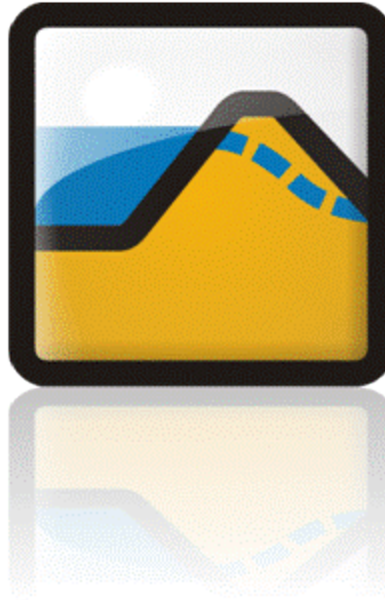




Seepage Modeling with SEEP/W

An Engineering Methodology

June 2015 Edition

GEO-SLOPE International Ltd.

Copyright © 2004-2015 by GEO-SLOPE International, Ltd.

All rights reserved. No part of this work may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying, recording, or by any information storage or retrieval system, without the prior written permission of GEO-SLOPE International, Ltd.

Trademarks: GEO-SLOPE, GeoStudio, SLOPE/W, SEEP/W, SIGMA/W, QUAKE/W, CTRAN/W, TEMP/W, AIR/W and VADOSE/W are trademarks or registered trademarks of GEO-SLOPE International Ltd. in Canada and other countries. Other trademarks are the property of their respective owners.

GEO-SLOPE International Ltd

1400, 633 – 6th Ave SW

Calgary, Alberta, Canada T2P 2Y5

E-mail: info@geo-slope.com

Web: <http://www.geo-slope.com>

Table of Contents

1	Introduction	1
2	Numerical Modeling: What, Why and How	3
2.1	Introduction	3
2.2	What is a numerical model?	3
2.3	Modeling in geotechnical engineering	5
2.4	Why model?	7
	Quantitative predictions	8
	Compare alternatives	9
	Identify governing parameters	10
	Discover & understand physical process - train our thinking.....	11
2.5	How to model	14
	Make a guess	14
	Simplify geometry	16
	Start simple.....	17
	Do numerical experiments.....	18
	Model only essential components	19
	Start with estimated material properties	20
	Interrogate the results.....	21
	Evaluate results in the context of expected results	21
	Remember the real world	21
2.6	How not to model	22
2.7	Closing remarks	23
3	Geometry and Meshing	25
3.1	Introduction	25
3.2	Geometry objects in GeoStudio	26
	Soil regions, points and lines.....	27
	Free points.....	29
	Free lines.....	29
	Interface elements on lines.....	31
	Circular openings.....	33
3.3	Mesh generation	34
	Structured mesh	35
	Unstructured quad and triangle mesh	35
	Unstructured triangular mesh	35

	Triangular grid regions.....	36
	Rectangular grid of quads	37
3.4	Surface layers	37
3.5	Joining regions	40
3.6	Meshing for transient analyses	41
3.7	Finite elements	42
3.8	Element fundamentals	43
	Element nodes.....	43
	Field variable distribution.....	43
	Element and mesh compatibility.....	44
	Numerical integration.....	46
	Secondary variables	47
3.9	General guidelines for meshing	47
	Number of elements	48
	Effect of drawing scale	48
	Mesh purpose.....	49
	Simplified geometry	50
4	Material Models and Properties.....	53
4.1	Soil behavior models	53
	Material models in SEEP/W	53
4.1	Soil water storage – water content function	54
	Factors affecting the volumetric water content.....	56
4.2	Storage function types and estimation methods	56
	Estimation method 1 (grain size - Modified Kovacs)	57
	Estimation method 2 (sample functions)	59
	Closed form option 1 (Fredlund and Xing, 1994).....	60
	Closed form option 2 (Van Genuchten, 1980).....	61
4.3	Soil material function measurement	62
	Direct measurement of water content function.....	62
4.4	Coefficient of volume compressibility	63
4.5	Hydraulic conductivity	64
4.6	Frozen ground hydraulic conductivity	66
4.7	Conductivity function estimation methods	68
	Method 1 (Fredlund et al, 1994)	68
	Method 2 (Green and Corey, 1971).....	69
	Method 3 (Van Genuchten, 1980)	70

4.8	Interface model parameters	71
4.9	Sensitivity of hydraulic results to material properties	72
	Changes to the air-entry value (AEV).....	72
	Changes to the saturated hydraulic conductivity	74
	Changes to the slope of the VWC function.....	76
	Changes to the residual volumetric water content.....	78
5	Boundary Conditions	80
5.1	Introduction	80
5.2	Fundamentals	80
5.3	Boundary condition locations	82
	Region face boundary conditions	83
5.4	Head boundary conditions	83
	Definition of total head.....	83
	Head boundary conditions on a dam.....	85
	Constant pressure conditions	87
	Far field head conditions	87
5.5	Specified boundary flows	88
5.6	Sources and sinks	91
5.7	Seepage faces	92
5.8	Free drainage (unit gradient)	94
5.9	Ground surface infiltration and evaporation	95
5.10	Boundary functions	97
	General.....	97
	Head versus time.....	97
	Head versus volume.....	99
	Nodal flux Q versus time	100
	Unit flow rate versus time	101
	Modifier function	102
6	Analysis Types	105
6.1	Steady state	105
	Boundary condition types in steady state	106
6.2	Transient	106
	Initial conditions	106
	Drawing the initial water table.....	108
	Activation values.....	108
	Spatial function for the initial conditions	109

	No initial condition	109
6.3	Time stepping - temporal integration	109
	Finite element temporal integration formulation	110
	Problems with time step sizes	110
	General rules for setting time steps	111
	Adaptive time stepping	111
6.4	Staged / multiple analyses	112
6.5	Axisymmetric	112
6.6	Plan view (confined aquifer only)	113
7	Functions in GeoStudio	115
7.1	Spline functions	115
	Slopes of spline functions	116
7.2	Linear functions	117
7.3	Step functions	117
7.4	Closed form curve fits for water content functions	118
7.5	Add-in functions	118
7.6	Spatial functions	118
8	Numerical Issues	121
8.1	Convergence	121
	Significant figures	122
	Minimum difference	122
8.2	Evaluating convergence	122
	Mesh view	122
	Graphs	123
	Commentary	125
8.3	Under-relaxation	125
8.4	Gauss integration order	125
8.5	Equation solvers (direct or parallel direct)	126
8.6	Time stepping	127
	Estimating a starting time step size	127
	Time steps too small	127
9	Flow nets, seepage forces, and exit gradients	129
9.1	Flow nets	129
	Equipotential lines	130
	Flow paths	130

	Flow channels.....	131
	Flow quantities.....	133
	Uplift pressures.....	134
	Limitations	134
9.2	Seepage forces	135
	Forces on a soil element	135
9.3	Exit gradients	136
	Gradients	136
	Critical gradients.....	137
	Geometry considerations.....	137
	Effective stress and soil strength.....	140
	Flow velocity	141
9.4	Concluding remarks	141
10	Visualization of Results	143
10.1	Transient versus steady state results	143
10.2	Node and element information	143
10.3	Graphing node and gauss Data	146
10.4	“None” values	148
10.5	Water table	148
10.6	Isolines	149
10.7	Projecting Gauss point values to nodes	149
10.8	Contours	150
10.9	Animation in GeoStudio	151
10.10	Velocity vectors and flow paths	151
	Calculating gradients and velocities	151
	Velocity vectors	152
	Flow paths	152
10.11	Flux sections	153
	Flux section theory	153
	Flux section application	155
11	Modeling Tips and Tricks.....	156
11.1	Introduction	156
11.2	Problem engineering units	157
11.3	Flux section location	157
11.4	Drain flux values	158
11.5	Unit flux versus total flux?	159

11.6	Flow above phreatic line	160
11.7	Pressure boundary with depth	161
11.8	Summing graphed data	162
12	Illustrative Examples.....	167
13	Theory	169
13.1	Darcy's law	169
13.2	Partial differential water flow equations	169
13.3	Finite element water flow equations	171
13.4	Temporal integration	172
13.5	Numerical integration	173
13.6	Hydraulic conductivity matrix	175
13.7	Mass matrix	176
13.8	Flux boundary vector	177
13.9	Density-dependent flow	180
14	Appendix A: Interpolating Functions.....	183
14.1	Coordinate systems	183
14.2	Interpolating functions	185
	Field variable model	185
	Interpolation function derivatives	186
	References	189
	Index	194

1 Introduction

The flow of water through soil is one of the fundamental processes in geotechnical and geo-environmental engineering. In fact, there would be little need for geotechnical engineering if water were not present in the soil. This is a nonsensical statement: if there were no water in the soil, there would be no way to sustain an ecosystem, no humans on earth and no need for geotechnical and geo-environmental engineering. However, the statement does highlight the importance of water in working with soil and rock.

Flow quantity is a key parameter in quantifying seepage losses from a reservoir or in identifying a potential water supply for domestic or industrial use. Pore-pressures associated with groundwater flow are of particular concern in geotechnical engineering. The pore-water pressure, whether positive or negative, is an integral component of the stress state within the soil and consequently has a direct bearing on the shear strength and volume change behavior of soil. Research in the last few decades has highlighted the importance of moisture flow dynamics in unsaturated surficial soils as it relates to the design of soil covers.

Historically, analyses of groundwater flow have focused on flow in saturated soils and flow problems were typically categorized as being confined or unconfined situations. Flow beneath a structure would be a confined flow problem, while flow through a homogeneous embankment would be an unconfined flow problem. Unconfined flow problems were often considered more difficult to analyze because the determination of the location of the phreatic surface (i.e., the transition from positive to negative pore water pressures) was integral to the analyses. The phreatic surface was considered an upper boundary and any flow that may have existed in the capillary zone above the phreatic line was ignored.

It is no longer acceptable to simply ignore the movement of water in unsaturated soils above the phreatic surface. Not only does it ignore an important component of moisture flow in soils, but it greatly limits the types of problems that can be analyzed. It is central to the analysis of problems involving infiltration and moisture redistribution in the vadose zone. Transient flow problems such as the advance of a wetting front within an earth structure after rapid filling are typical examples of situations in which it is impossible to simulate field behavior without correctly considering the physics of flow through unsaturated soils. Fortunately, it is no longer necessary to ignore the unsaturated zone. With the help of this document and the associated software, flow through unsaturated soils can be incorporated into numerical models so that almost any kind of seepage problem can be analyzed.

In general, all water flow is driven by energy gradients associated with the total head of water as represented by the components of pressure head (or pore water pressure) and elevation. The term *seepage* often is used to describe flow problems in which the dominant driving energy is gravity, such as a case in which seepage losses occur from a reservoir to a downstream exit point. In other situations such as consolidation, the primary driving energy may be associated with the creation of excess pore-water pressures as a result of external loading. However, both of these situations can all be described by a common set of mathematical equations describing the water movement. As a result, the formulation used to analyze seepage problems can also be used to analyze the dissipation of excess pore-water pressures resulting from changes in stress conditions. In the context of the discussions and examples in this document and in using the SEEP/W software, the term seepage is used to describe all movement of water through soil regardless of the creation or source of the driving energy or whether the flow is through saturated or unsaturated soils.

Simulating the flow of water through soil with a numerical model can be very complex. Natural soil deposits are generally highly heterogeneous and non-isotropic. In addition, boundary conditions often change with time and cannot always be defined with certainty at the beginning of an analysis. In some

cases the correct boundary conditions themselves are part of the outcome from the solution. Furthermore, when a soil becomes unsaturated, the coefficient of permeability or hydraulic conductivity becomes a function of the negative pore-water pressure in the soil. Since the pore-water pressure is related to the primary unknown (total head) and must be computed, iterative numerical techniques are required to compute the appropriate combination of pore-water pressure and the material property. This is referred to as a non-linear problem. These complexities make it necessary to use some form of numerical analysis to analyze seepage problems for all but the simplest cases.

While part of this document is about using SEEP/W to do seepage analyses, it is also about general numerical modeling techniques. Numerical modeling, like most things in life, is a skill that needs to be acquired. It is nearly impossible to pick up a tool like SEEP/W and immediately become an effective modeler. Effective numerical modeling requires some careful thought and planning and it requires a good understanding of the underlying fundamental theory and concepts. Steps in the analyses such as creating the finite element mesh and applying boundary conditions are not entirely intuitive at first. Time and practice are required to become comfortable with these aspects of numerical modeling.

A large portion of this book focuses on general guidelines for conducting effective analyses using a numerical model. Chapter 2, Numerical Modeling: What, Why and How, is devoted exclusively to discussions on this topic. The general principles discussed apply to all numerical modeling situations, but are used in the context of seepage analyses in this document.

Broadly speaking, there are three main parts to a finite element analysis. The first is creating the numerical domain, including the selection of an appropriate geometry and creating the discretized mesh. The second part requires the specification of material properties to the various sub-regions of the domain. The third is the specification of the appropriate boundary conditions. Separate chapters have been devoted to each of these three key components within this document.

The analysis of flow through saturated and unsaturated soils using numerical models is a highly non-linear problem that requires iterative techniques to obtain solutions. Numerical convergence is consequently a key issue. Also, the temporal integration scheme, which is required for a transient analysis, is affected by time step size relative to element size and material properties. These and other numerical considerations are discussed in Chapter 8, Numerical Issues.

Chapter 11, Modeling Tips and Tricks, should be consulted to see if there are simple techniques that can be used to improve your general modeling method. You will also gain more confidence and develop a deeper understanding of finite element methods, SEEP/W conventions and data results.

Chapter 12 has been dedicated to presenting examples in a brief, introductory format. The details of all examples along with the actual project files are available on a separate DVD or by download from our web site. You should scan over this chapter and see the many verification examples and case study examples to understand the capabilities of the software.

Chapter 13, Theory, is dedicated to theoretical issues associated with the finite element solution of the partial differential flow equation for saturated and unsaturated soils. Additional details regarding finite element interpolating functions are given in Appendix A, Interpolating Functions.

In general, this book is not a “how to use SEEP/W” manual. It is a book about how to model. It also describes how to engineer seepage problems using a powerful calculator; SEEP/W. Details of how to use the various program commands and features of SEEP/W are given in the online help inside the software.

2 Numerical Modeling: What, Why and How

2.1 Introduction

The unprecedented computing power now available has resulted in advanced software products for engineering and scientific analysis. The ready availability and ease-of-use of these products makes it possible to use powerful techniques such as a finite element analysis in engineering practice. These analytical methods have now moved from being research tools to application tools. This has opened a whole new world of numerical modeling.

Software tools such as SEEP/W do not inherently lead to good results. While the software is an extremely powerful calculator, obtaining useful and meaningful results from this useful tool depends on the guidance provided by the user. It is the users' understanding of the input and their ability to interpret the results that make it such a powerful tool. In summary, the software does not do the modeling, the user does the modeling. The software only provides the ability to do highly complex computations that are not otherwise humanly possible. In a similar manner, modern day spreadsheet software programs can be immensely powerful as well, but obtaining useful results from a spreadsheet depends on the user. It is the user's ability to guide the analysis process that makes it a powerful tool. The spreadsheet can do all the mathematics, but it is the user's ability to take advantage of the computing capability that leads to something meaningful and useful. The same is true with finite element analysis software such as SEEP/W.

Numerical modeling is a skill that is acquired with time and experience. Simply acquiring a software product does not immediately make a person a proficient modeler. Time and practice are required to understand the techniques involved and learn how to interpret the results.

Numerical modeling as a field of practice is relatively new in geotechnical engineering and, consequently, there is a lack of understanding about what numerical modeling is, how modeling should be approached and what to expect from it. A good understanding of these basic issues is fundamental to conducting effective modeling. Basic questions such as, What is the main objective of the analysis?, What is the main engineering question that needs to be answered? and, What is the anticipated result?, need to be decided before starting to use the software. Using the software is only part of the modeling exercise. The associated mental analysis is as important as clicking the buttons in the software.

This chapter discusses the “what”, “why” and “how” of the numerical modeling process and presents guidelines on the procedures that should be followed in good numerical modeling practice.

2.2 What is a numerical model?

A numerical model is a mathematical simulation of a real physical process. SEEP/W is a numerical model that can mathematically simulate the real physical process of water flowing through a particulate medium. Numerical modeling is purely mathematical and in this sense is very different than scaled physical modeling in the laboratory or full-scaled field modeling.

Rulon (1985) constructed a scale model of a soil slope with a less permeable layer embedded within the slope and then sprinkled water on the crest to simulate infiltration or precipitation. Instruments were inserted into the soil through the side walls to measure the pore-water pressures at various points. The results of her experiment are shown in Figure 2-1. Modeling Rulon's laboratory experiment with SEEP/W gives the results presented in Figure 2-2, which are almost identical to the original laboratory measurements. The positions of the equipotential lines are somewhat different, but the position of the water table is the same. In both cases there are two seepage exit areas on the slope, which is the main

important observation in this case. (Details of the SEEP/W analysis of this case are presented in Chapter 12, Illustrative Examples).

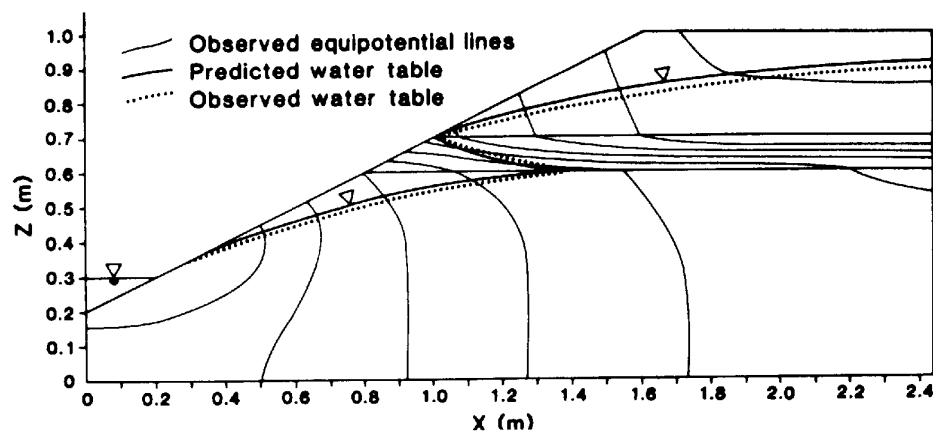


Figure 2-1 Rulon's laboratory scaled model results

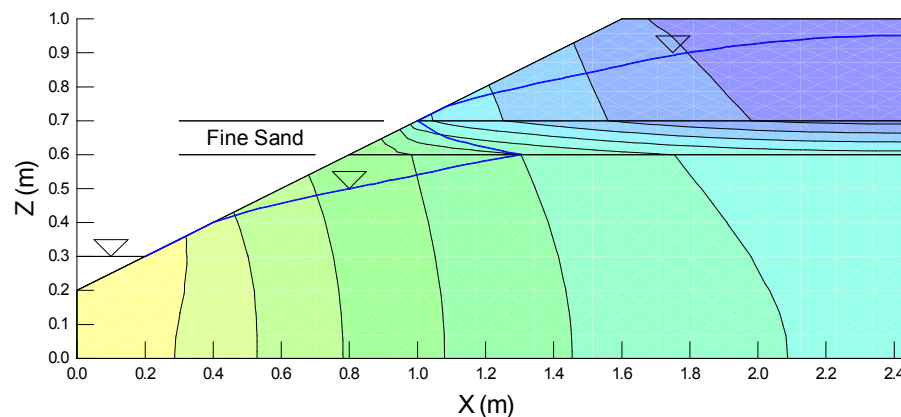


Figure 2-2 SEEP/W analysis of Rulon's laboratory model

The fact that mathematics can be used to simulate real physical processes is one of the great wonders of the universe. Perhaps physical processes follow mathematical rules, or mathematics has evolved to describe physical processes. Obviously, we do not know which came first, nor does it really matter. Regardless of how the relationship developed, the fact that we can use mathematics to simulate physical processes leads to developing a deeper understanding of physical processes. It may even allow for understanding or discovering previously unknown physical processes.

Numerical modeling has many advantages over physical modeling. The following are some of the more obvious advantages.

Numerical models can be set up very quickly relative to physical models. Physical models may take months to construct while a numerical model can be constructed in minutes, hours or days.

A physical model is usually limited to a narrow set of conditions. A numerical model can be used to investigate a wide variety of different scenarios.

Numerical models have no difficulty accounting for gravity. Gravity cannot be scaled, which is a limitation with laboratory modeling. A centrifuge is often required to overcome this limitation.

With numerical modeling, there is no danger of physical harm to personnel. Physical modeling sometimes involves heavy equipment and worker safety is consequently a concern.

Numerical modeling provides information and results at any location within the cross-section. Physical modeling only provides external visual responses and data at discrete instrumented points.

Numerical models can accommodate a wide variety of boundary conditions, whereas physical models are often limited in the types of boundary conditions possible.

It would be wrong to think that numerical models do not have limitations. Associated with seepage flow there may also be temperature changes, volume changes and perhaps chemical changes. Including all these processes in the same formulation is not possible, as the mathematics involved simply become too complex. In addition, it is not possible to mathematically describe a constitutive relationship, due to its complexity. Some of these difficulties can and will be overcome with greater and faster computer processing power. It is important to understand that numerical modeling products like SEEP/W will have limitations that are related to the current capability of hardware or integral to the formulation of the software, since it was developed to consider specific conditions. SEEP/W is formulated only for flow that follows Darcy's Law. Near the ground surface moisture may leave the ground as vapor. This component is not included in the SEEP/W formulation, like it is in another product called VADOSE/W. Consequently, SEEP/W has limitations when it comes to modeling moisture leaving the system at the ground surface. A real physical model would not have this type of limitation.

The important point to remember is that the mathematical formulations implemented in software like SEEP/W result in a very powerful and versatile means of simulating real physical processes.

“A mathematical model is a replica of some real-world object or system. It is an attempt to take our understanding of the process (conceptual model) and translate it into mathematical terms.” National Research Council Report (1990).

2.3 *Modeling in geotechnical engineering*

The role and significance of analysis and numerical modeling in geotechnical engineering has been vividly illustrated by Professor John Burland, Imperial College, London (UK). In 1987 Professor Burland presented what is known as the Nash Lecture. The title of the lecture was “The Teaching of Soil Mechanics – a Personal View”. In this lecture he advocated that geotechnical engineering consists of three fundamental components: the ground profile, the soil behavior and modeling. He represented these components as the apexes of a triangle, as illustrated in Figure 2-3. This has come to be known as the Burland triangle (Burland, 1987; Burland, 1996).

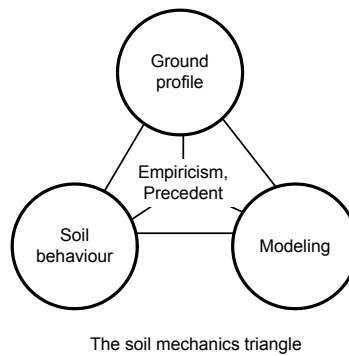


Figure 2-3 The Burland triangle (after Burland 1996)

The soil behavior component includes laboratory tests, *in situ* tests and field measurements. The ground profile component basically involves site characterization: defining and describing the site conditions. Modeling may be conceptual, analytical or physical.

Of great significance is that, in Burland's view, all three components need to be tied together by empiricism and precedent. This is the part inside the triangle.

The Burland triangle idea has been widely discussed and referred to by others since it was first presented. An article on this topic was presented in an issue of Ground Engineering (Anon. 1999). Morgenstern (2000) discussed this at some length in his keynote address titled "Common Ground" at the GeoEng2000 Conference in Melbourne Australia in 2000. With all the discussion, the triangle has been enhanced and broadened somewhat, as shown in Figure 2-4.

One important additional feature has been to consider all the connecting arrows between the components as pointing in both directions. This simple addition highlights the fact that each part is distinct yet related to all the other parts.

The Burland triangle vividly illustrates the importance of modeling in geotechnical engineering. Characterizing the field conditions and making measurements of behavior is not sufficient. Ultimately, it is necessary to do some analysis of the field information and soil properties to complete the triangle.

As Burland pointed out, modeling may be conceptual, analytical or physical. However, with the computing power and software tools now available, modeling often refers to numerical modeling. Accepting that modeling primarily refers to numerical modeling, the Burland triangle shows the importance that numerical modeling has in geotechnical engineering.

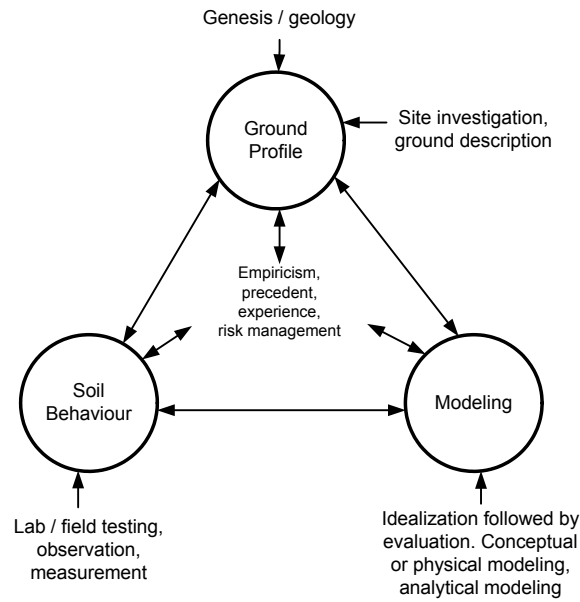


Figure 2-4 The enhanced Burland triangle (after Anon. 1999)

Making measurements and characterizing site conditions is often time consuming and expensive. This is also true with modeling, if done correctly. A common assumption is that the numerical modeling component is only a small component that should be undertaken at the end of a project, and that it can be done simply and quickly. This is somewhat erroneous. Good numerical modeling, as we will see later in the section in more detail, takes time and requires careful planning in the same manner that it takes time and planning to collect field measurements and adequately characterize site conditions.

Considering the importance of modeling that the Burland triangle suggests for geotechnical engineering, it is prudent that we do the modeling carefully and with a complete understanding of the modeling processes. This is particularly true with numerical modeling. The purpose of this book is to assist with this aspect of geotechnical engineering.

2.4 Why model?

The first reaction to the question, “why model?” seems rather obvious. The objective is to analyze the problem. Upon more thought, the answer becomes increasingly complex. Without a clear understanding of the reason for modeling or identifying what the modeling objectives are, numerical modeling can lead to a frustrating experience and uncertain results. As we will see in more detail in the next section, it is wrong to set up the model, calculate a solution and then try to decide what the results mean. It is important to decide at the outset the reason for doing the modeling. What is the main objective and what is the question that needs to be answered?

The following points are some of the main reasons for modeling, from a broad, high level perspective. We model to:

- make quantitative predictions,
- compare alternatives,
- identify governing parameters, and
- understand processes and train our thinking.

Quantitative predictions

Most engineers, when asked why they want to do some modeling, will say that they want to make a prediction. They want to predict the seepage quantity, for example, or the time for a contaminant to travel from the source to a seepage discharge point, or the time required from first filling a reservoir until steady-state seepage conditions have been established in the embankment dam. The desire is to say something about future behavior or performance.

Making quantitative predictions is a legitimate reason for doing modeling. Unfortunately, it is also the most difficult part of modeling, since quantitative values are often directly related to the material properties. The quantity of seepage, for example, is in large part controlled by the hydraulic conductivity and, as a result, changing the hydraulic conductivity by an order of magnitude will usually change the computed seepage quantity by an order of magnitude. The accuracy of quantitative prediction is directly related to the accuracy of the hydraulic conductivity specified. Unfortunately, for a heterogeneous profile, there is not a large amount of confidence about how precisely the hydraulic conductivity can be specified. Sometimes defining the hydraulic conductivity within an order of magnitude is considered reasonable. The confidence you have defining the hydraulic conductivity depends on many factors, but the general difficulty of defining this soil parameter highlights the difficulty of undertaking modeling to make quantitative predictions.

Carter et al. (2000) presented the results of a competition conducted by the German Society for Geotechnics. Packages of information were distributed to consulting engineers and university research groups. The participants were asked to predict the lateral deflection of a tie-back shoring wall for a deep excavation in Berlin. During construction, the actual deflection was measured with inclinometers. Later the predictions were compared with the actual measurements. Figure 2-5 shows the best eleven submitted predictions. Other predictions were submitted, but were considered unreasonable and consequently not included in the summary.

There are two heavy dark lines superimposed on Figure 2-5. The dashed line on the right represents the inclinometer measurements uncorrected for any possible base movement. It is likely the base of the inclinometer moved together with the base of the wall. Assuming the inclinometer base moved about 10 mm, the solid heavy line in Figure 2-5 has been shifted to reflect the inclinometer base movement.

At first glance one might quickly conclude that the agreement between prediction and actual lateral movement is very poor, especially since there appears to be a wide scatter in the predictions. This exercise might be considered as an example of our inability to make accurate quantitative predictions.

However, a closer look at the results reveals a picture that is not so bleak. The depth of the excavation is 32 m. The maximum predicted lateral movement is just over 50 mm or 5 cm. This is an extremely small amount of movement over the length of the wall – certainly not big enough to be visually noticeable. Furthermore, the actual measurements, when corrected for base movement fall more or less in the middle of the predictions. Most important to consider are the trends presented by many of the predicted results. Many of them predict a deflected shape similar to the actual measurements. In other words, the predictions simulated the correct relative response of the wall.

Consequently, we can argue that our ability to make accurate predictions is poor, but we can also argue that the predictions are amazingly good. The predictions fall on either side of the measurements and the deflected shapes are correct. In the end, the modeling provided a correct understanding of the wall behavior, which is more than enough justification for doing the modeling, and may be the greatest benefit of numerical modeling, as we will see in more detail later.

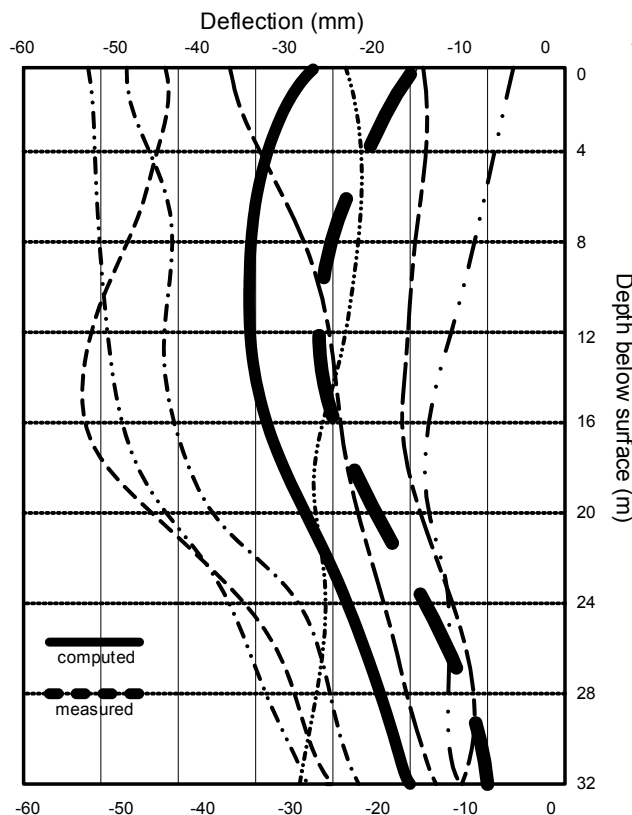


Figure 2-5 Comparison of predicted and measured lateral movements of a shoring wall (after Carter et al, 2000)

Numerical modeling is sometimes dismissed as being useless due to the difficulty with defining material properties. There are, however, other reasons for doing numerical modeling. If some of the other objectives of numerical modeling are completed first, then quantitative predictions often have more value and meaning. Once the physics and mechanisms are completely understood, quantitative predictions can be made with a great deal more confidence and are not nearly as useless as first thought, regardless of our inability to accurately define material properties.

Compare alternatives

Numerical modeling is useful for comparing alternatives. Keeping everything else the same and changing a single parameter makes it a powerful tool to evaluate the significance of individual parameters. For modeling alternatives and conducting sensitivity studies it is not all that important to accurately define some material properties. All that is of interest is the change between simulations.

Consider the example of a cut-off wall beneath a structure. With SEEP/W it is easy to examine the benefits obtained by changing the length of the cut-off. Consider two cases with different cut-off depths to assess the difference in uplift pressures underneath the structure. Figure 2-6 shows the analysis when the cutoff is 10 feet deep. The pressure drop and uplift pressure along the base are shown in the left graph in Figure 2-7. The drop across the cutoff is from 24 to 18 feet of pressure head. The results for a 20-foot cutoff are shown in Figure 2-7 on the right side. Now the drop across the cutoff is from 24 to about 15 feet of pressure head. The uplift pressures at the downstream toe are about the same.

The actual computed values are not of significance in the context of this discussion. It is an example of how a model such as SEEP/W can be used to quickly compare alternatives. Secondly, this type of

analysis can be done with a rough estimate of the conductivity, since in this case the pressure distributions will be unaffected by the conductivity assumed. There would be no value in carefully defining the conductivity to compare the base pressure distributions.

We can also look at the change in flow quantities. The absolute flow quantity may not be all that accurate, but the change resulting from various cut-off depths will be of value. The total flux is $6.26 \times 10^{-3} \text{ ft}^3/\text{s}$ for the 10-foot cutoff and $5.30 \times 10^{-3} \text{ ft}^3/\text{s}$ for the 20-foot cutoff, only about a 15 percent difference.

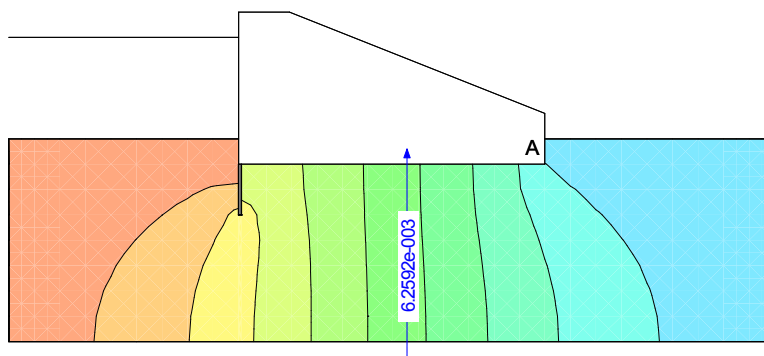


Figure 2-6 Seepage analysis with a cutoff

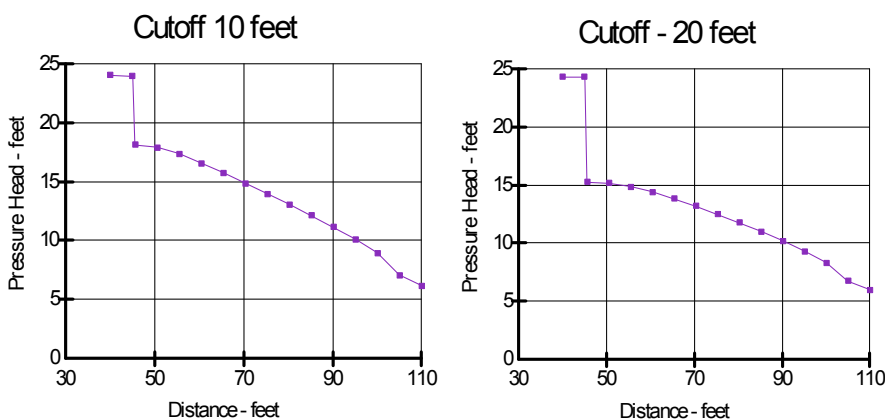


Figure 2-7 Uplift pressure distributions along base of structure

Identify governing parameters

Numerical models are useful for identifying critical parameters in a design. Consider the performance of a soil cover over waste material. What is the most important parameter governing the behavior of the cover? Is it the precipitation, the wind speed, the net solar radiation, plant type, root depth or soil type? Running a series of VADOSE/W simulations, keeping all variables constant except for one makes it possible to identify the governing parameter. The results can be presented as a tornado plot such as shown in Figure 2-8.

Once the key issues have been identified, further modeling to refine a design can concentrate on the main issues. If, for example, the vegetative growth is the main issue then efforts can be concentrated on what needs to be done to foster the plant growth.

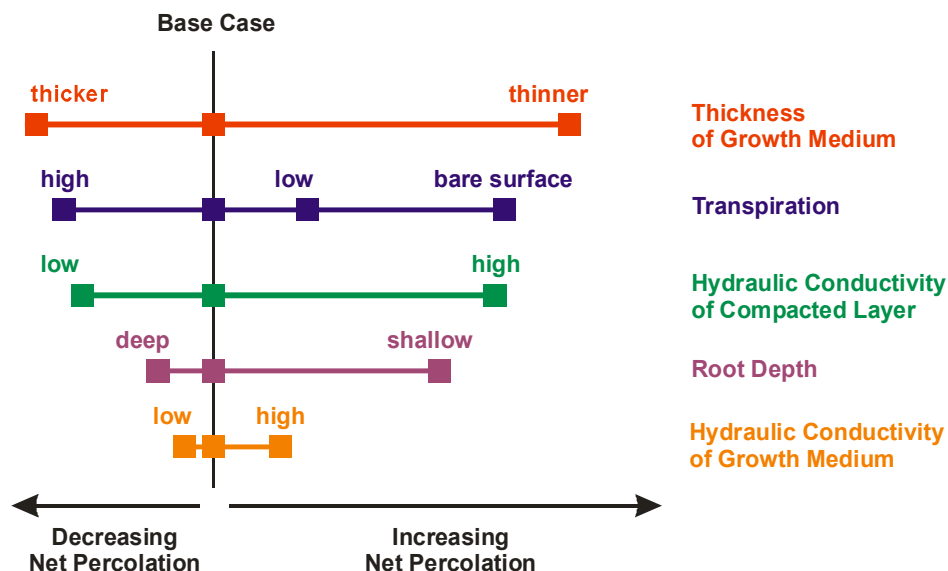


Figure 2-8 Example of a tornado plot (O'Kane, 2004)

Discover & understand physical process - train our thinking

One of the most powerful aspects of numerical modeling is that it can help us to understand physical processes in that it helps to train our thinking. A numerical model can either confirm our thinking or help us to adjust our thinking if necessary.

To illustrate this aspect of numerical modeling, consider the case of a multilayered earth cover system such as the two possible cases shown in Figure 2-9. The purpose of the cover is to reduce the infiltration into the underlying waste material. The intention is to use the earth cover layers to channel any infiltration downslope into a collection system. It is known that both a fine and a coarse soil are required to achieve this. The question is, should the coarse soil lie on top of the fine soil or should the fine soil overlay the coarse soil? Intuitively it would seem that the coarse material should be on top; after all, it has the higher conductivity. Modeling this situation with SEEP/W, which handles unsaturated flow, can answer this question and verify if our thinking is correct.

For unsaturated flow, it is necessary to define a *hydraulic conductivity function*: a function that describes how the hydraulic conductivity varies with changes in suction (negative pore-water pressure = suction). Chapter 4, Material Properties, describes in detail the nature of the hydraulic conductivity (or permeability) functions. For this example, relative conductivity functions such as those presented in Figure 2-10 are sufficient. At low suctions (i.e., near saturation), the coarse material has a higher hydraulic conductivity than the fine material, which is intuitive. At high suctions, the coarse material has the lower conductivity, which often appears counterintuitive. For a full explanation of this relationship, refer to Chapter 4, Materials Properties. For this example, accept that at high suctions the coarse material is less conductive than the fine material.

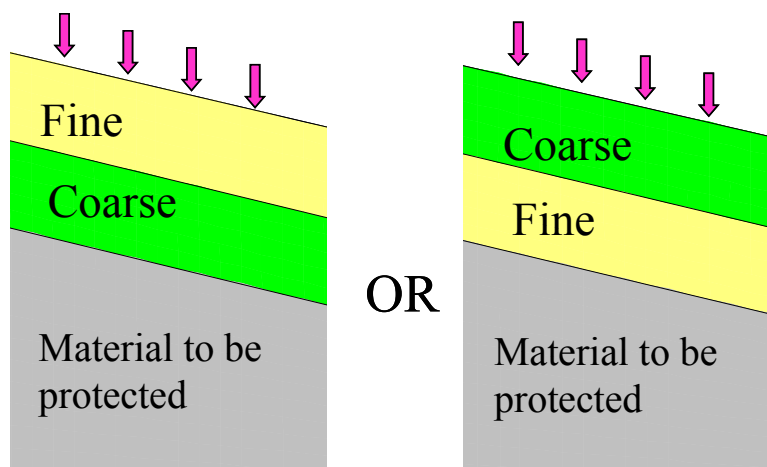


Figure 2-9 Two possible earth cover configurations

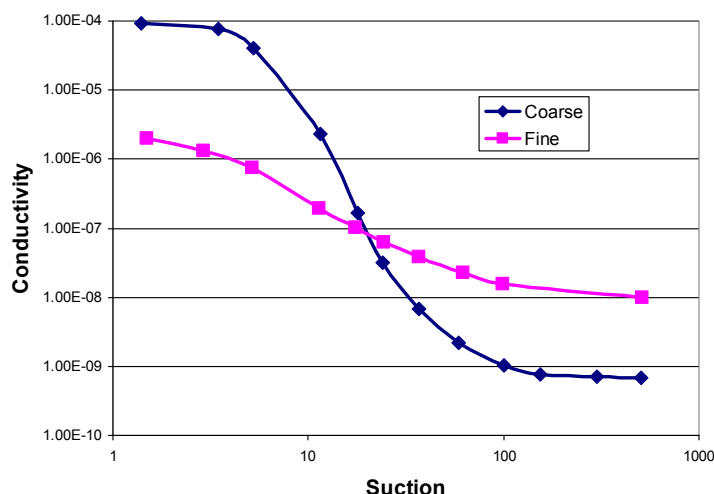


Figure 2-10 Hydraulic conductivity functions

After conducting various analyses and trial runs with varying rates of surface infiltration, it becomes evident that the behavior of the cover system is dependent on the infiltration rate. At low infiltration rates, the effect of placing the fine material over the coarse material results in infiltration being drained laterally through the fine layer, as shown in Figure 2-11. This accomplishes the design objective of the cover. If the precipitation rate becomes fairly intensive, then the infiltration drops through the fine material and drains laterally within the lower coarse material as shown in Figure 2-12. The design of fine soil over coarse soil may work, but only in arid environments. The occasional cloud burst may result in significant water infiltrating into the underlying coarse material, which may result in increased seepage into the waste. This may be a tolerable situation for short periods of time. If most of the time precipitation is modest, the infiltration will be drained laterally through the upper fine layer into a collection system.

So, for an arid site the best solution is to place the fine soil on top of the coarse soil. This is contrary to what one might expect at first. The first reaction may be that something is wrong with the software, but it may be that our understanding of the process and our general thinking is flawed.

A closer examination of the conductivity functions provides a logical explanation. The software is correct and provides the correct response given the input parameters. Consider the functions in Figure 2-13. When the infiltration rate is large, the negative water pressures or suctions will be small. As a result, the

conductivity of the coarse material is higher than the finer material. If the infiltration rates become small, the suctions will increase (water pressure becomes more negative) and the unsaturated conductivity of the finer material becomes higher than the coarse material. Consequently, under low infiltration rates it is easier for the water to flow through the fine, upper layer soil than through the lower more coarse soil.

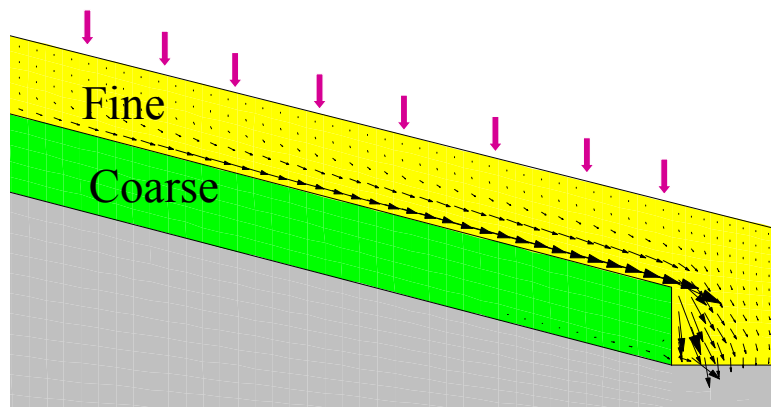


Figure 2-11 Flow diversion under low infiltration

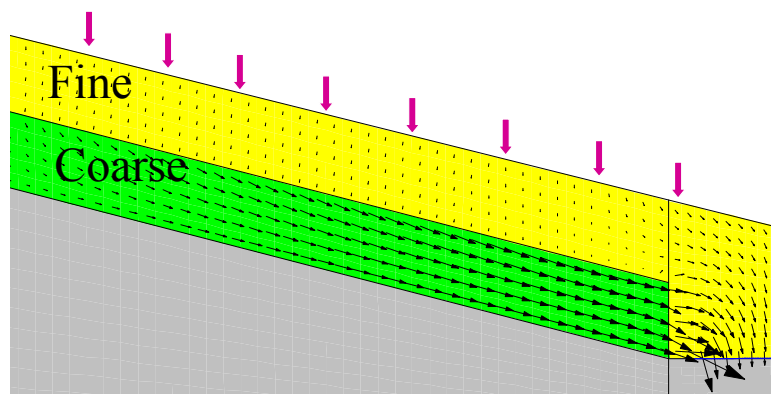


Figure 2-12 Flow diversion under high infiltration

This type of analysis is a good example where the ability to utilize a numerical model greatly assists our understanding of the physical process. The key is to think in terms of unsaturated conductivity as opposed to saturated conductivities.

Numerical modeling can be crucial in leading us to the discovery and understanding of real physical processes. In the end the model either has to conform to our mental image and understanding or our understanding has to be adjusted.

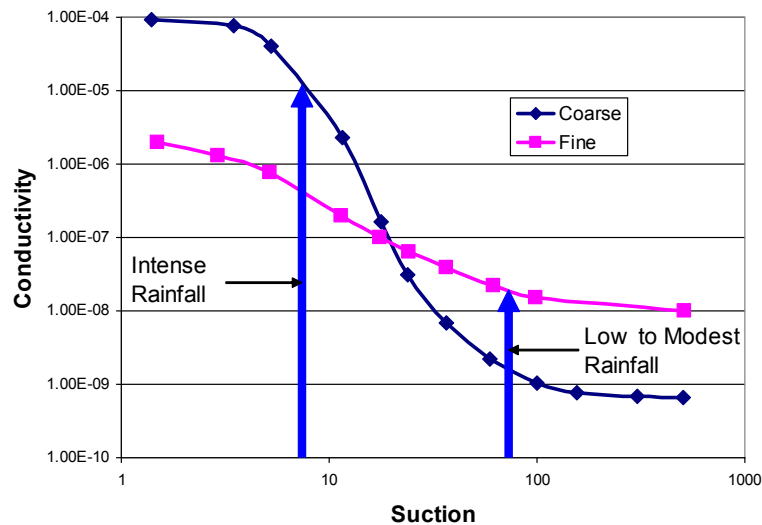


Figure 2-13 Conductivities under low and intense infiltration

This is a critical lesson in modeling and the use of numerical models in particular. The key advantage of modeling, and in particular the use of computer modeling tools, is the capability it has to enhance engineering judgment, not the ability to enhance our predictive capabilities. While it is true that sophisticated computer tools greatly elevated our predictive capabilities relative to hand calculations, graphical techniques, and closed-form analytical solutions, still, prediction is not the most important advantage these modern tools provide. Numerical modeling is primarily about ‘process’ - not about prediction.

“The attraction of ... modeling is that it combines the subtlety of human judgment with the power of the digital computer.” Anderson and Woessner (1992).

2.5 How to model

Numerical modeling involves more than just acquiring a software product. Running and using the software is an essential ingredient, but it is a small part of numerical modeling. This section talks about important concepts in numerical modeling and highlights important components in good modeling practice.

Make a guess

Generally, careful planning is involved when undertaking a site characterization or making measurements of observed behavior. The same careful planning is required for modeling. It is inappropriate to acquire a software product, input some parameters, obtain some results, and then decide what to do with the results or struggle to decide what the results mean. This approach usually leads to an unhappy experience and is often a meaningless exercise.

Good modeling practice starts with some planning. If at all possible, you should form a mental picture of what you think the results will look like. Stated another way, we should make a rough guess at the solution before starting to use the software. Figure 2-14 shows a very quick hand sketch of a flow net. It is very rough, but it gives us an idea of what the solution should look like.

From the rough sketch of a flow net, we can also get an estimate of the flow quantity. The amount of flow can be approximated by the ratio of flow channels to equipotential drops multiplied by the conductivity and the total head drop. For the sketch in Figure 2-14 the number of flow channels is 3, the number of equipotential drops is 9 and the total head drop is 5 m. Assume a hydraulic conductivity of $K = 0.1$ m/day. A rough estimate of the flow quantity then is $(5 \times 0.1 \times 3) / 9$, which is between 0.1 and 0.2 m^3/day . The SEEP/W computed flow is 0.1427 m^3/day and the equipotential lines are as shown in Figure 2-15.

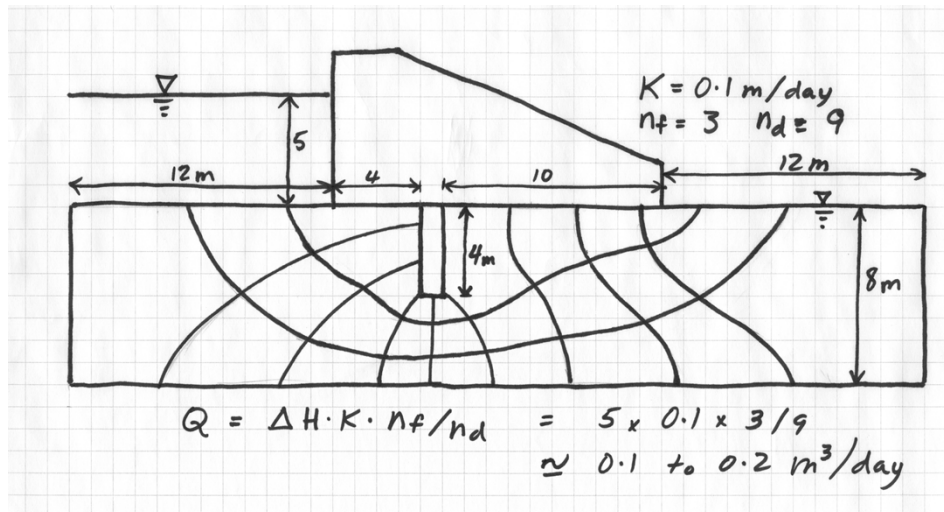


Figure 2-14 Hand sketch of flow net for cutoff below dam

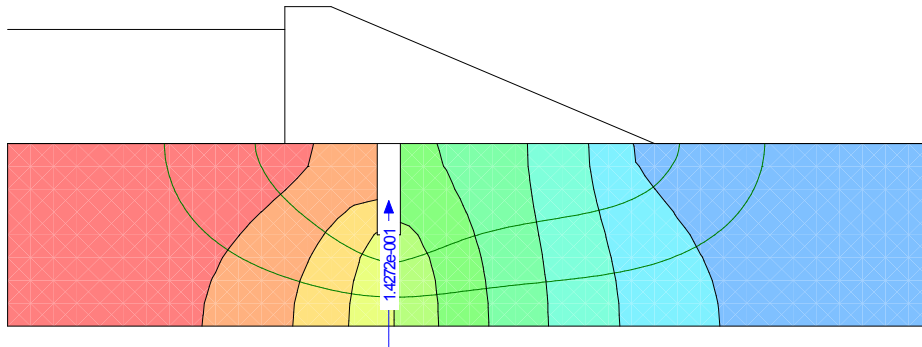


Figure 2-15 SEEP/W results compared to hand sketch estimate

The rough flow net together with the estimated flow quantity can now be used to judge the SEEP/W results. If there is no resemblance between what is expected and what is computed with SEEP/W then either the preliminary mental picture of the situation was not right or something has been inappropriately specified in the numerical model. Perhaps the boundary conditions are not correct or the material properties specified are different than intended. The difference ultimately needs to be resolved in order for you to have any confidence in your modeling. If you had never made a preliminary guess at the solution then it would be very difficult to judge the validity the numerical modeling results.

Another extremely important part of modeling is to clearly define at the outset, the primary question to be answered by the modeling process. Is the main question the pore-water pressure distribution or is the quantity of flow. If your main objective is to determine the pressure distribution, there is no need to spend a lot of time on establishing the hydraulic conductivity – any reasonable estimate of conductivity is adequate. If on the other hand your main objective is to estimate flow quantities, then a greater effort is needed in determining the conductivity.

Sometimes modelers say “I have no idea what the solution should look like - that is why I am doing the modeling”. The question then arises, why can you not form a mental picture of what the solution should resemble? Maybe it is a lack of understanding of the fundamental processes or physics, maybe it is a lack of experience, or maybe the system is too complex. A lack of understanding of the fundamentals can possibly be overcome by discussing the problem with more experienced engineers or scientists, or by conducting a study of published literature. If the system is too complex to make a preliminary estimate then it is good practice to simplify the problem so you can make a guess and then add complexity in stages so that at each modeling interval you can understand the significance of the increased complexity. If you were dealing with a very heterogenic system, you could start by defining a homogenous cross-section, obtaining a reasonable solution and then adding heterogeneity in stages. This approach is discussed in further detail in a subsequent section.

If you cannot form a mental picture of what the solution should look like prior to using the software, then you may need to discover or learn about a new physical process as discussed in the previous section.

Effective numerical modeling starts with making a guess of what the solution should look like.

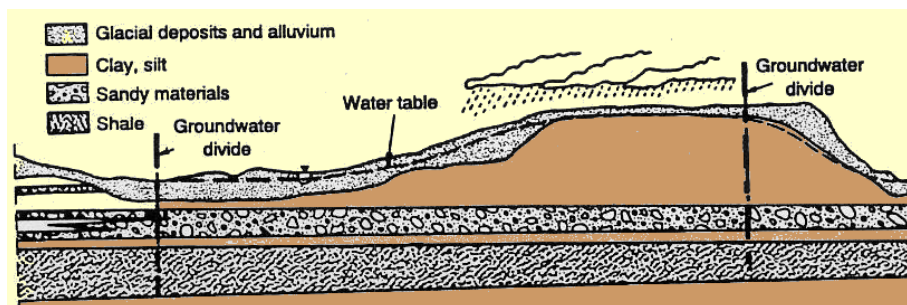
Other prominent engineers support this concept. Carter (2000) in his keynote address at the GeoEng2000 Conference in Melbourne, Australia, when talking about rules for modeling, stated verbally that modeling should “**start with an estimate.**” Prof. John Burland made a presentation at the same conference on his work with righting the Leaning Tower of Pisa. Part of the presentation was on the modeling that was done to evaluate alternatives and while talking about modeling he too stressed the need to “start with a guess”.

Simplify geometry

Numerical models need to be a simplified abstraction of the actual field conditions. In the field the stratigraphy may be fairly complex and boundaries may be irregular. In a numerical model the boundaries need to become straight lines and the stratigraphy needs to be simplified so that it is possible to obtain an understandable solution. Remember, it is a “model”, not the actual conditions. Generally, a numerical model cannot and should not include all the details that exist in the field. If attempts are made at including all the minute details, the model can become so complex that it is difficult and sometimes even impossible to interpret or even obtain results.

Figure 2-16 shows a stratigraphic cross section (National Research Council Report 1990). A suitable numerical model for simulating the flow regime between the groundwater divides is something like the one shown in Figure 2-17. The stratigraphic boundaries are considerably simplified for the finite element analysis.

As a general rule, a model should be designed to answer specific questions. You need to constantly ask yourself while designing a model, if this feature will significantly affects the results. If you have doubts, you should not include it in the model, at least not in the early stages of analysis. Always start with the simplest model.



**Figure 2-16 Example of a stratigraphic cross section
(from National Research Report 1990)**

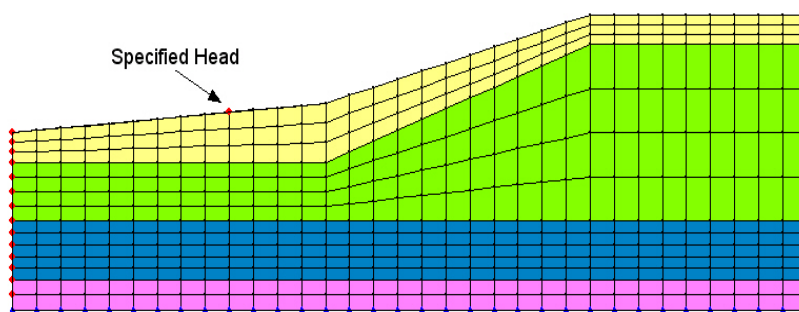


Figure 2-17 Finite element model of stratigraphic section

The tendency of novice modelers is to make the geometry too complex. The thinking is that everything needs to be included to get the best answer possible. In numerical modeling this is not always true. Increased complexity does not always lead to a better and more accurate solution. Geometric details can, for example, even create numerical difficulties that can mask the real solution.

Start simple

One of the most common mistakes in numerical modeling is to start with a model that is too complex. When a model is too complex, it is very difficult to judge and interpret the results. Often the result may look totally unreasonable. Then the next question asked is - what is causing the problem? Is it the geometry, is it the material properties, is it the boundary conditions, or is it the time step size or something else? The only way to resolve the issue is to make the model simpler and simpler until the difficulty can be isolated. This happens on almost all projects. It is much more efficient to start simple and build complexity into the model in stages, than to start complex, then take the model apart and have to rebuild it back up again.

A good start may be to take a homogeneous section and then add geometric complexity in stages. For the homogeneous section it is likely easier to judge the validity of the results. This allows you to gain confidence in the boundary conditions and material properties specified. Once you have reached a point where the results make sense, you can add different materials and increase the complexity of your geometry.

Another approach may be to start with a steady-state analysis even though you are ultimately interested in a transient process. A steady-state analysis gives you an idea as to where the transient analysis should end up: to define the end point. Using this approach you can then answer the question of how does the process migrate with time until a steady-state system has been achieved.

It is unrealistic to dump all your information into a numerical model at the start of an analysis project and magically obtain beautiful, logical and reasonable solutions. It is vitally important to not start with this expectation. You will likely have a very unhappy modeling experience if you follow this approach.

Do numerical experiments

Interpreting the results of numerical models sometimes requires doing numerical experiments. This is particularly true if you are uncertain as to whether the results are reasonable. This approach also helps with understanding and learning how a particular feature operates. The idea is to set up a simple problem for which you can create a hand calculated solution.

Consider the following example. You are uncertain about the results from a flux section or the meaning of a computed boundary flux. To help satisfy this lack of understanding, you could do a numerical experiment on a simple 1D case as shown in Figure 2-18. The total head difference is 1 m and the conductivity is 1 m/day. The gradient under steady state conditions is the head difference divided by the length, making the gradient 0.1. The resulting total flow through the system is the cross sectional area times the gradient which should be 0.3 m³/day. The flux section that goes through the entire section confirms this result. There are flux sections through Elements 16 and 18. The flow through each element is 0.1 m³/day, which is correct since each element represents one-third of the area.

Another way to check the computed results is to look at the node information. When a head is specified, SEEP/W computes the corresponding nodal flux. In SEEP/W these are referred to as boundary flux values. The computed boundary nodal flux for the same experiment shown in Figure 2-18 on the left at the top and bottom nodes is 0.05. For the two intermediate nodes, the nodal boundary flux is 0.1 per node. The total is 0.3, the same as computed by the flux section. Also, the quantities are positive, indicating flow into the system. The nodal boundary values on the right are the same as on the left, but negative. The negative sign means flow out of the system.

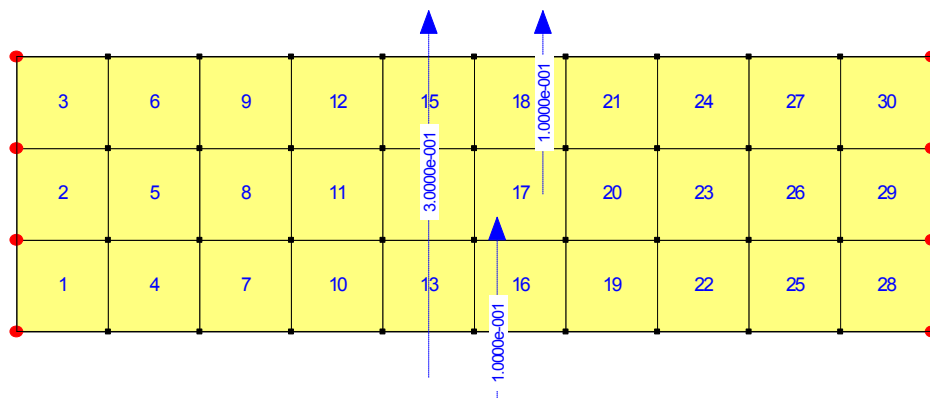


Figure 2-18 Horizontal flow through three element section

A simple numerical experiment takes only minutes to set up and run, but can be invaluable in confirming to you how the software works and in helping you interpret the results. There are many benefits: the most obvious is that it demonstrates the software is functioning properly. You can also see the difference between a flux section that goes through the entire problem versus a flux section that goes through a single element. You can see how the boundary nodal fluxes are related to the flux sections. It verifies for you the meaning of the sign on the boundary nodal fluxes. Fully understanding and comprehending the results of a simple example like this greatly helps increase your confidence in the interpretation of results from more complex problems.

Conducting simple numerical experiments is a useful exercise for both novice and experienced modelers. For novice modelers it is an effective way to understand fundamental principles, learn how the software functions, and gain confidence in interpreting results. For the experienced modeler it is an effective means of refreshing and confirming ideas. It is sometimes faster and more effective than trying to find appropriate documentation and then having to rely on the documentation. At the very least it may enhance and clarify the intent of the documentation.

Model only essential components

One of the powerful and attractive features of numerical modeling is the ability to simplify the geometry and not to have to include the entire physical structure in the model. A very common problem is the seepage flow under a concrete structure with a cut-off as shown in Figure 2-19. To analyze the seepage through the foundation it is not necessary to include the dam itself or the cut-off as these features are constructed of concrete and assumed impermeable.

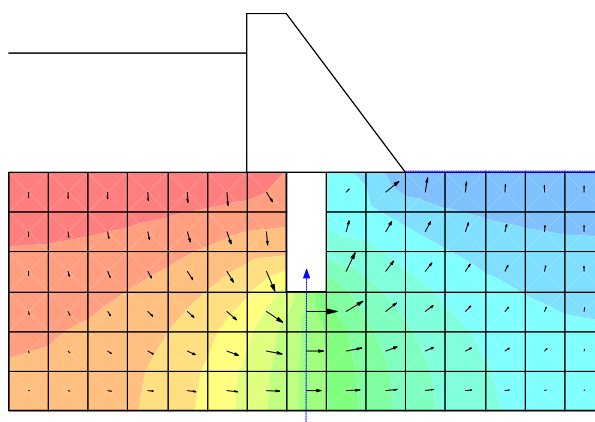


Figure 2-19 Simple flow beneath a cutoff

Another common example is the downstream toe-drain or horizontal under drain in an embankment (Figure 2-20). The drain is so permeable relative to the embankment material that the drain does not contribute to the dissipation of the head (potential energy) loss through the structure. Physically, the drain needs to exist in the embankment, but it does not need to be present in a numerical model. If the drain becomes clogged with fines so that it begins to impede the seepage flow, then the situation is different and the drain would need to be included in the numerical model. With any material, the need to include it in the analysis should be decided in the context of whether it contributes to the head loss.

Another example is the downstream shell of a zoned dam as illustrated in Figure 2-21. Often the core is constructed of fine-grained soil while the shells are highly permeable coarse granular material. If there is a significant difference between core and shell conductivities then seepage that flows through the core will drip along the downstream side of the core (usually in granular transition zones) down to an under drain. If this is the case, the downstream shell does not need to be included in the seepage analysis, since the shell is not physically involved in the dissipation of the head loss. Once again the shell needs to exist physically, but does not need to be included in the numerical seepage model.

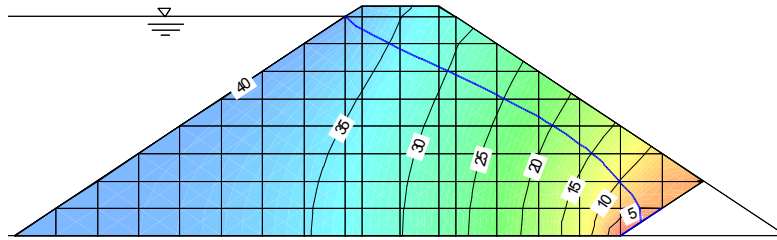


Figure 2-20 Flow through a dam with coarse toe drain

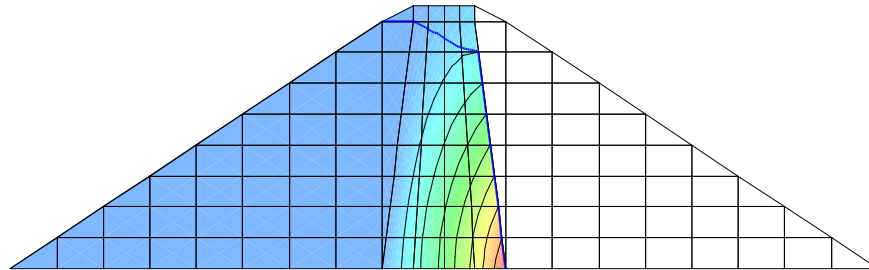


Figure 2-21 Head loss through dam core with downstream shell

Including unnecessary features and trying to model adjacent materials with extreme contrasts in material properties create numerical difficulties. The conductivity difference between the core and shell of a dam may be many, many orders of magnitude. The situation may be further complicated if unsaturated flow is present and the conductivity function is very steep, making the solution highly non-linear. In this type of situation it can be extremely difficult if not impossible to obtain a good solution with the current technology.

The numerical difficulties can be eased by eliminating non-essential segments from the numerical model. If the primary interest is the seepage through the core, then why include the downstream shell and complicate the analysis? Omitting non-essential features from the analysis is a very useful technique, particularly during the early stages of an analysis. During the early stages, you are simply trying to gain an understanding of the flow regime and trying to decide what is important and what is not important.

While deliberately leaving components out of the analysis may at first seem like a rather strange concept, it is a very important concept to accept if you want to be an effective numerical modeler.

Start with estimated material properties

In the early stages of a numerical modeling project it is often good practice to start with estimates of material properties. Simple estimates of material properties and simple property functions are more than adequate for gaining an understanding of the flow regime for checking that the model has been set up properly or to verify that the boundary conditions have been properly defined. Estimated properties are usually more than adequate for determining the importance of the various properties for the situation being modeled.

The temptation exists when you have laboratory data in hand that the data needs to be used in its entirety and cannot be manipulated in any way. There seems to be an inflexible view of laboratory data which can sometimes create difficulties when using the data in a numerical model. A common statement is; "I measured it in the lab and I have full confidence in my numbers". There can be a large reality gap that exists between laboratory determined results and actual insitu soil behavior. Some of the limitations arise because of how the material was collected, how it was sampled and ultimately quantified in the lab. Was

the sample collected by the shovelful, by collecting cuttings or by utilizing a core sampler? What was the size and number of samples collected and can they be considered representative of the entire profile? Was the sample oven-dried, sieved and then slurried prior to the test being performed? Were the large particles removed so the sample could be trimmed into the measuring device? Some of these common laboratory techniques can result in unrealistic property functions. Perhaps the amount of data collected in the laboratory is more than is actually required in the model. Because money has been spent collecting and measuring the data, it makes modelers reticent to experiment with making changes to the data to see what effect it has on the analysis.

It is good modeling practice to first obtain understandable and reasonable solutions using estimate material properties and then later refine the analysis once you know what the critical properties are going to be. It can even be more cost effective to determine ahead of time what material properties control the analysis and decide where it is appropriate to spend money obtaining laboratory data.

Interrogate the results

Powerful numerical models such as SEEP/W need very careful guidance from the user. It is easy to inadvertently and unintentionally specify inappropriate boundary conditions or incorrect material properties. Consequently, it is vitally important to conduct spot checks on the results to ensure the constraints and material properties are consistent with what you intended to define and the results make sense. It is important to check, for example that the boundary condition that appears in the results is the same as what you thought was specified defining the model. Is the intended property function being applied to the correct soil? Or, are the initial conditions as you assumed?

SEEP/W has many tools to inspect or interrogate the results. You can view node or element details and there are a wide range of parameters that can be graphed for the purpose of spot checking the results.

Inspecting and spot checking your results is an important and vital component in numerical modeling. It greatly helps to increase your confidence in a solution that is understandable and definable.

Evaluate results in the context of expected results

The fundamental question that should be asked during modeling is; “Do the results conform to the initial mental picture?” If they do not, then your mental picture needs to be fixed, there is something wrong with the model or both the model and your concept of the problem need to be adjusted until they agree. The numerical modeling process needs to be repeated over and over until the solution makes perfect sense and you are able to look at the results and feel confident that you understand the processes involved.

Remember the real world

While doing numerical modeling it is important to occasionally ask yourself how much you really know about the input compared to the complexity of the analysis. The following cartoon portrays an extreme situation, but underscores a problem that exists when uneducated or inexperienced users try to use powerful software tools.

“If we can incorporate boundary elements rather than simple finite elephants, enhance the statistical evaluation of parameter generation and stick with the fuzzy sets, I am confident that accuracy will be increased to at least the fourth decimal place.”



“Put another shovel full in Pat! It is full-cores they’re wanting!”

Note: origins of this figure are unknown at time of printing.

2.6 *How not to model*

As mentioned earlier in this chapter, it is completely unrealistic to expect to set up a complex model at the start of a project and immediately obtain realistic, understandable and meaningful results. There are far too many parameters and issues which can influence the results, so if this is your expectation, then modeling is going to lead to major disappointments.

For novice modelers; the initial reaction when faced with incomprehensible results is that something must be wrong with the software. It must be a limitation of the software that the solution is inappropriate or completely senseless. It is important to remember that the software is very powerful; it can keep track of millions of pieces of information and do repetitive computations which are far beyond the capability of the human mind. Without the software it would not be possible to make these types of analyses. The software by itself is extremely powerful numerically speaking, but essentially unintelligent. Conversely, the human mind has the capability of logic and reasoning, but has significant limitations retaining large amounts of digital data. It is the combination of the human mind together with the capability of a computer that makes numerical modeling so immensely powerful. Neither can do the task in isolation. The software can only be used effectively under the careful guidance and direction of the modeler.

Sometimes it is suggested that due to a time limitation, it is not possible to start simple and then progress slowly to a more complex analysis. A solution is needed quickly and since the budget is limited, it is necessary to immediately start with the ultimate simulation. This approach is seldom, if ever, successful. Usually this leads to a lot of frustration and the need to retreat to a simpler model until the solution is understandable and then build it up again in stages. Not following the above “how to” modeling procedures generally leads to requiring more time and financial resources than if you follow the recommended modeling concepts.

Remember, the software is only as good as your ability to guide and direct it. The intention of this document is to assist you in providing this guidance and direction so that you can take full advantages of the power the software can offer.

2.7 *Closing remarks*

As noted in the introduction, numerical modeling is a relatively new area of practice. Most university educational curricula do not include courses on how to approach numerical modeling and, consequently, the skill is often self-taught. As software tools such as SEEP/W become increasingly available at educational institutions and educators become comfortable with these types of tools, classes and instruction should improve with respect to numerical modeling.

When the numerical analysis software tool, SEEP/W 2007, is effectively utilized as it was intended to be used, it becomes an immensely powerful tool, making it possible to do highly complex analyses. It can even lead to new understandings about actual physical process.

The process of modeling is a journey of discovery, a way of learning something new about the complex behavior of our physical world. It is a process that can help us understand highly complex, real physical process so that we can exercise our engineering judgment with increased confidence.

3 Geometry and Meshing

3.1 Introduction

Finite element numerical methods are based on the concept of subdividing a continuum into small pieces, describing the behavior or actions of the individual pieces and then reconnecting all the pieces to represent the behavior of the continuum as a whole. This process of subdividing the continuum into smaller pieces is known as *discretization* or *meshing*. The pieces are known as *finite elements*.

In GeoStudio, the geometry of a model is defined in its entirety prior to consideration of the discretization or meshing. This is an approach that is new in GeoStudio 2007. Furthermore, automatic mesh generation algorithms have now advanced sufficiently to enable a well behaved, numerically robust default discretization often with no additional effort required by the user. Of course, it is still wise to view the default generated mesh but any required changes can easily be made by changing a single global element size parameter, by changing the number of mesh divisions along a geometry line object, or by setting a required mesh element edge size.

Figure 3-1 shows the fully defined model for a soil excavation project. The entire model was built using various geometry items.

- Soil regions were specified;
- Geometry lines were drawn at the locations of tie-back anchors and cutoff wall;
- Soil material models were created and assigned onto the geometry objects and
- Pre-defined boundary conditions were drawn on the region edges.

As a final step before solving, the mesh properties were viewed and adjustments made. In this case, the global element size was specified as 1.0 meter. In addition, the geometry line representing the grouted section of the anchors was discretized. The pre-stressed length of the anchors was intentionally left non-discretized. The final model is shown in Figure 3-2.

Now that you have a basic introduction to the concept of building your model using geometry objects, we can discuss each type of object in more detail. We must also have a discussion about the finite elements themselves, as these are the backbone to the entire finite element method.

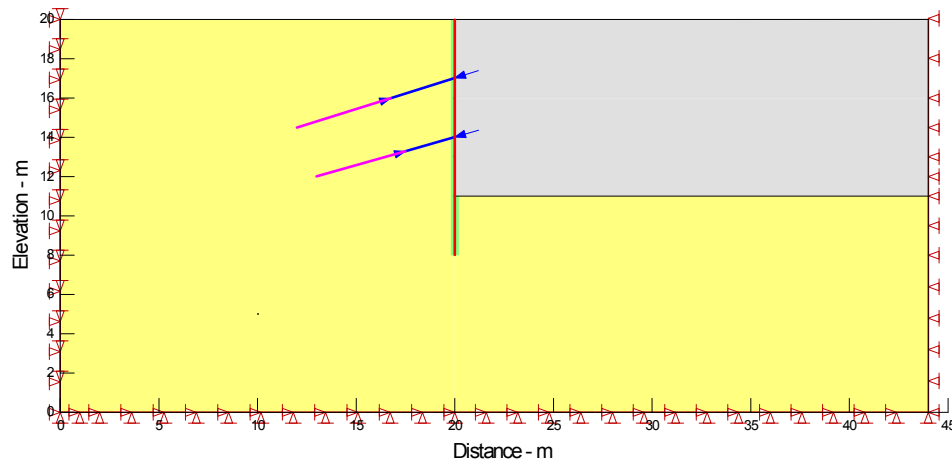


Figure 3-1 Fully defined geometry for soil excavation model

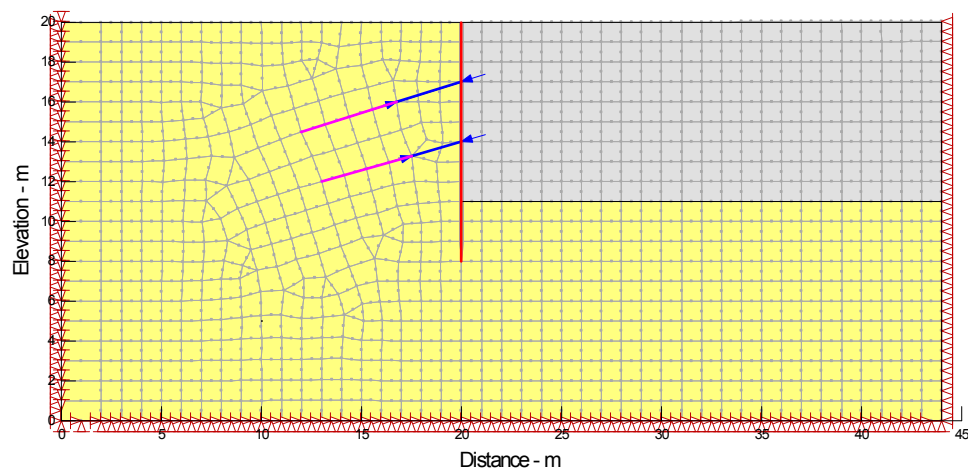


Figure 3-2 Default element discretization for model using 1m global element size constraint

3.2 Geometry objects in GeoStudio

New to GeoStudio in 2007 is the concept that the entire model is defined as a series of geometry objects. These objects can be soil regions, circular openings line objects, surface regions, and point objects. These objects are shown in the images below – as defined and then with the mesh applied.

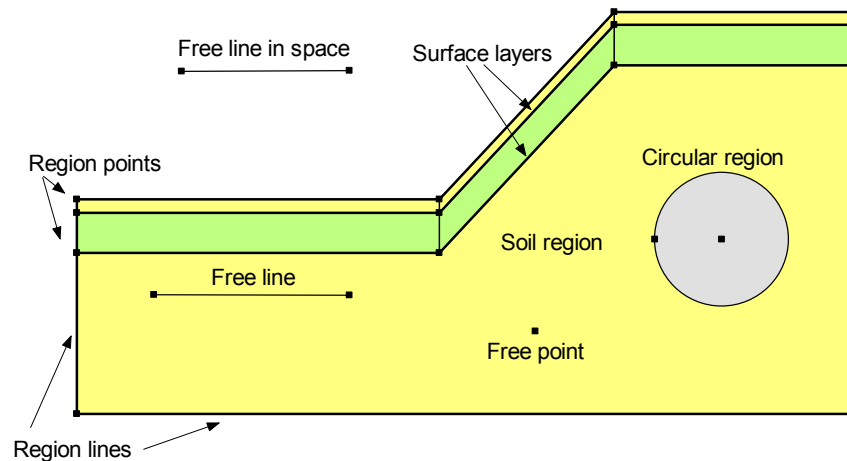


Figure 3-3 Available geometry objects

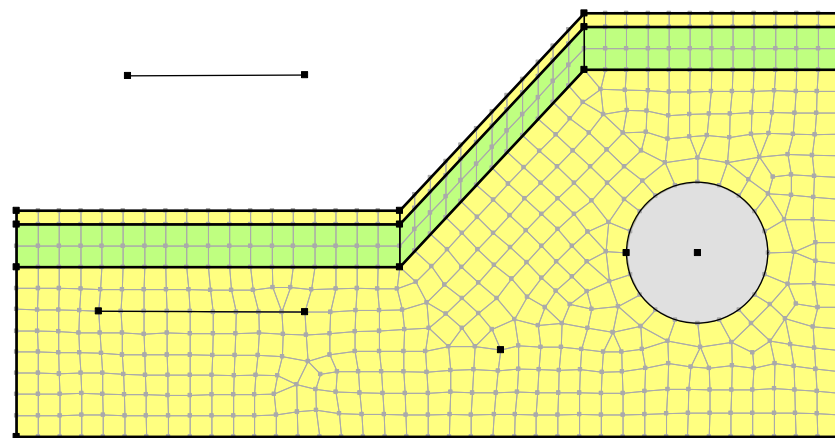


Figure 3-4 Mesh pattern for example model

Each of these geometry objects can have additional objects assigned to them such as material or boundary conditions objects. They can also have special properties, such as mesh element type, size and integration order.

Let us consider each in turn.

Soil regions, points and lines

GeoStudio uses the concept of regions and points to define the geometry of a problem and to facilitate discretization of the problem. The attraction of using regions is that they replicate what we intuitively do as engineers and scientists to illustrate concepts and draw components of a system. To draw a stratigraphic section, for example, we intuitively draw the different soil types as individual regions.

The use of regions offers all the advantages of dividing a large domain into smaller pieces, working and analyzing the smaller pieces, and then connecting the smaller pieces together to obtain the behavior of the whole domain, exactly like the concept of finite elements. Generally, all physical systems have to be broken down into pieces to create, manage and control the whole body.

A collection of highly adaptive individual pieces that can be joined together makes it possible to describe and define almost any complex domain or physical system. Such an approach is more powerful and can

be applied to a wider range of problems than any system that attempts to describe the whole domain as a single object.

Regions may be simple straight-sided shapes like quadrilaterals or triangles or a free form, multi-sided polygon. Figure 3-5 illustrates a domain constructed using one quadrilateral and two triangular regions. Also shown in this figure are the region points and the region lines. Each segment of a region edge between any two adjacent points is called a line. Both points and lines can have special properties as discussed in the next sections. In this figure the lines and points are not “free” as they belong to a region. They do, however have similar behavior to free points and lines.

Figure 3-6 shows a multi-sided polygonal region defined using 10 points. There is no restriction on the number of points in a region. However, the rule of thumb to keep things simple is always encouraged.

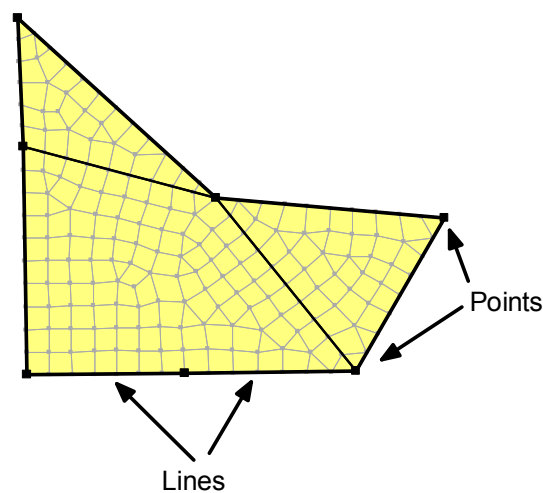


Figure 3-5 Illustration of a region's lines and points

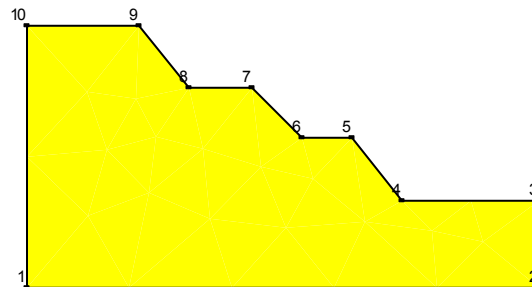


Figure 3-6 A multi-side polygonal region

Points can be selected and moved to modify the shape and position of regions, which provides for great flexibility in making adjustments and alterations to a problem definition.

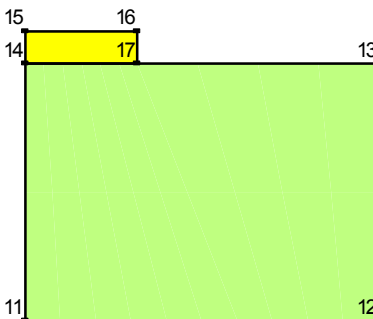


Figure 3-7 Regions of different size

Points are also required in order to join regions of different sizes and to control the meshing for specific purposes. Figure 3-7 shows a homogeneous soil region with a concrete footing region. The foundation region is made up of Points 11, 12, 13, 17 and 14. The footing region is made up of Points 14, 17, 16, and 15. Points 14 and 17 are common to both regions and therefore the two regions are properly joined and connected along this edge. In addition, Point 17 ensures that an element node will be created and will exist at the edge of the footing, which is required for proper meshing. It also breaks up the region edge between points 13 and 14 so that a unique boundary condition may be placed along the edge sub-section.

When a region is defined, it is restricted to having:

- One type of material,
- One type of element meshing pattern (or no mesh),
- One order of elements; either first- or second-order, and
- One integration order.

More information on finite elements is provided later in this chapter.

Free points

As seen in the illustrations above, regions are made up of a series of points. It is also possible for a point to exist within a region or outside of a region on its own. By default, a finite element “node” must exist at the location of all points, whether region corner points or free points. The advantage of this is that by placing a “free point” you can ensure that a boundary condition is applied at the desired location.

In past versions of GEO-SLOPE software, all boundary conditions were applied directly to mesh nodes or mesh element edges. This is no longer the approach to use. Now, all boundary conditions must be applied directly to region lines, region (or free) points or geometry lines. The power in this new approach will be readily evident to the user who decides to change the default generated mesh. In the past, changing the mesh required that all boundary conditions and soil properties (in some cases) were lost or attempted to be re-applied. Now, because properties and boundary condition exist as objects on geometry items, the mesh can be changed with no threat of having to re-do parts of the model set up.

Free lines

A free line is a line object that does not make up any part of a region edge. They can be very useful for applying anchors to a model or for specifying a geo-fabric or insulation layer. They can also be used for creating structural components that are partially in the soil and partially outside the soil. Here are some examples...

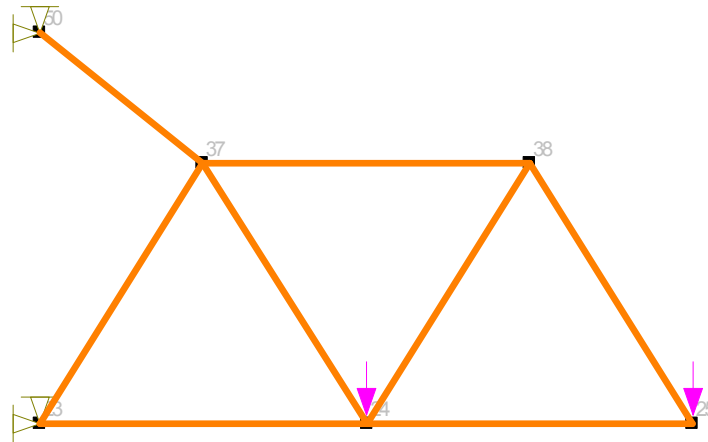


Figure 3-8 Truss loading model

In Figure 3-8 structural bar properties have been applied to a pattern of free lines in order to determine displacements under loading for the truss system.

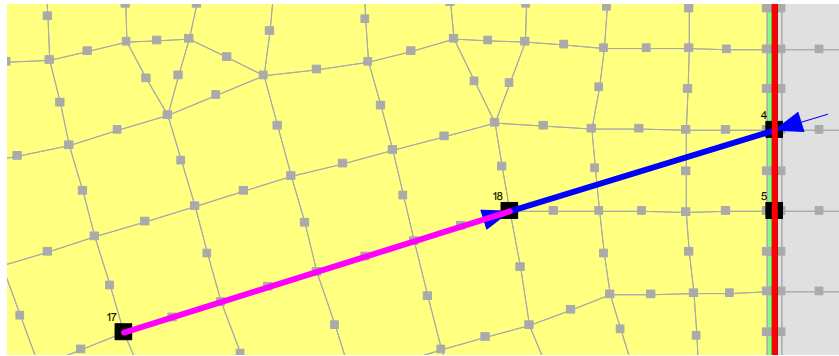


Figure 3-9 Anchors on lines (line is partially meshed)

Figure 3-9 shows two key aspects of free lines. A free line in two segments has been drawn to model a tie back anchor. The upper length of the free line was left un-discretized (i.e., independent of the finite element mesh) while the lower length is incorporated into the mesh. The lower end represents a beam structural member which requires that it is aligned with the surrounding soil elements. The upper length represents a structural bar element which only acts in tension or compression and only has an active force and stiffness at its end points which do coincide with nodal locations. It does not interact with the soil so does not need to share a mesh with the soil except at its end points.

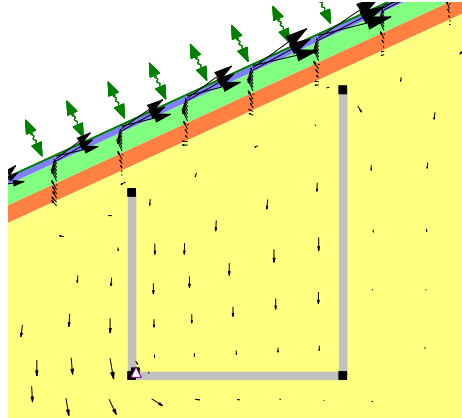


Figure 3-10 Lines with material model “none” (no flow) assigned to simulate lysimeter collection basin in waste beneath an engineered soil cover

Figure 3-10 shows the use of free lines to construct a lysimeter collection basin beneath an engineered soil cover system. The line was assigned a material model of “none” to simulate a no flow condition (i.e., a null material). This is a key point to understand, that the line was assigned a material model. When this is done, the line inherits the behavioral properties of the material assigned and a special interface element is added to the line mesh. The interface behavior depends on the application being solved. In this example, the interface has “no flow” across it. In a stress-deformation model, the interface on the line may be assigned soil – structure friction/slippage properties. Interface elements are discussed in more detail in the next section.

Interface elements on lines

In the previous section the concept of applying a material model to a single line was introduced. The actual material models that can be used are dependent on the analysis being solved. For example, in SEEP/W an interface model may be used to represent a geo-fabric or a null material to represent a barrier to flow. In TEMP/W it may be a thin insulation layer. In SIGMA/W, the material model may describe the friction properties between soils, or a soil and a structure such as a cutoff wall. You can read about all of these models in the respective engineering books.

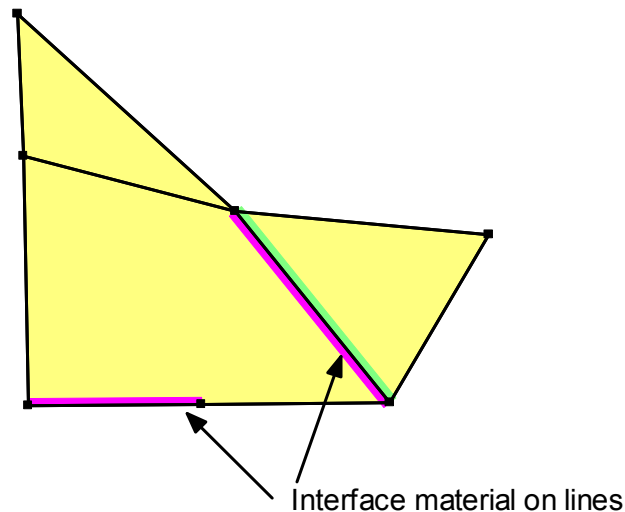


Figure 3-11 Illustration of "interface elements" on geometry lines

The discussion now will focus on how to apply an interface region to a line object. There are two ways to do this. There is a Draw Line Material Properties command in which you can choose a material model you have defined and apply it to a line by clicking just next to a line as shown in Figure 3-12. You can assign a different property to either side of a line. If you use this option, you are specifying the material as well as creating special thin “interface” elements. You can then go back and change the element thickness from its default value using the Draw Mesh Properties command and choosing the line.

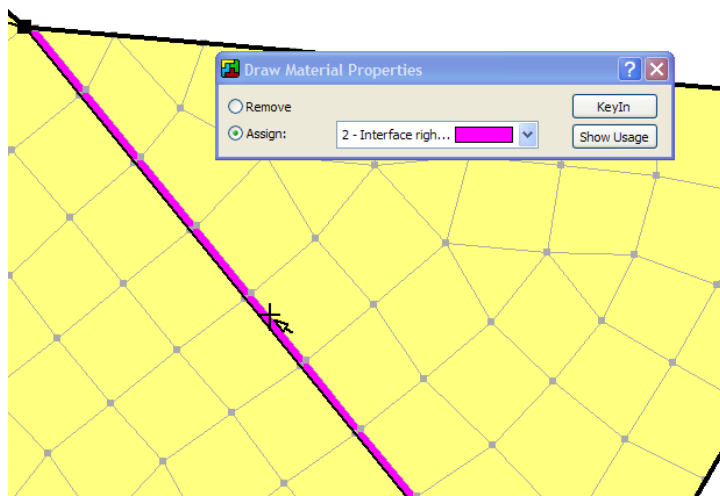


Figure 3-12 Using the Draw Line Material command to assign an interface model to a line

The second option for specifying an interface model on a line is to first create the thin elements and then assign the material to the line. You can use the Draw Mesh Properties command, select the line, choose the Generate Interface Elements option and specify an interface element thickness. This process is illustrated in Figure 3-13.

The actual thickness of the interface elements may or may not have physical meaning but the material model assigned to them will hold some meaning. If, for example, the interface represents an insulation layer in TEMP/W, then the thickness is relevant. However, if the interface describes the frictional behavior between two sliding blocks, then the thickness specified is not factored in the solution and it can be specified only to satisfy your presentation needs.

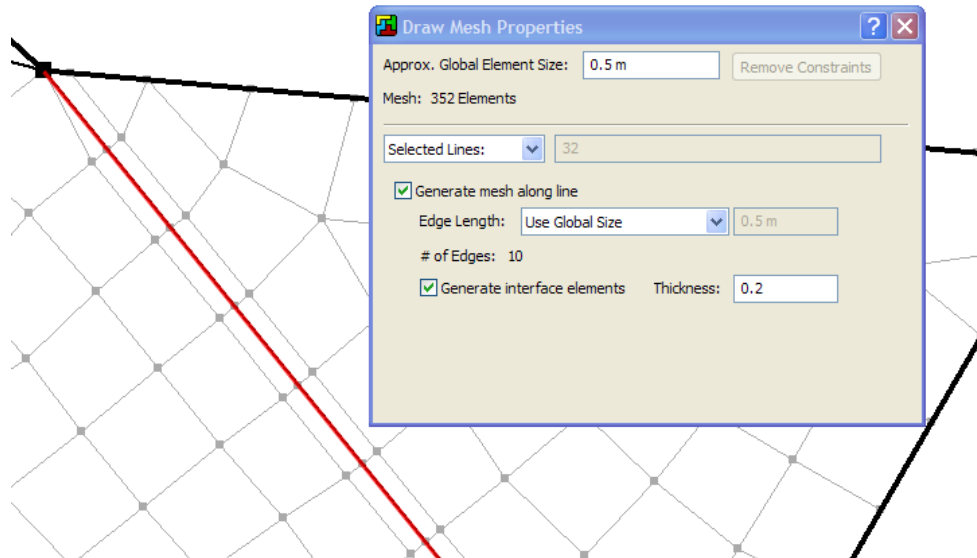


Figure 3-13 Using Draw Mesh Properties to create interface elements on selected line

Circular openings

A circular opening is a type of region that “floats” over top of another soil region. It is created using a Draw command and is defined by its center point and one point on its circumference. The region can be dragged to a different location or its circumference point can be moved to change the size of the opening. Like all other regions, it can have a mesh assigned to it; it can have material properties assigned to its edge (such as a tunnel liner interface material); and it can have boundary conditions assigned to its edge or center point.

Figure 3-14 shows a circular opening region that was placed on top of an existing soil region. The circular region was applied by clicking on the desired center point and then dragging the radius point to a desired location. Once defined, the region can be designated to be a mesh opening or an un-meshed opening or hole. A mesh may be necessary to obtain in-situ stresses prior to excavation. A hole may be necessary to simulate a pipe or culvert. In the opening presented in the figure, interface elements have been added to the tunnel face. This will make it possible to apply a structural beam to the face with a soil-structure interaction model applied between the beam and soil.

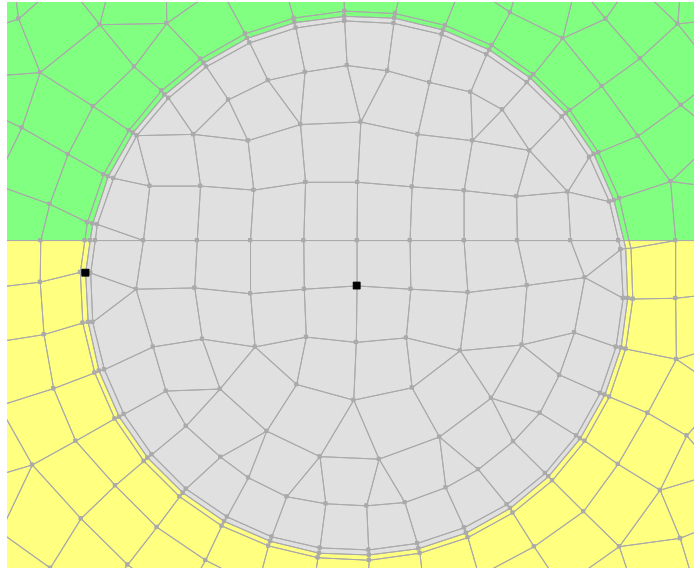


Figure 3-14 Circular region with defined center point, radius point and interface elements

3.3 Mesh generation

In GeoStudio all meshing is now fully automatic. There is no longer the ability to draw individual “finite elements.” In addition, there is no worry whether the mesh will be compatible across different regions or whether your material properties or boundary conditions will disappear if you change the mesh.

When a geometry region or line is initially drawn it is by default un-meshed. A default mesh is generated for the soil regions when you first use the Draw Mesh Properties command, which you may accept or modify. You may alter the size of the elements at a global level for the entire mesh, within any one or more regions, or along a line or around a point. You may specify mesh density as a real length unit, as a ratio of the global mesh size, or as the number of divisions along a line edge. Generally, however, it is recommended that you limit altering the mesh to changing the global density and then, if necessary, at a few limited locations where finer or coarser density is needed.

Meshing options and available patterns are shown in the image below.

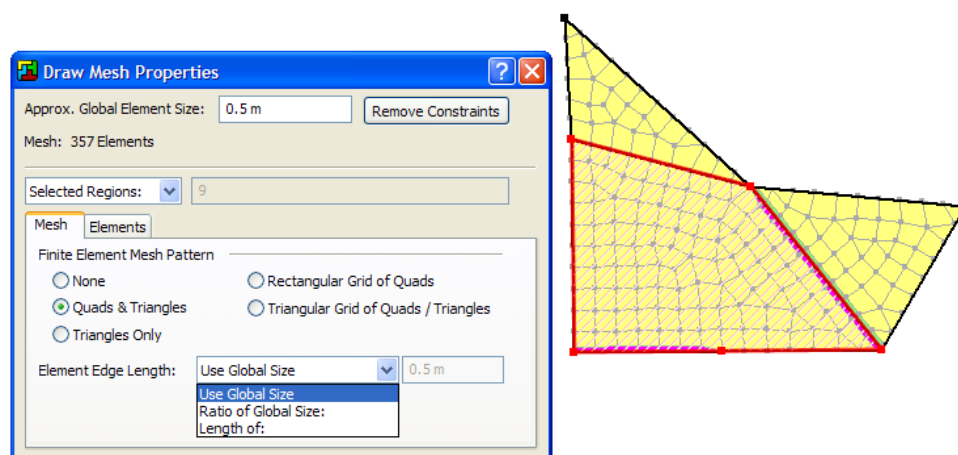


Figure 3-15 Draw Mesh Properties options

Structured mesh

Figure 3-16 presents what is known as a structured mesh because the elements are ordered in a consistent pattern and are of only two shapes and sizes. A structured mesh for a non-symmetrical geometry shape requires that several soil regions are created and the meshing is controlled within each region. This is likely more work to accomplish and will not yield a significant improvement in results. More efficient, automatic meshing options with good numerical performance are available and will be discussed in the next section. A structured mesh is created using either a rectangular grid of quads or a triangular grid of quads/triangles.

Unstructured quad and triangle mesh

The fully structured mesh shown in Figure 3-16 may require several regions to be defined so that you can control the meshing at a detailed level in order to maintain structure. A new meshing pattern is available that will automatically generate a well behaved unstructured pattern of quadrilateral and triangular elements as shown in Figure 3-17. In our opinion, this mesh option should be the first one you choose as it will meet your needs in most cases.

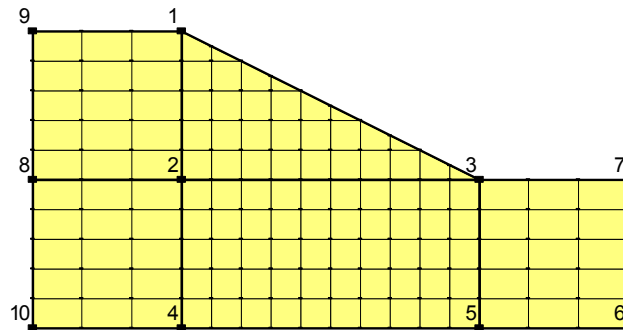


Figure 3-16 Structured mesh

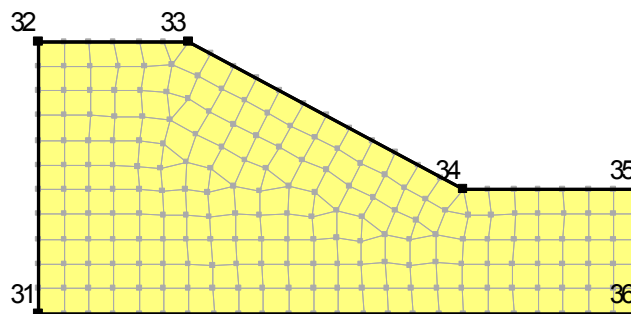


Figure 3-17 Mixed quad and triangle unstructured mesh

Unstructured triangular mesh

The diagram in Figure 3-18 shows the same section as in Figure 3-16, but this time with an unstructured triangular mesh. In this case the mesh is automatically created using Delaunay triangulation techniques. One of the great attractions of unstructured meshing is that almost any odd-shaped region can be meshed.

This meshing simplicity however has some numerical and interpretation consequences as discussed in more detail in the Structured versus Unstructured Section below.

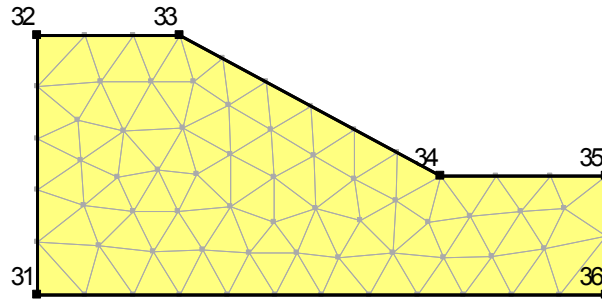


Figure 3-18 Unstructured triangular mesh

Triangular grid regions

GeoStudio has a special structured pattern for triangular regions called a triangular grid of quads/triangles. The next figure (Figure 3-19) shows a typical triangular region with the resulting structured mesh. The elements are a mixture of squares, rectangles, trapezoids and triangles. The use of this pattern is fairly general, but it does have some limitations and restrictions.

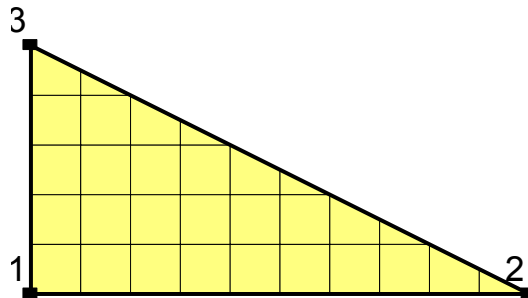


Figure 3-19 Triangular grid region

It is useful to think of the region as having three sides: a short side, an intermediate length side and a long side. The algorithm attempts to sort the edges so that the sides go from the shortest to the longest in a counter-clockwise direction. In this example, the shortest side is 3-1, the intermediate 1-2 and the longest 2-3.

The meshing algorithm works best when the number of divisions is controlled on the shortest and intermediate sides. To retain the even pattern shown in Figure 3-19, the number of divisions should be defined on the shortest side first and then on the intermediate side. The number of divisions on the intermediate side can be an even multiple of the number on the shortest side. In the above example, the shortest side has 5 divisions and the intermediate side can have 10, or 2 times that of the shortest side. The algorithm works best and gives the best structured mesh if the numbers of divisions on the longest side are left undefined allowing the algorithm to compute the appropriate number of divisions.

If a triangular region is mixed in with other more general regions, GeoStudio will attempt to ensure mesh compatibility. Sometimes however it may not be possible to adhere to the requirements for generating a structured mesh in a triangular region and then GeoStudio will substitute an unstructured mesh.

Rectangular grid of quads

Figure 3-20 shows the same region geometry meshed with the rectangular grid of quads pattern. This is a structured mesh but has the potential to be more difficult to control. The mesh pattern on the left side is very nice but near the base of the slope the quad shape is starting to distort. This distortion could be controlled by adding more region points along the bottom edge but this will still result in a mesh with large elements on the left and thinner elements on the right. This mesh pattern is ideally suited for four sided regions only.

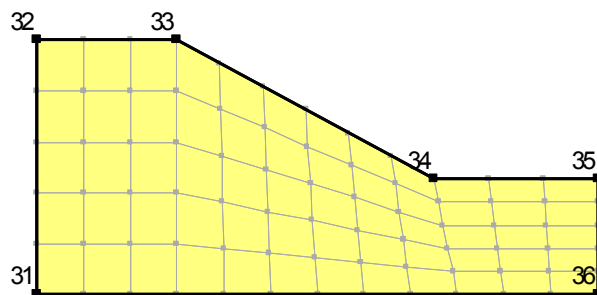


Figure 3-20 Rectangular grid of quad elements

3.4 Surface layers

At the ground surface conditions change in response to the climate and climatic conditions can change dramatically over short periods of time. For example, the ground maybe highly desiccated near the surface on a hot day before a thunderstorm. In a short period of time, the soil changes from being very dry to being saturated. Another example may be penetration of frost from the ground surface. To numerically deal with rapid and dramatic boundary changes it is necessary to have fine discretization near the ground surface. GeoStudio has a special procedure for constructing a surface layer that can be finely discretized. Figure 3-21 illustrates a surface layer placed over the surface of a larger region. The surface layer capability is also invaluable for discretizing features such as engineered soil covers over waste material, which may consist of several relatively thin layers of soil which also require fine discretization.

The ability to construct a surface layer is available in VADOSE/W, SEEP/W and TEMP/W. In SEEP/W the surface layer is used to tell the solver that it should track seepage face flows and infiltration events for any unit flux boundary condition. As a result, water that does not immediately infiltrate the ground is not considered lost from the analysis, but is allowed to pond and build up a positive pressure head in any user-defined low points along the surface. The other GeoStudio modules cannot be used to construct a surface layer, but once the surface mesh has been created it will exist in all the other modules. Consequently, if a layer has been created for a SEEP/W analysis, the surface layer will also be part of a SLOPE/W analysis, since GeoStudio uses only one geometry definition within a single data file.

Once the main soil profile has been meshed, a special Draw Surface Layer command can be used to build up a single or multi layer region along all or part of a ground surface. Parameters such as the soil type and individual layer geometry are defined and a quadrilateral element mesh with vertically oriented nodes is automatically built on top of the existing ground region. The structure of the mesh will ensure optimum numerical stability during the solution.

Quadrilateral elements are much better for modeling ground surface processes because the primary unknown gradients are usually steeper in a direction perpendicular to the surface. The presence of triangular elements in thin layers near the surface causes excessive fluctuation in the computed results

relative to the orientation of the triangular elements. Also, dealing with plant root zones in the VADOSE/W model necessitates that element nodes in the surface layer all fall on vertical lines. Moreover, using quadrilaterals greatly reduces the number of elements required, an important consideration when dealing with situations that will be very computationally intensive.

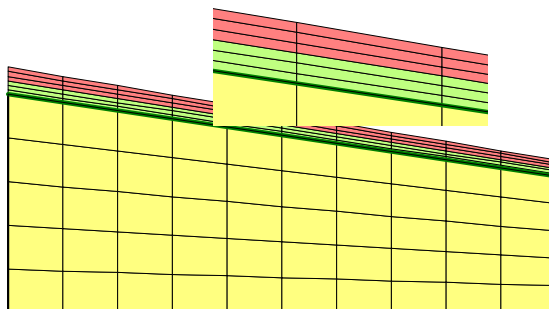


Figure 3-21 Illustration of a surface layer mesh

Surface layers have special viewing options. Consider the two meshes illustrated in Figure 3-22. The left diagram shows a surface layer without all the cluttering details as illustrated on the right. When many thin elements are located in a close proximity to each other, they can appear indistinguishable when viewed from a far away scale. By optionally turning off the surface mesh details a clearer image of the structure of the near surface soil layers can be viewed.

Figure 3-23 is another illustration of this optional viewing concept. The left diagram in the figure shows the detailed mesh and soil layers across the 0.75m thick surface region and the right diagram leaves the details out, but still shows the layer colors. A couple of additional key points can be made in regards to the figure. Notice that bottom two elements of the left diagram are the same soil type as the main underlying soil. This is a good mesh design strategy – that being to have the bottom most layer of the surface mesh be made of the same soil as the existing ground. Consider if the bottom layer of the surface soil was VERY different from the underlying soil. If a finely spaced mesh was placed directly on top of the different underlying soil then the numerical integration of material properties at the common mesh node between the two soils would be less accurate because of the influence of the large element area from the material below the common nodal point. By having the bottom layer of the surface layer be the same as the underlying soil, the element shapes are very similar in size and aspect at the common nodal point between the two very contrasting soils.

The second point to note from Figure 3-23 is that in the right diagram the nodes that are located at the interface between two soils are still viewable even though the main mesh details are not. This is intentional so that you can easily see and graph data at nodes that are used for automatic tracking of interlayer fluxes in the VADOSE/W model.

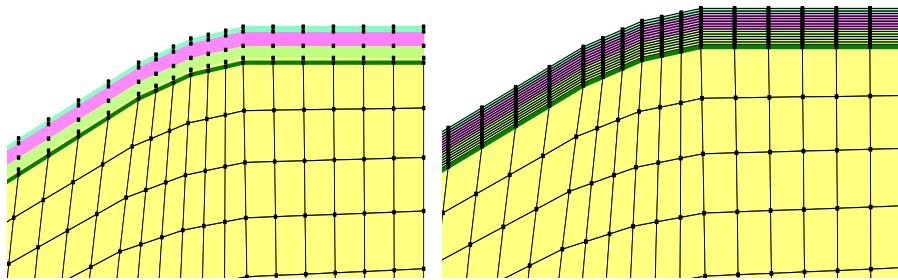


Figure 3-22 Surface region mesh with details off (left) and on (right)

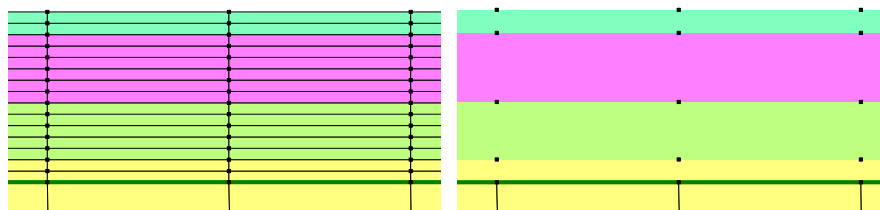


Figure 3-23 Close up of surface details on and off (note inter-layer nodes still visible in figure on right)

Boundary flux modeling with rainfall infiltration, runoff, snow melt etc. can be very numerically demanding from a convergence perspective. Potential problems can be made worse if the shape of the surface mesh is not “realistic.” Consider the two meshes illustrated in Figure 3-24 and Figure 3-25. In the first figure, the ground profile has rounded corners which are much more natural and much more numerically friendly. In the second figure, changes in slope angle are represented by a sharp break. This sharp break is not only un-natural, but the shape of the individual elements right at the transition points creates numerical problems if there are large changes in boundary condition type at different nodes within the same element. This would be the case when the corner node at the bottom of the slope becomes a seepage face point while the next node up slope is still an infiltration node. Basically, it is better to build the mesh to look somewhat natural.

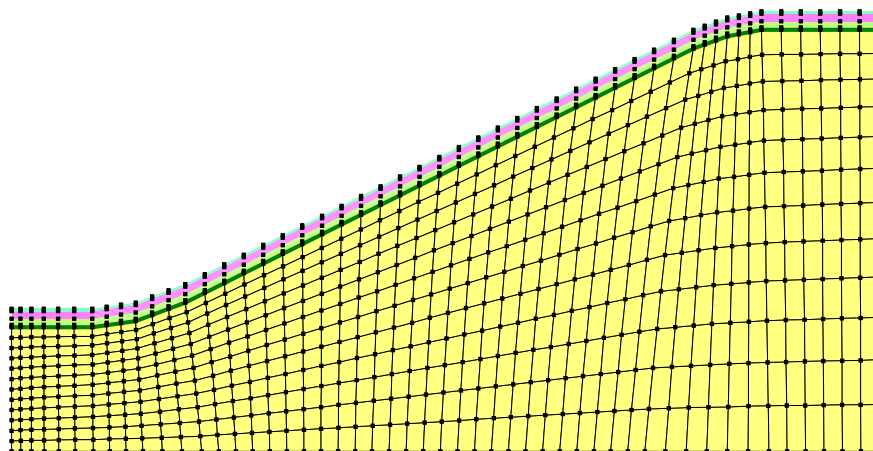


Figure 3-24 Mesh showing rounded surface slope breaks

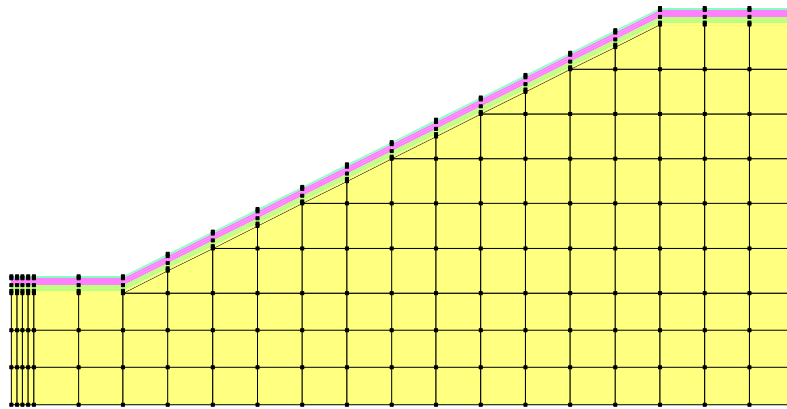


Figure 3-25 Mesh showing angular surface slope breaks

In order to create a surface mesh with more rounded features it is necessary to build the underlying soil mesh with the same rounded profile. This is easily accomplished in GeoStudio by adding additional region points near a slope break such that the region points can be moved slightly to create a rounded profile. This is the case in Figure 3-26 below where three region points are used at both the toe and crest of the slope. Also notice that three region points are used on the bottom of the mesh beneath the toe and crest location. This is a useful tip to remember. When you want to have more control over the trans-finite element mesh you should add region points on opposite sides of the mesh from where you need the detail. As a final note, adding region points can be done at any time – even after the surface layer is created. When the region beneath a surface layer is changed, the surface layer above it will be automatically regenerated to ensure mesh compatibility with the region below.

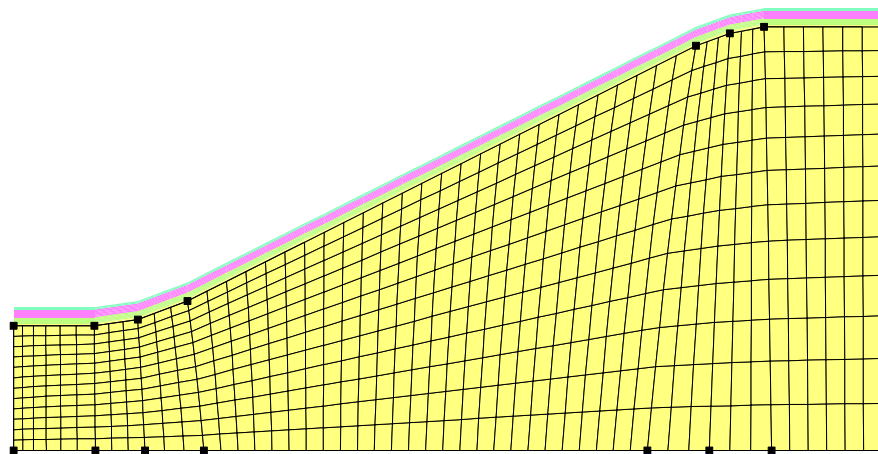


Figure 3-26 Region mesh with region corner points viewed and surface details not viewed

3.5 Joining regions

Compatibility must be maintained between regions to ensure the regions are connected. Regions must be joined at the region points and points must be common to adjoining regions for the regions to be properly connected. GeoStudio has a number of features to assist in achieving region compatibility.

The following are some of the main characteristics:

If the cross-hair symbol moves close to an existing point, the symbol will snap to the existing point.

A new point will be created if the cursor is on the perimeter of an existing region. The new point will then be common to the new region and to the existing region.

Points in between selected points are automatically selected along an existing region edge unless the Ctrl key is held down.

Consider the diagram in Figure 3-27. Region 1 is drawn first and Region 2 can be drawn by clicking on Points 7, 3, 8 and 9. Points 4, 5 and 6 are automatically added to Region 2.

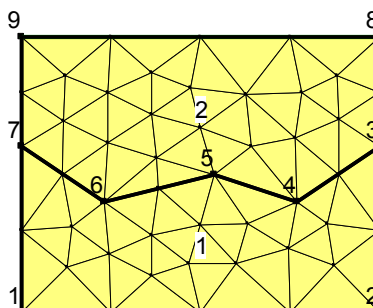


Figure 3-27 Regions joined along jagged line

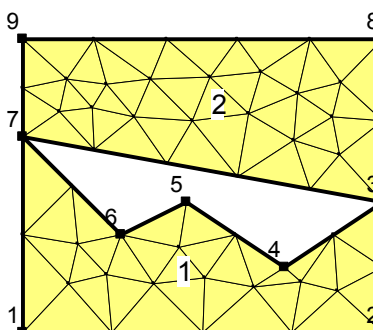


Figure 3-28 Adjoining regions with an open space

Sometimes it may be desirable to create an open area in a mesh and then it is necessary to hold down the Ctrl key when going from Point 7 to 3 or 3 to 7. Doing this results in a mesh as shown in Figure 3-28. In this case the Ctrl key was held down after clicking on Point 7, but before clicking on Point 3.

Additional details on joining regions are presented in the on-line help.

3.6 Meshing for transient analyses

Modeling transient processes requires a procedure to march forward in time increments. The time increments are referred to in GeoStudio as time steps. Selecting and controlling the time step sequence is a topic in itself and will be dealt with later. Obtaining acceptable transient solutions is not only influenced by the time steps, but also by the element size. In a contaminant transport advection-dispersion analysis (CTTRAN/W) it is necessary to have a time step sufficiently large to allow an imaginary contaminant

particle to move a significant distance relative to the element size, while at the same time not have the time step size be so large as to allow the particle to jump across several elements. The particle should, so to speak, make at least one stop in each element. In CTRAN/W this is controlled by the Peclet and Courant criteria.

In a simulation of consolidation, the time step size for the first time step needs to be sufficiently large so that the element next to the drainage face consolidates by at least 50 percent. Achieving this is related to the element size; the larger the element the greater the required initial time step. If the time step size is too small, the computed pore-water pressures may be unrealistic.

The important point in this section on meshing is to realize that meshing, more particularly element sizes, comes into play in a transient analysis. Rules and guidelines for selecting appropriate time stepping are discussed elsewhere with reference to particular types of analysis.

3.7 *Finite elements*

Discretization or meshing is one of the three fundamental aspects of finite element modeling. The other two are defining material properties and boundary conditions. Discretization involves defining geometry, distance, area, and volume. It is the component that deals with the physical dimensions of the domain.

A numerical book-keeping scheme is required to keep track of all the elements and to know how all the elements are interconnected. This requires an ordered numbering scheme. When finite element methods were first developed, creating the mesh numbering was very laborious. However, many computer algorithms are now available to develop the mesh and assign the element numbering. Developing these algorithms is in some respects more complex than solving the main finite element equations. GeoStudio has its own system and algorithms for meshing, which are designed specifically for the analysis of geotechnical and geo-environmental problems.

Some human guidance is required to develop a good finite element mesh in addition to using the powerful automatic meshing algorithms available. One of the issues, for example, is mesh size. Computers, particularly desktop or personal computers, have limited processing capability and therefore the size of the mesh needs to be limited. Variable mesh density is sometimes required to obtain a balance between computer processing time and solution requirements. Ensuring that all the elements are connected properly is another issue. Much of this can be done with the meshing algorithm, but it is necessary for the user to follow some fundamental principles. In finite element terminology this is referred to as ensuring mesh compatibility. GeoStudio ensures mesh compatibility within a region and for the most part ensures mesh compatibility across adjacent regions, but it is still possible to create a situation whereby mesh incompatibility exists. The user needs to provide some guidance in ensuring compatibility between regions.

The purpose of this chapter is to introduce some of the basic concepts inherent in meshing and outline some procedures which must be followed when developing a mesh. An understanding of these fundamentals is vital to proper discretization.

Much of this chapter is devoted to describing the meshing systems and the features and capabilities available in GeoStudio. In addition, there are also discussions on the selection, behavior and use of various element types, sizes, shapes and patterns. A summary of practical guidelines for good meshing practice are also outlined.

3.8 *Element fundamentals*

Element nodes

One of the main features of a finite element are the nodes. Nodes exist at the corners of the elements or along the edges of the elements. Figure 3-29 and Figure 3-30 show the nodes, represented as black dots.

The nodes are required and used for the following purposes:

- The positions of the nodes in a coordinate system are used to compute the geometric characteristics of the element – such as length, area or volume.
- The nodes are used to describe the distribution of the primary unknowns within the element. In the SEEP/W formulation, the primary field variable is the hydraulic head or pore-water pressure.
- The nodes are used to connect or join all the elements within a domain. All elements with a common node are connected at that node. It is the common nodes between elements that ensure compatibility, which is discussed in further detail below.
- All finite element equations are formed at the nodes. All elements common to a single node contribute to the characteristics and coefficients that exist in the equation at that node, but it is the equation at the node that is used to compute the primary unknown at that node. In other words, the seepage equation is developed for each node and the material properties which are used within the equations are contributed from the surrounding elements.

There can be multiple finite element equations developed at each node depending on the degrees of freedom. In seepage analysis there is only one degree of freedom at each node, which is the head or pore-water pressure. The number of finite element equations to be solved is equal to the number of nodes used to define the mesh. In a 2D stress-deformation analysis, there are two degrees of freedom at each node – displacement x and displacement y . Consequently, the number of equations for the whole domain is equal to two times the number of nodes. In a coupled consolidation analysis there are three degrees of freedom at each node – displacement x , displacement y and pore-water pressure. For a coupled consolidation analysis the total number of equations required to solve the problem is three times the number of nodes.

Since the number of finite element equations is related to the number of nodes, the number of nodes in a problem is one of the main factors in the computing time required to solve for the primary unknowns.

Field variable distribution

In a finite element formulation it is necessary to adopt a model describing the distribution of the primary variable within the element (e.g., total head). The distribution could be linear or curved.

For a linear distribution of the primary unknown, nodes are required only at the element corners. The two nodes (points) along an edge are sufficient to form a linear equation. Figure 3-29 illustrates this situation. Elements with nodes existing at the corners are referred to as first-order elements.

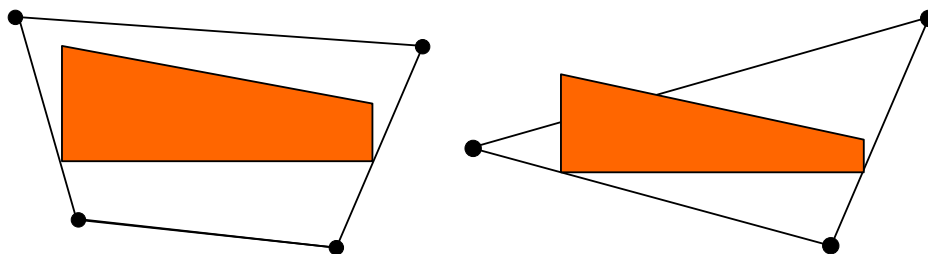


Figure 3-29 Primary field variable distribution in first-order elements

The derivative of the primary unknown with respect to distance is the gradient. For a linear distribution the gradient is consequently a constant. In the context of a seepage formulation the primary unknown is the hydraulic head. The derivative of head with respect to distance is the seepage gradient and the gradient is therefore constant within a first order element.

With three nodes defined along an edge, we can write a quadratic equation describing the distribution of the primary unknown within the element. Consequently the distribution of the primary unknown can be curved as shown in Figure 3-30. The derivative of the quadratic head distribution results in a linear gradient distribution. Elements with three or more nodes along an edge are referred to as higher order elements. More specifically, an element with three nodes along an edge is known as a second-order element.

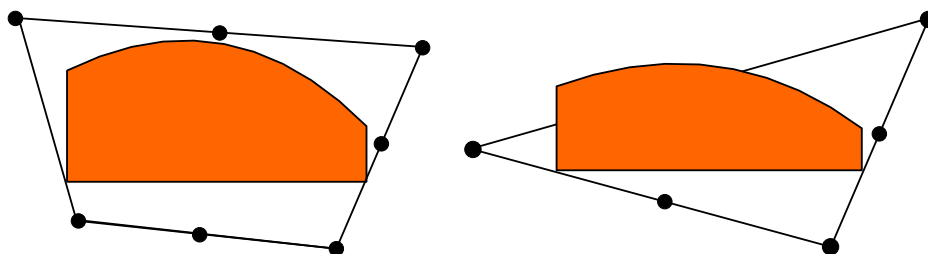


Figure 3-30 Primary field variable distribution in higher-order elements

Higher order elements are more suited to problems where the primary unknowns are vectors as in a stress-deformation analysis (deformation x and y). When the primary unknown is a scalar value as in a seepage formulation, there is often little to be gained by using higher-order elements. Smaller first-order elements can be as effective as larger higher-order elements. This is discussed in more detail in the meshing guidelines at the end of this chapter.

Element and mesh compatibility

Element and mesh compatibility are fundamental to proper meshing. Elements must have common nodes in order to be considered connected, and the distribution of the primary unknown along an element edge must be the same for an edge common to two elements.

Consider the illustration in Figure 3-31. Element numbers are shown in the middle of the element and node numbers are presented beside the nodes. Even though elements 4, 5 and 6 appear to be connected to elements 7, 8 and 9, they are actually not connected. Physically, the elements would behave the same as the two element groups shown with a physical separation on the right side of Figure 3-31. Common nodes are required to connect the elements as shown in Figure 3-32. Node 11, for example, is common to Elements 5, 6, 8 and 9.

Mixing elements of a different order can also create incompatibility. Figure 3-33 shows 4-noded quadrilateral elements connected to 8-noded elements. Elements 1 and 2 are 8-noded elements while

Elements 3 to 10 are 4-noded first-order elements. The field variable distribution in Element 1 along edge 9 to 11 could be curved. In Elements 3 and 4 the field variable distribution between 9 and 10 and between 10 and 11 will be linear. This means the field variable distributions between Elements 1 and 2 are incompatible with the field variable distributions in Elements 3 to 6.

The meshing algorithms in GeoStudio ensure element compatibility within regions. A special integer-based algorithm is also included to check the compatibility between regions. This algorithm ensures that common edges between regions have the same number of elements and nodes. Even though the software is very powerful and seeks to ensure mesh compatibility, the user nonetheless needs to be careful about creating adjoining regions. The illustration in Figure 3-31 can also potentially exist at the region level. At the region level, region points need to be common to adjoining regions to ensure compatibility.

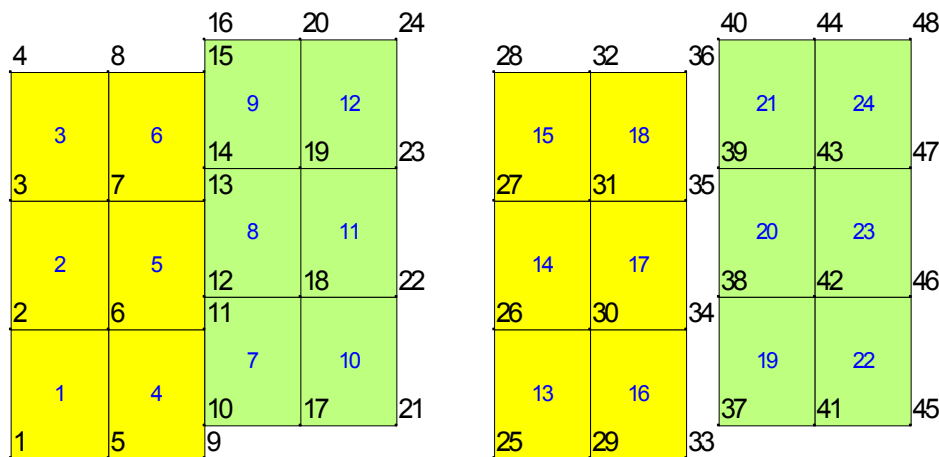


Figure 3-31 Disconnected elements – lack of compatibility



Figure 3-32 Connected elements – compatibility satisfied

5	8	13	18	23
4	2	12 6	17 10	22
3	7	11 5	16 9	21
2	1	10 4	15 8	20
1	6	9 3	14 7	19

Figure 3-33 Element incompatibility

The integer programming algorithm in GeoStudio seeks to ensure that the same number of element divisions exist between points along a region edge. The number of element divisions are automatically adjusted in each region until this condition is satisfied. It is for this reason that you will often notice that the number of divisions along a region edge is higher than what was specified. The algorithm computes the number of divisions required to achieve region compatibility.

Numerical integration

In a finite element formulation there are many integrals to be determined, as shown in the Theory chapter. For example, the integral to form the element characteristic matrix is:

$$\int [B]^T [C] [B] dv$$

For simple element shapes like 3-noded or 4-noded brick (rectangular) elements, it is possible to develop closed-formed solutions to obtain the integrals, but for higher-order and more complex shapes it is necessary to use numerical integration. GeoStudio uses the Gauss quadrature scheme. Basically, this scheme involves sampling the element characteristics at specific points known as Gauss points and then adding up the sampled information. Specific details of the numerical integration in GeoStudio are presented in the Theory Chapter.

Generally, it is not necessary for most users to have a comprehensive understanding of the Gauss integration method, but it is necessary to understand some of the fundamentals since there are several options in the software related to this issue and some results are presented at the Gauss sampling points.

The following table shows the options available. Use of the defaults is recommended except for users who are intimately familiar with numerical integration and understand the significance of the various options. The integration point options are part of the meshing operations in GeoStudio.

Element Type	Integration Points	Comments
4-noded quadrilateral	4	Default
8-noded quadrilateral	4 or 9	4 is the default
3-noded triangle	1 or 3	3 is the default
6-noded triangle	3	Default

Some finite element results are computed at the Gauss sampling points. GeoStudio presents the results for a Gauss region, but the associated data is actually computed at the exact Gauss integration sampling point. Even though a Gauss region is displayed, the data is not necessarily constant within the region.

With the View Object Information command, you can click inside a region and the geometry and material information is displayed together. By expanding the mesh folder, you can review the mesh information that has been assigned to the region. The number of Gauss regions within an element is equal to the number of Gauss integration points used in the analysis.

It is important to be cognizant of the impact of Gauss points on computing time and data storage. Nine-point integration in a quadrilateral element, for example, means that the element properties need to be sampled nine times to form the element characteristic matrix and element data is computed and stored at nine points. This requires more than twice the computing time and disk storage than for four-point integration. Sometimes nine-point integration is necessary, but the option needs to be used selectively.

Secondary variables

Earlier it was noted that finite element equations are formed at the nodes and the primary unknowns are computed at the nodes. Again, in a seepage formulation the primary unknowns are the total heads at the nodes. Once the primary unknowns have been computed, other variables of interest can be computed such as the seepage gradients within the element. Since these parameters are computed after the primary values are known, they are called secondary variables.

Secondary quantities are computed at the Gauss integration points. GeoStudio displays a Gauss region, but the associated values are strictly correct only at the Gauss integration point.

For contouring and graphing, the secondary values are projected and then averaged at the nodes. This can sometimes result in unrealistic values if the parameter variations are excessive between Gauss points. The procedure and consequence of the projection from Gauss points to the nodes is discussed further in the Visualization of Results Chapter. The important point is to be aware of the fact that secondary parameters are computed at Gauss integration points.

3.9 General guidelines for meshing

Meshing, like numerical modeling, is an acquired skill. It takes practice and experience to create the ideal mesh. Experience leads to an understanding as to how the mesh is related to the solution and vice versa. It is when you can anticipate an approximation of the solution that you will be more proficient at meshing.

The attraction of the GeoStudio system is that a mesh can quickly be created with relative ease and later modified with relative ease. This makes it convenient to try various configurations and observe how the meshing influences the results.

An appropriate finite element mesh is problem-dependent and, consequently, there are no hard and fast rules for how to create a mesh. In addition, the type of mesh created for a particular problem will depend on the experience and creativity of the user. However, there are some broad guidelines that are useful to follow. They are as follows:

- Use as few elements as possible at the start of an analysis. Seldom is it necessary to use more than 1000 elements to verify concepts and get a first approximate solution.

- All elements should be visible to the naked eye when the mesh is printed at a zoom factor of 100 % and when the horizontal and vertical scales are the same. The exception to this guideline are the elements found in a surface layer.

- The mesh should be designed to answer a specific question, and it should do not include features that do not significantly influence the system behavior.

The mesh should represent a simplified abstraction of the actual complex geometric field configuration.

Number of elements

Based on many years of responding to GEO-SLOPE user support questions, most users start with a mesh that is too complex, containing too many elements for the objective of the analysis. The thinking when users first start doing finite element analyses seems to be the more elements, the better; that a large number of elements will somehow improve the accuracy of the solution. This is not necessarily true. If the mesh is too large, the time required to obtain a solution can become unattainable. Sometimes it also becomes very difficult to interpret the results, particularly if the solutions appear to be unreasonable. The effort required to determine the reason for an unreasonable solution increases dramatically with mesh size.

We highly recommend that you try and create a mesh with less than 1000 elements, particularly at the start of an analysis. Our experience is that most geotechnical problems can be modeled with 1000 elements or less. Obviously there are exceptions, but this number is a good goal to strive for. Later, once you have a good first understanding of the critical mechanisms in your problem, you can increase the mesh density to refine the analysis.

Effect of drawing scale

Another good guideline is that all elements should be visible to the naked eye when the mesh is printed or viewed at a 100% zoom factor. Groups of elements that appear as a solid or nearly solid black smudge on the drawing are too small. This means a suitable element size is related to the drawing scale. A drawing at a scale of 1:100 can have much smaller elements than a drawing at a scale of 1:2000. In other words, if it is necessary to zoom in on an area of the drawing to distinguish the elements, the elements may be unnecessarily small.

All elements should be readily distinguishable when a drawing is viewed when the vertical scale is equal to the horizontal scale. It is possible to draw a nice looking mesh at a vertical exaggerated scale, but when viewed at a true vertical scale the mesh appears as a wide black line. This effect is illustrated in Figure 3-34. The top part of the figure shows a nice mesh at 10V:100H, a 10 times vertical exaggeration. The same mesh at a scale of 100V:100H appears at the bottom of Figure 3-34. At an exaggerated scale the elements appear suitable, but at a true scale they are not appropriate.

It is important to remember that the main processor which solves the finite element equations sees the elements only at the true scale. The vertical exaggeration is used only in DEFINE and CONTOUR for presentation purposes.

A good rule to follow is to always view the mesh at a true scale before solving the problem to check that the mesh is reasonable for the purpose of the analysis.

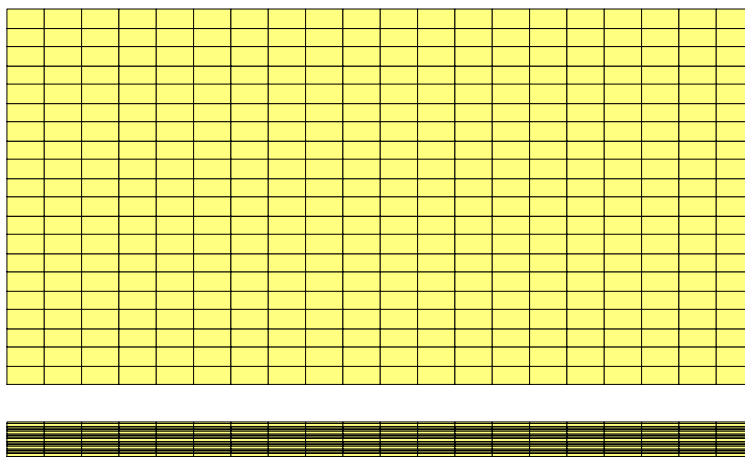


Figure 3-34 Mesh at an exaggerated scale (upper) and at a true scale (lower)

Mesh purpose

The “How To” modeling chapter notes that in good numerical modeling practice it is important to form a mental image of what the solution may possibly look like and to clearly define the purpose of the model before trying to create a model. Meshing is closely tied to this guideline. The mesh should be designed to answer specific questions. Trying to include all possible details in a mesh makes meshing unnecessarily time consuming and can sometimes make it difficult to interpret the results.

Let us assume that we are interested in estimating the seepage through the clay core of a zoned dam with rock shells. Figure 3-35 shows a typical case. The rock shells are considered to be many orders of magnitude more permeable than the core. In addition, the granular drain filter layers between the clay and the rock are clean and can easily handle any seepage through the core without impeding the drainage. In other words, the granular filter layers and rock shells make no contribution to dissipating the hydraulic head on the upstream side of the core. If this consideration is true, then there is nothing to be gained by including the highly permeable materials in the analysis. A mesh such as in Figure 3-35 is adequate to analyze the seepage through the core.

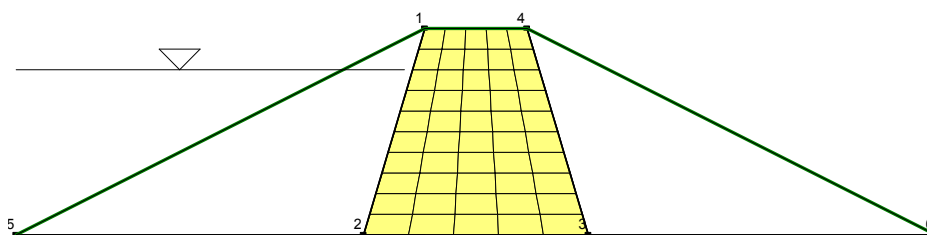


Figure 3-35 Modeling core of zoned dam

Figure 3-36 shows the total head contours (equipotential lines) in the core. From this the seepage quantities through the core can be computed.

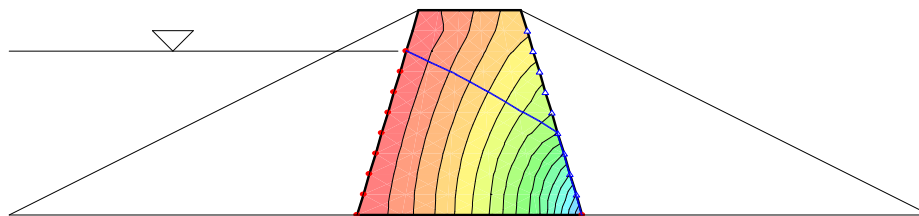


Figure 3-36 Equipotential lines in core of dam

Sometimes a mesh may be required to include the shells in the analysis for other reasons, such as a stress-deformation analysis. In such a case, the mesh can exist, but does not need to be included in the analysis. This is accomplished using null elements as shown in Figure 3-37. Elements in GeoStudio can be null (not active) by leaving a key material property undefined. In SEEP/W the elements are null if there is no specified conductivity function for the material. In the example in Figure 3-37 the rock shells have no conductivity function assigned to the material.

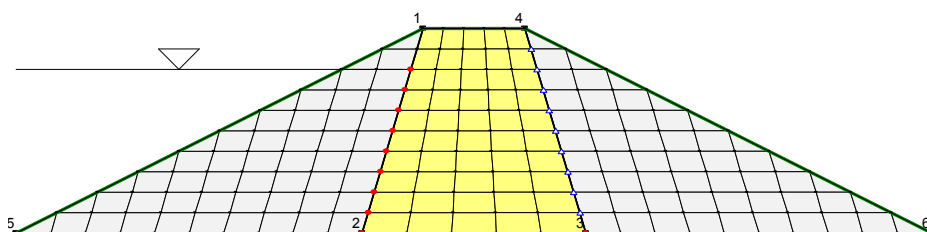


Figure 3-37 Mesh with null elements in shells of dam

One of the attractions inherent to numerical modeling is that the geometry and finite element mesh do not necessarily have to conform strictly to the physical conditions. As in Figure 3-35, the core can be analyzed in isolation. This would not be possible in physical modeling. The dam with a toe drain in Figure 3-38 is another good example. The toe drain does not have to be included in the numerical analysis. This, of course, would not be possible in a physical model.

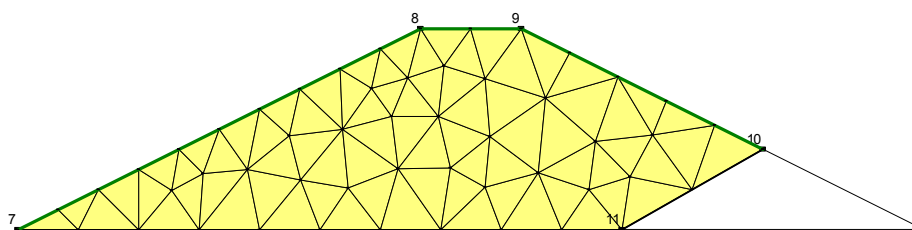


Figure 3-38 Dam with a toe drain

Simplified geometry

A numerical model needs to be a simplified abstraction of the actual field conditions. This is particularly true when it comes to the geometry. Including all surface irregularities is unnecessary in most situations. Geometric irregularities can cause numerical irregularities in the results, which distract from the main overall solution. The main message can be lost in the numerical noise.

Simplifying the geometry as much as possible is particularly important at the start of an analysis. Later, once the main processes involved are clear, the geometry can be altered to determine if the geometric details are important to the main conclusions.

The situation is different if the main objective of the analysis is to study the effects of surface irregularities. Then the irregularities of course need to be included. So, once again, the degree of geometric complexity depends on the objectives of the analysis.

Also, the level of geometric detail that needs to be included in the problem must be evaluated in light of the certainty with which other factors such as the boundary conditions and material properties are known. There is little to be gained by defining a very detailed geometry if the material properties are just a rough estimate. A simplified geometry is more than adequate if the material properties are rough estimates. There needs to be a balance in complexity between all the aspects of a finite element analysis, including the geometry.

Over-complicating the geometry is a tendency when users first get into numerical modeling. Then as modelers gain more experience they tend to use more simple geometries. This obviously comes from understanding how the mesh can influence the results and what level of complexity is required. The situation should be the reverse. It is the modelers with limited experience who should use simplified geometries.

The main message to remember when starting to model is to keep the problem as simple as possible until the main engineering issues are well understood.

4 Material Models and Properties

This chapter describes the various soil hydraulic properties that are required in the solution of the seepage partial differential equation. It is important to have a clear understanding of what the soil properties mean and what influence they have on the type of results generated. This chapter is not meant to be an all-inclusive discussion of these issues, but to highlight the importance of various parameters and the implications associated with not defining them adequately.

Well defined soil properties can be critical to obtaining an efficient solution of the finite element equations. When is it sufficient to guess at a function and when must you very carefully define one? This chapter will address these issues.

4.1 Soil behavior models

There are four different material models to choose from when using SEEP/W. A summary of these models and the required soil properties are given below and a discussion of the individual parameters and functions are provided in the next section.

Material models in SEEP/W

- 1) None (used to removed part of a model in an analysis)
- 2) Saturated / Unsaturated model
 - Hydraulic conductivity function, ratio and direction
 - Water content function
 - Air conductivity function (only if AIR/W added)
- 3) Saturated only model
 - Hydraulic saturated conductivity (Ksat), ratio and direction
 - Saturated water content
 - Coefficient of volume compressibility (Mv)
 - Air conductivity set to zero (only if AIR/W added)
- 4) Interface model
 - Hydraulic normal and tangent conductivity
 - Air conductivity (only if AIR/W added)

The Saturated Only soil model is very useful for quickly defining a soil region that will always remain below the phreatic surface, but it should not be used for soils that will at some point during the analysis become partially saturated. If this happens, the model will continue to solve but you will be saying, in effect, that the unsaturated zone can transmit the water at the same rate as for the saturated soil. This will result in an over estimate of flow quantity and can result in an unrealistic water table.

If there is any doubt about which model to choose for a soil region (as opposed to an interface region), you should select the Saturated / Unsaturated model. Also, if you intend to do a coupled analysis with CTRAN/W you MUST use the fully specified soil property functions.

The Interface soil model is to be used in conjunction with “interface elements” that are added to the mesh to represent geo-membranes, wick drains, or cut off walls. In these cases, you can specify a different

tangent and normal conductivity – which would be the case in a wick drain where the smearing effect caused during installation results in a lower conductivity in the normal flow direction than in the tangential direction.

4.1 **Soil water storage – water content function**

It is important to understand the relationship between pore-water pressure and water content in seepage analyses. Soil consists of a collection of solid particles and interstitial voids. The pore spaces or voids can be filled either with water or air, or with a combination of both. In a saturated soil, all the voids are filled with water and the volumetric water content of the soil is equal to the porosity of the soil according to:

$$\Theta_w = nS$$

where:

- Θ_w = the volumetric water content,
- n = the porosity of the soil, and
- S = the degree of saturation (in saturated soil equal to 1.0 or 100%).

The porosity, n , is related to the void ratio, e , by:

$$e = \frac{n}{1-n} = \frac{wG_s}{S}$$

where:

- w = the gravimetric water content, and
- G_s = the particle specific gravity.

In an unsaturated soil, the volume of water stored within the voids will vary depending on the matric suction within the pore-water, where matric suction is defined as the difference between the air (U_a) and water pressure (U_w) as follows: $U_a - U_w$.

There is no fixed water content in time and space and so a function is required to describe how the water contents change with different pressures in the soil.

The volumetric water content function describes the capability of the soil to store water under changes in matric pressures. A typical function for a drying soil is shown in Figure 4-1 where the function was measured for an air pressure of zero.

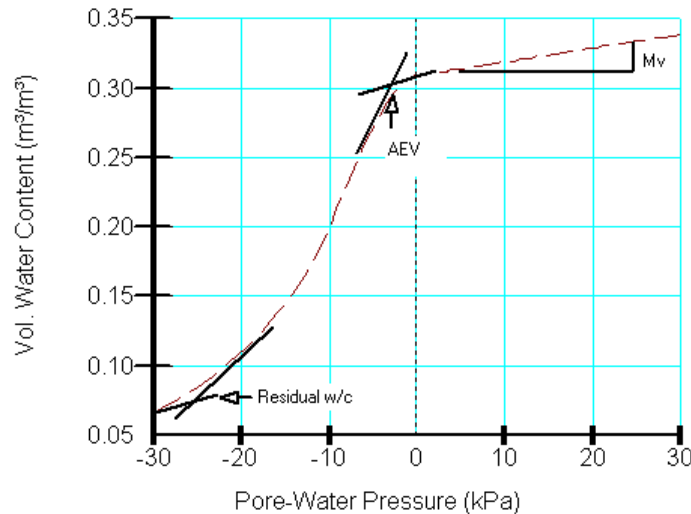


Figure 4-1 Volumetric water content (storage) function

The volumetric water content function describes what portion or volume of the voids remains water-filled as the soil drains. The three main features that characterize the volumetric water content function are the air-entry value, (AEV), the slope of the function for both the positive and negative pore-water pressure ranges (designated as m_w), and the residual water content or saturation, (Θ_r or S_r). The air-entry value (AEV) corresponds to the value of negative pore-water pressure when the largest voids or pores begin to drain freely. It is a function of the maximum pore size in a soil and is also influenced by the pore-size distribution within a soil. Soils with large, uniformly shaped pores have relatively low AEV's.

Adequately representing the slope of the water content function, in both the positive and negative pore-water pressure regions, can be very important in a seepage analysis, since water can be released from the soil in two ways. Water can be released by draining the water-filled voids thus desaturating the soil profile by gravitation forces, or by compressing the soil skeleton and reducing the size of the voids, effectively squeezing water out of a saturated system. In the positive pore-water pressure region, m_w becomes equivalent to m_v , the coefficient of volume compressibility for one-dimensional consolidation. The slope of the volumetric water content function in the negative pore-water pressure range represents the rate at which the volume of water stored within the soil changes as the pressure changes, over a range of values from the AEV to the pressure at the residual water content.

Another key feature of the volumetric water content function is the residual volumetric water content, which represents the volumetric water content of a soil where a further increase in negative pore-water pressure does not produce significant changes in water content. This point can also be expressed in terms of the degree of saturation by dividing the residual volumetric water content by the porosity of the soil. It is possible to remove water to a state less than the residual water content value, but this process is controlled by evaporation and / or osmotic forces. Evaporative drying is excluded from SEEP/W, but is included in VADOSE/W by considering simultaneous coupled heat, mass and vapor flow.

Water can be released by draining the water-filled voids and de-saturating the soil profile by gravitation force; by compressing the soil skeleton and reducing the size of the voids, effectively squeezing water out of a saturated system; or by applying evaporative demand.

Factors affecting the volumetric water content

The three main features of the volumetric water content function as described above are most strongly influenced by the size of the individual soil particles and the distribution of particle sizes in the soil. By way of example, consider the differences between the volumetric water content functions for sand, silt and clay shown in Figure 4-2

There are several factors that influence the shape of the volumetric water content function. As mentioned earlier, the air-entry value (AEV) reflects how much negative pore-water pressure can be applied to the pore-water before the largest pores or voids start to drain.

Consider the uniform sand function. Since the particles are relatively large and approximately the same size, water can easily be removed under relatively small negative pore-water pressures and the resulting air entry value is smaller than that for the silt or clay. The uniform nature of the pores means that all the pores drain over a small range of negative pore-water pressures, which makes the slope of the function steeper than the others.

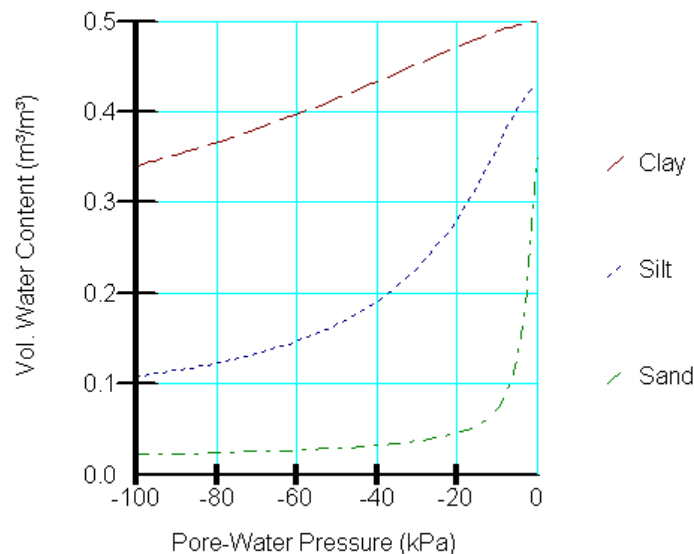


Figure 4-2 Typical storage functions for 3 soil types

The silt has a wider distribution of pore sizes. Some of the pores between the larger sand particles are filled with silt particles, making the largest pore sizes smaller than those of uniform sand. Consequently, a more negative pore-water pressure must be applied before drainage begins, thereby increasing the AEV. In addition, the pore sizes are not as uniform and the slope of the function is less steep.

The pores between the individual clay particles in the clay are small. It is often difficult to identify a specific air-entry value for clay, as consolidation and changes to the structure of the clay can release water from the system over a significant range before air actually enters the pores. Since clay tends to be compressible, the slope of the function in the positive pore-water pressure region tends to be steeper and becomes a significant parameter to consider in saturated seepage analyses.

4.2 Storage function types and estimation methods

It is not especially difficult to obtain a direct measurement of a volumetric water content function in a laboratory, but it does require time and it requires finding a geotechnical laboratory that performs the service. It is, however, standard practice to obtain a grain-size distribution curve and many companies

have the capability and facilities to develop their own curves. The development of the grain-size distribution curve is inexpensive and can be quickly accomplished.

One of the required input parameters for a transient analysis is the volumetric water content function. Since it can sometimes be difficult or time consuming to obtain a volumetric water content function, it may be of benefit to be able to develop an estimation of the volumetric water content function using either a closed-form solution that requires user-specified curve-fitting parameters, or to use a predictive method that uses a measured grain-size distribution curve. SEEP/W has four methods available to develop a volumetric water content function. One is to estimate a data point function using a predictive method based on grain size, one is to base your function of a sample set of functions built into the software, and two are closed form equations based on known curve fit parameters.

Estimation method 1 (grain size - Modified Kovacs)

Aubertin et al (2003) presented a method to predict the volumetric water content function which is modified from the method proposed by Kovacs (1981). The modifications were made to Kovacs's method to better represent materials such as tailings from hard-rock mines. A further modification extended the method for clay type soils. The Aubertin et al. method predicts the volumetric water content function using basic material properties which can be useful, particularly for preliminary analysis. It should be cautioned that, especially for clay type materials, it is critical to base a final design on measured material properties as opposed to estimated ones.

The function is initially determined as a degree of saturation function and then is later converted to a volumetric water content function. The function is developed by defining the degree of saturation for two main components. The first component contributes to the amount of water that is stored in a soil by capillary forces that exist at relatively small negative pore-water pressures. The second component contributes to the volumetric water content function at large negative pore-water pressures where the amount of water that exists in the soil is primarily a function of adhesion. Both of these components can be evaluated from the negative pore-water pressure and material property information such as particle-size, the shape of the particles and the porosity.

The degree of saturation as determined based on the capillary and adhesive components is as follows:

$$S_r = \frac{\Theta_w}{n} = S_c + S_a^*(1 - S_c)$$

where:

S_r = the degree of saturation,

Θ_w = the volumetric water content,

n = the porosity,

S_c = the degree of saturation due to capillary forces, and

S_a^* = the bounded degree of saturation due to adhesion (S_a).

where:

$$S_a^* = \langle 1 - S_a \rangle + 1$$

The adhesive component is a bounded value since it is possible at low suctions for the value S_a to be greater than 1. The bounded value ensures that for a S_a greater or equal to 1, $S_a^* = 1$ and if S_a is less than 1, then $S_a^* = S_a$.

The adhesion component is associated with the thin film of water that covers the surface of the soil grain and depends on basic material properties such as the negative pore-water pressure in the soil and the particle-size, shape coefficient and porosity of the soil. It is determined by the following equation:

$$S_a = aC_\Psi \frac{\left(\frac{h_{co}}{\Psi_n}\right)^{2/3}}{e^{1/3} \left(\frac{\Psi}{\Psi_n}\right)^{1/6}}$$

where:

- a = a curve fitting parameter,
- Ψ = the suction,
- Ψ_n = a suction term introduced to ensure dimensionless component,
- e = the void ratio,
- h_{co} = the mean capillary rise (cm) determined for capillary soils by:

$$h_{co} = \frac{b(cm^2)}{eD_{10}(cm)}$$

or

$$h_{co} = \frac{\xi w_L^{1.75}}{e}$$

for cohesion type soils where:

- D_{10} = the particle diameter (cm) corresponding to 10% passing on a grain-size curve,

$b(cm^2)$ = is given by:

$$b(cm^2) = \frac{0.75}{1.17 \log C_u + 1}$$

where:

- C_u = the coefficient of uniformity,
- w_L = the liquid limit (%),
- ξ = a constant approximately equal to 402.2 cm^2 ,
- C_Ψ = a correction coefficient that allows a progressive decrease in water content at high suctions, forcing the function through a water content of zero at one million kPa suction as initially proposed by Fredlund and Xing (1994) and described by:

$$C_{\Psi} = 1 - \frac{\ln\left(1 + \frac{\Psi}{\Psi_r}\right)}{\ln\left(1 + \frac{\Psi_o}{\Psi_r}\right)}$$

where:

Ψ_r = the suction corresponding to the residual water content at which point an increase in suction will not effectively remove more liquid water from the soil and given by:

$$\Psi_r = 0.86 \left(\frac{\xi}{e} \right)^{1.2} w_L^{1.74}$$

The capillary saturation, which depends essentially on the pore diameter and the pore size distribution, is given by:

$$S_c = 1 - \left[\left(\frac{h_{co}}{\Psi} \right)^2 + 1 \right]^m \exp \left[-m \left(\frac{h_{co}}{\Psi} \right)^2 \right]$$

where:

m = a fitting parameter that takes into account the pore size distribution and controls the shape and position of the volumetric water content function in the capillary zone.

For plastic-cohesive soils considered here, both the value of parameters m and a can be taken as constants with $m=3 \times 10^{-5}$ and $a=7 \times 10^{-4}$ in the predictive applications. For the capillary based soils, m and a can be taken as 1 and 0.01 respectively.

Estimation method 2 (sample functions)

GeoStudio provide several “typical” water content functions for different types of soils. In using these sample functions, it is up to you to specify the saturated water content and the residual water content (if any) based on your understanding of field conditions. These functions are provided as a means to letting you set up some test models quickly, change functions easily, decide how sensitive your results are to function shape, and ultimately to have you decide if you need to spend more time and money obtaining more accurate data. In the past, our list of “real” functions ended up being used as final design material properties with little thought about their relevance. This is not good modeling practice.

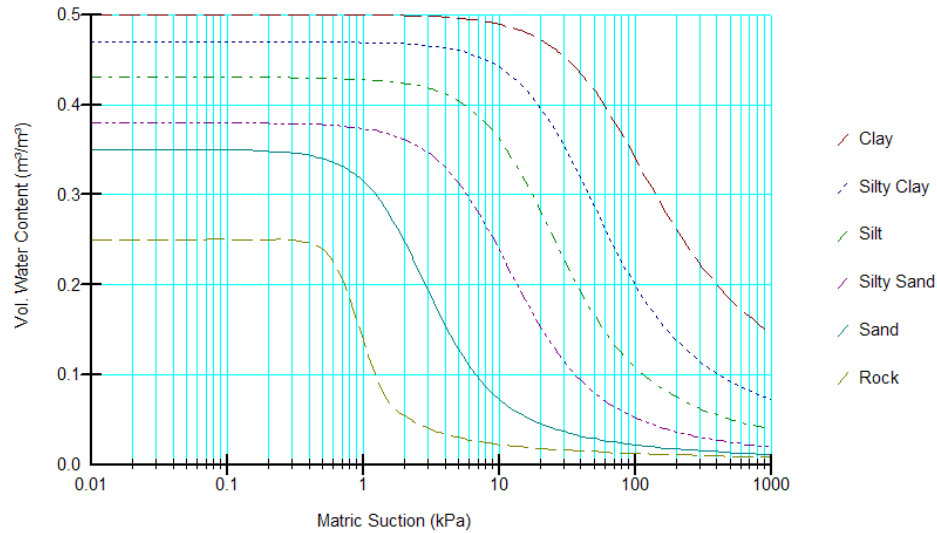


Figure 4-3 Sample Functions in GeoStudio

Closed form option 1 (Fredlund and Xing, 1994)

The Fredlund and Xing (1994) method is a closed-form solution that can be used to develop the volumetric water content function for all possible negative pressures between zero and minus one million kPa based on the user's knowledge of a group of three parameters. The governing equation is as follows:

$$\Theta_w = C_\psi \frac{\Theta_s}{\left\{ \ln \left[e + \left(\frac{\Psi}{a} \right)^n \right] \right\}^m}$$

where:

- Θ_w = the volumetric water content,
- C_ψ = the correction function described above,
- Θ_s = the saturated volumetric water content,
- e = the natural number (2.71828),
- Ψ = the negative pore-water pressure, and
- a, n, m = curve fitting parameters.

The 'a' parameter, which has units of kPa, is the inflection point of the volumetric water content function. It is generally slightly larger than the air-entry value. The parameter n controls the slope of the volumetric water content function and the m parameter controls the residual water content. The three parameters a, n, and m are determined as follows:

$$a = \Psi_i$$

$$m = 3.67 \ln \left(\frac{\Theta_s}{\Theta_i} \right)$$

$$n = \frac{1.31^{m+1}}{m\Theta_s} 3.72s\Psi_i$$

where:

Ψ_i = the suction pressure corresponding to the water content occurring at the inflection point of the curve, and
 s = the slope of the line tangent to the function that passes through the inflection point.

The Fredlund and Xing, 1994 method is only functional if you know values of a , n and m . In general, the a , n and m values can be determined using a fitting algorithm and applying it to measured data points. SEEP/W has this ability.

It is important to understand that this method is not intended to predict a volumetric water content function from grain-size curves, but was developed to obtain a smooth function over the complete range of negative pore-water pressure values (0 to 1,000,000 kPa).

Closed form option 2 (Van Genuchten, 1980)

In 1980, Van Genuchten proposed a four-parameter equation as a closed form solution for predicting the volumetric water content function. The governing equation is as follows:

$$\Theta_w = \Theta_r + \frac{\Theta_s - \Theta_r}{\left[1 + \left(\frac{\Psi}{a} \right)^n \right]^m}$$

where:

Θ_w = the volumetric water content,
 Θ_s = the saturated volumetric water content,
 Ψ = the negative pore-water pressure, and
 a, n, m = curve fitting parameters (note: “ a ” has units of pressure, not 1/pressure head as in some formulations of this equation)

Although the terminology of the a , n and m parameters are similar to those of Fredlund and Xing (1994), the definitions are slightly different. The a parameter in particular cannot be estimated by the air-entry value, but instead is a pivot point about which the n parameter changes the slope of the function. The parameter m affects the sharpness of the sloping portion of the curve as it enters the lower plateau. The Van Genuchten closed form method can only be used then the curve fit parameters are known, but there are some references to these values in the literature that can be applied in the model.

CAUTION: the units of the “ a ” value should be checked to make sure they are consistent between your data source and SEEP/W, which requires the units be in terms of pressure and not 1/pressure.

4.3 Soil material function measurement

Obtaining the saturated conductivity value of a soil is a fairly simple process that involves measuring a fixed head gradient across a saturated soil sample and then measuring the quantity of water that passes through at steady state conditions. For coarse grained soils this can be as simple as setting up a column of soil and using a pump or tap water to force water through. A standpipe manometer can be used at two fixed locations in the column to obtain the total head values needed to back-calculate the saturated conductivity value. For more fine grained soils, the rate of water expelled during an Oedometer test can be related to its saturated conductivity value. This is a useful approach as the saturated conductivity versus void ratio data can be plotted and used in subsequent sensitivity analysis relating to as built field conditions.

The hydraulic conductivity versus pore-water pressure relationship can be established directly from laboratory measurements. This involves measuring the hydraulic conductivity of soil samples at various negative pore-water pressure levels, but the process can often take very long due to the low conductivity values that can be present in partially saturated soil, which affects the time it takes to reach equilibrium conditions.

Making measurements of unsaturated hydraulic conductivity is a fairly complex and involved task. Difficulties associated with the measurements have been discussed by Brooks and Corey, 1966, and Green and Corey, 1971. In addition to the time for testing, the difficulties are generally related to problems with air diffusion and measuring small flow quantities.

The techniques for measuring unsaturated hydraulic conductivity have been documented by Klute, 1965. This document describes the fundamentals of the equipment and procedures involved. Another paper on the subject has been presented by Corey, 1957. Hillel, 1990, discusses the measurement of unsaturated hydraulic conductivity in situ.

Direct measurement of water content function

The water content of soil at a particular negative pore-water pressure can be measured with a commercially available apparatus known as a pressure plate cell. Figure 4-4 shows a schematic diagram of the cell.

It is important that the soil sample is placed in direct contact with a porous ceramic plate located on the bottom of the inner chamber. The ceramic plate acts like a semi-permeable membrane between the soil sample and the water filled reservoir at the bottom of the cell. Positive air pressure is applied to the top of the cell and increases the air pressure in the chamber. The increase in air-pressure causes pore-water from the soil sample to be pushed out through the ceramic plate and air enters the previously water-filled pores in the soil sample. It is very important that the air entering the soil is only from the air chamber and not a result of diffusion through the ceramic plate at the bottom of the cell. This is achieved by using a high air-entry plate, which will allow water to readily flow through the plate, but restrict the flow of air up to a certain maximum pressure.

Each incremental increase in air pressure results in an incremental decrease in water content within the soil sample. Equilibrium conditions must be established following each incremental increase in air pressure, then the entire cell is weighed and the change in weight is recorded. At the end of the test, the changes in cell weight are used together with the final dry weight of the sample to compute the water content of the sample that existed at each of the various applied pressures. In this manner, the volumetric water content versus negative pore-air pressure relationship can be developed.

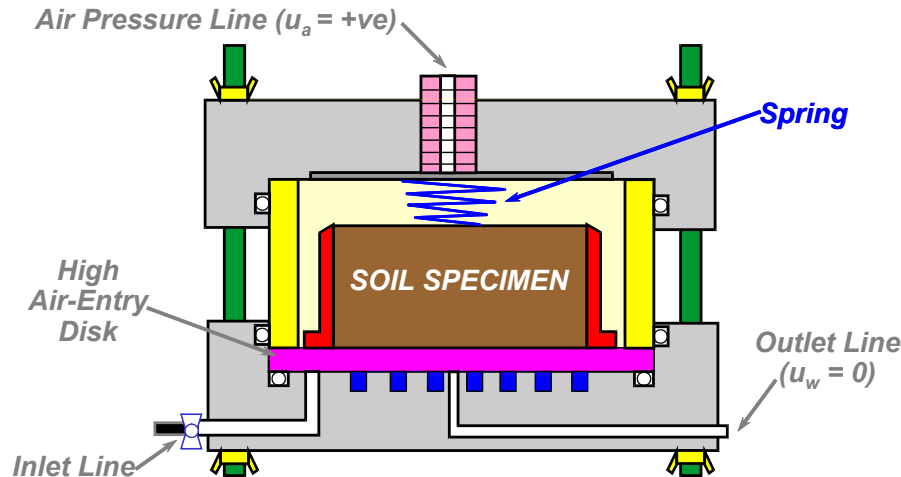


Figure 4-4 Schematic cross-section view of a pressure plate cell apparatus

4.4 Coefficient of volume compressibility

The coefficient of volume compressibility m_v is the slope of the volumetric water content function in the positive pore pressure range. The coefficient characterizes the volume of water stored or released from the soil when the pore-water pressure changes. More fundamentally, the coefficient of volume compressibility embodies the compressibility of the soil structure and water. The coefficient of volume compressibility can be calculated from specific storage, which is given by:

$$S_s = \rho g (\alpha + n\beta) = \rho g (m_v)$$

where:

- ρ = the water density,
- g = the gravitational constant,
- α = the compressibility of the soil structure,
- n = the soil porosity, and
- β = the compressibility of the water.

Water can be assumed incompressible for most geotechnical applications. The coefficient of volume compressibility is often assumed equivalent to the coefficient obtained from a 1D compression test; however, this assumption is not strictly valid unless a constrained flow system is being simulated. In that case, or as an approximation, the coefficient of volume change can be calculated as:

$$m_v = \frac{1}{M} = \frac{a_v}{1 + e_o}$$

where:

- M = the modulus of elasticity in confined compression. It is also given by:
- a_v = the coefficient of compressibility, and

e_0 = the initial void ratio.

The metric units for the coefficient of volume compressibility are 1/kPa and the value generally ranges from $1e-6$ 1/kPa to $1e-3$ 1/kPa for most soils.

4.5 Hydraulic conductivity

The ability of a soil to transport or conduct water under both saturated and unsaturated conditions is reflected by the hydraulic conductivity function. In a saturated soil, all the pore spaces between the solid particles are filled with water. Once the air-entry value is exceeded, air enters the largest pores and the air-filled pores become non-conductive conduits to flow and increase the tortuosity of the flow path as shown schematically in Figure 4-5. As a result, the ability of the soil to transport water (the hydraulic conductivity) decreases. As pore-water pressures become increasingly more negative, more pores become air-filled and the hydraulic conductivity decreases further. By this description, it is clear that the ability of water to flow through a soil profile depends on how much water is present in the soil, which is represented by the volumetric water content function. Actually measuring the hydraulic conductivity function is a time-consuming and expensive procedure, but the function can be readily developed using one of several predictive methods that utilize either a grain-size distribution curve or a measured volumetric water content function and the saturated hydraulic conductivity. SEEP/W has built-in predictive methods that can be used to estimate the hydraulic conductivity function once the volumetric water content function and a K_{sat} value have been specified.

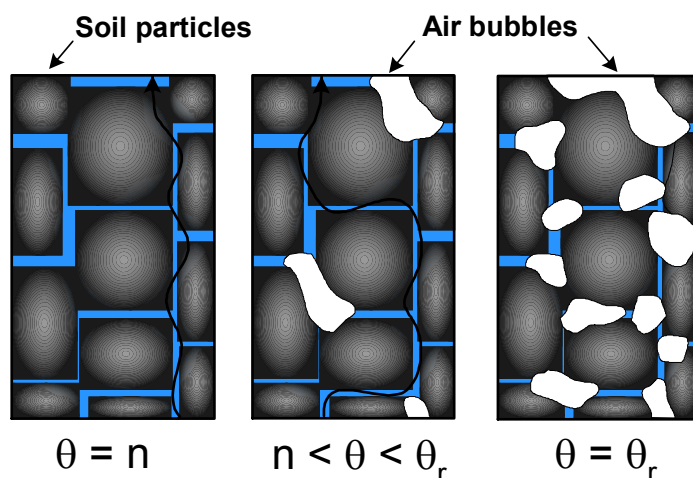


Figure 4-5 Availability of water filled flow paths from saturation to residual

A hydraulic conductivity function should be specified for all materials in a problem that will have an unsaturated zone. Even if the hydraulic conductivity function is an estimate, the results will be more realistic than if the function is entered as a flat horizontal line. In a unsaturated seepage analysis with negative surface fluxes (such as evaporation) where negative pressures can become extreme, the conductivity function should be defined over a pressure range that exceeds several hundred thousand kPa (or equivalent) of negative pressure. If the function is not defined over the full range, the lowest specified value will be used for any increasingly negative pressures.

Adopting a perfectly flat hydraulic conductivity function (i.e., a constant conductivity) for an unsaturated soil can lead to unrealistic results. The phreatic surface may end up at an unrealistic position, and the proportion of flow through the unsaturated zone may be too high. This occurs because with a horizontal conductivity function, water can flow through the unsaturated zone with the same ease as through the

saturated zone. In other words, for a given constant head differential, the volume of flow is the same in the unsaturated zone as in the saturated zone when the hydraulic conductivities in the two zones are the same. In general, water cannot flow through unsaturated soil with the same ease as through saturated soil, because the unsaturated hydraulic conductivity is lower than that of a saturated soil.

To illustrate the effect of assuming that hydraulic conductivity is independent of negative pore-water pressure (i.e., a perfectly flat conductivity function), consider the example of seepage flow through a rectangular screened box, as shown in Figure 4-6. Initially, the box is filled with clay. The phreatic surface will have the form of a hyperbolic curve (top of figure). In the bottom of the figure, the box is enlarged, the upstream half is filled with clay, and the downstream half is filled with sand. The sand is assigned a perfectly flat hydraulic conductivity function. In this case, the phreatic surface in the clay will be at a lower position. The reason for this is that a significant portion of the flow passes through the unsaturated sand. Since the resistance to flow is the same in the unsaturated sand as in the saturated sand, there is no reason for the sand to be saturated in order to conduct the water. Intuition alone suggests that this is not the case. The phreatic surface in the clay should be approximately the same in both configurations, and the seepage that arrives at the clay-sand contact should flow vertically down the contact and then horizontally along the bottom of the box to the exit point at the lower right corner.

To model the clay-sand configuration, the sand needs to be assigned a very steep function, so that as soon as the sand desaturates, the hydraulic conductivity drops dramatically. This ensures that there is no significant flow in the unsaturated sand. However, a nearly vertical function may cause convergence difficulties. A compromise would be to use a moderately steep hydraulic conductivity function, which would eliminate the majority of the flow in the unsaturated sand and yet produce a reasonable result. It would certainly be closer to the correct solution than for the first case where the sand has a perfectly flat hydraulic conductivity function.

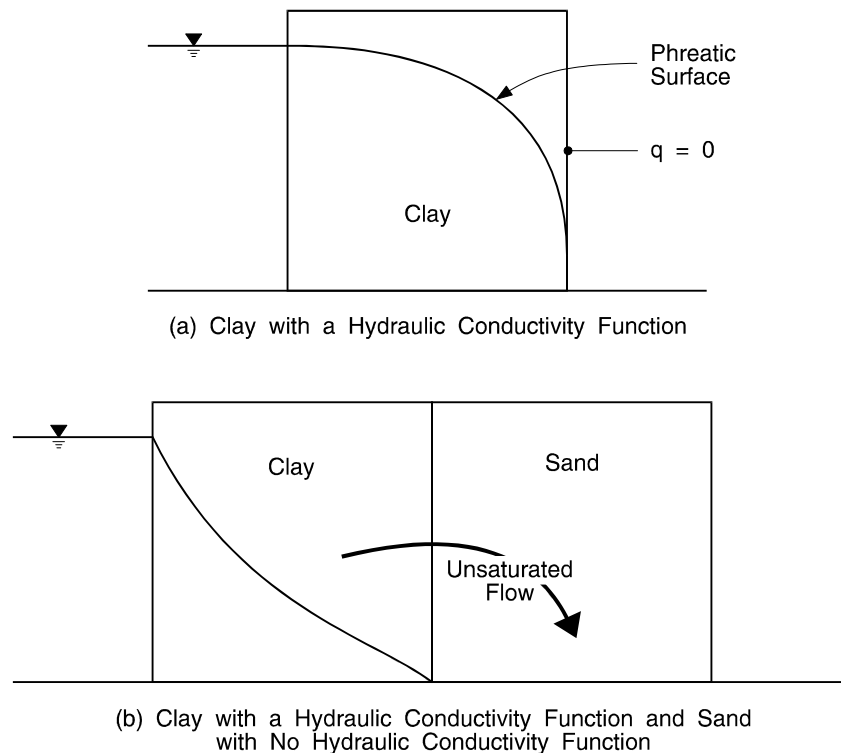


Figure 4-6 Comparison of flow with and without a K function

Coarse granular materials essentially have an infinitely steep (vertical) hydraulic conductivity function when unsaturated. The soil de-saturates completely when the pore-water pressure is zero or negative; consequently, no flow passes through such a soil when it is unsaturated. As a result, the hydraulic conductivity in the unsaturated zone should be infinitely low.

Whenever a problem contains a coarse granular soil that ideally has a near vertical hydraulic conductivity function when unsaturated, it is necessary to ask the question, "Does the material contribute to the dissipation of the head?" If it does not, then consideration should be given to excluding the material from the analysis. In the clay-sand box example, the sand may not contribute to dissipating the head. Consequently, a reasonable solution might be obtained by excluding the sand from the analysis and treating the vertical contact between the two materials as a boundary. The decision as to whether the sand should be included in the analysis must also be made in light of the question, "Does the negative pore-water pressure in the sand contribute to increasing the gradient in the clay?" If it does, the sand must be included in the analysis.

The accuracy with which the hydraulic conductivity needs to be specified depends to some extent on the objective of the analysis. If the primary objective is to compute the distribution of pore-water pressure, then an approximate function may be adequate. On the other hand, if the objective of the analysis is to make reliable time predictions, then it may be necessary to define the storage and hydraulic conductivity with the assistance of laboratory tests.

The level of effort required to define the material functions can be evaluated by performing several analyses with different assumed functions. Performing such a sensitivity analysis can greatly increase the confidence level of the computed results.

In summary, a hydraulic conductivity function must be specified for each material included in an analysis, even if the function is only an approximation.

An approximated curved conductivity relationship in the unsaturated zone results in a much better solution than using a straight, horizontal line.

4.6 Frozen ground hydraulic conductivity

SEEP/W can be used in conjunction with TEMP/W to model transient seepage behavior in frozen, partially frozen, or actively freezing ground. An example of seepage flow being diverted around an active freezing region is illustrated in Figure 4-7 and Figure 4-8. This type of analysis is controlled by the TEMP/W program, but information is passed back and forth between the two solvers as the solution progresses. TEMP/W requires knowledge of the water content in the soil as well as seepage velocities, so that it can compute the convective heat transfer associated with flowing water. SEEP/W on the other hand requires knowledge of the soil temperature so that it can estimate the reduction in hydraulic conductivity associated with pore-water becoming pore-ice. This estimate is based on knowledge of the soil's unfrozen water content function.

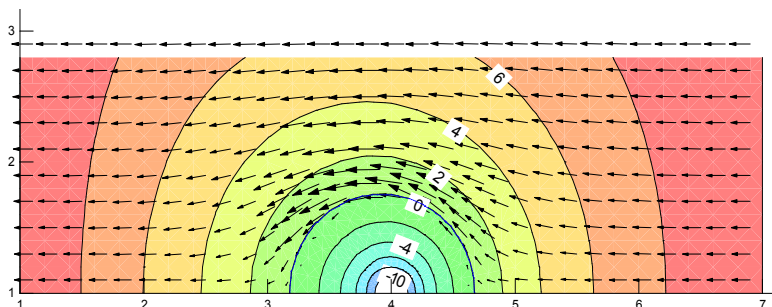


Figure 4-7 Seepage diversion around actively freezing soil region (Temperature contours from TEMP/W displayed)

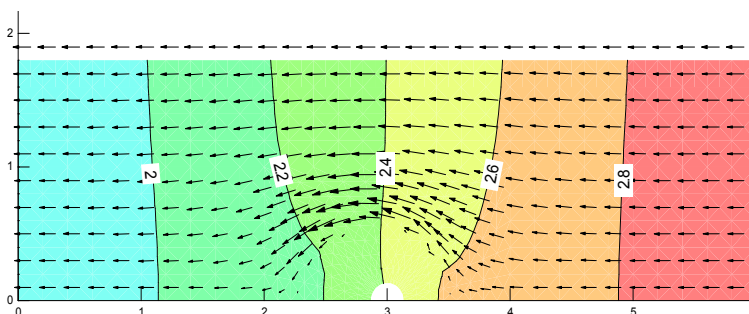


Figure 4-8 Seepage diversion around actively freezing soil region (Head contours from SEEP/W displayed)

The unfrozen water content function relates the amount of unfrozen water to a temperature below freezing and its curve is very similar in appearance to a soil water characteristic curve when plotted on a semi-log scale. The unfrozen water content curve can serve three purposes. It can be used to determine the freezing point depression for pore-water in soils at a given water content below saturation; it can be used to determine the amount of water that remains unfrozen at any given temperature below freezing; and the slope of the curve determines how much latent heat is added to the system by the phase change during the heat and mass transfer analysis.

Ideally, a soil freezing curve should be measured, but this is difficult to do. It is possible to estimate the curve using a measured soil water characteristic (storage) curve and the Clapeyron equation, which relates changes in suction to change in temperature based on equilibrium thermodynamics. Analysis of the Gibbs free energy for any two phases in equilibrium can be used to derive the Clapeyron equation, which relates how the equilibrium pressure changes with a change in temperature. The reduced form of the Clapeyron equation as applied to the soil freezing scenario is given by Black and Tice (1989) as follows:

$$\text{Equation 4-1} \quad \Delta\Psi = -1110\Delta T$$

where:

$\Delta\Psi$ = the soil matric suction (kPa), and

T = the soil temperature below zero Celsius.

The constant value equal to 1110 kPa/°C combines the latent heat of fusion value, specific volume, and the conversion between the freezing temperature of water in Kelvin and degrees Celsius.

If the soil temperature below zero Celsius is passed to SEEP/W from TEMP/W, then the seepage program can use Equation 4-1 to estimate what the approximate frozen condition suction would be such that this

suction is used to determine the hydraulic conductivity at each gauss point with freezing temperatures (Newman, 1996).

Seepage analysis in freezing ground can be very complicated, especially in the direct vicinity of the phase change region. At this location it is possible for certain types of soil to experience “cryogenic suction” which results in very steep pressure gradients that can draw water towards the freezing front where it can accumulate and cause frost heave. In the SEEP/W model, this phenomenon is not accounted for. While the suctions are estimated based on temperature and used to determine frozen ground hydraulic conductivity, they are not directly coupled with the thermal equation and therefore only change due to the solution of the seepage partial differential equation.

4.7 Conductivity function estimation methods

The difficult task of measuring the unsaturated hydraulic conductivity function directly is often overcome by predicting the unsaturated hydraulic conductivity from either a measured or predicted volumetric water content function, such as the one illustrated in Figure 4-1. Consequently, this is the preferred approach if a suitable predictive model is available. These estimation methods generally predict the shape of the function relative to the saturated conductivity value which is easily obtained.

SEEP/W has three separate methods built into the model that can be used to predict unsaturated hydraulic conductivity functions using either a measured or estimated volumetric water content function or a saturated hydraulic conductivity function.

Method 1 (Fredlund et al, 1994)

This method consists of developing the unsaturated hydraulic conductivity function by integrating along the entire curve of the volumetric water content function. The governing equation of this method is:

$$k_w = k_s \frac{\sum_{i=j}^N \frac{\Theta(e^y) - \Theta(\Psi)}{e^{y_i}} \Theta'(e^{y_i})}{\sum_{i=1}^N \frac{\Theta(e^y) - \Theta_s}{e^{y_i}} \Theta'(e^{y_i})}$$

where:

- k_w = the calculated conductivity for a specified water content or negative pore-water pressure (m/s),
- k_s = the measured saturated conductivity (m/s),
- Θ_s = the volumetric water content,
- e = the natural number 2.71828,
- y = a dummy variable of integration representing the logarithm of negative pore-water pressure,
- i = the interval between the range of j to N ,
- j = the least negative pore-water pressure to be described by the final function,
- N = the maximum negative pore-water pressure to be described by the final function,
- Ψ = the suction corresponding to the j^{th} interval, and
- Θ' = the first derivative of the equation ...

$$\Theta = C(\Psi) \frac{\Theta_s}{\left\{ \ln \left[e + \left(\frac{\Psi}{a} \right)^n \right] \right\}^m}$$

where:

- a = approximately the air-entry value of the soil,
 n = a parameter that controls the slope at the inflection point in the volumetric water content function,
 m = a parameter that is related to the residual water content, and
 $C(\Psi)$ = a correcting function defined as

Equation 4-2

$$C(\Psi) = 1 - \frac{\ln \left(1 + \frac{\Psi}{C_r} \right)}{\ln \left(1 + \frac{1,000,000}{C_r} \right)}$$

where:

- C_r = a constant related to the matric suction corresponding to the residual water content.

A typical value is about 1500 kPa. The value 1,000,000 in the above equation corresponds to the matric suction (in kPa) at which there is zero moisture remaining in the soil in a liquid or vapor phase.

Method 2 (Green and Corey, 1971)

The Green and Corey equation is:

Equation 4-3

$$k(\Theta)_i = \frac{k_s}{k_{sc}} \cdot \frac{30 T^2}{\mu g \eta} \cdot \frac{\xi^p}{n^2} \cdot \sum_{j=i}^m \left[(2j + 1 - 2i) h_i^{-2} \right]$$

where:

- $K(\Theta)_i$ = the calculated conductivity for a specified water content or negative pore-water pressure (cm/min),
 $\frac{k_s}{k_{sc}}$ = the matching factor (measured saturated conductivity / calculated saturated conductivity),
 i = the last water content class on the wet end (e.g. $i=1$ identifies the pore class corresponding to the lowest water content, and $i=m$ identifies the pore class corresponding to the saturated water content),
 h_i = the negative pore-water pressure head for a given class of water-filled pores (cm of water),
 n = the total number of pore classes between i and m ,
 Θ = volumetric water content (cm³/cm³),
 T = surface tension of water (Dyn/cm),
 ξ = the water-saturated porosity,

- η = the viscosity of water (g/cm •s-1),
 g = the gravitational constant (cm/s-1),
 μ = the density of water (g/cm3), and
 p = a parameter that accounts for the interaction of pore classes.

The following are some suggested values of p given by various authors: Marshall (1958): 2.0; Millington and Quirk (1961): 1.3; and Kunze, Vehara and Graham (1968): 1.0.

The shape of the conductivity function is controlled by the term:

$$\sum_{j=i}^m \left[(2j + 1 - 2i) h_i^{-2} \right]$$

in Eq. 4.3.

The term:

$$\frac{30 T^2}{\mu g \eta} \cdot \frac{\xi^p}{n^2}$$

is a constant for a particular function and can be taken to be 1.0 when determining the shape of the hydraulic conductivity function. This is the assumption made in SEEP/W.

SEEP /W first computes the hydraulic conductivity at the zero pressure value using the equation,

$$k_{sc} = \sum_{j=i}^m \left[(2j + 1 - 2i) h_i^{-2} \right]$$

The saturated conductivity k_s is a user-defined value in SEEP /W. When k_s is specified, the entire conductivity function is moved up or down by a constant ratio of k_s / k_{sc} .

In summary, SEEP /W uses the Green and Corey equation to estimate the shape of the conductivity function and then moves the curve up or down so that the function passes through the user-specified value of k_s .

Method 3 (Van Genuchten, 1980)

Van Genuchten (1980) proposed the following closed form equation to describe the hydraulic conductivity of a soil as a function of matric suction:

$$k_w = k_s \frac{\left[1 - (a\Psi^{(n-1)}) \left(1 + (a\Psi^n)^{-m} \right) \right]^2}{\left((1 + a\Psi^n)^{\frac{m}{2}} \right)}$$

where:

- k_s = saturated hydraulic conductivity,
 a, n, m = curve fitting parameters,

$$n = 1/(1-m), \text{ and}$$

$$\psi = \text{required suction range.}$$

From the above equations, the hydraulic conductivity function of a soil can be estimated once the saturated conductivity and the two curve fitting parameters, a and m are known.

Van Genuchten (1980) showed that the curve fitting parameters can be estimated graphically based on the volumetric water content function of the soil. According to van Genuchten, the best point to evaluate the curve fitting parameters is the halfway point between the residual and saturated water content of the volumetric water content function.

The slope of the function can be calculated as:

$$S_p = \frac{1}{(\Theta_s - \Theta_r)} \left| \frac{d\Theta_p}{d(\log \Psi_p)} \right|$$

where:

$$\Theta = \text{the saturated and residual volumetric water contents respectively,}$$

$$\Theta_p = \text{the volumetric water content at the halfway point of the volumetric water content function, and}$$

$$\Psi_p = \text{the matric suction at the same point.}$$

Van Genuchten (1980) proposed the following formula to estimate the parameters m and a when S_p is calculated:

$$m = 1 - \exp(-0.8S_p)$$

for S_p between 0 and 1;

$$m = 1 - \frac{0.5755}{S_p} + \frac{0.1}{S_p^2} + \frac{0.025}{S_p^3}$$

for $S_p > 1$; and

$$a = \frac{1}{\Psi} \left(2^{\frac{1}{m}} - 1 \right)^{(1-m)}$$

4.8 Interface model parameters

The interface model allows you to assign a material model to a line and to give that line a thickness. In a seepage application, you may want to use an interface model to simulate a thin liner or a wick drain.

When you assign an interface model to a line you must give it hydraulic conductivity values that are both normal and tangent to the direction of the line as shown in Figure 4-9.

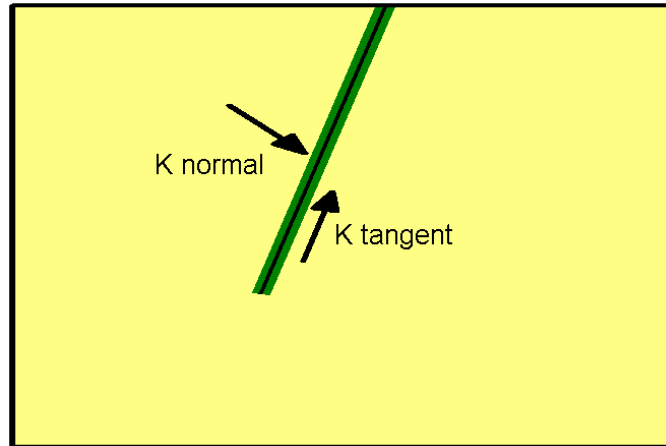


Figure 4-9 Illustration of interface model assigned to a line

4.9 Sensitivity of hydraulic results to material properties

How sensitive is a model to changes to the air-entry value, the slope of the function, the residual volumetric water content and the saturated hydraulic conductivity? The effect of altering each of the four material property functions is highlighted below, through a series of steady-state and transient analyses where only one parameter is changed at a time to clearly evaluate the influence of each parameter. Figure 4-10 shows a cross-section of a system where both saturated and unsaturated conditions exist. This cross-section represents a two-dimensional view of a flow system in which water from a canal passes through an unconfined, homogeneous, fine sand aquifer and is collected in a series of collection wells located along the right edge of the cross-section.

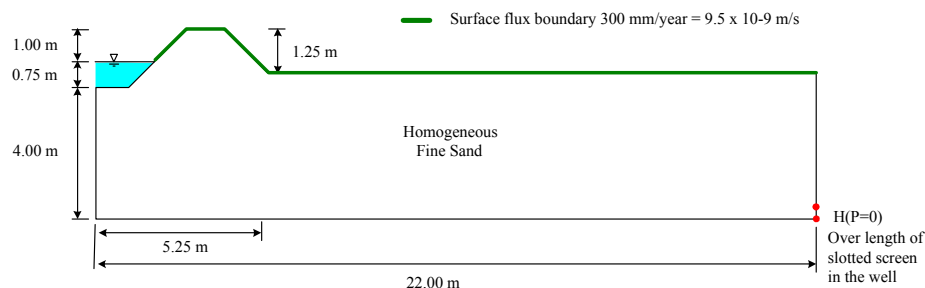


Figure 4-10 Diagram of system used in sensitivity analysis

Changes to the air-entry value (AEV)

To show how sensitive the model is to changes in air-entry value, a steady-state simulation was conducted where the AEV of a function was increased from 3 to 10 kPa. A volumetric water content function is not required for a steady-state analysis, however it is good practice to always make changes to the volumetric water content function when conducting a sensitivity analysis, and then let the changes be reflected in the hydraulic conductivity function by recalculating the function from the value of the saturated hydraulic conductivity and the newly specified volumetric water content function. Figure 4-11 shows the adjusted volumetric water content function and the predicted hydraulic conductivity functions used in the simulations. It is important to note that if the K-function was presented on a log-log plot (hydraulic

conductivity functions are usually presented in the literature on a log-log scale), the effect of increasing the air-entry value and adjusting the rest of the curve accordingly would appear to also steepen the slope of the function. In VADOSE/W however, the functions are always presented on a log-arithmetic scale and the AEV can be increased while the slope of both functions remain similar in shape. Figure 4-12 and Figure 4-13 show the VADOSE /W modeling results for both the 3 kPa and 10 kPa AEV simulations respectively.

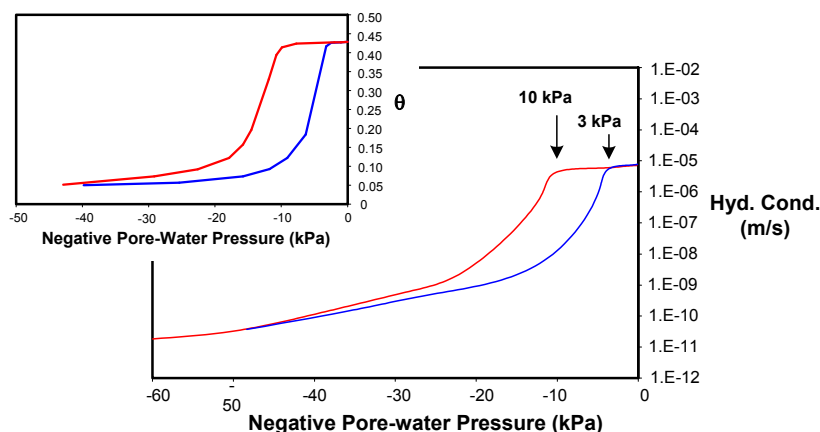


Figure 4-11 Material property functions used in the AEV sensitivity analyses

To compare the results in Figure 4-12 and Figure 4-13, it is probably easiest to consider the height of the capillary fringe that is emphasized in the magnified sections. A dimension arrow has been superimposed to show the extent of capillary rise that develops for both simulations. The capillary rise is the height above the water table where negative pore-water pressures exist, but the soil remains saturated due to capillary tension. The air-entry value, when converted from pressure (kPa) to a pressure head (m), is approximately equal to the height of the capillary fringe. In the capillary fringe, water is transported through the soil at a rate equal to the saturated hydraulic conductivity, so more water can be transported in a larger capillary fringe than in a smaller one. As an interesting aside, note how the structure of the model (i.e., the downstream side of the berm) controls the shape of the unsaturated flow system. Even though saturated flow occurs in the berm, the water table is still at depth and negative pore-water pressures exist on the downstream face so a seepage face never develops.

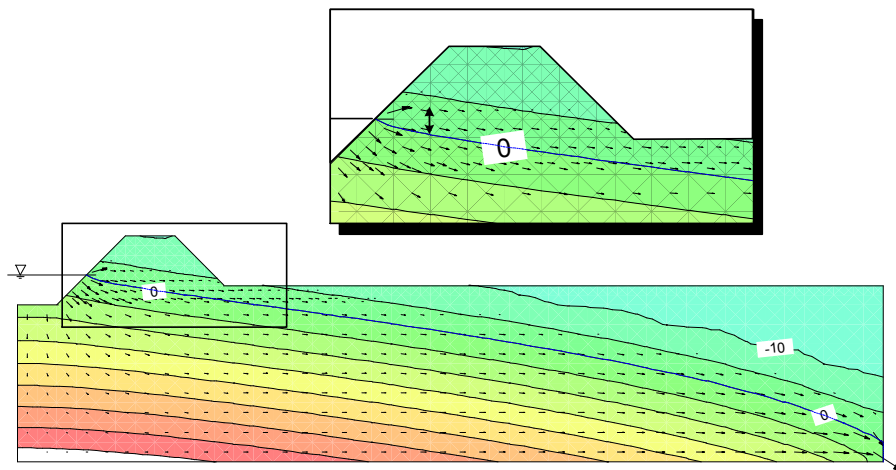


Figure 4-12 Pressure contours and flow vectors for a 3 kPa AEV material

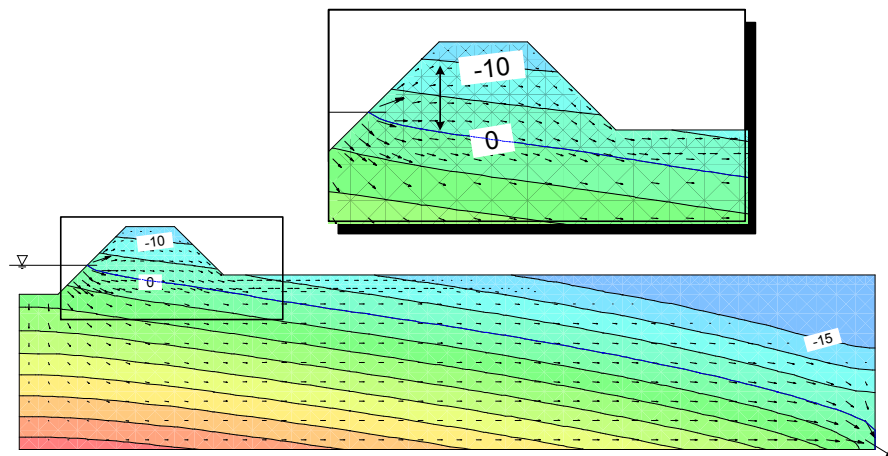


Figure 4-13 Pressure contours and flow vectors for a 10 kPa AEV material

Changes to the saturated hydraulic conductivity

Six steady state simulations were conducted to evaluate the effect of changing the saturated hydraulic conductivity (Table 4-1). Basically, the first three simulations compared the effect of both increasing and decreasing the saturated hydraulic conductivity and the last three showed the effect of conducting a saturated hydraulic conductivity sensitivity analysis with the added influence of an applied surface flux boundary condition.

Table 4-1 Summary of simulations for Ksat sensitivity analyses

Simulation	Ksat (m/s)	Surface Flux (m/s)
1	7.5×10^{-4}	none
2	7.5×10^{-6}	none
3	7.5×10^{-8}	none
4	7.5×10^{-4}	9.5×10^{-9}
5	7.5×10^{-6}	9.5×10^{-9}
6	7.5×10^{-8}	9.5×10^{-9}

The three hydraulic conductivity functions that were used in the simulations are presented in Figure 4-14. The general shape of the function remained unchanged while the saturated hydraulic conductivity was adjusted.

Figure 4-15 to Figure 4-17 show the results obtained from simulations 1 to 3 (see Table 4-1). A surface flux was not applied and the Ksat was varied by two orders of magnitude between each simulation. The resulting total head contours and total flux values that were determined near the well screen are included in the figures. One of the most significant comparisons to make is with respect to the total head contours. Altering the saturated hydraulic conductivity does not alter the shape of the flow net, so the total head contours should be and are the same. The only obvious difference between the results can be found in the value associated with the flux section near the well. The flux varies along the same order of magnitude that the Ksat was varied, so increasing the saturated hydraulic conductivity results in a greater flow rate to the well and decreasing it reduces the amount of flow to the well.

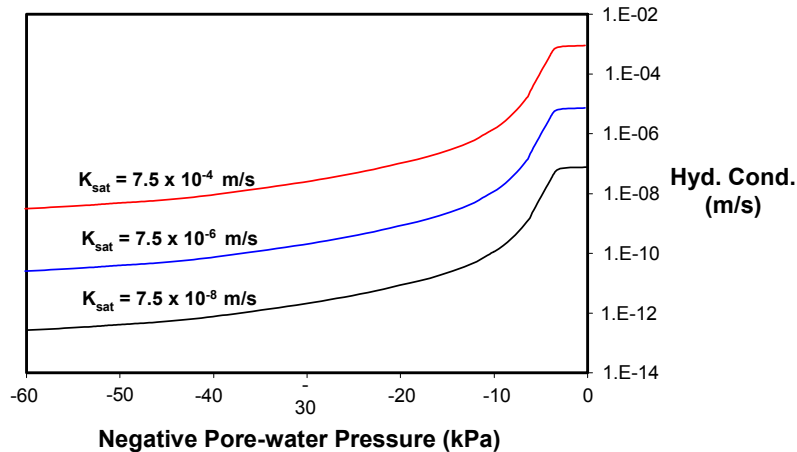


Figure 4-14 Hydraulic conductivity functions used for Ksat sensitivity analyses

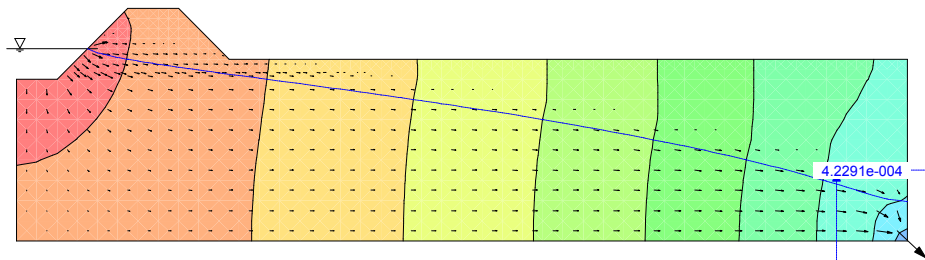


Figure 4-15 Simulation #1 total head contours (Ksat 7.5×10^{-4} m/s)

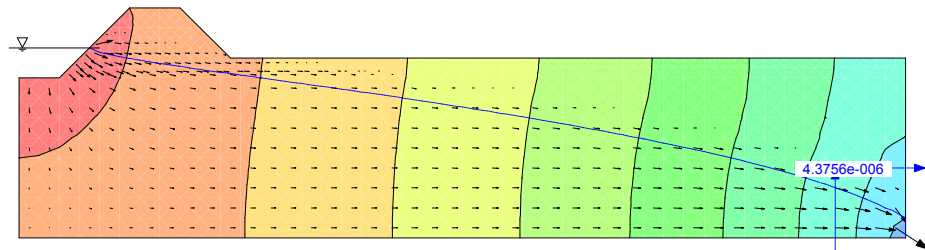


Figure 4-16 Simulation #2 total head contours (Ksat = 7.5×10^{-6} m/s)

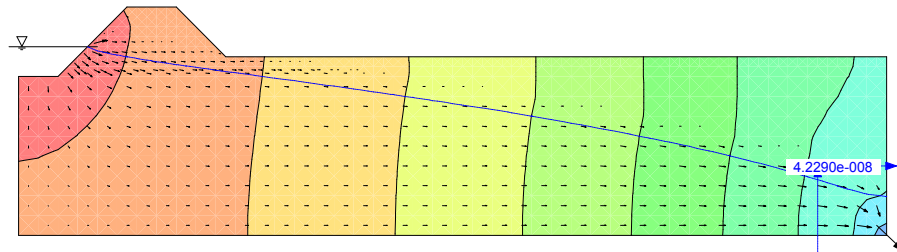


Figure 4-17 Simulation #3 total head contours (Ksat = 7.5×10^{-8} m/s)

The results presented in the above figures did not include a surface flux boundary condition. Results obtained from sensitivity analyses (regarding Ksat) where surface fluxes are applied, are not nearly as easy to compare. The next three figures show the results from simulations 4 to 6 from Table 4-1. In these cases, a surface flux boundary condition of 9.5×10^{-9} m/s (equivalent to 300 mm/year net infiltration)

was applied over the surface and the K_{sat} was once again varied by two orders of magnitude between simulations.

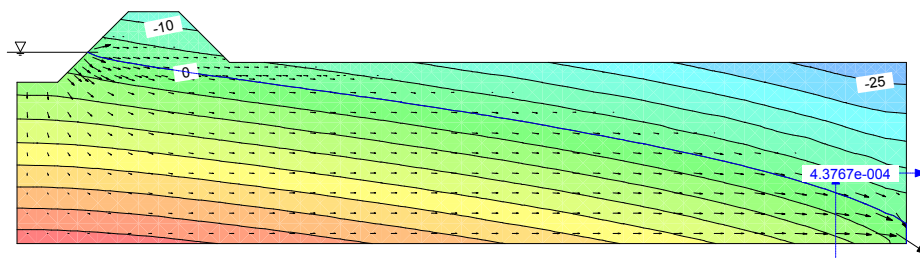


Figure 4-18 Simulation #4 pressure head contours ($K_{sat} = 7.5 \times 10^{-4}$ m/s)

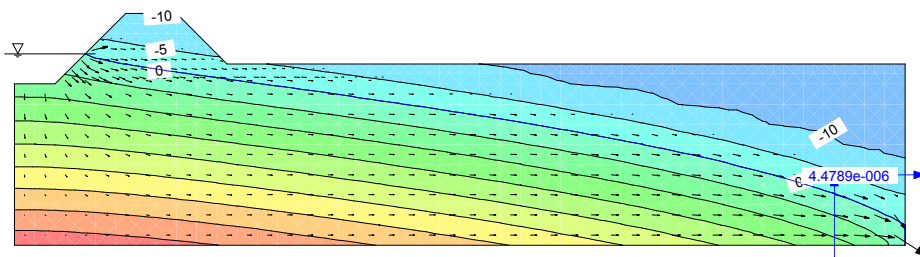


Figure 4-19 Simulation #5 pressure head contours ($K_{sat} = 7.5 \times 10^{-6}$ m/s)

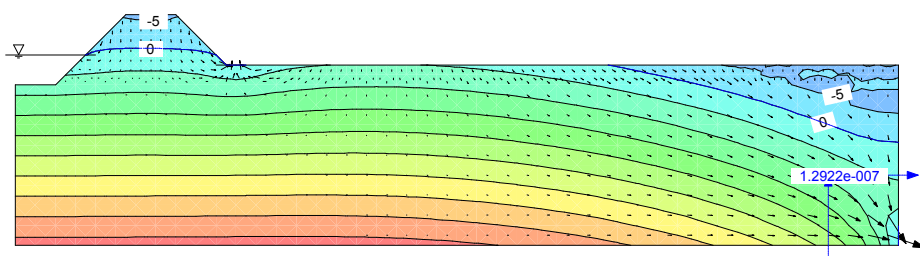


Figure 4-20 Simulation #6 pressure head contours ($K_{sat} = 7.5 \times 10^{-8}$ m/s)

The pressure head contours (and consequently the total head contours) are very different between the three simulations where a surface flux was applied. This time the system has two water sources to contribute to the total flow; the upstream ponded water and the applied surface flux. The effect on the flow system is minimal in Figure 4-18 because the K_{sat} is much greater than the applied surface flux (10^{-4} compared to 10^{-9} m/s). As the difference between the K_{sat} and the applied flux gets smaller, the effect on the pressure profiles becomes more significant. In Figure 4-20, the system is close to becoming fully saturated and a seepage face develops on the downstream side of the berm. The rate at which the water can leave the system (i.e., the well), is still controlled by the saturated hydraulic conductivity, but in the case of analysis with a surface flux applied, the total flux value leaving the profile is greater.

Changes to the slope of the VWC function

In a steady-state analysis, the amount of flow into the system corresponds to the flow rate out of the system. However, in a transient seepage analysis the flow into the system may differ from the flow out of the system because the system stores or releases water. It therefore becomes necessary to account for the change of water stored within a soil profile with time. The amount of water stored or retained is a function of the pore-water pressures and the characteristics of the soil structure and is described by the volumetric water content function. A transient analysis can be used to evaluate the effect of altering the slope of the

volumetric water content function (mw). In any transient analysis there are two main considerations; how fast the water is flowing (a function of the hydraulic conductivity) and how much water is flowing (a function of the change in storage and the amount of water in the system). As a result, both material property functions must be defined. Storage is the amount of water that remains in the pores of a soil under negative pore-water pressures. If the slope of the VWC function is flat, the change in volumetric water content for increasingly negative pore-water pressures would be less than for a soil with a steeper function. Figure 4-21 shows the volumetric water content and hydraulic conductivity functions used for the sensitivity analysis regarding the slope of the VWC function. Creating a function with a flatter slope represents a soil which is non-uniform and has a larger distribution of pore sizes. The air-entry value (a function of the largest pore size) has not been changed, nor has the residual water content (a function of the smallest pore size). The modifications were made to the VWC function and the changes were then reflected in the hydraulic conductivity function through the use of predictive methods.

In order to obtain initial head conditions, steady-state analyses were conducted using both the modified and unmodified material property functions. A pond depth of 0.75m was included for the steady-state analyses and was then removed for the start of the transient analyses, allowing the system to drain into the well for 40 days. The results from both transient analyses are presented in Figure 4-22.

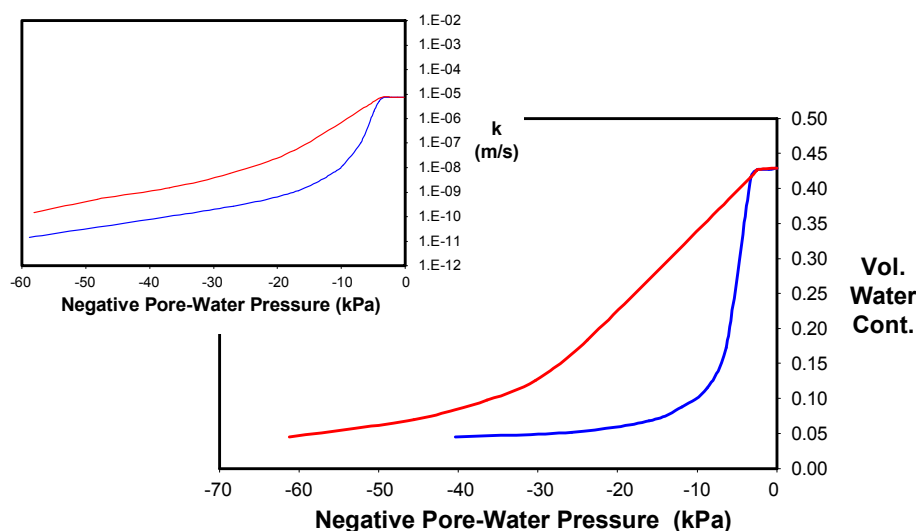


Figure 4-21 Material property functions used in the sensitivity analyses for changes to the slope of the storage function

One way to evaluate the effect of altering the slope of the VWC function is to compare how long it takes each simulation to lower the water table to the same elevation. As can be seen in Figure 4-22, it only took 22 days of drainage, using the modified function, to lower the $P=0$ contour (water table) to the same elevation as that of the unmodified function after 40 days of drainage. The time difference can be explained in part by comparing water content profiles taken at the same location. Figure 4-22 shows the initial water content for both simulations as a vertical, solid black line. The red line indicates the water content profile after 22 days of drainage using the modified function and the blue line indicates the water content profile after 40 days of drainage using the unmodified function (the one with the steeper slope). The amount of water removed from the system for each soil type can be estimated as the area between the black line and the red or blue line respectively. The water content profile of the modified soil (red) shows that the soil is wetter, having stored more water in the unsaturated zone than the unmodified soil (blue). As a result, the amount of water released from the system is less than that of the unmodified VWC

function. The majority of the water removed for both soils was through the saturated flow system, and since the K_{sat} remained unaltered between the simulations, it took less time to drain less water.

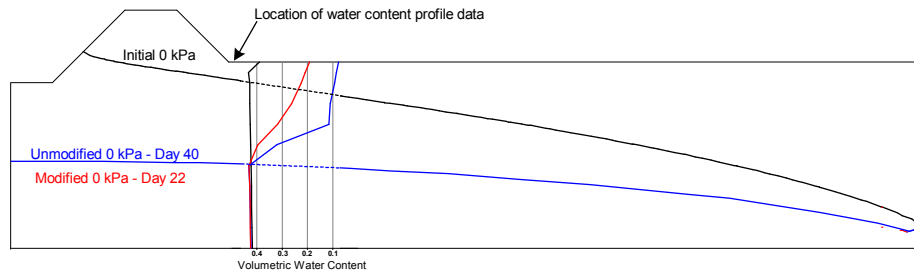


Figure 4-22 Location of $P=0$ kPa contour and water content profiles for the modified function (day 22) and unmodified function (day 40)

Changes to the residual volumetric water content

The last feature of the volumetric water content function to evaluate in the sensitivity analysis is the residual water content. The effect of changing the residual volumetric water content does not alter the hydraulic conductivity function, so only the VWC function was adjusted such that the residual water content for the modified function was much greater than the unmodified function as shown in Figure 4-23.

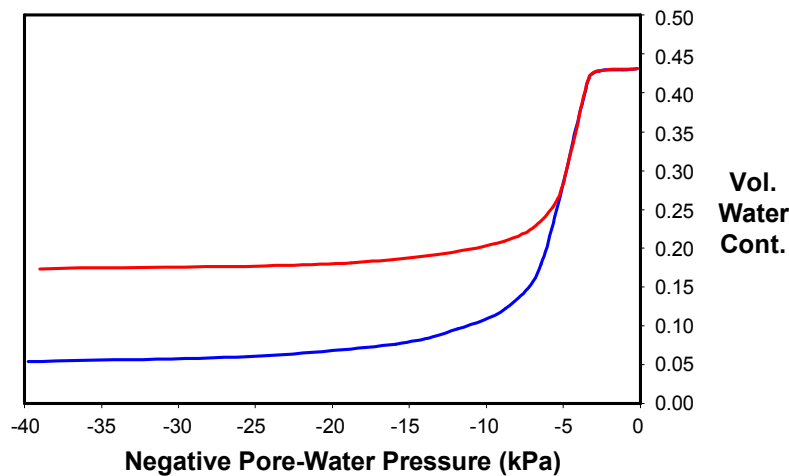


Figure 4-23 Modified and unmodified residual water contents used in analyses

Both steady state and transient analyses were conducted in a manner similar those described in the last section in terms of having the pond in place for the steady-state simulation and then letting the system drain over a 40 day period. Intuitively, altering the volumetric water content to have higher volumetric water content at residual should result in a wetter unsaturated profile. To confirm this thinking, the length of time that it took to lower the $P=0$ pressure contour (water table) to the same level as in Figure 4-22 was determined. The results are presented in Figure 4-24.

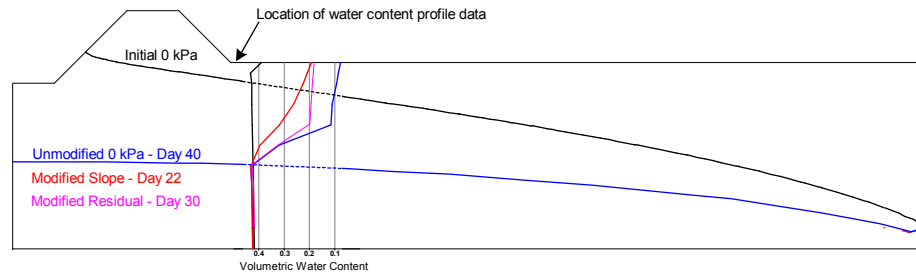


Figure 4-24 Location of $P=0$ kPa contours and water content profiles for the modified slope function (day 22), the modified residual (day 30) and the unmodified function (day 40)

With the completely unmodified function, it took 40 days to lower the $P=0$ contour (water table) to the location represented in Figure 4-24. It took the function with the shallower slope 22 days to have the $P=0$ contour lower to the same elevation. Increasing the residual water content of the unmodified function resulted in the $P=0$ contour reaching the same elevation after 30 days.

Therefore, increasing the residual water content of the volumetric water content function resulted in a wetter profile above the $P=0$ contour (as shown by the pink water content profile) than that of the unmodified function (as shown by the blue water content profile). Less water was released from the system and so it took less time to release the water. The greatest amount of water storage results in the least amount of water being released from the system. This occurred using the volumetric water content function with the shallow slope.

5 Boundary Conditions

5.1 Introduction

Specifying conditions on the boundaries of a problem is one of the key components of a numerical analysis. This is why these types of problems are often referred to as “boundary-valued” problems. Being able to control the conditions on the boundaries is also what makes numerical analyses so powerful.

Solutions to numerical problems are a direct response to the boundary conditions. Without boundary conditions it is not possible to obtain a solution. The boundary conditions are, in essence, the driving force. What causes seepage flow? It is the hydraulic total head difference between two points or some specified rate of flow into or out of the system. The solution is the response inside the problem domain to the specified conditions on the boundary.

Sometimes specifying conditions is fairly straightforward, such as defining the conditions that exist along the bottom of a reservoir. It is simply the elevation head at the top of the reservoir. Many times, however, specifying boundary conditions is complex and requires some careful thought and planning. Sometimes the correct boundary conditions may even have to be determined through an iterative process, since the boundary conditions themselves are part of the solution, as for instance along a seepage face. The size of the seepage face is not known and needs to be determined from the solution. Furthermore, the conditions on the boundaries may change with time during a transient analysis, which can also add to the complexity.

Due to the extreme importance of boundary conditions, it is essential to have a thorough understanding of this aspect of numerical modeling in order to obtain meaningful results. Most importantly, it is essential to have a clear understanding of the physical significance of the various boundary condition types. Without a good understanding it can sometimes be difficult to interpret the analysis results. To assist the user with this aspect of an analysis, SEEP/W has tools that make it possible verify that the results match the specified conditions. In other words, do the results reflect the specified or anticipated conditions on the boundary? Verifying that this is the case is fundamental to confidence in the solution.

This Chapter is completely devoted to discussions on boundary conditions. Included are explanations on some fundamentals, comments on techniques for applying boundary conditions and illustrations of boundary condition types applicable for various conditions.

5.2 Fundamentals

All finite element equations just prior to solving for the unknowns ultimately simplify down to:

$$[K]\{X\} = \{A\}$$

where:

$[K]$	=	a matrix of coefficients related to geometry and materials properties,
$\{X\}$	=	a vector of unknowns which are often called the field variables, and
$\{A\}$	=	a vector of actions at the nodes.

For a seepage analysis the equation is:

Equation 5-1 $[K]\{H\} = \{Q\}$

where:

$\{H\}$ = a vector of the total hydraulic heads at the nodes, and

$\{Q\}$ = a vector of the flow quantities at the node.

The prime objective is to solve for the primary unknowns, which in a seepage analysis is the total hydraulic head at each node. The unknowns will be computed relative to the H values specified at some nodes and/or the specified Q values at some other nodes. Without specifying either H or Q at some nodes, a solution cannot be obtained for the finite element equation. In a steady-state analysis, at least one node in the entire mesh must have a specified H condition. The specified H or Q values are the boundary conditions.

A very important point to note is that boundary conditions can only be one of two options. We can only specify either the H or the Q at a node. It is very useful to keep this in mind when specifying boundary conditions. You should always ask yourself the question: “What do I know? Is it the head H or the flow, Q?” Realizing that it can be only one or the other and how these two variables fit into the basic finite element equation is a useful concept to keep in mind when you specify boundary conditions.

As we will see later in this Chapter, flow across a boundary can also be specified as a gradient or a rate per unit area. Such specified flow boundary conditions are actually converted into nodal Q values. So, even when we specify a gradient, the ultimate boundary condition options still are either H or Q.

There are cases where we may not know either H or Q. A seepage face is a boundary condition where this is the case. We actually know that the pore-water pressure is zero (H equals elevation) at the location where the seepage face develops, but we do not know the size of the seepage face that will exist. In such a situation, it is necessary to use an iterative procedure to try either H or Q boundary conditions until the correct solution is obtained. A complete section is devoted to this special case later in this Chapter. The point here is that in the end we can still only specify H or Q - which one is to be specified needs to be guided by the solution itself.

Remember! When specifying seepage boundary conditions, you only have one of two fundamental options – you can specify H or Q. These are the only options available, but they can be applied in various ways.

Another very important concept you need to fully understand is that when you specify H, the solution to the finite element Equation 5-1 will provide Q. Alternatively, when you specify Q, the solution will provide H. The equation always needs to be in balance. So when an H is specified at a node, the computed flux, Q, is the amount that is required to maintain the specified H, you cannot control the Q as it is computed. When Q is specified, the computed head, H, is the value that is required to maintain the specified flow Q.

Recognizing the relationship between a specified nodal value and the corresponding computed value is useful when interpreting results. Let's say you know the specified flow across a surface boundary. Later when you check the corresponding computed head at that node, you may find that a significant depth of water impoundment is required to provide the computed head. Then you can judge whether that is reasonable or not.

SEEP/W always provides the corresponding alternative when conditions are specified at a node. When H is specified, Q is provided, and when Q is specified, H is provided. The computed Q values at nodes where a head is specified are referred to as Boundary Flux values with units of volume per time. These Boundary Flux values are listed with all the other information provided at nodes.

A third important fundamental behavior that you need to fully understand is that when neither H nor Q is specified at a node, then the computed Q is zero. Physically, what it means is that the flow coming towards a node is the same as the flow leaving the node. Another way to look at this is that no flow is entering or leaving the system at these nodes. Water leaves or enters the system only at nodes where H or a non-zero Q has been specified. At all nodes without a specified condition, Q is always zero. This, as we will see later in this chapter, has important implications when simulating features such as drains.

The heads in a seepage analysis are the primary unknowns or field variables. A boundary condition that specifies the field variable (H) at a node is sometimes referred to as a Type One or a Dirichlet boundary condition. Flow gradient (flux) boundary conditions are often referred to as Type Two or Neumann boundary conditions. You will encounter these alternate names in the literature, but they are not used here. This document on seepage modeling simply refers to boundary conditions as head (H) or flux (Q) boundary conditions. Later we will differentiate between nodal flux Q and specified gradients (rates of flow per unit area) across an element edge.

5.3 Boundary condition locations

In GeoStudio 2007 all boundary conditions must be applied directly on geometry items such as region faces, region lines, free lines or free points. There is no way to apply a BC directly on an element edge or node. The advantage of connecting the BC with the geometry is that they become independent of the mesh and the mesh can be changed without losing the boundary condition specification. If you keep the concept of BC's on geometry in mind, you will find that you can specify any location for a BC quite easily. Consider the following examples which show the desired location of boundary conditions, the boundary condition applied to the geometry, and finally the underlying mesh with boundary conditions visible.

If you look carefully at Figure 5-2 and Figure 5-3 you will see that the BC symbols along the slope edge are spaced differently. In the view with no mesh visible, the BC's are displayed at a spacing that depends on the scale and zoom factor of the page. In the image with the mesh visible, the BC's are drawn exactly where they will appear. They are always at a node for this type of BC. Notice also that the free point location forces a mesh node to be at the exact location. This way, you can always define a BC anywhere you want and when the mesh changes, the BC location will remain fixed.

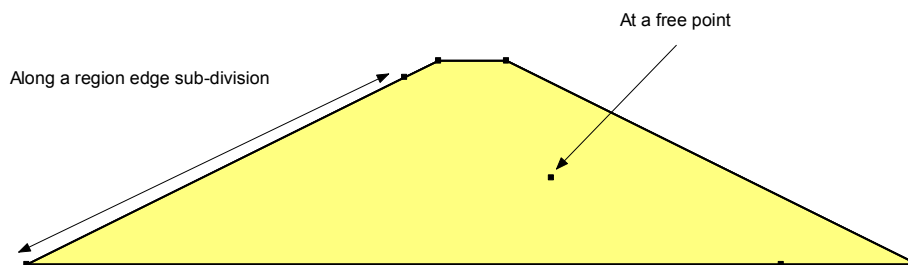


Figure 5-1 Desired BC locations

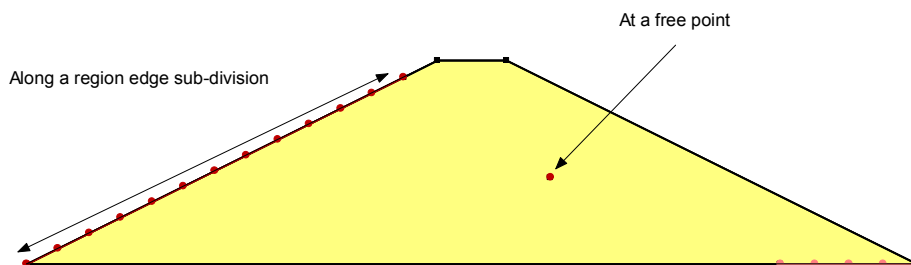


Figure 5-2 BC's attached to geometry

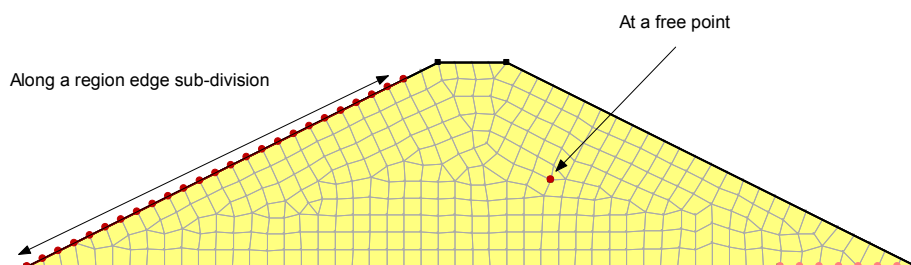


Figure 5-3 BC's with underlying mesh visible

Region face boundary conditions

A flux or stress boundary condition, which is given in units per area, can be applied to the “face” of a region. GeoStudio uses the contributing area that surrounds each node to calculate the corresponding flow rate or force. For example, a stress boundary condition applied to a region face is multiplied by the area that surrounds a node and converted into a force at the nodes. Similarly, a heat flux rate of 1 kJ/m²/day would be converted into a flow rate (kJ/day) after integration.

5.4 Head boundary conditions

Definition of total head

In SEEP/W the primary unknown or field variable is the total hydraulic head, which is made up of pressure head and elevation. The elevation represents the gravitational component. In equation form the total head is defined as:

$$H = \frac{u}{\gamma_w} + y$$

where:

- | | | |
|------------|---|--|
| H | = | the total head (meters or feet), |
| u | = | the pore-water pressure (kPa or psf), |
| γ_w | = | the unit weight of water (kN/m ³ or pcf), and |
| y | = | the elevation (meters or feet). |

The term u / γ_w is referred to as the pressure head – represented in units of length.

When specifying head boundary conditions, it is necessary to always think of both the pressure and elevation components. The significance of these components is illustrated with the following series of examples of flow through a pipe filled with sand.

Figure 5-4 shows a situation where the pipe is horizontal. The elevation is the same at both ends, but the pressure head is higher on the left than on the right. The total head on the left is 20 while on the right the total head is 10. There is a total head difference, and consequently, there will be flow from the left to the right. In equation form, the total difference is:

$$\Delta H = (20 + 5) - (10 + 5) = 10$$

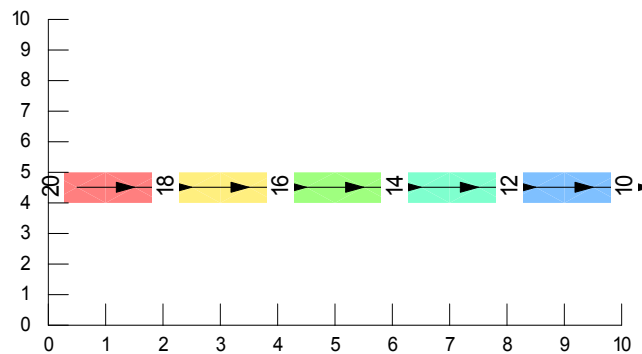


Figure 5-4 Flow in horizontal pipe due to head gradient only

Now consider the case where the left end of the pipe is higher than the right end as in Figure 5-5. In this case, the pressure head is the same at both ends. There is, however, a total head difference between the two ends due to the elevation difference and, consequently, once again there will flow from the left to the right. Now the total head difference is:

$$\Delta H = (0 + 10) - (0 + 0) = 10$$

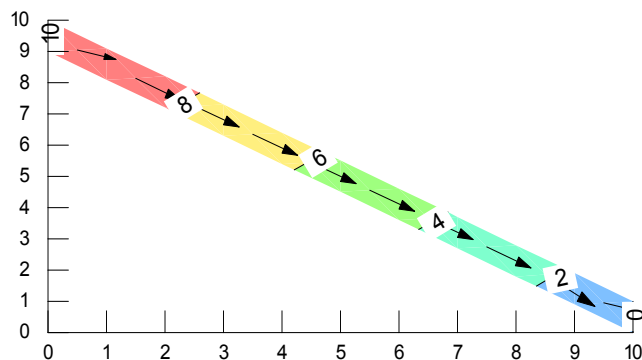


Figure 5-5 Flow in inclined pipe due to elevation head gradient only

In Figure 5-6 the pipe is again inclined as was the previous case, but the water pressures are different at each end. The total head difference now is zero and, consequently, there is no flow.

$$\Delta H = (5 + 10) - (15 + 0) = 0$$

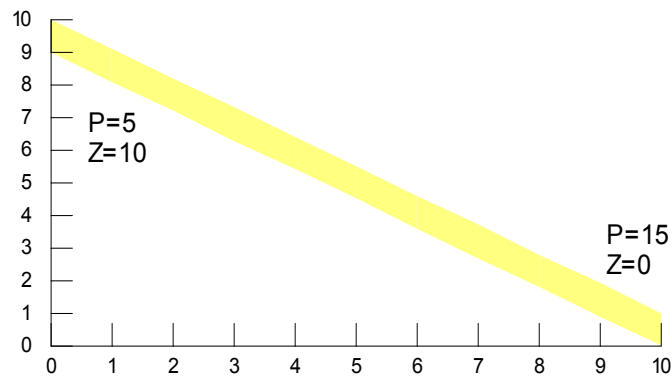


Figure 5-6 No flow in inclined pipe if zero total head change

Now consider the case where the total head is higher on the right than on the left, as in Figure 5-7, even though the pipe slopes from left to right. The total head difference is:

$$\Delta H = (5 + 10) - (20 + 0) = -5$$

The negative value indicates the flow is from right to left.

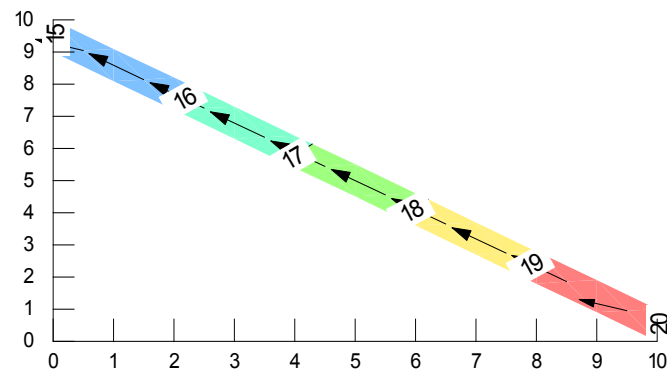


Figure 5-7 Upward flow due to pressure head and lower elevation head gradient

These simple examples all illustrate that the flow is in response to a total head difference between two points and, when defining head boundary conditions, it is necessary to always consider the pressures head together with the elevation.

At least one node in a steady-state analysis needs to have a head-type boundary condition in order to obtain a solution.

Head boundary conditions on a dam

Now let us specify the head boundary condition for a seepage analysis through the dam shown in Figure 5-8. The full supply level of the reservoir is 20 m. The ground elevation at the bottom of the reservoir is 10 m. Downstream of the dam the water table is at the ground surface and at the downstream end there is an under-drain. The granular material in the under-drain is highly permeable, making it reasonable to assume that the water level in the drain is the same as the water table downstream of the dam. This

implies that the permeability is high enough that there is no head loss in the drain relative to the small amount of seepage that will come through the low permeability embankment and foundation material.

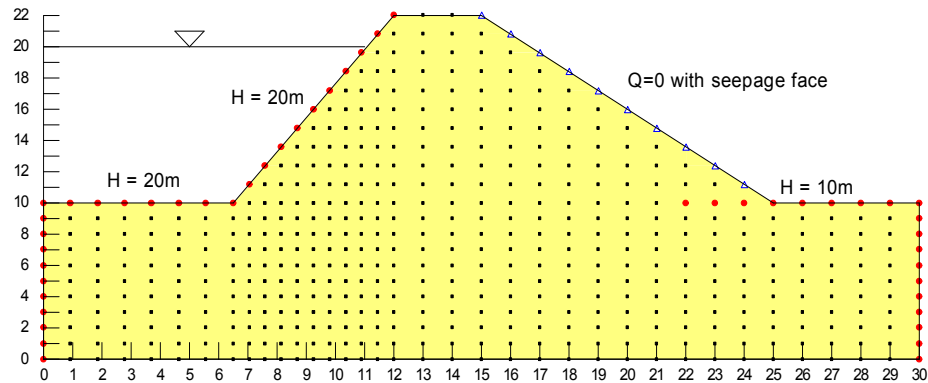


Figure 5-8 Head and seepage face boundary for flow through a dam

On the upstream side, where the reservoir contacts the embankment dam and the original ground surface, the total head is equal to the full supply level. On the upstream face of the dam at the reservoir level, the water pressure is zero, but the elevation is 20 m. The total head at this point then is 20 m. At the bottom of the reservoir, the pressure head is 10 m of water and the elevation is 10 m, making the total head also 20 m. The upstream boundary condition is therefore a constant total head equal to 20 m. This shows the convenience of using total head as a boundary condition. Even though the water pressures are different at every node on the upstream sloping face under the reservoir, the total head is nonetheless a constant. Without using total head as a boundary condition a different pressure would have to be specified at each node on the upstream dam face. Specifying this condition as a constant total head is more convenient.

Downstream of the dam toe, the water table is at the ground surface; that is, the water pressure is zero at the ground surface. The total head, therefore, is the elevation of the ground surface, which in Figure 5-8 is 10 m.

The water level in the drain is the same as the water table beyond the downstream toe as reasoned earlier. Therefore, the conditions around the perimeter of the drain are known. The total head around the drain perimeter in this case is 10 m.

The total head difference between the upstream and downstream conditions is 10 m, resulting in the flow shown in Figure 5-9.

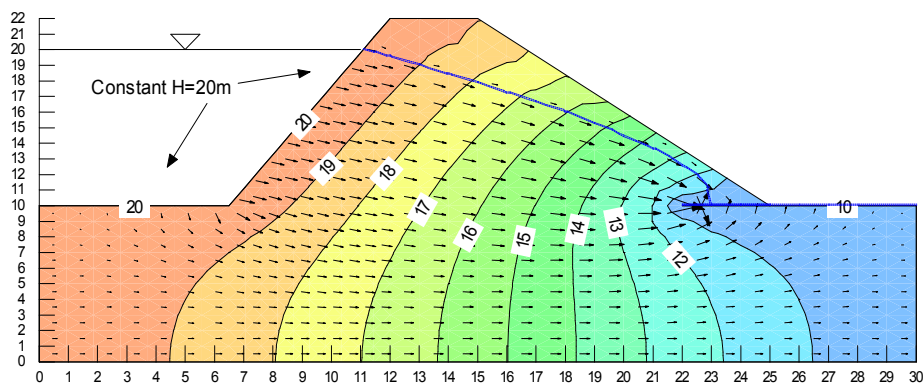


Figure 5-9 through reservoir dam with toe under-drain

Constant pressure conditions

Sometimes it is the pressure head that is constant on sloping ground and the total head varies at every node. Figure 5-10 illustrates a case like this. The natural ground surface beyond the embankment toe slopes away from the embankment. The water table is considered to be at the ground surface. From a seepage analysis point of view, the water pressure is zero at the ground surface. The elevation of each node in this area is different and therefore the total head is different at each node.

SEEP/W has a special feature to handle this condition. You can specify the head with a special condition that makes the pressure zero. When you select this option, SEEP/W finds the y-coordinate of each node and sets the total head equal to the y-coordinate. You will notice that the boundary condition is specified as a head-type with the action equal to the y-coordinate of the node.

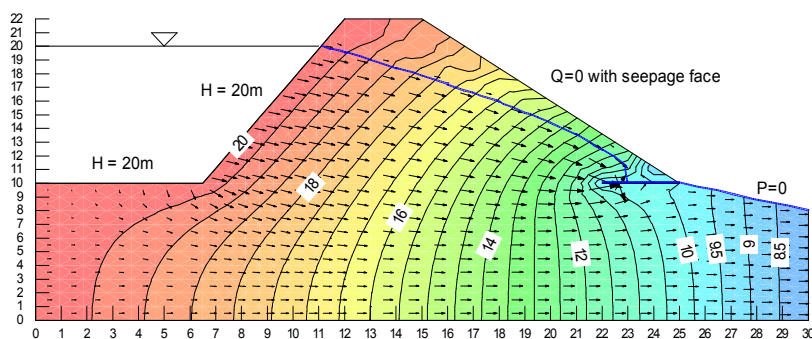


Figure 5-10 Sloping down-stream face with $P=0$ boundary (non constant heads)

Far field head conditions

There are many situations where it is necessary to specify the boundary conditions at the far extents of a problem. Figure 5-11 shows an example where the main interest is the seepage into an excavation. The excavation is below the initial water table. In a case like this it is necessary to specify the condition on the vertical boundary some distance away from the excavation. This is often referred to as the far field boundary.

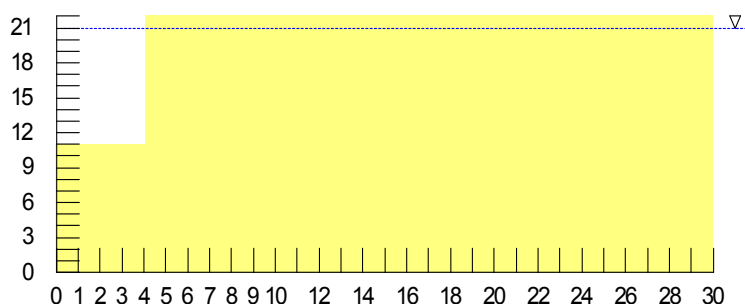


Figure 5-11 Excavation region with initial water table shown

In reality, the boundary conditions at the far field are unknown. It is common, however, to approximate the boundary condition by specifying a head condition reflective of the initial water table. In other words, the boundary condition is specified as a total head equal to the elevation of the water table. This is a reasonable thing to do, but it is important to understand the physical significance of this type of a boundary condition.

Specifying a constant total head as a far field boundary condition implies that sufficient water is somehow going to come to that location so that the head remains a constant. Physically this is the same as if there was a water reservoir up against the far field end of the problem, as illustrated in Figure 5-12, and that the reservoir remains at a constant level. This is not reality, but specifying the constant head has the same physical significance.

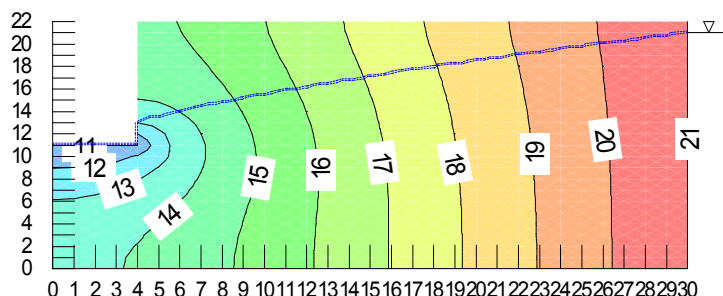


Figure 5-12 Far field boundary with head equal to water table elevation

Specifying a constant head far field boundary condition is legitimate in a transient analysis until the point in time when the water table would naturally start to drop at the far field. For further time steps, it is only an approximation of reality the same as it would be if used in a steady-state analysis. A steady-state analysis is physically a transient analysis run to infinity, and during this long time water somehow arrives at the far field to maintain the constant head.

The objective in a finite element analysis is to extend the far field boundary far enough away from the point of engineering interest that the head approximation does not significantly influence the engineering results.

Once again as discussed in Section 5.2, it is important to remember that when you specify a head, the software will always compute a flux Q into or out of the flow system at that node.

5.5 Specified boundary flows

The second of two options for specifying boundary conditions is to specify a rate of flow across the edge of an element. An example may be the rate of infiltration or application of precipitation. This is often referred to as a flux boundary.

Thinking back to the discussion on fundamentals in Section 5.2, when it comes to solving the unknowns in the finite element equations, it is necessary to specify or compute the flow at the nodes. Consequently, when specifying a unit rate of flow across the edge of an element, it is necessary to integrate along the edge of the element and convert the unit rate of flow (q) into nodal flows (Q). Even though SEEP/W automatically does the integration, it is nonetheless useful to have an understanding of how the integration is done. Understanding the relationship between unit rates of flow across an element edge and nodal flows can be useful in understanding and verifying results. Sometimes in unusual situations, it can also be useful to know how to manually compute the nodal Q flows from unit rates of flow (q) and then specify Q at the node instead of q on the edge of the element.

First of all, SEEP/W only handles uniform rates of flow across the edge of an element. The uniform rate of flow can be set to vary with time, but not with distance across an edge of any individual element. Formulations for non-uniform rates of flow across an edge are available, but these have not been implemented in SEEP/W.

Since the flow across the edge of an element is uniform, the total flow across the edge is simply the flow rate (q) times the length of the element edge. The manner in which the total flow across the edge gets distributed to the nodes depends on the number of nodes on the edge. In SEEP/W the edge can have either 2 or 3 nodes.

When the edge of an element has only two nodes, the total specified flow across the edge is evenly divided between the end nodes. Half goes to one end node, the other half goes to the other end node. This is graphically illustrated in Figure 5-13. When two or more edges have a common node, the contributions from each edge accumulate at the common node as illustrated in Figure 5-13.

Higher-order elements with three nodes along the edge of an element are usually not required in a seepage analysis. Sometimes, however, it is desirable to do a SIGMA/W or QUAKE/W analysis on the same mesh, and for these types of analyses, higher order elements are required. Consequently, sometimes higher order elements are used in a seepage analysis not because they improve the seepage analysis results, but because mesh consistency is required when integrating with other types of analyses.

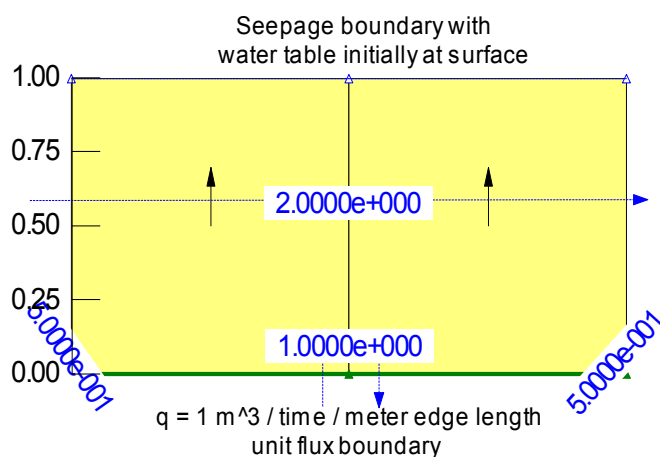


Figure 5-13 Unit and total nodal flux boundary condition relationships

When there are three nodes along the edge of an element, the total flow across the edge is distributed as $1/6$ to each corner node and $4/6$ to the middle node. This is graphically illustrated in Figure 5-14. Once again, when one node is common to more than one edge the $1/6$ end contribution from each edge is accumulated at the common node. The distributions of the nodal fluxes come directly from the numerical integration of a variable, such as a gradient along the edge of an element. Further details on this are presented later in the Theory Chapter and the Visualization of Results Chapter.

For an axisymmetric case the Q boundary flux values are a function of the radial distance from the vertical symmetric axis. For elements with nodes located only at the corners as shown in Figure 5-15, the nodal boundary flux, Q , is the contributing area from each element multiplied by the radial distance from the symmetric axis times the element thickness which is expressed in terms of radians. The nodes further away from the axis of symmetry have more contributing area so their flux values are greater. In the case of the middle node, it has a total flux value of 1.0, but this is comprised of two flux quantities of 0.5, each contributed by the element edges on either side of it. The node on the right has a total flux of 0.833333, but only half the contributing area as the middle node. Consequently, as the distance from the axis of symmetry increases so does the flux quantity.

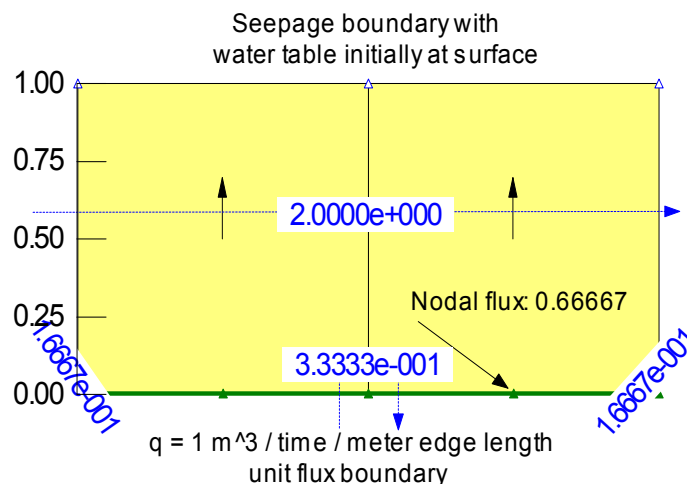


Figure 5-14 Distribution of unit flux in element with secondary nodes

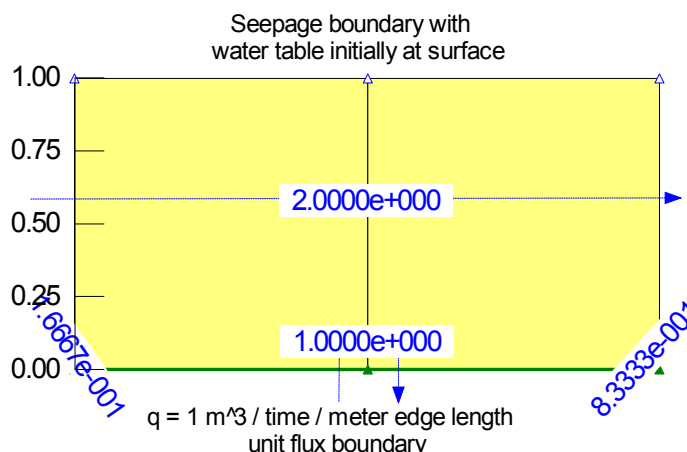


Figure 5-15 Distribution of unit flux in axisymmetric analysis without secondary nodes

The situation is not quite as straightforward for higher order elements where three nodes exist along the element edge, as illustrated in Figure 5-16. The detailed formulas for these cases are provided in the Theory Chapter. The case described in Figure 5-15 is fairly straightforward and easy to remember for quick spot-checking the computed nodal boundary flux values.

Conversion of specified flow rates (gradients) across element edges is dependent on the specified element thickness. For a two-dimensional vertical section the thickness, by default, is one unit, although the thickness can be some other specified value. For axisymmetric cases, the thickness is in terms of radians. The default is 1 radian. Sometimes it is useful and convenient to specify the thickness as 2π radian so that the computed nodal boundary flux values are for the entire circumferential area. Specifying the thickness is simply a user preference. It is important to remember the element thickness when interpreting the computed Q values.

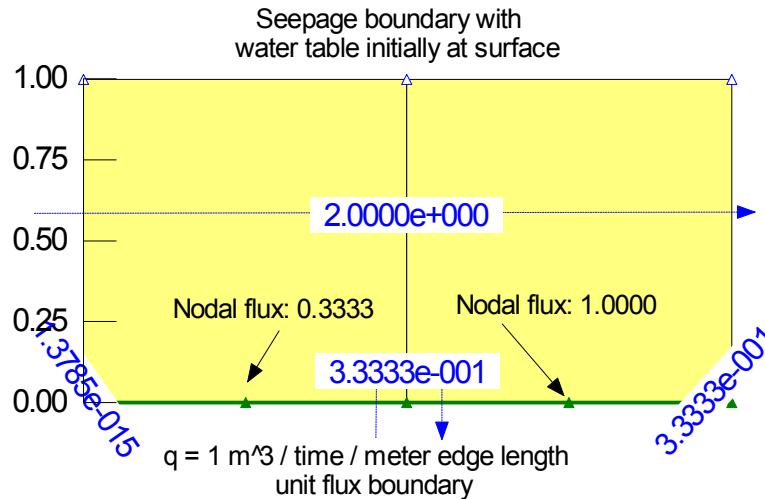


Figure 5-16 Distribution of unit flux in axisymmetric analysis with secondary nodes

In SEEP/W, the specified in-flow or out-flow rates (gradients) are always deemed to be normal to the element edges.

In SEEP/W, flow into the system is positive. Flow out of the system is negative.

Another important concept to keep in mind when specifying and interpreting flow quantities is that nodal boundary flux values Q are scalar values. Nodal Q values have no specified or computed direction. The direction of flow can only be inferred from the computed gradients or unit flow vectors inside the elements. SEEP/W can display velocity vectors in each element, which are a graphical representation of flow rates and direction.

5.6 Sources and sinks

There is another type of boundary condition called a source or a sink. These boundary conditions are sometimes referred to as a Type 3 boundary condition. A typical sink might represent a drain at some point inside a mesh. A source could be an injection well. The important concept about sinks and sources is that they represent flow into or out of the system.

Consider the situation in Figure 5-17, which simulates the effect of an under-drain on the downstream face of a dam; in particular at nodes 415, 416, and 417. The simplest way of including the effect of a drain at that location is to specify a “sink” type boundary condition at those nodes. We assume that the drain will be capable of removing all the seepage that arrives at the drain. In other words, the drain will not be under positive pressures at any time; the water pressure in the drain will be zero. We can specify this boundary condition by specifying a head (H) at the desired drain nodes equal to the elevation (y -coordinate) of those nodes. This is equivalent to specifying the pressure to be zero at those nodes ($P=0$). By specifying the H condition, SEEP/W will compute the boundary flux (Q) that must exist to ensure the pressure is always 0 at each node.

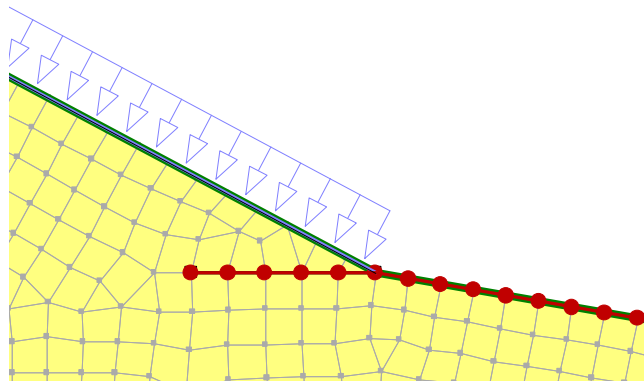


Figure 5-17 Under-drain nodes beneath toe of down stream dam face

In this example the specified H is a sink, since water leaves or disappears from the flow system as a result.

From a behavioral point of view, any specified H or Q on the perimeter of a problem has the same action as a specified H or Q inside the mesh. The difference is only in terminology. Along the problem perimeter the specified H and Q are usually referred to as boundary conditions. Inside the mesh, a specified nodal H or Q is often referred to as a sink or source type of boundary condition.

It is often tempting to specify piezometric measurements at certain nodes inside a mesh. This leads to erroneous results. The physical consequence of including piezometric measurements in a seepage analysis is that water needs to enter or escape from the flow system in order to achieve the specified water pressure at the point where the piezometer is located. Remember that when we specify H (piezometric measurement), SEEP/W will compute Q . Piezometers are not a sink or a source and therefore it is not appropriate to specify the field measurements as a boundary condition. A useful way of thinking about this situation is that the flow regime was created before the piezometer was installed and the installation of the piezometer does not change the flow regime. Specifying the piezometric measurements may alter the flow regime since the specified conditions may act like a sink or a source, which is not the intention.

5.7 Seepage faces

It has been noted several times earlier that in seepage analyses only the head (H) or flow (Q or q) can be specified as a boundary condition. There are, however, situations where neither H nor Q are known. A typical situation is the development of a downstream seepage face such as illustrated in Figure 5-18. Another common situation is the seepage face that develops on the upstream face after rapid drawdown of a reservoir. Other examples may include the drain/soil interface inside a lysimeter where water accumulates until pressures build sufficiently to form a drip surface. For this type of situation, there are periods of time where there is no flow across the drain boundary.

The pore pressure on a seepage face is zero, but this cannot be specified for all times because a $P=0$ situation also reflects the location of the phreatic surface and it may inadvertently become a source for water. Another problem with seepage faces is that the size of the seepage face may not be known and consequently needs to be determined as part of the solution through an iterative process.

In SEEP/W a potential seepage face needs to be flagged as a boundary condition type. At the end of each iteration, the conditions along the specified potential seepage face are “reviewed” to see if they meet the correct criteria. This is why it is called a potential seepage face review

Consider the simplest case where the boundary flux is zero along a seepage face. In this situation, the boundary conditions along the entire potential seepage face are set to $Q = 0$ (flux-type), indicating that no additional flux is going to be added or removed at these nodes, and with the specification that the conditions will be reviewed and adjusted as necessary. At the end of the first iteration after the heads are computed for all nodes along the potential seepage face, SEEP/W checks to see if any of the nodes have a positive pressure (H greater than elevation). Nodes with computed pressures greater than zero are not allowed, as positive pressure on the surface indicates ponding, which cannot happen along the sloping boundary. The water would run off, not pond. The specified boundary condition of Q equal to zero is therefore not correct. Physically, it means water wants to exit the system, but the Q equal zero condition does not allow the water to exit. To allow water to leave, SEEP/W converts the boundary condition to a head-type with H equal to the y-coordinate (zero water pressure) at each node with a computed positive pressure, and a new solution is computed.

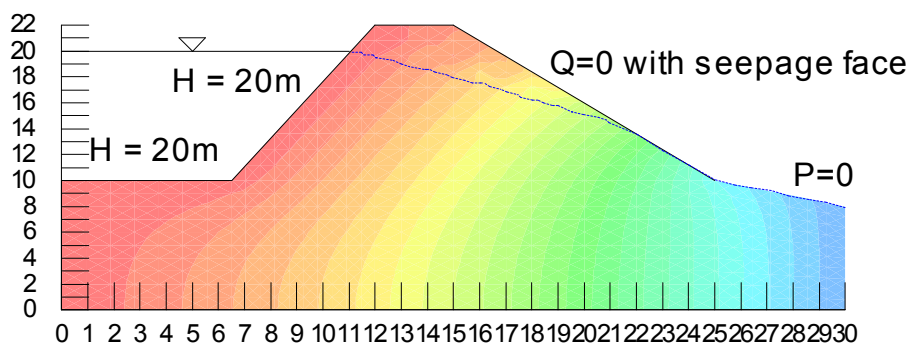


Figure 5-18 Seepage on down slope dam face (no toe under drain in this case)

At the end of the repeated solution step, water can now leave or enter the system at the nodes that were converted from a Q -type to an H -type. Now the program checks to see whether the computed Q is negative (out of the system) or positive (into the system). Computed flow out (negative) of the system is acceptable – a computed flow into (positive) the system is not acceptable. At nodes along the potential seepage face with a positive flux, SEEP/W then converts the H -type boundary back to the original specified Q -type and then repeats the analysis. The process is repeated until all nodes on the potential seepage face have either a computed zero water pressure ($H = y$) or a nodal Q equal to zero, the original specified boundary condition. In a steady state analysis, once the seepage face is determined, the solution solves to completion. In a transient analysis, the seepage face is determined for every transient time step as necessary.

The initially specified flux on a potential seepage face does not have to be zero. The initially-specified flux can be some other positive or negative value. A positive value could represent infiltration and a negative value could represent evaporation. Nodes that are not converted to an H -type along a potential seepage face in the end must have the specified flux.

During a transient analysis, potential seepage face review nodes are set to a flux-type boundary condition at the start of each time step. This includes all nodes that were converted to a head type boundary condition during the previous time step. The applied flux is set to the initial action specified at the review nodes. If the review nodes follow a flux-type boundary function, the flux at the review nodes is computed from the function for the start of each time step. The size of the required seepage face must be determined separately for each time step in a transient analysis.

Review nodes can be located anywhere along a boundary. For example, the nodes may be located on the downstream seepage face and on the upstream drawdown face at the same time. All review boundary

nodes are considered in each review procedure. Consequently, depending on the solution, the modification of boundary conditions may jump from one area to another with each successive iteration.

Sometimes conditions may follow a head-type boundary function; such as the upstream face of an embankment dam experiencing drawdown. The head changes as the reservoir is lowered. Above the changing surface of the reservoir, there is potential for a seepage face to develop as the pore-water pressures within the embankment redistribute. In this case, a function that describes the changing head boundary with time can be developed and a secondary condition that will change the head boundary to a flux boundary if the total head becomes less than the elevation head at any node is automatically inferred. When the head boundary is assigned to the reservoir nodes, the potential seepage face review option can then be selected. The boundary review procedure is then followed to determine the required seepage for that particular time step. This is required to ensure that all nodes with a y-coordinate greater than the boundary function head (pool elevation) have a condition of flux equal to zero at the start of each time step.

It is important to remember that the node along a potential seepage face must have a flux type boundary at the start of the time step. Nodes with a specified constant head boundary condition cannot be reviewed.

Care must taken to not specify potential seepage boundary conditions that are unrealistic or unnecessarily too large. The computing time and associated potential convergence difficulties increase significantly as the number of nodes along all potential seepage faces increases. A little preliminary planning and guessing as to where seepage face may develop and specifying the boundary conditions accordingly helps to mitigate these issues.

5.8 Free drainage (unit gradient)

A free drainage boundary condition can be useful in certain circumstances, but it can also be easily misused. Consider the pressure profile of the Kisch problem for steady state infiltration through a clay barrier over a coarse sandy soil as illustrated in Figure 5-19. In this figure, both the surface and base boundary conditions are specified as head boundaries. There is in effect a pressure equal to zero at the surface and a water table at the base. Consider now what the bottom boundary condition would be if the water table was not 2.5 meters below the surface, but was 30 meters below the surface. While technically it would be possible to construct a mesh to represent this situation, it would likely not be practical.

We know from the illustrated Kisch pressure profiles that at some point beneath the clay (near surface) layer, the water contents and pressures in the sand become constant with depth. When this is the case, the total head gradient within the sand is equal to unity and the downward flux is equal to the hydraulic conductivity at that point. Closer inspection of the pressures in Figure 5-19 indicate that the unit gradient condition could exist under the clay soil until a specific depth, which is dependent on the infiltration rate and the hydraulic properties of the underlying soil, when the pressures have to transition towards $P=0$ near the water table. If we do not want to model the entire profile and we are certain that for our real field conditions there is a fairly constant water content and pressure with depth, we can use the free drainage or unit gradient boundary condition option for the base of our geometry and apply it at some fixed distance AWAY from our prime area of interest in the model. Figure 5-20 shows the correct pressure profile for the Kisch problem with a free drainage base boundary condition applied one meter beneath the clay cover.

Use of the free drainage boundary condition should ideally be applied in transient analysis and only specified on bottom mesh boundaries that are relatively far away from your main area of focus. They will work for a steady state analysis, but the solution can be hard to obtain from a convergence perspective, as you are in effect applying an unknown Q boundary with unknown conductivity values at the base of the problem and the solution can tend to oscillate if not carefully controlled.

In a transient analysis, however, the unit gradient boundary condition is obtained by setting the Q flux boundary to be equal to the actual hydraulic conductivity value present at the nearest gauss point from the previous iteration multiplied by the contributing edge boundary length. Therefore, the solution of the correct Q to apply is closely controlled by the initial or previous time step and actual pressure and conductivity conditions in the finite element at the point of application. In a transient analysis the solved Q drainage is free to increase or decrease with time, depending on the small changes in pressures in the elements above the free drainage boundary. In turn, the water content at the bottom can, with time, slowly increase or decrease in response to the free drainage or infiltration from above.

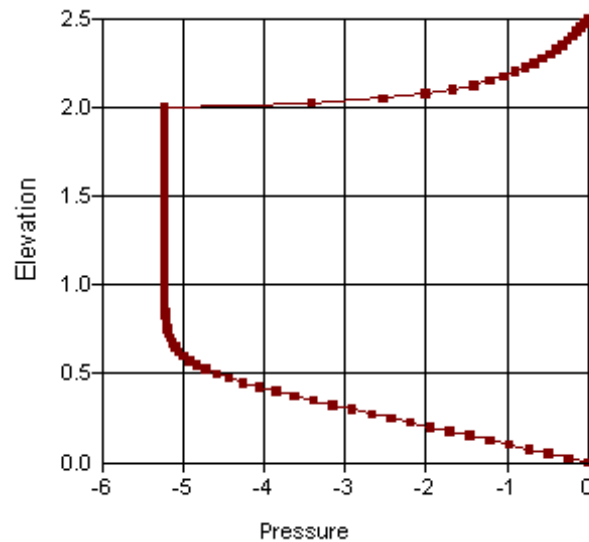


Figure 5-19 Kisch pressure profile with head top and head base boundary condition

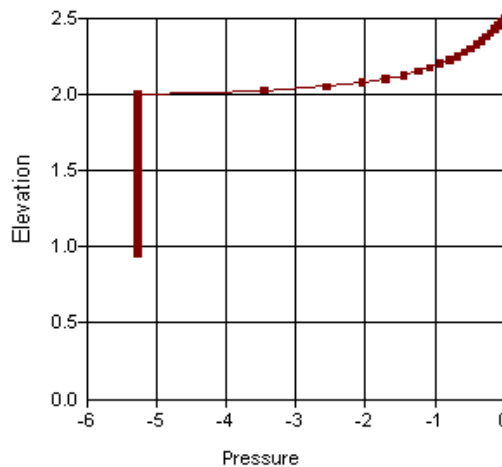


Figure 5-20 Kisch pressure profile with head top and free drainage base boundary condition

5.9 Ground surface infiltration and evaporation

SEEP/W is formulated for saturated-unsaturated flow, making it possible to model infiltration or evaporation at the ground surface. Figure 5-21 shows a situation where surface infiltration is considered. The infiltration in this example is specified as a positive surface flux.

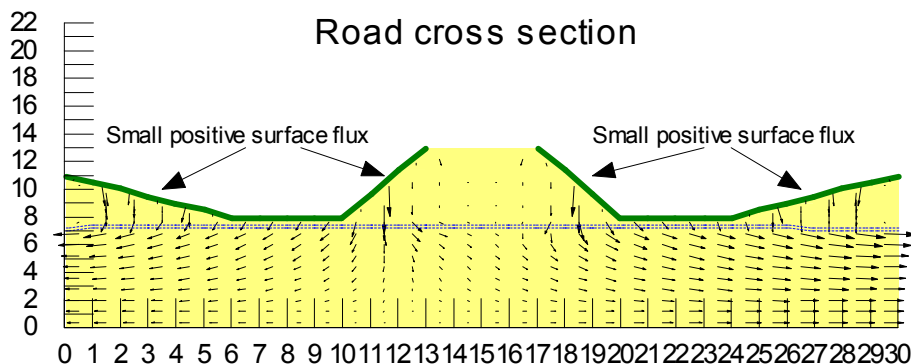


Figure 5-21 Cross section of paved road with infiltration flux boundary

While SEEP/W can readily accommodate flux boundary conditions representing infiltration and evaporation, relating the specified rate of surface flux to environmental conditions is not trivial. There are many factors involved, as depicted in Figure 5-22. On the surface there is rainfall, snow melt, wind speed, solar radiation, air temperature, and run off. If the ground experiences high temperatures, water in the ground may turn to vapor and evaporate. Additionally, water may leave the ground through plant transpiration. The amount of water leaving the ground surface is related to the tension in the water. A special formulation which has been implemented in VADOSE/W is required to correctly simulate the ground surface flux and relate the flux to environmental conditions.

The decision as to when it is appropriate to estimate the ground surface flux and simply specify the flux as a boundary condition in SEEP/W and when it is necessary to more rigorously correlate the ground surface flux with the environmental conditions has to be made in the context of the objectives and significance of each particular project. The important point here is that SEEP/W can accommodate flux boundary conditions, but to correctly and rigorously correlate the ground surface flux with environmental conditions it is necessary to use VADOSE/W. Further discussions are presented in the VADOSE/W documentation.

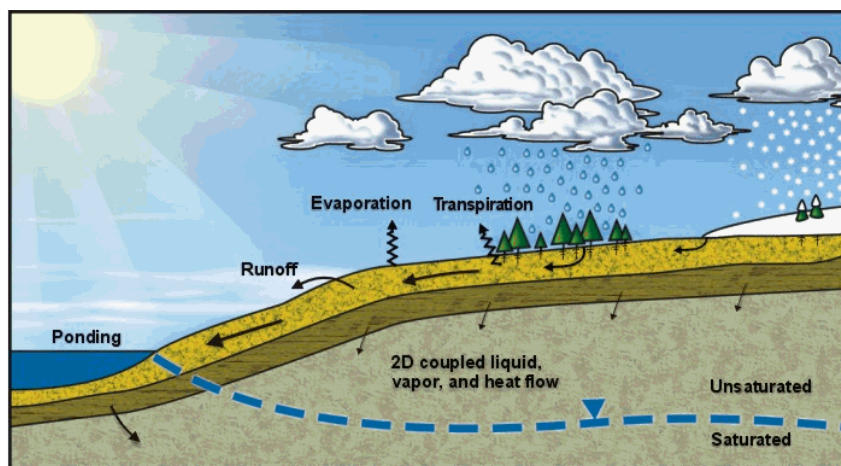


Figure 5-22 Summary of near ground surface flow processes

One response worth remembering when specifying flux boundary conditions is that the solution is related to the saturated hydraulic conductivity (K_{sat}). If the specified per unit area flux rate (q) is higher than K_{sat} , the solution will usually indicate positive water pressures developing on the surface. In other words, water would need to pond on the surface in order for the specified flux to enter the ground. Also, in a

steady-state analysis, it may completely saturate the system. The specified surface flux in Figure 5-21 is less than K_{sat} and, consequently, there is an unsaturated zone that exists within the ground above the water table. Once again, when a flow rate is specified, a head will be computed, and it is possible that if the specified flow rate is too high then unrealistic heads will be computed. In summary, it is always useful to be thinking about K_{sat} when specifying ground surface flux rates.

5.10 Boundary functions

SEEP/W is formulated to accommodate a very wide range of boundary conditions. In a steady state analysis, all of the boundary conditions are either fixed heads (or pressure) or fixed flux values. In a transient analysis however, the boundary conditions can also be functions of time or in response to flow amounts exiting or entering the flow regime. SEEP/W accommodates a series of different boundary functions. Each one is discussed in this section.

General

A non-fixed boundary condition must be entered as a user-defined function. In SEEP/W all functions are defined using a combination of manually entered or cut and paste data points, and all functions can be customized to suite your exact needs. In certain cases, it may be desirable to have a stepped function. The additional functionality of automatically fitting the data with a step function has been added to the program. There is also an option to have a cyclic function repeat itself over time, which saves you the task of defining it repeatedly. Additional information regarding all function types are described in the Chapter called Functions in GeoStudio.

In general, functions are comprised of a series of x and y data points that are fit by a spline curve. A spline curve is a mathematical trick to fit a curved shape between a series of points. The simplest way to fit a series of data points is to draw a straight line between the points. This is often a very poor way to represent a non-linear function. The advantage of the spline is that it joins all data points with a continuous smooth curve.

During the solution process, the solver uses the “y” value along the spline curve for any required “x” value. It is therefore important to make sure the spline fit, not your original data, portrays how you want the boundary condition applied.

Head versus time

A very useful boundary function is user-specified Head versus Time. Consider the example in Figure 5-26 where the drawdown in the reservoir is to be applied as a boundary condition so that the dissipation or pore-water pressures in the dam can be modeled. A Head versus Time function (Figure 5-25) with the $Q=0$ if $H < \text{elevation}$ option has been specified on the reservoir side of the dam.

The advantage of using a Head versus Time function on the reservoir side of the dam is that it avoids a “shock” unloading of the water pressure on the dam if the water level in the reservoir were instantly drawn down to the 11 meter elevation. Whenever possible, it is a good idea to apply changing boundary conditions as realistically as possible, to avoid shocking the system.

In the drawdown example, the boundary function has an added criteria imposed on it that requires the boundary flux on the head nodes to be zero if the head at the node becomes less than the elevation of the node. This means that seepage cannot flow out of the upstream face of the dam above the water table. This may not be desired in some cases, such as if there were a higher water table on the right side of the dam that caused reverse flow through the dam, or if there were some surface infiltration that could cause water to flow back into the reservoir through the upstream face.

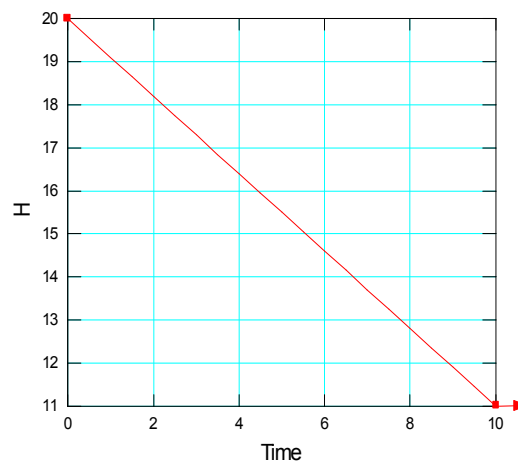


Figure 5-25 Head versus time function for reservoir draw down

There are occasions where the boundary function may cycle, or repeat itself over time. This may be the case if the reservoir water level rises and drops on an approximately constant cycle. Instead of defining the cyclic nature of the function from start to finish, you have the option to define it over the first “period” of time and then have it repeat itself for as long as you defined time steps to solve. An example of a cyclic function is given in Figure 5-27 where the data represents the water level in the reservoir fluctuating between 20 and 16 meters over a 100 day period.

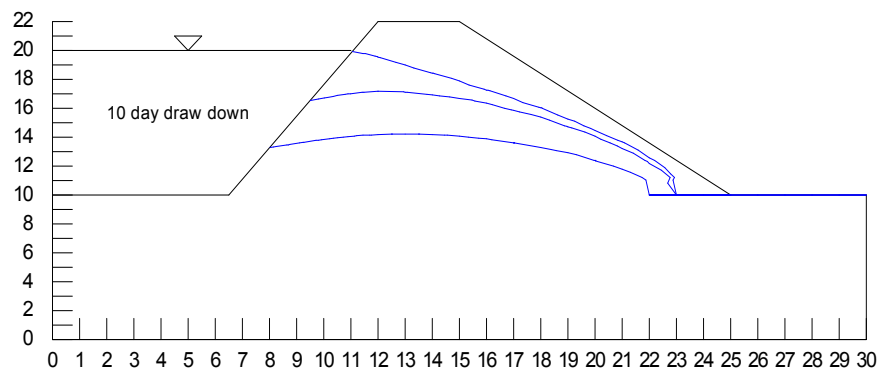


Figure 5-26 Phreatic surface in dam over time after reservoir draw down

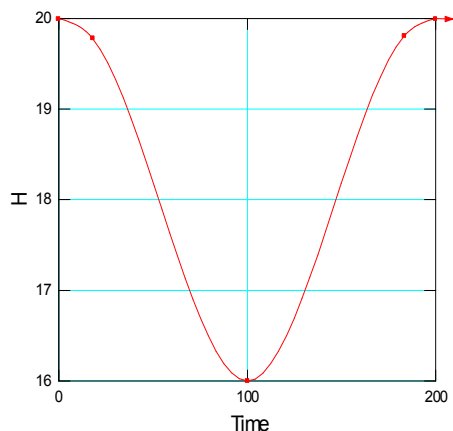


Figure 5-27 Cyclic head versus time function

In any “time” function, make sure the units of time you specify in the function match those used to define the conductivity function (e.g. seconds, hours, days etc.) For long transient solutions, it is often useful to define the conductivity in terms of m/day so that all time steps and functions can be set up with “day” units. This avoids having to input time values of 864000 seconds to represent 10 days.

Head versus volume

In certain circumstances you may want to determine the amount of seepage that exists and how deep a pond may be as a result of the outflow. Consider that you have a rising river level and you need to determine the depth of ponded water that develops in a depression adjacent to the river.

SEEP/W is formulated to automatically track and accumulate the volume of water from a user-specified area, such that the volume of water can be compared with a user input function to determine the actual head value at that point. Consider the simple illustration in Figure 5-28. In this example, a river level is rising according to a user input Head versus Time function. The objective is to determine how deep the seepage pond will get when the river reaches its peak.

To carry out this analysis, you must create a Head versus Volume function for the area being filled by seepage water. A schematic of this function is given in the figure below. As the analysis solves, the total outflow from each Head versus Volume boundary node is obtained and summed. The total summation of these nodal volumes is equal to the total volume of ponded water. Prior to the next time step, the program will consider the total volume of ponded water and use the Head versus Volume function to determine the head boundary to apply at each node.

It is important to consider the ponded water head in this case, because the total outflow into the pond is a result of the difference in the driving head from below AND from above.

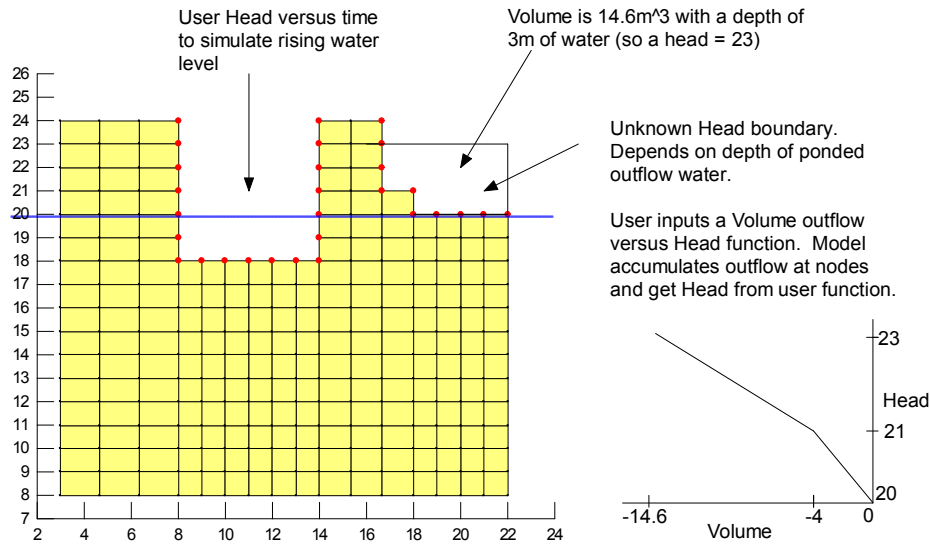


Figure 5-28 Simple illustration of head versus volume boundary function

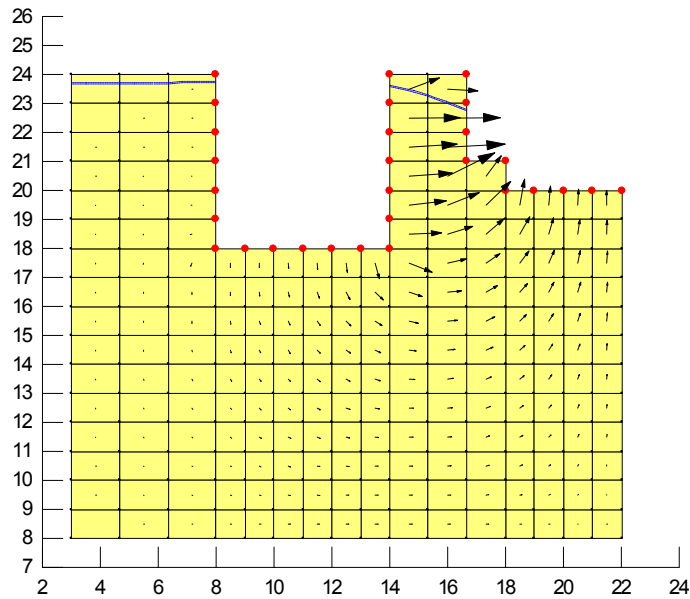


Figure 5-29 Seepage into ponded volume

The illustration in Figure 5-29 shows the position of the phreatic line at the end of the analysis when the river level is at its peak. You can see that the flow vectors indicate there is still some seepage out of the soil and into the pond at this stage. Had the analysis been carried out for slightly longer with the river level remaining at its peak, then the ponded depth would rise to match that of the river. However, in a real life application, it is likely that the river level would peak and then subside. This analysis method could be used to determine the full transient nature of the rise and decline of the ponded water in response to the river's rise and drop.

Nodal flux Q versus time

As discussed previously, there are only two fundamental types of boundary conditions: either a head or flux boundary. Head boundaries can be applied as total head, pressure head, or a time/volume function of

head. Likewise, flux boundaries can be applied as fixed values of total or unit flux or a time dependent function of either.

In many cases it is useful to apply a total nodal flux (Q) boundary that is a function of time such as the function illustrated in Figure 5-30. In this case, a total nodal flux is being applied at one or more nodes such that the flux represents a decaying source of water at those nodes. If the function “ y ” value is positive, this indicates a water source at the node and if the “ y ” value is negative, this would indicate a nodal sink flux. When you apply a total nodal flux at a node, you are not taking into account any mesh geometry in the flux quantity. You are simply stating that you are introducing a set volume of water at a single point in the mesh, and the amount that you are introducing changes with time.

During the solver process, the solver will take the elapsed time of the solution and look up from the defined function what the total flux rate of water should be at that time. It will then multiply the function value by the current time step duration to obtain a total volume of water to add at that time step. Be careful when using functions like this. It is very important to make sure the time steps you set up in the solver are small enough to follow the desired shape of the function. For example, for the function shown in the figure below, if the elapsed time was 3 days and the time step being solved was 3 days, then a water volume of $0.05 \text{ m}^3/\text{day} * 3 \text{ days} = 0.15 \text{ m}^3$ would be applied between day 3 and day 6. The specified function amounts over these three days would be $0.05 \text{ m}^3/\text{day}$ between day 3 and day 4, 0.04 m^3 between day 4 and day 5, and 0.025 m^3 between day 5 and day 6. Therefore, the correct total for these three days would be $0.05 + 0.04 + 0.025 = 0.115 \text{ m}^3$. Using too large a time step in this case would in effect introduce more water than indicated by the function for the same time period. As a rule, make the time steps small enough to follow the shape of the function.

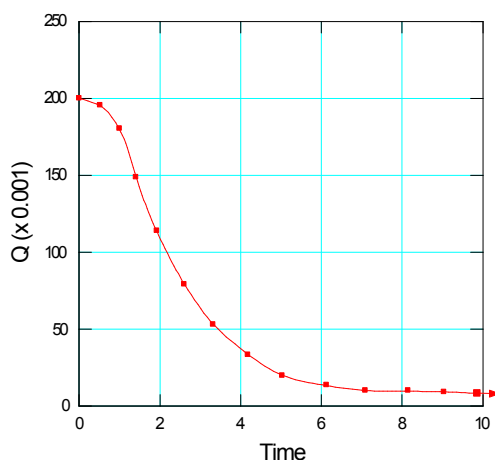


Figure 5-30 Nodal flux (Q) versus time function

Unit flow rate versus time

The previous section discussed application of a total nodal flux boundary function. In this section, a unit flux boundary function is discussed. A unit flux is a flux value applied along a mesh edge such that the length of the mesh edge determines how much flux is applied at the nodes adjacent to the edge. While a unit flux can be applied as a fixed action value, it is very useful to be able to apply it as a function.

A very simple example of a unit flux boundary function applied to an element edge is that of a ground surface daily rainfall minus potential evaporation rate that is known per square meter of surface area.

Consider one such function as illustrated in Figure 5-31. In this function, the daily rainfall minus potential evaporation is entered as a step function. If the solver time steps are set up to be less than or equal to a day, they will follow the shape of the function and include data from each step. The data for this type of function can easily be set up in a spreadsheet program and cut and pasted into the function edit box.

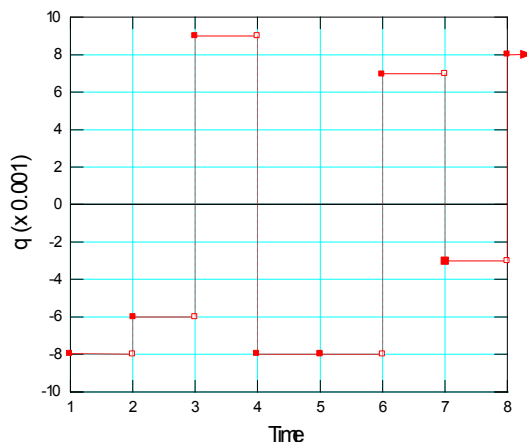


Figure 5-31 Unit flux function showing net rainfall minus evaporation

Modifier function

The previous two types of functions allow you to apply a set total or unit flux at element edges or mesh nodes. They are functions that will let the solver apply the appropriate action value for the stated elapsed time.

A modifier function is a powerful way of having the applied flux be dependent on the pore-water pressures in the soil. Consider the process of evaporation from a soil surface. As the soil dries out at the surface, the amount of actual water that is evaporated is reduced from the potential amount. Wilson (1990) showed that the rate of actual evaporation to potential evaporation is a function of the soil surface temperature and relative humidity. Ideally, the relationship is a result of coupled heat and mass flow processes in the soil (e.g. VADOSE/W); however, the process can be represented in simplified terms by having knowledge of the soil water pressure. It is possible to use SEEP/W to compute an approximate actual evaporation rate from the soil surface by combining a unit flux boundary condition (such as that specified above) with a modifier function such as the one illustrated in Figure 5-32. In this figure, the ratio of actual to potential evaporation is shown as a function of the negative pore-water pressure. As the soil dries out (the pressure becomes more negative), the percentage of the daily potential evaporation as specified by the unit flux versus time function is reduced. At each time step, the solve will look up the appropriate potential evaporation rate and then multiply that value by the percentage value obtained from the modifier function for any given soil pressure state.

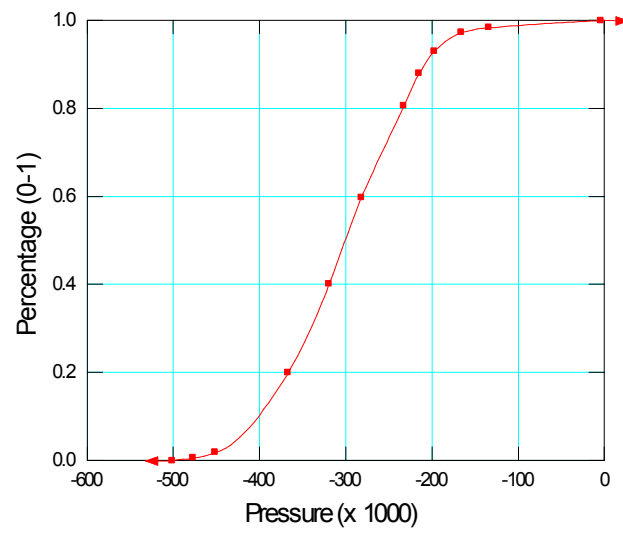


Figure 5-32 Modifier function showing possible ratio of actual to potential evaporation

6 Analysis Types

There are two fundamental types of finite element seepage analyses, steady-state and transient. Numerous additional constraints and conditions can be applied within each fundamental type. A description of each type and the implications associated with each type are discussed in this chapter.

6.1 Steady state

Think about the meaning of the words “steady state.” They describe a situation where the state of the model is steady and not changing. In a seepage analysis for example, the “state” means the water pressures and water flow rates. If they have reached a steady value, it means that they will be in that state forever. In many cases where the geotechnical problem is exposed to nature with its cyclical conditions, it is possible that a steady state will never be reached. Flow beneath a cutoff wall may come close to steady state if the upstream conditions are held fairly constant over time. Net surface infiltration and evaporation will never reach a steady long term constant value. When simulating a constantly changing boundary condition such as surface infiltration and evaporation, the model does not know what the steady-state reality might be, it only knows how the initial input parameters change in response to the boundary conditions applied over the length of time being modeled. If you make the assumption that the boundary conditions are constant over time, then the model can compute the long term ground conditions in response to your assumption.

This type of analysis does not consider how long it takes to achieve a steady condition and that is something you need must understand. The model will reach a solved set of pressure and flow conditions for the given set of unique boundary conditions applied to it and that is the extent of the analysis.

It’s very important to understand that when you are doing a steady state analysis you are not making any estimation of how long it takes the final condition to develop, nor how long it will last. You are only predicting what the ground will look like for a given set of boundary values, and it is implied that you are pretending the boundary values have been in place forever and will be in place forever.

Because steady state analyses are taking out the “time” component of the problem it greatly simplifies the equations being solved. However, at the same time it can make convergence harder to achieve – depending on the degree of non-linearity of your soil property functions. The steady state seepage equation leaves out the actual “time” variable and omits the entire volumetric water content function. They are not needed in the solution. The volumetric water content is used for telling how much water is gained or lost in the soil if there is a change in pressure. In a steady state analysis, there is no starting pressure, and so there is no change in pressure to worry about. Remember, it is only looking to find out what the pressures will be throughout the problem geometry, given the fact you know what they are at only a few known locations and for all points in time.

Unfortunately, simplifying the equations does not mean that there is a reduced set of numerical issues associated with solving them. Recall that the soil hydraulic conductivity can and usually does change several orders of magnitude between the wet and dry states of the soil. Because the solution is not starting from a known water content or pressure state at all points in the finite element mesh, it does not have the numerical luxury of letting the solution march forward in time from a start condition to its end condition. Since there is no start condition, the solver also has no idea what the soil hydraulic conductivity value is at all points in the soil. The objective, therefore, is to be able to get the pressures and hydraulic conductivities to match throughout the mesh domain by only giving the solver a “known” condition at a very small part of the domain when the solve process begins. The way the solver gets the final answers is by iterating – that is, trying different things and moving slowly towards the singular answer. The answer is singular because for the fixed set of boundary conditions (whether pressures or fluxes) there is only

ONE unique perfectly converged solution. The objective is to get as close to the unique solution as possible – without driving yourself crazy!

Boundary condition types in steady state

In a steady state analysis there are two choices of boundary conditions: a constant pressure (or head) and a constant flux rate. For convenience the flux rate can be specified as a total nodal flux or a unit flux applied to an element edge, but the end result applied to the equations is identical. It is either a known pressure at this point, or there is a continuous inflow or outflow of water.

A few other boundary condition features have been added to the analysis to help model different real life scenarios such as seepage faces, but these are just different ways of telling the model how to come up with an H or Q boundary value to put into the equation solver. A more detailed discussion of boundary condition options is given the dedicated chapter dedicated to Boundary Conditions.

6.2 Transient

A transient analysis by definition means one that is always changing. It is changing because it considers how long the soil takes to respond to the user boundary conditions. Examples of transient analyses include predicting the time it takes the core of a dam to “wet up” when the reservoir is filled quickly; or predicting where the seepage will exit the face of a dam if a heavy rainfall event is applied over the ground surface.

In order to move forward in time during a transient analysis, you must tell the solver what the soil pressure conditions are at the start of the time period in question. In other words, you must provide initial conditions as well as current or future boundary conditions.

Initial conditions

For a transient analysis, it is essential to define the initial (starting) total head at all nodes. SEEP/W allows you to specify the initial conditions by either reading the data from an initial conditions file created in a separate analysis, or by drawing the initial water table position. It is important to recognize that the initial conditions for a transient analysis can have a significant effect on the solution. Unrealistic initial conditions will lead to unrealistic solutions that may be difficult to interpret, especially in the early stage of the transient analysis.

You can specify initial conditions by instructing the solver to use data from a previously completed analysis. The initial conditions file must be identical in geometry to the current file and can be from one of the following sources:

- A file created by a steady-state seepage analysis,
- A file created by a transient seepage analysis at a specific time step,
- A file created by the current analysis at a previously saved time step to that which the current analysis is starting,
- A file created by a SIGMA/W stress/deformation analysis, or
- A file created by a QUAKE/W earthquake dynamic analysis.

In most cases, the initial conditions are established by running a steady-state analysis. Figure 6-1 illustrates two examples of steady-state flow problems that can be used to define the initial conditions for a transient analysis. The initial conditions for seepage from a pond (part “a” in figure) might be the steady-state regional groundwater flow defined by the water table elevation underneath the pond. The

initial conditions for seepage through a dam (part “b” in figure) might be the steady-state flow condition of the dam due to the small water impoundment upstream from the dam.

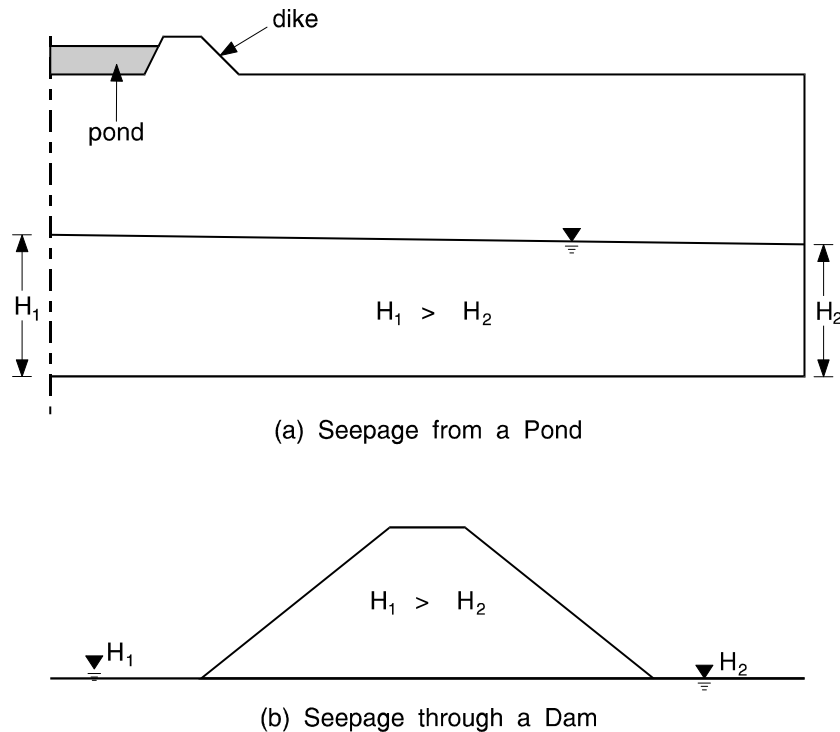
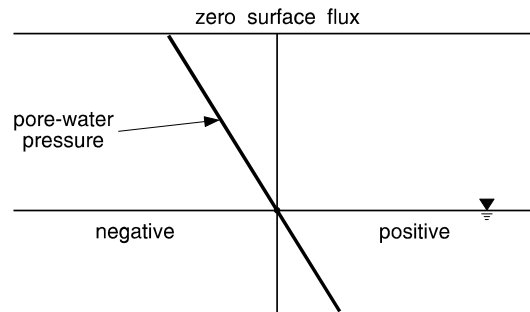


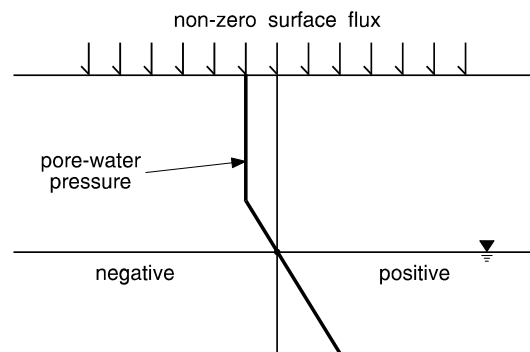
Figure 6-1 Establishing initial conditions

In a steady-state analysis, the pore-water pressure in the unsaturated zone above the water table will vary in a linear manner when the surface flux is specified as zero, as shown in Figure 6-2 part “a”. This means that the negative pore-water pressure near the surface may become too high. A more realistic estimate of the initial negative pore-water pressure can be established by specifying a small infiltration along the ground surface. The small surface flux has the effect of changing the pore-water pressure profile as illustrated in Figure 6-2 part “b”.

The magnitude of the maximum negative pore-water pressure is dependent on the shape of the hydraulic conductivity function and, to a lesser extent, on the rate of infiltration. The infiltration rate must be less than the saturated conductivity. In general, applying a small non-zero surface flux tends to give a more realistic initial pore-water pressure estimate than a zero surface flux.



(a) Pore-Water Pressure with Zero Surface Flux



(a) Pore-Water Pressure with Non-Zero Surface Flux

Figure 6-2 Effect of surface flux on negative pore-water pressures under steady state conditions

Drawing the initial water table

With this option, you specify initial conditions directly by using the Draw Initial Water Table command in SEEP/W. This is particularly useful when the location of the initial water table is known in advance. When you define an initial water table, the initial total head at each node is computed proportionally to the vertical distance between the node and the defined water table. The effect is that the pore-water pressure varies hydrostatically with distance above and below the water table. Above the water table, the negative pore-water pressure can be set to a limit to produce a pressure distribution such as shown in Figure 6-2 part “b”. Below the water table, the pressures increase hydrostatically with depth.

The definition of an initial water table gives an accurate pore-water pressure distribution when the water table is perfectly horizontal. If the water table is curved, this option gives an approximation of the actual initial conditions. Whether this approximation is or is not acceptable will depend on the nature of the problem. If this approximation is not considered adequate, then you will have to perform a SEEP/W analysis to establish the initial conditions.

Activation values

If you have a new soil region becoming active and you know it has a certain initial pressure or air pressure (AIR/W), you can use the material property activation value to initialize the parameter in that region. This value is only applied the first time a new region is active in the analysis. This approach can be used to set initial value at the start of any analysis, not just a construction sequence analysis.

Spatial function for the initial conditions

A third option is to specify directly what you think the starting values conditions will be by applying a spatial function. You can define a spatial function for pressure and have the solver point to this function result. An example of a spatial function is shown below.

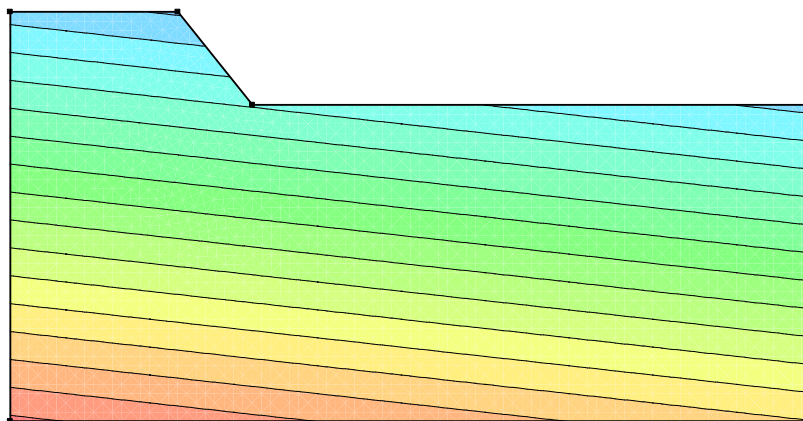


Figure 6-3 Spatial function assigned for initial pressures

No initial condition

In the event no initial condition is specified, SEEP/W will not proceed with the transient analysis. The solver will bring up an error message informing you that no initial data is available and no water table has been defined.

In summary, a reasonable initial condition must be specified in a transient analysis.

6.3 Time stepping - temporal integration

An incremental time sequence is required for all transient analyses and the appropriate time sequence is problem-dependent. In most cases it will likely be necessary to try a reasonable sequence and then adjust the sequence as necessary in response to the computed results. For example, if the migration of the wetting front is too rapid, the time steps need to be decreased; if the migration is too slow, the time steps need to be increased.

The accuracy of the computed results depends to some extent on the size of the time step. Over the period of one time increment, the process is considered to be linear. Each time step analysis is equivalent to a mini steady-state analysis. The incremental stepping forward in time is, in reality, an approximation of the nonlinear process. For the same rate of change, large time steps lead to more of an approximation than small time steps. It follows that when the rate of change is high, the time steps should be small, and when the rate of change is low, the time steps should be large.

Many seepage processes related to the dissipation of excess pore-water pressures and infiltration follow an exponential form. The dissipation or infiltration is rapid at first and then decreases with time. A typical example is the consolidation of a soil. To model this situation, the time step sequence should approximately follow an exponential form. The time steps should be small at first and then progressively increase.

Finite element temporal integration formulation

The following discussion is presented here and again in more detail in the Theory chapter, where the full development of the equations is given. At this point, it is important now to simply show the equation to highlight a key point that will enhance your understanding of the transient finite element method.

The finite element solution for a transient analysis has total head changes being a function of time as indicated by the $\{H\}$ term in the finite element equation below. In order to solve for the total heads at some point in the future, time integration can be performed by a finite difference approximation scheme.

Writing the finite element equation in terms of a “Backward Difference” finite difference form leads to the following equation (see Segerlind, 1984, pp. 183-185):

$$(\Delta t [K] + [M])\{H_1\} = \Delta t \{Q_1\} + [M]\{H_0\}$$

or

$$\{H_1\} = \frac{\Delta t \{Q_1\} + [M]\{H_0\}}{\Delta t [K] + [M]}$$

where:

Δt	=	the time increment,
H_1	=	the head at end of time increment,
H_0	=	the head at start of time increment,
Q_1	=	the nodal flux at end of time increment,
K	=	the element hydraulic conductivity matrix, and
M	=	the element water storage matrix.

Look at this equation carefully for a moment. You can ignore the braces and brackets, as they just indicate a grouping of node and element information with some geometry tied in. The thing to focus on is that in order to solve for the new heads at the end of the time increment, it is necessary to know the heads at the start of the increment, along with the average material properties calculated at the average of the new and old heads. If you do not have reasonable values for starting heads, then you make the equation difficult to solve because you use these starting heads directly in the equation AND also in the calculation of the average material properties.

Problems with time step sizes

The detailed discussion of how to compute optimum time steps is very advanced and is left to those with advanced understanding of finite element mathematics (see Segerlind, 1984, pp 190-199). For our purposes, let us just point out some key issues and make some safe conclusions.

As discussed directly above, the shape and size of an element is tied into the assembly of the $[K]$ and $[M]$ matrices. It is also plain to see that changing the time step magnitude can change the solution of the computed heads. Small time steps can cause overshoot in the calculation of the new heads and time steps that are too large can result in poor averaging of material properties at the mid point of the time step. Elements that are too large can also result in poor material property averaging, while elements that are too small can lead to overshoot problems as well.

The mathematical formulations for time step sizes generally take into account the soil's water storage capability, its hydraulic conductivity, and the size of the element. In a homogeneous, fully-saturated soil

sample, the optimum time step can be readily estimated. However, the problem in a 2D analysis with multiple soil types quickly becomes apparent. What may be an optimum time step for one region of the problem can also be a very poor choice of time step for another region; and, unfortunately, the entire problem must be solved with the same time step.

Some programs have attempted to use adaptive meshing and adaptive time stepping to deal with these issues, but the bottom line is that they cannot be applicable to all points in a mesh at all times. So, you are going to have to try some different things and use some common sense.

General rules for setting time steps

So, now that we have made the issue of time steps somewhat unclear, what do we recommend are some methods to deal with it? Some of the following points have a strong theoretical basis and some are just common sense based on years of experience.

The finite element shape is important. Triangular elements should not have any interior angles greater than 90° . Square elements can have double the time step size as triangular elements.

Since the element size is directly proportional to time step size, doubling the element size means you can double the time step. The corollary of this is that decreasing the element size and not decreasing the time steps accordingly will not improve the calculated results.

If you have a “instantaneous loading” of the system, such as instantaneous pore pressure increases due to external loading, you should set the initial time step size to be approximately equal in order of magnitude to the mv value divided by the saturated conductivity value of the soil that is instantaneously loaded.

If you are modeling a rapid filling of a reservoir, you can “instantaneous load” the model by having it be filled instantaneously, or you can use a Head versus Time boundary function to more accurately simulate the filling of the reservoir over a short, finite time period.

If you are using a detailed surface boundary flux versus time function with many different values (such as simulating the on / off nature of rainfall) make sure your time steps are small enough to allow the solver to follow the shape of the function. For example, 2 week long time steps don't work if rain falls on day 3, 8 and 17. The program would make its first flux value be that at day 14 (2 weeks), but you have no value on day 14 in your function. Also, note that if the rainfall on day 8 is 5 mm/day, and if you have solver time steps of 4 days, your function will get a value from the function on day 8, but the actual applied flux in the model would be 5 mm/day applied over 4 days for a total of 20mm of rainfall. Make sure you use 1 day time steps in this case.

To get started, take the total time you need to model and divide it by 5. Set the model to have 5 equal time steps and run it. Watch how long it takes to converge and if it even does converge. If it does not converge and you suspect it is a time step issue, choose 10 smaller time steps. Once the model solves all right, as a test, decrease the time steps some and see if the converged results change at all. The bottom line is... the ONLY way to do time steps properly is by trial and error.

Adaptive time stepping

SEEP/W will permit you to activate an adaptive time stepping routine that will insert extra time steps between your specified time steps in the event the solution is not meeting your specified time stepping or convergence criteria. Full details of the adaptive time stepping options are discussed in the Numerical Issues chapter.

6.4 Staged / multiple analyses

Multiple analyses can be included in a single GeoStudio project. Fundamentally, multiple analyses in a single Project allows different material properties and different boundary conditions to be specified across time and space. This facilitates the modeling of staged construction in which soil is added or removed over time and/or boundary conditions or material properties that change with time. Including multiple analyses in a single Project can be used for a variety of reasons such as:

- 1) Conducting sensitivity analyses for variations in material properties and boundary conditions;
- 2) Analyzing staged construction;
- 3) Establishing initial conditions for a transient analysis;
- 4) Integrating various GeoStudio products; and,
- 5) Linking together multiple transient analyses.

GeoStudio uses a parent-child terminology to describe the relative position of each analysis within a Project. Figure 6-4 displays an example of an Analysis Tree for a slope stability project. The SEEP/W steady-state analysis is the Parent and is used to define the initial pore-water pressure conditions for the two transient SEEP/W analyses. The indentation in the tree indicates that both analyses 2 and 3 have the same Parent. SLOPE/W analyses 2a and 3a are children of transient SEEP/W analyses. This naturally suggests that the pore-water pressure conditions for both stability analyses are defined using the transient seepage results.

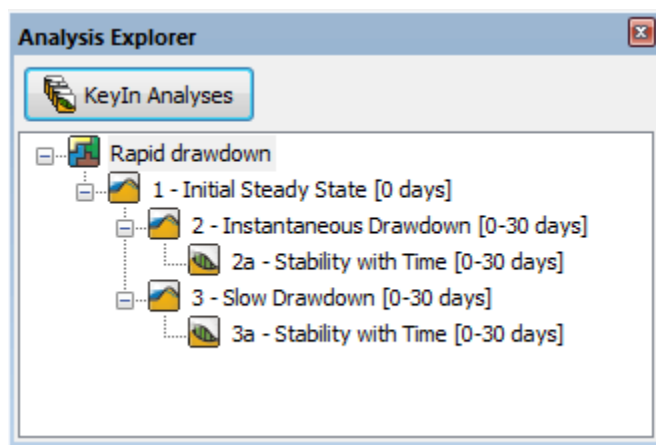


Figure 6-4 Example of an Analysis Tree in GeoStudio

One significant benefit of the Analysis tree is that all analyses related to a specific project are contained within a single file. It is no longer necessary to reference other files to establish initial conditions or integrate the various GeoStudio products.

6.5 Axisymmetric

An axisymmetric analysis can be used to simulate three-dimensional problems with symmetry about a vertical axis of rotation. The problem is defined in two dimensions, but for analysis it is as if the section is rotated about a vertical central axis. A typical example of an axisymmetric analysis is the flow into a single pumping well or flow out of a single recharge well into a uniform aquifer.

In SEEP/W the vertical symmetric axis of rotation is always at x-coordinate equal to zero. The x-coordinates in an axisymmetric finite element mesh must, therefore, all be greater or equal to zero. A typical case for an axisymmetric analysis is the drawdown cone formed by a single pumping well, as illustrated in Figure 6-5. In this figure, the pressure head contours are plotted for a steady state condition of continual pumping from the well at a rate of 0.22 cubic meters per day per radian about the x-axis.

For an axisymmetric analysis, the computed flux is per unit radian if the element thickness is specified as 1.0. If you want the computed flux value for the entire circumferential area, you must either specify the element thickness as 6.2832 (i.e., 2π radians) before you do the analysis, or simply multiply the above value by 2π after the solution is finished. You can change the element thickness for the entire mesh with the Draw Mesh Properties command.

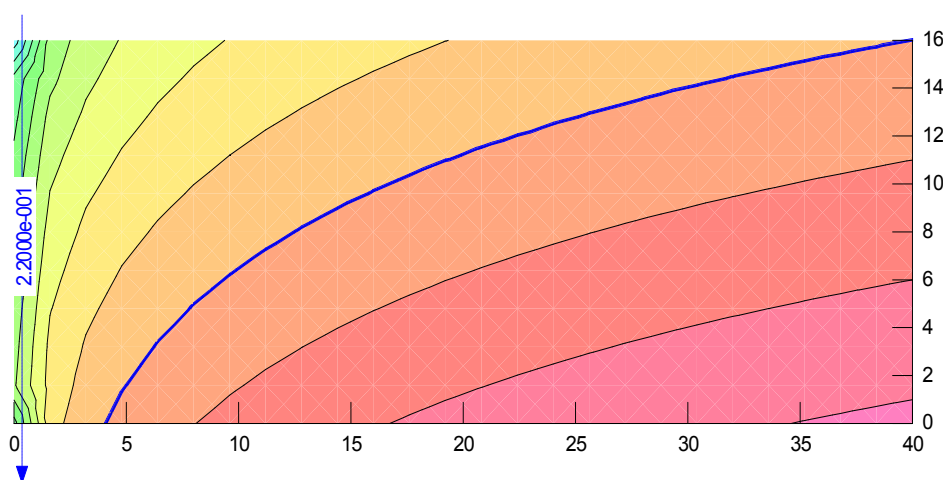


Figure 6-5 Well drawdown for an axisymmetric case (note zero x- coordinate axis)

It is not relevant to apply a small “q” unit flux boundary to the left edge of the mesh if it is at an “x” coordinate of zero because then there is no area (no radius) to compute an area over which the flux should be applied. You can, however, apply a big “Q” flux, because when you do this you are including the area of flow inside the Q value you specify.

6.6 Plan view (confined aquifer only)

A plan view analysis views the finite element mesh as lying on its side instead of standing upright in a vertical plane. This makes it possible to model the piezometric surface changes that result from extracting or injecting fluid into an aquifer. However, the analysis is limited to a perfectly flat hydraulic conductivity function because the ground must always be saturated inside an aquifer (that is, the hydraulic conductivity cannot be a function of the pore-water pressure). The limitation is not serious in the case of confined aquifers where the water pressure remains positive at all locations and at all times. However, it may be a serious limitation in the case of an unconfined aquifer. Therefore, the plan view analysis type is intended for confined aquifer only, and application to unconfined problems must be conducted with caution. The position of the piezometric surface as computed for unconfined aquifers should be viewed only as a rough approximation.

For a plan view analysis, you can define the z-coordinate (elevation) for each node and the thickness of the elements. If the z-coordinate is not specifically specified, it is taken to be zero by default. Similarly, if the thickness is not specifically specified, the thickness of the element is set to 1.0. The plan view

elevation and thickness can be set after you have defined the x- and y-coordinates of the plan view problem with the Key In Generate Plan View command. The elevation (z-coordinate) and thickness are generated on a planar basis by defining the x-, y- and z-coordinates as well as the thickness at three points.

The thickness of the elements influences the computed flux quantity across a flux section. Therefore, when you are interpreting the flow across a flux section you must take into account the aquifer thickness at the location of the flux section. Again, if you have not specified the element thickness, then the thickness is by default unity (1.0).

The z-coordinate can be thought of as the depth down to the top of the aquifer. Adopting this definition helps with interpreting the results. The pore-water pressure in a plan view analysis is computed as total head (H) minus the elevation (Z). This means that when the water pressure is positive, the water table is above the top of the aquifer and the aquifer remains saturated. When the water pressure is negative, the water table is below the top of the aquifer and the aquifer in part has de-saturated. To maintain a positive water pressure in the aquifer, the specified and computed head must be greater than the z-coordinate.

You can specify a rate of infiltration or evaporation on the ground surface of a plan view analysis using the q (unit flux) boundary condition. SEEP/W computes the contributing surface area for each node to get the nodal flux required for solving the finite element equations.

7 Functions in GeoStudio

User specified functions are used throughout GeoStudio to specify soil material properties, to specify modifier parameters for constants or other functions, or to specify boundary conditions that change over time. It is important to have an understanding of how the functions are specified and used by the solver and also to know what your options are for inputting these functions. A functional relationship between “x” and “y” data can be defined using:

- Natural and weighted splines between data points

- Linear lines between data points

- A step function between data points

- A closed form equation that is based on parameters and does not require data points

- A user written externally compiled code (dll library) that connects with GeoStudio data or data from another process (eg, Excel)

New in GeoStudio 7.1 are spatial functions which return a different value based on both x and y coordinate within any given soil material. This option is available for strength parameters in SLOPE/W and for activation values in the other finite element codes. Activation values are starting pressures or temperatures, for example, that are present in the ground when the model first starts a transient process.

The type of function you choose to use will depend on what your needs are.

In many cases a function you need can be estimated from other data you have input. An example is the hydraulic conductivity function for soils that is based on a user input water content function. Several GeoStudio material models require functions that may be estimated if you do not already have a full set of data.

7.1 *Spline functions*

A spline function is a mathematical technique to fill in the gaps between adjacent data points with curved line segments. Unfortunately, all our data points do not always fit nicely along a path with a predictable curvature such as a logarithmic or exponential decay. Many of the functions in geo-technical engineering have double curvature with an inflection point between. Consider the water content function that is initially concave downwards, and then at higher suctions is concave upwards. Splining is an advantageous technique to fit lines through these data points because the spline settings can be altered to fit almost any set of data.

In GeoStudio you can control the look of a spline function by adjusting its degree of curvature and its level of fit with the input data points. Consider the two images below.

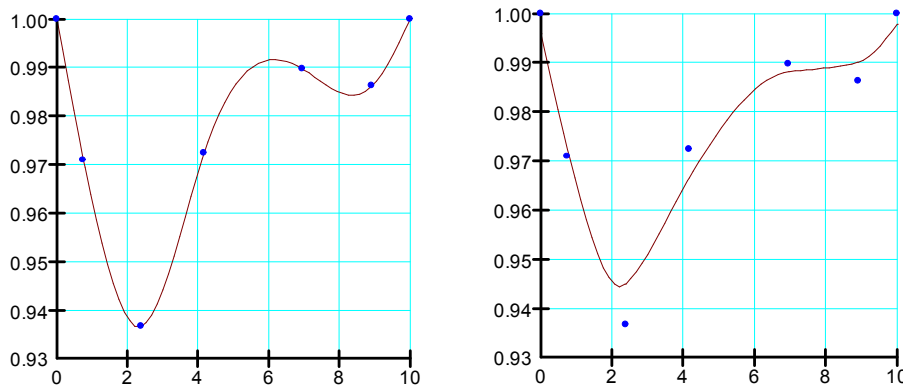


Figure 7-1 Spline functions with different settings

The left image has the spline fit almost exactly through the data points with fairly curved segments. The right image has more linear segments that only fit the data approximately. The range of fit and curvature is controlled by two “slider controls” and can range between values of zero and 100%. The important thing to note is that the solver will use the data represented by the splined fit. What you see in the function set up is EXACTLY what the solver will use when needed.

Slopes of spline functions

Sometimes, the solver does not require the “Y” value of a function at a given “X” value but the slope of the function at a given “X” value. This is the case for the water content function where the slope is used directly in the solution of the transient seepage and air flow equations. You must be careful when setting spline values because while a spline may look smooth, its slope may not be so.

The two images below are the slopes of the two functions shown above. You can see that the more natural curved function (left side images) with 100% curvature and exactness in the spline settings produces a much smoother slope function than the approximated function. While not usually critical, you should know if the function you are using is dependent on its slope being well behaved.

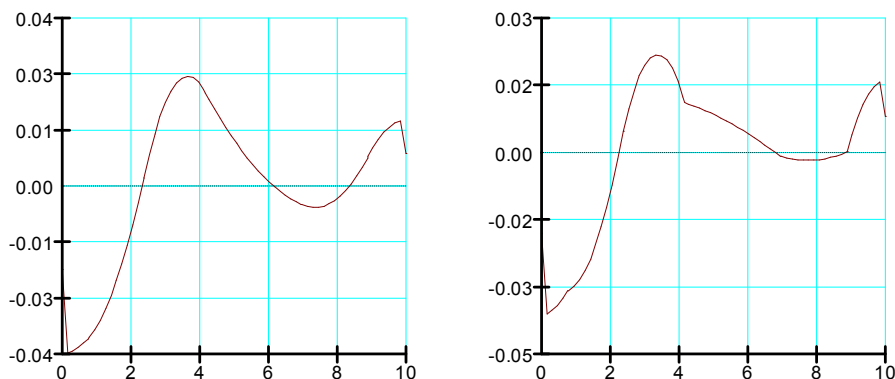


Figure 7-2 Slope of spline functions

7.2 Linear functions

A linear function is a spline function with the curvature setting to 0% and the fit set to 100% exact as shown below.

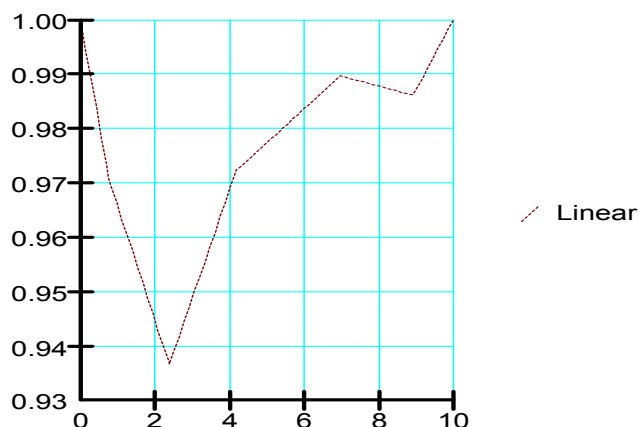


Figure 7-3 Linear fit spline

7.3 Step functions

GeoStudio 2007 has an option for functions that result in “steps” between data points. This can be useful if your data changes abruptly over time, for example, rainfall on different days. When you use a step function, you need to be careful of the location of the blue data point. You can see that the functions will assume the starting time of the step is at the data point and that its duration extends just up to but not reaching the next data point.

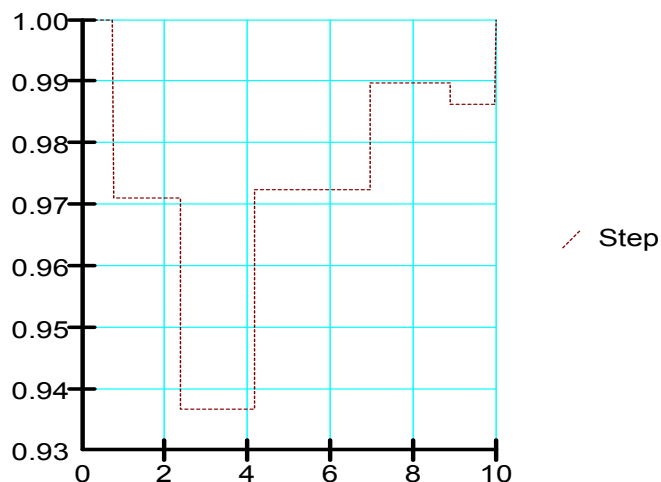


Figure 7-4 Step function

A comparison of all four data point functions is shown below on one image. When multiple functions are viewed simultaneously in GeoStudio, the data points are hidden and just the computed functions are displayed.

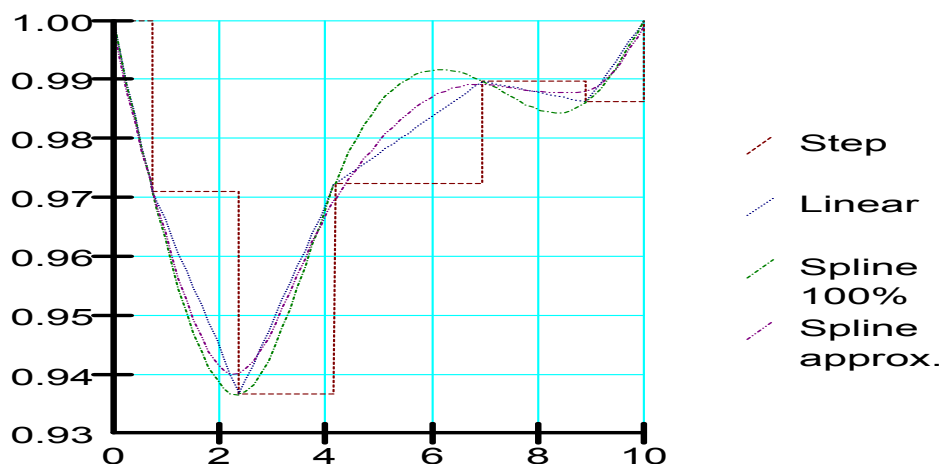


Figure 7-5 Comparison of all data point functions

7.4 Closed form curve fits for water content functions

The storage function is defaulted to be represented by a spline function; however, it is possible to have the function represented by a closed form equation that is fit to the data. Two common methods exist in the literature for water content functions: the Fredlund and Xing method, and the Van Genuchten method. Each of these curve fits require that you enter fitting parameters that are usually published or provided by soil laboratories. The only advantage to using these techniques in GeoStudio 2007 is that you do not have to enter a series of data points. If you know the fit parameters, you may enter them directly to obtain the function. More information about these two fit equations are provided in the chapter on Material Models and Soil Properties in this book.

7.5 Add-in functions

GeoStudio Add-Ins are supplemental programs run by the solver as part of a GeoStudio analysis. A Function Add-In is an object that takes the place of a function defined in GeoStudio, and offers the flexibility of computing function results that vary dynamically based on the current analysis state. For example, Add-Ins can be assigned to Slip Surface Slices (via strength functions), Mesh nodes (via boundary condition functions), and Mesh gauss points (via material property functions). Please consult the Add In Developers Kit (SDK) available on the website (www.geo-slope.com/downloads) for full details.

7.6 Spatial functions

A spatial function in SEEP/W can be used to establish starting pressure profile over a two-dimensional domain. When you first create a spatial function you will not see its contoured colors appear on the geometry. However, once you assign the function as the initial condition in Key In Analysis Settings, you can return to the Key In Spatial Function command, make changes and edits to the function data, and

see instantly what the new function will look like when applied to your model. An example of this is shown below for initial pore-water pressures which would be applied in the seepage part of the analysis.

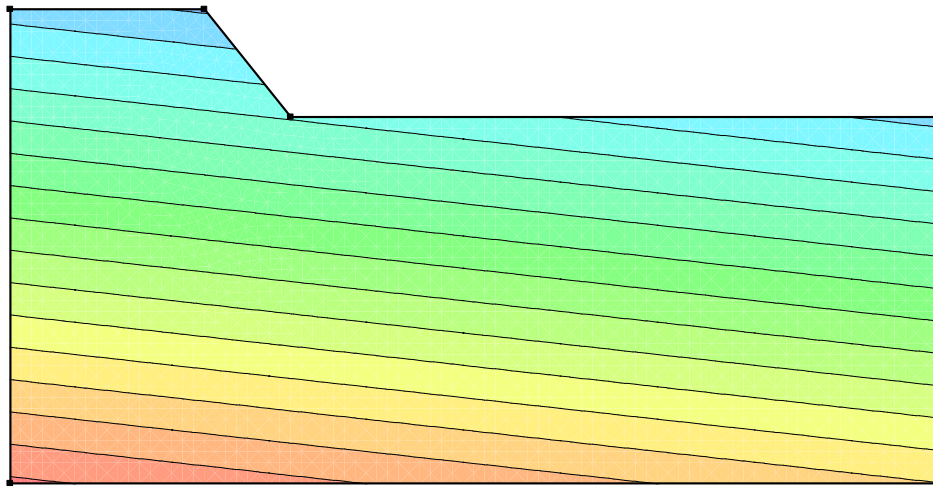


Figure 7-6 Example of spatial function assigned to model

8 Numerical Issues

Entire textbooks are written on numerical issues related to finite element analysis. While modern computers and powerful graphics can make defining an analysis relatively fast and easy, they cannot necessarily deal with all of the challenges of taking a complex, non-linear, transient physical process and trying to replicate it in terms of discretized time and space.

A variety of approaches have been used to deal with many of the numerical issues, but unfortunately there is no single method for dealing with all problems. Some numerical issues relate to restrictions in computer hardware such as rounding off of non-integer variables during mathematical operations. Some issues relate to highly non-linear soil properties or that the equations do not apply to all cases (for example, flow through unsaturated coarse gravel is not likely Darcian flow). Other issues relate to our spatial discretization being too coarse, or our temporal discretization being too small.

There are numerical solvers that make use of adaptive meshing or adaptive time stepping or both in an attempt to be more suited to a wider range of problems. All of these, however, have mathematical limitations regardless of the claims of the software developer. It becomes somewhat risky to rely on a solver that “handles it all” if you do not know what the limitations.

Some finite element solutions attempt to march forward in time by evaluating soil properties at the previous, the current or a mid-time step average. Some solvers simply make assumptions that limit the ability of the software to handle real world problems. Finally, some solvers may only work if the user provides an initial guess of the solution that is close to the desired solution. In other words, the solution is started by pointing the solver ‘in the right direction’.

Fortunately, sound judgment and common sense can usually overcome most of these challenges and result in meaningful interpretations of soil behavior.

It is not always possible to get an exact solution for particularly challenging cases, so you should not necessarily be seeking an exact solution. If the problem is so difficult that it is not solving reasonably, then it is very likely that either mistakes have been made in the input, or, that you are pushing the envelope of the physical theory applied in the model. This chapter looks at some of these issues as they pertain to SEEP/W.

8.1 Convergence

The finite element equation generally takes the form:

$$[K]\{h\} = \{Q\}$$

where $[K]$ is the global property matrix, $\{h\}$ is the vector of nodal primary values, and $\{Q\}$ the right side forcing vector. The global assemblage of finite element matrices contains material properties that could be a function of the solution. A commonly used numerical procedure for coping with material non-linearity involves repeatedly solving the finite element equations and updating the material properties based the solution at the previous iteration. Convergence is obtained if successive solutions are equal within a specified tolerance or the maximum number of iterations is reached. The GeoStudio products determined convergence based on two parameters:

- Significant figures
- Minimum difference

Significant figures

Significant figures of a number are those digits that carry meaning as to the precision of the number. Leading and trailing zeros simply provide a reference as to the scale of the number. Consider a number like 5123.789. If we say that the number is precise to two significant figures its precision is 5.1×10^3 ; if it is three significant figures then its precision is 5.12×10^3 , and if it is four significant figures then its precision is 5.124×10^3 .

In GeoStudio, the user is allowed to specify the desired significant figures or digits for comparison of the primary variable(s) from the finite element solution. Specifying two significant digits means that when the primary variable(s) from two successive iterations are the same to a precision of two significant figures, they are deemed to be the same or they are said to have converged.

Minimum difference

Computer computations inherently produce numerical noise; that is, digits that have no significance. So when comparing floating point numbers it is necessary to filter out the insignificant digits.

GeoStudio does this with a user specified minimum-difference value. If the difference between two successive primary variables at a node is less than this minimum specified value, the two values are deemed to be the numerically equivalent and the solution is converged without giving consideration to the significant figure criteria. For example, convergence at a node would be designated if the minimum difference was specified as 0.001 and the difference in the primary variable(s) between successive iterations was less than this value.

Consider two numbers such as 1.23×10^{-6} and 1.23×10^{-7} . These two numbers have the same number of significant digits but the difference (1.11×10^{-6}) is small and may have no physical meaning in the context of the analysis. The two numbers are consequently deemed to be equivalent within the tolerance.

8.2 Evaluating convergence

GeoStudio provides several tools for judging or evaluating whether the computed results have converged to an acceptable solution.

Mesh view

View Preferences | Node Convergence paints the convergence state of the nodes when in Results. Figure 8-1 illustrates that an (X) is painted if the solution does not meet the significant figure requirement and a circle is painted on nodes that have a differences in the primary variable with iteration that is less than the minimum. The nodal convergence painting assists with identifying areas of the domain that are not meeting the convergence criteria. The solution in these regions can then be further inspected other features.

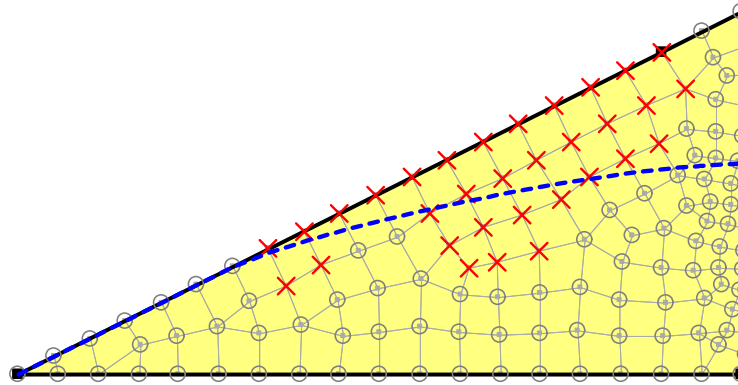


Figure 8-1 Nodes that have not achieved the convergence criteria

Graphs

A number of graphs can be generated to judge convergence based on Data from Nodes (Selected by the User) or from all nodes or Gauss regions within the entire domain. Figure 8-2, for example, shows the number of unconverged nodes versus iteration. The number of unconverged nodes steadily decreases towards 0 after 223 iterations. Graphs of this type are useful for examining the number of iterations required to achieve convergence, to detect whether the iterative process stopped at the specified maximum number of iterations and to detect whether there were oscillations during iteration.

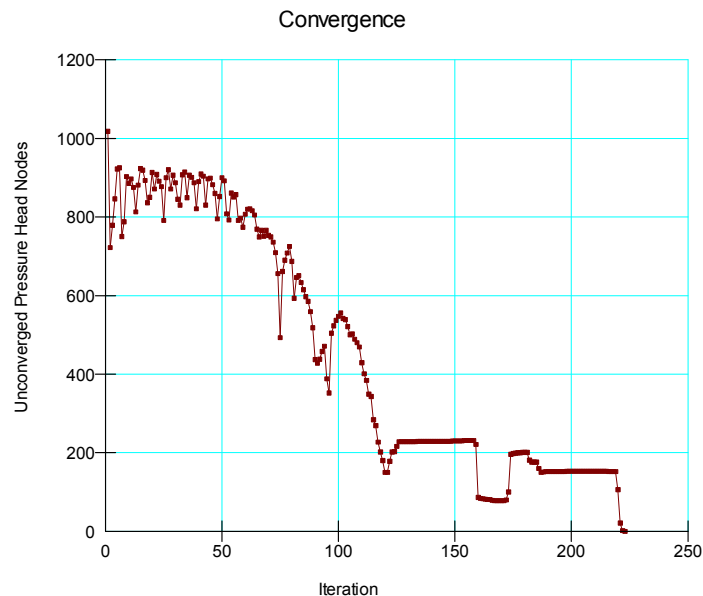


Figure 8-2 Unconverged nodes versus number of iterations

Figure 8-3 shows a graph of the hydraulic conductivity at every Gauss integration point with the entire domain that was used to obtain the solution versus the input function. The solution has not converged;

consequently the ‘Actual’ and input functions do not match. In contrast, Figure 8-4 shows a near match between the ‘Actual’ and input functions when convergence is obtained.

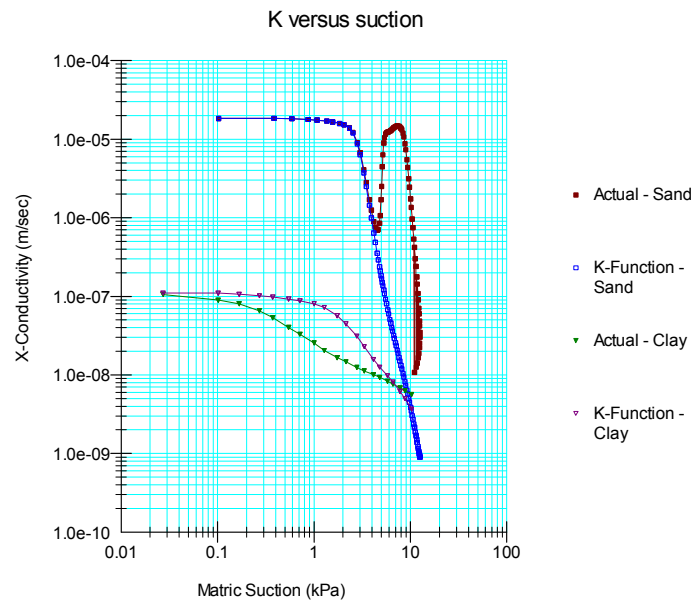


Figure 8-3 Conductivity versus suction illustrating non-convergence

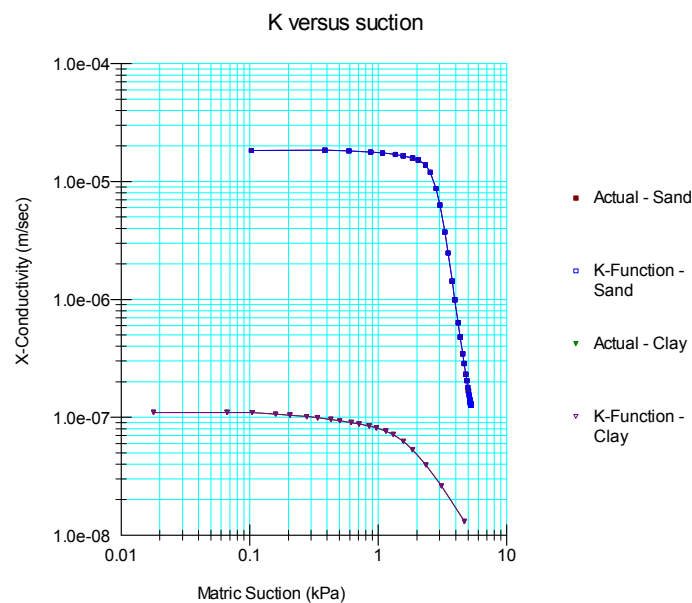


Figure 8-4 Conductivity versus suction graph illustrating a converged solution

Commentary

An unsaturated seepage analysis is a highly nonlinear process and consequently it can sometimes be difficult to obtain a converged solution. Evaluating the convergence in detail using the aforementioned techniques is highly advisable.

These tools are available at the end of a steady-state analysis or at the end of every *saved* time step in a transient analysis.

8.3 Under-relaxation

Successive solutions can diverge and/or oscillate if the material properties are highly non-linear. In this case, some form of under-relaxation is required. Under-relaxation procedures attempt to mitigate large variations in the material properties that are the source of the non-linearity. For instance, the conductivity parameter characterizing water transfer can vary by many orders of magnitude over a small pressure range. The inclusion of latent heat effects in an energy transfer analysis is another example of extreme material non-linearity. Divergence of the solution after two successive iterations can therefore be mitigated by limiting – under-relaxing – the variation of the material properties used to calculate the finite element matrices. This in turn exerts a control on the difference between successive solutions and produces a less chaotic progression towards a converged solution. The under-relaxation parameters are specified in the Convergence settings of the analysis definition and include:

1. Initial Rate (e.g. 1);
2. Minimum Rate (e.g. 0.1);
3. Rate Reduction Factor (e.g. 0.65);
4. Reduction Frequency (e.g. 10 iterations).

The Initial Rate essentially controls the allowable variation in the solution between successive iterations. A value of 1 corresponds to repeated substitution with no under-relaxation. The under-relaxation rate is reduced by multiplication of the Rate Reduction Factor once the Reduction Frequency is exceeded. For example, the under-relaxation rate would be 0.65 after the 10th iteration, 0.65² after the 20th iteration and so on until the under-relaxation rate is less than or equal to the minimum rate.

The default parameters may not be ideal for some numerically challenging problems. For example, it may be advantageous to immediately commence under-relaxation if the material properties are highly non-linear by specifying an Initial Rate that is less than 1 (e.g. 0.65). The Minimum Rate might also have to be reduced if the solution oscillates slightly around the solution (e.g. 0.01). Other variations on this strategy are possible, such as retaining the Initial Rate of 1 but reducing the Reduction Frequency (e.g. 5 iterations) and Minimum Rate (e.g. 0.01). Ultimately some form of numerical experimentation is required and convergence must be judged by using the previously mentioned techniques.

8.4 Gauss integration order

The details of numerical integration are provided in the appendices, along with a discussion of how different integration orders can affect results for various types of elements. Part of this discussion is repeated here as it pertains to improving solution convergence.

The appropriate integration order is a function of the presence of secondary nodes. When secondary nodes are present, the interpolating functions are nonlinear and consequently a higher integration order is required. Table 8-1 gives the acceptable integration orders.

Table 8-1 Acceptable element integration orders

Element Type	Secondary Nodes	Integration Order
Quadrilateral	no	4
Quadrilateral	yes	9
Triangular	no	1
Triangular	yes	3

It is also acceptable to use four-point integration for quadrilateral elements that have secondary nodes. This is called a reduced integration order (see Bathe, 1982). Acceptable results can be obtained with reduced integration. For example, reduced integration is useful in saturated zones where the hydraulic gradient is low and the hydraulic conductivity is constant. Selective use of reduced integration can greatly reduce the required number of computations.

It is also possible to use three-point and nine-point integration with elements that have no secondary nodes. However, the benefits of this are marginal, particularly for quadrilateral elements. Nine-point integration for a quadrilateral element involves substantially more computing than four point integration, and there is little to be gained from the additional computations. As a general rule, quadrilateral elements should have secondary nodes to achieve significant benefits from the nine point integration.

The situation is slightly different for triangular elements. One-point integration means the material properties and flow gradients are constant within the element. This can lead to poor performance of the element, particularly if the element is in an unsaturated zone where the hydraulic conductivity varies sharply with changes in pore-water pressure. Using three point integration, even without using secondary nodes, can improve the performance, since material properties and gradients within the elements are distributed in a more realistic manner. The use of three point integration in triangular elements with no secondary nodes is considered acceptable for triangular elements in a mesh that has predominantly quadrilateral elements. This approach is not recommended if the mesh consists primarily of triangular elements with no secondary nodes.

In general, it is sufficient to use three-point integration for triangular elements and four-point integration for quadrilateral elements. In situations where there is an unsaturated zone with hydraulic conductivity that varies sharply within an element, it is best to use quadrilateral elements with secondary nodes together with nine-point integration.

8.5 Equation solvers (direct or parallel direct)

SEEP/W has two types of equation solvers built into it; a direct equation solver and a parallel direct equation solver. Both offer certain advantages.

Select the direct equation solver option if you want the system equations to be solved using a Gauss elimination skyline direct solver. The processing speed of the direct solver is bandwidth (the maximum node number difference of all the elements in a domain) dependent. In other words, the direct solver is very fast when solving simple problems with small bandwidth, but it can be quite slow when solving more complex problems with a large bandwidth. SIGMA/W automatically sorts the nodes so that the bandwidth is the smallest possible value, which helps the solution solve faster using the direct solver. By default, the direct equation solver is selected.

Select the parallel direct equation solver option if you have a larger mesh. The parallel solver will save the matrices in a compressed format to eliminate zero's and it has many advanced schemes to solve large systems of equations more efficiently. It also offers the ability to make use of multiple processors on a

computer if they are available. The disadvantage of this solver is that it is a bit slower when the models are smaller in size.

If in doubt, try each solver and choose the one that offers the best performance.

8.6 Time stepping

An incremental time sequence is required for all transient analyses and the appropriate time sequence is problem-dependent. In most cases, it will likely be necessary to try a reasonable sequence and then adjust the sequence as necessary in response to the computed results. For example, if the migration of the wetting front is too rapid, the time steps need to be decreased; if the migration is too slow, the time steps need to be increased.

The accuracy of the computed results is dependent to some extent on the size of the time step. Over the period of one time increment, the process is considered to be linear. Each time step analysis is equivalent to a mini steady-state analysis. The incremental stepping forward in time is, in reality, an approximation of the nonlinear process. For the same rate of change, large time steps lead to more of an approximation than small time steps. It follows that when the rate of change is high, the time steps should be small, and when the rate of change is low, the time steps should be large.

Many seepage processes related to the dissipation of excess pore-water pressures and infiltration follow an exponential form. The dissipation or infiltration is rapid at first and then decreases with time. A typical example is the consolidation of a soil. To model this situation, the time step sequence should approximately follow an exponential form. The time steps should be small at first and then progressively increase.

Estimating a starting time step size

Appropriate time steps are related to the element material properties and the element size. The initial time step for the dissipation of excess pore-pressure from a saturated column can, for example, be estimated from:

$$\Delta t = (0.1) \gamma_w \frac{m_v}{K_{sat}} L^2$$

Where m_v is the coefficient of volume compressibility as related to consolidation, K_{sat} is the saturated hydraulic conductivity and L is the length of the drainage path in one element, or simply the element size. More simply, Δt is proportional to:

$$\Delta t \approx \frac{m_v}{K_{sat}} L^2$$

This equation is sometimes useful for estimating an approximate initial Δt .

Time steps too small

Time steps in a numerical analysis can sometimes be too small. The thought that the accuracy of the solution will increase as the time step size decreases is not always true. Time steps that are too small can cause the solution to “over-shoot” the specified initial and boundary conditions. The computed head for example can be larger than the starting conditions.

As a general rule, it is better make the time steps bigger rather than smaller than the ideal, particularly in the early stages of an analysis.

After achieving a realistic solution, confidence in the solution can be elevated by experimenting with the time stepping to determine if it has a significant effect on the results.

9 Flow nets, seepage forces, and exit gradients

This chapter discusses three aspects of pre-finite element seepage analyses that often cause users some concern. The concern usually arises because the user is trying to fit finite element output into the context of more traditional, and often approximate, analysis techniques and ideas. Issues discussed include flow nets, seepage forces and exit gradients. This chapter is, in some ways, about helping you to change your thinking.

9.1 Flow nets

Seepage flow through a homogeneous isotropic medium under steady state conditions is described by the Laplacian equation. The form of the equation is,

$$\frac{\partial^2 h}{\partial x^2} + \frac{\partial^2 h}{\partial y^2} + \frac{\partial^2 h}{\partial z^2} = 0$$

This is one of the most basic partial differential equations known in mathematics. A graphical solution to this equation is what is known as a flow net. A flow net is in essence is map of contours of equal potential crossed with flow lines. For the flow net to represent a correct solution to the Laplacian equation, the equipotential lines and flow lines must follow certain rules. The flow lines must for example cross the equipotential lines at right angles. Also, the area between two adjacent flow lines is called a flow channel and the flow in each channel has to carry the same amount of flow. A correctly constructed flow net is a graphical solution to the Laplacian equation.

Prior to the availability of computers and numerical methods such as finite element analyses, flow nets were often the only available solution to solving seepage analysis problems. Consequently, over the years, flow nets have become deeply entrenched in any published discussion and presentation on the subject of seepage. Most textbooks that have a section on seepage talk about how to construct and interpret flow nets. In spite of the apparent simplicity, constructing a good flow net that meets all the criteria is not a trivial task. Anybody who has ever tried to construct a flow net can attest to the difficulties.

Fortunately, now that computers and numerical software tools are so readily available, it is no long necessary to rely on graphical solutions to the Laplacian equation. Numerical solutions can now be obtained more quickly and easily than constructing a flow net. Moreover, a numerical solution provides all the information obtainable from a flow net plus much more. Furthermore, numerical analyses can provide solutions for highly complex situations for which it is not possible to construct a flow net. Flow nets, for example, are nearly impossible to construct for a complex stratigraphy or when there is both saturated and unsaturated flow, or when the flow is transient. With a software tool like SEEP/W these types of situations can be readily considered in a seepage analysis.

SEEP/W does not create a true flow net because flow nets can only be created for a few special situations. SEEP/W, however, does compute and display many elements of a flow net which are useful for interpreting results in the context of flow net principles. For example, flow lines must be approximately perpendicular to equipotential lines. Features like this provide a reference point for judging the SEEP/W results.

This chapter describes the SEEP/W features which make it possible to view the results in the context of a flow net and outlines the simple cases for which an approximate flow net can be created. Also, examples are presented for which it is not possible to construct a flow net.

Equipotential lines

SEEP/W is formulated in terms of total hydraulic head. Contours of total head are the equivalent of equipotential lines. So equipotential lines can be drawn and displayed by creating a plot of total head contours. Figure 9-1 shows total head contours. They are identical to equipotential lines in a flow net. In this example there are eight equipotential drops from 20 to 12, each 1 meter.

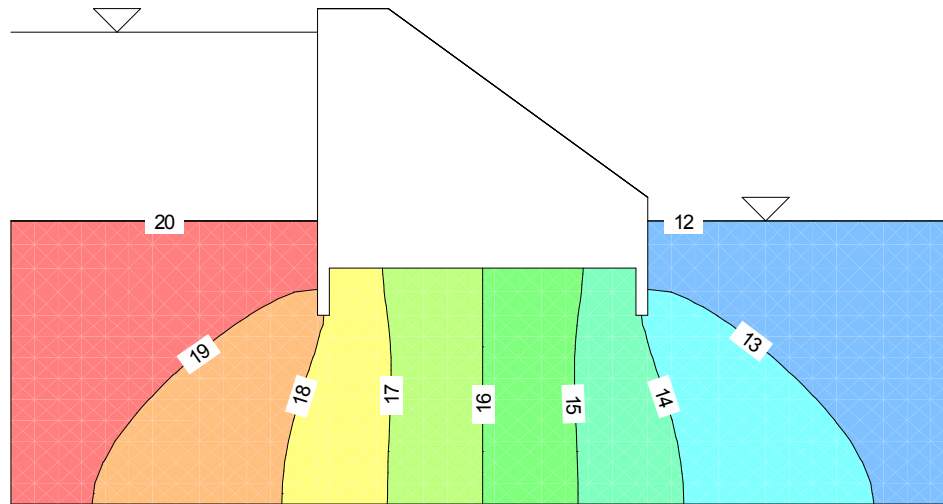


Figure 9-1 Plot of total head contours or equipotential lines

Flow paths

In SEEP/W you can draw paths as illustrated in Figure 9-2. These are lines that an imaginary droplet of water would follow from entrance to exit; they are not flow lines in the true context of a flow nets.

It is very important to recognize that flow paths drawn on top of total head contours are not flow channels. In flow net terminology, the area between two flow paths is called a flow channel. In a flow net, the amount of flow between each flow line must be the same; that is, the amount of flow is the same in each flow channel. Drawing flow paths in SEEP/W does not guarantee this. When visually the combination of total head contours and flow paths in SEEP/W look very much like a flow net as in Figure 9-2, the amount of flow in each channel will be approximately the same. This is true only by implication. SEEP/W makes no calculation to this effect.

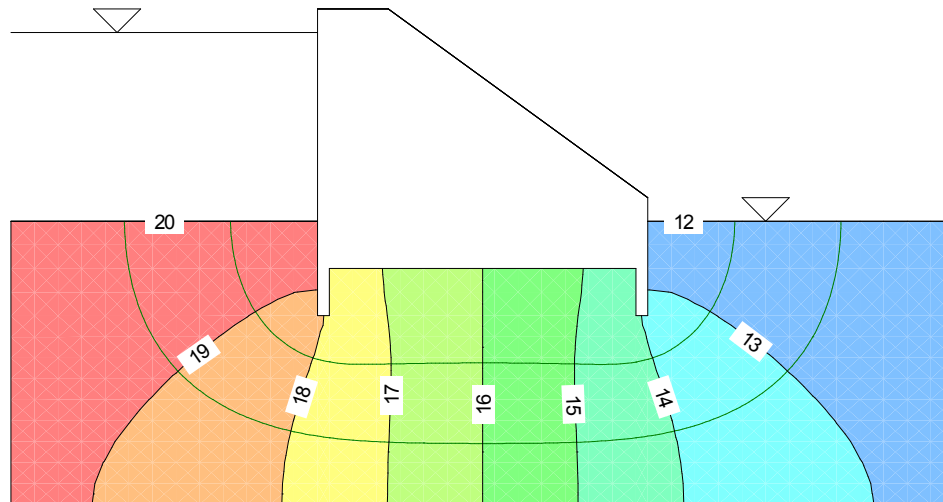


Figure 9-2 Flow net approximation

Note in Figure 9-2 that the flow paths cross the equipotential lines at right angles, as they rightly must for a homogenous section. This is useful information when judging SEEP/W results. If the paths do not cross the total head contours at right angles (or at least close to right angles) for a homogenous section, there is reason to believe something is wrong perhaps with the input data. This is a useful visual tool to mentally check the validity of your analysis.

As is evident in Figure 9-2, a very good approximation of a flow net can be created with SEEP/W by visually placing flow paths on top of total head contours at strategic points. The fact that you can so quickly draw flow paths and then delete and replace them makes it possible with a little experimenting to make a nice-looking flow net.

Flow channels

It is possible for simple cases to compute flow lines so that they create exact true flow channels. SEEP/W can be tricked into making this calculation by reversing boundary conditions. Head boundaries become flow boundaries and flow boundaries become head boundaries.

Say that in the above example of seepage under a structure with two cutoffs, we want to create six flow channels. This can be done by specifying a total head of zero along the vertical ends of the problems and along the bottom, by specifying a total of 6 along the perimeter of the structure and by specifying a Q (flux) of zero along the flow entrance and exit areas. The blue triangles in Figure 9-3 indicate a flow boundary and the red circles represent a head boundary.

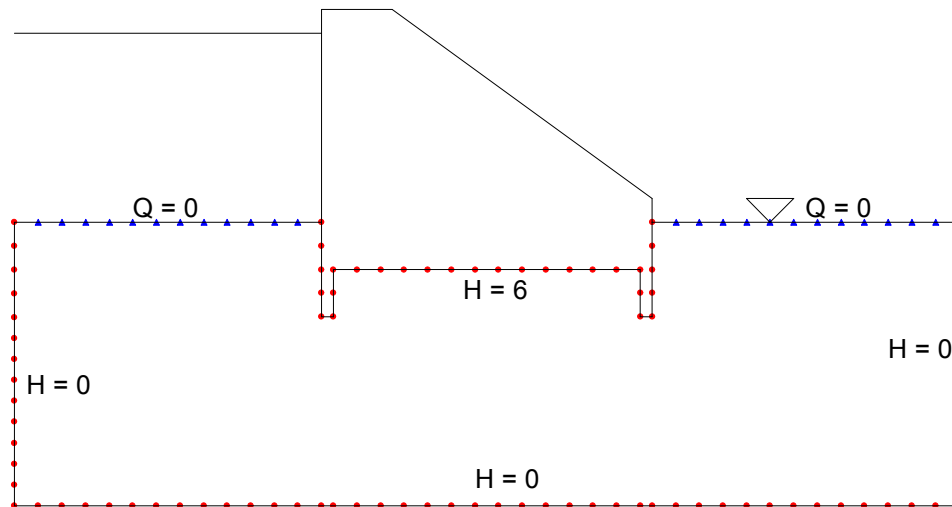


Figure 9-3 Boundary condition for computing flow channels

The resulting computed flow channels and true flow lines are shown in Figure 9-3. In SEEP/W the flow lines are plotted as contours of total head, but they actually represent flow lines (Figure 9-4). The SEEP/W drawn flow lines are now actually equipotential lines. Again, by visually placing the flow lines you can create a good-looking approximation of a flow net as pictured in Figure 9-5. Now the drop in head between equipotential lines is only approximate, not like in Figure 9-1 where the drop is exactly the same between equipotential lines.

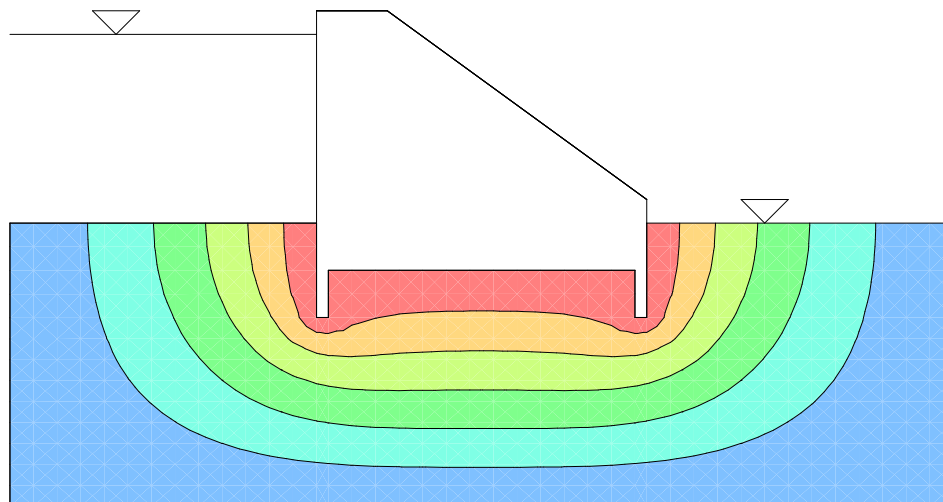


Figure 9-4 Computed flow lines and flow channels

Creating a picture of flow channels in practice is not all that useful. Usually the main objectives of a seepage analysis are to obtain a plot of pore-water pressure distribution and to determine the quantity of seepage. This information is obtainable from a conventional seepage analysis; a flow channel analysis is not required for this purpose.

A flow channel analysis is primarily of interest to show that SEEP/W can be tricked into doing this type of analysis and that the results are reasonable which in a sense validates the SEEP/W formulation.

Also, if a reasonably good flow net is required for discussion and communication purposes, this is a technique that can be useful.

The technique is also good for teaching students flow net principles.

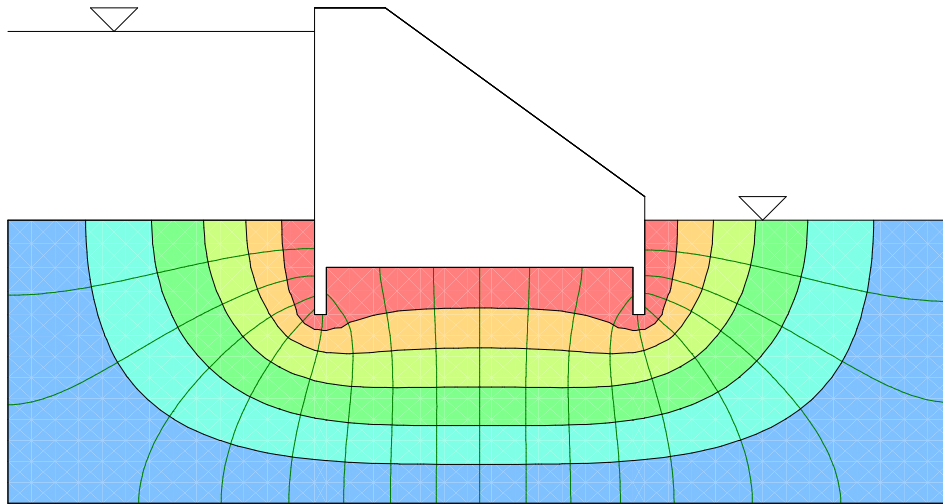


Figure 9-5 SEEP/W flow paths represent equipotential lines in a flow channel analysis

Flow quantities

Computing flow quantities is usually of primary interest in a seepage analysis. With SEEP/W this is done by defining a flux section. Figure 9-6 shows the example under discussion here with a flux section. The total seepage under the structure is $2.5 \times 10^{-4} \text{ m}^3$ per second, as indicated on the flux section. The conductivity was specified as $1 \times 10^{-4} \text{ m/sec}$.

Flow quantities can be estimated from a flow net as the total head drop times the conductivity times a ratio of the number of flow channels to the number of equipotential drops. The ratio in Figure 9-6 is 3 divided by 8. The total head drop is 8. In equation form,

$$Q = (\Delta H)K \frac{n_f}{n_d}$$

$$Q = (8)(1 \times 10^{-4}) \frac{3}{8}$$

$$Q = 3 \times 10^{-4}$$

This approximation from the flow net equation is reasonably close to the SEEP/W computed flux. The difference between the two values is confirmation that the estimated flow paths are only approximate. To get the same value from the flow net, 2.5 flow channels are required for the 8 equipotential drops.

The SEEP/W computed value is the correct Q , since flow nets are at best always an approximation.

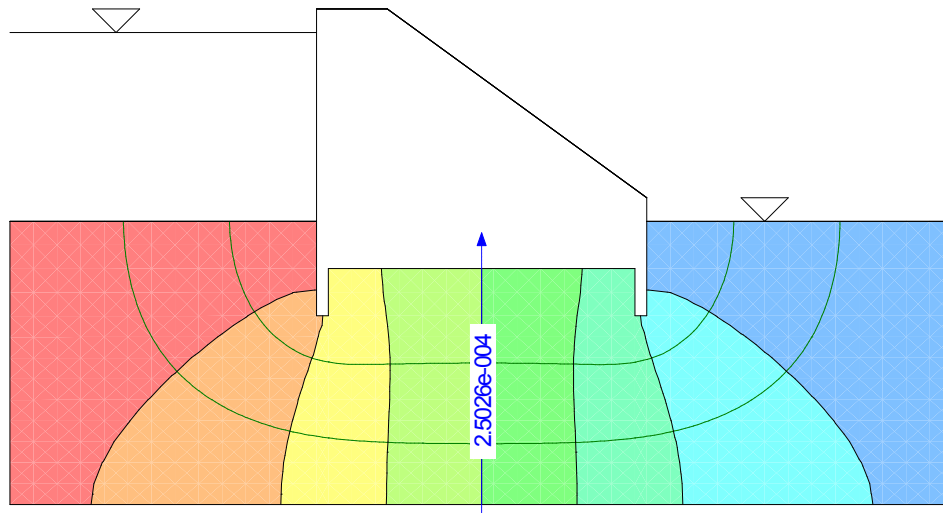


Figure 9-6 Analysis with a flux section

Uplift pressures

Almost any soil mechanics text book that has a section on seepage includes an example on how a flow net can be used to estimate the uplift pressure on a hydraulic structure such as discussed here. Producing such a plot is easy and straight forward with the SEEP/W graph function. Figure 9-7 shows the pressure from a point just outside the structure on the upstream side to a point downstream of the structure at the base of the structure. Note the pressure drops across the cutoff walls at both ends of the structure.

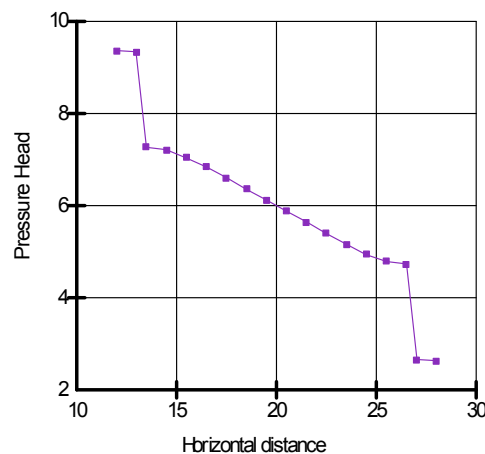


Figure 9-7 Distribution of uplift pressure on base of structure

Limitations

Flow nets can be approximated with SEEP/W provided the seepage domain is completely saturated and there are no extreme contrasts in hydraulic conductivity. However, when a section has an unsaturated zone, it is not possible to create even an approximate flow net. In the unsaturated zone there is a large variation in conductivity. Each Gauss integration point has its own conductivity depending on the pore-water pressure. In some parts of the unsaturated zone the conductivity may get so low that there is essentially no flow. This is illustrated in Figure 9-8. There is one flow path that stops in the section.

Physically this means it has reached a dead zone; that is, the conductivity is so low the path cannot be drawn beyond this point.

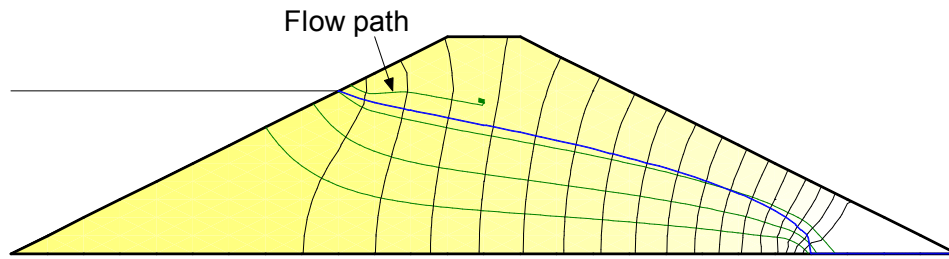


Figure 9-8 Flow paths in unconfined flow

Flow paths can be drawn below the phreatic surface and in the capillary rise zone above the phreatic surface, but not above the capillary rise. If flow paths are drawn high up in the unsaturated zone, they usually have a jagged irregular shape and as result have no meaning. They certainly do not adhere to flow net principles.

The equipotential lines shown in Figure 9-8 are valid in both the saturated and unsaturated, but the flow paths are not.

Worth noting here as well is that in a saturated-unsaturated seepage formulation such as SEEP/W, the phreatic surface is not a true flow line as in the context of a flow net. The phreatic surface is merely a line of zero pressure. A flow path can actually cross the line of zero pressure as illustrated in Figure 9-8.

In traditional flow nets for unconfined flow as in embankments, the phreatic surface is a no-flow boundary and a line of zero pressure. When finite element methods were first applied to unconfined seepage problems, it was necessary to adjust the mesh until the pressure along the top of the mesh was zero. Numerically this was awkward and difficult to achieve, and greatly limited the types of problems that could be modeled. So while this made it possible to construct flow nets in the saturated zone, it was suitable only for a few simple cases. The SEEP/W formulation and approach is applicable to a much wider range of problems and more accurately represents the real physical conditions. Moreover, SEEP/W provides a more complete picture of pressure distributions and flow quantities than what is available from a traditional flow net for unconfined flow.

9.2 Seepage forces

Soil Mechanics by Lambe and Whitman (1969, John Wiley and Sons Inc.) has a detailed discussion on seepage forces and when they are or are not relevant. The discussion that follows is based on their work.

If you have concern regarding seepage forces then you should read this section. If you are not concerned or aware of seepage forces, then please skip over this section... you don't need to know about seepage forces in GeoStudio because it is important to understand that a rigorous seepage analysis can be done without any reference to seepage force if the analysis formulation uses a consistent approach which considers total unit weight and boundary forces... as do all GeoStudio models.

Forces on a soil element

In the image below we can see different ways of considering forces acting on a soil element.

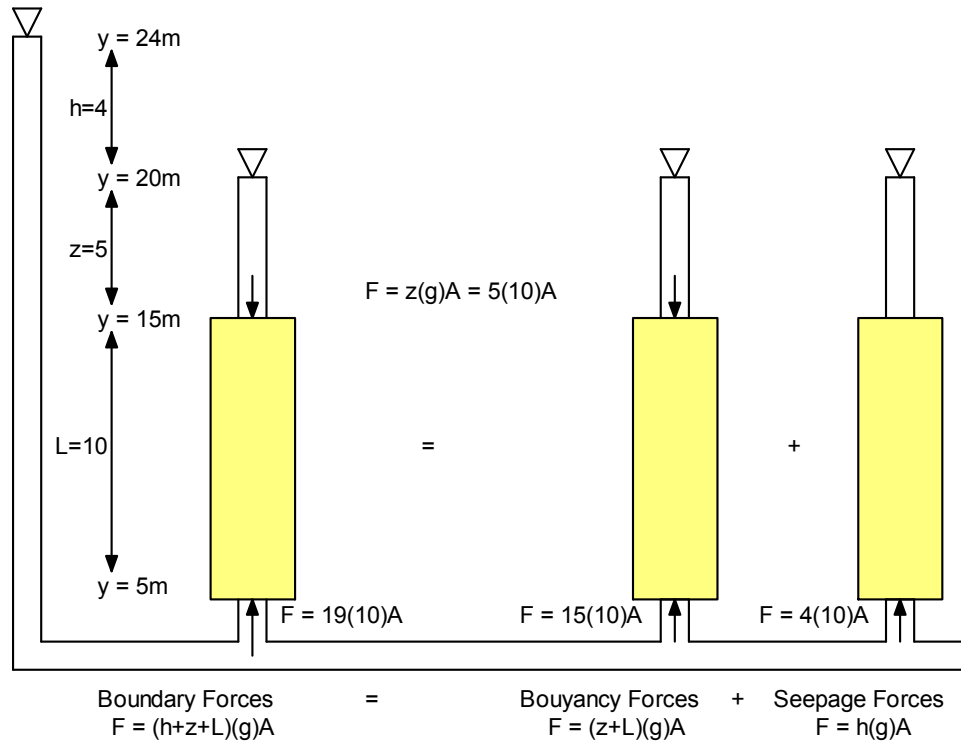


Figure 9-9 Forces on a soil element

In the left case, we look at the boundary forces at the top and bottom of the element. The force acting on the top is the depth of water above multiplied by the water weight (assumed 10 KN/m^3) and by the cross-sectional area of the element, A . Likewise, the boundary force acting at the base is the depth of bottom boundary below the surface multiplied by the water weight and area.

In the right two figures the forces are broken down into two parts, a buoyancy force and a seepage force. The buoyancy forces act on the bottom and top based on the static pressure as if there were no flow (e.g., the water acting down from above). The seepage force is the force that is all spent due to moving the water through the resisting soil.

In both cases, the same force equilibrium will apply to the soil element. However, it is often more clear to consider the boundary force approach because there is only one equation to consider and it applies to any point on the soil element. The boundary force approach is used in all GeoStudio models.

9.3 Exit gradients

Gradients

The hydraulic gradient in the soil is computed as the total head loss divided by distance of flow between the two measured head locations, or

$$i = \frac{dH}{dL}$$

In a finite element formulation, the gradient matrix is termed the $[B]$ matrix and it is computed for all points within a single element based on the coordinates of the element nodes and a shape function which determines how the total head is distributed within the element. There is a discussion in the chapter on

meshing about element shapes and how the primary variable is distributed throughout the element. Details of the [B] matrix itself are provided in the appendices.

It is important to know that in a finite element formulation the gradients within elements on either side of a shared element edge are not necessarily going to be the same. This is because the shape functions we use to know how parameters are distributed within an element are not continuous to adjacent elements across a common edge. The finite element method ensures we have flow continuity at the nodes (e.g., mass balance) but it does not ensure we have energy (or gradient) continuity at element edges. If you make the mesh finer, you may improve the apparent gradient continuity across an element edge but you will likely not improve the nodal mass balance that significantly assuming that you had a converged solution before and after you fine-tuned the element size.

Critical gradients

A quick condition exists when the upward force on a soil particle equals the total particle weight. Said another way, it is when the seepage force equals the submerged weight of the soil. Since in GeoStudio we choose not use the seepage force approach (as discussed in the previous section) we will stick to total unit weights and pore-water pressures in our discussion... in other words, we will deal with effective stresses.

The effective stress is computed as the total stress minus the pore-water pressure; where the total stress is computed in the y-direction as the total unit weight of the soil multiplied by the depth from the surface. If the pore-water pressure is high enough such that the effective stress is reduced to zero or less, then it is possible a quick condition exists.

The concept of critical gradient = 1 was developed using hand calculated flow nets for upward vertical flow in one direction. It has traditionally been computed by considering the state when the buoyant weight divided by the unit weight of water is equal to 1.0 but this is only true for forces in the vertical direction. The value of critical gradient = 1 does not necessarily apply in two dimensional flow as we will show below.

Geometry considerations

Consider the images in Figure 9-10 below. There are four meshes in one analysis. The left two are vertical flow only because the ground surface is exactly flat. The right two have a very slight curvature to the ground surface. In both cases, the head difference is equal to the elevation difference so the “hand calculated” gradient should be 1. In the vertical flow cases only, regardless of course or fine mesh, the vertical gradient is computed to be exactly 1.0. In the curved surface cases, there is a gradient concentration at the point of maximum curvature because there are gradients in both the X and Y directions that create an overall higher hydraulic energy state in the element. The maximum Y gradient is now about 1.15. It is even higher in the mesh with secondary nodes because the gradients are better defined over the element when there are secondary nodes.

It is now obvious that if this were a real life soil, we would expect no difference in the stability of all four cases because our engineering judgment lets us know they are the same. Numerically, and physically, there is a higher gradient for the curved surface soils but we are trying to interpret that higher gradient (1.15) in the context of what we think should be happening for one dimensional, vertical flow only. It is not the software that is wrong; it is our frame of reference that is wrong.

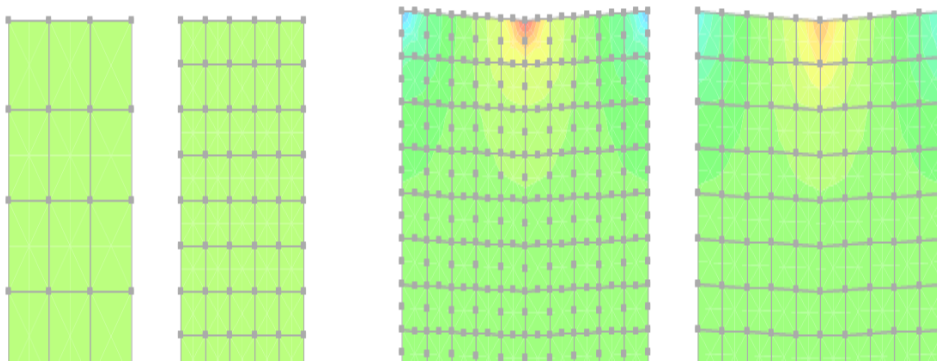


Figure 9-10 Four column meshes - left to flat on top, right two slightly curved

It is possible to look at the gradients across all profiles at one time as shown in Figure 9-11 below. You can see that for the perfectly flat cases the gradient in the y-direction is 1.0 at all elevations. However, for the sloping ground cases, the gradients vary across the profiles at all depths. In this case, the ground surface is even flatter than shown above. The left and right sides of each curved surface column shown here are 1cm higher than the center point over a 10m high column. In comparing the more curved case above with this case, it is very clear that the sharper the break in slope, the more effect the 2D flow has on increasing gradients at the break location. It is also clear that the phenomenon is not just localized to the surface where there is a break in slope angle. The phenomenon extends throughout the entire column where there is 2 dimensional flow.

The real issue we have to deal with when considering seepage is: what is a critical gradient for two dimensional exit flow? It is NOT 1.0. One can appreciate that it is hard to say to a client “we don’t want to design to your 1.0 criterion” but you may have to. If you want to do more detailed analysis, then you need to perhaps consider a SIGMA/W coupled stress/pore-water pressure analysis to consider effective stress. This is done below.

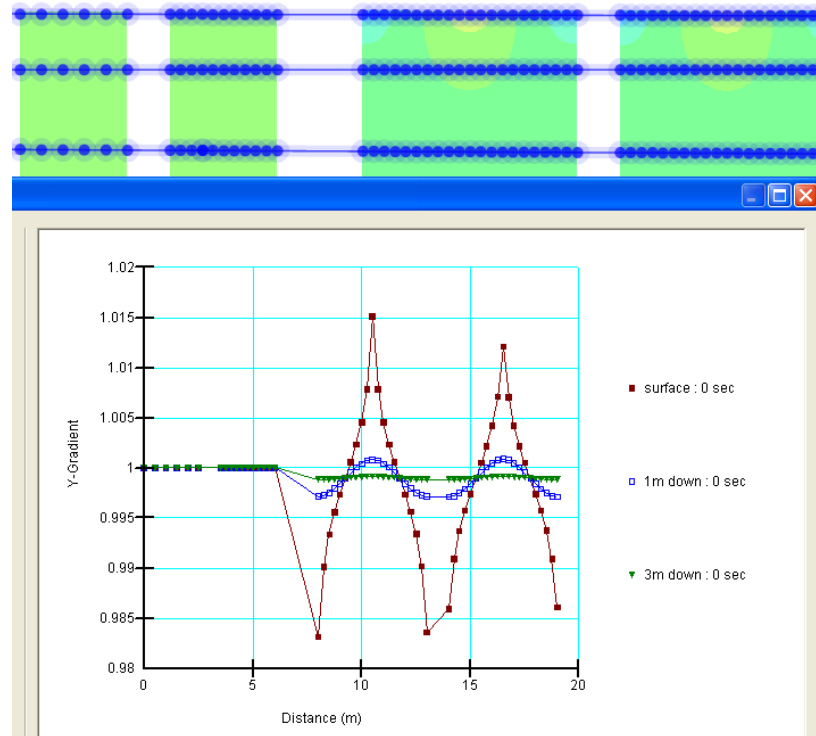


Figure 9-11 Y gradients at different depths in a column

Now let us consider the case of exit flow on a sloping ground with a break in the slope. Figure 9-12 shows two slope geometries for essentially the same model. The left image has a sharp break in the slope while the right image has several regions points added such that a smooth or rounded break in the slope is accounted for. Both images show contours of x-y gradients and you can see that there is a gradient concentration at the location of the break in the slope where water is seeping out.

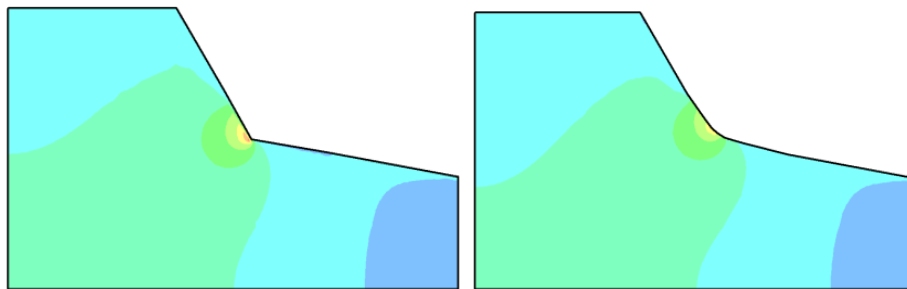


Figure 9-12 Comparison of gradients with sharp and rounded corners

Figure 9-13 shows a plot of gradient versus distance into the slope on a line drawn normal to the break in the slope. You can see that for both the rounded and angular case there is an x-y gradient greater than zero right at the surface but that at a short distance into the slope, say 1m, the gradients are less than 1.0. If this were a flow net solution, it is not likely that the flow net would be drawn to incorporate a grid size of less than 1m (given that the model is 30m in width) which means the hand computed gradient at the exit location would be the average gradient over 1 or more meters distance.

We recommend that when you consider exit gradients and you want to compare them with the traditional flow net critical gradient concept of 1.0, you take an average gradient over a 1 to 2 meter distance in the vicinity of the exit location.

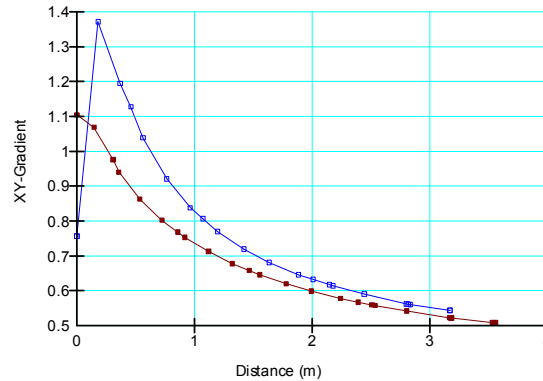


Figure 9-13 Gradients on a line normal to the slope from the break location

Effective stress and soil strength

A key issue to consider when thinking about critical gradients is the strength of the soil. If a soil has zero effective stress and has no cohesive strength, then it is in a serious state from a stability and quick or piping perspective. However, if there is any cohesive strength under zero effective stress, then the concept of critical gradient being equal to 1.0 is not valid. The gradient may be 1 or more, but the soil has shear strength to resist a piping effect.

Figure 9-14 shows the effective stresses in a coupled SIGMA/W stress/pore-water pressure analysis for the column with a curved surface case. You can see that there is a small section of soil just below the ground surface at the point of maximum curvature where there are very small negative y-effective stresses. A negative effective stress signifies a “quick” condition. This first analysis assumed a soil unit weight of 20 KN/m³.

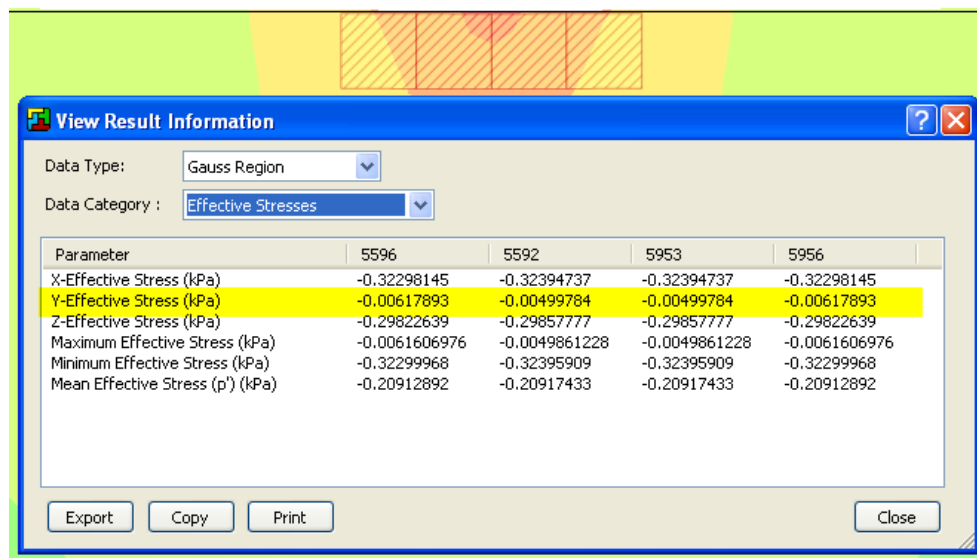


Figure 9-14 Gauss point stresses using unit weight of 20 KN/m³

The second image below shows the result with an assumed unit weight of 21 kN/m³. You can see in the second analysis that there are now positive effective stresses at all points just below the surface. A seepage analysis alone considers none of this. In both cases below, the hydraulic gradients are greater than 1.0 but depending on soil weight and the effective stress and associated strength, there may or may not be “quick” conditions. Even in a soil with frictional strength only, the presence of a positive effective stress near the ground surface may resist piping conditions.

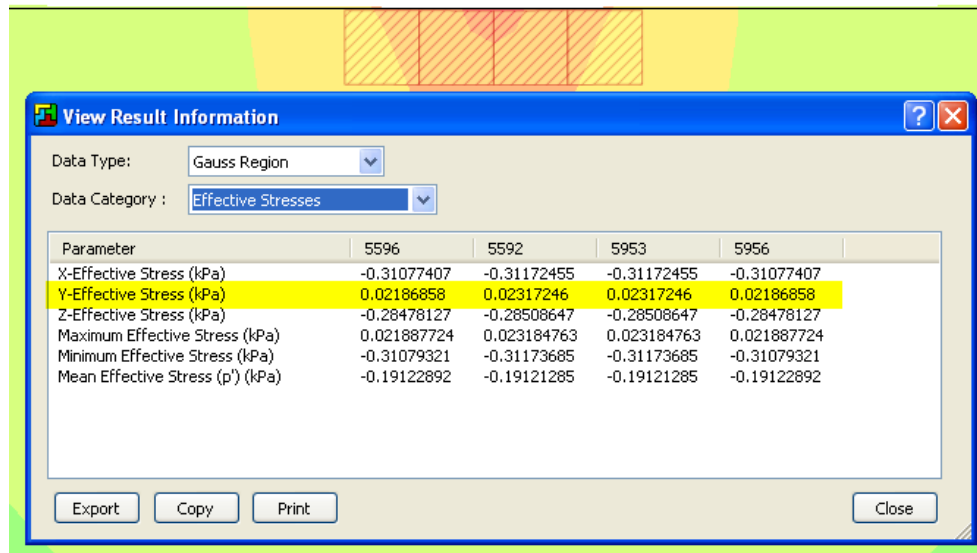


Figure 9-15 Gauss point stresses using unit weight of 21 kN/m³

A seepage analysis alone, when used for determining gradients, must be done with some understanding that the critical gradient approach of 1.0 was based on very simplified assumptions developed using hand drawn flow nets under vertical flow scenarios.

Flow velocity

One final issue to consider is the fact a high gradient may exist at the ground surface but if there is not sufficient exit flow velocity, any loose particles of soil cannot be lifted and transported. Consider a vacuum cleaner... it has a very high suction gradient but if you block off the flow of air, no particles are moved. The gradient remains but there is insufficient flow to transport dirt. A detailed discussion of overland flow and transport is beyond the scope of this book and there are experts in industry who can be consulted. One approach is to determine the exit flow volumes from a seepage analysis and use them in a transport analysis.

9.4 Concluding remarks

With a software tool like SEEP/W, it is no longer necessary to construct traditional flow nets. SEEP/W can provide all the information obtainable from a flow net plus much more, and SEEP/W makes it possible to analyze a much wider range of problems. Furthermore, doing a SEEP/W analysis is much simpler and faster than trying to create a flow net.

There is still value in being able to approximate flow nets, since so much of the textbook literature still presents flow nets and discusses techniques for constructing flow nets. Consequently, flow nets are sort of a historical reference point. Having the ability to view SEEP/W results in the context of past methods is useful, but attempting to use flow nets in practice is no longer practical nor necessary.

Seepage forces are used in the literature and are taught in some classes on ground-water but they are not necessary to consider if the analysis approach uses total soil weight and boundary forces as opposed to buoyant and seepage forces. This is the approach used throughout GeoStudio.

Finally, exit gradients and critical gradients must be considered in terms of geometry, effective stress and associated soil shear strength, and flow velocity. A computed gradient of 1.0 does not necessarily mean there will be stability issues or quick or piping conditions. We recommend you consider an average gradient over a reasonable depth of soil near the exit location, not just the exit gradient right at the ground surface where it is affected by model geometry, two dimensional flow, low effective stresses etc.

10 Visualization of Results

When you get to the visualization of results stage of a finite element analysis you can congratulate yourself for having completed the hardest parts – setting up the geometry, defining meaningful soil property functions, and applying appropriate boundary conditions to the model.

This chapter describes the various types of output data that are computed by the solver and it attempts to get you thinking about what the data is trying to tell you. For example, did the solution solve properly? Did the boundary conditions you applied get reflected in the actual solution? Did the soil respond how you thought it would respond? If not, how to you methodically determine what to check next?

The chapter is structured to explain what type of data is available for visualization. In the various sections, comments are provided that relate the type of result data in question to how it should be used in the overall thought process. It's a good idea to read this entire chapter.

10.1 *Transient versus steady state results*

The type of data you can view is somewhat dependent on the type of analysis you have completed. For example, if you have done a steady state analysis, you cannot interpret any data related to changes in time. You can view instantaneous flux values that have units of volume per time; however, you cannot see how these values may change with time because steady state, by nature, means things do not change with time.

In a steady state analysis you are not required to define a volumetric water content function, because it is the slope of this function that is needed in the solution of the transient, not steady state, finite element equation. If you do not define a water content function, then it stands to reason that you cannot view water contents. Water contents can only be viewed in a steady state solution if you have defined a water content function that the solver can access to report water contents based on solved pressures. No water content function... no reported water contents!

In a transient analysis, you can look at how all of the various output data values change with respect to time and/or position, whereas in steady state analysis, you can only graph how the data changes with position.

Finally, in a transient analysis you cannot plot the flow path of a particle of water across the entire geometry. This is because there are no single flow lines in a transient process. Sure, water is flowing somewhere, but a single particle of water at one point in a mesh has an infinite number of possible places to flow with time, and so it is not known what the path of the particle will be. You can plot the flow vector within any element at any point in time because by the simple nature of the “finite element” we are assuming the flow within any single element is known at all points in time. In a steady state analysis, the flow line is useful to follow the path of a particle of water from its entrance into the geometry to its exit. The entire path is known, because once the flow is established, it is at a steady value and therefore is fixed at that position. There is a more detailed discussion on flow lines and flow vectors later in this chapter.

10.2 *Node and element information*

In order to understand what type of information can be viewed as results output, it helps a bit to know how the data is obtained. So, to recap, you set up the problem geometry, define material properties, and apply boundary conditions of either known head (or pressure) or flux. The solver assembles the soil property and geometry information for every Gauss point in every element and applies it to the flow equation that is written for every node. Therefore, at each node we have applied boundary data,

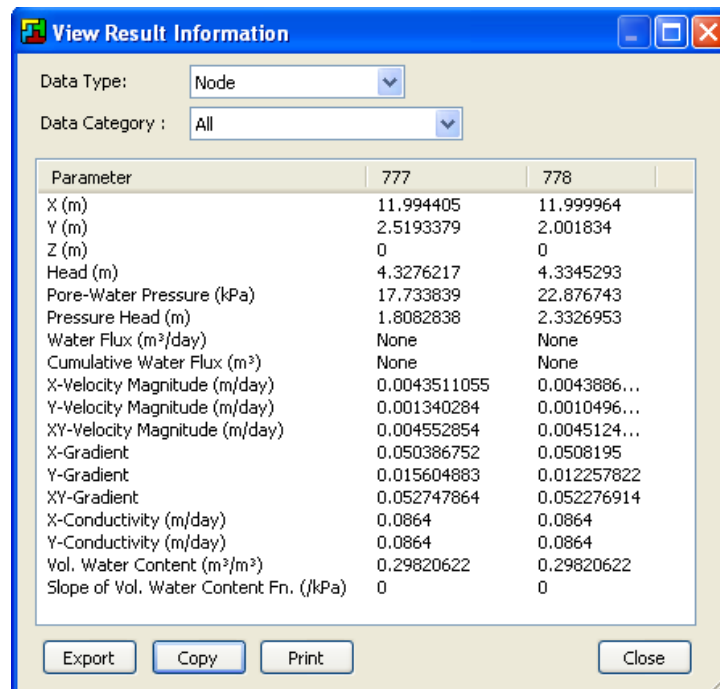
interpolated soil property data and geometry data. The solver then computes the unknown value in the equation for each node – the unknown value being either head or flux. It is the Gauss point data that is used to set up the nodal equations, so the Gauss point data written to the output file is the actual data used in the solver.

In GeoStudio, all output data for nodes and gauss points anywhere in the model is accessible using the View Results Information command. With the command selected, you can click the mouse on any single node to view the output at the node. You can also hold down the shift key to multi-select many points. If you click beside a node and within the element itself, you will get the Gauss point data at that location. You can multi-select Gauss point to see a table of data.

Figure 10-1 is an illustration of the type of information that can be viewed for each node in the finite element mesh. You can view the types of data in the list to see that there is a combination of heads, pressures, fluxes, velocities, gradients, conductivities and water contents. There is also a summary of the position of the node within the problem domain. In effect, the node information is a summary of the problem geometry, the soil material properties, and the boundary conditions – the three main parts of any finite element analysis. If your model included an air flow analysis, the list would contain all the related air flow parameters too.

One key point to note in the figure below is that the nodal Boundary Flux quantity is None. This is an important point to understand because it can help with your overall interpretation of results. This boundary flux is computed by summing the contributing fluxes from each of the four Gauss points that surround this node. So, if water is flowing out of one Gauss region, it HAS TO be flowing into an adjacent Gauss region. For all internal nodes with no user boundary applied to them, the sum of all the fluxes at a node should equal zero and is reported as None – meaning there is no data at this point in time or space.

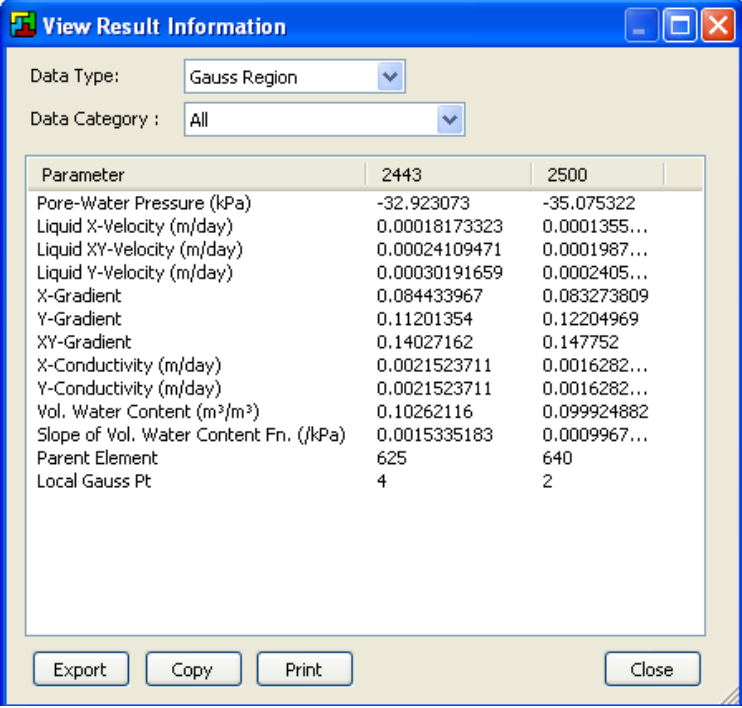
If the node being viewed is a boundary condition node (not necessarily at the edge of the geometry, but with an allowed influx or outflux) then the summation of all the fluxes at that node will not be zero, because water is either gained or lost at that point.



Parameter	777	778
X (m)	11.994405	11.999964
Y (m)	2.5193379	2.001834
Z (m)	0	0
Head (m)	4.3276217	4.3345293
Pore-Water Pressure (kPa)	17.733839	22.876743
Pressure Head (m)	1.8082838	2.3326953
Water Flux (m ³ /day)	None	None
Cumulative Water Flux (m ³)	None	None
X-Velocity Magnitude (m/day)	0.0043511055	0.0043886...
Y-Velocity Magnitude (m/day)	0.001340284	0.0010496...
XY-Velocity Magnitude (m/day)	0.004552854	0.0045124...
X-Gradient	0.050386752	0.0508195
Y-Gradient	0.015604883	0.012257822
XY-Gradient	0.052747864	0.052276914
X-Conductivity (m/day)	0.0864	0.0864
Y-Conductivity (m/day)	0.0864	0.0864
Vol. Water Content (m ³ /m ³)	0.29820622	0.29820622
Slope of Vol. Water Content Fn. (/kPa)	0	0

Figure 10-1 Visualization of node information

Figure 10-2 is the corresponding Gauss point information for two adjacent Gauss points. Because the element is rectangular, each Gauss area is equal to one quarter of the total area of the element. The inset in the figure below shows the type of data that can be viewed at each Gauss point. If you consider the water content value of 0.10262%, for example, you should realize that this water content is assumed to exist throughout the Gauss point area displayed; and you should next realize that if the element size is increased, the estimate of the water content becomes less accurate, as we are averaging it over a larger area. The real trick to getting good finite element analysis results is to create a finite element mesh with just the right sized elements that are not too big or too small, that can represent the highly non-linear soil properties within them, and that can handle the potentially extreme boundary conditions you apply. It is not always easy and there is no sure quick or automatic method to make that happen.



Parameter	2443	2500
Pore-Water Pressure (kPa)	-32.923073	-35.075322
Liquid X-Velocity (m/day)	0.00018173323	0.0001355...
Liquid XY-Velocity (m/day)	0.00024109471	0.0001987...
Liquid Y-Velocity (m/day)	0.00030191659	0.0002405...
X-Gradient	0.084433967	0.083273809
Y-Gradient	0.11201354	0.12204969
XY-Gradient	0.14027162	0.147752
X-Conductivity (m/day)	0.0021523711	0.0016282...
Y-Conductivity (m/day)	0.0021523711	0.0016282...
Vol. Water Content (m ³ /m ³)	0.10262116	0.099924882
Slope of Vol. Water Content Fn. (/kPa)	0.0015335183	0.0009967...
Parent Element	625	640
Local Gauss Pt	4	2

Figure 10-2 Visualization of element Gauss point information

10.3 Graphing node and gauss Data

The Draw Graph command allows you to plot a graph of any computed value as a function of time, position or both time and position. In past versions of GeoStudio, all graphing was based on user selected nodes. Moving forward, GeoStudio now requires the user to select graph data locations based on one or more points, a cut line, or a region of points. It is possible to select all three types of data locations within a single graph. Figure 10-3 shows a combination of all three graph data objects in a single dam cross section.

The advantage of using this type of data selection is that the location and type of data used in any graph can be named and saved. Each time you return to the graphing command, you can choose from your saved list of graphs and you do not have to re-define them. Even if you change the mesh, the model will know the new nodes nearest to your graph selections and it will draw the graph using the most recent solution for those nodes.

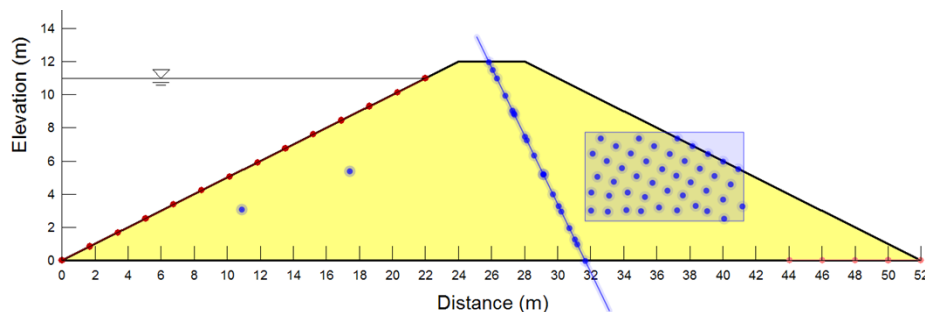


Figure 10-3 Graph data selection options (points, lines, planes)

In the previous image, the graph points were selected at any point in the domain. Sometimes it is easier to select all points along a given geometry object such as a region line or point. Consider Figure 10-4 where the entire up stream region edge line has been selected for graphing. In this case, it was easier to just select one point along the entire edge and have the model capture all nodes along that edge. The option of selecting custom points or geometry points is totally a user preference.

Once the graph is visible there are many options to change the font, apply a legend, rotate the image, copy the image to paste it into a report, copy the data to paste to Excel or another program, or export it as a comma separated text file.

You can even hover the mouse directly on a graphed point to see the actual data as shown in Figure 10-5 below.

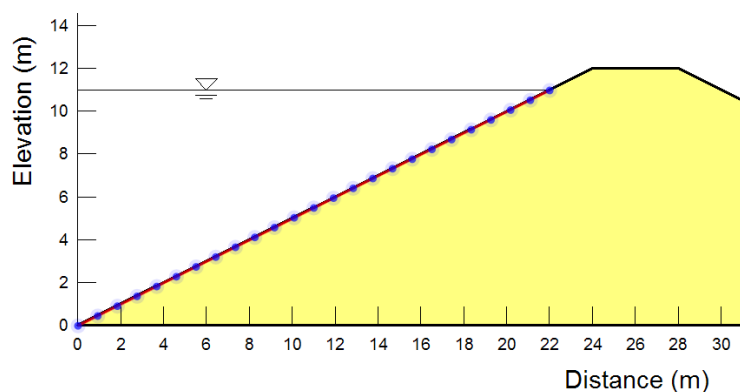


Figure 10-4 Graph selections based on geometry item (upstream region edge line)

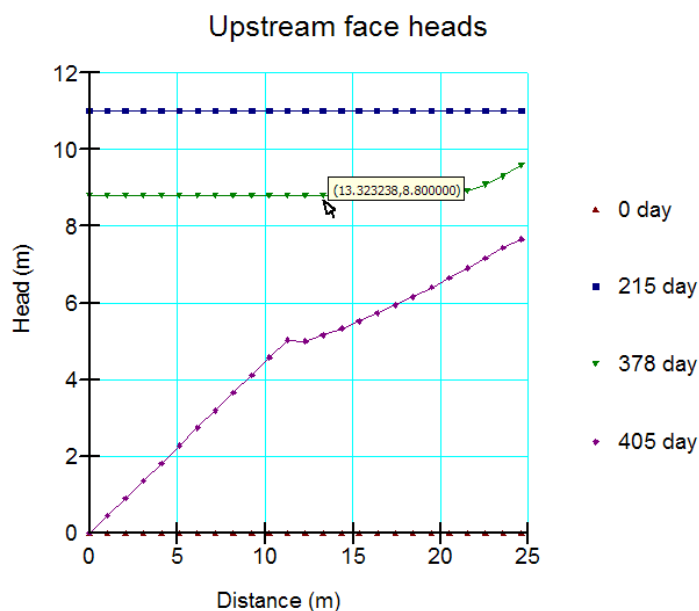


Figure 10-5 Upstream total head as a function of position for each time

10.4 “None” values

In GeoStudio, an attempt is made to distinguish between data values that have a true value of zero, and those that are missing. A missing value is labeled as “none” in a data list or is not printed to file when you save the data for export or pasting into another program such as Excel. A missing value is simply a data type that is not relevant to the current set of analysis parameters. For example, in Figure 10-1 above, the node boundary flux values are set to “none”. This is because there are no nodal flows at internal, non boundary condition nodes.

“None” or missing values, are simply a way for GeoStudio to not erroneously report data values as zero (which has meaning) when they really just do not exist. Consider the following graph generated by GeoStudio of pore-water pressures in a soil as it is placed during a construction sequence. At the 0 second time, the soil surface is at 10m. At 10 seconds, 2 meters of more soil is added. At 8010 seconds, another 2 meters is added. Notice that for the two added lifts of soil, the pressure values are not graphed as zero prior to their placement time. The data is “missing” in the program so is not reported or graphed.

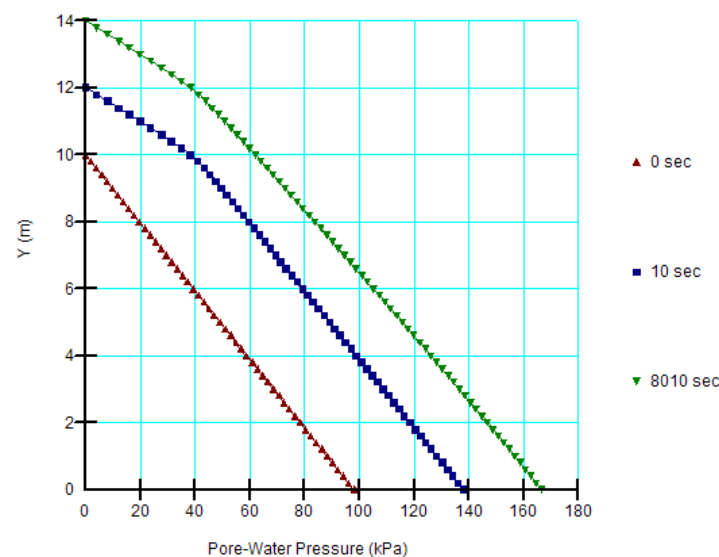


Figure 10-6 Graph showing how missing data is excluded and not printed as zero

10.5 Water table

In SEEP/W, the location of the water table is drawn in Contour along an isoline where the water pressure is zero. This is technically correct for the case where the air pressure is assumed to be zero but it is not correct when air pressures are considered. The location of the water table is actually where the matric pressure ($U_a - U_w$) is zero.

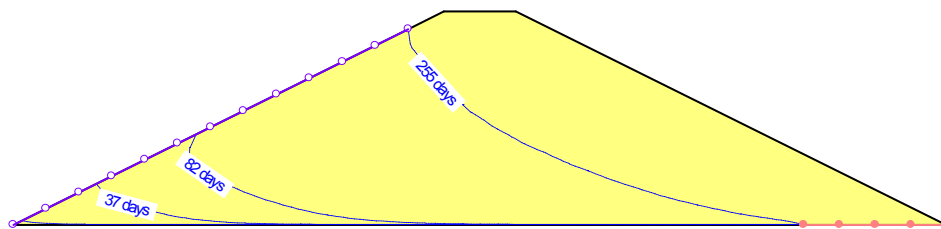


Figure 10-7 Water table (isolines) over time

10.6 Isolines

You can use the Draw Isolines command to choose which parameter you want the water table calculation based on. It can be water pressure or matric suction. You can also choose to draw an isoline contour of any other parameter at an instance in time or over multiple times. If you draw an isoline at multiple time steps then you can not also view contour shading as it only exists for any instance in time. The isolines are a way to track a single value of a parameter as it changes over time... such as a water table.

10.7 Projecting Gauss point values to nodes

SEEP/W performs contouring calculations based on parameter values at the nodes. Since the primary parameters, (total head, pressure, and pressure head), are computed at the nodes, these parameters can be contoured directly. However, secondary parameters, (velocity, gradient, conductivity, and volumetric water content), are computed at the element Gauss points and must therefore be projected to the nodes for contouring purposes.

In triangular elements, the Gauss point values are projected on the basis of a plane that passes through the three Gauss points. For one-point integration, the value at the Gauss point is also taken to be the value at the nodes (i.e., the Gauss point value is constant within the element).

In quadrilateral elements, the Gauss point values are projected using the interpolating functions. (For more information about interpolating functions, see the appendix). In equation form,

$$x = \langle N \rangle \{X\}$$

where:

- x = the projected value outside the Gauss points at a local coordinate greater than 1.0,
- $\langle N \rangle$ = a matrix of interpolating functions, and
- $\{X\}$ = the value of Gauss point variable.

The local coordinates at the element nodes are the reciprocal of the Gauss point local coordinates when forming the element characteristic matrix. Figure 10-8 is an example of the local coordinates at the element corner nodes when projecting outwards from the four Gauss points in the element. The value of 1.7320 is the reciprocal of the Gauss point coordinate 0.57735.

This projection technique can result in some over-shoot at the corner nodes when variation in the parameter values at the Gauss points is large. For example, consider that we wish to contour volumetric water content and that in some elements the water content at the Gauss points varies over the complete

range of the volumetric water content function. Projecting such a large variation to the nodes can result in projected nodal water contents beyond the range of the volumetric water content function.

Extreme changes in the parameter values at the Gauss points within an element often indicate numerical difficulties (the over-shoot at the nodes being just a symptom of the problem). This over-shoot can potentially be reduced by a finer mesh discretization. Smaller elements within the same region will result in a smaller variation of parameter values within each element, therefore lowering the potential for encountering unrealistic projections.

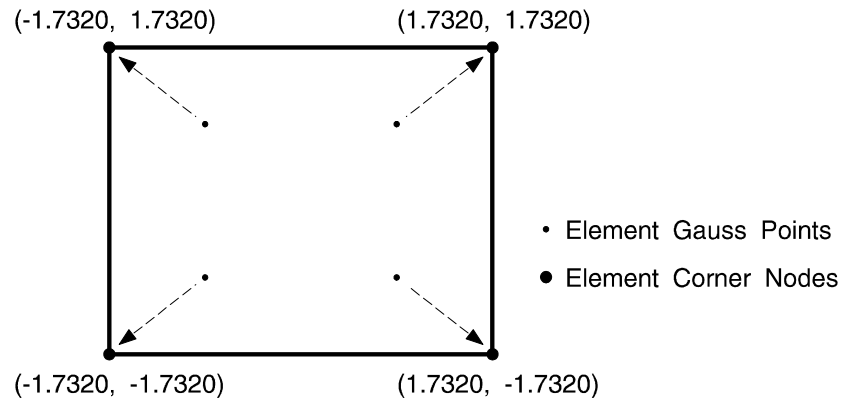


Figure 10-8 Local coordinates at the corner nodes of an element with four integration points

10.8 Contours

The power of using advanced graphical interfaces with finite element analysis is that the computer can quickly convert thousands of pieces of data into meaningful pictures. In a section above we introduced isolines and showed how their relative positions give an indication of the change in a parameter over time and space. In this section, we show that it is quite simple, and much more meaningful, to interpret parameters over space if we can view all values for a model at one time over the entire domain.

In GeoStudio, you have the ability to set up a contour profile, give it a name, and save it in a list. You can then easily change contour views using a drop down list on the toolbar of any saved contours without having to re-set the viewing parameters.

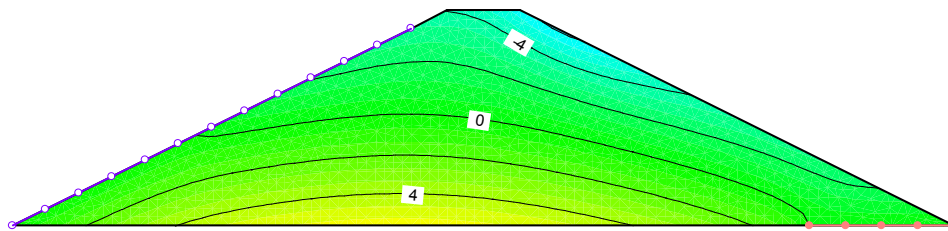


Figure 10-9 Contour of pressures at a fixed time

10.9 Animation in GeoStudio

Movie files (*.avi) can be created in GeoStudio to illustrate a physical process in a transient analysis. The first step in creating a movie is to define the contours and specify any View Preferences that need to be visible (e.g. flux vectors or the displaced mesh). The View Animation command is selected and the time steps and viewing area are defined. After saving the movie file to the appropriate location, GeoStudio joins together all of the individual images for each time step, creating a seamless animated movie.

10.10 Velocity vectors and flow paths

Calculating gradients and velocities

Once the solution has converged and the nodal heads are known, SEEP/W computes the hydraulic gradients and Darcian flow velocities at each of the integration points within each element. The gradient at each Gauss or integration point is computed from the equation:

$$\begin{Bmatrix} i_x \\ i_y \end{Bmatrix} = [B]\{H\}$$

where:

- i_x = the gradient in x direction,
- i_y = the gradient in y direction,
- $[B]$ = the gradient matrix as defined in the theory chapter, and
- $\{H\}$ = the vector of total head at the nodes.

The Darcian velocities at each Gauss point are computed from the equation:

$$\begin{Bmatrix} v_x \\ v_y \end{Bmatrix} = [C][B]\{H\}$$

where:

- v_x = the velocity in x direction,
- v_y = the velocity in y direction, and
- $[C]$ = the hydraulic conductivity matrix.

SEEP/W stores the hydraulic conductivity at each Gauss point used in the formulation of the finite element equations in an array. The same hydraulic conductivity values are later used to compute the velocities.

The SEEP/W velocity is actually the specific discharge, q , which is the total flux Q divided by the full cross-sectional area (voids and solids alike); it is not the actual speed with which the water moves between the soil particles. The actual microscopic velocity is:

$$v = \frac{q}{n}$$

where:

v = the average linear velocity,
 q = the specific discharge, and
 n = the soil porosity.

SEEP/W does not compute the actual pore-channel velocity. Once the velocities and gradients are calculated, they are saved to a data file for later use in visualizing the node or Gauss point information, or for generating flow vectors and flow lines.

Velocity vectors

Velocity vectors are a useful way of seeing not only where the flow is occurring, but how much flow there is relative to other regions of the domain. SEEP/W uses the magnitude of the actual velocity in its calculation of how large to display the vector so that you have a visual representation of where the velocities are high or low. For each element, the average x-velocity and average y-velocity from the Gauss point velocity values are computed and then vectorally summed to obtain an average velocity vector for the element. This average velocity vector is plotted with the tail of the vector at the center of the element.

When displaying vectors, SEEP/W finds the maximum velocity vector and draws it at the length specified in the Draw Vectors dialog box. All other vectors are drawn in proportion to the element velocity relative to the maximum velocity. For example, if the element velocity is one quarter of the maximum velocity, then the length of the velocity vector is one-quarter of the length specified in the Draw Vectors dialog box.

You have the option of controlling the size of the vectors as they are displayed by controlling the magnification scale when you issue the command to create the vectors. Specifying a magnification value allows you to control the scale at which all vectors are drawn. When you type a value in the magnification edit box, the maximum length edit box is updated to display the length at which the maximum vector will be drawn. You can control the vector length either by specifying a magnification value or by specifying a maximum display length value. If you specify a length, the magnification value is computed by dividing the maximum length by the maximum velocity and adjusting the value for the scale of the page and engineering units.

Flow paths

The SEEP/W flow paths are not flow lines or stream lines as in a traditional flow net. In many cases the flow paths are a very good approximation of stream lines, but they are not the same. The flow paths are simply a line based on velocity vectors in an element that a drop of water would follow under steady-state conditions; and slight variations between a flow path and a flow line should not be of concern because they are computed in entirely different ways. At best, the SEEP/W flow path should be viewed as a reasonable approximation of the flow lines within a flow net.

The flow paths will always be the most realistic in saturated zones where there is significant velocity. In zones where there is little or no flow, the SEEP/W flow paths may not be realistic. Several cases are illustrated in Figure 10-10. In the upstream toe area where there is little flow, the flow path ends along the bottom boundary. This is not physically correct and therefore is not a realistic flow path. Within the unsaturated zone in the figure there is a flow path that ends within the dam section. The reason for this is that the path has reached an area where there is essentially no flow. Once again, such a flow path has no meaning. Since it is possible to click in an area with relatively no seepage and create an unrealistic flow

path, some judgment is required by you when drawing flow paths. You should discard flow paths that you judge to be unrealistic.

When you create a flow path, a message will be displayed if you attempt to draw it in an area where there is little or no flow. After accepting the message, the flow path will be drawn, but it may not be complete; that is, the path will end inside the flow regime and not at an external mesh boundary. This message will also be displayed if the flow path encounters a no flow perimeter boundary.

Another important point is that in a saturated / unsaturated flow system, the phreatic line is not a flow line. It is simply a line of zero water pressure. Since water can flow from the saturated to the unsaturated zone, and vice versa, flow can take place across the phreatic surface. Consequently, a flow path may cross the phreatic surface as illustrated Figure 10-10. This is acceptable and realistic.

You can select any point to draw a flow path by clicking on a point within the flow domain. The path is drawn strictly on the basis of the velocity vectors within each element. The path is projected forward and backward incrementally within each element until the path encounters a boundary. The flow path is simply a graphical representation of the route a molecule of water would have traveled under steady-state conditions from the entrance to exit point within the flow regime.

Flow paths can only be drawn for steady-state conditions. Flow paths based on velocity vectors at an instant in time during a transient process have no physical meaning. For this reason, SEEP/W does not permit you to draw flow paths for transient conditions

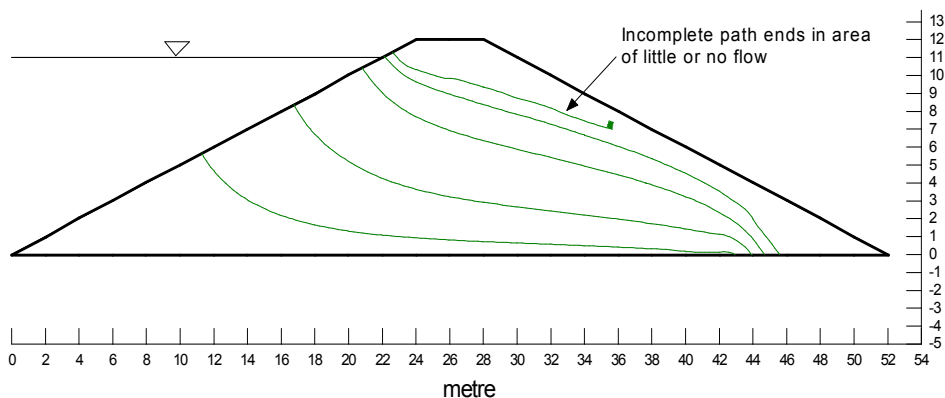


Figure 10-10 Illustration of flow lines in a steady state analysis before reservoir drained

10.11 Flux sections

SEEP/W has the ability to compute the instantaneous seepage volume rate that flows across a user-defined section for either a steady state or transient analysis. This is a very useful tool for isolating flow volumes to specific regions of interest and it can save you manually adding up individual nodal flows in the case of a drain or seepage face that is comprised of many nodes.

Flux section theory

This value can be computed from the nodal heads and the coefficients of the finite element equation. For example, consider a mesh with only one element, as illustrated in Figure 10-11. The objective is to compute the total flow across a vertical section of the element.

In the Theory chapter, the finite element form of the mass flow equation is presented. It can be re-written with the flux value isolated on one side as follows:

$$[K]\{H\} + [M]\frac{\Delta H}{\Delta t} = \{Q\}$$

In a steady-state analysis, the storage term $[M]\frac{\Delta H}{\Delta t}$ becomes zero, and the equation can be reduced to:

$$[K]\{H\} = \{Q\}$$

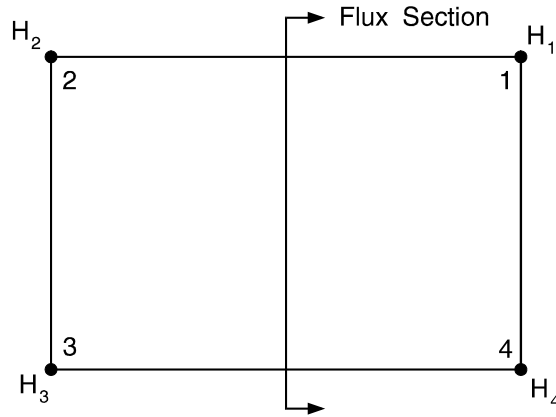


Figure 10-11 Illustration of a flux section across a single element

The global set of finite equations for one element is as follows:

$$\text{Equation 10-1} \quad \begin{bmatrix} c_{11} & c_{12} & c_{13} & c_{14} \\ c_{21} & c_{22} & c_{23} & c_{24} \\ c_{31} & c_{32} & c_{33} & c_{34} \\ c_{41} & c_{42} & c_{43} & c_{44} \end{bmatrix} \begin{Bmatrix} H_1 \\ H_2 \\ H_3 \\ H_4 \end{Bmatrix} = \begin{Bmatrix} Q_1 \\ Q_2 \\ Q_3 \\ Q_4 \end{Bmatrix}$$

From Darcy's Law, the total flow between two points is:

$$\text{Equation 10-2} \quad Q = k A \frac{\Delta H}{l}$$

The coefficients, c , in Equation 10-1 are a representation of $\frac{KA}{l}$ in Equation 10-2. Therefore, the flow from Node i to Node j is:

$$Q_{ij} = c_{ij} (H_i - H_j)$$

In a transient analysis, because of material storage, the calculation of the total flow quantity must include the storage effect. The change in flow quantity due to the storage term can be expressed as:

$$\frac{1}{\Delta t} \begin{bmatrix} m_{11} & m_{12} & m_{13} & m_{14} \\ m_{21} & m_{22} & m_{23} & m_{24} \\ m_{31} & m_{32} & m_{33} & m_{34} \\ m_{41} & m_{42} & m_{43} & m_{44} \end{bmatrix} \begin{Bmatrix} \Delta H_1 \\ \Delta H_2 \\ \Delta H_3 \\ \Delta H_4 \end{Bmatrix} = \begin{Bmatrix} Q_1 \\ Q_2 \\ Q_3 \\ Q_4 \end{Bmatrix}$$

where $\Delta H_{1,2,3,4}$ etc. are the changes of total head at the various nodes between the start and the end of a time step. In general, the average change of total head from Node i to Node j can be expressed as:

$$\Delta H_{ij} = \frac{\Delta H_i + \Delta H_j}{2}$$

Therefore, the change in flow quantity from Node i to Node j due to a change in storage is:

$$Q_{ij} = m_{ij} \frac{\Delta H_{ij}}{\Delta t}$$

The total flow quantity from Node i to Node j for a transient analysis then becomes:

$$Q_{ij} = c_{ij} (H_i - H_j) + m_{ij} \frac{\Delta H_{ij}}{\Delta t}$$

The total flow quantity through the flux section shown in Figure 10-11 is:

$$Q = Q_{21+} + Q_{24} + Q_{31} + Q_{34}$$

The imaginary flow lines from one side of the section to the other side are known as subsections. SEEP/W identifies all subsections across a user-defined flux section, computes the flow for each subsection, and then sums the subsection flows to obtain the total flow across the flux section.

Flux section application

Flux sections can be used in many ways, because they can be drawn any place across which you want to know the flux. You may want to check that an influx is equal to an out flux such as illustrated in Figure 10-12; or you may want to know the total drain flux as illustrated in Figure 10-13.

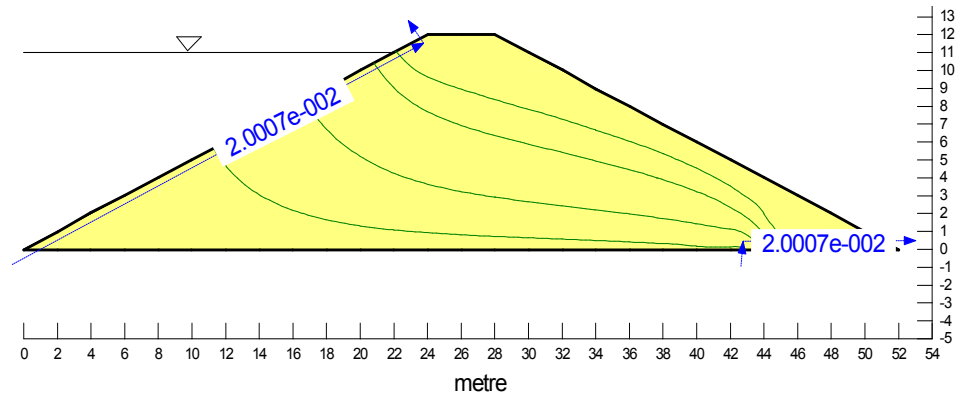


Figure 10-12 Flux section used to check balance of inflow and outflow

Flux sections do not have to be drawn as single straight lines. They can be made of continuous attached segments as illustrated in both figures above. When a multiple segmented flux section is drawn, the value of flux reported for the section applies to the entire section, not any individual segment.

The key point to note when defining a flux section is to make the flux section go completely through an element if you want the value associated with that element to be included in the flux summation. Also, if you want to check the flux around a closed loop as illustrated for the drain nodes in Figure 10-13, make sure the end of the flux section crosses over the tail of the first segment of the flux section.

Two words of caution: flux sections MUST be defined before you solve the problem, because the program needs to calculate the values during the solution sequence, not afterward. In addition, all flux values are reported as positive, which means direction is not taken into account. This is required because the sign of the flux value will depend on which way you draw the section. To avoid any misinterpretation, all flux section values are reported as positive, and then you can plot flux vectors in order to determine the direction of flow, if it is not obvious based on your problem definition.

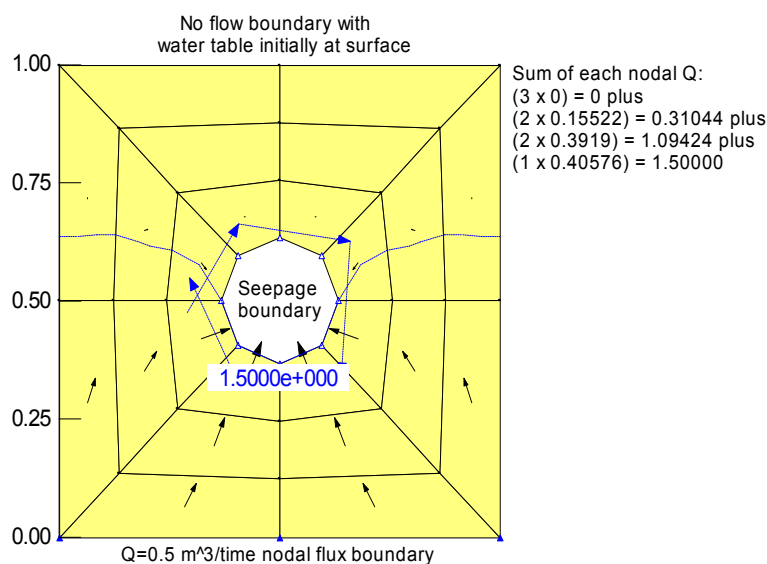


Figure 10-13 Flux section used around series of drain nodes to check flow

11 Modeling Tips and Tricks

11.1 Introduction

This chapter contains many useful hints about using the software and understanding what it does.

There have been many occasions where GEO-SLOPE has been contacted by clients with questions about how the model behaves in response to changes in various parameters. If we do not know the answer, we conduct a numerical experiment to test what will happen. The first few sections of this chapter illustrate a few common examples of numerical experiments. You are strongly encouraged to learn why these types of simple tests are so powerful in testing how the program computes results but they will also enhance your understanding of how the physical mechanisms of flow through porous medium occurs.

A numerical experiment is carried out by making a very simple finite element problem. It is useful to use a mesh that is one distance unit wide and one distance unit high. This makes hand-calculating flux values very simple and they can easily be checked against the computed flux values. The following discussion illustrates how some simple numerical experiments have been carried out to test some simple, yet valid, questions.

When setting up these experiments, it is a good idea to input simple soil property functions. In most cases, two data points are sufficient to define the conductivity and storage function. Just as a reminder, give both functions some slope – don't make them horizontal!

11.2 Problem engineering units

New in GeoStudio is the addition of units to all data input and output. The displayed units will be based on the set of units specified in the Set menu and any changes to units do not change the actual data. The units are present as an aide when setting up the model and considering output results.

Any system of units can be used for a seepage analysis; the only requirement is that you must be consistent. Fundamentally, you must select the units for length (geometry), time, and force. Once you have selected units for these parameters, all other units must be consistent. Table 11-1 presents some typical sets of consistent units.

Table 11-1 Consistent SI and Imperial units

Parameter	Symbol	SI Units	Imperial Units
Length	L	metres	feet
Time	t	seconds	hours
Force	F	kN	lbs
Pressure	F/L^2	kN/m ²	psf
Unit Weight of Water	F/L^3	kN/m ³	pcf
Hydraulic conductivity	L/t	m/sec	ft/hr
Total / Pressure head	L	m	ft
Nodal Flux (Q)	L^3/t	m ³ /sec	ft ³ /hr
Boundary Flux (q)	L/t	m/sec	ft/hr
Flux Section	L^3/t	m ³ /sec	ft ³ /hr
Volume	L^3	m ³	ft ³

The units of time are established once you select the units for hydraulic conductivity. The units of pressure are established once you select the unit weight of water. Generally, all units are defined by selecting the units of length for the problem geometry, units for hydraulic conductivity, and the units for the unit weight of water.

In summary, the key requirement is that the system of units be consistent.

11.3 Flux section location

Question: Does the location of a flux section within an element have any influence on the computed flux value?

Answer: No. The flux section value will be the same regardless of whether the section is drawn near the element edge or element middle.

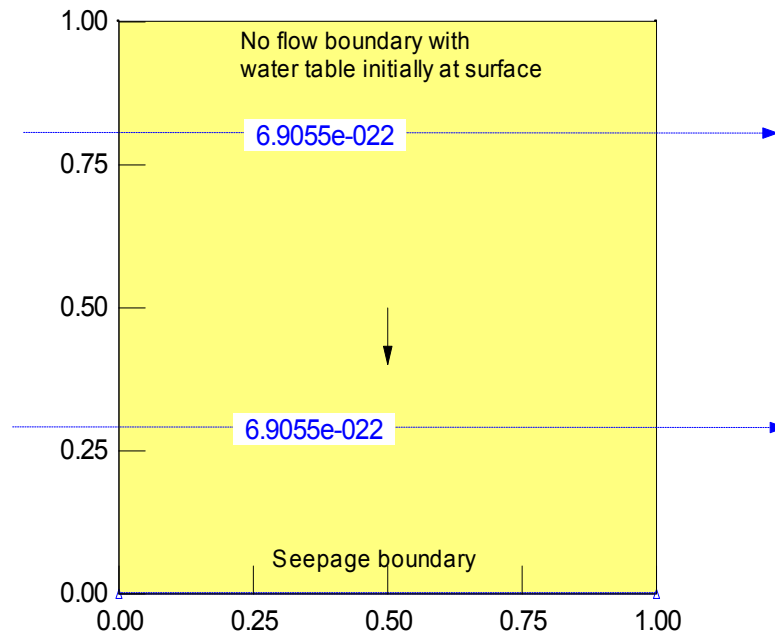


Figure 11-1 Test to check flux section locations

11.4 Drain flux values

Question: What is the best way to compute a total drain flux?

Answer: You can sum each individual nodal flux, or, draw a flux section around the nodes.

The figure below shows a total flux into the system through the three bottom nodes of 1.5 m^3 per time unit (each node contributing a total nodal flux of 0.5 m^3). This example also shows that with no other place for the water to flow except the drain, the flux into the base equals the flux out of the drain nodes.

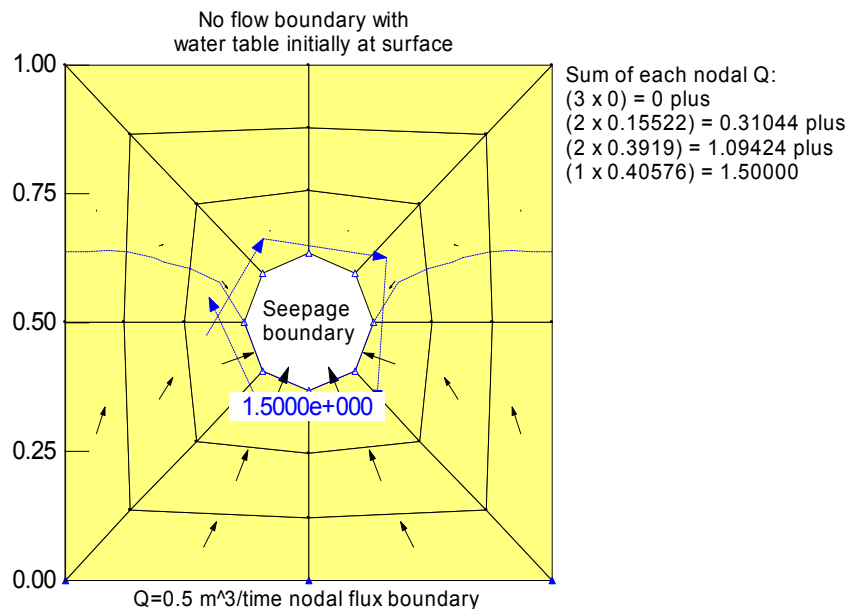


Figure 11-2 Test to see best method for observing drain flux

11.5 Unit flux versus total flux?

There are many people who are unsure of the difference between a unit flux and a total nodal flux. Do a simple test if you are unsure.

Question: How is a unit flux related to a total nodal flux in a 2D analysis?

Answer: The total nodal flux should be exactly equal to the unit flux multiplied by the total length of the element edges that contribute to that node.

In the figure below, a unit flux of 1 m³/ time / meter edge length has been applied to the bottom of the element. The top is a seepage face which will let the water out. The flux sections drawn in the element confirm that the total edge flux of 1 m³/ time has been converted by the solver into two equal total nodal fluxes of 0.5 m³/ time each.

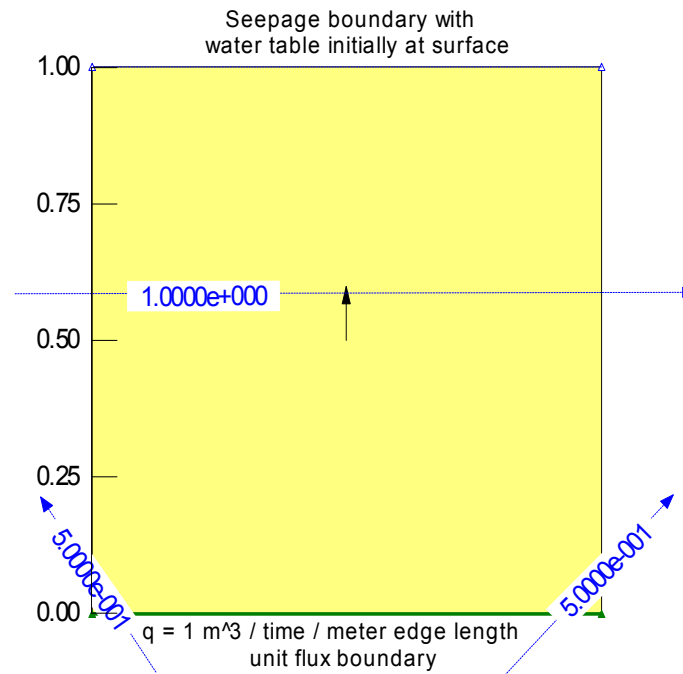


Figure 11-3 Test to compare unit flux and total flux

11.6 Flow above phreatic line

Question: Is there really flow above a water table?

Answer: The amount of flow through the unsaturated zone depends on its conductivity value as a function of how saturated the ground is. If the conductivity is quite high in a partially saturated state, the flow can be quite high.

In the numerical test below, there is a head gradient from right to left and the water table sits at the mid point of the mesh. You can see from the flux section values that 45% of the total flow passes through the soil ABOVE the phreatic surface in the unsaturated zone.

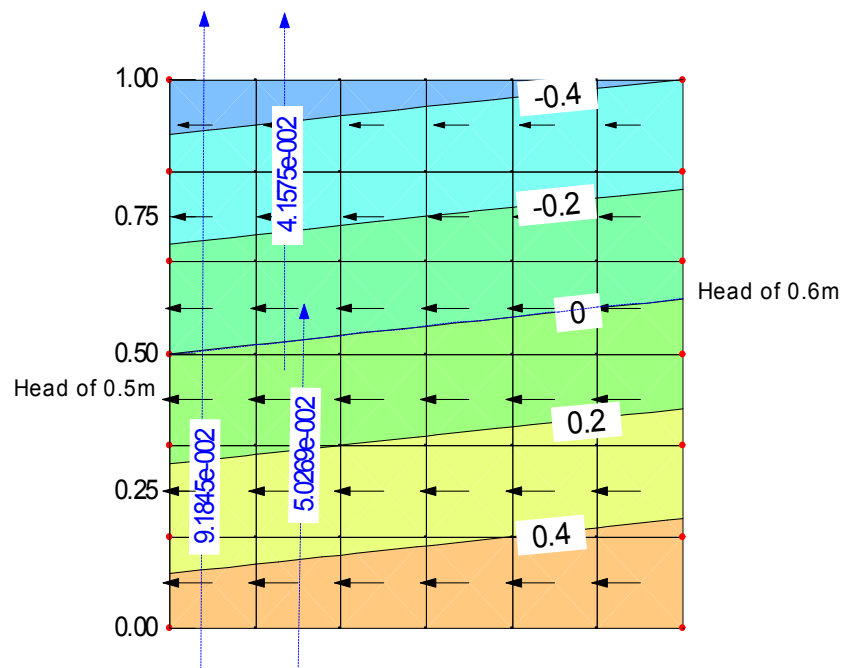


Figure 11-4 Test to see flow above phreatic line with pressure head contours shown.

11.7 Pressure boundary with depth

Question: If I know the pressure boundary with depth, do I have to put a different pressure value at each node?

Answer: If the pressure change is hydrostatic, simply specify a total head of a constant value along the entire edge and let the solver figure out the pressure at each node. The total constant head value should be equal to the elevation of the water table along that edge.

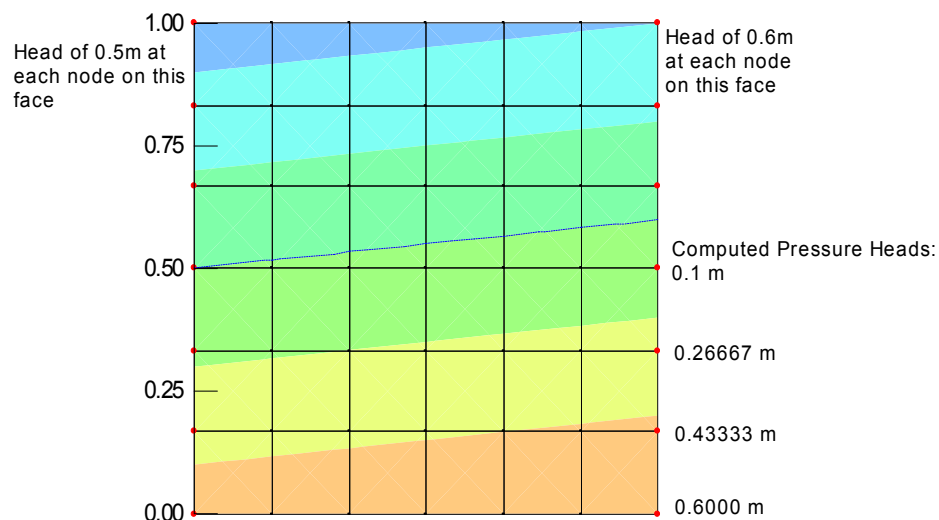


Figure 11-5 Test to see how pressure boundary can be applied as constant total head value along entire edge

11.8 Summing graphed data

In some cases you may want to know the total flow that crosses a boundary or the total change in stored water within the system. There is a trick you can do with the graphing feature to aid in this task.

The graphing feature has a check box that will “Sum(y) vs Average(x)” values. This option will result in all the “y” data points being added together and plotted versus the average of the “x” value each data point. If the “x” category is time, then there is no averaging and the resulting graph is a sum of all “y” values versus time.

Figure 11-6 shows the nodes selected for graphing on the upstream face of a reservoir during the filling stage. The objective is to know how much water crossed this boundary over time. One option is to draw a flux section but this is not the most accurate way in a transient analysis because the flux section is drawn through the elements and there is change in storage within the element. The most accurate way is to plot cumulative flow at each boundary node.

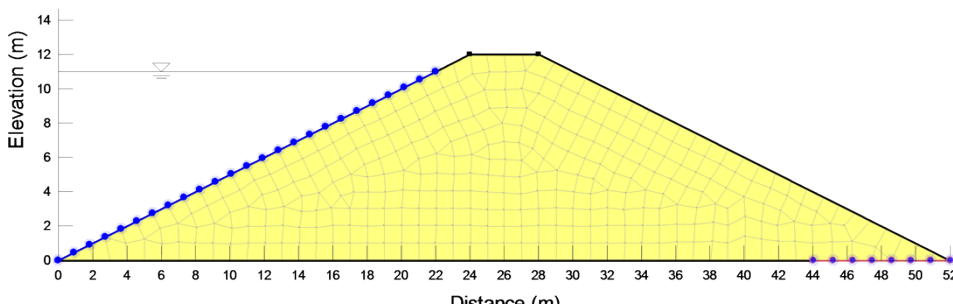


Figure 11-6 Graphing nodes selected on upstream dam face

The figure below shows the individual nodal flows at each selected node. You can see that each node has a different flow value. The second figure is much easier to interpret. In that graph, the check box for Sum(y) vs Average(x) was selected. This second graph is the cumulative total flow across the entire face.

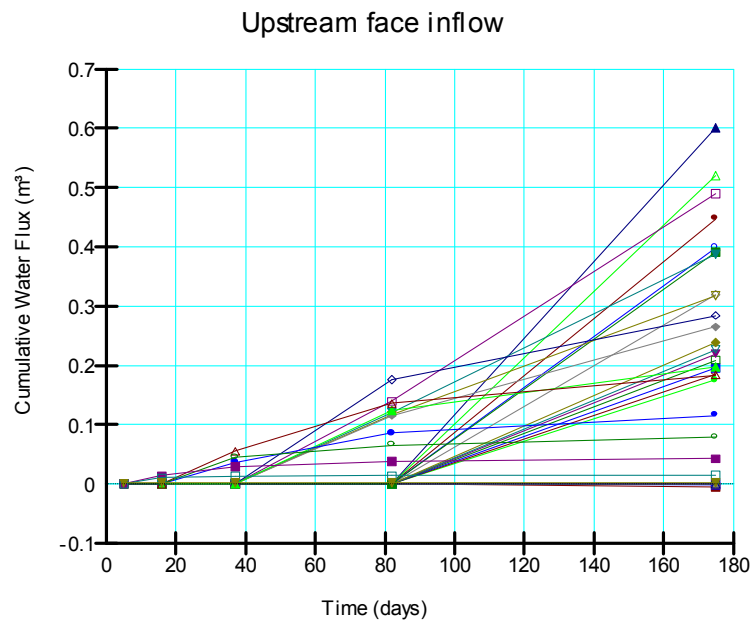


Figure 11-7 Individual nodal flow on upstream face

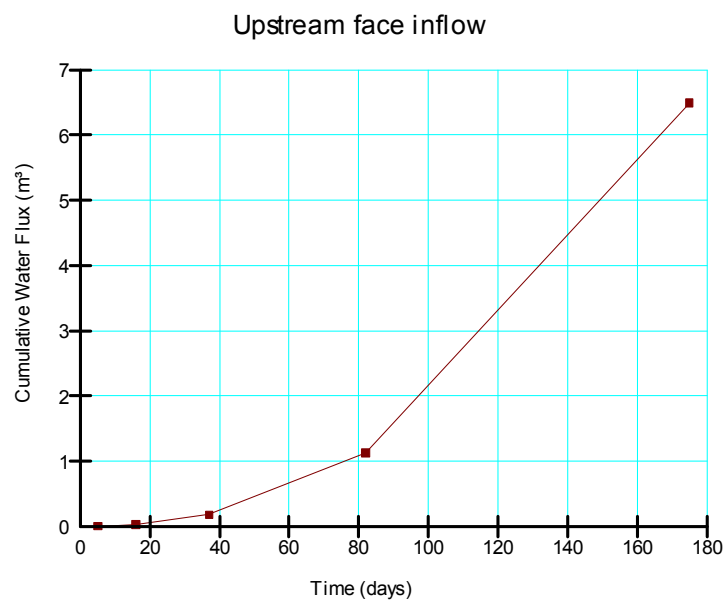


Figure 11-8 Sum of all nodal flow on upstream face

Suppose now we wanted to look at the change in stored water in the dam throughout a filling and draining cycle. We could select ALL the nodes in the mesh and use the Sum option as shown next.

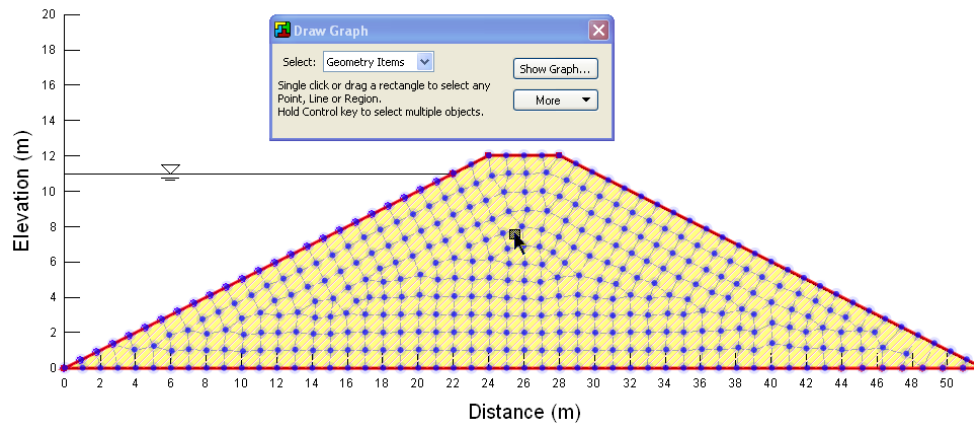


Figure 11-9 Selected the region geometry to graph data at all nodes

The resulting graph would be...

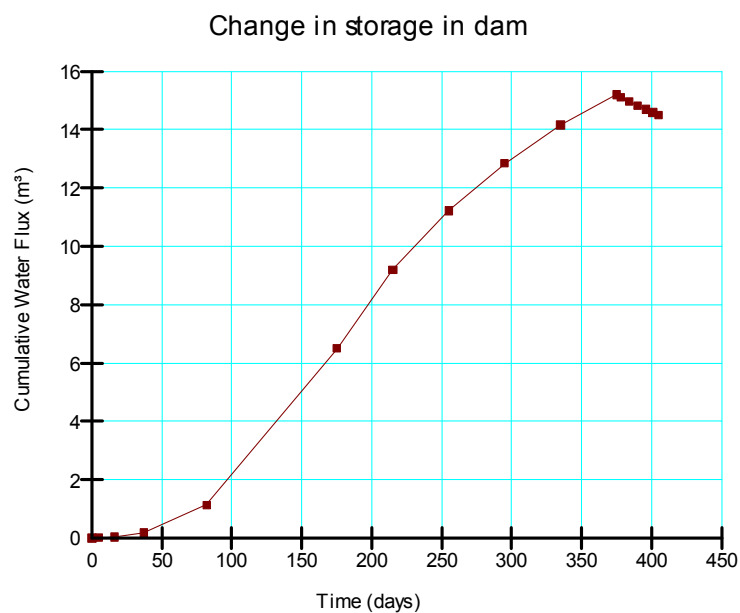


Figure 11-10 Change in stored water within dam over time

12 Illustrative Examples

A variety of verification and illustrative examples has been developed and are available with the software. These examples can be useful for learning how to model various problems, particularly in the selection and application of boundary conditions. Each example comes with a PDF document that provides explanations on the problem setup, comments on modeling techniques and a commentary on interpreting the results. Verification examples are discussed in terms of closed-form solutions, published information and/or laboratory measurements.

All of the examples can be downloaded and installed from GEO-SLOPE's web site (www.geo-slope.com). Once installed, it is possible to search for a particular type of analysis on the GeoStudio desktop. Conversely, the search feature is available directly on the website. It should be noted that a product-specific search is possible (e.g. search for TEMP/W or SIGMA/W).

The GeoStudio example files can be reviewed using the **free** GeoStudio Viewer license.

13 Theory

This chapter describes the theoretical engineering basis for the SEEP/W program. More specifically, it deals with the fundamental flow laws for steady state and transient flow, and it shows how these laws are represented in numerical form. This chapter is not a study of groundwater and flow in porous media. For advanced discussion of these topics the reader is referred to the textbooks *Groundwater*, by Alan Freeze and John Cherry (Prentice Hall, 1979); and *Unsaturated Soil Mechanics*, by Del Fredlund and Harianto Rahardjo (John Wiley & Sons, 1995).

13.1 Darcy's law

SEEP/W is formulated on the basis that the flow of water through both saturated and unsaturated soil follows Darcy's Law which states that:

$$q = ki$$

where:

$$\begin{aligned} q &= \text{the specific discharge,} \\ k &= \text{the hydraulic conductivity, and} \\ i &= \text{the gradient of total hydraulic head.} \end{aligned}$$

Darcy's Law was originally derived for saturated soil, but later research has shown that it can also be applied to the flow of water through unsaturated soil (see Richards, 1931 and Childs & Collins-George, 1950). The only difference is that under conditions of unsaturated flow, the hydraulic conductivity is no longer a constant, but varies with changes in water content and indirectly varies with changes in pore-water pressure.

Darcy's Law is often written as:

$$v = ki$$

where:

$$v = \text{the Darcian velocity.}$$

Note that the actual average velocity at which water moves through the soil is the linear velocity, which is equal to Darcian velocity divided by the porosity of the soil. In unsaturated soil, it is equal to Darcian velocity divided by the volumetric water content of the soil. SEEP/W computes and presents only the Darcian velocity.

13.2 Partial differential water flow equations

The general governing differential equation for two-dimensional seepage can be expressed as:

$$\text{Equation 13-1} \quad \frac{\partial}{\partial x} \left(k_x \frac{\partial H}{\partial x} \right) + \frac{\partial}{\partial y} \left(k_y \frac{\partial H}{\partial y} \right) + Q = \frac{\partial \theta}{\partial t}$$

where:

$$H = \text{the total head,}$$

k_x	=	the hydraulic conductivity in the x-direction,
k_y	=	the hydraulic conductivity in the y-direction,
Q	=	the applied boundary flux,
θ	=	the volumetric water content, and
t	=	time.

This equation states that the difference between the flow (flux) entering and leaving an elemental volume at a point in time is equal to the change in storage of the soil systems. More fundamentally, it states that the sum of the rates of change of flows in the x- and y-directions plus the external applied flux is equal to the rate of change of the volumetric water content with respect to time.

Under steady-state conditions, the flux entering and leaving an elemental volume is the same at all times. The right side of the equation consequently vanishes and the equation reduces to:

$$\frac{\partial}{\partial x} \left(k_x \frac{\partial H}{\partial x} \right) + \frac{\partial}{\partial y} \left(k_y \frac{\partial H}{\partial y} \right) + Q = 0$$

Changes in volumetric water content are dependent on changes in the stress state and the properties of the soil. The stress state for both saturated and unsaturated conditions can be described by two state variables (see Fredlund and Morgenstern, 1976 and Fredlund and Morgenstern, 1977). These stress state variables are $(\sigma - u_a)$ and $(u_a - u_w)$ where σ is the total stress, u_a is the pore-air pressure, and u_w is the pore-water pressure.

SEEP/W is formulated for conditions of constant total stress; that is, there is no loading or unloading of the soil mass. SEEP/W also assumes that the pore-air pressure remains constant at atmospheric pressure during transient processes. This means that $(\sigma - u_a)$ remains constant and has no effect on the change in volumetric water content. Changes in volumetric water content are consequently dependent only on changes in the $(u_a - u_w)$ stress state variable, and with u_a remaining constant, the change in volumetric water content is a function only of pore-water pressure changes. As a result, the change in volumetric water content can be related to a change in pore-water pressure by the following equation:

$$\text{Equation 13-2} \quad \partial\theta = m_w \partial u_w$$

where:

$$m_w = \text{the slope of the storage curve.}$$

The total hydraulic head, H , is defined as:

$$\text{Equation 13-3} \quad H = \frac{u_w}{\gamma_w} + y$$

where:

$$u_w = \text{the pore-water pressure,}$$

γ_w = the unit weight of water, and
 y = the elevation.

Equation 13-3 can be rearranged as:

$$u_w = \gamma_w (H - y)$$

Substituting Equation 13-3 into Equation 13-2 gives the following equation:

$$\partial\theta = m_w \gamma_w \partial(H - y)$$

which now can be substituted into Equation 13-1, leading to the following expression:

$$\frac{\partial}{\partial x} \left(k_x \frac{\partial H}{\partial x} \right) + \frac{\partial}{\partial y} \left(k_y \frac{\partial H}{\partial y} \right) + Q = m_w \gamma_w \frac{\partial(H-y)}{\partial t}$$

Since the elevation is a constant, the derivative of y with respect to time disappears, leaving the following governing differential equation used in SEEP/W finite element formulation:

$$\frac{\partial}{\partial x} \left(k_x \frac{\partial H}{\partial x} \right) + \frac{\partial}{\partial y} \left(k_y \frac{\partial H}{\partial y} \right) + Q = m_w \gamma_w \frac{\partial H}{\partial t}$$

13.3 Finite element water flow equations

Applying the Galerkin method of weighed residual to the governing differential equation, the finite element for two-dimensional seepage equation can be derived as:

Equation 13-4

$$\begin{aligned} & \tau \int_A ([B]^T [C] [B]) dA \{H\} + \tau \int_A (\lambda \langle N \rangle^T \langle N \rangle) dA \{H\}, t \\ & = q \tau \int_L (\langle N \rangle^T) dL \end{aligned}$$

where:

$[B]$ = the gradient matrix,
 $[C]$ = the element hydraulic conductivity matrix,
 $\{H\}$ = the vector of nodal heads,
 $\langle N \rangle$ = the vector of interpolating function,
 q = the unit flux across the edge of an element,
 τ = the thickness of an element,
 t = time,
 λ = storage term for a transient seepage equals to $m_w \gamma_w$,
 A = a designation for summation over the area of an element, and

L = a designation for summation over the edge of an element.

In an axisymmetric analysis, the equivalent element thickness is the circumferential distance at different radius, R about the symmetric axis. The complete circumferential distance is 2π radian times R , since SEEP/W is formulated for one radian, the equivalent thickness is R . Therefore, the finite element equation for the axisymmetric case is:

$$\int_A ([B]^T [C] [B] R) dA \{H\} + \int_A (\lambda \langle N \rangle^T \langle N \rangle R) dA \{H\}, t \\ = q \int_L (\langle N \rangle^T R) dL$$

Note that the radial distance R is not a constant within an element as in the case of the thickness τ in the two-dimensional analysis; consequently, R is a variable inside the integral.

In an abbreviated form, the finite element seepage equation can be expressed as:

$$\text{Equation 13-5 } [K] \{H\} + [M] \{H\}, t = \{Q\}$$

where:

$[K]$ = the element characteristic matrix

$[M]$ = the element mass matrix

$\{Q\}$ = the element applied flux vector

Equation 13-5 is the general finite element equation for a transient seepage analysis. For a steady-state analysis, the head is not a function of time and, consequently, the term $\{H\}, t$ vanishes, reducing the finite element equation to:

$$[K] \{H\} = \{Q\}$$

which is the abbreviated finite element form of the fundamental seepage equation, Darcy's Law.

13.4 Temporal integration

The finite element solution for a transient analysis is a function of time as indicated by the $\{H\}, t$ term in the finite element equation. The time integration can be performed by a finite difference approximation scheme. Writing the finite element equation in terms of finite differences leads to the following equation (see Segerlind, 1984, pp. 183-185):

$$(\omega \Delta t [K] + [M]) \{H_1\} = \\ \Delta t ((1 - \omega) \{Q_0\} + \omega \{Q_1\}) + ([M] - (1 - \omega) \Delta t [K]) \{H_0\}$$

where:

t = the time increment,

ω = a ratio between 0 and 1,

H_1 = the head at end of time increment,

- H_0 = the head at start of time increment,
 Q_1 = the nodal flux at end of time increment, and
 Q_0 = the nodal flux at start of time increment.

SEEP/W uses the Backward Difference Method, a method that sets ω to 1.0, the finite element equation is then simplified to:

$$\text{Equation 13-6} \quad (\Delta t [K] + [M]) \{H_1\} = \Delta t \{Q_1\} + [M] \{H_0\}$$

As indicated by Equation 13-6, in order to solve for the new head at the end of the time increment, it is necessary to know the head at the start of the increment. Stated in general terms, the initial conditions must be known in order to perform a transient analysis.

13.5 Numerical integration

SEEP/W uses Gaussian numerical integration to evaluate the element characteristic matrix $[K]$ and the mass matrix $[M]$. The integrals are evaluated by sampling the element properties at specifically defined points and then summed together for the entire element.

Using the characteristic matrix $[K]$ as an example, the following integral (from Equation 13-4):

$$[K] = \tau \int_A ([B]^T [C] [B]) dA$$

can be replaced by:

$$[K] = \tau \sum_{j=1}^n [B_j]^T [C_j] [B_j] \det |J_j| W_{1j} W_{2j}$$

where:

- j = an integration point,
 n = the number of integration points,
 $[C_j]$ = the element hydraulic conductivity matrix at the integration point,
 $[B_j]$ = the element matrix at the integration point,
 $\det |J_j|$ = the determinant of the Jacobian matrix, and
 $W_{1j} W_{2j}$ = a weighting factors.

The number of sample (integration) points required in an element depends on the number of nodes and the shape of the elements. The tables below contain the number and location of sampling points that are used by SEEP/W.

Table 13-1 Sample point locations and weightings for four point quadrilateral element

Point	r	s	w1	w2
1	+0.57735	+0.57735	1.0	1.0
2	-0.57735	+0.57735	1.0	1.0
3	-0.57735	-0.57735	1.0	1.0
4	+0.57735	-0.57735	1.0	1.0

Table 13-2 Sample point locations and weightings for nine point quadrilateral elements

Point	r	s	w1	w2
1	+0.77459	+0.77459	5/9	5/9
2	-0.77459	+0.77459	5/9	5/9
3	-0.77459	-0.77459	5/9	5/9
4	+0.77459	-0.77459	5/9	5/9
5	0.00000	+0.77459	8/9	5/9
6	-0.77459	0.00000	5/9	8/9
7	0.00000	-0.77459	8/9	5/9
8	+0.77459	0.00000	5/9	8/9
9	0.00000	0.00000	8/9	8/9

Table 13-3 Sample point locations and weighting for one point triangular element

Point	r	s	w1	w2
1	0.33333	0.33333	1.0	0.5

Table 13-4 Sample point locations and weightings for three point triangular element

Point	r	s	w1	w2
1	0.16666	0.16666	1/3	1/2
2	0.66666	0.16666	1/3	1/2
3	0.16666	0.66666	1/3	1/2

One-point integration for a triangular element results in a constant gradient throughout the element. The number of integration points is denoted as the integration order. The appropriate integration order is a function of the presence of secondary nodes. When secondary nodes are present, the interpolating functions are nonlinear and consequently a higher integration order is required. Table 13-5 gives the acceptable integration orders.

Table 13-5 Acceptable element integration orders

Element Type	Secondary Nodes	Integration Order
Quadrilateral	no	4
Quadrilateral	yes	4 or 9
Triangular	no	1 or 3
Triangular	yes	3

It is also acceptable to use four-point integration for quadrilateral elements that have secondary nodes. This is called a reduced integration order (see Bathe, 1982, p. 282). Acceptable results can be obtained with reduced integration. For example, reduced integration is useful in saturated zones where the hydraulic gradient is low and the hydraulic conductivity is constant. Selective use of reduced integration can greatly reduce the required number of computations.

It is also possible to use three-point and nine-point integration with elements that have no secondary nodes. However, the benefits of this are marginal, particularly for quadrilateral elements. Nine point integration for a quadrilateral element involves substantially more computing than four point integration, and there is little to be gained from the additional computations. As a general rule, quadrilateral elements should have secondary nodes to achieve significant benefits from the nine point integration.

The situation is slightly different for triangular elements. One-point integration means the material properties and flow gradients are constant within the element. This can lead to poor performance of the element, particularly if the element is in an unsaturated zone where the hydraulic conductivity varies sharply with changes in pore-water pressure. Using three point integration, even without using secondary nodes, can improve the performance, since material properties and gradients within the elements are distributed in a more realistic manner. The use of one point integration in triangular elements with secondary nodes is not acceptable.

In general, it is sufficient to use three-point integration for triangular elements and four-point integration for quadrilateral elements. In situations where there is unsaturated zone with hydraulic conductivity varies sharply within an element, it is best to use quadrilateral elements with secondary nodes together with nine-point integration.

13.6 Hydraulic conductivity matrix

The general form of the SEEP/W element hydraulic conductivity matrix is:

$$[C] = \begin{bmatrix} C_{11} & C_{12} \\ C_{21} & C_{22} \end{bmatrix}$$

where:

$$\begin{aligned} C_{11} &= k_x \cos^2 \alpha + k_y \sin^2 \alpha \\ C_{22} &= k_x \sin^2 \alpha + k_y \cos^2 \alpha \\ C_{12} &= k_x \sin \alpha \cos \alpha + k_y \sin \alpha \cos \alpha \\ C_{21} &= C_{12} \end{aligned}$$

The parameters k_x , k_y and α are defined in Figure 13-1.

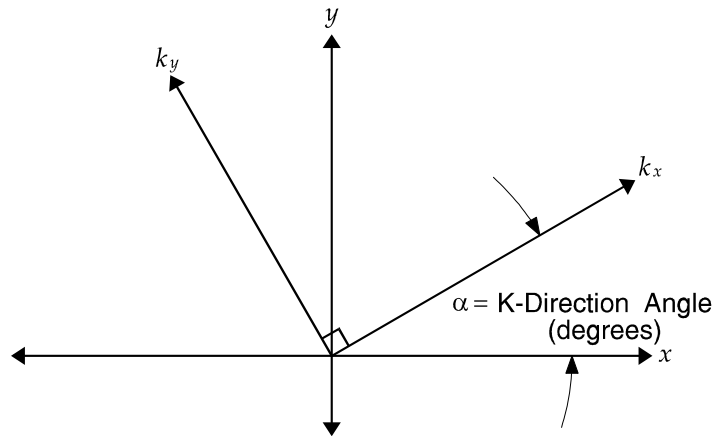


Figure 13-1 Definition of hydraulic conductivity matrix parameters

When α is zero, $[C]$ is reduced to:

$$[C] = \begin{bmatrix} k_x & 0 \\ 0 & k_y \end{bmatrix}$$

The parametric k_x is always determined from the hydraulic conductivity function. Parameter k_y is then computed as k_x multiplied by k_{Ratio} . In equation form,

$$k_y = k_x \times k_{Ratio}$$

13.7 Mass matrix

As first presented in Equation 13-4 the element mass (or storage) matrix for a two-dimensional analysis is defined as:

$$[M] = \tau \int_A (\lambda < N >^T < N >) dA$$

Similar to the element characteristic matrix, the mass matrix is also evaluated by numerical integration as shown below:

$$[M] = \tau \sum_{j=1}^n \lambda_j \langle N \rangle^T \langle N \rangle \det |J_j| W_{1j} W_{2j}$$

When the mass matrix is evaluated, SEEP/W uses a lumped formulation to lump the off-diagonal coefficients in the mass matrix to the diagonal terms. For higher order elements, diagonal scaling is also used. The lumped formulation, together with diagonal scaling, is adopted to improve stability in transient analysis. The detail of the lumped formulation is given by Segerlind, 1984, pp. 178-182.

To evaluate λ , SEEP/W obtains a m_w value from the volumetric water content function (the storage function) for each integration point in one of two ways. In most cases it computes m_w from the slope of a straight line between the old and new pore-water pressures at a Gauss point, as illustrated in Figure 13-2. The slope of this straight line can be viewed as the average rate of change during one increment of time. This is considered to be a more realistic value than taking the derivative of the function at a specific point. An exception to this procedure is when the old and new pore-water pressures are nearly identical. In this case, SEEP/W computes m_w by calculating the slope of the function at the average of the old and new pore-water pressures.

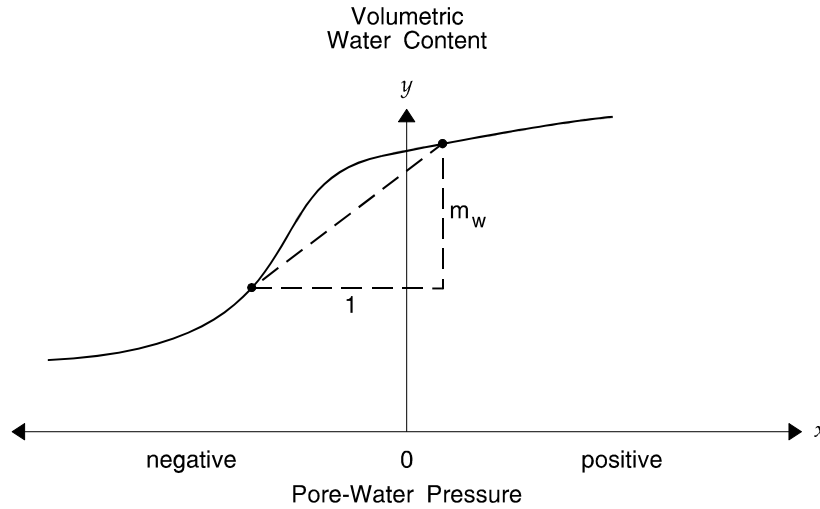


Figure 13-2 Computation of storage term, m_w

13.8 Flux boundary vector

The nodal flux boundary vector $\{Q\}$ for a two-dimensional analysis is defined as:

$$\{Q\} = q\tau \int_L \langle N \rangle^T dL$$

or for an axisymmetric analysis as:

$$\{Q\} = q \int_L \langle N \rangle^T R dL$$

and for a plan view analysis as:

$$\{Q\} = q \int_A (\langle N \rangle^T \langle N \rangle) dA$$

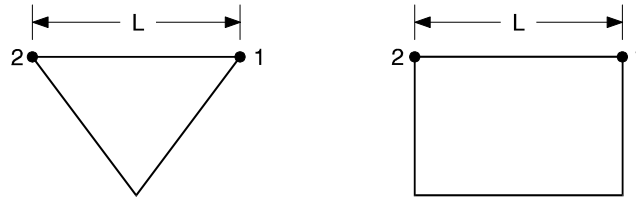
where:

- q = the unit flux across the side of an element,
- τ = the element thickness,
- A = the area of the element in plan view, and
- R = the radial distance from the symmetric axis to the element corner nodes.

Solutions to the integrals are dependent on the analysis type and on the presence of secondary nodes. For two-dimensional (i.e., vertical section) and axisymmetric analyses, the integrals are solved by closed form solutions as illustrated in Figure 13-3 and Figure 13-4. However, for plan view analysis, the contributing area of a node is computed by numerical integration in the same way as forming the mass matrix. In other words, the contributing area per node in a plan view depends on the partial contributing areas of all elements surrounding that node, and this must be computed using the Gauss region area integration scheme for each element, not standard shape relationships as discussed here.

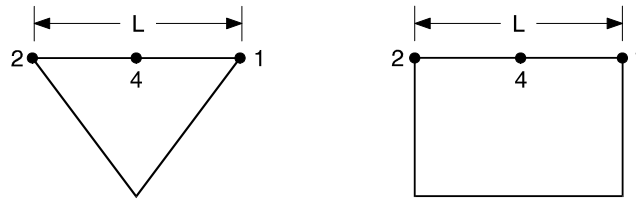
Two types of flux boundaries may be specified in SEEP/W namely: a nodal flux boundary (Q) and a unit flux boundary (q). A nodal flux boundary (Q) can be specified directly on the boundary nodes. A unit flux boundary (q) must be specified along the boundary edges of the elements, except for a plan view analysis where the unit flux is applied per unit area on the plan view. When you set up a boundary condition, you identify the edges of the elements across which a q boundary should be applied. Based on this specific element edge information, the solver performs the integration and determines the applied flux Q at the nodes. The solver needs Q, not q, to solve the finite element equations.

For two-dimensional (i.e., vertical section) and axisymmetric analyses, the nodal flux Q computed by the solver is dependent on the specified element thickness. For plan view analysis, since the surface area is independent of the element thickness, the nodal flux Q is also independent of the element thickness.



$$a_1 = a_2 = \frac{L}{2}$$

(a) No intermediate node



$$a_1 = a_2 = \frac{L}{6}$$

$$a_4 = \frac{4L}{6}$$

(b) With an intermediate node

Figure 13-3 Contributing area for section view elements with unit thickness

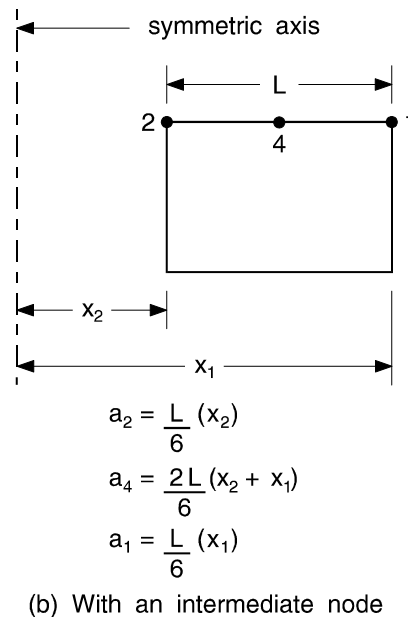
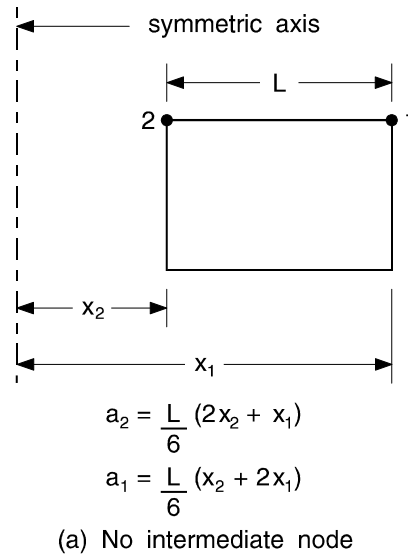


Figure 13-4 Contributing area for axisymmetric elements over unit Radian thickness

13.9 Density-dependent flow

SEEP/W may be used together with CTRAN/W to perform density-dependent contaminant transport analyses. For density-dependent analyses, an instance of SEEP/W SOLVE is started and controlled by CTRAN/W SOLVE, so that the groundwater flow equation can be solved simultaneously with the contaminant transport equation at each time step. The simultaneous solution of the groundwater flow equation and the contaminant transport equation is required for density-dependent flow problems, because the groundwater flow velocities are dependent on the contaminant densities, which in turn are dependent on the contaminant concentrations.

The formulation used in SEEP/W for density-dependent flow was proposed by Frind, (1982). To accommodate density-dependent flow analyses, a density body force term is added to the governing

groundwater flow equation. After discretizing the problem domain into finite elements, the density body force term results in a body force vector, $[G]$, which is added to the element flow equation in SEEP/W.

In matrix form, the body force vector can be expressed as:

$$\text{Equation 13-7} \quad [G] = \int_A \bar{K}_y \cdot \gamma \cdot \bar{C} \cdot [B_y] dA$$

where:

- $[G]$ = element density body force vector,
- $\bar{K}_y$ = element average hydraulic conductivity in the y-direction,
- $\bar{C}$ = element average contaminant concentration,
- γ = contaminant density contrast equals to $(\rho_r - 1)$
- ρ_r = contaminant relative density at reference concentration, and
- $[B_y]$ = element gradient matrix in the y-direction

Equation 13-7 shows that the body force vector is a function of the element average contaminant concentration, hydraulic conductivity of the material and the density contrast between contaminated water and freshwater. Note that if the contaminant relative density at the reference concentration is 1.0, the contaminant density contrast $(\rho_r - 1)$ is zero. In other words, the density body force vector $[G]$ becomes zero when there is no density contrast between the contaminated water and the freshwater.

14 Appendix A: Interpolating Functions

14.1 Coordinate systems

The global coordinate system used in the formulation of SEEP/W is the first quadrant of a conventional x y Cartesian system.

The local coordinate system used in the formulation of element matrices is presented in Figure 14-1. Presented as well in Figure 14-1 is the local element node numbering system. The local coordinates for each of the nodes are given in Table 14-1.

Table 14-1 Local element node numbering system

Element Type	Node	r	s
Quadrilateral	1	+1	+1
	2	-1	+1
	3	-1	-1
	4	+1	-1
	5	0	+1
	6	-1	0
	7	0	-1
	8	1	0
Triangular	1	0	0
	2	1	0
	3	1	1
	4	—	—
	5	$\frac{1}{2}$	0
	6	$\frac{1}{2}$	$\frac{1}{2}$
	7	0	$\frac{1}{2}$
	8	—	—

SEEP/W uses the fourth node to distinguish between triangular and quadrilateral elements. If the fourth node number is zero, the element is triangular. If the fourth node number is not zero, the element is quadrilateral.

In the case of quadrilateral elements, Nodes 5, 6, 7, and 8 are secondary nodes. In the case of triangular elements, Nodes 5, 6, and 7 are secondary nodes.

The local and global coordinate systems are related by a set of interpolation functions. SEEP/W uses the same functions for relating the coordinate systems as for describing the variation of the field variable (head) within the element. The elements are consequently isoperimetric elements.

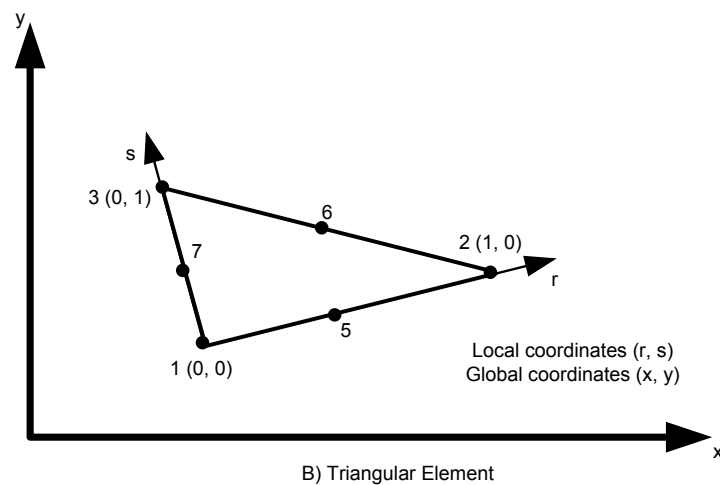
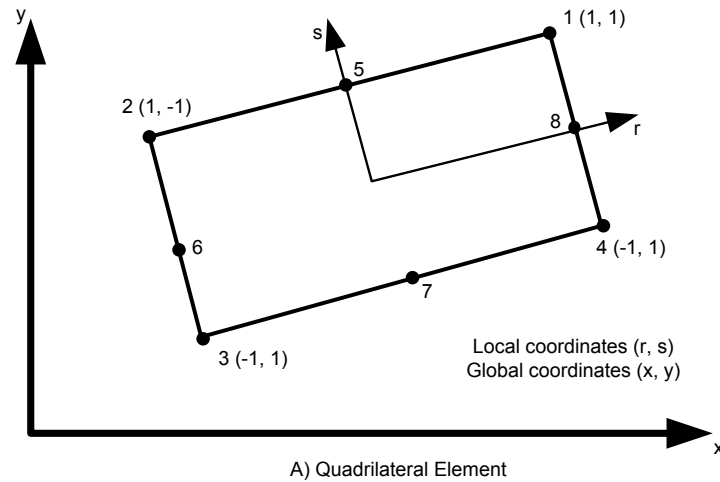


Figure 14-1 Global and local coordinate system

The x- and y-coordinates anywhere in the element are related to the local coordinates and to the x- and y-coordinates of the nodes by the following equations:

Equation 14-1

$$x = \langle N \rangle \{X\}$$

$$y = \langle N \rangle \{Y\}$$

where:

- $\langle N \rangle$ = is a vector of interpolating shape functions,
- $\{X\}$ = the global x-coordinates of the element nodes, and
- $\{Y\}$ = the global y-coordinates of the element node.

The interpolating functions are expressed in terms of local coordinates. Therefore, once a set of local coordinates (r,s) have been specified, the corresponding global coordinates can be determined by Equation 14-1.

14.2 Interpolating functions

SEEP/W uses a general set of interpolating functions presented by Bathe (1982). These general functions are suitable for elements which have none, some, or all of the secondary nodes defined. This allows for considerable versatility in the types of elements that can be used.

The interpolating functions in terms of local coordinates r and s for quadrilateral and triangular elements are given in Table 14-2 and Table 14-3, respectively.

The functions represent a linear equation when the secondary nodes are missing, and a quadratic (nonlinear) equation when the secondary nodes are included.

Table 14-2 Interpolation functions for quadrilateral elements

Function	Include in function if node is present			
	5	6	7	8
$N1 = \frac{1}{4}(1+r)(1+s)$	$-\frac{1}{2}N5$	—	—	$-\frac{1}{2}N8$
$N2 = \frac{1}{4}(1-r)(1+s)$	$-\frac{1}{2}N5$	$-\frac{1}{2}N6$	—	—
$N3 = \frac{1}{4}(1-r)(1-s)$	—	$-\frac{1}{2}N6$	$-\frac{1}{2}N7$	—
$N4 = \frac{1}{4}(1+r)(1-s)$	—	—	$-\frac{1}{2}N7$	$-\frac{1}{2}N8$
$N5 = \frac{1}{2}(1-r^2)(1+s)$	—	—	—	—
$N6 = \frac{1}{2}(1-s^2)(1-r)$	—	—	—	—
$N6 = \frac{1}{2}(1-s^2)(1-r)$	—	—	—	—
$N7 = \frac{1}{2}(1-r^2)(1-s)$	—	—	—	—

Table 14-3 Interpolation functions for triangular elements

Function	Include in function if node is present		
	5	6	7
$N1 = 1-r-s$	$-\frac{1}{2}N5$		$-\frac{1}{2}N7$
$N2 = r$	$-\frac{1}{2}N5$	$-\frac{1}{2}N6$	
$N3 = s$		$-\frac{1}{2}N6$	$-\frac{1}{2}N7$
$N5 = 4r(1-s)$			
$N6 = 4rs$			
$N7 = 4s(1-r-s)$			

Field variable model

To formulate a finite element analysis, it is necessary to adopt a model for the distribution of the field variable within the element. Since the field variable in the seepage analysis is the total head (H), it is necessary to adopt a model for the distribution of H within the element.

SEEP/W assumes that the head distribution within the element follows the adopted interpolating functions. This means that the head distribution is linear when the secondary nodes are missing, and the head distribution is nonlinear when the secondary nodes are present.

In equation form, the head distribution model is:

$$\text{Equation 14-2} \quad h = \langle N \rangle \{H\}$$

where:

$$\begin{aligned} h &= \text{the head at any local coordinate,} \\ \langle N \rangle &= \text{a vector of interpolation function, and} \\ \{H\} &= \text{a vector of heads at the nodes.} \end{aligned}$$

Interpolation function derivatives

The constitutive relationship for a seepage analysis is Darcy's Law. The equation is:

$$q = ki$$

The gradient i is one of the key parameters required in the finite element formulation. The following presents the procedure used by SEEP/W to compute the gradient.

From the adopted head distribution model, the head at any point within the element in terms of the nodal heads is obtained from Equation 14-2

The gradients in the x and y directions are then known by:

$$\begin{aligned} i_x &= \frac{\partial h}{\partial x} = \frac{\partial \langle N \rangle}{\partial x} \{H\} \\ i_y &= \frac{\partial h}{\partial y} = \frac{\partial \langle N \rangle}{\partial y} \{H\} \end{aligned}$$

Equation 14-3

The interpolating functions are written in terms of r and s and not in terms of x and y . The derivatives must consequently be determined by the chain rule of differentiation, as follows:

$$\frac{\partial \langle N \rangle}{\partial s} = \frac{\partial \langle N \rangle}{\partial x} \frac{\partial x}{\partial s} + \frac{\partial \langle N \rangle}{\partial y} \frac{\partial y}{\partial s}$$

This can be written as:

$$\left\{ \frac{\partial \langle N \rangle}{\partial r} \right\} = [J] \left\{ \frac{\partial \langle N \rangle}{\partial x} \right\}$$

where $[J]$ is the Jacobian matrix and is defined as:

Equation 14-4

$$[J] = \begin{bmatrix} \frac{\partial x}{\partial r} & \frac{\partial y}{\partial r} \\ \frac{\partial x}{\partial s} & \frac{\partial y}{\partial s} \end{bmatrix}$$

The derivatives of the interpolation function with respect to x and y is called the B matrix and can be determined by inverting the Jacobian matrix and rewriting the equations as:

$$[B] = \begin{bmatrix} \frac{\partial \langle N \rangle}{\partial x} \\ \frac{\partial \langle N \rangle}{\partial y} \end{bmatrix} = [J]^{-1} \begin{bmatrix} \frac{\partial \langle N \rangle}{\partial r} \\ \frac{\partial \langle N \rangle}{\partial s} \end{bmatrix}$$

Recalling Equation 14-1 and taking their derivative with respect to s and r as follows:

$$\begin{aligned} \frac{\partial x}{\partial r} &= \frac{\partial \langle N \rangle}{\partial r} \{X\} \\ \frac{\partial x}{\partial s} &= \frac{\partial \langle N \rangle}{\partial s} \{X\} \\ \frac{\partial y}{\partial r} &= \frac{\partial \langle N \rangle}{\partial r} \{Y\} \\ \frac{\partial y}{\partial s} &= \frac{\partial \langle N \rangle}{\partial s} \{Y\} \end{aligned}$$

then substituting these values into Equation 14-4, the Jacobian matrix becomes:

$$[J] = \begin{bmatrix} \frac{\partial(N_1, N_2, \dots, N_8)}{\partial r} \\ \frac{\partial(N_1, N_2, \dots, N_8)}{\partial s} \end{bmatrix} \begin{bmatrix} X_1 & Y_1 \\ X_2 & Y_2 \\ \vdots & \vdots \\ X_8 & Y_8 \end{bmatrix}$$

The derivatives of the interpolation functions with respect to r and s are required to compute the Jacobian matrix and to compute the flow gradients (Equation 14-3).

The derivatives of the interpolation functions with respect to r and s used by SEEP/W for quadrilateral and triangular elements are given in Table 14-4 and

Table 14-5 respectively.

Table 14-4 Interpolation function derivatives for quadrilateral elements

Derivative of Function	Include in derivative if node is present			
	5	6	7	8
$N1,r = \frac{1}{4}(1+s)$	$-\frac{1}{2}(N5,r)$	—	—	—
$N2,r = -\frac{1}{4}(1+s)$	$-\frac{1}{2}(N5,r)$	$-\frac{1}{2}(N6,r)$	—	—
$N3,r = -\frac{1}{4}(1-s)$	—	$-\frac{1}{2}(N6,r)$	$-\frac{1}{2}(N7,r)$	—
$N4,r = \frac{1}{4}(1-s)$	—	—	$-\frac{1}{2}(N7,r)$	$-\frac{1}{2}(N8,r)$
$N5,r = -\frac{1}{2}(2r+2sr)$	—	—	—	—
$N6,r = -\frac{1}{2}(1-s^2)$	—	—	—	—
$N7,r = -\frac{1}{2}(2r-2sr)$	—	—	—	—
$N8,r = \frac{1}{2}(1-s^2)$	—	—	—	—
$N1,s = \frac{1}{4}(1+r)$	$-\frac{1}{2}(N5,s)$	—	—	$-\frac{1}{2}(N8,s)$
$N2,s = \frac{1}{4}(1-r)$	$-\frac{1}{2}(N5,s)$	$-\frac{1}{2}(N6,s)$	—	—
$N3,s = -\frac{1}{4}(1-r)$	—	$-\frac{1}{2}(N6,s)$	$-\frac{1}{2}(N7,s)$	—
$N4,s = -\frac{1}{4}(1+r)$	—	—	$-\frac{1}{2}(N7,s)$	$-\frac{1}{2}(N8,s)$
$N5,s = \frac{1}{2}(1-r^2)$	—	—	—	—
$N6,s = -\frac{1}{2}(2s-2sr)$	—	—	—	—
$N7,s = -\frac{1}{2}(1-r^2)$	—	—	—	—
$N8,s = -\frac{1}{2}(2s+2sr)$	—	—	—	—

Table 14-5 Interpolation function derivatives for triangular elements

Derivative of Function	Include in derivative if node is present		
	5	6	7
$N1,r = -1.0$	$-\frac{1}{2}(N5,r)$	—	—
$N2,r = 1.0$	$-\frac{1}{2}(N5,r)$	$-\frac{1}{2}(N6,r)$	—
$N3,r = 0.0$	—	$-\frac{1}{2}(N6,r)$	$-\frac{1}{2}(N7,r)$
$N5,r = (4-8r-4s)$	—	—	—
$N6,r = 4s$	—	—	—
$N7,r = -4s$	—	—	—
$N1,s = -1.0$	$-\frac{1}{2}(N5,s)$	—	—
$N2,s = 0.0$	$-\frac{1}{2}(N5,s)$	$-\frac{1}{2}(N6,s)$	—
$N3,s = 1.0$	—	$-\frac{1}{2}(N6,s)$	$-\frac{1}{2}(N7,s)$
$N5,s = -4r$	—	—	—
$N6,s = 4r$	—	—	—
$N7,s = (4-4r-8s)$	—	—	—

The following notation is used in the preceding tables:

$$N_{i,r} = \frac{\partial N_i}{\partial r}$$

$$N_{i,s} = \frac{\partial N_i}{\partial s}$$

The Jacobian matrix is a 2x2 matrix:

$$[J] = \begin{bmatrix} J_{11} & J_{12} \\ J_{21} & J_{22} \end{bmatrix}$$

The inverse of [J] is:

$$[J]^{-1} = \begin{bmatrix} J_{22} & -J_{12} \\ -J_{21} & J_{11} \end{bmatrix}$$

The determinant of [J] is:

$$\det[J] = J_{11} \times J_{22} - J_{21} \times J_{12}$$

References

- Anonymous. 1999. Definition of Geotechnical Engineering, Ground Engineering Magazine, Vol. 32, No. 11, p. 39.
- Arya, L.M., and Paris, J.F. 1981. A Physico-empirical Model to Predict the Soil Moisture Characteristic from Particle-size Distribution and Bulk Density Data. Soil Science Society of America Journal, 45:1023-1030.
- Aubertin, M., Mbonimpa, M., Bussière, B. and Chapuis, R.P. 2003. A model to predict the water retention curve from basic geotechnical properties. Canadian Geotechnical Journal, 40(6): 1104-1122 (2003)
- Bathe, K-J. 1982. Finite Element Procedures in Engineering Analysis. Prentice-Hall.
- Black, P., and Tice, A. 1989. Comparison of Soil Freezing Curve and Soil Water Curve Data for Windsor Sandy Loam. Water Resources Research 25(10):2205-2210.
- Brooks, R.H., and Corey, J.C. 1966. Properties of Porous Media Affecting Fluid Flow. ASCE Journal, Irrigating and Drainage Division.
- Bruch, P.G. 1993. A laboratory study of evaporative fluxes in homogeneous and layered soils. M.Sc. Thesis, Department of Civil Engineering, University of Saskatchewan, Saskatoon, Canada.
- Burland, J.B. 1987. Nash Lecture: The Teaching of Soil Mechanics – a Personal View. Proceedings, 9th ECSMFE, Dublin, Vol. 3, pp 1427-1447.
- Burland J.B. 1996. Closing Session Discussions, Proceedings of the First International Conference on Unsaturated Soils Conference, Paris, Vol. 3, p. 1562. (Proceedings published by A.A. Balkema; Editors, E.E. Alonso and P. Delage).

- Carter, J.P., Desai, C.S, Potts, D.M., Schweiger, H.F, and Sloan, S.W. 2000. Computing and Computer Modeling in Geotechnical Engineering, Invited Paper, Conference Proceedings, GeoEng2000, Melbourne, Australia.
- Childs, E.C., and Collis-George, N., 1950. The Permeability of Porous Materials. Proceedings of the Royal Society, pp. 392-405.
- Corey, J.C., 1957. Measurement of Water and Air Permeability in Unsaturated Soil. Soil Science of America, Vol. 21.
- Croucher and O'Sullivan, 1985 – CTRAN Henry example reference
- Elzeftawy, A., and Cartwright, K., 1981. Evaluating the Saturated and Unsaturated Hydraulic Conductivity of Soils. Permeability and Groundwater Contaminant Transport, ASTM STP, T.F. Zimmie and C.D. Riggs Editors, pp. 168-181.
- Fredlund, D.G., and Morgenstern, N.R., 1976. Constitutive Relations for Volume Change in Unsaturated Soils. Canadian Geotechnical Journal, Vol. 13, pp. 261-276.
- Fredlund, D.G. and Morgenstern, N.R., 1977. Stress State Variables for Unsaturated Soils. ASCE, Vol. 103, pp. 447-464.
- Fredlund, D.G. and Rahardjo, H., 1993. Soil Mechanics for Unsaturated Soils. John Wiley & Sons, Inc.
- Fredlund, D. G., and Anqing Xing. 1994. Equations for the soil-water characteristic curve. Canadian Geotechnical Journal. Vol. 31, pp: 521-532.
- Freeze, R.A. and Cherry, J.A., 1979. Groundwater. Prentice-Hall.
- Frind, E.O., 1982. Simulation of Long-Term Transient Density-Dependent Transport in Groundwater. Advance Water Resources, 5, pp.73-88.
- Gonzalez, P.A., and Adams, B.J., 1980. Mine tailings disposal: I. laboratory characterization of tailings. Publication 80-05, Department of Civil Engineering, University of Toronto, Toronto, Canada.
- Green, R.E. and Corey, J.C., 1971. Calculation of Hydraulic Conductivity: A Further Evaluation of Some Predictive Methods. Soil Science Society of America Proceedings, Vol. 35, pp. 3-8.
- Hillel, D., 1980. Fundamentals of Soil Physics, Academic Press, New York.
- Ho, P.G., 1979. The Prediction of Hydraulic Conductivity from Soil Moisture Suction Relationship. B.Sc. Thesis, University of Saskatchewan, Saskatoon, Canada.
- Huang, S.Y., 1994. Measurement and Prediction of the Coefficient of Permeability in Unsaturated Soils. Ph.D. Thesis, Department of Civil Engineering, University of Saskatchewan, Saskatoon, Canada.
- Kisch, M., 1959. The Theory of Seepage From Clay-Blanketed Reservoirs. Geotechnique, Vol. 9.
- Klute, A., 1965. Laboratory Measurements of Hydraulic Conductivity of Unsaturated Soil. Methods of Soil Analysis, Part 1. Agronomy 9, C.A. Black Edition, pp. 253-261.

- Kovács, G. (1981). Seepage Hydraulics. Developments in Water Science 10. Elsevier Science Publishers, Amsterdam.
- Kunze, R.J., Vehara, G., and Graham, K., 1968. Factors Important in the Calculation of Hydraulic Conductivity. Soil Science of America Proceedings, Soil Physics, Vol. 32.
- Lambe, T.W. and Whitman, R.V., 1969. Soil Mechanics. John Wiley and Sons.
- Lancaster, P. and Salkauskas, K., 1986. Curve and Surface Fitting: An Introduction. Academic Press.
- Massmann, J.W., Applying groundwater flow models in vapor extraction system design, *J. Environ. Engr.*, 115(1), 129-49, 1989.
- Marshall, T.J., 1958. A Relation Between Permeability and Size Distribution of Pores. Soil Science of America Journal, Vol. 9.
- Mercer, J.W. and C.R. Faust. 1981. Ground-Water Modeling. National Water Well Association
- Milly, P.C.D, 1982. Moisture and heat transport in hysteretic, inhomogeneous porous media: A matric head based formulation and a numerical model. Water Resources Research, 18(3): 489-498.
- Millington, R.J., and Quirk, J.P., 1961. Permeability of Porous Solids. Transaction of the Faraday Society, Vol. 57, pp. 1200 1207.
- Morgenstern, N.R. 2000. Common Ground, Invited Paper, Conference Proceedings, GeoEng2000, Melbourne, Australia.
- National Research Council, 1990. Groundwater Models: Scientific and regulatory applications. Water Science and Technology Board, Committee on Ground Water Modelling Assessment. Commission on Physical Sciences, Mathematics and Resources. National Research Council, National Academy Press, Washington, D.C.
- Newman, G. P. 1995. Heat and Mass Transfer of Unsaturated Soils During Freezing. M.Sc., Thesis, Univ. of Sask.
- O’Kane Consultants, 2004. Personal communication.
- Richards, L.A., 1931. Capillary Conduction of Liquids Through Porous Mediums. Physics, Vol. 1.
- Rulon, J.J, and Freeze, R.A., 1985. Multiple Seepage Faces on Layered Slopes and their Implications for Slope Stability Analysis. Canadian Geotechnical Journal, Vol. 22.
- Rulon, J.J., 1984. The Development of Multiple Seepage Faces along Heterogeneous Hillsides. Ph.D. Thesis, University of British Columbia, Vancouver, Canada.
- Salkauskas, K. 1974. C' Splines For Interpolation of Rapidly Varying Data. Rocky Mountain Journal of Mathematics 14(1):239-250.
- Segerlind, L.J. 1984. Applied Finite Element Analysis. John Wiley and Sons.
- Spitz, K., and Moreno, J. 1996. A Practical Guide to Groundwater and Solute Transport Modelling. John Wiley, New York.

-
- Swanson, D. 1991. The Effects of Loading on the Moisture Characteristic and Permeability-Suction Relationships for Unsaturated Soils. B.Sc. Thesis, Department of Civil Engineering, University of Saskatchewan, Saskatoon, Canada.
- van Genuchten, M. Th. 1980. A closed-form equation for predicting the hydraulic conductivity of unsaturated soils. Soil Science Society of America Journal 44:892-898.
- Wilson, G.W. 1990. Soil Evaporative Fluxes for Geotechnical Engineering Problems. Ph.D. Thesis, University of Saskatchewan, Saskatoon, Canada.

Index

Adaptive time stepping	113	Drain flux	161
Add-in	121	Effective stress	142
air-entry value	72	Element and mesh compatibility	44
Analysis Types	107	Element fundamentals	43
Axisymmetric	114	Element nodes	43
Boundary Conditions	80	Equipotential lines	132, 151
boundary flows	88	Evaluate results in the context of expected results	21
Boundary functions	98	evaporation	95
Burland	6	Far field head	87
Burland triangle	6	Field variable distribution	43
Circular openings	33	Finite element water flow equations	173
Closed form curve fits	121	Flow channels	133
Coefficient of volume compressibility	63	Flow Nets	131
Compare alternatives	9	Flow paths	132
Constant pressure conditions	87	Flux boundary vector	179
Contours	152	Flux sections	155
Convergence	123	Fredlund and Xing, 1994	60
Critical gradients	139	Fredlund et al, 1994	68
Darcy's law	171	Free drainage	94
Density-dependent flow	182	Frozen ground hydraulic conductivity	66
Do numerical experiments	18		

Functions.....	118	Mass matrix.....	178
Gauss integration	127	Material Properties.....	53
Gauss point values	151	Meshing.....	25
General guidelines for meshing	47	Meshing for transient analyses.....	41
Geometry objects	26	Model only essential components..	19
Gradients	138	Modeling Tips & Tricks.....	159
Graphing	148	Modified Kovacs.....	57
Green and Corey, 1971	69	Modifier function	104
Head boundary conditions	83	Nodal flux Q versus time	102
Head versus volume.....	101	Node and element information.....	145
How not to model.....	22	Numerical integration	46, 175
How to model.....	14	Numerical Issues	123
Hydraulic conductivity.....	64	Numerical modeling.....	3
Hydraulic conductivity matrix	177	phreatic line.....	163
Identify governing parameters	10	Plan view.....	115
infiltration	95	points.....	27
Initial conditions	108	Product integration illustrations ..	169
initial water table.....	110	Quantitative predictions	8
Interface elements	31	regions.....	27
Interpolating functions	185	residual volumetric water content..	78
Interrogate the results.....	21	saturated hydraulic conductivity	74
Joining regions	40	Secondary variables	47
lines	27	Seepage faces	92

Seepage forces	137	Time stepping.....	111, 129
Sensitivity of results to material properties.....	72	Transient	108
Simplify geometry	16	Triangular regions.....	36
slope of the VWC function	76	understand physical process.....	11
Soil water storage.....	54	Unit flow rate versus time.....	103
Sources and sinks.....	91	Unit flux versus total flux	162
Spatial functions.....	121	unit gradient	94
Spline	118	Unstructured mesh	35
Start simple	17	Uplift pressures	136
Start with estimated material properties.....	20	van Genuchten, 1980	70
Steady state	107	Velocity vectors and flow paths...	153
Structured mesh	35	Visualization of Results	145
Surface mesh.....	37	volumetric water content.....	56
Theory	171	Why model?	7

A consistent set of units

Parameter	Symbol	SI Units	Imperial Units
Length	L	meters	feet
Time	t	seconds	hours
Force	F	kN	Ibs
Pressure	F/L^2	kN/m ²	psf
Unit weight water	F/L^3	kN/m ³	pcf
Hydraulic conductivity	L/t	m/sec	ft/hr
Total/pressure head	L	m	ft
Nodal flux (Q)	L^3/t	m ³ /sec	ft ³ /hr
Boundary flux (q)	L^3/t	m/sec	ft/hr
Flux section	L^3/ft	m ³ /sec	ft/hr
Volume	L^3	m ³	ft ³

Note: it is also possible to use time units of days.