

*THE DEPARTMENT OF DEFENSE*

---

***Groundwater Modeling System***

***SEEP2D PRIMER***



*SEEP2D Primer*

Copyright © 1998 Brigham Young University - Engineering Computer Graphics Laboratory

All Rights Reserved

Unauthorized duplication of the *GMS* software or documentation is strictly prohibited.

THE BRIGHAM YOUNG UNIVERSITY ENGINEERING COMPUTER GRAPHICS LABORATORY MAKES NO WARRANTIES EITHER EXPRESS OR IMPLIED REGARDING THE PROGRAM *GMS* AND ITS FITNESS FOR ANY PARTICULAR PURPOSE OR THE VALIDITY OF THE INFORMATION CONTAINED IN THIS USER'S MANUAL

The software *GMS* is a product of the Engineering Computer Graphics Laboratory of Brigham Young University. For more information about this software and related products, contact the ECGL at:

Engineering Computer Graphics Laboratory  
Rm. 300, Clyde Building  
Brigham Young University  
Provo, Utah 84602  
Tel.: (801) 378-2812  
Fax: (801) 378-2478  
e-mail: *GMS@byu.edu*  
WWW: <http://www.ecgl.byu.edu/software/GMS/>

Last revision: October 19, 1999



# TABLE OF CONTENTS

|          |   |            |
|----------|---|------------|
| <b>1</b> | <b>OVERVIEW OF SEEP2D MODELING .....</b>          | <b>1-1</b> |
| 1.1      | INTRODUCTION .....                                | 1-1        |
| 1.2      | APPLICATIONS .....                                | 1-1        |
| 1.3      | GOVERNING EQUATION .....                          | 1-2        |
| 1.4      | MODELING PROCESS .....                            | 1-2        |
| 1.4.1    | <i>Mesh Construction</i> .....                    | 1-3        |
| 1.4.2    | <i>Boundary Conditions</i> .....                  | 1-3        |
| 1.4.3    | <i>SEEP2D</i> .....                               | 1-3        |
| 1.4.4    | <i>Post-Processing With GMS</i> .....             | 1-3        |
| 1.5      | PRIMER OVERVIEW .....                             | 1-3        |
| <b>2</b> | <b>MODEL CONCEPTUALIZATION.....</b>               | <b>2-1</b> |
| 2.1      | INTRODUCTION .....                                | 2-1        |
| 2.2      | DETERMINING APPROPRIATE BOUNDARY CONDITIONS ..... | 2-1        |
| 2.2.1    | <i>No Flow Boundary Conditions</i> .....          | 2-1        |
| 2.2.2    | <i>Constant Head Boundary Conditions</i> .....    | 2-2        |
| 2.2.3    | <i>Exit Face Boundary Conditions</i> .....        | 2-2        |
| 2.2.4    | <i>Flow Rate Boundary Conditions</i> .....        | 2-2        |
| 2.2.5    | <i>Flux Boundary Conditions</i> .....             | 2-2        |
| 2.3      | CONFINED FLOW PROBLEMS .....                      | 2-2        |
| 2.4      | UNCONFINED FLOW PROBLEMS .....                    | 2-4        |
| 2.4.1    | <i>Deforming Mesh</i> .....                       | 2-4        |
| 2.4.2    | <i>Unsaturated Flow</i> .....                     | 2-5        |
| 2.5      | IMPERMEABLE FLOW BARRIERS .....                   | 2-5        |
| 2.6      | INTERNAL DRAINS .....                             | 2-6        |
| 2.7      | CORES OF DAMS .....                               | 2-7        |
| 2.8      | FLOW TO A WELL .....                              | 2-8        |
| 2.9      | PLAN OR AERIAL VIEW MODELS .....                  | 2-9        |
| <b>3</b> | <b>MESH CONSTRUCTION GUIDELINES.....</b>          | <b>3-1</b> |
| 3.1      | INTRODUCTION .....                                | 3-1        |
| 3.2      | MESH CONSTRUCTION .....                           | 3-1        |
| 3.2.1    | <i>Mesh Construction With GMS</i> .....           | 3-2        |
| 3.2.2    | <i>Basic Element Types</i> .....                  | 3-2        |
| 3.2.3    | <i>Material IDs</i> .....                         | 3-2        |
| 3.2.4    | <i>Renumbering</i> .....                          | 3-2        |
| 3.3      | MESH DENSITY .....                                | 3-4        |
| <b>4</b> | <b>MODEL PARAMETERS.....</b>                      | <b>4-1</b> |
| 4.1      | INTRODUCTION .....                                | 4-1        |
| 4.2      | ANALYSIS PARAMETERS .....                         | 4-1        |
| 4.2.1    | <i>Title</i> .....                                | 4-1        |
| 4.2.2    | <i>Datum</i> .....                                | 4-1        |
| 4.2.3    | <i>Unit Weight of Water</i> .....                 | 4-2        |
| 4.2.4    | <i>Type of Flow</i> .....                         | 4-2        |
| 4.2.5    | <i>Confined vs. Unconfined</i> .....              | 4-2        |

|          |  |            |
|----------|--|------------|
| 4.2.6    | <i>Saturated/Unsaturated Flow Modeling</i> .....                             | 4-2        |
| 4.2.7    | <i>Flow Lines</i> .....  | 4-5        |
| 4.3      | <b>MATERIAL PROPERTIES</b> .....   | 4-5        |
| 4.3.1    | <i>Hydraulic Conductivity</i> .....  | 4-6        |
| 4.3.2    | <i>Unsaturated Zone Parameters</i> .....                                     | 4-6        |
| <b>5</b> | <b>RUNNING SEEP2D</b> .....  | <b>5-1</b> |
| 5.1      | <b>INTRODUCTION</b> .....  | 5-1        |
| 5.2      | <b>FILES</b> .....   | 5-1        |
| 5.2.1    | <i>Input Files</i> .....   | 5-1        |
| 5.2.2    | <i>Output Files</i> .....  | 5-1        |
| 5.3      | <b>RUNNING SEEP2D</b> .....  | 5-2        |
| 5.4      | <b>TROUBLE SHOOTING</b> .....  | 5-2        |
| 5.4.1    | <i>Floating Point Error</i> .....  | 5-2        |
| 5.4.2    | <i>Maximum Number of Nodes or Elements Exceeded</i> .....                    | 5-2        |
| 5.4.3    | <i>Unable to Compute Flow Lines</i> .....                                    | 5-2        |
| 5.4.4    | <i>Mesh is Deformed when Modeling a Confined Problem</i> .....               | 5-3        |
| 5.4.5    | <i>Maximum Iterations Reached</i> .....                                      | 5-3        |
| 5.4.6    | <i>Problem Unexpectedly Aborts, or Takes an Unusual Amount of Time</i> ..... | 5-3        |

---

## Overview Of SEEP2D Modeling

### 1.1 Introduction

---

This document is a primer for the *GMS/SEEP2D* numerical modeling system. It should be read completely before any modeling is attempted with *SEEP2D*. The purpose of this primer is as follows:

1. To provide a general introduction to the *SEEP2D* program.
2. To provide the user with some guidelines for mesh generation and selection of boundary conditions and model parameters to help minimize improper application of *SEEP2D*.

The *SEEP2D* software was developed by the United States Army Engineer Waterways Experiment Station to model a variety of problems involving seepage. It is assumed that the reader will be using *GMS* in conjunction with *SEEP2D*. *GMS* is a graphical pre- and post-processor for *SEEP2D*. *GMS* was developed by the Brigham Young University Environmental Modeling Research Laboratory in cooperation with the Waterways Experiment Station.

### 1.2 Applications

---

The following conditions can be modeled using *SEEP2D*:

1. Isotropic and anisotropic soil properties.
2. Confined and unconfined flow for profile models.

3. Saturated/unsaturated flow for unconfined profile models.
4. Confined flow for plan (areal) models.
5. Flow simulation in the saturated and unsaturated zones.
6. Heterogeneous soil conditions.
7. Axisymmetric models such as flow from a well.
8. Drains.

The following conditions cannot be modeled using *SEEP2D*:

1. Transient or time varying problems
2. Unconfined plan models

### 1.3 Governing Equation

---

The governing equation used in the *SEEP2D* models is:

$$\nabla \cdot (\mathbf{K} \cdot \nabla h) = 0 \dots\dots\dots 1.1$$

or

$$\frac{\partial}{\partial x} \left( K_{xx} \frac{\partial h}{\partial x} + K_{xy} \frac{\partial h}{\partial y} \right) + \frac{\partial}{\partial y} \left( K_{yy} \frac{\partial h}{\partial y} + K_{yx} \frac{\partial h}{\partial x} \right) = 0 \dots\dots\dots 1.2$$

where:

h = total head (elevation head plus pressure head)

K = hydraulic conductivity

This equation is often referred to in the literature as the Laplace equation.

### 1.4 Modeling Process

---

In a typical modeling problem involving the *SEEP2D* software, a series of tasks are performed in a specific sequence. Each of these steps is described briefly.



### 1.4.1 Mesh Construction

---

First of all, a finite element mesh must be constructed which represents the region being modeled. This mesh is typically constructed using *GMS*. A variety of mesh generation and interactive editing tools are provided in *GMS*. These tools are described in more detail in the *GMS Tutorial* and *GMS Reference Manual*.

### 1.4.2 Boundary Conditions

---

Once a mesh has been constructed, boundary conditions are applied to the mesh. Boundary conditions are typically entered as constant head at a node, head equals elevation (exit face) at a node, or as an incoming flux at a node or along an element edge. Hydraulic conductivities must also be entered for the two principal directions for different regions (representing different soil types) in the mesh. All of these parameters can be input interactively using *GMS*. The mesh geometry and boundary conditions are saved by *GMS* to a *SEEP2D* input file.

### 1.4.3 SEEP2D

---

Once the mesh is constructed, the *SEEP2D* program can be executed to calculate the head, flow, discharge (Darcian) velocity, and pore pressure at every node in the mesh. For problems encountered when running *SEEP2D*, refer to section 5.4 of this primer.

### 1.4.4 Post-Processing With GMS

---

After running *SEEP2D*, results may be viewed in *GMS*. *GMS* displays contours of equipotential total or pressure head, contours of pore pressures, velocity vectors, and, for many classes of problems, flow lines. In addition, the summed flow of all selected nodes can be viewed. Upon viewing the solution, the user must ascertain if the results are reasonable. If necessary, the mesh should be refined, boundary conditions modified, or input coefficients altered, and a new solution computed.

## 1.5 Primer Overview

---

Each of the steps of the modeling process outlined above are described in more detail in the remainder of this primer. Some general guidelines concerning model conceptualization, particularly with regard to the use of boundary conditions, are given in Chapter 2. Mesh construction guidelines are outlined in Chapter 3. Chapter 4 describes *SEEP2D* analysis parameters other than boundary conditions, such as the datum and material properties. Finally, in Chapter 5, details concerning the execution of *SEEP2D* are given, including a section on trouble shooting of commonly encountered problems.



---

## ***Model Conceptualization***

### **2.1 Introduction**

---

*SEEP2D* is used to model seepage conditions for actual physical problems. Before a problem can be modeled, the subregion of the actual site to be modeled must be determined and a set of appropriate boundary conditions must be selected. This process is called model conceptualization. Model conceptualization is perhaps the most important part in developing any *SEEP2D* model. The accuracy of the model will be significantly influenced by the choices made during the model conceptualization process.

### **2.2 Determining Appropriate Boundary Conditions**

---

Deciding what type of boundary conditions to use is the most important part of model conceptualization. The boundary conditions define the flow model. Constant head boundary conditions are typically used to represent standing bodies of water. However, velocity, exit face, and flux type boundaries are also important for modeling certain situations.

#### **2.2.1 No Flow Boundary Conditions**

---

If a boundary condition has not been explicitly applied to a boundary node, the node is assumed to be a "no-flow" boundary and the flow direction will be computed parallel to the boundary. Thus, all boundary nodes are no flow boundaries by default.

### **2.2.2 Constant Head Boundary Conditions**

---

Constant head boundary conditions represent boundaries where the head is known. They typically are found where water is ponding or at the boundary of a region where the water table is known to remain constant. Since the head along such boundaries cannot change, they represent regions of the model where flow enters or exits the system (flow lines are always orthogonal to constant head boundaries).

### **2.2.3 Exit Face Boundary Conditions**

---

Exit face boundary conditions imply that the head is equal to the elevation (assuming that the datum is 0). They are used when modeling unconfined flow problems and should be placed along the face where the free surface is likely to exit the model. This boundary condition must be used if the option for deforming the mesh to the phreatic surface has been selected. It may also be used with a saturated/unsaturated flow model. In this case, if the head at a node on the boundary becomes greater than the node elevation during the iteration process, the head at the node is fixed at the nodal elevation and the node acts as a specified head boundary. Thus, water is allowed to exit the boundary above the tailwater. If an exit face boundary is not used with a saturated/unsaturated flow model, all of the flow will be forced through the tailwater.

### **2.2.4 Flow Rate Boundary Conditions**

---

Flow rate boundary conditions are used to specify nodes at which a certain flow rate is known to exist. They are used primarily when modeling wells and the flow specified represents the pumping rate. Negative values represent extraction of fluid from the system whereas positive values represent injection.

### **2.2.5 Flux Boundary Conditions**

---

Flux boundary conditions are used to specify a known flux rate [L/T] along a sequence of element edges on the perimeter of the mesh. They are often used to simulate infiltration. Flux into the system is positive and flux out of the system is negative.

## **2.3 Confined Flow Problems**

---

The governing equation defined in the previous chapter assumes that the porous media is saturated. This is always the case for confined flow problems where the upper mesh boundary defines the limit of the saturated zone. These types of boundaries typically occur in areas where a standing body of water or an impervious clay liner or material overlays the region of interest as demonstrated in Figure 2.1 and Figure 2.2. Confined flow problems are

modeled using a combination of constant head and no flow boundary conditions.

The vertical boundaries on the sides of the models shown in Figure 2.1 and Figure 2.2 are modeled using no flow boundary conditions. They can also be modeled using constant head boundary conditions. Both approaches will give similar solutions provided the region is extended a suitable distance as discussed below.

In addition to determining which boundary conditions are appropriate for the ends of the models shown in Figure 2.1 and Figure 2.2, the user must also determine how far to extend the model in either direction. In either case, most of the flow is concentrated in the vicinity of the sheet piles. As the mesh is extended in either direction, a point is reached where extending the mesh further makes very little difference in the solution. A good approach to determining how far to extend a mesh is to compute a series of solutions where the mesh is extended for each subsequent solution until the extension is seen to have little effect on the solution.

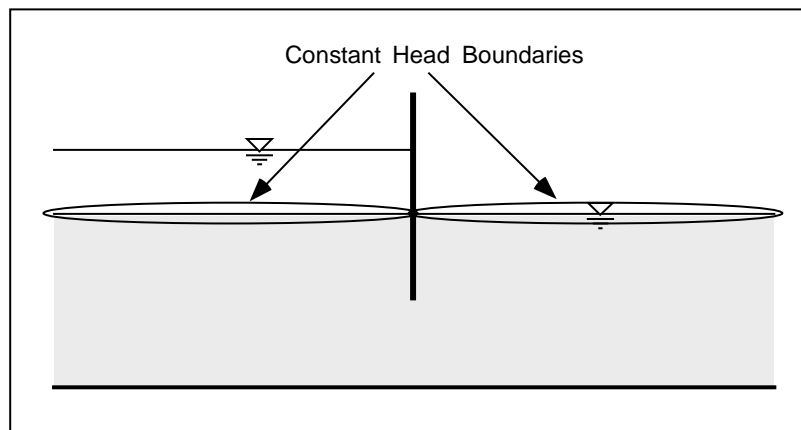


Figure 2.1 Confined Flow With Standing Water on Either Side of a Flow Barrier.

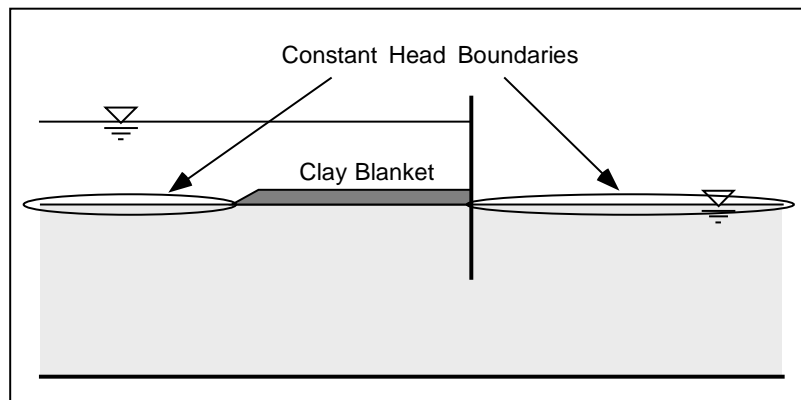


Figure 2.2 Confined Flow Where a Clay Blanket Exists.

## 2.4 Unconfined Flow Problems

With unconfined problems, the position of the free surface and the point downstream where the free surface exits is unknown. The Dupuit problem shown in Figure 2.3 is a classic example of an unconfined flow problem. This same type of condition exists when modeling flow through a dam as shown in Figure 2.4. Unconfined flow problems can be modeled in *SEEP2D* by either deforming the mesh to the phreatic surface so that flow only occurs in the saturated zone or by simulating flow in both the saturated and unsaturated zone

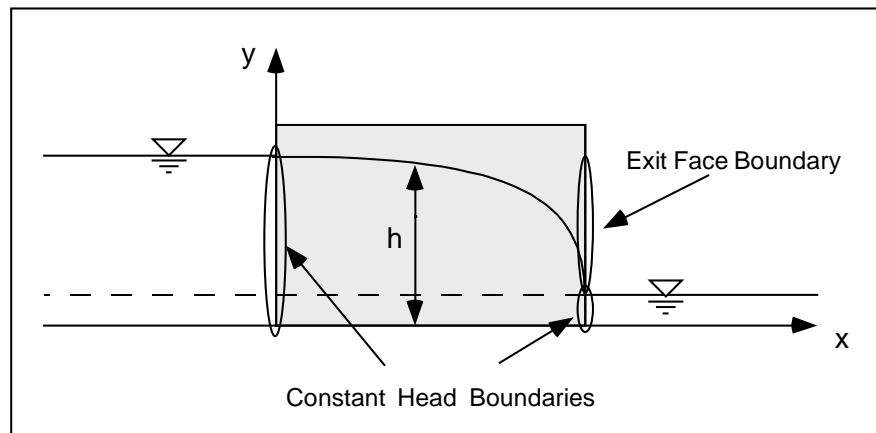


Figure 2.3 Unconfined Flow Represented by the Dupuit Problem.

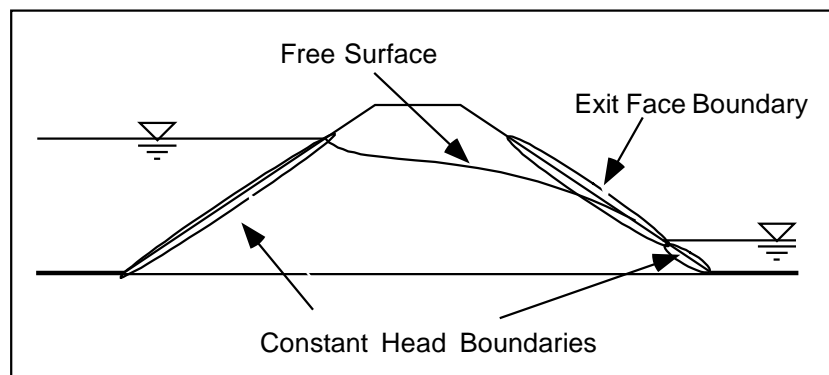


Figure 2.4 Unconfined Flow Through a Dam

### 2.4.1 Deforming Mesh

If the deforming mesh option is selected, the boundary conditions are implemented by placing constant head boundary conditions in the locations where the head and tail water are known, and placing exit face boundary conditions along the boundary where the phreatic surface is assumed to exit as shown in Figure 2.3 and Figure 2.4. *SEEP2D* will automatically solve for the free surface and deform the mesh to this boundary. The resulting set of

solution files will include a geometry file containing the nodes and elements in the deformed mesh.

### **2.4.2 Unsaturated Flow**

---

If the option to model saturated/unsaturated flow is selected, the mesh is not deformed. Rather the flow in the entire problem domain, both in the saturated and the unsaturated zone is modeled. In this case, a method must be selected for defining the relative conductivity in the unsaturated zone and parameters related to that method must be specified for each of the material zones. These options are defined in more detail in sections 4.2.5 and 4.2.6. If an exit face boundary is used, water is allowed to exit the model along the exit face. Otherwise, all of the water is forced to exit at the tail water.

## **2.5 Impermeable Flow Barriers**

---

Sheet piles, grout curtains and other structures used to create impermeable flow barriers can be modeled in *SEEP2D* by creating a discontinuity or "crack" in the mesh. By default the boundary of a finite element mesh is a "no flow" boundary, therefore flow barriers such as the ones shown in Figure 2.5a are modeled by making the mesh boundary extend in a crack like fashion down to the depth of the barrier as shown in Figure 2.5b (the "crack" in the mesh has been exaggerated for illustration purposes). Since gradients are typically higher around the tip of such barriers, it is often useful to refine the mesh more in these regions.

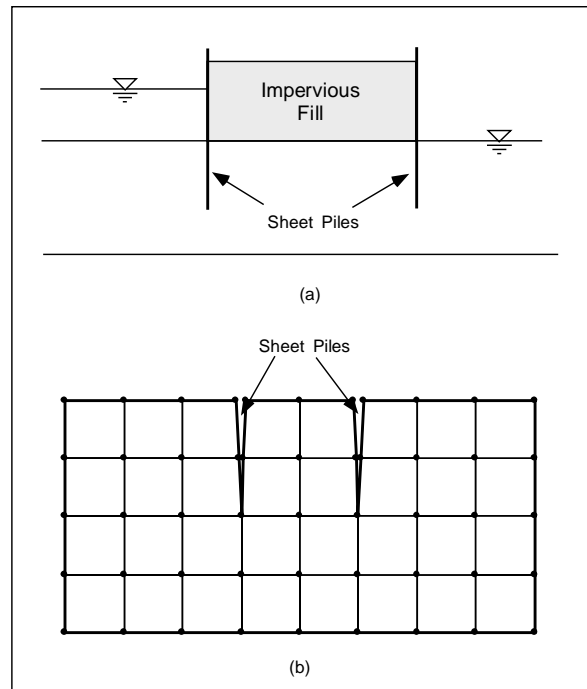


Figure 2.5 a) Sheet Piles Creating an Impermeable Flow Barrier (b) A Mesh Representing the Sheet Piles.

## 2.6 Internal Drains

Internal drains can be modeled in *SEEP2D* by placing a hole in the mesh and by applying either constant head or exit face boundary conditions to the nodes on the perimeter of the hole. If the drain is placed below a region where water is ponding or in a region where it is anticipated that the soil will remain saturated, constant head boundary conditions should be assigned to the drain. For example, the model shown in Figure 2.6 contains a sheet pile with a drain at the bottom of the sheet pile. The water is expected to pond on both sides of the sheet pile.

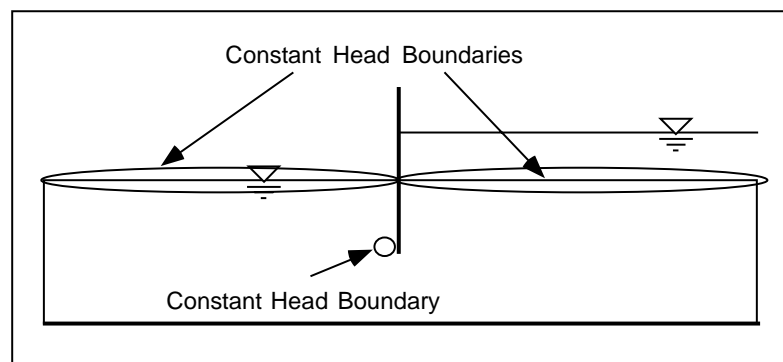


Figure 2.6 Confined Approach for Modeling an Internal Drain.



In some cases, there is not a sufficient supply of water for the water to pond above a drain and the drain should be modeled as an unconfined problem using the saturated/unsaturated option with the boundary conditions shown in Figure 2.7. Another approach is to use the deforming mesh option with exit face boundary conditions as shown in Figure 2.8. In this example it is assumed that water is ponded on the downstream side of the flow barrier but not on the upstream side.

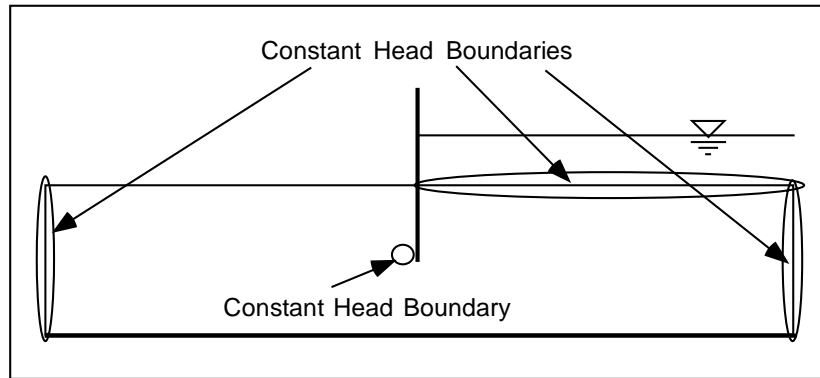


Figure 2.7 Saturated/Unsaturated Approach for Modeling an Internal Drain.

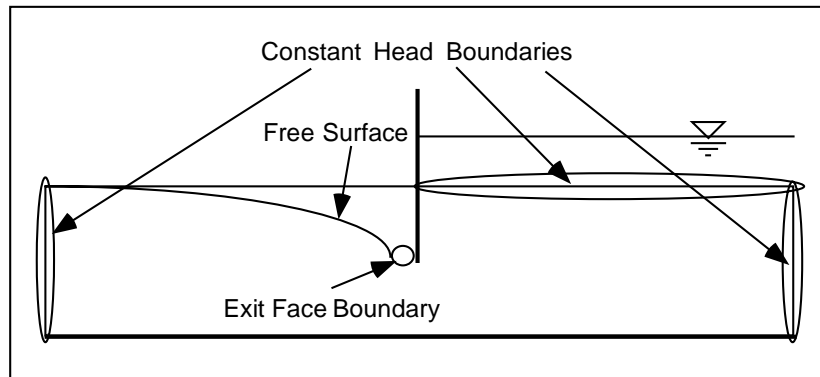


Figure 2.8 Deforming Mesh Approach for Modeling an Internal Drain.

## 2.7 Cores of Dams

When modeling an earth dam with a core, the core can be modeled by forcing the element edges to conform to the boundary of the core and by assigning separate material properties to the core and shell of the dam (Figure 2.9).

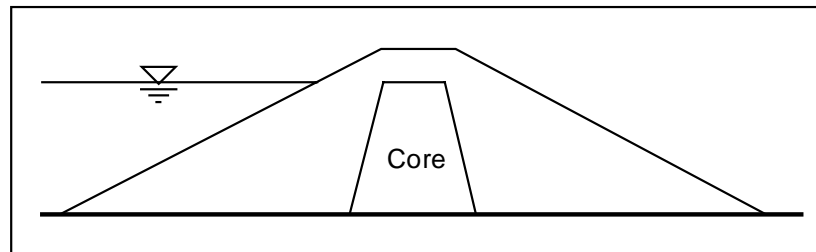


Figure 2.9 Earth Dam with a Core.

If the deforming mesh option is selected and if the hydraulic conductivity of the core material differs by more than two orders of magnitude from the hydraulic conductivity of the shell material, then *SEEP2D* may have difficulties computing the location of the phreatic surface. This is due to the fact that the slope of the phreatic surface is very steep in the core but is very flat in the shell. The transition from a steep slope to a flat slope may cause *SEEP2D* to have difficulty converging on a solution. In such cases, the portion of the model downstream from the core can be omitted as shown in Figure 2.10 without significant effect on the solution since most of the drawdown occurs in the core of the dam.

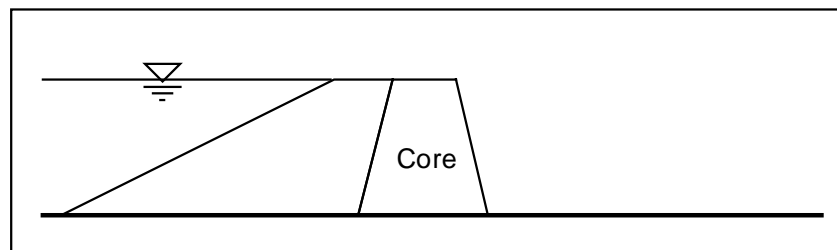


Figure 2.10 Suggested Boundaries when Modeling a Core.

## 2.8 Flow to a Well

Flow to a well is a classic flow problem which can be solved using *SEEP2D*. For such a condition the axisymmetric option of *SEEP2D* should be selected (see section 4.2.4). The axis of symmetry for the model is the axis of the well. In other words, the well should be established along the  $y$  axis. The well should be placed on the left edge of the model and the origin for the coordinate system should be placed at the lower left corner of the model as shown in Figure 2.11. For a fully penetrating well, the left edge of the mesh should begin at  $x=r$  where  $r$  is the well radius. In the case of a partially penetrating well, a notch the thickness of the well radius should be placed in the mesh to the depth of the well as shown in Figure 2.11

The boundary conditions assigned to a well problem depend on the type of aquifer being modeled. In most cases, a constant head boundary condition should be applied to the right end of the model. For a confined aquifer, a flow rate boundary condition should also be placed at the bottom of the well. For

an unconfined aquifer, a flow rate boundary condition should be placed at the bottom of the well and, if the deforming mesh option is being used, exit face boundary conditions should be applied to the nodes on the well above the bottom. If the flow to the well is modeled as a point sink then the units on the flow rate would be length<sup>3</sup> / time. However, if the flow to the well is modeled over the screened interval of the well with a flux boundary condition then the units on the flow rate would be length<sup>3</sup> / time divided by the surface area of the screened interval resulting in units of length / time.

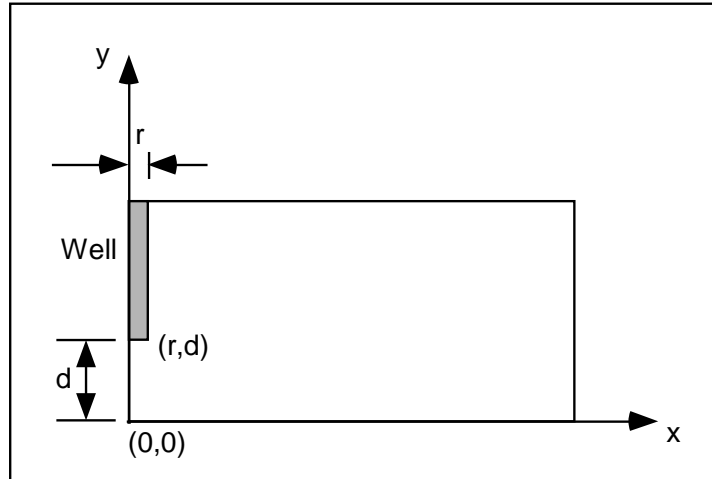


Figure 2.11 Boundary for a Partially Penetrating Well.

## 2.9 Plan or Aerial View Models

The governing equation used for solving profile seepage models of confined aquifers is as follows:

$$\nabla \cdot (T \cdot \nabla h) = 0 \dots\dots\dots 2.8$$

or

$$\frac{\partial}{\partial x} \left( T_{xx} \frac{\partial h}{\partial x} + T_{xy} \frac{\partial h}{\partial y} \right) + \frac{\partial}{\partial y} \left( T_{yy} \frac{\partial h}{\partial y} + T_{yx} \frac{\partial h}{\partial x} \right) = 0 \dots\dots\dots 2.9$$

where:

h = total head (elevation head plus pressure head)

T = transmissivity.

This equation is the same as the equation used in *SEEP2D* except that the hydraulic conductivity terms (k) have been replaced by transmissivity terms (T = k X aquifer thickness). Therefore, profile seepage models of confined

aquifers can be solved using *SEEP2D* as long as transmissivities are entered for the hydraulic conductivities.

Since the governing equation used for modeling unconfined aquifers is different than the equation used by *SEEP2D*, unconfined plan view models **cannot** be modeled using *SEEP2D*.

For plan view models, the values used for the constant head boundary conditions may need to be shifted by a constant value. For example, the model shown in Figure 2.12a represents a simple plan view model with constant head boundary conditions on two sides and no flow boundary conditions on the other two sides. Since the y dimension of the model is 500', the y coordinates of the nodes at the top of the model will be 500'. If a constant head boundary condition of 10' is assigned to the nodes at the top of the model, *SEEP2D* will assume that the model is an unconfined profile model (since  $h=500' > h=10'$ ) and it will attempt to draw down a phreatic surface. This problem can be avoided by ensuring that all constant head boundary conditions are given a value that is greater than the value of any of the y coordinates in the model as shown in Figure 2.12b. The constant added to the heads can be subtracted from the computed heads. The problem could also be solved by setting the datum to 500' and the respective heads to 10' and 6'.

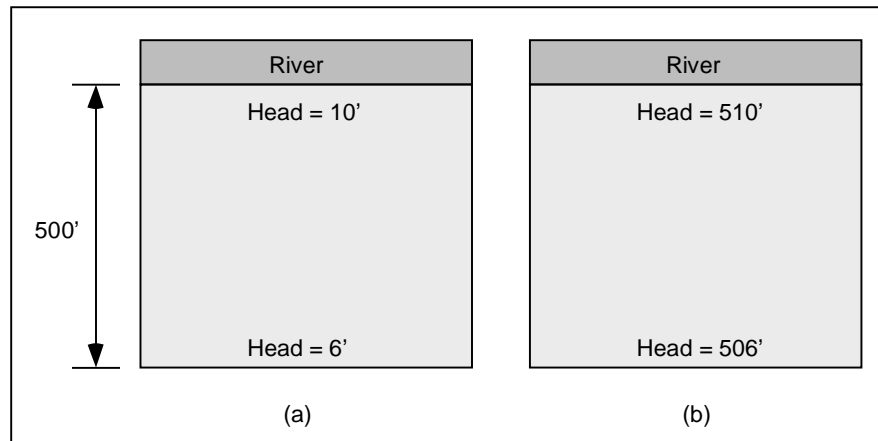


Figure 2.12 Plan View Modeling (a) Actual Parameters (b) Model Parameters

---

## ***Mesh Construction Guidelines***

### **3.1 Introduction**

---

A fundamental part of solving a seepage problem using *SEEP2D* is the construction of a two-dimensional finite element mesh representing the cross section or region being modeled. This mesh is typically constructed using *GMS*. Some general guidelines concerning the construction of meshes for input into *SEEP2D* are described in this chapter.

### **3.2 Mesh Construction**

---

The finite element mesh used by *SEEP2D* is composed of nodes and elements. A sample mesh is shown in Figure 3.1. A finite element mesh can be thought of as a cross section representing the region to be modeled that is formed by piecing together a large number of small triangular and quadrilateral patches called "elements". The nodes are the xy points that define the geometry of the mesh. The elements define the mesh topology and are formed by connecting a set of nodes in the mesh with edges.

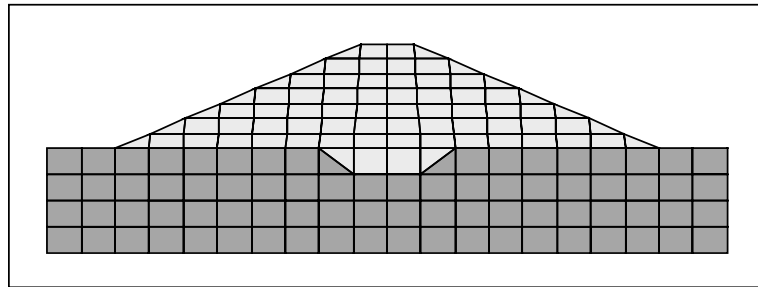


Figure 3.1 Sample Finite Element Mesh.

### 3.2.1 Mesh Construction With *GMS*

---

Finite element meshes can be constructed using *GMS*. There are a large number of tools available in *GMS* to assist the user in the construction and editing of meshes. Details concerning the mesh construction and editing tools are provided in the *GMS Tutorial* and *GMS Reference Manual*.

### 3.2.2 Basic Element Types

---

Two types of elements, triangles and quadrilaterals are commonly used in the construction of two-dimensional meshes (Figure 3.2). Quadratic elements (elements with midside nodes) are not supported by *SEEP2D*.

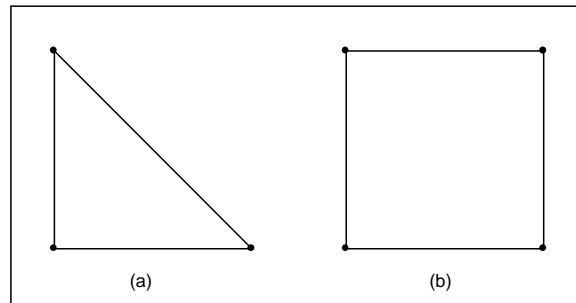


Figure 3.2 The Two Basic Elements: (a) Linear Triangles (b) Linear Quadrilaterals

### 3.2.3 Material IDs

---

Each element in the mesh should have an associated material ID. The material ID is an index to a list of material properties (hydraulic conductivities). The material properties are described in more detail in Chapter 4.

### 3.2.4 Renumbering

---

Each node and element in the mesh has an associated ID. The order in which the nodes are numbered is very critical and should be well understood by the

user. The node and element numbering sequence can be altered using *GMS*. The first step in altering the numbering sequence is to select a node string. A node string is a sequence of nodes which is typically on the boundary of the mesh. *GMS* can be used to automatically renumber the nodes and elements using this node string. The nodes and elements are numbered by first numbering the nodes and elements connected to the string and then numbering the remainder of the mesh by progressing outward from the string in a sweeping fashion (Figure 3.3).

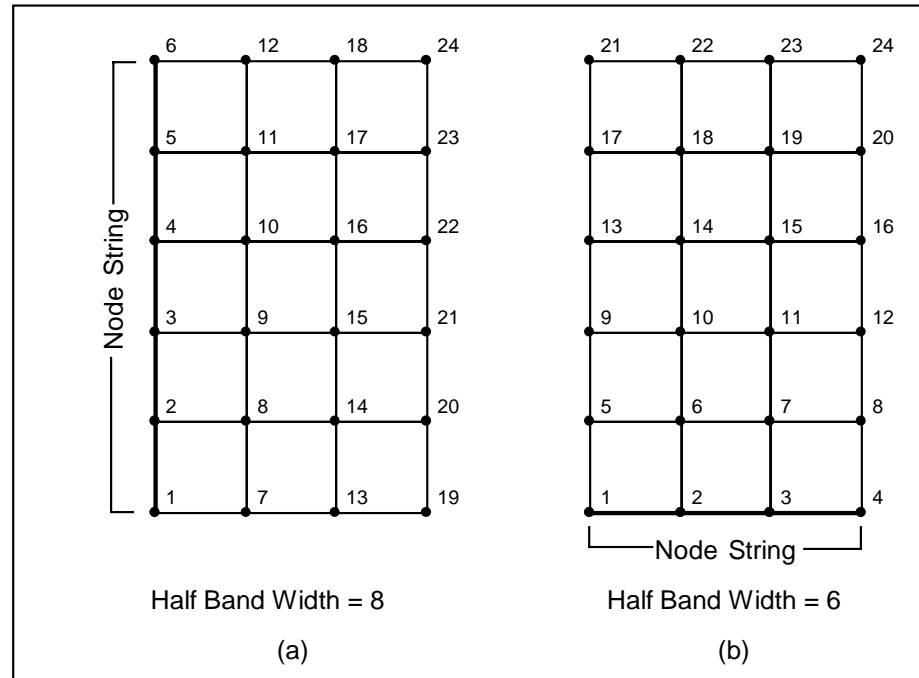


Figure 3.3 Sample Results of Renumbering Process. (a) Results With Node String on End of Mesh. (b) Results With Node String on Side of Mesh.

As seen in Figure 3.3, the location of the node string controls the node and element numbering. The result of the numbering sequence is represented by the maximum nodal half band width. This parameter is related to the maximum difference in ID's of the nodes defining an element. Since the solution time and the memory requirements for *SEEP2D* are proportional to the square of the maximum nodal half band width, different numbering sequences on the same mesh can produce drastically different solution times. Different node strings can be selected and tested to find the string resulting in the smallest half band width. In many cases, the optimal location of the node string is immediately obvious. If the mesh is shaped such that there are distinct longitudinal (major axis) and lateral (minor axis) dimensions, the node string should be placed on one end of the mesh such that the numbering sequence progresses longitudinally along the mesh as shown in Figure 3.3b. This tends to minimize the nodal band widths.

Since the numbering sequence has such a dramatic effect on the *SEEP2D* solution time and memory requirements, the mesh should always be renumbered after the mesh has been constructed or after the mesh has been edited.

### 3.3 Mesh Density

---

In general, the higher the resolution of the mesh, the more accurate the solution. While theoretically the size (number of nodes and elements) of a *SEEP2D* model is unlimited, practically speaking there are some limitations. *SEEP2D* is a FORTRAN program with constant length array sizes. By default, *SEEP2D* allows meshes as large as 2000 nodes and 2000 elements. If needed, the *SEEP2D* program can be recompiled with larger dimensions.

When increasing the density of a mesh to increase the accuracy, one approach is to increase the density of the mesh globally (by subdividing each element). A more efficient approach is to only refine the mesh in areas where there is high flow or high gradient in head. For example, in constrictions around flow barriers, at wells, and near drains the node density should be higher. Most of the computational error is concentrated in such areas.



---

## ***Model Parameters***

### **4.1 Introduction**

---

Once a finite element mesh is created, and boundary conditions have been applied, several model parameters must be set to complete the definition of the model. These parameters include global analysis parameters and material properties. All model parameters can be specified interactively using *GMS*.

### **4.2 Analysis Parameters**

---

Several global options for controlling how *SEEP2D* solves the seepage problem must be specified. These options define the units and the type of problem.

#### **4.2.1 Title**

---

A title can be input to *SEEP2D*. This title is used in the header of the text output file.

#### **4.2.2 Datum**

---

By default, the datum of the model is at zero, but it can be specified to any convenient value, such as the value corresponding to the base or lowest y coordinate of the model.

### 4.2.3 Unit Weight of Water

---

The unit weight of water must be entered. *SEEP2D* uses this value to compute pore pressures. The weight and length units defined in this value should be consistent with the units used elsewhere in the model.

### 4.2.4 Type of Flow

---

The type of flow must be specified either as plane flow or axisymmetric flow. The axisymmetric option should be selected for models corresponding to flow to a single well as described in section 2.8. All other models should use the plane flow option.

### 4.2.5 Confined vs. Unconfined

---

The type of model should be specified as either confined or unconfined. For confined models, the entire model domain is assumed to be saturated. No exit face boundary conditions should be applied and the unsaturated zone material properties are not required.

For unconfined models, two options are available for dealing with the unsaturated zone: (1) deforming mesh and (2) saturated/unsaturated flow modeling. For both types of problems, exit face boundary conditions should be applied along the boundary of the mesh where the free surface is expected to exit. With the deforming mesh option, *SEEP2D* iterates to find the location of the phreatic surface and the mesh is deformed or truncated so that the upper boundary of the mesh matches the phreatic surface. The solution files from this type of simulation include a geometry file containing the deformed mesh. With the saturated/unsaturated option, the mesh is not modified and the flow in both the saturated and unsaturated zone is modeled. The hydraulic conductivity in the unsaturated zone is modified (reduced) using either the linear frontal method or the Van Genuchten method. The hydraulic conductivity in the unsaturated zone is modified using the equations described in the following section.

### 4.2.6 Saturated/Unsaturated Flow Modeling

---

For unconfined problems, *SEEP2D* can be used to simulate flow in the unsaturated zone. Equation 1.1 on page 1-2 is the governing differential equation which is solved with *SEEP2D*. The solution to the equation is a function describing the total head,  $h$ , as a function of  $x$  and  $y$ . The following equation is the general form for computing heads:

$$h = h_p + h_{el} - h_d \dots\dots\dots 2.1$$

where:

$h$  = total head.

$h_p$  = pressure head

$h_{el}$  = elevation head

$h_d$  = datum.

The hydraulic conductivity term,  $K$ , in equation 1.1 is typically thought of as the saturated hydraulic conductivity. In other words, it is only valid when the  $h_p$  in equation 2.1 is positive. In order to model flow in unsaturated regions (negative  $h_p$ ) the hydraulic conductivity can be expressed as the product of the saturated hydraulic conductivity  $k_s$  and the relative hydraulic conductivity  $k_r$  as follows:

$$k = k_r k_s \dots\dots\dots 2.2$$

As long as the porous medium is saturated ( $h_p > 0$ ),  $k_r = 1$ , but as  $h_p$  goes negative, the value of  $k_r$  decreases towards zero. By using equation 2.2 for hydraulic conductivity, *SEEP2D* can be used to simulate flow in unsaturated portions of a model. The parameter  $k_r$  is specified on a material-by-material basis. Two approaches can be used to define  $k_r$ : a frontal function or the Van Genuchten model.

**Frontal Function**

The frontal function method is used by numerous models for simulating flow in unsaturated regions and is defined by specifying a  $k_{r0}$  and an  $h_0$  for each material zone in the model. In the saturated zone ( $h_p > 0$ ) and  $k_r = 1$ . In the unsaturated zone where  $h_p < h_0$ ,  $k_r = k_{r0}$ , and for values of  $h_p$  between 0 and  $h_0$ ,  $k_r$  varies linearly from 1 to  $k_{r0}$ . This is illustrated in Figure 4.1.

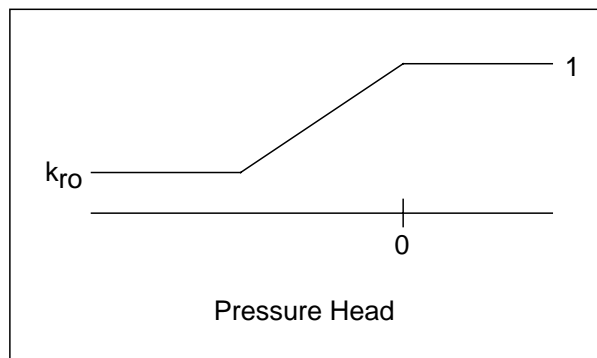


Figure 4.1 Frontal Function.

Notice that if  $h_0=0$  then the front becomes a step function as shown in Figure 4.2.

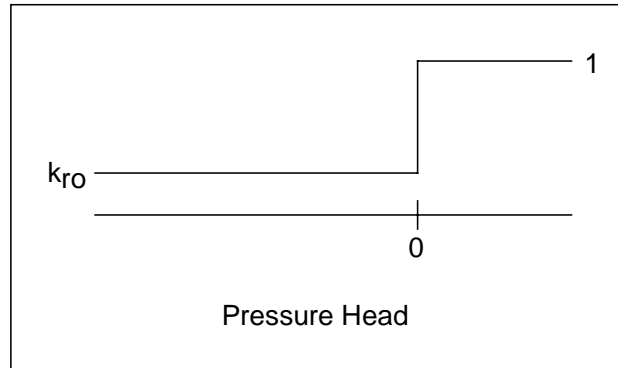


Figure 4.2 Frontal Function Degenerates to a Step Function for  $h_0 = 0$ .

Equations 2.3-2.5 summarize how  $k_r$  is computed for different values of  $h_p$ .

$$k_r = 1 \quad h_p > 0 \dots\dots\dots 2.3$$

$$k_r = (k_{r0} - 1) \left( \frac{h_p}{h_0} \right) + 1 \quad h_0 < h_p < 0 \dots\dots\dots 2.4$$

$$k_r = k_{r0} \quad h_p < h_0 \dots\dots\dots 2.5$$

**Van Genuchten Model**

Van Genuchten (1980) developed a model for computing the relative hydraulic conductivity using a term for effective saturation,  $\bar{S}$  defined by equation 2.6.

$$\bar{S} = \left[ 1 + (\alpha h_p)^n \right]^{-m} \dots\dots\dots 2.6$$

where

$\bar{S}$  = effective saturation

$\alpha, n$  = Van Genuchten parameters

and

$$m = 1 - \frac{1}{n}$$

The relative hydraulic conductivity,  $k_r$ , can then be defined by equation 2.7.

$$k_r = \bar{S}^{\frac{1}{2}} \left[ 1 - \left( 1 - \bar{S}^{\frac{1}{m}} \right)^m \right]^2 \dots\dots\dots 2.7$$

Table 2.1 provides some representative values for  $\alpha$  and  $n$  for different soil types.

| Soil Type       | $\alpha$ [cm <sup>-1</sup> ] | $n$  |
|-----------------|------------------------------|------|
| Clay**          | 0.008                        | 1.09 |
| Clay Loam       | 0.019                        | 1.31 |
| Loam            | 0.036                        | 1.56 |
| Loam Sand       | 0.124                        | 2.28 |
| Silt            | 0.106                        | 1.37 |
| Silt Loam       | 0.020                        | 1.41 |
| Silty Clay      | 0.005                        | 1.09 |
| Silty Clay Loam | 0.010                        | 1.23 |
| Sand            | 0.145                        | 2.68 |
| Sandy Clay      | 0.027                        | 1.23 |
| Sandy Clay Loam | 0.059                        | 1.48 |
| Sandy Loam      | 0.075                        | 1.89 |

\*\* Agricultural soil, less than 60% clay  
Source: Carsel and Parrish (1988)

References:

Carsel, R.F., and R.S. Parrish, Developing joint probability distributions of soil-water retention characteristics, *Water Resources Research*, Vol. 24, No. 5, pp. 755-769, 1988.

Van Genuchten, M. Th., 1980, "A Closed-Formed Equation for Predicting the Hydraulic Conductivity of Unsaturated Soils," *Soil Science Society of America Journal*, Vol. 44, No. 5.

#### 4.2.7 Flow Lines

The primary result of a *SEEP2D* analysis is the total head at every node in the model. By contouring these values in *GMS*, lines of equipotential head can be displayed. Flow lines or stream function (orthogonal to the equipotential lines) can be computed using an equation identical to the one used for solving heads. *SEEP2D* computes flow lines by first computing the head values and then redefining the boundary conditions and solving for pseudo-flow values ("flow potential" values) at the nodes. These flow values are contoured in *GMS* to generate flow lines.

### 4.3 Material Properties

In order for a solution to be computed, material or soil properties for all elements must be defined. The material properties include hydraulic conductivity and unsaturated zone parameters.

### 4.3.1 Hydraulic Conductivity

---

SEEP2D allows for different hydraulic conductivities along the major and minor axes (anisotropic conditions) to be defined. In addition, an orientation angle can be entered which defines the angle between the x axis of the model and the major axis of hydraulic conductivity as shown in Figure 4.3 below.

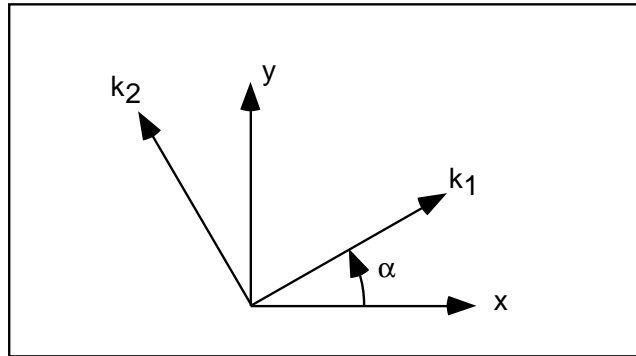


Figure 4.3 Material Orientation Angle.

Heterogeneous models can be created by specifying different values of hydraulic conductivity for the elements representing the different layers or regions.

### 4.3.2 Unsaturated Zone Parameters

---

If one of the saturated/unsaturated flow modeling options has been selected for an unconfined model, a set of unsaturated zone parameters must also be defined for each material. If the linear front option is selected, a minimum pressure head ( $h_0$ ) and a minimum relative conductivity must be specified ( $k_{ro}$ ). If the Van Genuchten option is selected, the Van Genuchten  $\alpha$  and  $n$  parameters must be specified. These options are described in more detail in Section 4.2.6.

---

## ***Running SEEP2D***

---

### **5.1 Introduction**

Once a finite element mesh has been constructed and boundary conditions and material properties have been defined, *SEEP2D* can be used to compute the head, flow, and Darcy velocity at each node in the mesh. The steps necessary to run *SEEP2D* are described in this chapter.

---

### **5.2 Files**

The files associated with *SEEP2D* analysis can be divided into two categories: input files and output files:

---

#### **5.2.1 Input Files**

The input files to *SEEP2D* consist of two files: a super file and the *SEEP2D* input file. The super file is a short text file that contains the names of the input file and the output files. The *SEEP2D* input file contains the mesh, boundary conditions, and model parameters.

---

#### **5.2.2 Output Files**

The output from *SEEP2D* consists of three files: a printed output file, a geometry file, and a data set file. The printed output file is a text file containing a summary of the input data and listing of the computed solution. The geometry file contains the modified mesh and is only output if the

problem is unconfined and the deformed mesh option is selected. This file must be read into *GMS* prior to reading and viewing the solution file. The data set file is a special file used by *GMS* for post-processing. It contains the total head, the pressure head, the Darcy velocity, and flow potential values (for computing flow lines).

### 5.3 Running *SEEP2D*

---

*SEEP2D* can be executed two ways: directly from the *GMS* menu or from the command line. When executing from the command line, the user is prompted to input the name of the super file. *SEEP2D* then computes a solution and writes the solution to the output files specified in the super file.

### 5.4 Trouble Shooting

---

Some of the common problems which can occur while running *SEEP2D*, along with possible remedies are given below.

#### 5.4.1 Floating Point Error

---

If a hydraulic conductivity for one of the material properties referenced by an element is zero then a floating point divide by zero can occur. Be sure to check all the material properties being referenced to make sure that valid hydraulic conductivities are given.

#### 5.4.2 Maximum Number of Nodes or Elements Exceeded

---

While *GMS* will allow you to create large meshes, the node and element arrays in *SEEP2D* have fixed dimensions. By default, the number of nodes and the number of elements are both set to 2000. If your model has more nodes or elements than this you will have to either redefine your mesh so that it has 2000 or less or resize the *SEEP2D* arrays. To resize the arrays you will need a FORTRAN compiler. The dimensions can be changed by editing the SEEP.INC file found in the source directory. Change the parameters to the desired limits (be sure to change MXBNDW proportional to the amount MXNODES is changed). Follow the directions provided by your compiler to recreate a *SEEP2D* executable file. If you do not have a FORTRAN compiler contact *GMS* technical support for information regarding the acquisition of a large array sized executable.

#### 5.4.3 Unable to Compute Flow Lines

---

This error occurs when *SEEP2D* is unable to reverse the boundary conditions to compute flow lines. However, all other results are accurate and valid.



#### **5.4.4 Mesh is Deformed when Modeling a Confined Problem**

---

Check to be sure that you have accounted for the elevation of the node when assigning the boundary condition. If the constant head boundary condition assigned to a node is smaller than the elevation of the node, *SEEP2D* will attempt to deform the mesh in that region.

#### **5.4.5 Maximum Iterations Reached**

---

When solving unconfined problems, *SEEP2D* deforms the boundary of the mesh to match the phreatic surface. The boundary is moved in an iterative fashion. The progress of the iteration and a convergence parameter are printed to the screen. In some cases, the iteration does not converge after the maximum number of iterations (30) has been reached. In such cases, you should review the assigned boundary conditions to ensure that they are correct. You may also wish to import the computed solution to *GMS*. Often, the solution can give clues to where the mesh deformation process is having difficulty. Sometimes the problem can often be fixed by increasing the density of the mesh in the problem area.

#### **5.4.6 Problem Unexpectedly Aborts, or Takes an Unusual Amount of Time**

---

Be sure to check the band width before saving the file. If the model has a high band width try renumbering again, or try renumbering using a different node string.



# INDEX

|  |          |
|--|----------|
| <b>A</b>   |          |
| Analysis Parameters .....  | 4-1      |
| anisotropic .....  | 4-6      |
| Applications .....   | 1-1      |
| axisymmetric .....   | 2-8      |
| Axisymmetric flow .....  | 4-2      |
| <b>B</b>   |          |
| Basic Element Types .....  | 3-2      |
| Boundary Conditions .....  | 1-3, 2-1 |
| Brigham Young University Engineering Computer<br>Graphics Laboratory ..... | 1-1      |
| <b>C</b>   |          |
| confined aquifers .....  | 2-9      |
| Confined flow .....  | 4-2      |
| Confined Flow Problems .....   | 2-2      |
| Confined vs. unconfined .....  | 4-2      |
| constant head boundary conditions .....                                    | 2-3      |
| constant head boundary conditions .....                                    | 2-4      |
| Constant Head Boundary Conditions .....                                    | 2-2      |
| Cores of Dams .....  | 2-7      |
| <b>D</b>   |          |
| Datum .....  | 4-1      |
| deforming mesh .....   | 4-2      |
| Dupuit problem .....   | 2-4      |
| <b>E</b>   |          |
| Exit Face Boundary Conditions .....  | 2-2      |
| <b>F</b>   |          |
| Files .....  | 5-1      |
| Floating Point Error .....   | 5-2      |
| Flow Lines .....   | 4-5      |
| Flow Rate Boundary Conditions .....  | 2-2      |
| Flow to a Well .....   | 2-8      |
| Flux Boundary Conditions .....   | 2-2      |
| frontal function .....   | 4-3      |
| <b>G</b>   |          |
| Governing Equation .....   | 1-2      |
| grout curtains .....   | 2-5      |
| <b>H</b>   |          |
| Heterogeneous models .....   | 4-6      |
| <b>I</b>   |          |
| hydraulic conductivities .....   | 3-2      |
| hydraulic conductivity .....   | 2-7, 2-9 |
| <b>J</b>   |          |
| <b>K</b>   |          |
| <b>L</b>   |          |
| Laplace equation .....   | 1-2      |
| linear front .....   | 4-3      |
| <b>M</b>   |          |
| material ID .....  | 3-2      |
| Material Properties .....  | 4-5      |
| Maximum Iterations Reached .....   | 5-3      |
| Maximum Number of Nodes or Elements Exceeded<br>.....                      | 5-2      |
| memory requirements .....  | 3-3      |
| Mesh Construction .....  | 1-3      |
| Mesh Construction .....  | 3-2      |
| Mesh Construction Guidelines .....   | 3-1      |
| Mesh Density .....   | 3-4      |
| Mesh is Deformed when Modeling a Confined<br>Problem .....                 | 5-3      |
| Model Conceptualization .....  | 2-1      |
| Model Parameters .....   | 4-1      |
| Modeling Process .....   | 1-2      |
| <b>N</b>   |          |
| no flow boundary conditions .....  | 2-3      |
| No Flow Boundary Conditions .....  | 2-1      |
| nodal half band width .....  | 3-3      |
| <b>O</b>   |          |
| <b>P</b>   |          |
| phreatic surface .....   | 2-7      |
| Plan or Aerial View Models .....   | 2-9      |
| Plane flow .....   | 4-2      |
| Post-Processing .....  | 1-3      |
| Problem Unexpectedly Aborts, or Takes an<br>Unusual Amount of Time .....   | 5-3      |
| <b>Q</b>   |          |
| Quadratic elements .....   | 3-2      |
| quadrilaterals .....   | 3-2      |
| <b>R</b>   |          |
| relative conductivity .....  | 4-3      |

Renumbering ..... 3-2  
Running *SEEP2D* ..... 5-1, 5-2

**S**

Saturated/unsaturated flow ..... 4-2  
*SEEP2D* ..... 1-3  
sheet pile ..... 2-6  
Sheet piles ..... 2-5  
solution time ..... 3-3

**T**

Title ..... 4-1  
triangles ..... 3-2  
Trouble Shooting ..... 5-2  
Type of Flow ..... 4-2

**U**

Unable to compute flow lines ..... 5-2

unconfined aquifer ..... 2-8  
unconfined aquifers ..... 2-9  
Unconfined flow ..... 4-2  
Unconfined Flow Problems ..... 2-4  
Unconfined vs. confined ..... 4-2  
Unit Weight of Water ..... 4-2  
United States Army Engineer Waterways  
    Experiment Station ..... 1-1  
Unsaturated flow ..... 4-2

**V**

Van Genuchten model ..... 4-4

**W**

Waterways Experiment Station ..... 1-1  
well ..... 2-8