Thermal Modeling with TEMP/W

An Engineering Methodology

September 2014 Edition

GEO-SLOPE International Ltd.

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
1.1 Typical applications 1
Thermal design for roads and airstrips 1
Ground freezing for soil stabilization 2
Insulation design for shallow buried piping 4
Thawing or freezing beneath heated or chilled structures 5
Freezing around chilled pipelines 6
Convective cooling of surfaces 7
1.2 About this book 8
2 Numerical Modeling – What, Why and How 11
2.1 Introduction 11
2.2 What is a numerical model 11
2.3 Modeling in geotechnical engineering 15
2.4 Why model 16
Quantitative predictions 17
Compare alternatives in design or modeling methodology 18
Identify governing parameters 20
Discover and understand physical process - train our thinking 20
2.5 How to model 23
Make a guess 23
Simplify geometry 24
Start simple 26
Do numerical experiments 26
Model only essential components 27
Start with estimated material properties 28
Interrogate the results 28
Evaluate results in the context of expected results 29
Remember the real world 29
2.6 How not to model 29
2.7 Closing remarks 30
3 Geometry and Meshing 31
3.1 Introduction 31
3.2 Geometry Objects in GeoStudio 32

Soil Regions, Points and Lines 33
Free Points 35
Free Lines 35
Interface Elements on Lines 36
Circular Openings 38
3.3 Mesh Generation 39
Structured mesh 40
Unstructured quad and triangle mesh 40
Unstructured triangular mesh 40
Triangular grid regions 41
Rectangular grid of quads 42
3.4 Surface layers 42
3.5 Joining regions 45
3.6 Meshing for transient analyses 46
3.7 Finite Elements 47
3.8 Element fundamentals 48
Element nodes 48
Field variable distribution 48
Element and mesh compatibility 49
Numerical integration 51
Secondary variables 52
3.9 General guidelines for meshing 52
Number of elements 53
Effect of drawing scale 53
Mesh purpose 54
Simplified geometry 55
4 Material Models and Properties 57
4.1 Material models in TEMP/W 57
4.2 Interface model parameters 58
4.3 Thermal functions 58
Unfrozen water content function 58
Thermal conductivity 60
Estimating thermal conductivity for soils 61
Typical values of thermal conductivity 62
Volumetric heat capacity 63
Estimating volumetric heat capacity 64

Typical values of volumetric heat capacity 65
4.4 Sensitivity of results to thermal material properties and water content in soil 67
5 Boundary Conditions 71
5.1 Introduction 71
5.2 Fundamentals 71
5.3 Boundary condition locations 73
Region face boundary conditions 74
5.4 Temperature boundary conditions 74
5.5 Heat flow boundary conditions 74
5.6 Sources and sinks 77
5.7 Ground surface energy balance 77
5.8 Rigorous approach to calculation of ground surface temperatures 78
No snow present 78
With snow pack 79
5.9 Traditional approach to computing ground temperatures 80
Empirical approach to estimating ground surface temperature 80
5.10 Convective heat transfer surfaces 82
Artificial ground freezing application 83
Pyrex tile cooling application 85
Time steps for convective surface analysis 86
5.11 Thermosyphon boundary conditions 86
5.12 Boundary functions 88
General 89
Temperature versus time 89
Nodal and unit heat flux versus time (Q and q) 90
Modifier functions 90
6 Analysis Types 93
6.1 Steady-state 93
Boundary condition types in steady-state 93
6.2 Transient 93
Initial conditions 94
Activation values 95
Spatial function for the initial conditions 95
No initial condition 95
6.3 Time stepping - temporal integration 95
Finite element temporal integration formulation 96

Problems with time step sizes 96
General rules for setting time steps 97
6.4 Staged / multiple analyses 98
6.5 Axisymmetric 98
6.6 Plan view 100
7 Functions in GeoStudio 103
7.1 Spline functions 103
Slopes of spline functions 104
7.2 Linear functions 105
7.3 Step functions 105
7.4 Closed form curve fits for water content functions 106
7.5 Add-in functions 106
7.6 Spatial functions 106
8 Numerical Issues 108
8.1 Convergence 108
Significant figures 109
Minimum difference 109
8.2 Evaluating convergence 109
Mesh view 109
Graphs 110
Commentary 111
8.3 Under-relaxation 111
8.4 Gauss integration order 111
8.5 Equation solvers 112
8.6 Time stepping 113
Automatic adaptive time stepping 113
9 Visualization of Results 116
9.1 Transient versus steady-state results 116
9.2 Node and element information 116
9.3 Graphing Node and Gauss Data 118
9.4 “None” values 120
9.5 Isolines 121
9.6 Projecting Gauss point values to nodes 122
9.7 Contours 124
9.8 Animation in GeoStudio 125

9.9 Energy flow vectors and flow paths 125
Calculating gradients and velocities 125
Energy flow vectors 126
Flow paths 127
9.10 Flux sections 127
Flux section theory 127
Flux section application 129

10 Modeling Tips and Tricks 131
10.1 Introduction 131
10.2 Problem engineering units 131
10.3 Flux section location 133
10.4 Source or sink flux values 133
10.5 Unit energy flux versus total energy flux? 134
10.6 Summing graphed data 135

11 Illustrative Examples 139

12 Theory 140
12.1 Flow law 140
12.2 Governing equations 140
12.3 Finite element heat flow equations 141
12.4 Temporal integration 143
12.5 Numerical integration 143
12.6 Thermal conductivity matrix 145
12.7 Heat storage matrix 146
12.8 Flux boundary vector 147
12.9 Convective heat transfer with SEEP/W or AIR/W 150

13 Appendix A: Interpolating Functions 153
13.1 Coordinate systems 153
13.2 Interpolating functions 155
Field variable model 155
Interpolation function derivatives 156

References 160

Index 162

1 Introduction

TEMP/W is a finite element software product that can be used to model the thermal changes in the ground due to environmental changes, or due to the construction of facilities, such as buildings or pipelines. The comprehensive formulation makes it possible to analyze both simple and highly complex geothermal problems, with or without temperatures that result in freezing or thawing of soil moisture.

TEMP/W cannot be used to predict frost heave at this time. While TEMP/W can be used with SEEP/W to model the affect of moving water on heat transfer, it cannot deal with the very complex and little understood physical coupling between water, ice, air and soil during heaving.

An assumption inherent in all TEMP/W analysis is that the moisture content does not change during the solution. The ground does not need to be saturated, but the total moisture content is assumed to be fixed. This means that the summation of ice and water always adds up to the total moisture content by volume.

It is also possible to model situations where there is no moisture in the soil, which means that TEMP/W can actually be used to model heat transfer through any type of porous or solid material. Heat transfer in solid materials is actually a very simplified case of what TEMP/W is capable of analyzing. While this simplified case will be discussed periodically throughout the book, the main focus is heat transfer through porous media containing soil, water, ice and air at the same time.

1.1 Typical applications

Thermal design for roads and airstrips

For designing roads and airstrips in areas that experience cold winters, the use of insulation to control the depth of frost penetration may be used to eliminate freezing of frost susceptible soils located beneath the subgrade. TEMP/W can be used to compute the transient distribution of subsurface temperatures during design.

Figure 1-1 shows the maximum frost penetration below a cleared highway designed without insulation. Additional TEMP/W modeling could be used to assess alternative designs using insulation and various depths of drained subgrade to minimize the frost penetration directly beneath the highway.

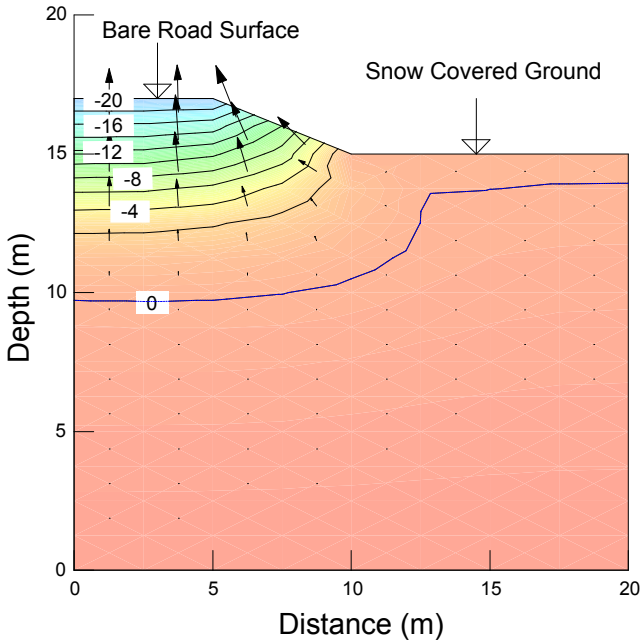


Figure 1-1 Frost penetration beneath a cleared highway

Ground freezing for soil stabilization

For many geotechnical engineering projects, excavation through soft soils is required. Ground freezing is sometimes used before excavation to provide stabilized soil. For a successful soil freezing system, things like the appropriate number of wells, well placement, energy flux requirements, freezing time requirements, and coverage of frozen zones must all be estimated during design. TEMP/W is a useful tool for obtaining these estimates, because the ground freezing process can be modeled on a site-specific basis.

Figure 1-2 shows the results of modeling axisymmetric freezing around a single brine pipe using TEMP/W. This particular modeling example can be used to estimate energy flux requirements, freezing time, and freezing coverage for a single well. An axisymmetric analysis works ok for a single freeze pipe, but multiple freeze pipes should be modeled in plan view to show the inter-relationship between adjacent pipes and growth of the frozen wall after adjacent frozen columns have connected. This is discussed in more detail in the illustrative examples chapter.

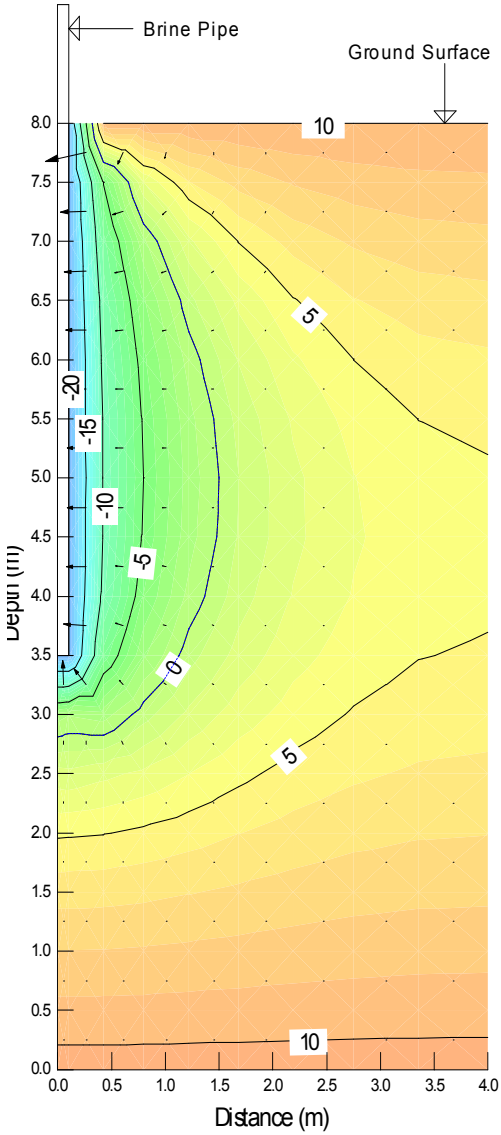


Figure 1-2 Frost bulb around freezing pipe

In Figure 1-3, the convective heat transfer boundary condition option has been used to compute the actual heat removal from each freeze pipe in a mine shaft freezing program. The heat removed is a function of ground temperature, brine temperature, brine flow rate, and pipe geometry. TEMP/W computes this value at each time step as it changes during the active freezing period. In this figure, the actual pipe geometry is entered into TEMP/W and then the heat flux boundary condition is applied at a single node in end view. This way, you do not have to create a fine mesh around the pipe and a coarse mesh far away from the pipe. The other advantage of the convective heat transfer boundary condition approach is that you do not have to assume the freeze pipe wall temperature, which is always less than the actual brine temperature.

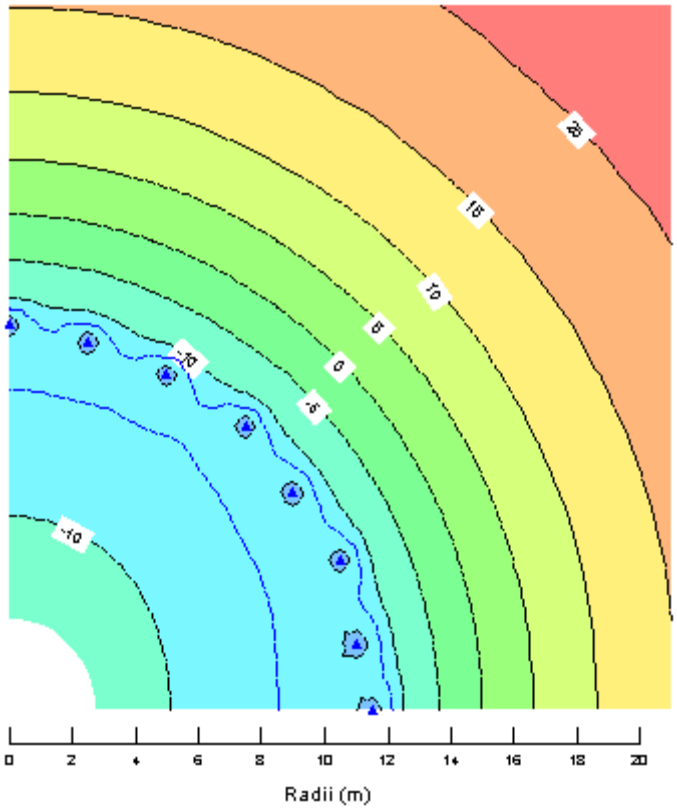


Figure 1-3 Plan view of freeze wall growth around a shaft excavation

Note: in Figure 1-3, the ground water being frozen is Sodium Chloride rich so the actual freeze point temperature was set in the model to be minus 11 degrees Celsius. The "frozen ground" isotherm is the curvy blue line. In this case, the ground water being frozen was set to freeze at zero degrees Celsius, so the frozen ground isotherm is at the zero degree point.

Insulation design for shallow buried piping

Many buried pipes, such as water supply pipes, do not require insulation design because they are typically placed below the frost line. Even when frost does penetrate to buried pipes, sufficient fluid flow through the pipes carries enough latent heat that the fluid does not freeze while flowing through the pipe. However, in cases where there are limitations on trench depth and/or intermittent flow through the buried pipe, a design which includes insulation to protect against freezing may be required. TEMP/W can be used in these cases.

Figure 1-4 shows the results of an analysis performed during the design of a shallow buried piping system for a groundwater remediation project. The pipes were located beneath inverted-U insulation at a depth of about 1.0 m below the ground surface. Insulation design in this case is important because of the limitations on trench depth and the intermittent water flow within the buried pipes. The figure below shows the maximum frost penetration, which occurs in early spring as the thaw front is migrating downward from the ground surface.

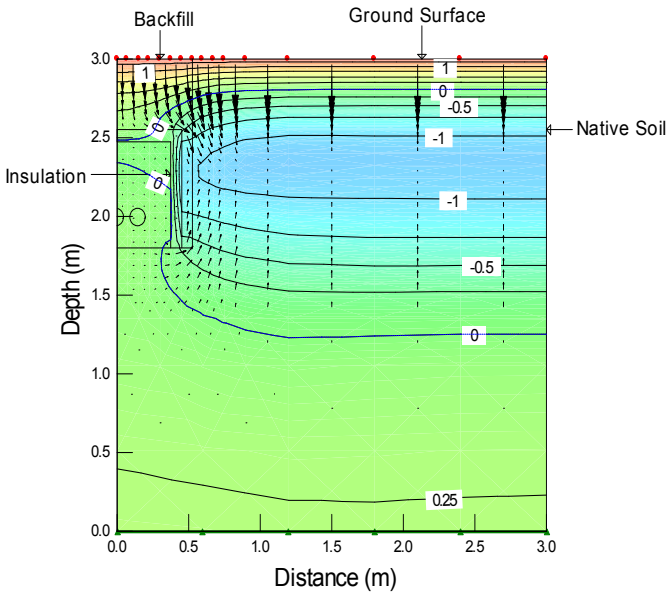


Figure 1-4 Frost penetration around shallow buried piping

Thawing or freezing beneath heated or chilled structures

Construction of heated structures on permafrost is complicated by the fact that excessive permafrost thawing can have detrimental effects on the structure's foundation. Successful design of these structures therefore depends on an analysis of the heat transfer from the structure into the underlying soil. TEMP/W can be used to perform these analyses.

Figure 1-5 illustrates the results of an axisymmetric analysis of heat transfer below a warm circular structure. The results show the long term thawing of the permafrost beneath the structure. This analysis shows the permafrost thawing in the absence of any insulation in the foundation of the structure. Additional modeling could show the effect of installing insulation to minimize thawing.

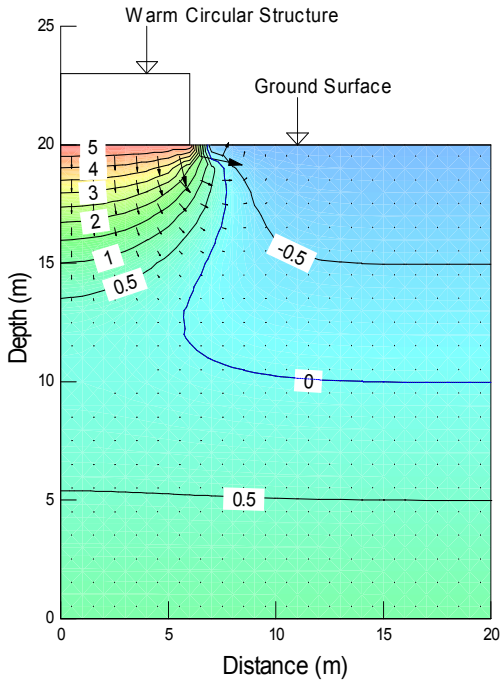


Figure 1-5 Thawing permafrost beneath a heated structure

In the next example, thermosyphons have been installed beneath the building to help stabilize the permafrost. Outside the building, the climate data option has been used to predict the ground temperature beneath the snow or on the exposed ground surface. The thermosyphon is a special type of boundary condition in TEMP/W.

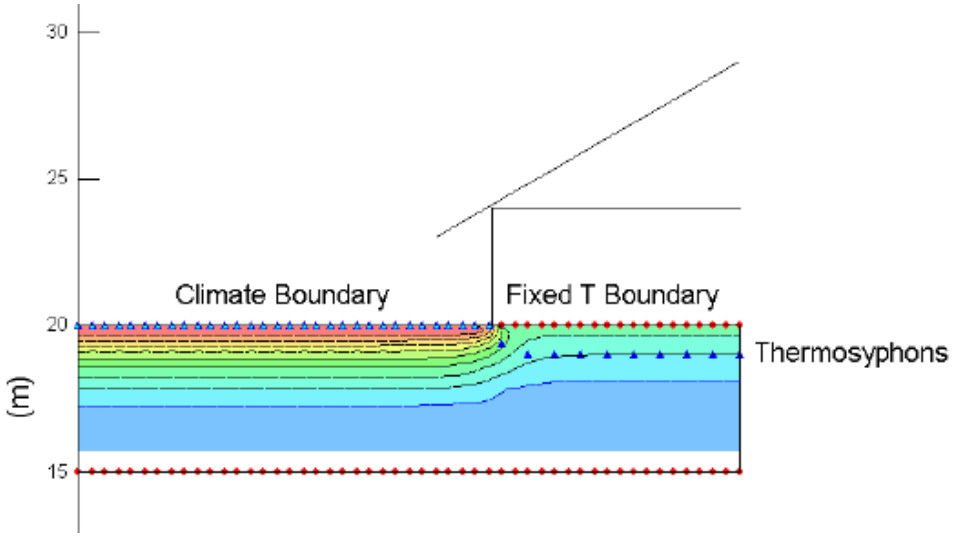


Figure 1-6 Thermosyphons used to support permafrost beneath a structure

Freezing around chilled pipelines

Chilled pipelines, for transporting commodities such as natural gas, typically traverse large distances through a diversity of soil types. An important factor to consider in the design of chilled pipelines is the size of the frost bulb that will develop in the soil around the pipe. In frost susceptible soils, the frost action around the pipe will cause frost heave which will lift the pipe vertically. This is especially

troublesome where the pipeline crosses the interface between a frost susceptible soil and soil which is not very frost susceptible. In these cases, differential frost heave will cause the pipe to bend, thus placing stresses on the pipe. Pipeline design therefore depends on factors such as native soil type, backfill material, pipe diameter, pipe wall thickness, pipe temperature, and amount of insulation applied to the outside of the pipe. Central to the design is the heat transfer from the soil to the pipe in both the transient, start-up phase, and in the long term operational phase. TEMP/W can be used to perform the heat transfer analyses required for pipeline design.

Figure 1-7 shows the long term distribution of temperature around a chilled pipeline. The analysis shows the depth of frost penetration below the pipe and can be used to determine the heat flux into the pipe from the surrounding soil. In this example, fluid flow is affecting the shape of the thermal profile. TEMP/W can account for transient fluid flow in the heat transfer analysis. It does so my simultaneously solving SEEP/W which provides water contents and flow velocities.

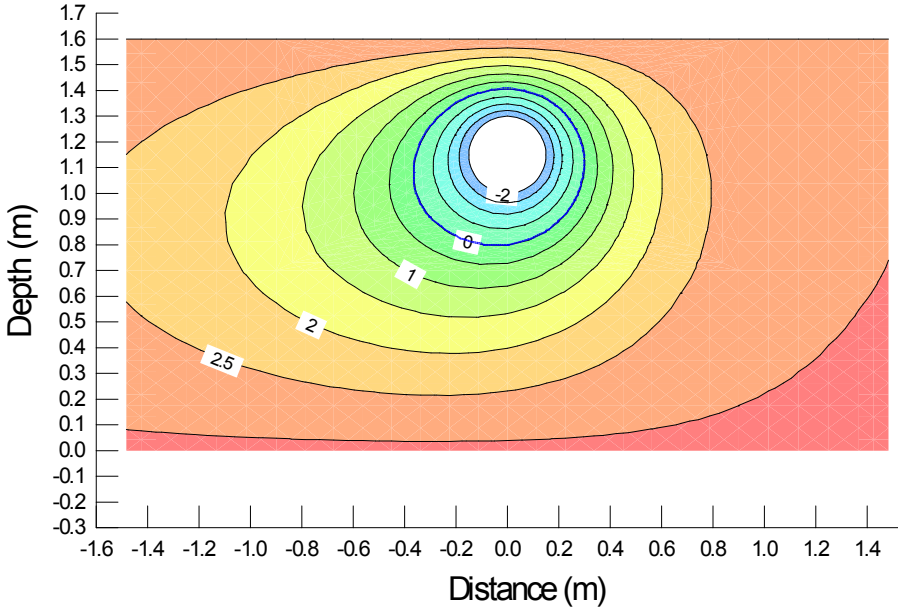


Figure 1-7 Freezing around a chilled pipeline

Convective cooling of surfaces

TEMP/W has a special boundary condition that considers the heat removal from the model due to a fluid flowing past the edge of the model. Consider Figure 1-8 which shows a model set up to compute the cooling of a fabricated 10 mm thick Pyrex tile for two cases: still air and forced air. In this case the tile cools from 140°C to about 25°C; therefore, phase change is not an issue and the model is quite easy to set up and solve. TEMP/W can compute the actual removal of heat from the model surface based on the specified fluid temperature and the continually changing model boundary temperature.

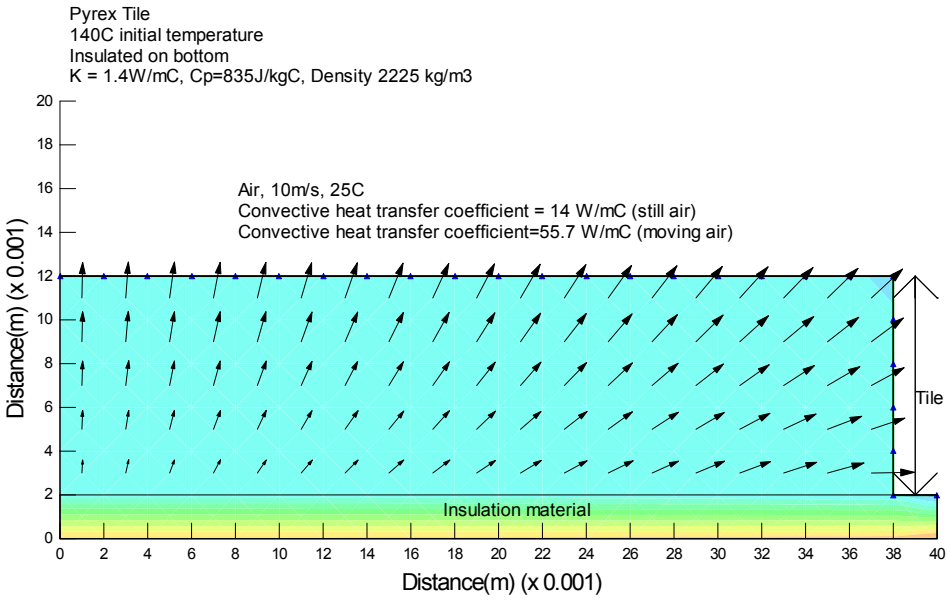


Figure 1-8 Heat flow vectors during air cooling of a fabricated Pyrex tile

1.2 About this book

Modeling the flow of heat through soil (with or without water changing phases) with a numerical solution 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 fact, the correct boundary condition can sometimes be part of the solution, as is the case for artificial ground freezing, where the amount of heat removed from the ground through the chilled pipe depends on the difference between the pipe fluid temperature and the ground temperature itself.

Furthermore, when a soil becomes partially frozen, the coefficient of thermal conductivity becomes a function of the negative temperature in the soil. It is the temperature that is the primary unknown and needs to be determined. Because of this, an iterative numerical technique is required to match the computed temperature and the material property, making the solution highly non-linear. These complexities make it necessary to use some form of numerical analysis to analyze thermal problems for all but the simplest cases. A common approach is to use finite element formulations, and TEMP/W, the subject of this book, is an example of a numerical software tool.

While part of this document is about using TEMP/W to do thermal 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 TEMP/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 physical fundamentals. Aspects such as discretization of a finite element mesh and applying boundary conditions to the problem 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 the general guidelines of how to conduct effective numerical modeling. In fact, Chapter 2 is devoted exclusively to discussions on this topic. The general principles discussed there apply to all numerical modeling, including thermal analysis. In this chapter, however, the discussion pertains to seepage analyses because it is more intuitive to most people. It is easier to mentally understand the flow of water than heat.

Broadly speaking, there are three main parts to a finite element analysis. The first is Discretization: dividing the domain into small areas called elements. The second part is specifying and assigning material properties. The third is specifying and applying boundary conditions. Separate chapters have been devoted to each of these three key components within this document.

Frozen / unfrozen numerical modeling 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 a chapter titled Numerical Issues.

Two chapters have been dedicated to presenting and discussing illustrative examples. One chapter deals with examples where geotechnical solutions are obtained by integrating more than one type of analysis, and the other chapter presents and describe how a series of different geotechnical problems can be solved.

A full chapter is dedicated to theoretical issues associated with the finite element solution of the partial differential heat flow equation for frozen and unfrozen soils. Additional finite element numerical details regarding interpolating functions and infinite elements are given in Appendix A.

The chapter entitled “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, TEMP/W conventions and data results.

In general, this book is not a “how to use TEMP/W” manual. This is a book about how to model. It is a book about how to engineer thermal problems using a powerful calculator; TEMP/W. Details of how to use various program commands and features are given in the on-line 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 GeoStudio 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, you do 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 GeoStudio.

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 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. TEMP/W is a numerical model that can mathematically simulate the real, physical process of heat flowing through a fully or partially saturated, frozen or thawed particulate or solid medium. Numerical modeling is purely mathematically and in this sense is very different than scaled physical modeling in the laboratory or full-scaled field modeling.

Newman and Maishman (2000) presented details of an artificial ground freezing project involving freezing a barrier wall around three sides of an underground uranium ore body in northern Canada. The freezing was necessary in order to seal off potential excessive inflow of high pressure water from the south, north and west sides of the ore body as illustrated in Figure 2-1. This figure shows the southern row of freeze pipes that were installed at 2m intervals around the region. A freeze pipe operates circulates

chilled brine which in turn removes heat from the ground and eventually freezes all water between the pipes. This results in a frozen barrier wall.

Figure 2-2 shows a plan view of the three freeze walls through different types of ground after about six months of active freezing. It is interesting to note the different rates of freeze wall thickness that develop in the different materials. While different materials have different thermal conductivity values, other factors come into play such as the ground water content by volume in each material and the initial ground temperature.

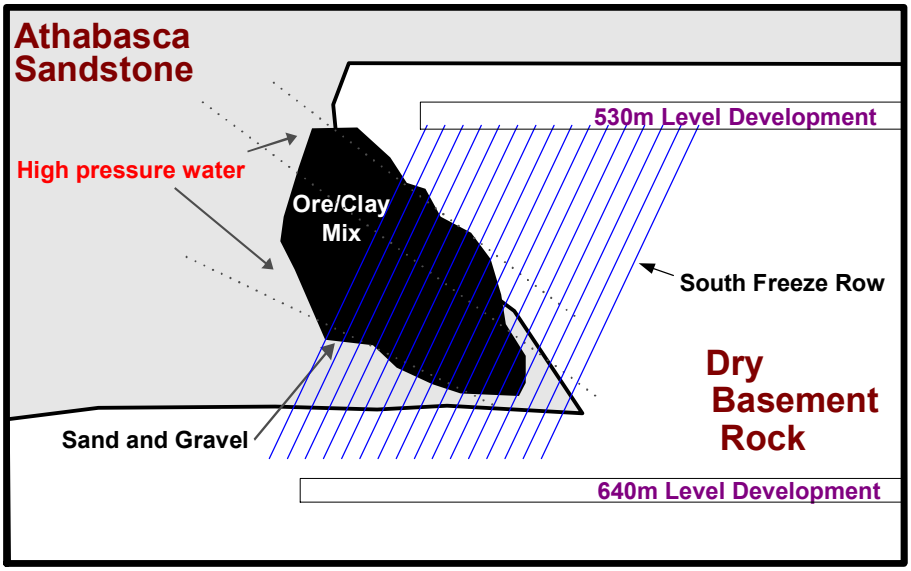


Figure 2-1 End view of freeze wall around underground ore body

Figure 2-3 shows a comparison of predicted and observed ground temperature decay over time for various ground types, and at various distances offset from the row of freeze pipes. TEMP/W was able to accommodate the wide range of ground types, water contents, thermal properties and initial in-situ ground temperatures in its analysis. In particular, once it was realized that the initial ground temperature in the ore rock itself was much higher than the surrounding ground temperature (+22 C versus +10 C), TEMP/W was able to model the actual time it would take to freeze this ground and, therefore, provide a more reasonable estimate of when production could begin once development of a safe water sealing barrier around the mining region was established.

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.

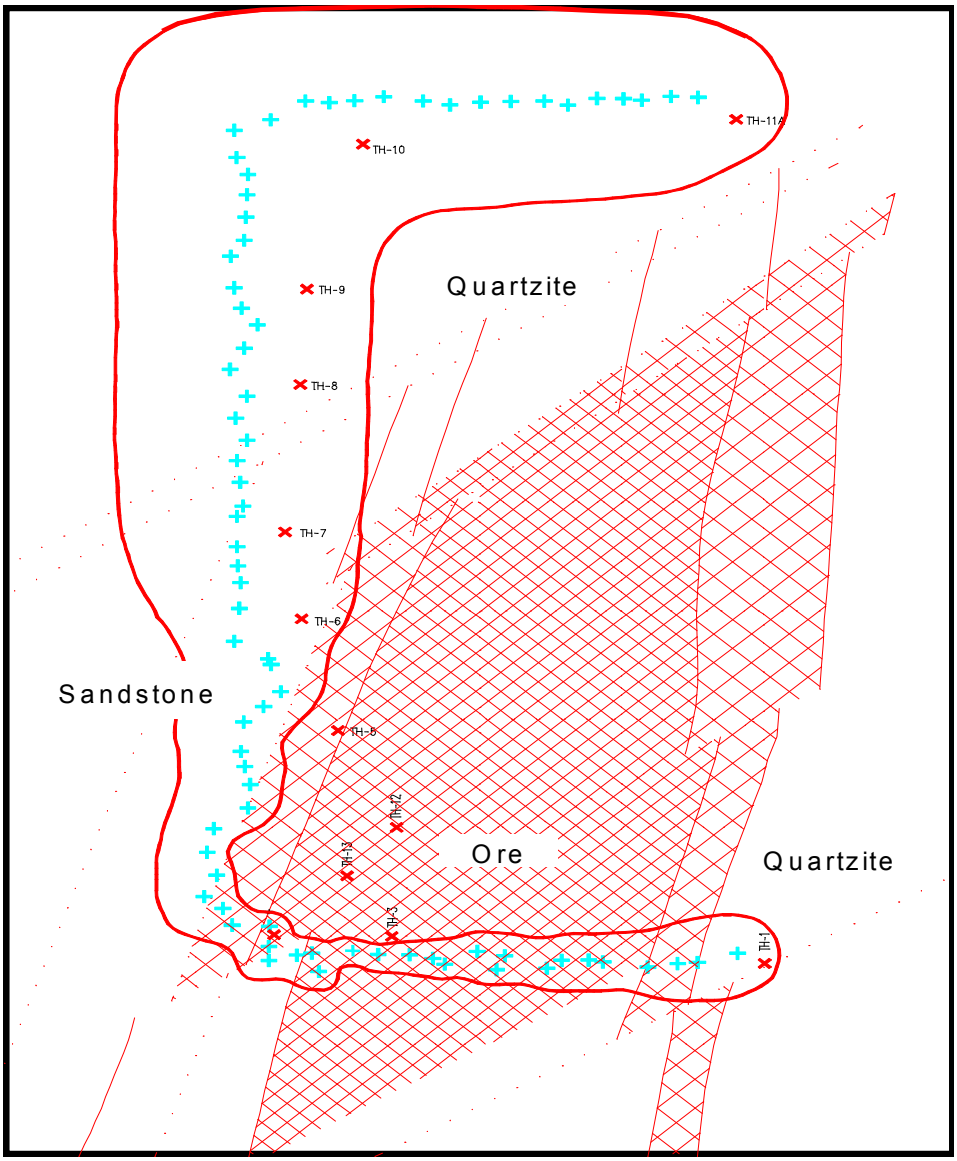


Figure 2-2 Plan view of freeze wall growth around ore body after 6 months of active freezing (south freeze wall is at bottom of figure, crosses mark surveyed position of freeze pipes at modeled elevation)

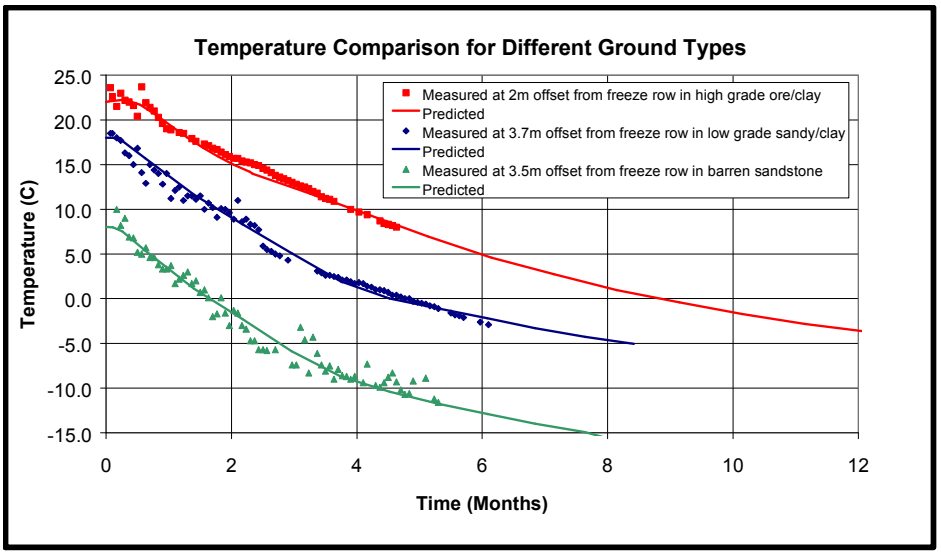


Figure 2-3 TEMP/W analysis versus measured data

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 thermal heat flow there may also be pore-water pressure 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 TEMP/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. TEMP/W is formulated only for heat flow that follows a Darcy type conduction law with a modification for convective heat flow when combined with SEEP/W.

The important point to remember is that the mathematical formulations implemented in software like GeoStudio 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 2-4. This has come to be known as the Burland triangle (Burland, 1987; Burland, 1996).

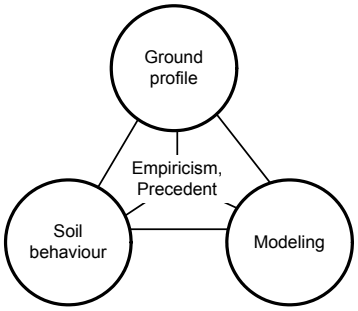


Figure 2-4 The Burland triangle (after Burland 1996)

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 2-5.

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.

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.

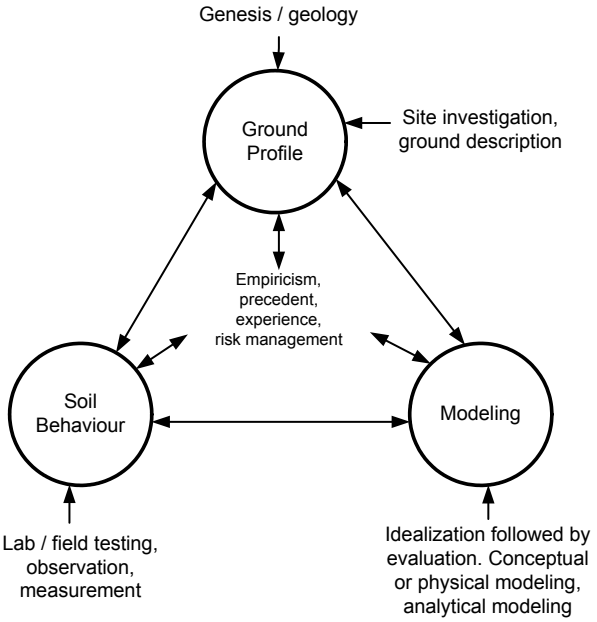


Figure 2-5 The enhanced Burland triangle (after Anon. 1999)

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 more 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 2-6 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 2-6. 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 2-6 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.

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.

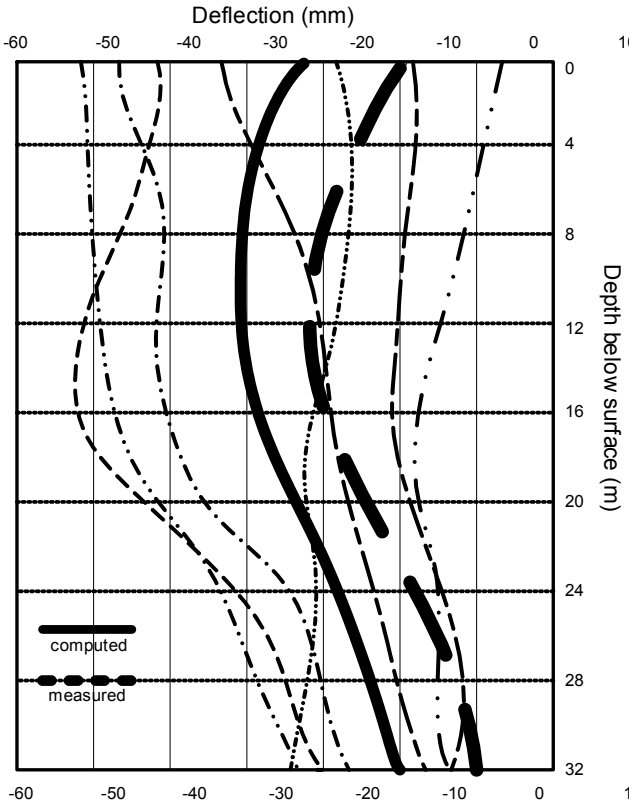


Figure 2-6 Comparison of predicted and measured lateral movements of a shoring wall (after Carter et al, 2000)

Compare alternatives in design or modeling methodology

Numerical modeling is useful for comparing alternatives. Keeping everything 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 frost penetration on a roadway over a winter. There are two types of boundary conditions that could be applied at the ground surface: a detailed climate analysis or a simplified analysis that assumes ground temperature is equal to average monthly air temperature (which also assumes the road is kept clear of any insulating snow cover). The detailed climate analysis considers the daily warming and cooling of the air while setting the ground temperature to be equal to the average daily air temperature ignores any diurnal variation or any affect on ground temperature from the ground heat flux itself or snow cover etc. A simple one-dimensional model can be set up as an initial test to just see how big the impact of both boundary options is. It may be significant, it may not. Regardless, it's MUCH

faster and easier to test these alternatives in a simple model before you build the detailed two-dimensional model.

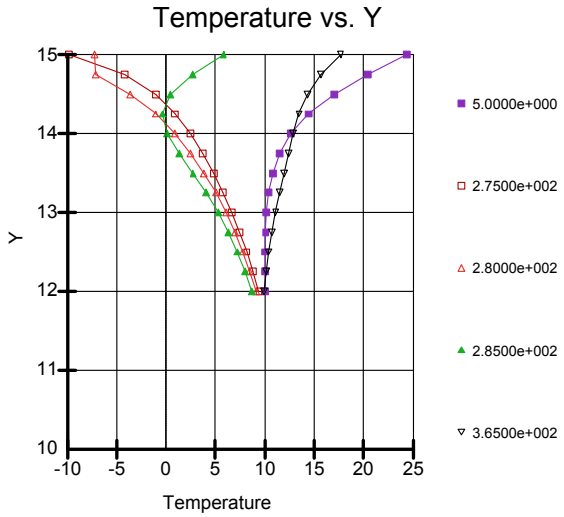


Figure 2-7 Near surface temperatures over time computed with daily detailed data

The actual computed values are not of significance in the context of this discussion. What is significant is that ignoring detail in these cases produces ground temperatures that get colder in winter and do not get warmer in summer, based on energy balances at the surface on bare soil or soil with a snow pack. This is an example how a simple model set up in TEMP/W can be used to quickly compare alternatives prior to a more detailed analysis. Another thing we may want to check at this stage of our modeling is how sensitive the results are to slight variations in thermal conductivity.

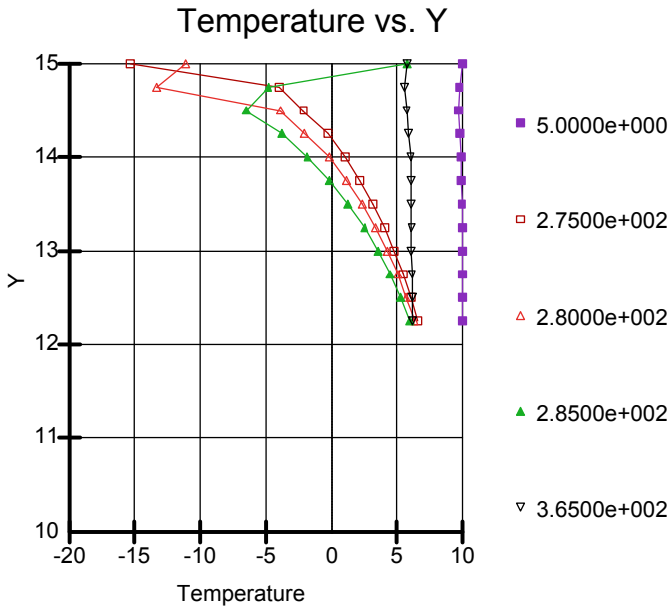


Figure 2-8 Near surface temperature profile using ground temperature set to average daily air temperature

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 2-9. A similar type plot may be generated based on TEMP/W simulations to determine the optimum insulation thickness, thermal conductivity or water content for a design that prevents freezing temperatures from developing beneath a slab foundation, for example.

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.

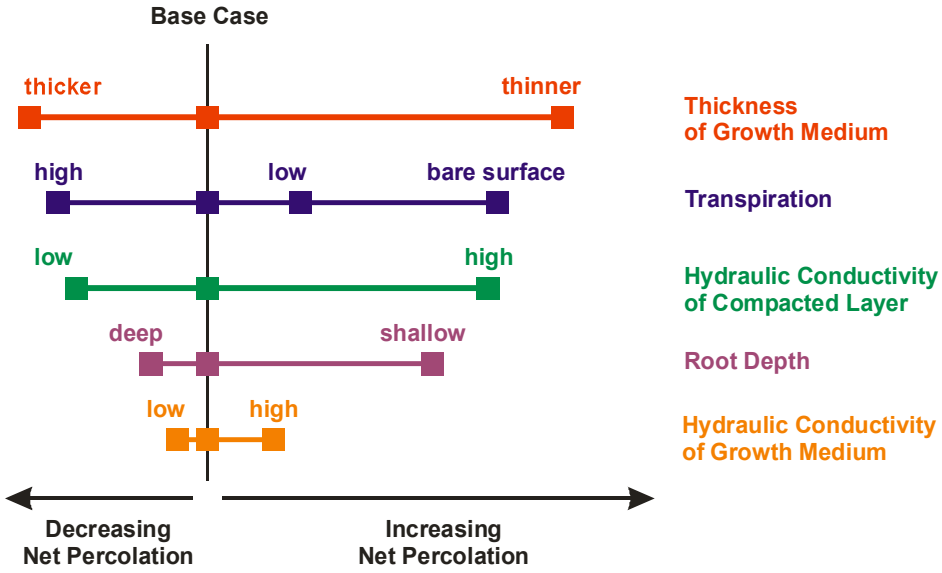


Figure 2-9 Example of a tornado plot (O'Kane, 2004)

Discover and 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 2-10. 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 down slope 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, in the SEEP/W Engineering book describes in detail the nature of the

hydraulic conductivity (or permeability) functions. For this example, relative conductivity functions such as those presented in Figure 2-11 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 the SEEP/W book. For this example, accept that at high suctions the coarse material is less conductive than the fine material.

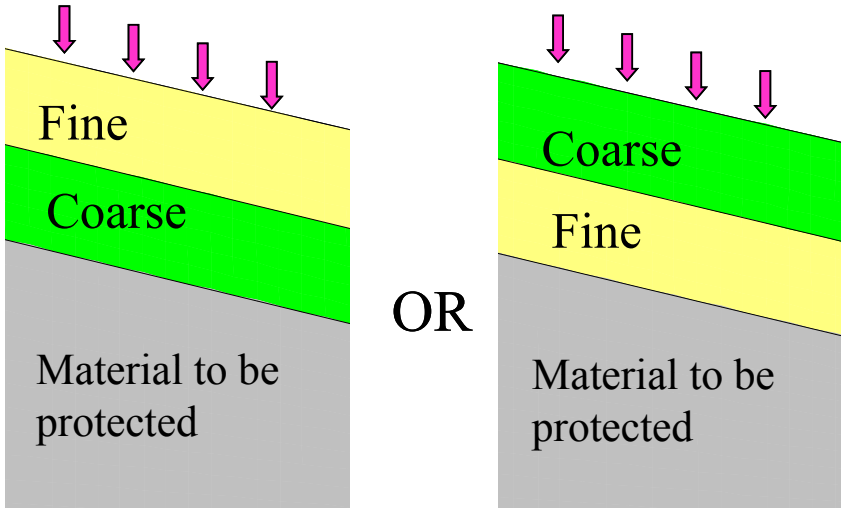


Figure 2-10 Two possible earth cover configurations

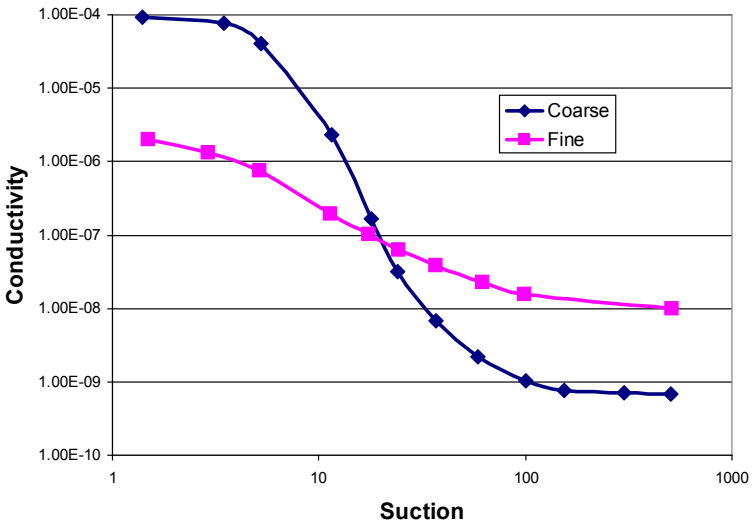


Figure 2-11 Hydraulic conductivity functions

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 2-12. 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 2-13. The design of fine soil over coarse soil may work, but only in arid environments. The occasional cloudburst 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 2-14. 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.

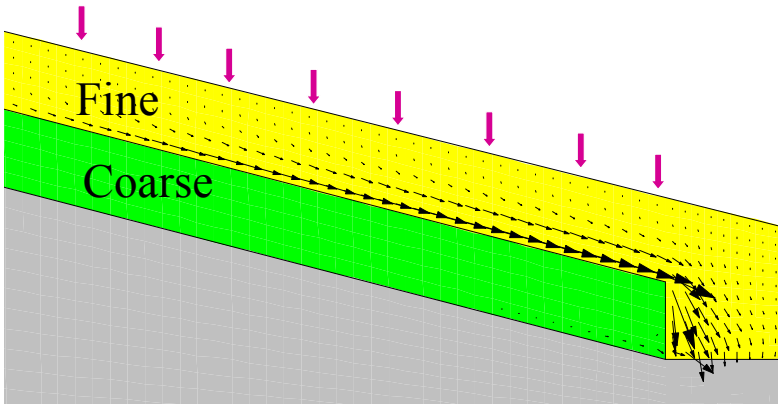


Figure 2-12 Flow diversion under low infiltration

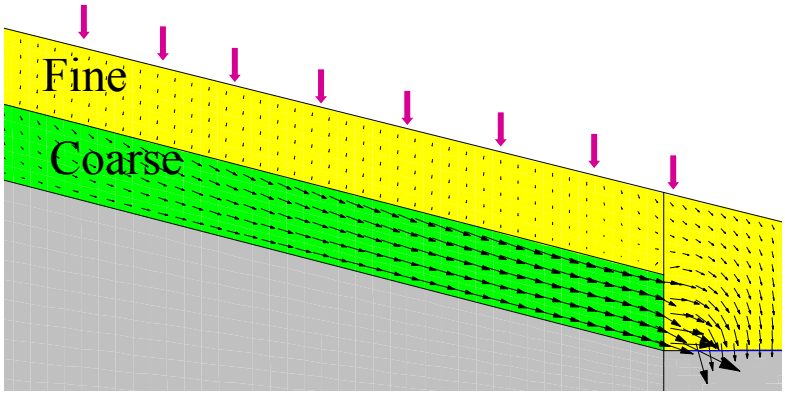


Figure 2-13 Flow diversion under high infiltration

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.

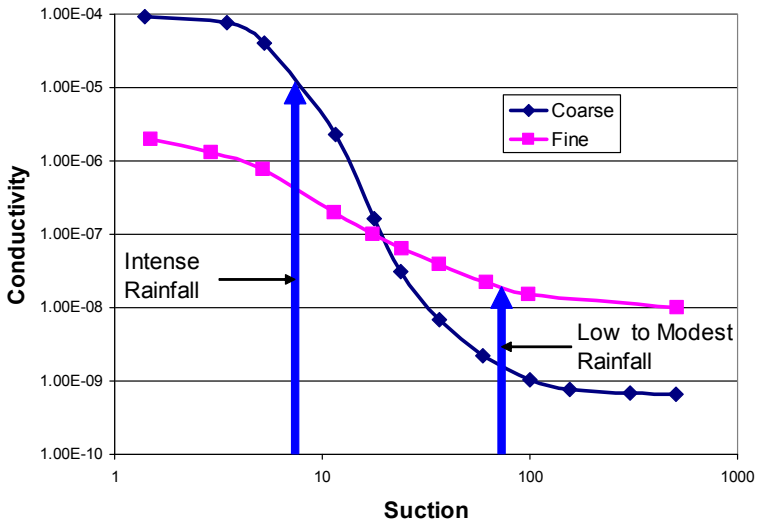


Figure 2-14 Conductivities under low and intense infiltration

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. If there is no resemblance between what is expected and what is computed, 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 thermal distribution, or is the quantity of heat added or removed from the system? If your main objective is to determine the temperature distribution, there is no need to spend a lot of time on establishing the thermal conductivity; any reasonable estimate of conductivity is adequate. If on the other hand your main objective is to estimate heat flow quantities, then a greater effort is needed in determining the thermal 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, such 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 2-15 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 2-16. 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.

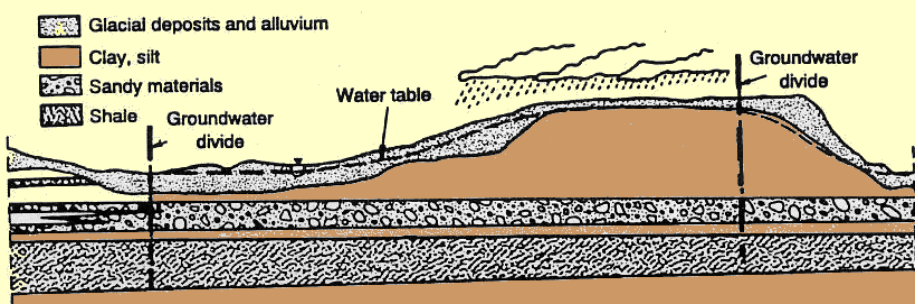


Figure 2-15 Example of a stratigraphic cross section (from National Research Report 1990)

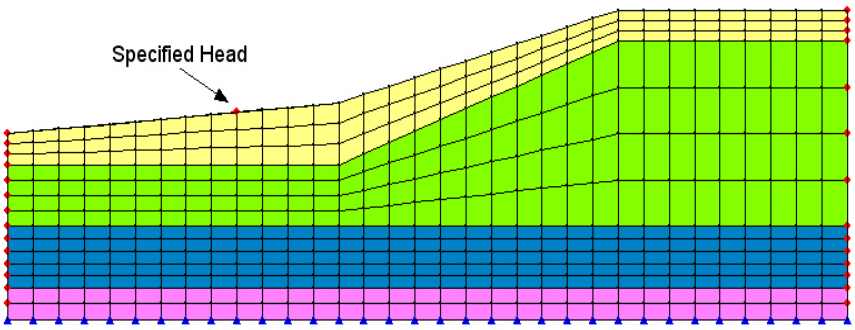


Figure 2-16 Finite element model of stratigraphic section

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.

Another example of keeping the geometry simple relates to modeling actual freeze pipe geometry, versus having a single freeze pipe be represented by a single node. Consider the illustrations in Figure 2-17, where on the left the actual pipe geometry is built into the mesh, and on the right the freeze pipe is represented by a single nodal boundary condition. Depending on the boundary condition applied in the analysis, the mesh on the right will give equally reliable results and it is much simpler to set up. This is especially true if you need to model many freeze pipes in a complex pattern to assess the freezing that develops over the region.

When a small geometry freeze pipe is modeled (e.g. several inch diameter), TEMP/W has the ability to apply a special convective heat transfer boundary condition that includes the actual pipe surface area in the boundary value. With this option, the actual heat removed from the ground will be almost identical in both models and there is no need to spend time building very small detail into the actual finite element mesh.

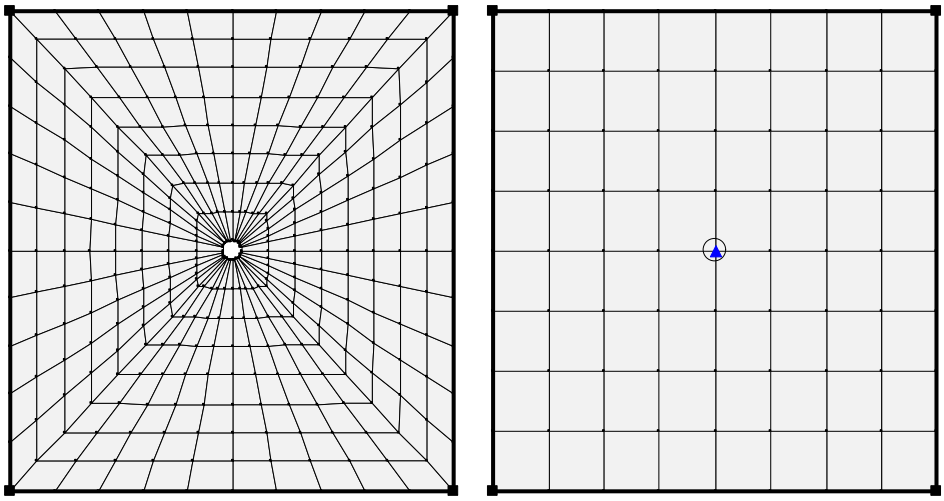


Figure 2-17 Detailed pipe geometry versus simplified geometry

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 build 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 the process migrates 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 that 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 2D case as shown in Figure 2-18. The total temperature difference is 1 degree and the thermal conductivity is 1 W/mC. The gradient under steady-state conditions is the temperature difference divided by the length, and is this case the gradient is 0.1. The resulting total heat flow through the system is the cross-sectional area multiplied by the gradient, which should be 0.3 W. The flux section that goes through the entire section confirms this result. There are flux sections through Elements 16 & 18. The flow through each element is 0.1 W, 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 temperature is specified, TEMP/W computes the corresponding nodal flux. In TEMP/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.

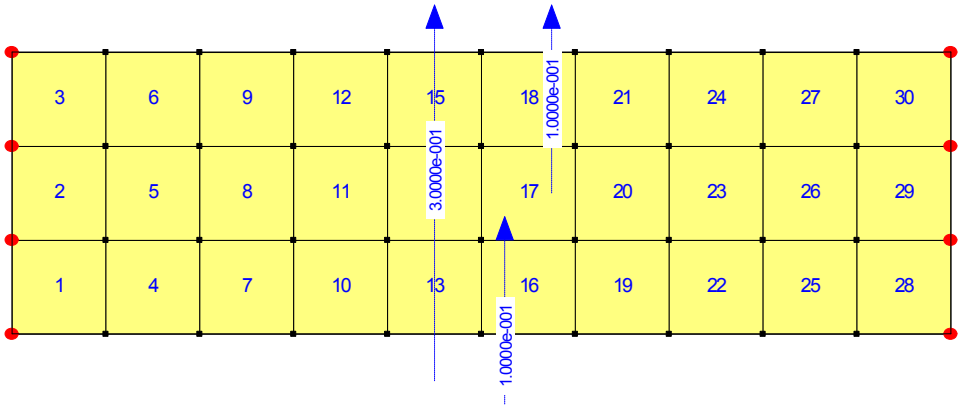


Figure 2-18 Horizontal heat flow through three-element section

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 that 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.

Including unnecessary features and trying to model adjacent materials with extreme contrasts in material properties create numerical difficulties. The hydraulic conductivity difference between the core and shell of a dam, for example, 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 in situ 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 TEMP/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

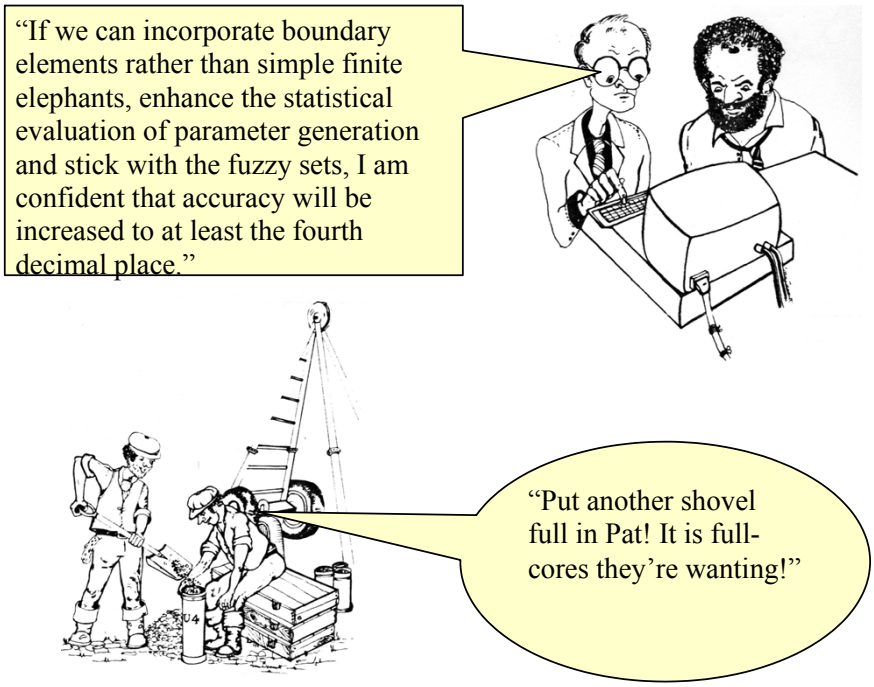
same as what you thought was specified defining the model. Is the intended property function being applied to the correct soil? Or, are the initial conditions as you assumed?

TEMP/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.



** Origins of this figure are unknown at time of printing

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.

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. Nether 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 advantage 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 GeoStudio, 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, GeoStudio, 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. 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.

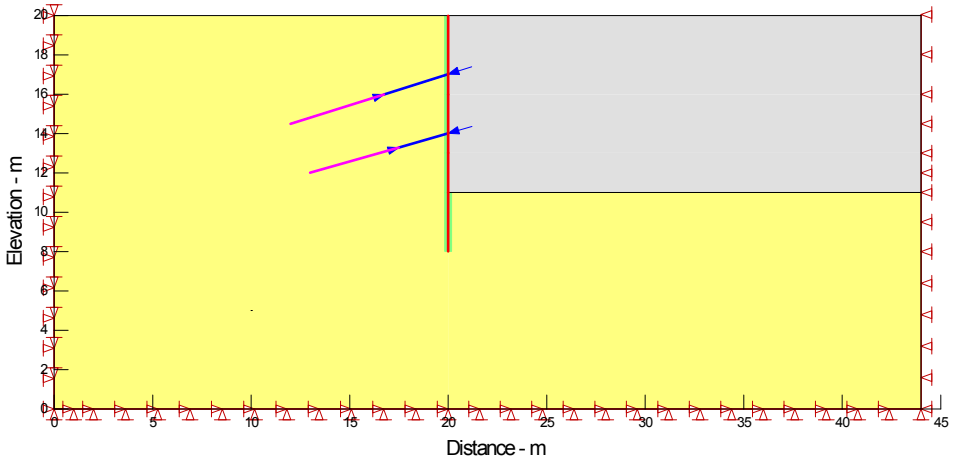


Figure 3-1 Fully defined geometry for soil excavation model

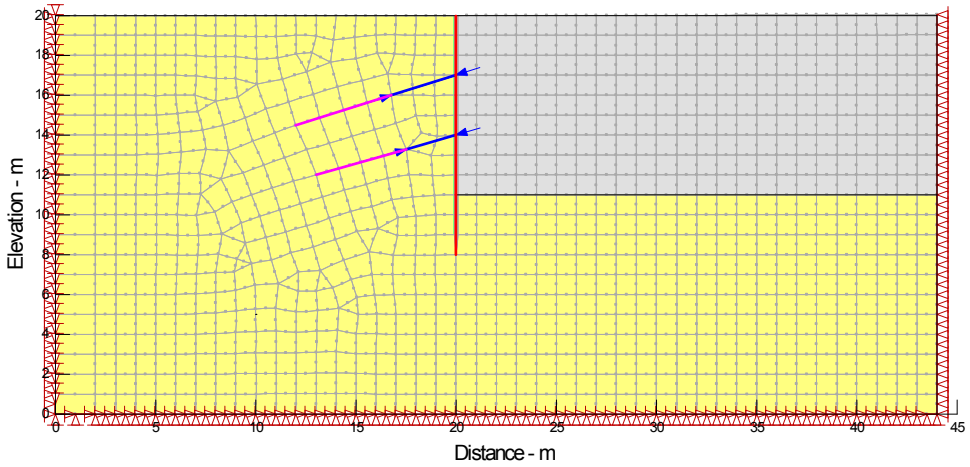


Figure 3-2 Default element discretization for model using 1m global element size constraint

3.2 Geometry Objects in GeoStudio

In GeoStudio, 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.

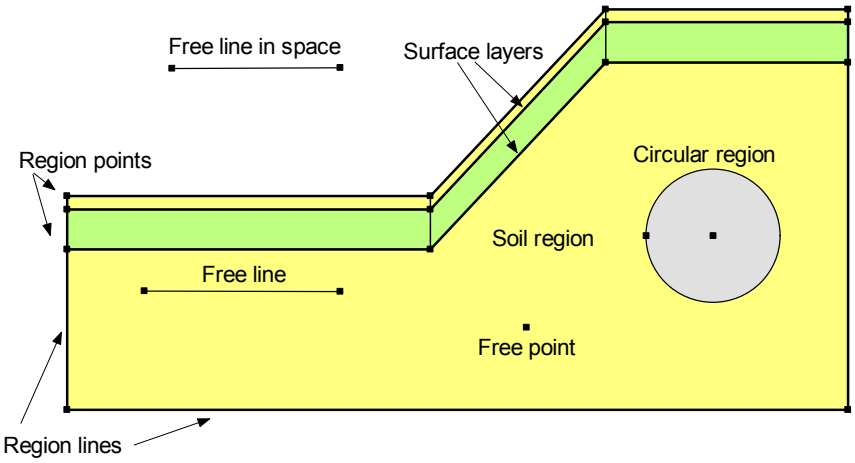


Figure 3-3 Available geometry objects

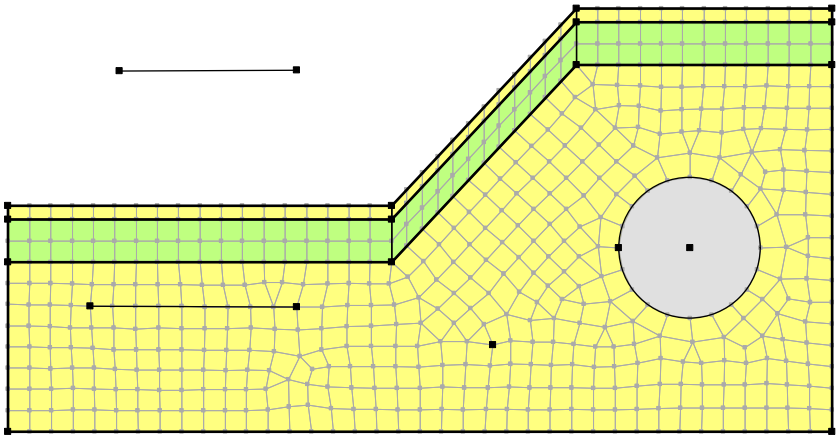


Figure 3-4 Mesh pattern for example model

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.

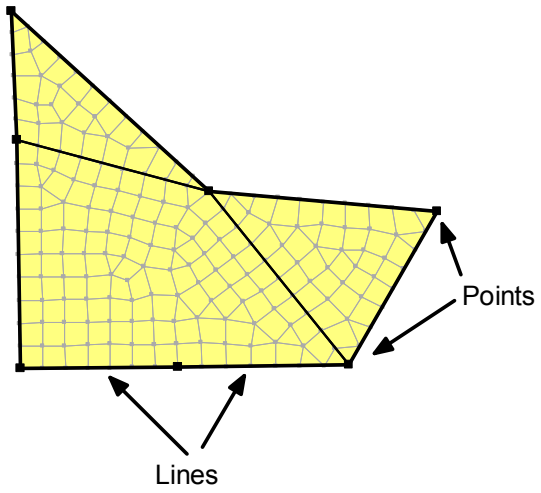


Figure 3-5 Illustration of a region's lines and points

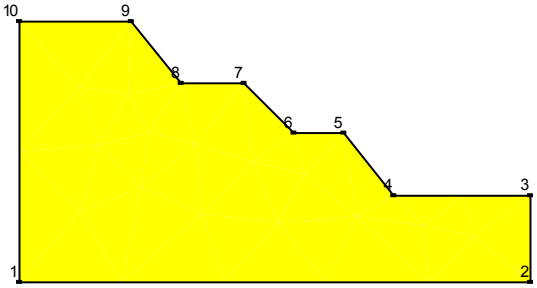


Figure 3-6 A multi-side polygonal region

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.

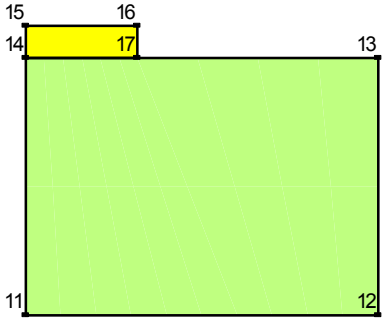


Figure 3-7 Regions of different size

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. 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…

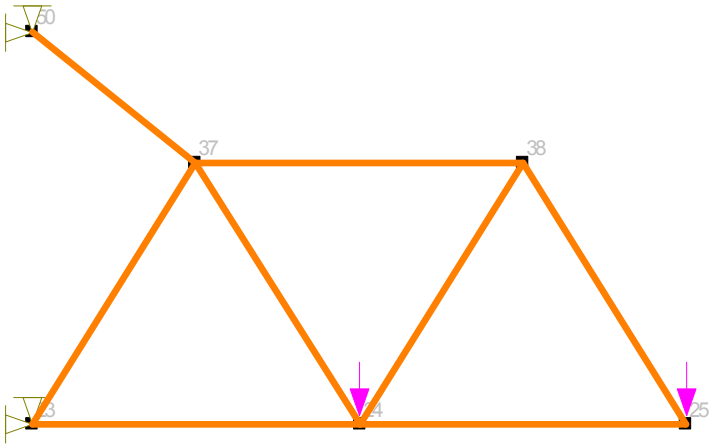


Figure 3-8 Truss loading model

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.

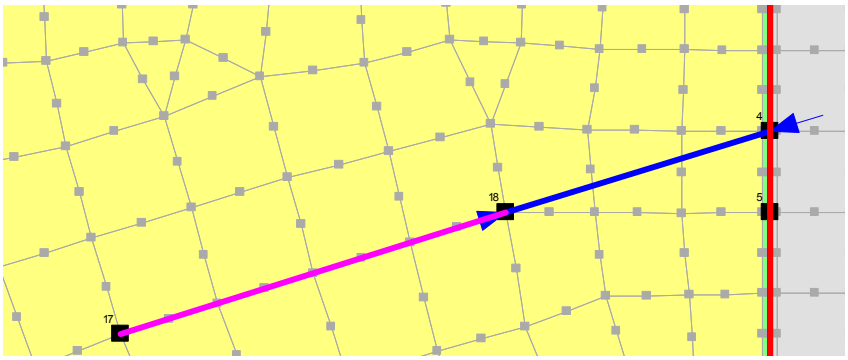


Figure 3-9 Anchors on lines (line is partially meshed)

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.

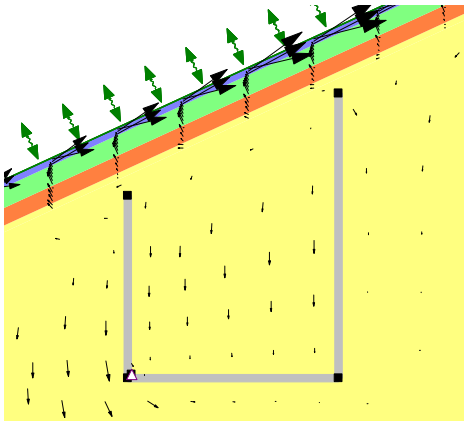


Figure 3-10 Lines with material model "none" (no flow) assigned to simulate lysimeter collection basin in waste beneath an engineered soil cover

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.

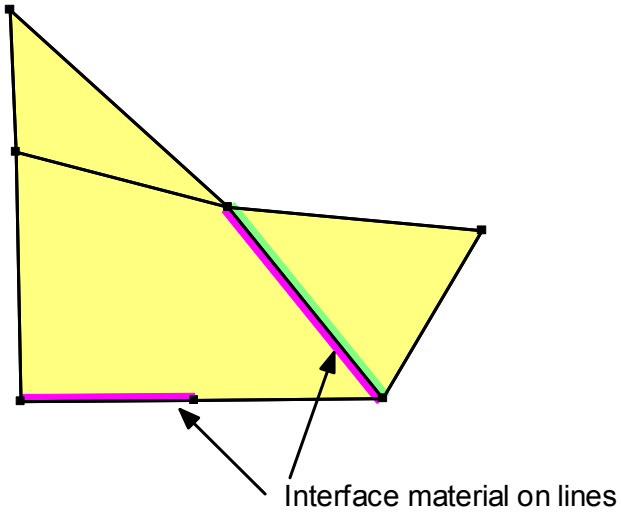


Figure 3-11 Illustration of "interface elements" on geometry lines

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.

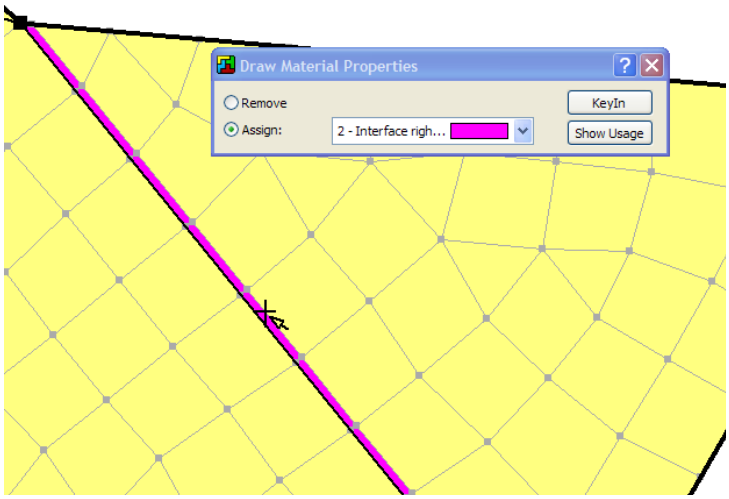


Figure 3-12 Using the Draw Line Material command to assign an interface model to a line

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.

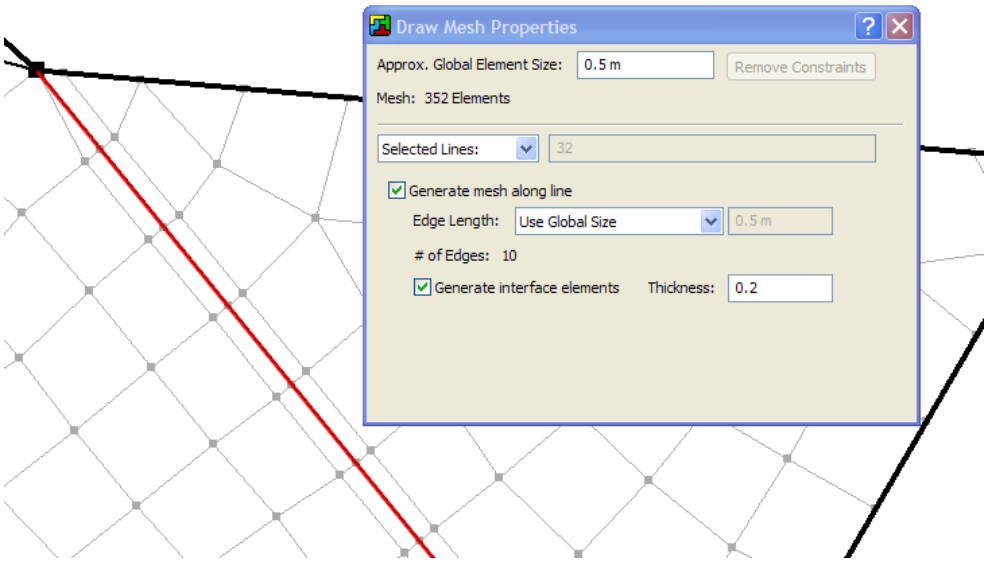


Figure 3-13 Using Draw Mesh Properties to create interface elements on selected line

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.

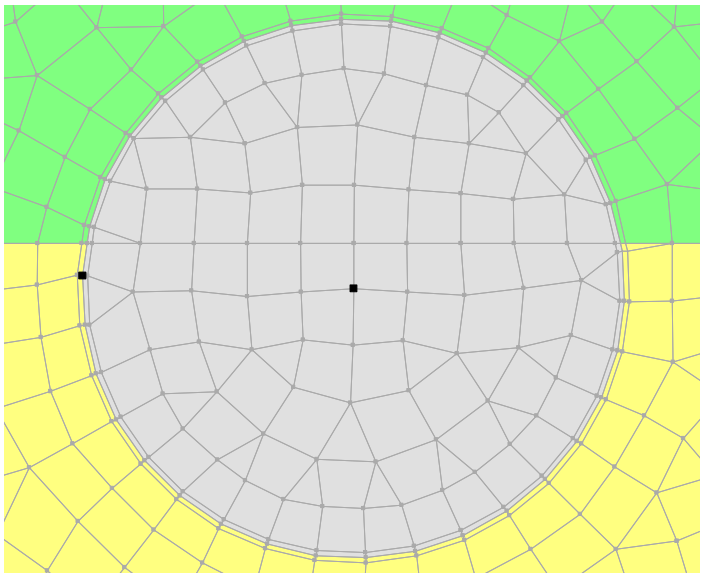


Figure 3-14 Circular region with defined center point, radius point and interface elements

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.

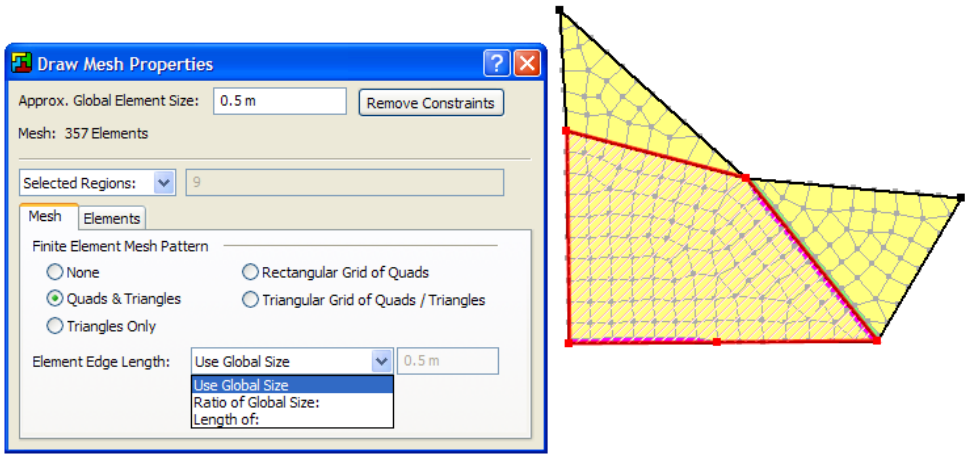


Figure 3-15 Draw Mesh Properties options

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.

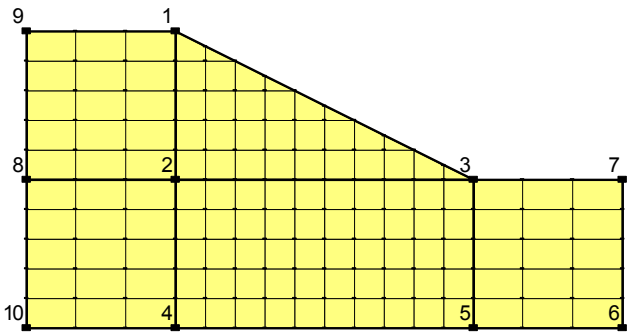


Figure 3-16 Structured mesh

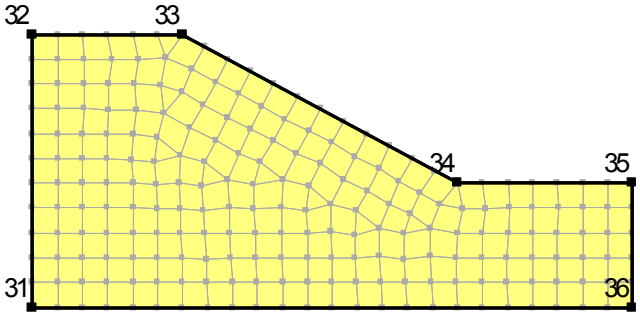


Figure 3-17 Mixed quad and triangle unstructured mesh

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.

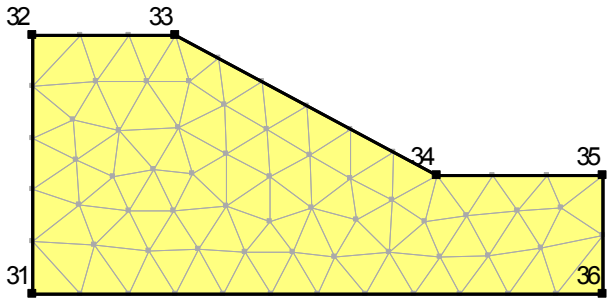


Figure 3-18 Unstructured triangular mesh

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.

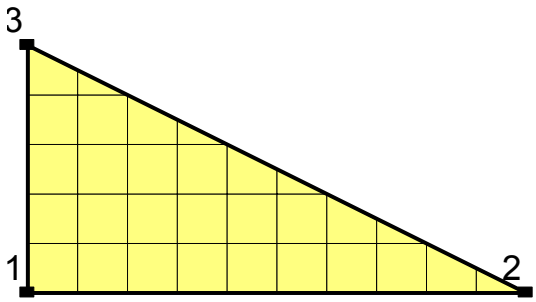


Figure 3-19 Triangular grid region

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.

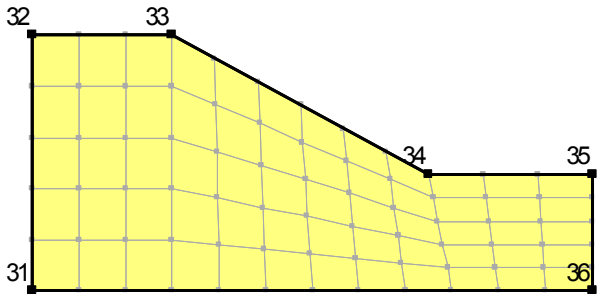


Figure 3-20 Rectangular grid of quad elements

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.

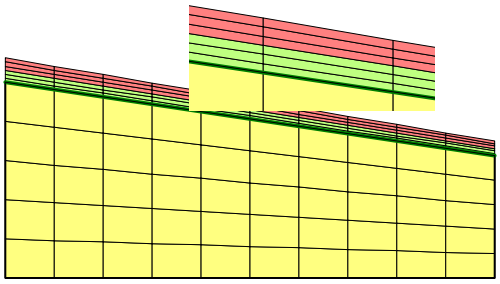


Figure 3-21 Illustration of a surface layer mesh

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.

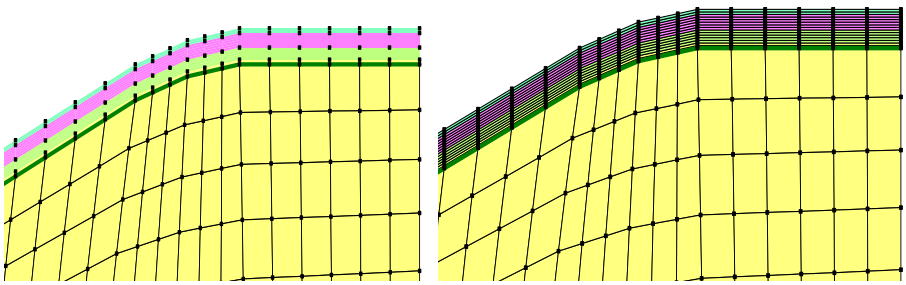


Figure 3-22 Surface region mesh with details off (left) and on (right)

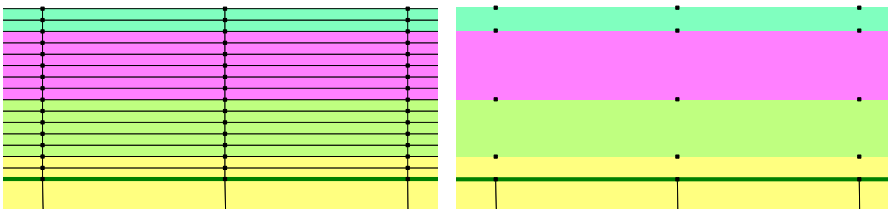


Figure 3-23 Close up of surface details on and off (note inter-layer nodes still visible in figure on right)

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.

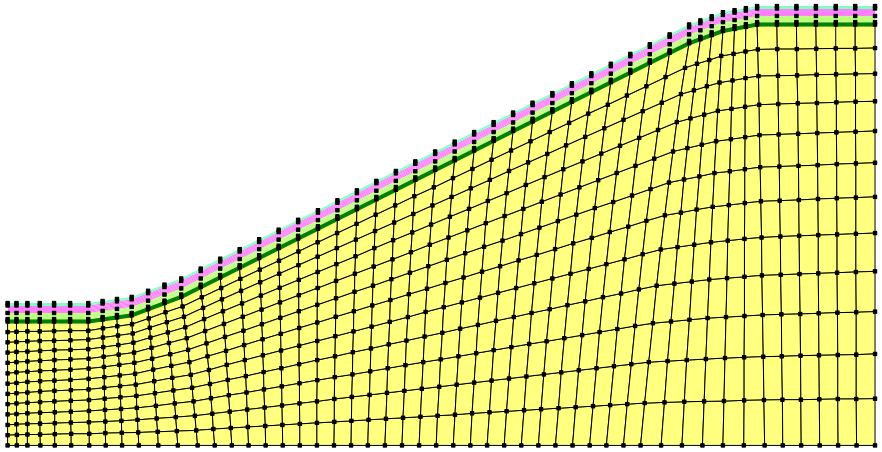


Figure 3-24 Mesh showing rounded surface slope breaks

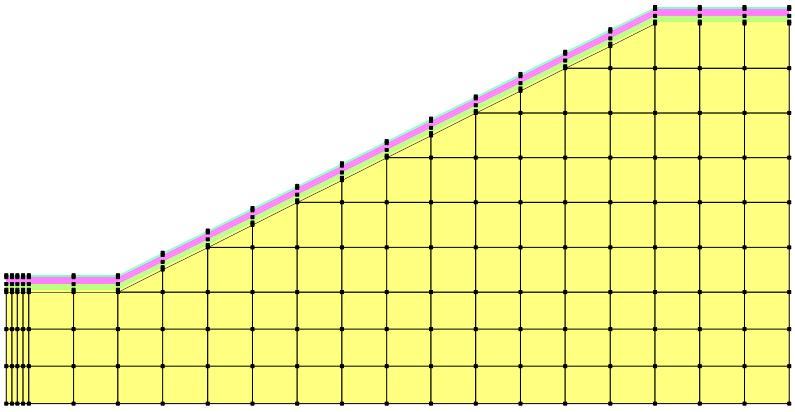


Figure 3-25 Mesh showing angular surface slope breaks

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.

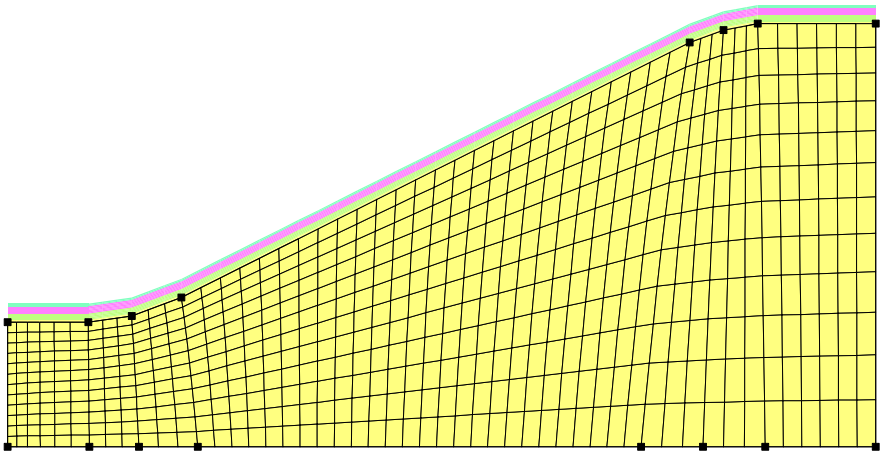


Figure 3-26 Region mesh with region corner points viewed and surface details not viewed

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.

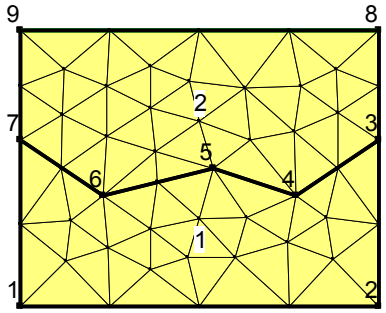


Figure 3-27 Regions joined along jagged line

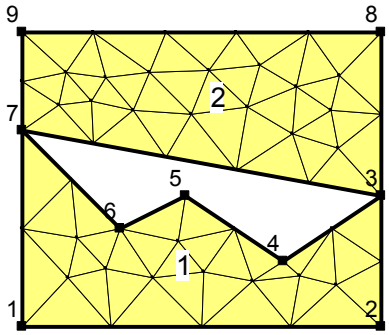


Figure 3-28 Adjoining regions with an open space

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 (CTRAN/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. In a thermal analysis, the primary variable is temperature. 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., 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.

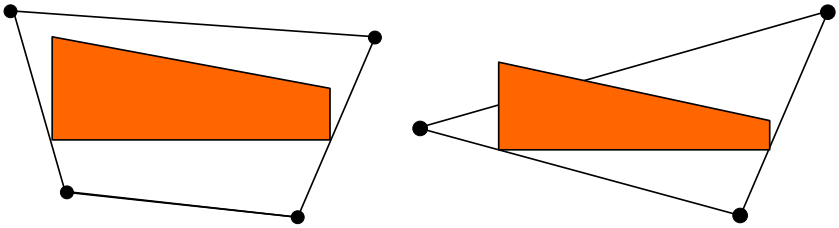


Figure 3-29 Primary field variable distribution in first-order elements

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.

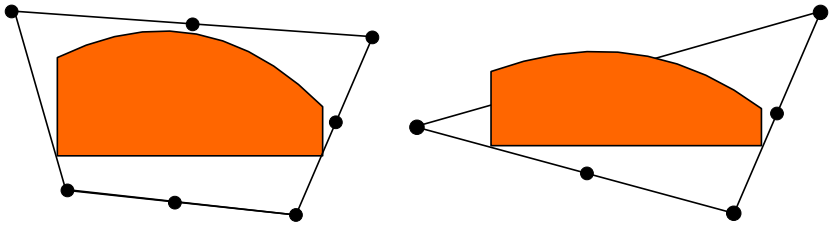


Figure 3-30 Primary field variable distribution in higher-order elements

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 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.

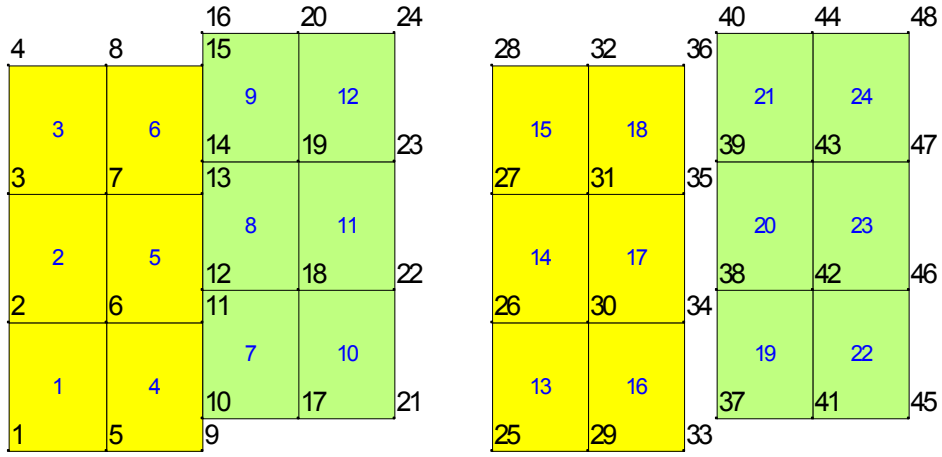


Figure 3-31 Disconnected elements – lack of compatibility

Figure 3-32 Connected elements – compatibility satisfied

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 vise 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.

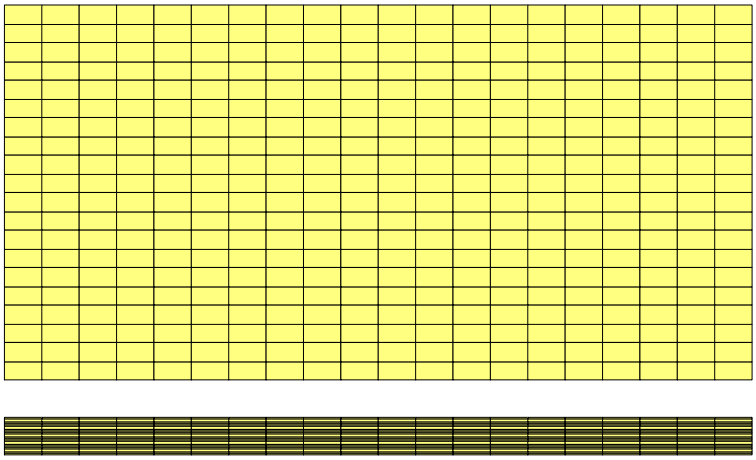


Figure 3-34 Mesh at an exaggerated scale (upper) and at a true scale (lower)

Mesh purpose

The "How To" modeling chapter notes that in good numerical modeling practice it is important to form a mental imagine 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 though 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 though 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 though the core.

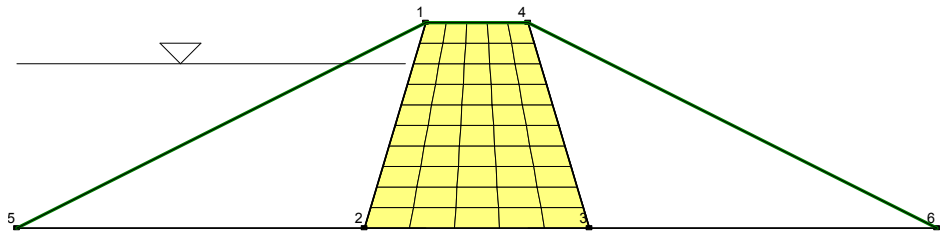


Figure 3-35 Modeling core of zoned dam

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

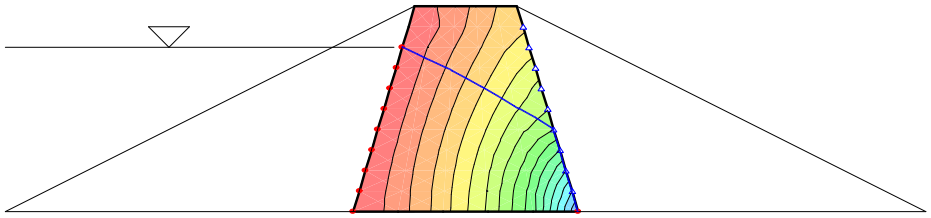


Figure 3-36 Equipotential lines in core of dam

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.

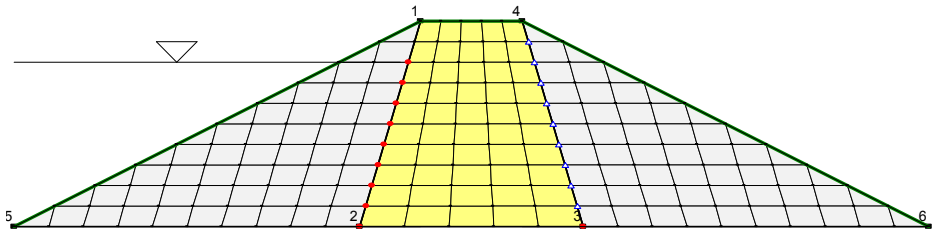


Figure 3-37 Mesh with null elements in shells of dam

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.

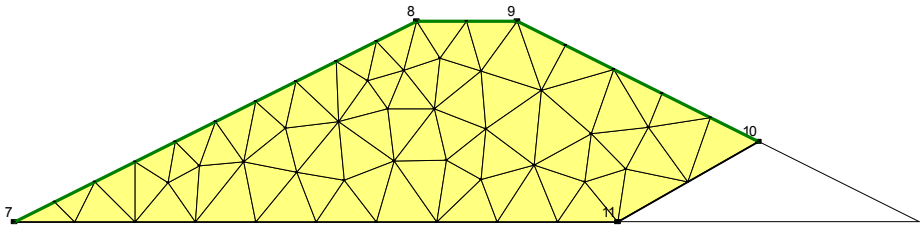


Figure 3-38 Dam with a toe drain

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 thermal properties that are required in the solution of the TEMP/W 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. It is meant 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 ok to guess at a function and when must you very carefully define one? This chapter will address these issues.

4.1 Material models in TEMP/W

There are five different material models to choose from in TEMP/W. A summary of these models and the required soil properties are given below, and discussion of the individual parameters and function are provided in the next section.

1. None (used to remove part of the domain in an analysis)
2. Full thermal model
   - Thermal conductivity function vs temperature, ratio and direction
   - Unfrozen water content function
   - Frozen and unfrozen volumetric heat capacities
   - Insitu volumetric water content
3. Simplified thermal model
   - Frozen and unfrozen volumetric heat capacities
   - Frozen and unfrozen thermal conductivity
   - Insitu volumetric water content
4. Coupled convective thermal model (with SEEP/W)
   - Thermal conductivity function verses water content, ratio and direction
   - Unfrozen water content function
   - Volumetric heat capacity verses water content function
5. Interface model
   - Thermal conductivity

The None model is useful to assign a soil region if you do not want that region to be included in the current analysis but you may want it to become active some time later in a staged construction sequence analyses.

The Full thermal model is the traditional TEMP/W model and requires that functions be specified for thermal conductivity and unfrozen water contents. These functions are dependent on temperature and are therefore only valid for constant water content. The water may turn to ice, but the overall amount of water/ice is fixed throughout the analysis.

The Simplified thermal model is intended for problems in which the latent heat of phase change is not that important to the freezing process. You can still specify different thermal properties for the frozen and unfrozen states, but phase change is assumed to occur at the phase change temperature instead of occurring over a range of temperatures, which is manifest in the unfrozen water content function used in the Full thermal model.

The Coupled convective thermal model requires that the thermal functions are dependent on water content and not just temperature. The water content is allowed to change during the analysis but it must be coupled with SEEP/W for this to happen. This model is also required for AIR/W coupling with TEMP/W.

Finally, the Interface model is used for specifying thin insulation or no heat flow barriers within the soil. These are discussed next.

4.2 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 thermal analysis, you may want to use an interface model to simulate a thin layer of insulation. When you assign an interface model to a line you must give it a thermal conductivity value. It is assumed that for this type of material the normal and tangent thermal conductivities are the same and the material is homogeneous and isotropic. In a hydraulic interface element, the tangent and normal conductivity may differ as shown in Figure 4-1. In an AIR/W model, the air conductivity value you specify is also independent of direction.

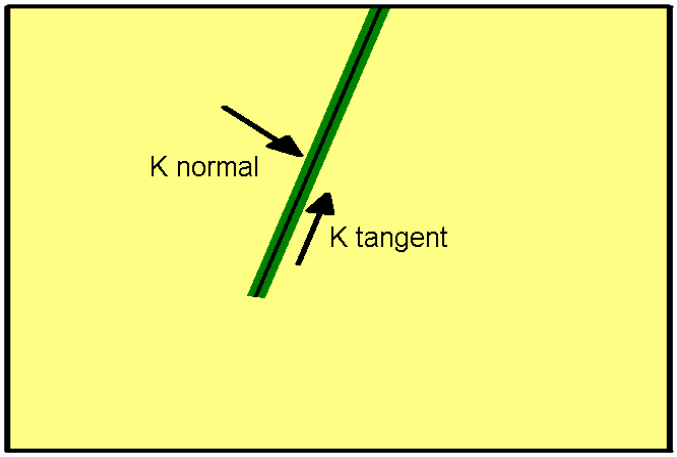


Figure 4-1 Illustration of interface model assigned to a line

4.3 Thermal functions

Unfrozen water content function

Fundamental to the formulation of a general freeze-thaw thermal analysis of soil systems is an understanding of how energy stored in the soil varies as the soil temperature changes. An example of this relationship is shown in Figure 4-2. The function represents the relative energy required for the soil medium to sustain a certain temperature. The steep part of the function in the region of the freeze-thaw front represents the latent heat absorbed or released by a soil due to phase changes of the soil water. The slope of the function away from the freeze-thaw zone represents the volumetric heat capacity of the frozen and unfrozen zones.

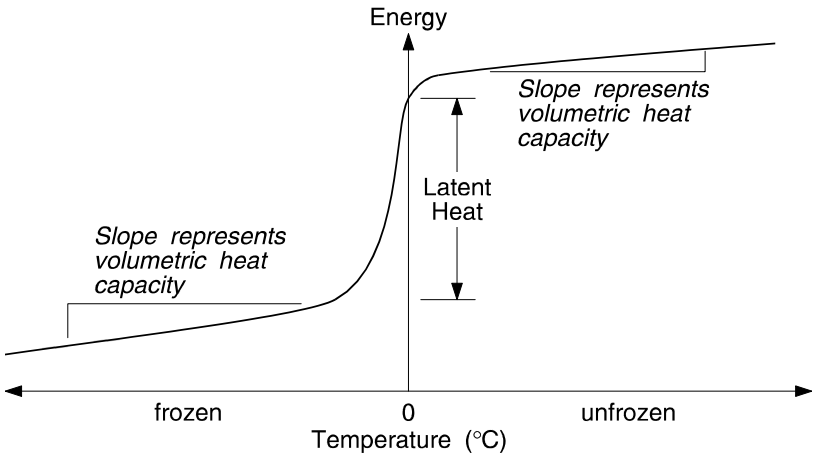


Figure 4-2 Significance of an energy storage function

Unlike the case of a pure water medium, water within a soil changes phase over a temperature range. In other words, not all of the water within the soil experiences a phase change at a single temperature. The percentage of the soil water volume that remains unfrozen at a certain temperature is referred to as the unfrozen water content. Examples of the unfrozen water content function for a variety of soils are illustrated in Figure 4-2. In general, the unfrozen volumetric water content function is very steep for coarse grained soils and shallower for fine grained soils.

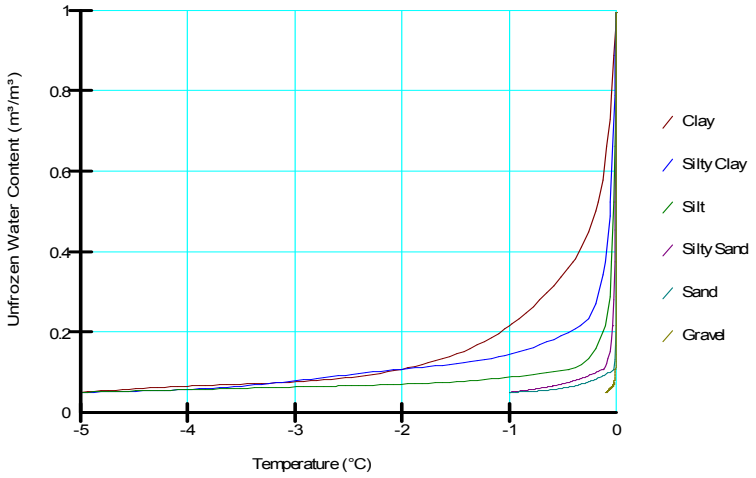


Figure 4-3 Sample unfrozen water content functions

In GeoStudio, the unfrozen water content is normalized to the porosity and therefore varies between 0 and 1.0. Accordingly, a single function could be used for different soils by adjusting the user-input insitu volumetric water content. Both the Full thermal and Coupled convective thermal models require an unfrozen volumetric water content function. The Simplified thermal model does not require an unfrozen volumetric water content function because it assumes that all water in the soil changes phase at a single temperature.

Thermal conductivity

Thermal conductivity k, which characterizes the ability of a soil medium to transmit heat by conduction, is defined as the quantity of heat that will flow through a unit area of a soil of unit thickness in unit time under a unit temperature gradient. Thermal conductivity units are commonly given as J/(sec.m.°C), kJ/(day.m.°C) or Btu/(hr.ft.°F).

At temperatures above zero (0°C), the thermal conductivity tends to increase with increasing water content (see Harlan and Nixon, 1978, pp. 103-163). At temperatures below zero (0°C), the thermal conductivity is usually greater than that of an unfrozen soil, since the thermal conductivity of ice is much higher than water. The thermal conductivity of frozen soil is highly dependent on the unfrozen water content within the soil.

Furthermore, coarse-grained soils are commonly dominated by quartz, which has a relatively large thermal conductivity; fine-grained soils are dominated by clayey minerals, which generally have a substantially lower thermal conductivity.

The Simplified thermal model assumes that the frozen and unfrozen thermal conductivities are constant with temperature. The material model also makes the tacit assumption that the water content is constant throughout the duration of the analysis. The Full thermal model also assumes constant water content, but the thermal conductivity is a function of temperature (Figure 4-4).

In a coupled convective analysis, the volumetric water content is known from the water transfer analysis; so the user must specify a thermal conductivity verses volumetric water content function for the unfrozen soil. The thermal conductivity of frozen soil is computed internally by the solver as a function of ice, water, air and soil mineral content.

If the ice content is greater than zero, the soil particle thermal conductivity is back-calculated. The frozen soil thermal conductivity at the given water content is calculated assuming all water is frozen according to:

$$K_f = (K_{ice})^{wc} (K_{soil})^{1-wc}$$

This relationship is based on that proposed by Johansen (1975) and described in the following section.

Once the fully frozen conductivity is known, the partially frozen thermal conductivity of the ground is assumed to be linearly partitioned between the unfrozen and frozen states by the ratio of ice content to water content.

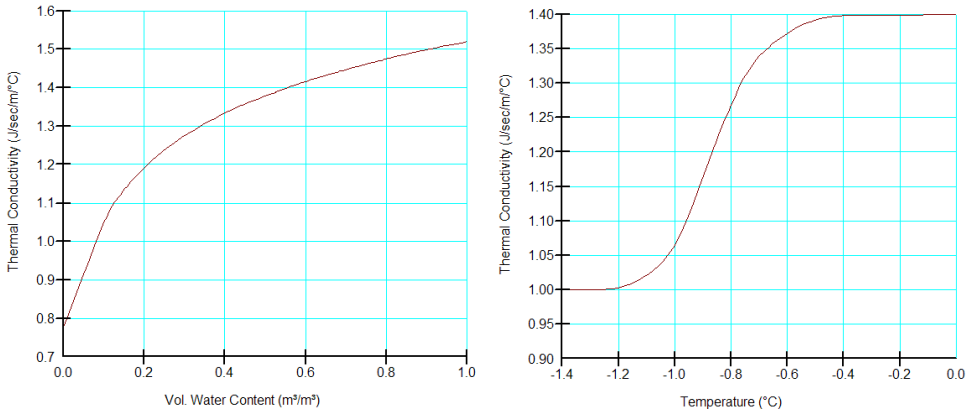


Figure 4-4 Two types of thermal conductivity functions

Estimating thermal conductivity for soils

Various empirical or semi-empirical methods have been developed for estimating the thermal conductivity of soils. These methods have been evaluated in detail by Farouki, (1981), who concluded that different types of soils may require different estimation methods. The method developed by Johansen (1975), however, appears to be the most general method. Since it is beyond the scope of this chapter to present the various estimation methods, only Johansen's method is presented in this section.

For dry, natural soils, the thermal conductivity k_{dry} can be estimated based on its dry density using the following equation:

$$K_{dry} = \frac{0.135\gamma_d + 64.7}{2700 - 0.947\gamma_d} \pm 20\%$$

where the dry density γd is in kg/m3 and the unit weight of soil particles is taken as 2700 kg/m3.

For dry crushed rock materials, the thermal conductivity k_{dry} can be estimated based on its porosity n using the following equation:

$$K_{dry} = 0.039n^{-2.2} \pm 25\%$$

For a saturated unfrozen soil, the thermal conductivity k_{sat} is estimated based on the thermal conductivities of its components and their respective volume fractions.

$$K_{sat} = \left(K_s\right)^{1-n}\left(K_w\right)^n$$

where, k_s is the thermal conductivity of the soil particles, and k_w is the thermal conductivity of the pore water.

For a saturated frozen soil containing some unfrozen water content, w_u, the thermal conductivity k_{sat} becomes:

$$K_{sat} = \left(K_s\right)^{1-n}\left(K_i\right)^{n-w_u}\left(K_w\right)^{w_u}$$

where ki is the thermal conductivity of ice.

For an unsaturated soil, the thermal conductivity k_{unsat} is estimated based on its saturated conductivity, dry conductivity and degree of saturation S using the following equation:

$$K_{unsat} = (K_{sat} - K_{dry})K_e + K_{dry}$$

where:

K_e = 0.7 LogS + 1.0 for unfrozen coarse grained soil,
K_e = LogS + 1.0 for unfrozen fine grained soil, and
K_e = S for frozen soil.

The above equations may be used for an approximate estimation of the thermal conductivity of a soil. It is the user's responsibility to ensure the applicability of the above equations to the respective soils.

Typical values of thermal conductivity

Table 4-1 provides typical values of thermal conductivity for various materials (extracted from Johnston, Ladanyi, Morgenstern, and Penner, 1981). When using this data in the function estimation routine in GeoStudio, make sure the engineering units of the data you select match the units set you have chosen for the model and make sure you have set your time units prior to estimating these functions.

Figure 4-5 and Figure 4-6 show typical thermal conductivity functions for both the full thermal and coupled convective thermal models. These functions were based on sample functions available directly in the function estimation routine. If you use the estimated functions, you are required to enter some guiding parameters which are available in the table below.

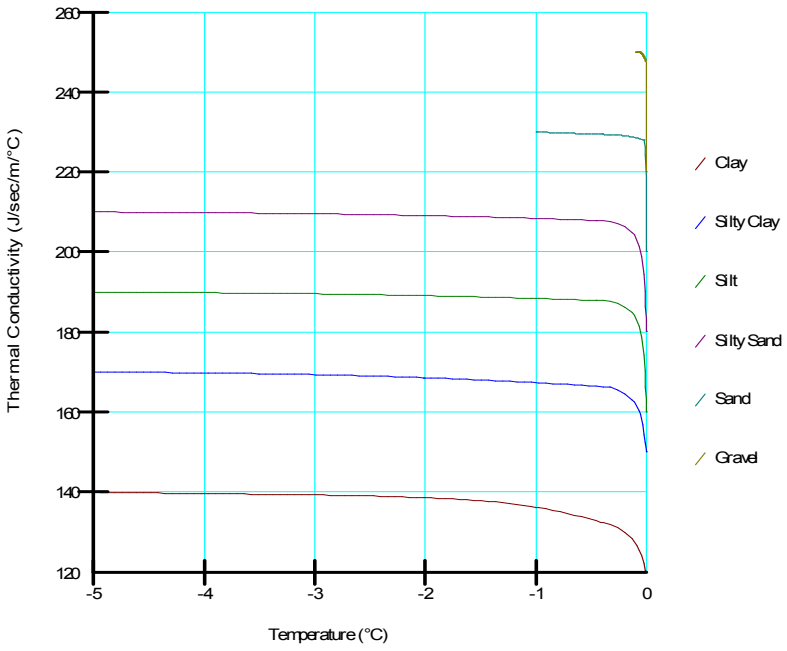


Figure 4-5 Sample K vs temperature functions

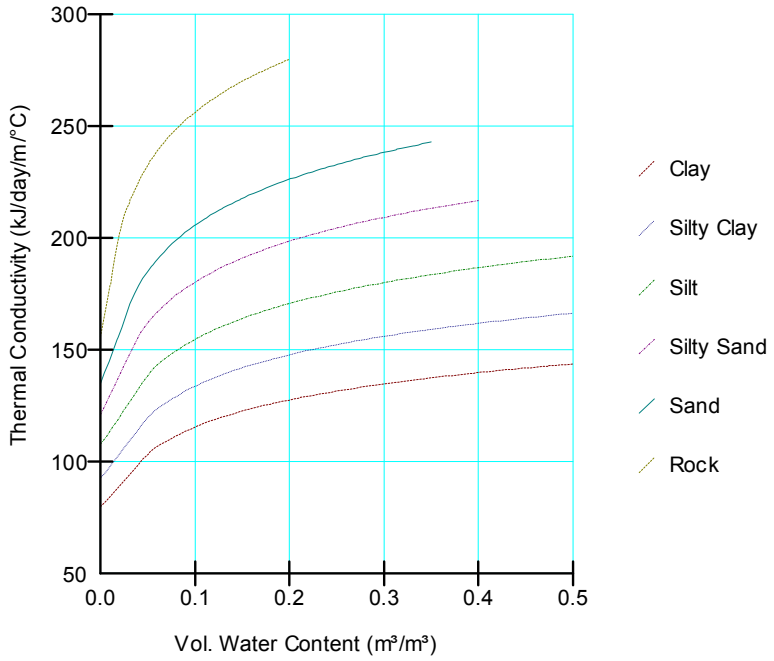


Figure 4-6 Sample K vs water content functions

Volumetric heat capacity

The heat capacity of a material is defined as the quantity of heat required to raise the temperature of the material by a unit degree. When expressed on a per unit weight basis, this quantity of heat is referred to as the specific heat capacity; when expressed on a unit volume basis, the quantity is known as the volumetric heat capacity. The units for specific heat capacity are J/(kg.°C) or kJ/(kg.°C) and Btu/(lb.°F), and the units for volumetric heat capacity are J/(m3.°C) or kJ/(m3.°C) and Btu/(ft3.°F).

TEMP/W users need only specify the function for the unfrozen zone and, based on this, the volumetric specific heat in the frozen zone will be computed internally by the solver as a function of ice, water, air and soil mineral content using the relationships described in the following section.

Table 4-1 Unit weight and thermal conductivity of various materials

Material	Unit Weight	Conductivity		
	lb/(ft^3)	Btu/(hr·ft·°F)	J/(sec·m·°C)	kJ/(day·m·°C)
Water	62.4	0.35	0.605	52.27
Ice	57	1.29	2.23	192.7
Air (dry, still)	0	0.014	0.024	2.07
Snow				
loose, new	–	0.05	0.086	7.43
on ground	–	0.07	0.121	10.45
dense, compacted	–	0.20	0.340	29.37
Soil and rock minerals				
shale	–	0.9	1.5	129.6
evaporites	–	3.1	5.4	466.6
limestone	168	0.75-2.9	1.3-5.0	112-432
dolomite	178	2.9	5.0	432
sandstone	–	1.1-2.4	1.8-4.2	155-86.4
schist	–	0.90	1.6	138.24
gneiss	–	1.4	2.5	216
greenstone	–	1.9	3.3	285.12
slate	–	2.2	3.8	328.3
argillite	–	1.9	3.3	285.12
quartzite	–	2.6-4.1	4.5-7.1	388.8-613
granite	–	1.0-2.3	1.7-4.0	146.9-345.6
diabase	–	1.2	2.1	181.4
gabbro	–	1.4	2.5	216
grandiorite	–	1.5	2.6	224.6

Estimating volumetric heat capacity

TEMP/W uses the volumetric heat capacity in its formulation. The volumetric heat capacity of a soil can be approximated by the dry density of the soil and the sum of the specific heat capacities of its different constituents (namely, soil particles, water, ice, and air). The air component is very small and generally is neglected. A general equation for estimating unfrozen and frozen volumetric heat capacity is: (see Johnston, Ladanyi, Morgenstern and Penner, 1981, pp. 73-147).

$$C = c\gamma = \gamma_d \left[c_s + c_w w_u + c_i w_f \right]$$

where:

C = volumetric heat capacity of the soil,

c = specific heat capacity of the soil,

γ = bulk density of the soil,

γ_d = dry density of the soil,

c_s = specific heat capacity of soil particle,

c_w = specific heat capacity of water,

c_i = specific heat capacity of ice,

w_u = unfrozen water content expressed in % of dry weight of the soil, and

w_f = frozen water content expressed in % of dry weight of the soil.

Examples

The following examples illustrate how to estimate the volumetric heat capacity for various soils.

Example 1

For a dry mineral soil with dry density of 120 lb/ft3 or 1923 kg/m3, the volumetric heat capacity of the soil can be estimated as:

C = 120 lb/ft3 * 0.17 Btu/(lb . °F)

= 20.4 Btu/(ft3 . °F)

or in SI units as:

C = 1923 kg/m3 * 0.71 kJ/(kg . °C)

= 1365 kJ/(m3 . °C)

Example 2

For an unfrozen mineral soil with dry density of 120 lb/ft3 or 1923 kg/m3, and unfrozen water content of 0.25, the volumetric heat capacity of the soil can be estimated as:

C = 120 lb/ft3 * [0.17 Btu/(lb . °F) + 1.0 Btu/(lb . °F) * 0.25]

= 50.4 Btu/(ft3 . °F)

or in SI units as:

C = 1923 kg/m3 * [0.71 kJ/(kg . °C) + 4.187 kJ/(kg . °C) * 0.25]

= 3378 kJ/(m3 . °C)

Typical values of volumetric heat capacity

Table 4-2 below provides typical values for the specific and volumetric heat capacity of various materials, (extracted from Johnston, Ladanyi, Morgenstern and Penner, 1981 and Harlan and Nixon, 1978). Sample functions based on water contents in the coupled convective model are given in Figure 4-7.

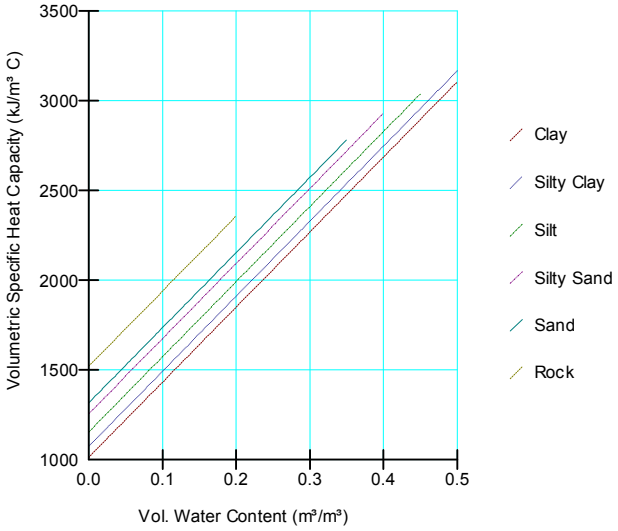


Figure 4-7 Sample heat storage capacity functions

Table 4-2 Specific and Volumetric Heat Capacities of Various Materials

Material	Mass Specific Heat Capacity		Volumetric Heat Capacity	
	Btu/(lb·°F)	kJ/(kg·°C)	Btu/(ft^3·°F)	kJ/(m^3·°C)
Water	1.00	4.187	62.4	4187
Ice	0.50	2.094	28.1	1880
Air	0.24	1.0	0.0187	1.25
Soil minerals	0.17	0.71	28.0	1875
Organic soil minerals	0.40	1.674	37.5	2520
Extruded polystyrene insulation	0.24	1.0	0.65	43.5
Concrete	0.21	0.895	30.0	2010
Asphalt	0.40	1.674	37.5	2520
Snow, fresh	–	–	3.11	209
Snow, drifted and compacted	–	–	7.80	523.5
Granite	–	–	37.1	2490
Limestone	0.29	1.2	48.9	3285
Dolomite	0.21	0.88	37.4	2510
Sandstone	–	–	37.4	2510
Shale	–	–	27.4	1840
Glass	–	–	26.2	1760
Steel	0.11	0.46	56.0	3890
Wood	0.19	0.8	7.79	523

4.4 Sensitivity of results to thermal material properties and water content in soil

In a seepage analyses, the hydraulic conductivity values can change many orders of magnitude. A sensitivity analysis usually considers multiplying or dividing values by a factor of 10. With thermal properties, the change in values (thermal conductivity or heat storage) generally occur within a 1/3 order of magnitude. In addition, the amount of unfrozen water that freezes and releases latent heat can be a significant factor.

In this discussion, the attempt to calibrate a thermal model to real site data is used for illustrative purposes. The geometry of the model was established and default material properties were used in the initial simulations based on previous modeling for the site. At iterative approach was then used to adjust material properties until a reasonable agreement between computed and actual observed temperatures was obtained. The calibrated model was needed for future site planning.

There are three primary material properties that can be adjusted. These are listed below with an explanation of the affect they have on the computed temperature trends. The comparison between actual measured and best fit calibrated data is provided in Figure 4-8.

- **Thermal Conductivity**: Increasing this value increases the rate at which the temperature drops to the freezing point and then after the phase change has occurred. It is evident in the slope of the curves in the figure at temperatures above freezing and at temperatures below about –4 C when most of the pore-water is frozen.

- **Volumetric Water Content**: Adjusting the water content affects the length of time the temperature profile "hovers" around the freezing point before it starts to drop off rapidly again. This length of time is directly related to the latent heat that is released so the more water there is, the more latent heat, and the longer it takes to be removed.
- **Slope of Unfrozen Water Content Function**: This parameter controls the rate at which the water is frozen at temperatures just below freezing. If the function is steep, all the water freezes just below 0 C and the temperature decay curve drops off quickly at a rate almost equal to the rate before the freezing point was reached. If the function is shallow and unfrozen water is still present at temperatures around –2 to –4 C, then the temperature decay curve shows a slower drop in the temperature range just below 0 C. Steep functions are typical of very coarse-grained material and shallow functions are typical of fine-grained materials.

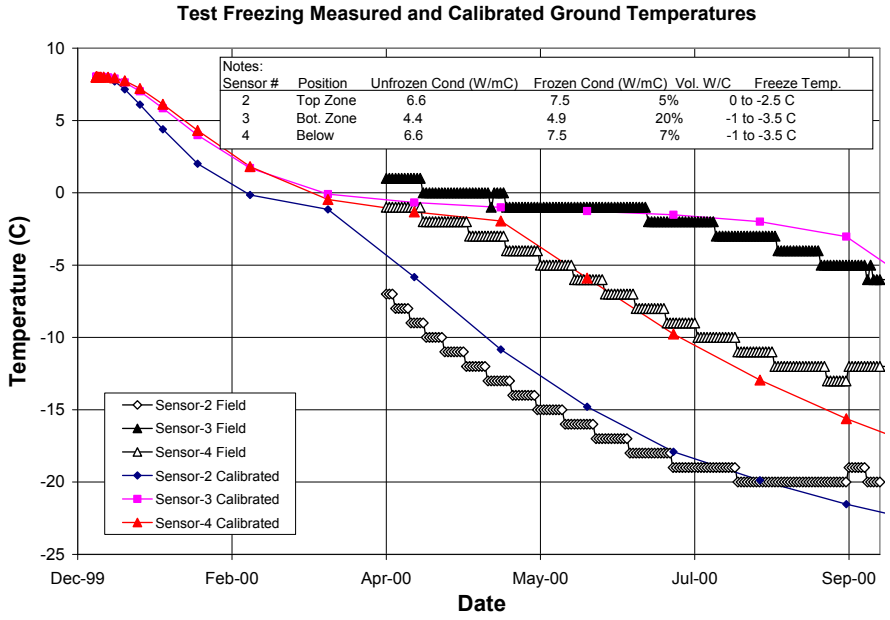


Figure 4-8 Comparison between actual and calibrated ground temperatures

When the model was initially run using assumed material properties the predicted freezing times were much longer than those observed. In fact, the temperatures were just starting to pass through the 0ºC point by the end of the test freezing period. The slopes of the decay curves from the initial temperature of +8 ºC to the 0ºC point indicated that the material thermal conductivity was too low. Sensors 2 and 3 were assumed to be in a clay/ore type ground with very low thermal conductivity and sensor 4 was assumed to be in a sandy type ground. The model was re-run a few times with slight increases to the thermal conductivity on each subsequent iteration. When the drop in temperature from +8ºC to 0ºC agreed somewhat with the measured data the thermal conductivity was held constant.

Once the thermal conductivity was adjusted it was necessary to adjust the volumetric water contents in order to match the time the temperature profile "hovered" near the freezing point. The final calibrated water contents are likely lower than one may expect, but even a slight increase in the calibrated values resulted in several more weeks of freezing delay near the freezing point. The final calibrated values are within the ranges originally reported by the project owner.

Finally, the shape of the unfrozen water content function was adjusted in order to approximate the rate of decay of the temperatures after they have passed through the freeze point. No extreme changes were made

to the material properties (compared to earlier modeling), but they were adjusted somewhat to approximate a more fine-grained, than coarse-grained material.

In carrying out all of the above changes it was important to ensure that the final combination of material properties for any given soil made sense for that soil. In other words, care was taken to not have water contents typical of clay, combined with a thermal conductivity obtained from a solid rock core sample, applied to a soil with an unfrozen water content function found in beach sand.

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. For example, it is the temperature difference between two points or some specified rate of heat flow into or out of the system that causes heat to flow. The solution is the response inside the problem domain to the specified conditions on the boundary.

Sometimes specifying conditions is fairly straight forward, such as defining the temperature or heat flux conditions that exist on a year round basis at some known depth in the soil. 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 the heat removal into a freeze pipe where the amount of heat removed depends on the temperature difference between the fluid in the pipe and the soil. The resulting heat removal changes the soil temperature which in turn changes the heat removal. 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, TEMP/W has tools which make it possible to verify that the results match the specified conditions.

This Chapter is completely devoted to discussions on boundary conditions. Included are explanations of 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 untimely boil 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 thermal analysis the equation is:

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

where:

{T} = a vector of the temperature at the nodes, and

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

The prime objective is to solve for the primary unknowns which in a thermal analysis are the temperatures at each node. The unknowns will be computed relative to the T values specified at some nodes and/or the specified Q values at some other nodes. Without specifying either T 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 T condition. The specified T or Q values are the boundary conditions.

A very important point to note here is that boundary conditions can only be one of two options. We can only specify either the T or the Q at a node. It is very useful to keep this in mind when specifying boundary condition. You should always ask yourself the question: "What do I know? Is it the T or the heat 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 T or Q.

Remember! When specifying thermal boundary conditions, you only have one of two fundamental options – you can specify T 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 T, the solution to the finite element Equation 5-1 will provide Q. Alternatively, when you specify Q, the solution will provide T. The equation always needs to be in balance. So when a T is specified at a node, the computed Q is the Q that is required to maintain the specified T; you cannot control the Q as it is computed. When Q is specified, the computed T is the T 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 temperature at that node you may find that it is unreasonably high or low. You would use your knowledge of the problem to assess if the heat flow applied was reasonable. The T values are computed based on Q and the soil properties so it must be one of three things. Knowing what to look for helps you to judge whether that is reasonable or not.

TEMP/W always provides the corresponding alternative when conditions are specified at a node. When T is specified, Q is provided, and when Q is specified, T is provided. The computed Q values at nodes where a temperature is specified are referred to as Boundary Flux values with units of heat per time (e.g. J/sec). 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 T nor Q is specified at a node, then the computed Q is zero. Physically, what it means is that the heat 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. Heat leaves or enters the system only at nodes where T, or a non-zero Q, has been specified. At all nodes for no specified condition, Q is always zero. This, as we will see later in this chapter, has important implications when simulating such conditions as freeze pipes at a single node.

The temperatures in a thermal analysis are the primary unknowns or field variables. A boundary condition that specifies the field variable (T) 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 thermal modeling simply refers to boundary conditions as temperature (T) or heat 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 2012 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 it becomes independent of the mesh and the mesh can be changed if necessary 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.

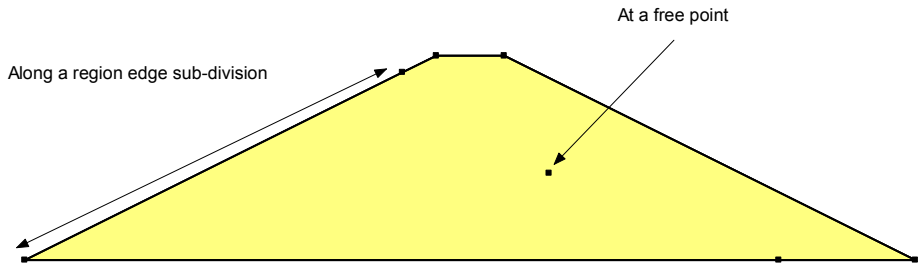


Figure 5-1 Desired BC locations

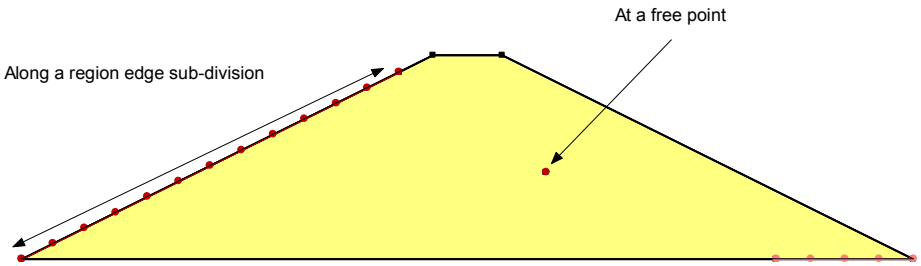


Figure 5-2 BC's attached to geometry

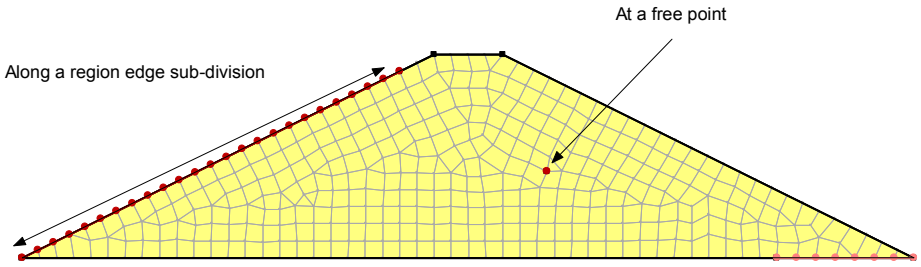


Figure 5-3 BC's with underlying mesh visible

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 Temperature boundary conditions

TEMP/W allows a temperature boundary condition to be specified in various ways: as a fixed value; as a time varying function; or even as a value that is modified based on the existing ground temperature, as would be the case for modifying an air temperature versus time function in order to apply a ground surface temperature boundary condition. This latter example is discussed in more detail later in this chapter.

One thing to keep in mind when applying a temperature boundary condition in a transient analysis is that the applied temperature should agree with the initial condition temperature at the node or else a large heat flux will be generated to transition the temperature from the initial condition value to the new boundary value. Initial conditions are discussed in more detail in the Numerical Issues chapter.

5.5 Heat flow boundary conditions

The second of two options for specifying boundary conditions is to specify a rate of heat flow across the edge of an element. An example may be the rate of heat flow from a pipe or across the ground surface. 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 heat flow (q) into nodal total heat flows (Q). Even though TEMP/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 (or in the case of GeoStudio, a region point) instead of q on the edge of the element.

First of all, TEMP/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. 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 TEMP/W the edge can have either two or three 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 5-4. When two or more edges have a common node, the contributions from each edge accumulate at the common node as illustrated in Figure 5-4. These illustrations refer to seepage quantities, but the same rates and concepts can apply to thermal models.

Higher-order elements with three nodes along the edge of an element are usually not required in a thermal 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, higher order elements are used in a thermal and seepage analysis not because they improve the results, but because mesh consistency is required when integrating with other types of analyses.

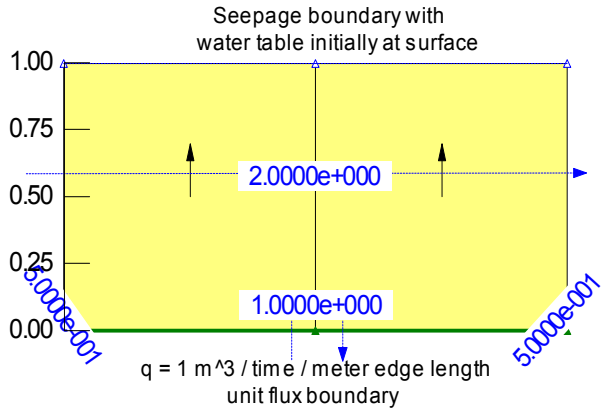


Figure 5-4 Unit and total nodal flux boundary condition relationships

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 5-5. 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.

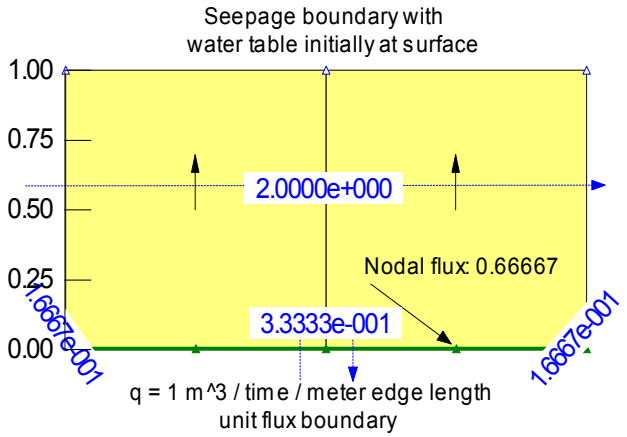


Figure 5-5 Distribution of unit flux in element with secondary nodes

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 5-6, the nodal boundary flux Q is the contributing area from each element times 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. As the distance from the axis of symmetry increases, so does the nodal flux quantity.

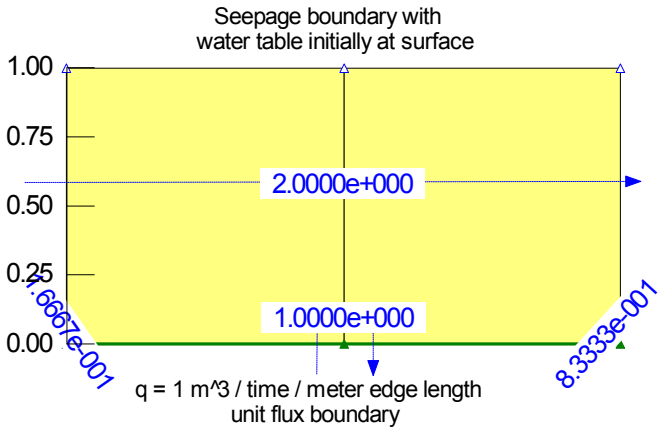


Figure 5-6 Distribution of unit flux in axisymmetric analysis without secondary nodes

The situation is not quite as straight forward for higher order elements where three nodes exist along the element edge as illustrated in Figure 5-7. The detailed formulas for these cases are provided in the Theory Chapter. The case described in Figure 5-6 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 2*Pi 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, however, to remember the element thickness when interpreting the computed Q values.

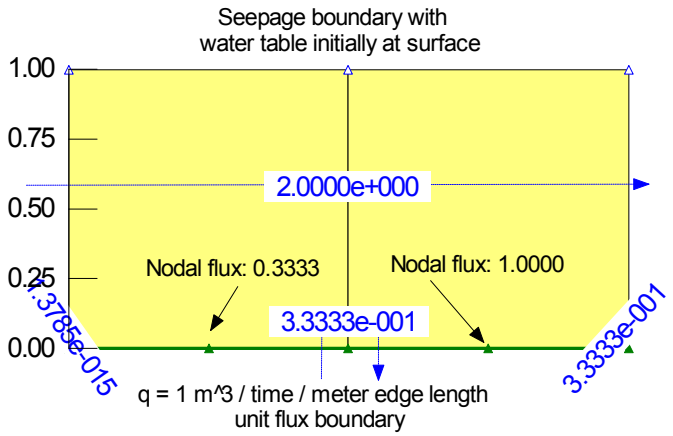


Figure 5-7 Distribution of unit flux in axisymmetric analysis with secondary nodes

In TEMP/W, the specified in or out flow rates (gradients) are always deemed to be normal to the element edges. In addition, flow into the system is positive, while 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. TEMP/W can display heat flow vectors in each element that 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 hot buried pipe while a sink may be a buried freeze pipe or thermosyphon pipe. The important concept about sinks and sources is that they represent flow into or out of the system.

In TEMP/W, flux boundary conditions can be applied along outside edges or points of the mesh or along inside edges or points. There is no difference in how the equations are solved in either case. The only thing to watch for is that you have an understanding of what the area the heat flows across is.

There are a couple special types of heat sink boundary conditions that have been formulated directly in TEMP/W: the convective heat transfer boundary, and the thermosyphon boundary. These are fancy ways of setting up and applying a simple flux boundary sink at a node or group of nodes. They are discussed in more detail below.

5.7 Ground surface energy balance

In nature, the ground surface boundary condition is the net heat flux arising from absorbed solar and long wave radiation and from sensible and latent heat transfer between the ground surface and the overlying air. Schematically, this net flux can be expressed by the energy balance equation as follows: (Goodrich and Gold, 1981)

$$Q_G = -k\frac{\partial T}{\partial n} + Q_{SW} + Q_{LW} + Q_H + Q_E$$

where:

QG	=	ground surface heat flux,
k	=	thermal conductivity at the ground surface,
$\partial T / \partial n$	=	heat gradient normal to and evaluated at the ground surface,
QSW	=	net flux of solar radiation,
QLW	=	net flux of long wave radiation,
QH	=	net flux of sensible heat, and
QE	=	flux of latent heat associated with evaporation of moisture from the surface

5.8 Rigorous approach to calculation of ground surface temperatures

Because the individual heat flux components cannot be measured or estimated with great precision, large errors in the determination of Q_G are possible. Recently however, iterative techniques have been developed that can use readily available climate data so that a prediction of the ground surface temperature on bare soil or beneath a snow pack can be estimated.

In order to use actual climate data for computing the ground surface temperature you must gather and format the necessary data so that it can be pasted into the Climate Data Set of the DEFINE program. There is also an option to import existing data from one or more of the 40+ data sets for various global sites. More than one data set can be applied in a single simulation if there are variations in topography which result in different boundary conditions along the surface.

TEMP/W requires that you enter a complete table of weather data for each day of the simulation. Because of the diurnal nature of all weather data, you should also enter time steps in the transient analysis so that the maximum time step is in the order of 1/8 to 1/4 a day. The use of larger time steps will result in poor averaging of daily climate values and may result in the application of unrealistically high temperature values over the entire day. Remember also that the thermal property "time" units must match the user-specified time step units.

Due to possible extreme changes in climate data from day to day or within any given day, the user-specified time steps may not be small enough to handle a large change in temperature. An adaptive time stepping scheme has been added to the program that will insert extra time steps if necessary. Extra time steps will be added if the percent change in temperature from one time step to the next exceeds the user-specified "tolerance" value as entered in the Key In Time Increments box. Adaptive time stepping should be used in addition to appropriate user established time steps. You should not specify one day time steps without adaptive stepping, and hope the solver follows the diurnal nature of climate data within the one large daily time step. More information on adaptive time stepping is provided later in this chapter.

No snow present

Temperatures within the soil profile are required for the solution of the moisture and heat flow equations. The surface temperature may be estimated (for conditions where no snow pack is present) with the following relationship (Wilson, 1990):

$$T_s = T_a + \frac{1}{\upsilon f(u)}(Q - E)$$

where:

Ts	=	temperature at the soil surface (°C),
Ta	=	temperature of the air above the soil surface (°C),
υ	=	psychrometric constant,
Q	=	net radiant energy (minus transpiration energy) available at the surface (mm/day), and
AE	=	actual vertical evaporative flux (mm/day),

This equation is basically stating that the ground temperature is increased by energy that was not consumed in the evaporation or transpiration process. When calculating the actual evaporation component of the above equation, TEMP/W assumes that the soil is about 80% saturated and that the actual evaporation rate will approach the potential evaporation rate. The VADOSE/W model computes actual evaporation and ground temperature based on actual soil moisture stress conditions; however, TEMP/W does not consider the water stress state.

With snow pack

If there is snow cover, then snow accumulation / melt and ground temperatures are based on an energy balance approach adopted from various methods presented in the following references: SNTHERM89 model, Bras (1990), Liang et al (1994), and Flerchinger (1989).

Full details of how GeoStudio determines snow accumulation and melt, as well as ground temperatures will not be given here, as there are many physical and empirical equations and an entire book could be written on the subject. In general, the snow routine in GeoStudio should balance user supplied precipitation data with applied snow melt during the freshet. The computed snow depths and ground temperatures are affected by many variables, consequently, exact correlation with field data may be hard to obtain. However, the intent of the snow routine in GeoStudio is not to predict snow drifting or frost heaving. The intent is to enable year round analysis that does not force the user to neglect what happens during winter months or during snow melt when it comes to applying a temperature boundary at the surface.

When the climate routine is called, the following activities occur:

The relative humidity, air temperature and precipitation are computed at the elapsed time of the day. If a sinusoidal climate distribution option is selected, the air temperature is assumed to be at its minimum value at sunrise and at its maximum value at 12:00 pm (noon). The relative humidity is assumed to be at its maximum at sunrise and its minimum at 12:00 pm. Both the air temperature and relative humidity are distributed sinusoidal pattern between their minimum and maximum times. The daily precipitation is assumed to follow a sinusoidal distribution over your specified hours of the day, with the peak rainfall (or snowfall) occurring at the mid-point of the specified interval.

If you select to have climate data applied as a constant value over the day then the temperature and relative humidity are fixed for the day and the amount of precipitation will be the daily total, divided by the duration of the day, multiplied by the current solver time step. If you select to have climate data be averaged from day to day, then the previous days and next day's average values are assumed to occur at mid day and then a function is created to look up any value based on the elapsed time since the previous mid day.

If there is precipitation and the air temperature is cold enough, then the albedo of the ground surface is set to be that for fresh snow. If there is no new snow or there is old snow, the albedo is computed accordingly based on techniques in the references cited above.

The net radiation for the time of day is computed using the albedo and other climate parameters as discussed in the references. The net radiation is assumed to be at its peak value at 12:00 pm and is assumed to be zero prior to sunrise or after sunset. It is possible to have a maximum radiation value that is negative. This occurs if the incoming long and short wave radiation is less than the reflected and surface emitted radiation.

Using the data calculated above, the snow accumulation or melt routine is called.

The heat storage capacity of any existing snowpack and any fresh snow is determined.

The snow surface temperature is assumed to be at the freezing point and then an energy balance is carried out to determine if there is a net surplus of energy or an energy deficit. Ground heat flux, sensible heat flux, and latent heat flux are included in the energy balance, as is the ground surface temperature at the last time step.

If the net energy balance across the snow is positive, then some snow is melted to balance the equation. If the net energy is negative, then an iterative scheme is called to compute the correct snow surface temperature to balance the equation. If the energy balance returns zero, then the assumed ground surface temperature is correct.

Once the snow surface temperature is calculated, the snow water equivalent and snow heat storage capacities are updated.

Using the new snow parameters, the snow density is updated as is the snow depth. The snow density of the main snowpack is assumed to be a function of time and is set to increase as the snow season increases or as the duration since the last snow fall increases. The maximum snow density is assumed as 500 kg per cubic meter. The snow depth is a function of the snow density and the snow water equivalent.

Once the snow layer parameters are established, the temperature beneath the snowpack is computed by using the fixed snow surface temperature and iteratively solving for a new ground surface temperature such that the ground heat flux into the bottom layer of the snowpack will balance with the heat storage capacity of the snowpack.

5.9 Traditional approach to computing ground temperatures

TEMP/W enables you to choose from a detailed climate data boundary condition option or an empirical approach that correlates ground temperatures with air temperatures. The empirical approach is discussed next.

Empirical approach to estimating ground surface temperature

In practice, ground surface boundary conditions are often correlated with the air temperature using an empirically determined coefficient called the “n-factor”. The n-factor (n_f and n_t) is defined as the ratio of the surface freezing or thawing index (I_{sf} and I_{st}) to the air freezing or thawing index (I_{af} and I_{at}) as represented in the following equations:

$$n_f = \frac{I_{sf}}{I_{af}}$$

$$n_t = \frac{I_{st}}{I_{at}}$$

The magnitude of freezing and thawing n-factors depends on the climatic conditions, as well as on the type of ground surface. Approximate values of the n-factor for several different surfaces have been suggested in the literature and are given in Table 5-1, (Goodrich and Gold, 1981).

In TEMP/W, the boundary conditions at the ground surface can be simulated with a combination of a temperature function and a modifier function. The temperature function describes the seasonal variation of the air temperature, while the modifier function describes the variation of the n factor as a function of the ground temperature at the surface. Example functions are presented in Figure 5-8 and Figure 5-9.

Table 5-1 N-factor values for various surfaces

Surface Type	Freezing (nf)	Thawing (nt)
Soil surface - spruce trees, brush, moss over peat	0.29 (under snow)	0.37
Soil surface - brush, moss over peat	0.25 (under snow)	0.73
Turf	0.5 (under snow)	1.0
Snow	1.0	———
Gravel	0.6 - 1.0	1.3 - 2
probable range for northern conditions	0.9 - 0.95	
Asphalt pavement	0.29 - 1.0 or greater	1.4 - 2.3
probable range for northern conditions	0.9 - 0.95	
Concrete pavement	0.25 - 0.95	1.3 - 2.1
probable range for northern conditions	0.7 - 0.9	

In TEMP/W, at the beginning of a new time step, the air temperature (Tair) at the elapsed time is obtained from the temperature function, and the n-factor is obtained from the modifier function based on the computed ground surface temperature from the previous time step. The phase change temperature (Tphase) is specified by you. The ground surface temperature (Tground) at the new time step is calculated as follows:

$$T_{ground} = (\text{n - factor})\,(T_{air} - T_{phase}) + T_{phase}$$

When no modifier function is specified, TEMP/W assumes the n-factor value to be 1.0 at all temperatures.

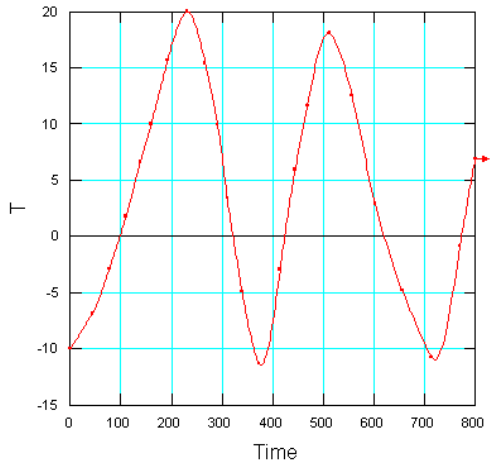


Figure 5-8 Example of T vs time boundary function

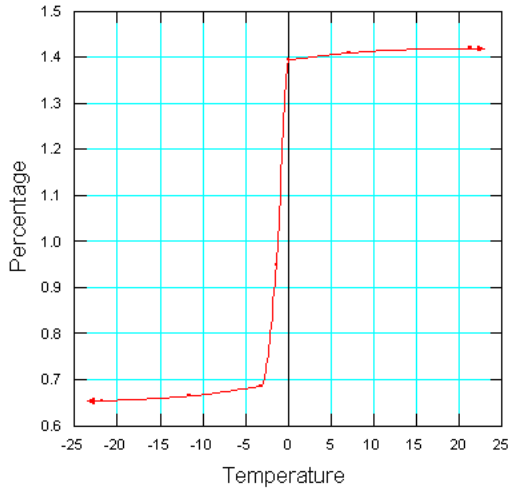


Figure 5-9 Example of modifier function for simulating n-factors

5.10 Convective heat transfer surfaces

TEMP/W can model convective heat transfer boundary conditions for transient analysis where the boundary is either an element edge or one or more individual nodes. Convective heat transfer is the process that removes heat from a surface when that surface is exposed to fluid (liquid or gas) of a different temperature flowing over it. The general rate equation for this type of heat transfer is:

$$Q = hA(T_g - T_{fluid})$$

where:

Q = the total heat flux,

h = the convective heat transfer coefficient,

A = the area in contact with the flowing fluid,

Tg = the surface temperature, and

Tfluid = the bulk temperature of the flowing fluid.

The units for convective heat transfer coefficient will depend on the user-specified heat and time units of the problem, but are in general heat/length/temperature (e.g. W/m/ºC).

Artificial ground freezing application

This type of boundary condition is particularly useful in TEMP/W for determining the ground temperature adjacent to a buried pipe - whether it is a heated pipe or a freeze pipe used for artificial ground freezing. Traditionally, ground freezing analysis has made an assumption about the actual temperature of the outside of the freeze pipe over time, and often the assumption is that the wall temperature is equal to or very close to the brine temperature inside the pipe. This is a very poor assumption, since if the two temperatures were close, then the amount of heat transfer would be quite low due to the small thermal gradient. A more accurate approach is to specify the heat transfer characteristics and let the model determine the heat removal rates and changing ground temperature.

In a typical chilled brine ground freezing application the flow of brine inside the pipe is laminar, the Nusselt number is therefore constant at a value of 4.1, and the convective heat transfer coefficient can be computed from:

$$h_2 = \frac{kN_u}{D_h}$$

where:

Nu = the Nusselt number,

k = is the thermal conductivity of the brine, and

Dh = is the hydraulic diameter (equal to diameter of outer pipe minus diameter of inner pipe).

For other applications, the convective heat transfer coefficient should be determined by consulting an appropriate heat transfer text book or by searching for methods to calculate it on the internet. The convective heat transfer coefficient is a function of fluid temperature, fluid velocity, mass flow rate, and thermal conductivity. Its value depends on whether the flow is laminar, turbulent or in transition. It is calculated based on knowledge of the Reynolds' number, the Nusselt number, and the Prantl number. It matters if the flow is forced convection or free convection.

The convective heat transfer for brine freezing applications typically ranges between 25 and 75 W/m^{2}ºC.

As discussed in the Chapter on meshing, if the freeze pipe diameter is known, then the actual boundary condition for the freeze pipe can be applied to a single node. The heat removed at that node will be a function of the pipe surface area, the heat transfer coefficient, and the difference in temperature between the brine and the ground. When this type of boundary condition is set up in TEMP/W, then you must enter the fluid temperature (as a constant value of function of time), the pipe perimeter length, and the convective heat transfer value. If the pipe is to be located at a mesh corner or edge node, the exposed pipe perimeter would be 1/2 or 1/4 of the whole pipe perimeter (i.e., 1/4 * 3.1416 * diameter).

The following is an example of a freeze pipe boundary condition applied to a shaft freezing project. Each individual pipe is represented by a big Q convective surface boundary condition, as illustrated by the blue triangles at various nodes in Figure 5-10. Notice that there are two defined convective surface boundaries

in this mesh depending on whether the nodes are internal or along the edge. Obviously, the pipe perimeter at the edge nodes is 1/2 of those at internal mesh nodes.

Figure 5-10 shows the temperature contours after 120 days of freezing. The starting temperature of the ground in this case was +8ºC and the phase change temperature of the ground water was set at ºC . Brine temperature was assumed to be -25ºC for all time steps.

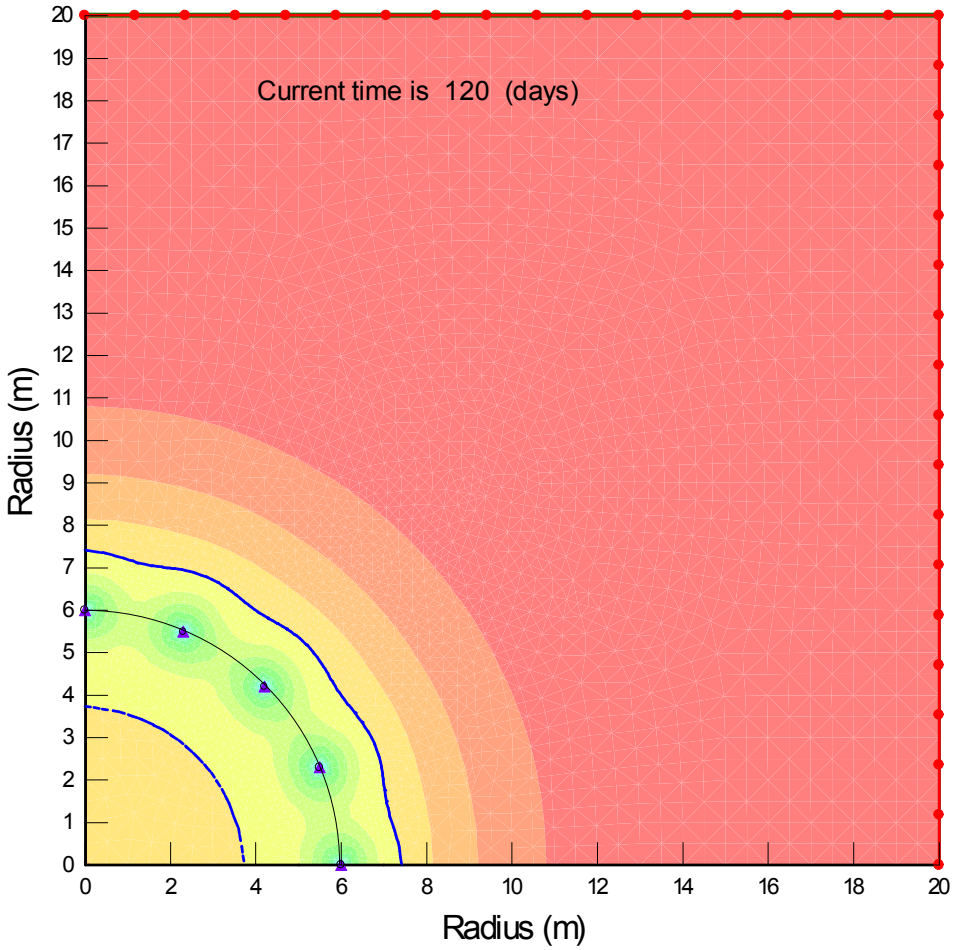


Figure 5-10 Ground freezing of a mine shaft to be excavated

Figure 5-11 shows the solved pipe surface temperature versus time (days) for the problem above. Observe that there is a smooth temperature decay at the pipe. Also observe that the pipe surface temperature is gradually approaching the brine fluid temperature (in this case -25 C). It is important to note that the pipe surface temperature will never equal the brine fluid temperature for if it did, there would be no temperature gradient across the surface and the heat flow would equal zero.

If you do not want to model the pipe as a single node/point, there is the option to specify the boundary condition along element edges and the model will compute the area across which the heat flows.

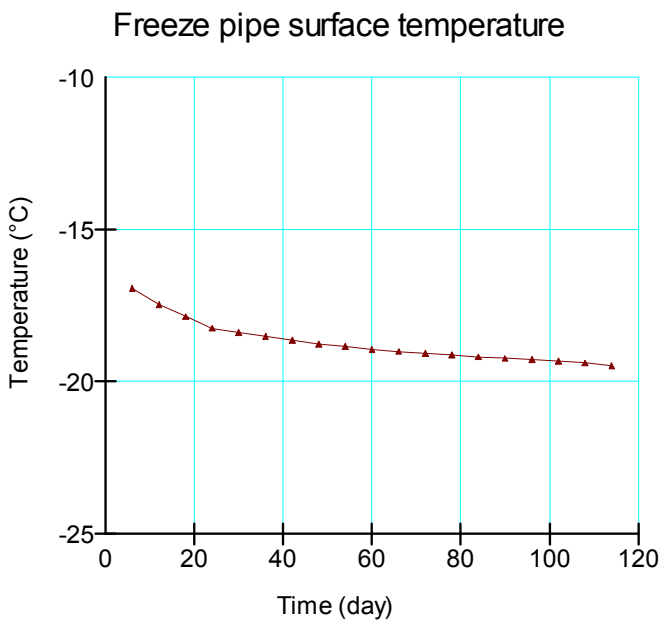


Figure 5-11 Computed pipe surface temperature for ground freezing model

Pyrex tile cooling application

The convective heat transfer boundary condition type is not only suited to buried hot or cold pipes. Consider the example in Figure 5-12 where a hot Pyrex tile 10mm thick is rapidly cooled from 140 degrees to 25 degrees after coming out of the oven. In this example, the model was set up to compare the cooling rates with either still air over the tile or air moving at 10 m/s. The only difference in the applied boundary condition is the computed heat transfer coefficient for the still or moving air.

In this example, the convective surface is not a single node, but the entire top and right edges of the tile. TEMP/W will compute the distances between nodes along these edges and apply the appropriate Q heat flux in the solution of the equations.

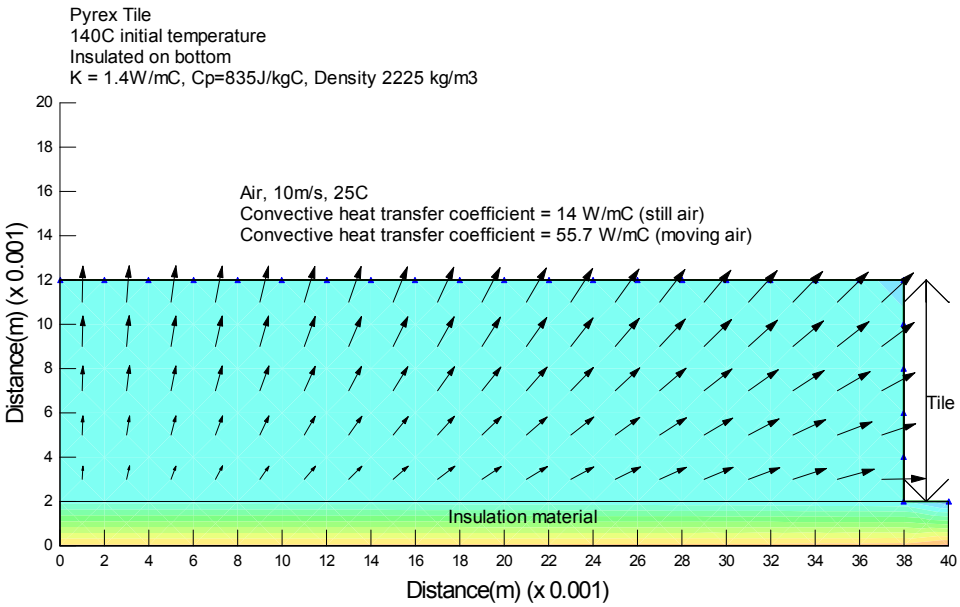


Figure 5-12 Convective cooling of a hot Pyrex tile after fabrication

Time steps for convective surface analysis

Depending on the starting temperature difference between the ground surface and the fluid, it may be necessary to specify very small starting time steps in order to get the problem to converge properly. If a brine temperature was -30ºC and the starting ground temperature was +20ºC, then it would not be uncommon for the first time step in the analysis to be in the order of 1/2 an hour (or its equivalent in whichever time units you have specified). If you notice a large jump or drop in the temperature at a given time step after the analysis is complete, then the analysis may have to be re-solved with small time steps. Likewise, if you notice that the ground temperature increases slightly above its starting value when it is supposed to be dropping, then the first time step is likely too small and should be increased somewhat.

5.11 Thermosyphon boundary conditions

TEMP/W has the ability to let you apply a thermosyphon boundary condition at individual nodes. Each node can represent a single thermosyphon as if you were looking at the end of the pipe. A thermosyphon cannot be represented along the edge of an element because it is a point source (per meter depth) and not a line source (which would make it a plane into the page).

In general, a thermosyphon is comprised of a sealed pipe which may contain a gas, such as Carbon Dioxide. When the gas is warmed, it has a lower density and it will rise to the top of the pipe. If the effect of the cold air blowing across a heat dissipating radiator mounted to the top of the pipe cools the gas, then it will condense and flow down the inside of the pipe wall. If the ground is warm enough at the bottom of the pipe, it will add heat to the condensed fluid, which will cause it to evaporate, become less dense, and rise to the top. The cycle repeats itself as long as the air temperature is less than the ground temperature.

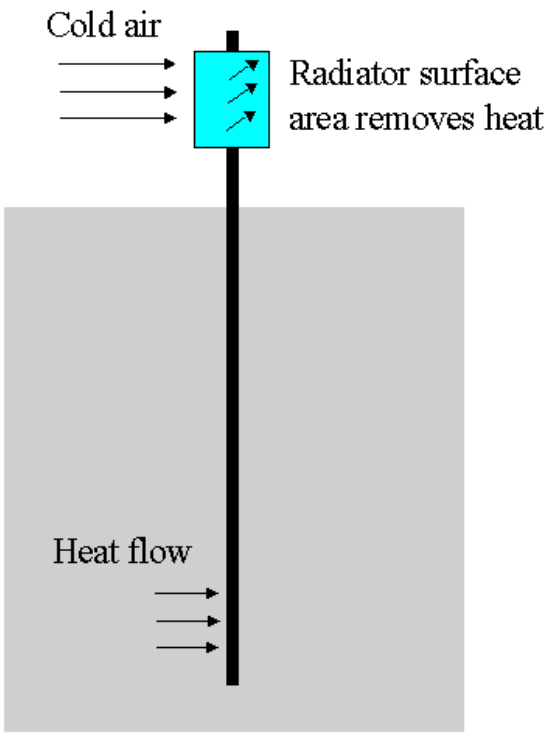


Figure 5-13 Schematic of thermosyphon operation

Figure 5-13 is a schematic of a thermosyphon installed in the ground. Heat flows into the thermosyphon at the base and is removed by the combination of wind speed, air temperature, and radiator surface area at the top. There are several different heat transfer processes occurring in this cycle. There is conduction and convection at the base of the pipe, convection along the length of the pipe, and convection between the surface radiator and the wind. These individual processes can be combined into an overall heat transfer process with the following rate equation:

$$Q = P(T_g - T_{air})$$

where:

P	=	the performance characteristic of the thermosyphon,
Tg	=	the ground temperature, and
Tair	=	the air temperature.

The key to using the thermosyphon boundary condition option is determining the performance characteristic, P, in the above equation.

Experimentation carried out by Artic Foundations Inc. (personal communication) has lead to the following relationship for determining P as a function of wind speed and radiator surface area. This equation is valid for thermosyphons installed at angles of inclination greater than five degrees (i.e., five degrees from horizontal to vertical).

$$P = (2.54 + 4wind^{0.62})area$$

where:

P = the performance for the entire length of pipe in W / °C,

wind = the wind speed in m/s, and

area = the radiator surface area in m^2.

This rate equation gives the maximum heat extraction capability of the thermosyphon over its entire length. Since TEMP/W only models thermosyphons per unit length, this value must be divided by the actual constructed length of pipe in the field.

Figure 5-14 is the thermosyphon data entry dialogue for TEMP/W. You can specify the pipe geometry two ways. If in end view, you must enter the actual length of pipe in the units of your mesh geometry. If in side view, the model will determine the length. You must also enter a maximum functioning air temperature and the minimum temperature difference between ground and air that is required for continuous operation of the thermosyphon.

In terms of heat extraction performance (P), you can enter your own fixed value or you can have TEMP/W compute the value based on the performance equation above and on the radiator size. If you specify a fixed value, the only unknown in the above rate equation is the air temperature and this is obtained from the climate data set that you must enter. Your fixed value must be in units of heat per time per degree temperature. If you use the radiator size option, then the rate equation will become a function of air temperature and wind speed, as well as the actual surface area of the radiator.

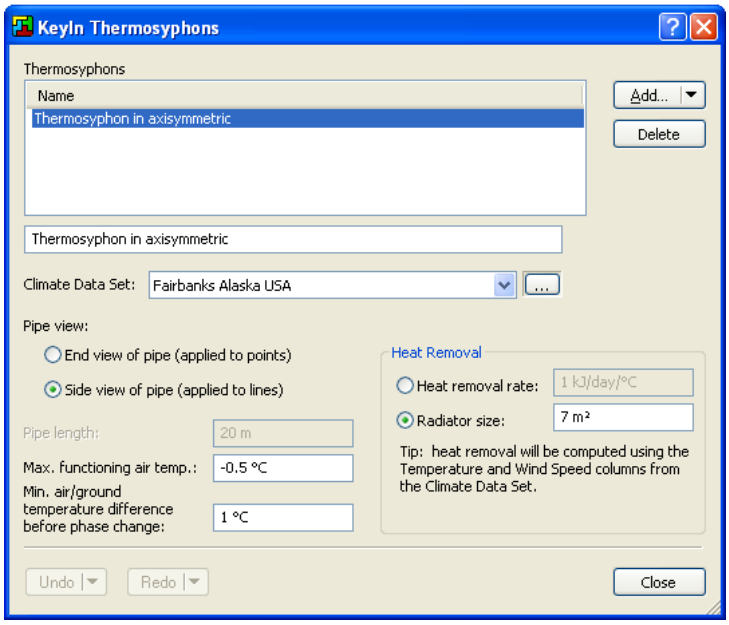


Figure 5-14 Data required for thermosyphon boundary

5.12 Boundary functions

TEMP/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 temperatures or fixed flux values. In a transient analysis however, the boundary conditions can also be functions of time, or a response to flow amounts exiting or entering the flow regime. TEMP/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 TEMP/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.

In general, all 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 is 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.

Temperature versus time

A very useful boundary function is user-specified Temperature versus Time. Consider the case of a ground freezing simulation where the very cold brine temperature is introduced into the much warmer ground. Applying a sudden change in boundary temperature can be numerically difficult to solve, but also physically incorrect. In reality, when the refrigeration plant is turned on, it takes a few days for the brine to cool to its operating temperature. This fact can be applied in the model as illustrated in Figure 5-15, where it takes about 12 days for the brine to reach its minimum operating temperature. This function can be used directly at a boundary node or in combination with the convective flux boundary option as discussed above.

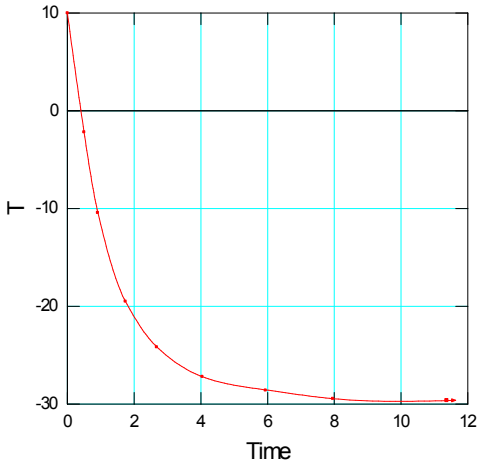


Figure 5-15 Temperature versus time brine cool down function

There are occasions where the boundary function may cycle, or repeat itself over time. This may be the case if a typical year of air temperature data were to be used in a single multi-year analysis. 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.

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 kJ/m/C/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.

Nodal and unit heat flux versus time (Q and q)

As discussed previously, there are only two fundamental types of boundary conditions: either a temperature or flux boundary. As with temperature boundaries, 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. If the function "y" value is positive, this indicates a heat source at the node and if the "y" value is negative, this would indicate a nodal heat sink flux. When you apply a total nodal flux function 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 amount of heat flow 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 heat should be at that time. It will then multiply the function value by the current time step duration to obtain a total amount of heat 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. Using too large a time step will result in the look up heat rate value from the function being applied over a longer time period, which will mean too much heat is applied as a boundary.

Modifier functions

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 or applied temperature be dependent on the current temperature in the soil. Consider the process of convective heat transfer where the heat removed from the ground depends on the fluid temperature flowing over the ground and the ground temperature itself (the difference in these two temperatures establishes the thermal gradient causing heat to flow from the ground to the passing fluid). A modifier function can be set up which establishes what the heat flow should be based on the changing ground temperature. This modifier function is then multiplied by a Q or q boundary function containing a value of -1 for all time. The -1 value simply forces the modifier function to become a sink value that will remove heat from the ground. Consider the function illustrated in Figure 5-16. This says that if the ground temperature is +10, the heat removed from the soil should be 3000*(Q=-1)=-3000. As the ground temperature drops due to this heat removal, the future heat removal drops. Thus, if the ground temperature gets to be -10, the heat removal would be 1000*(Q=-1)=-1000. The Q=-1 notation is the main boundary function that the modifier function is multiplied with to get the desired heat removal rate.

As a closing comment to this example, TEMP/W has a built in unique boundary condition option to deal with convective heat transfer so that it is not necessary to use a Q function with a modifier in this manner. This example is just an illustration of how the modifier function can be used.

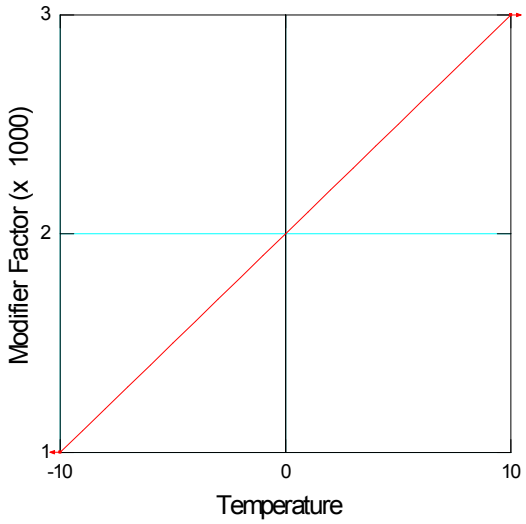


Figure 5-16 Thermal modifier function

Another example of modifier function usage is to adjust a temperature versus time air temperature function so that ground temperatures become a function of both ground and air temperature. This is discussed in detail earlier in this chapter.

One thing to note about the thermal modifier function, it can have any value greater than 0. The seepage modifier function is a percentage modifier, but the thermal modifier is not restricted in that way.

6 Analysis Types

There are two fundamental types of finite element thermal analyses: these are steady-state and transient. 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 mean that the state of the model is steady and not changing. In a thermal analysis, the “state” means the temperature and heat flow rates. If they have reached a steady value throughout the entire geometry, 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. Heat flow beneath an artificial skating rink surface may come close to steady-state if the ground thermal conditions are held fairly constant over time. Ground surface temperatures on an artic runway covered by snow will likely never reach a constant value. The model only knows what you tell it and if you make the assumption the boundary conditions are constant over time, then the model can compute the long term ground conditions in response to your assumption. When the analysis is specified as steady-state, the model can compute these long term conditions without marching forward over multiple time steps. It can solve right to the end “steady-state” condition.

This type of analysis does not consider how long it takes to get to a steady condition and you have to understand that. The model will reach a solved set of temperature and flow conditions for the given set of unique boundary conditions applied to it and that’s it.

That sounds a bit blunt, but 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 somewhat harder to achieve – depending on the degree of non-linearity of your soil property functions. The steady-state thermal equation leaves out the actual “time” variable and omits the entire unfrozen water content function – which means the release of latent heat due to the phase change is not considered. They are not needed in the solution. The unfrozen water content function gives an indication of the rate of release of latent heat as the temperature drops and turns to ice. In a steady-state analysis, the rate of heat release is not considered as any generated latent heat is assumed to have long since dissipated by the time the steady conditions are reached.

Boundary condition types in steady-state

In a steady-state analysis there are two choices of boundary conditions: a constant temperature and a constant heat flux rate. For convenience the heat 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. Convective heat transfer, thermosyphon and climate boundary conditions are not applicable to steady-state analysis, as all of these are time dependent processes.

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 frost to reach a certain depth, or the time it takes to cool a heated ceramic tile etc.

In order to move forward in time during a transient analysis you must tell the solver what the soil temperature conditions are at the start of the time period in question. You cannot move ahead in time if you don't have a starting point. 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 (or starting) temperature at all nodes. TEMP/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 temperatures at all nodes by applying the boundary condition to a region face. 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 which may be difficult to interpret, especially in the early stage of the transient analysis.

When 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 thermal analysis,
- A file created by a transient thermal analysis at a specific time step for the same mesh geometry, or
- A file created by the current analysis at a previously saved time step to that which the current analysis is starting.

Consider the following: you know that the in-situ temperature is -2°C at the ground surface and 0°C at a depth of 5 m. This initial condition can be established by running a steady-state analysis that has a surface boundary temperature of -2°C and a boundary temperature along the bottom of the problem of 0°C as illustrated in Figure 6-1. The computed steady-state temperature file can then be used as the initial conditions. The learn example in the getting started manual uses this approach.

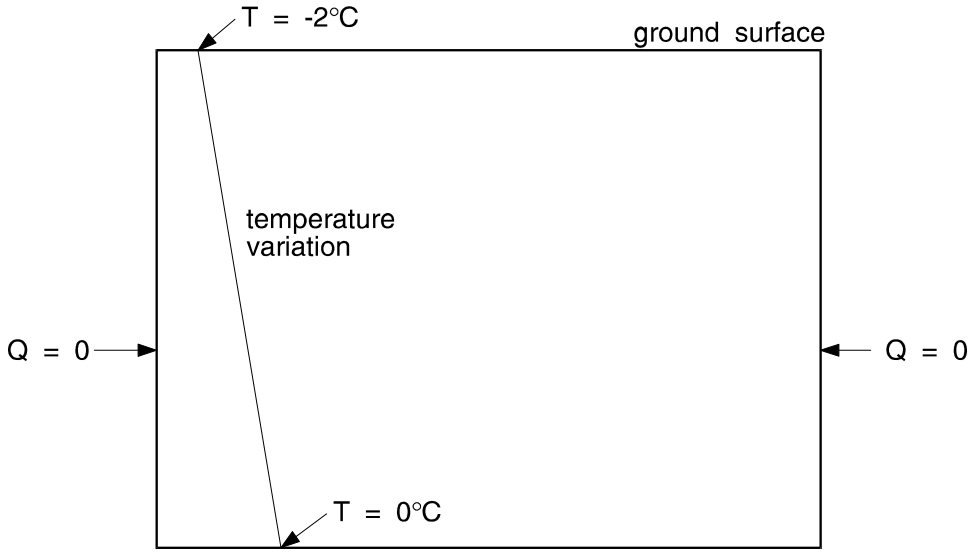


Figure 6-1 Using a steady-state analysis for initial conditions

Activation values

If you have new soil region becoming active and you know it has a certain initial temperature, 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 temperature and have the solver point to this function result. An example of a spatial function is shown below.

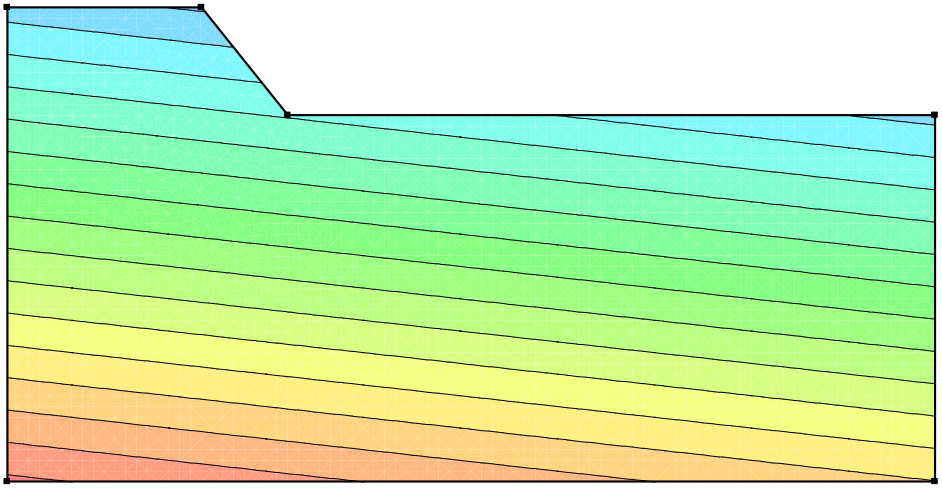


Figure 6-2 Spatial function assigned for initial temperature

No initial condition

In the event no initial condition is specified, TEMP/W will still solve the model. However, it makes the assumption that the ground temperature is uniform at a value of zero. This value of zero does not consider whether the problem thermal units are degrees Celsius or Fahrenheit, so it is not wise to omit leaving out a user-specified initial condition.

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 freezing front is too rapid, the time steps need to be decreased; if the migration is too slow, the time steps may 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 thermal processes related to the ground cooling follow an exponential form. The dissipation of heat is rapid at first due to higher thermal gradients and then decreases with time. A typical example is artificial ground freezing. 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. It is important now to just show the equation so that a key point can be highlighted that will enhance your understanding of the transient finite element method.

The finite element solution for a transient analysis has temperature changes being a function of time as indicated by the $\{T_1\}$ term in the finite element equation below. In order to solve for the temperature 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 (Segerlind, 1984):

$$\left(\Delta t\left[K\right]+\left[M\right]\right)\left\{T_1\right\}=\Delta t\left\{Q_1\right\}+\left[M\right]\left\{T_0\right\}$$

Equation 6-1 *or*

$$\left\{T_1\right\}=\frac{\Delta t\left\{Q_1\right\}+\left[M\right]\left\{T_0\right\}}{\Delta t\left[K\right]+\left[M\right]}$$

where:

dt	=	the time increment,
T1	=	the temperature at end of time increment,
T0	=	the temperature at start of time increment,
{Q1}	=	the nodal flux at end of time increment,
[K]	=	the element thermal conductivity matrix, and
[M]	=	the element heat storage matrix.

Look at this equation carefully for a moment. You can even 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 temperatures at the end of the time increment, it is necessary to know the temperatures at the start of the increment along with the average material properties calculated at the average of the new and old temperatures. If you do not have reasonable values for starting temperatures, then you make the equation difficult to solve because you use these starting temperatures 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 (Segerlind, 1984). 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 temperatures. Small time steps can cause overshoot in the calculation of the new temperatures

and time steps that are too big can result in poor averaging of material properties at the mid point of the time step. Elements that are too big can also result in poor material property averaging while elements that are too small can lead to over shoot problems as well.

The mathematical formulations for time step sizes generally take into account the soil's heat storage capability, its thermal conductivity, and the size of the element. In a homogeneous, non-freezing analysis, the optimum time step can be readily estimated. However, the problem in a 2D freezing 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. Consequently, 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 90^0. 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 application of a cold temperature you should set the initial time step size to be approximately equal in order of magnitude to the unfrozen volumetric specific heat value divided by the unfrozen thermal conductivity value of the soil that is instantaneously loaded. This approach of dividing the storage rate by the conductivity has relevance in consolidation theory, but can also be used as a guideline in this situation.

If you are modeling a rapidly cooling boundary condition, a better approach is to use a Temperature versus Time boundary function to more accurately simulate the initial cooling over a short, finite time period.

The best option is to try some time steps pattern and turn on the adaptive time stepping scheme which is discussed and in the Numerical Issues chapter. TEMP/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 the user's specified time stepping or convergence criteria.

Just to get started, take the total time you need to model and divide it by five. Set the model to have five 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, drop the time steps some more 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. Not too advanced an approach, we know, but it's the truth!

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 6-3 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.

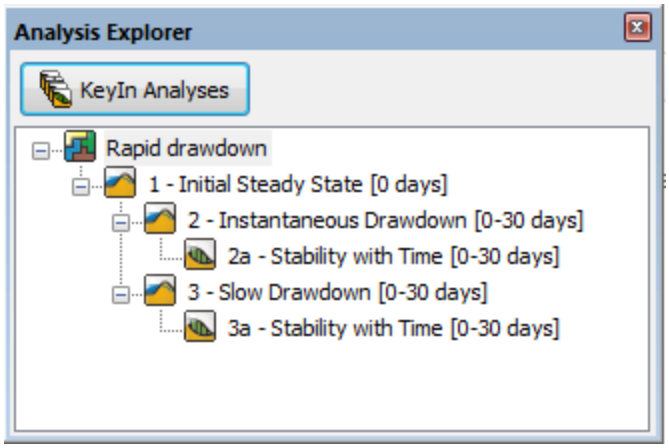


Figure 6-3 Example of an Analysis Tree in GeoStudio

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 freeze pipe installed in the ground as illustrated in Figure 6-4. In TEMP/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.

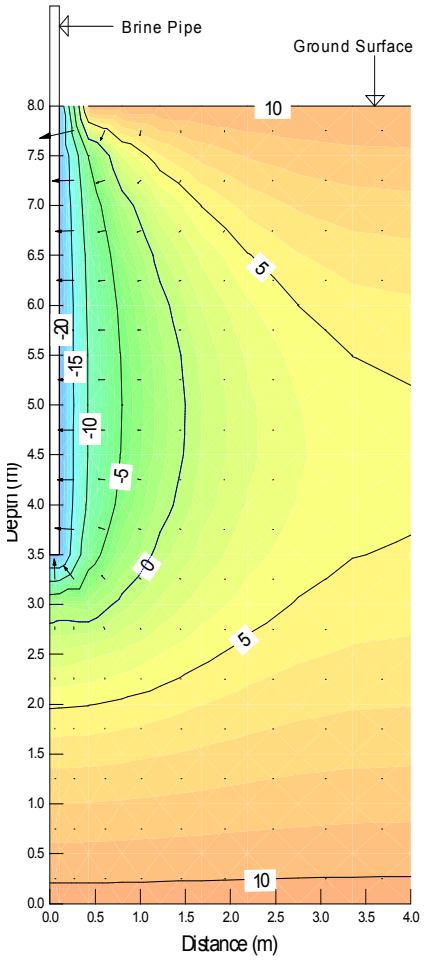


Figure 6-4 Frost bulb around freezing pipe

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., 2 pi radians) before you do the analysis, or simply multiply the above value by 2 pi after the solution is finished. You can change the element thickness for the entire mesh with the Draw Element Properties command.

Infinite elements may be used for the outside far field edge of the axisymmetric mesh (right side away from the axis of symmetry). However, the application of non-zero q (unit flux) boundary conditions along the infinite element edges is not allowed. This is because the nodal contributing area is dependent on the distance of the node from the rotation axis, and since the far nodes of the infinite elements are at infinity, the nodal contributing area becomes undefined. Even though TEMP/W may still compute a solution to the problem in some simple cases, the solution becomes suspect, and the use of q-type boundary conditions in this case is therefore not recommended.

It is also 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

A plan view analysis views the finite element mesh as lying on its side instead of standing upright in a vertical plane. In TEMP/W the only difference between a plan view and a 2D view analysis is the way the area of small "q" heat flux boundary is computed.

In a 2D view, the area the "q" heat flux rate is multiplied by comes from the length of the element edges that the boundary condition is specified on. This length is then multiplied by the specified thickness of the element (typically unit thickness into the screen) to obtain a total flux, Q, to apply at the relevant nodes.

In a plan view, the area the "q" heat flux rate is multiplied by comes from the plan view areas of each element that contribute data to the "q" specified nodes. The solver will compute the element areas and apply the total heat flux, Q, to the nodes.

In general, you should NOT need to use a plan view analysis. Even if you want to model a plan view of a given elevation of the soil, you will typically use a 2D section so that the flux boundary conditions consider the edge length of the elements and not the elemental area of the element when computing the correct total flux to apply.

You can use infinite elements in a Plan View analysis. However, the application of a non-zero q (unit flux) to infinite elements is not recommended. The outer edge of infinite elements is theoretically projected to infinity and consequently the contributing areas for the outer edge nodes are not well defined. The resulting temperatures may or may not be reasonable. If you are going to apply a surface heat flux to infinite elements, you will have to make a careful assessment of the results to make sure they are reasonable. You can apply T and Q type boundaries without difficulty.

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 complied code (dll library) that connects with GeoStudio data or data from another process (eg, Excel)

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

In many cases a function you 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.

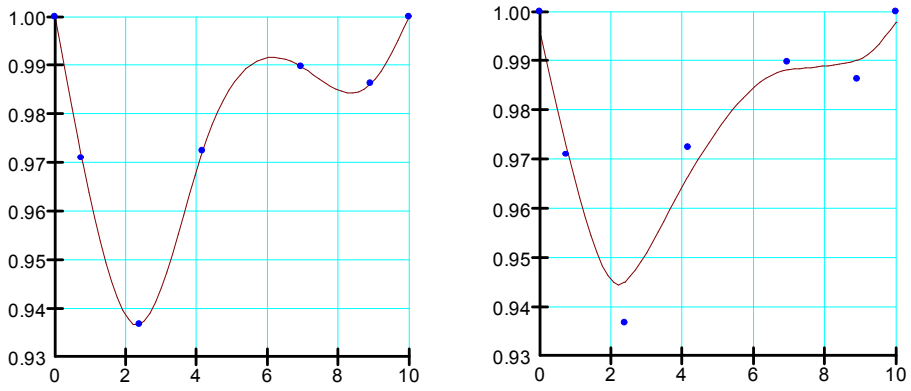


Figure 7-1 Spline functions with different settings

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.

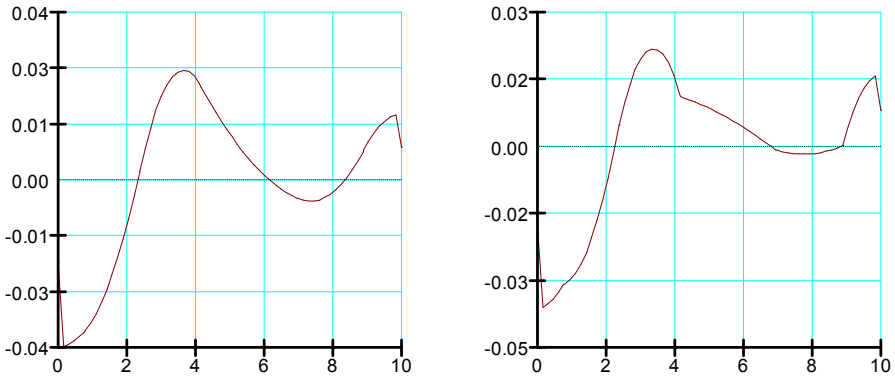


Figure 7-2 Slope of spline functions

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.

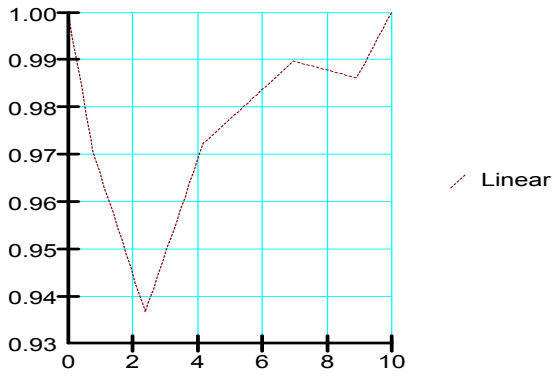


Figure 7-3 Linear fit spline

7.3 Step functions

GeoStudio 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.

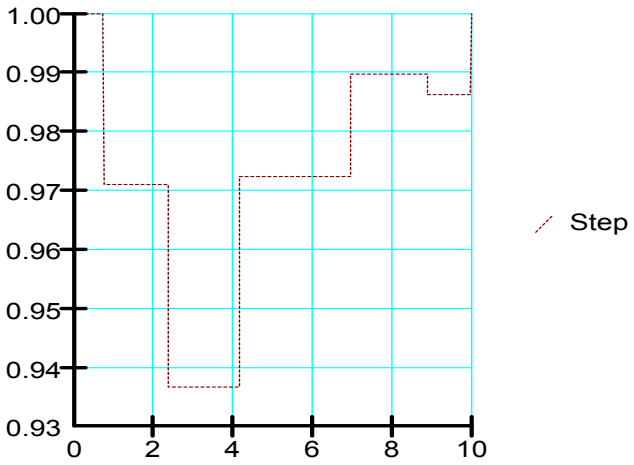


Figure 7-4 Step function

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.

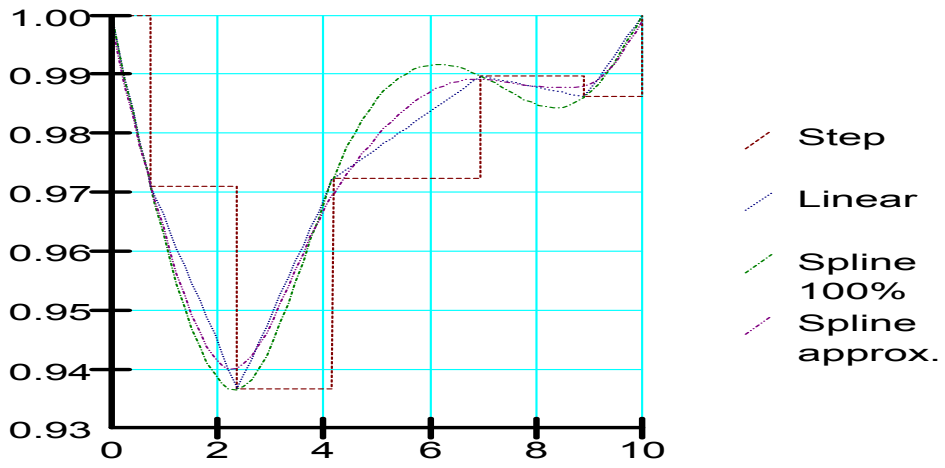


Figure 7-5 Comparison of all data point functions

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 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 TEMP/W can be used to establish starting temperature profiles 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 temperatures.

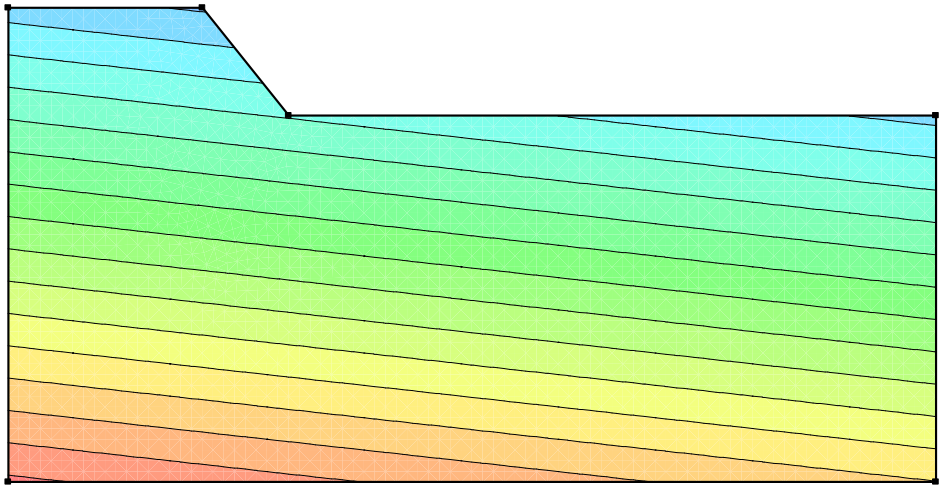


Figure 7-6 Example of spatial function assigned to model

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, TEMP/W does not account for ice lensing or frost heave). Other issues relate to our spatial discretization being too course, 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.

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.

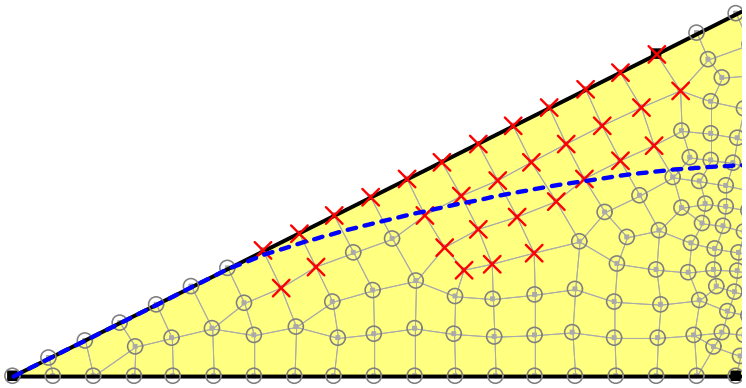


Figure 8-1 Nodes that have not achieved the convergence criteria

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

Commentary

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.

The solutions to thermal analyses involving phase change are highly nonlinear due primarily to the inclusion of latent heat effects. The Simplified Thermal Model is particularly prone to numerical difficulties because the water in the pore-space is assumed to freeze instantaneously instead of over a temperature range. In contrast, convergence can be promoted by using a thermal model that allows the unfrozen water content function to vary over a temperature range. Regardless, convergence may still be difficult.

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. Under-Relaxation Rate

The Under-Relaxation Rate essentially controls the allowable variation in the solution between successive iterations. A value of 1 corresponds to repeated substitution with no under-relaxation. An under-relaxation rate that is less than 1 is always required for problems involving phase change. The default parameters may not be ideal for some numerically challenging problems. 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 unfrozen zones where the thermal gradient is low and the thermal 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 a frozen zone where the thermal 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 frozen zone with thermal 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

TEMP/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. TEMP/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 thermal processes related to the freezing of soils follow an exponential form. The heat is rapid at first and then decreases with time. A typical example is the artificial ground freezing. 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.

Automatic adaptive time stepping

TEMP/W will permit you to activate an adaptive time stepping routine that will insert extra time steps between your specified time steps. There are four methods of calculating adaptive time steps, plus a fourth scenario should the solution reach its maximum allowable iteration count without reaching convergence.

In the first two methods, the change in nodal temperatures is used to determine an increase or decrease in time step between the allowable ranges specified. The first method checks each node to see if the temperatures between successive time steps are changing by more than your percentage. If the allowable percentage temperature change at any given node is too high, then the time steps will be reduced such that the percent change is upheld. In the second option, the vector norm of nodal temperatures is used as the criteria. The vector norm considers all temperatures simultaneously. Experience should show that the vector norm approach is faster for a large mesh with two dimensional heat flows. Analysis of a column study or a mesh with heat flow primarily in one direction is better solved with the individual nodal temperature comparison. The time stepping scheme that compares temperature changes as adopted in TEMP/W is that proposed by Milly (1982). As a general rule, nodal temperatures should not be allowed to vary more than 5 or 10 percent over any given time step. Keep in mind that this criterion depends somewhat on the actual temperatures being solved. For example, 5% of 1 degree is 0.05 degrees. 5% of 100 degrees is 5 degrees. A tighter tolerance may be necessary for problems with an overall higher temperature range.

Regardless of the option to choose nodal temperatures or vector norm of nodal temperatures, the equation used to adjust the time step up or down is:

$$MinMultFactor = \frac{Max\ Step}{Last\ Step}$$

$$MultFactor = abs\left(\frac{tolerance \times T_0}{T_1 - T_0}\right)$$

The actual time step multiplication factor used to modify the existing time step is the lesser of the two values calculated in the above equation.

The third adaptive time stepping control routine does not consider nodal temperature change percentage. It continually tracks the total iteration count required to achieve convergence. If the iteration count exceeds 5, then time step adjustments are assessed. If the iteration count is less than 5, then no time step adjustment is made for the next solve step. When deciding how much to increase or decrease time steps, the solver compares the last converged iteration count with the current average iteration count. So, if the average iterations to converge are 8 and the last time step took 10 iterations, then the solver will reduce the time step by 8/10. Likewise, if the average count is 8 and the most recent count is 6, then the time steps will increase by 8/6.

The fourth method for adaptive time stepping only applies if the analysis type is a coupled convective model with SEEP/W and, or AIR/W. In this case, it is possible to have the time steps adjusted such that the Courant criteria is met. This criteria attempts to adjust the time step size such that the advective component of air or flow flowing with heat transfer does not enter an unstable state. More information about the Courant number is provided in the AIR/W and CTRAN/W manuals.

In the adaptive scheme, the time steps are reduced for the first 2 iterations and are then held constant until convergence is reached. If the maximum number of iterations is reached without convergence, then the solver checks to see if the current time step is larger than the user-specified minimum allowable step. If the current step is larger then the solver will reduce the current step by half and repeat the time step. If there is no room to reduce the step by half without reducing it below the user's minimum value, then the minimum value will be applied and the step repeated.

This multi faceted time stepping logic can be quite powerful. Often you will find it quite useful to specify a lower number of allowable iterations to reach convergence rather than a higher number, because if the solver gets stuck on any given time step, it will reach the maximum allowable iterations more quickly and then try reducing the time step in half. The trade-off becomes repeating a time step or allowing more iterations. It is worth a simple trial and error test early on in a more lengthy analysis to see which option is better in the long run.

The adaptive scheme will always insert just enough time steps to return to the increments established by you. This way, no write out data steps are missed and you still have control over the general time stepping of the solution. In order to activate adaptive time stepping you must set up their time steps with preferred "save" time steps. Then you check the adaptive time stepping box and time step criteria as well as the maximum allowable change in nodal temperature per time step, the maximum adaptive step, and the minimum allowable time step. The time steps are entered in the same time units as the thermal conductivity.

9 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 mesh. If, at this point, you do not have the tools or the understanding of how to interpret the massive amount of data that may have been generated by the solver, then you have wasted your time.

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 I applied get reflected in the actual solution? Did the soil respond how I thought it would respond? If not, how do I 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 is should be used in the overall thought process. It's a good idea to read this entire chapter.

9.1 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 do a steady-state analysis, you cannot interpret any data related to changes in time. You can view instantaneous flux values that have units of heat 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 an unfrozen 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 an unfrozen content function, then it stands to reason that you cannot view unfrozen contents. Water contents can only be viewed in a steady-state solution if you have defined an unfrozen content function that the solver can access to report water contents based on solved steady-state temperatures. No water content function… no reported unfrozen 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, you can only graph how the data changes with position.

Finally, in a transient analysis, you cannot plot the heat flow path across the entire geometry. This is because in a transient process there are no single flow lines. Sure, heat is flowing, but a single source of heat 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 will be. You can plot the heat 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 heat flow 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.

9.2 Node and element information

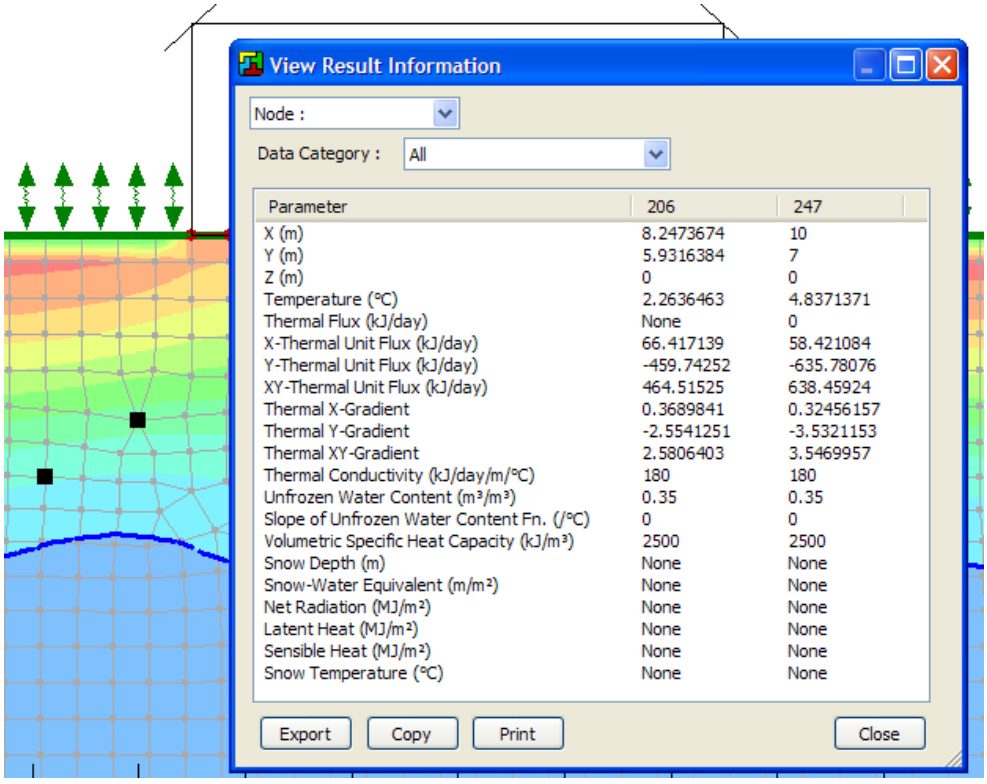
In order to understand what type of information can be viewed as results output, it helps a bit to know how the data is obtained. So, recapping… you set up the problem geometry, define material properties, and apply boundary conditions of either known temperature or flux. The solver assembles the soil property and geometry information for every Gauss point in every element and applies it to the heat flow equation that is written for every node. Therefore, at each node we have some applied boundary data, some 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 temperature 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.

Figure 9-1 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 temperatures, fluxes, velocities, gradients, conductivities, heat capacity and unfrozen 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.

One key point to note in the figure below is that one of the nodal Thermal Flux quantity is set to "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 heat 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 heat fluxes at a node should equal zero in a properly converged solution and has no heat flow (or a value of "none"). In other words, there is heat balance at the node. The node with the value of zero is a true value of zero. In this case, it is a heat pipe BC location and is not extracting heat at this time in the analysis.

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 heat fluxes at that node will not be zero, because heat is either gained or lost at that point.



Parameter	206	247
X (m)	8.2473674	10
Y (m)	5.9316384	7
Z (m)	0	0
Temperature (°C)	2.2636463	4.8371371
Thermal Flux (kJ/day)	None	0
X-Thermal Unit Flux (kJ/day)	66.417139	58.421084
Y-Thermal Unit Flux (kJ/day)	-459.74252	-635.78076
XY-Thermal Unit Flux (kJ/day)	464.51525	638.45924
Thermal X-Gradient	0.3689841	0.32456157
Thermal Y-Gradient	-2.5541251	-3.5321153
Thermal XY-Gradient	2.5806403	3.5469957
Thermal Conductivity (kJ/day/m/°C)	180	180
Unfrozen Water Content (m³/m³)	0.35	0.35
Slope of Unfrozen Water Content Fn. (/°C)	0	0
Volumetric Specific Heat Capacity (kJ/m³)	2500	2500
Snow Depth (m)	None	None
Snow-Water Equivalent (m/m²)	None	None
Net Radiation (MJ/m²)	None	None
Latent Heat (MJ/m²)	None	None
Sensible Heat (MJ/m²)	None	None
Snow Temperature (°C)	None	None

Figure 9-1 Visualization of node information

Figure 9-2 is the corresponding Gauss point information for the Gauss points located just above and to the right of the nodes illustrated in the previous figure. The shaded region in the figure shows the contributing

area of that Gauss point and in this case because the element is rectangular and has secondary nodes, this Gauss area is equal to one ninth 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 unfrozen water content value of 0.35%, 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's not always easy and there is no sure quick or automatic method to make that happen.

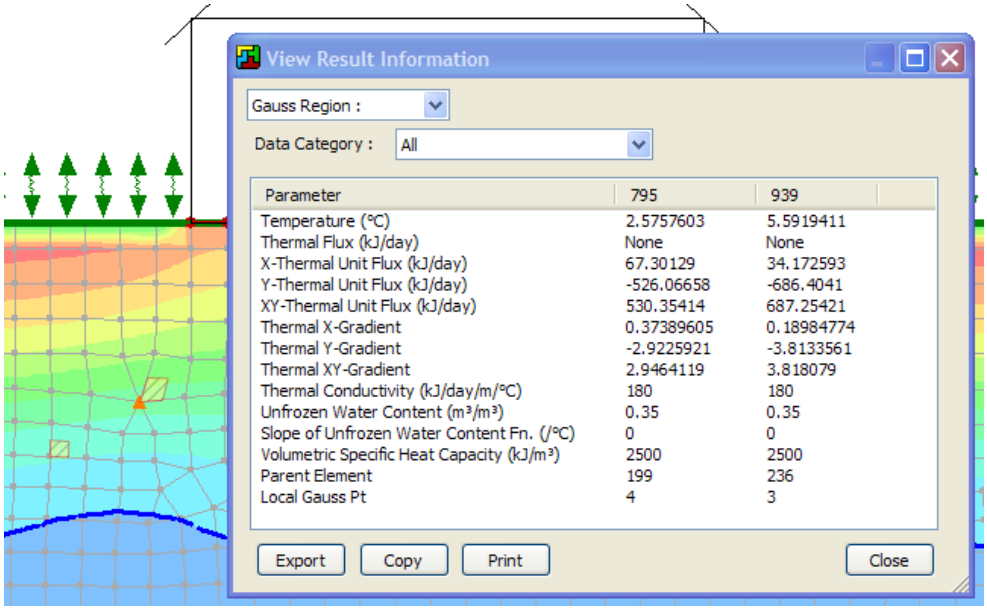


Figure 9-2 Visualization of element Gauss point information

9.3 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 9-3 shows a combination of all three graph data objects in a single model 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.

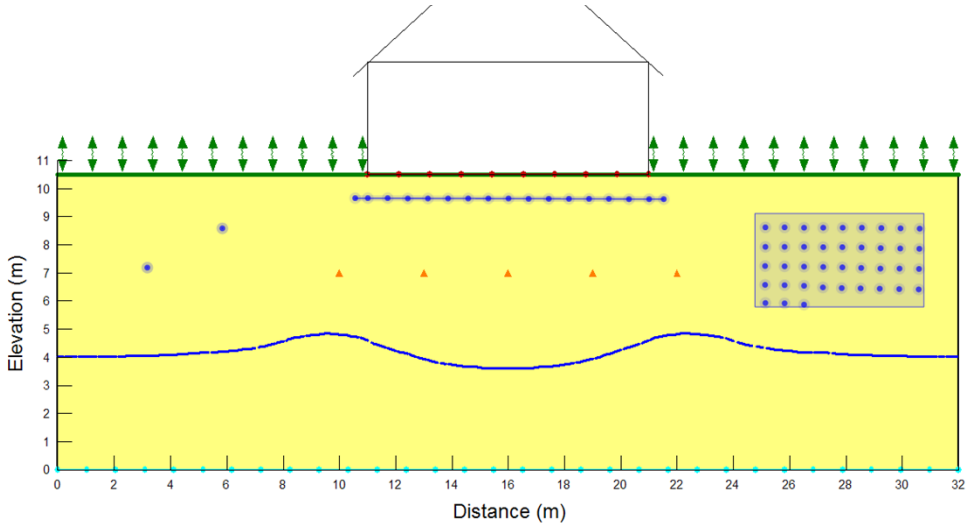


Figure 9-3 Graph data selection options (points, lines, planes)

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 9-4 where the entire up ground surface edge line beneath the building 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.

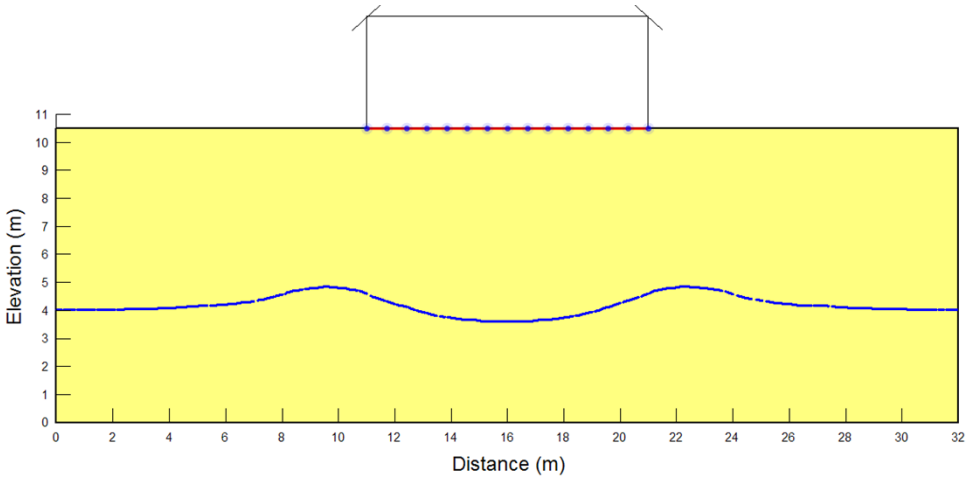


Figure 9-4 Graph selections based on geometry item (soil region edge line below building)

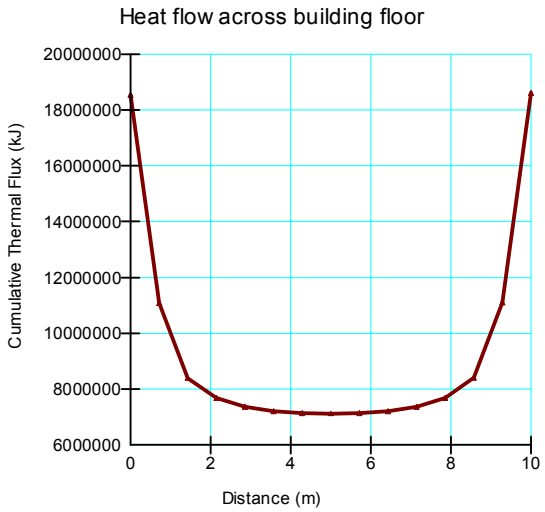


Figure 9-5 Cumulative heat flow beneath building after 2 years

9.4 "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 9-1 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.

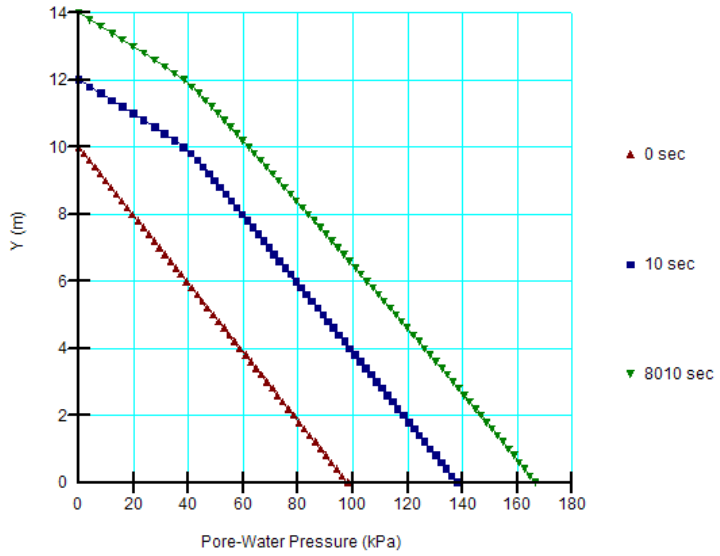


Figure 9-6 Graph showing how missing data is excluded and not printed as zero

9.5 Isolines

You can use the Draw Isolines command to draw an isoline contour of any 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 the phase change or freezing front position as shown in Figure 9-7.

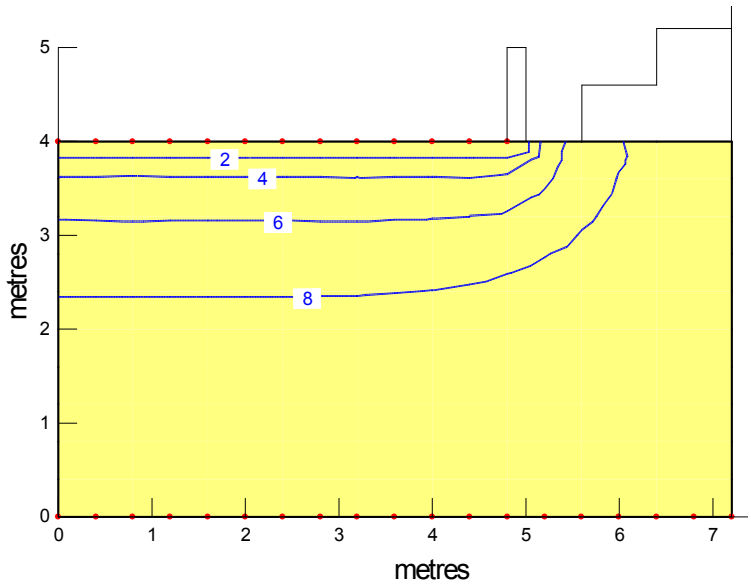


Figure 9-7 Transient position of freeze-thaw line during frost penetration beneath a skating rink

Most parameters cannot be contoured for more than a single time step at a time because, unlike plotting the change in freeze-thaw line, most contour values vary over the entire domain. In the event you want to view contours at different time steps, you must select the time step in question and re-plot the contour as

shown in Figure 9-8. In the event you want more specific data output as a function of time and position, you can use the graphing feature.

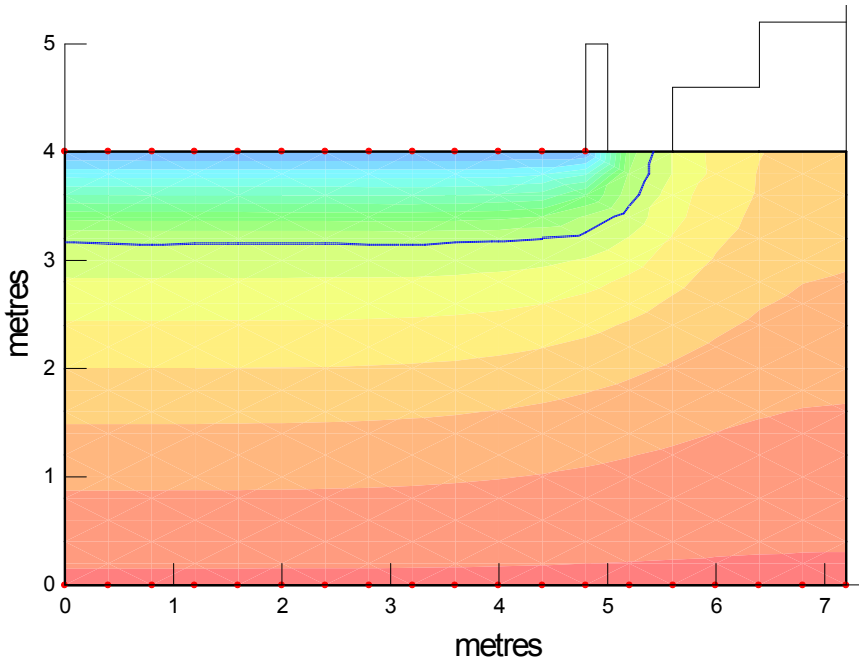


Figure 9-8 Freeze-thaw line and temperature contours after time step 6

9.6 Projecting Gauss point values to nodes

TEMP/W performs contouring calculations based on parameter values at the nodes. Since the primary parameter, temperature, is computed at the nodes, this parameter can be contoured directly. However, secondary parameters, (velocity, gradient, conductivity, and unfrozen 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).

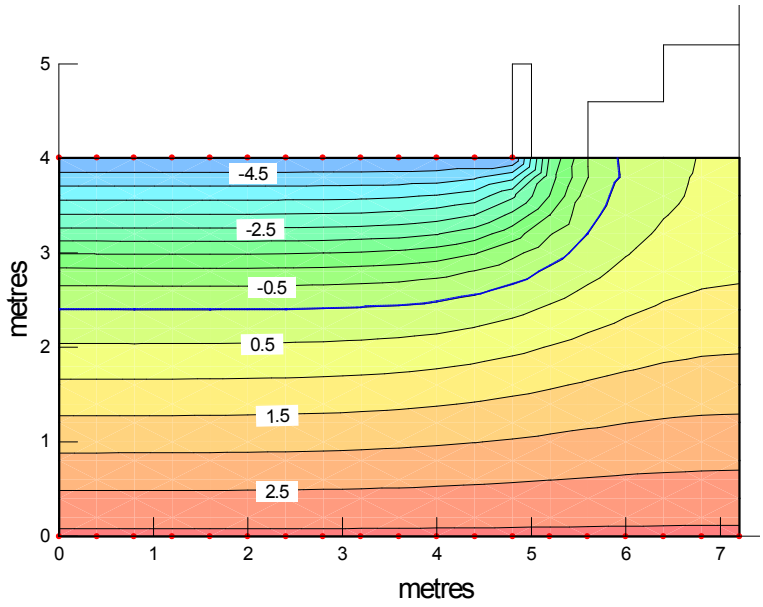


Figure 9-9 Illustration of isotherms generated in TEMP/W program

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,
<N>	=	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 9-10 is an example of the local coordinates at the element corner nodes when projecting outward 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 unfrozen water content and that in some elements the water content at the Gauss points varies over the complete range of the water content function. Projecting such a large variation to the nodes can result in projected nodal water contents beyond the range of the unfrozen 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.

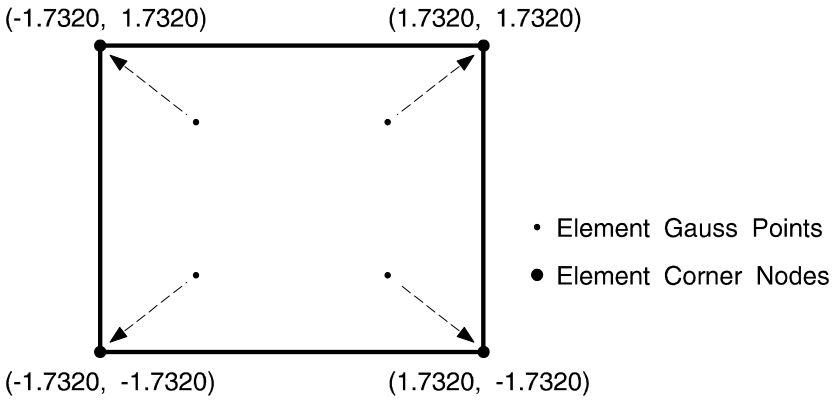


Figure 9-10 Local coordinates at the corner nodes of an element with four integration points

9.7 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 the section above we introduced isotherms. We can use a picture of the isotherms to tell us something about what is going on in the soil. In particular, if we consider how close adjacent isotherms are to each other, we are in effect, considering how steep the thermal gradient is. If we recall that the amount of heat flow is equal to the thermal gradient multiplied by the conductivity, we then have a fast and clear picture of where the areas of high heat flow are in the domain we have modeled.

TEMP/W is a powerful tool in that it will let you contour many different parameters, such as temperatures, gradients, heat fluxes, water contents and thermal conductivity. Figure 9-11 is a contour of the actual heat flux values for the temperature isotherms illustrated above. As we already determined, the isotherms were close together near the surface and we therefore realized the heat flux was higher in this region also. Figure 9-11 confirms our expectations.

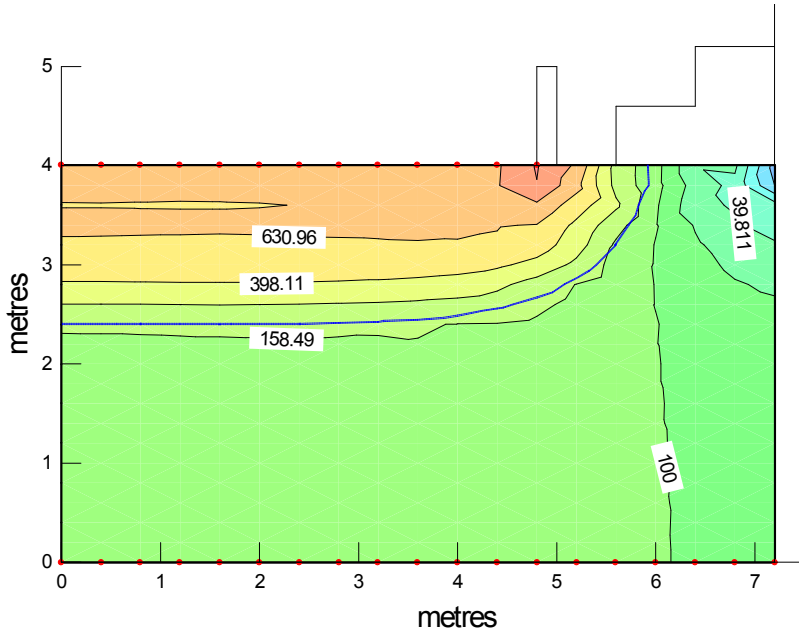


Figure 9-11 TEMP/W computed heat flux contours

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.

9.8 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.

9.9 Energy flow vectors and flow paths

Calculating gradients and velocities

Once the solution has converged and the nodal temperatures are known, TEMP/W computes the thermal gradients and heat flow velocities at each of the integration points within each element. The gradient at each Gauss or integration point is computed from the equation:

$$\begin{Bmatrix} i_x \\ i_y \end{Bmatrix} = [B]\{T\}$$

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

{T} = the vector of temperature at the nodes.

The heat flow velocities at each Gauss point are computed from the equation:

$$\begin{Bmatrix} v_x \\ v_y \end{Bmatrix} = [C][B]\{T\}$$

where:

v_x = the velocity in x direction,

v_y = the velocity in y direction, and

[C] = the thermal conductivity matrix.

TEMP/W stores in an array the thermal conductivity at each Gauss point used in the formulation of the finite element equations. The same thermal conductivity values are later used to compute the velocities.

Energy flow vectors

Energy flow vectors are a useful way of seeing not only where the heat flow is occurring, but how much flow there is relative to other regions of the domain. Examples of such energy flow vectors are given in Figure 9-12. TEMP/W uses the magnitude of the actual energy flow in its calculation of how large to display the vector, so that you have a visual representation of where the energy flow rates are high or low. For each element, the average x-energy flow rate and average y-energy flow rate from the Gauss point flow rate values are computed and then vectorally summed to obtain an average energy flow vector for the element. This average vector is plotted with the tail of the vector at the center of the element.

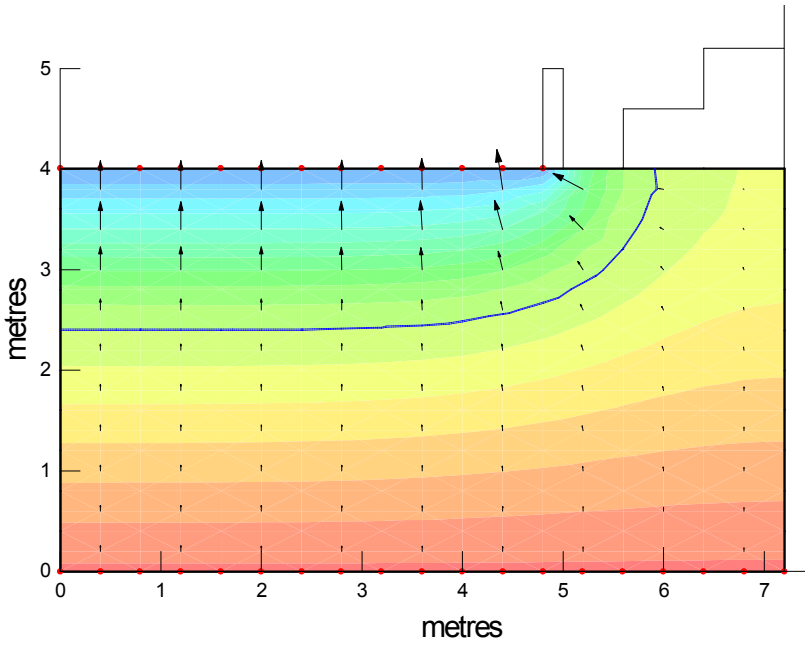


Figure 9-12 Heat flow vectors

When displaying vectors, TEMP/W finds the maximum vector and draws it at the length specified in the Draw Vectors dialog box. All other vectors are drawn in proportion to the element energy flow rate relative to the maximum flow rate. For example, if the element flow rate is one quarter of the maximum flow rate for the domain, then the length of the 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 flow rate and adjusting the value for the scale of the page and engineering units.

Vectors are only drawn in element regions being viewed. Choose View Element Regions if you wish to view different materials or to not view infinite elements. If no elements are viewed, an error message appears when you choose Draw Vectors.

Flow paths

The TEMP/W flow paths are not truly flow lines or stream lines as in a traditional seepage flow net sense of the word. 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 energy flow rate vectors in an element that a source of heat 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 TEMP/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 high thermal gradient zones where there is significant energy flow rate. In zones where there is little or no flow, the TEMP/W flow paths may not be realistic. 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.

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 energy flow rate 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 source of heat would move 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 energy flow rate vectors at an instant in time during a transient process have no physical meaning. For this reason, TEMP/W does not permit you to draw flow paths for transient conditions.

9.10 Flux sections

TEMP/W has the ability to compute the instantaneous total energy flow rate across a user-defined section for either a steady-state or transient analysis. This is a very useful tool for isolating quantity of heat to or from specific regions of interest, and it can save you manually adding up individual nodal flows in the case of a heat source or sink that is comprised of many nodes.

Flux section theory

This value can be computed from the nodal temperatures and the coefficients of the finite element equation. For example, consider a mesh with only one element, as illustrated in Figure 9-13. 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 heat flow equation is presented. It can be re-written with the flux value isolated on one side as follows:

$$[K]\{T\}+[M]\frac{\Delta T}{\Delta t} = \{Q\}$$

In a steady-state analysis, the storage term $[M]\frac{\Delta T}{\Delta t}$ becomes zero, and the equation can be reduced to:

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

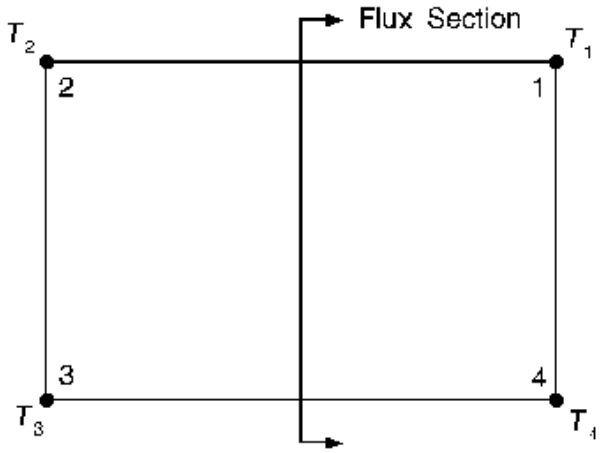


Figure 9-13 Illustration of a flux section across a single element

The global set of finite equations for one element is as follows:

Equation 9-1
$$\begin{bmatrix} c_{11} & c_{12} & c_{13} & c_{14} \\ c_{21} & c_{22} & c_{23} & c_{24} \\ c_{31} & c_{32} & c_{33} & c_{34} \\ c_{41} & c_{42} & c_{43} & c_{44} \end{bmatrix} \begin{Bmatrix} T_1 \\ T_2 \\ T_3 \\ T_4 \end{Bmatrix} = \begin{Bmatrix} Q_1 \\ Q_2 \\ Q_3 \\ Q_4 \end{Bmatrix}$$

From the basic heat flow equation, the total flow between two points is:

Equation 9-2
$$Q = k\, A \frac{\Delta T}{l}$$

The coefficients, c, in Equation 9-1 are a representation of $\frac{KA}{l}$ in Equation 9-2 Therefore, the flow from Node i to Node j is:

$$Q_{ij} = c_{ij} \left(T_i - T_j \right)$$

In a transient analysis, because of heat 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:

$$\frac{1}{\Delta t} \begin{bmatrix} m_{11} & m_{12} & m_{13} & m_{14} \\ m_{21} & m_{22} & m_{23} & m_{24} \\ m_{31} & m_{32} & m_{33} & m_{34} \\ m_{41} & m_{42} & m_{43} & m_{44} \end{bmatrix} \begin{Bmatrix} \Delta T_1 \\ \Delta T_2 \\ \Delta T_3 \\ \Delta T_4 \end{Bmatrix} = \begin{Bmatrix} Q_1 \\ Q_2 \\ Q_3 \\ Q_4 \end{Bmatrix}$$

where $\Delta T_{1,2,3,4}$ etc. are the changes of temperature at the various nodes between the start and the end of a time step. In general, the average change of temperature from Node i to Node j can be expressed as:

$$\Delta T_{ij} = \frac{\Delta T_i + \Delta T_j}{2}$$

Therefore, the change in flow quantity from Node i to Node j due to a change in storage is:

$$Q_{ij} = m_{ij} \frac{\Delta T_{ij}}{\Delta t}$$

The total flow quantity from Node i to Node j for a transient analysis then becomes:

$$Q_{ij} = c_{ij} \left(T_i - T_j \right) + m_{ij} \frac{\Delta T_{ij}}{\Delta t}$$

The total flow quantity through the flux section shown in Figure 9-13 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. TEMP/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 heat flow across the flux section.

Flux section application

Flux sections can be used in many ways because they can be drawn any place you want to know the flux across. You many want to check if an influx is equal to an outflux, such as illustrated in Figure 9-14. In this case the values are NOT the same, which is to be expected because the problem is a transient analysis and there is a change in the amount of heat storage over time. If this solution were allowed to solve for a much longer time frame, then the two fluxes would eventually come to a similar value, which would indicate that at that point in time, the solution has reached the equivalent of a steady-state condition.

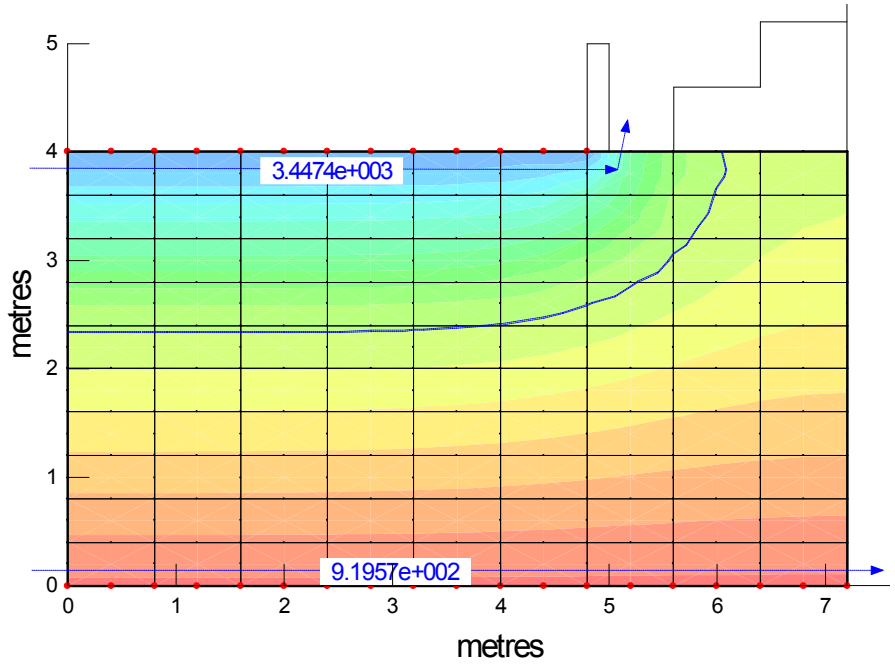


Figure 9-14 Flux section used to check balance of inflow and outflow

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 completely cross 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 SEEP/W seepage drain nodes in Figure 9-15, 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.

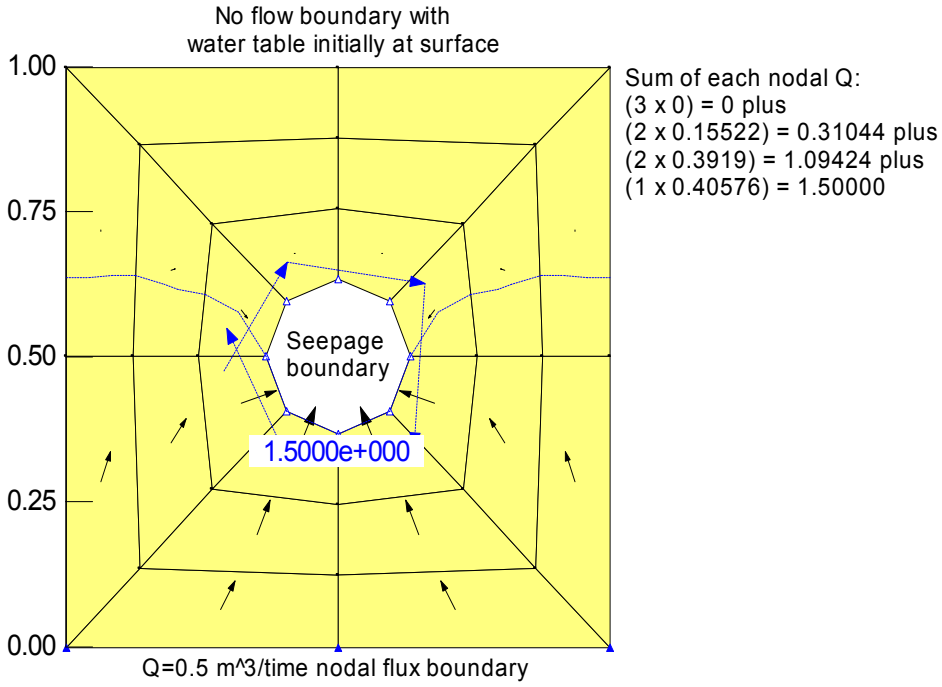


Figure 9-15 Flux section used in SEEP/W around series of drain nodes to check flow

10 Modeling Tips and Tricks

10.1 Introduction

This chapter contains many useful hints about using the software and understanding what it does. READ THIS CHAPTER!

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 ALSO in enhancing 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.

10.2 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 TEMP/W analysis; the only requirement is that you must be consistent. Fundamentally, you must select the units for length (geometry), time, heat and temperature. Once you have selected units for these parameters, all other units must be consistent. Table 10-1 to

Table 10-4 present some typical sets of consistent units.

Table 10-1 Consistent Set of Units in Meters, Seconds, Joules and Degrees Celsius

Parameter	Symbol	Units
Length	L	meters (m)
Time	t	seconds (sec)
Heat	H	Joules (J)
Temperature	T	°C
Latent heat of water	H/L^3	J/m^3
Thermal conductivity	$H/(t \times L \times T)$	$J/(sec \times m \times °C)$
Volumetric heat capacity	$H/(L^3 \times T)$	$J/(m^3 \times °C)$
Total flux (Q)	H/t	J/sec
Unit heat flux (q)	$H/(t \times L^2)$	$J/(sec \times m^2)$

Table 10-2 Consistent Set of Units in Meters, Days, Kilojoules Degrees Celsius

Parameter	Symbol	Units
Length	L	meters (m)
Time	t	days (day)
Heat	H	kilojoules (kJ)
Temperature	T	°C
Latent heat of water	H/L^3	kJ/m^3
Thermal conductivity	$H/(t \times L \times T)$	$kJ/(day \times m \times °C)$
Volumetric heat capacity	$H/(L^3 \times T)$	$kJ/(m^3 \times °C)$
Total flux (Q)	H/t	kJ/day
Unit heat flux (q)	$H/(t \times L^2)$	$kJ/(day \times m^2)$

Table 10-3 Consistent Set of Units in Feet, Hours, Btu and Degrees Fahrenheit

Parameter	Symbol	Units
Length	L	feet (ft)
Time	t	hours (hr)
Heat	H	Btu (Btu)
Temperature	T	°F
Latent heat of water	H/L^3	Btu/ft^3
Thermal conductivity	$H/(t \times L \times T)$	$Btu/(hr \times ft \times °F)$
Volumetric heat capacity	$H/(L^3 \times T)$	$Btu/(ft^3 \times °F)$
Total flux (Q)	H/t	Btu/hr
Unit heat flux (q)	$H/(t \times L^2)$	$Btu/(hr \times ft^2)$

Table 10-4 Consistent Set of Units in Meters, Hours, Kilocalories and Degrees Celsius

Parameter	Symbol	Units
Length	L	meters (m)
Time	t	hours (hr)
Heat	H	kilocalories (kcal)
Temperature	T	°C
Latent heat of water	H/L^3	$kcal/m^3$
Thermal conductivity	$H/(t \times L \times T)$	$kcal/(hr \times m \times °C)$
Volumetric heat capacity	$H/(L^3 \times T)$	$kcal/(m^3 \times °C)$
Total flux (Q)	H/t	kcal/hr
Unit heat flux (q)	$H/(t \times L^2)$	$kcal/(hr \times m^2)$

NOTE: The unit of J/sec is sometimes expressed as Watts, (1 J/sec = 1 W).

The latent heat of water expressed in the above four example units are:

Latent heat of water = 334 x 106 J/m^3 = 334 x 103 kJ/m^3 = 8975 Btu/ft^3 = 79760 kcal/m^3

In summary, many systems of units can be used in TEMP/W. The key requirement is that the system of units be consistent. Generally, all units are established once you select the units for thermal conductivity. For example, if the thermal conductivity is chosen to be in kJ/day/m/°C, then length must be in meters, heat energy must be in kJ, time must be in days and temperature must be in °C.

10.3 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 10-1 shows this to be the case, and it is true for a transient and steady-state solution.

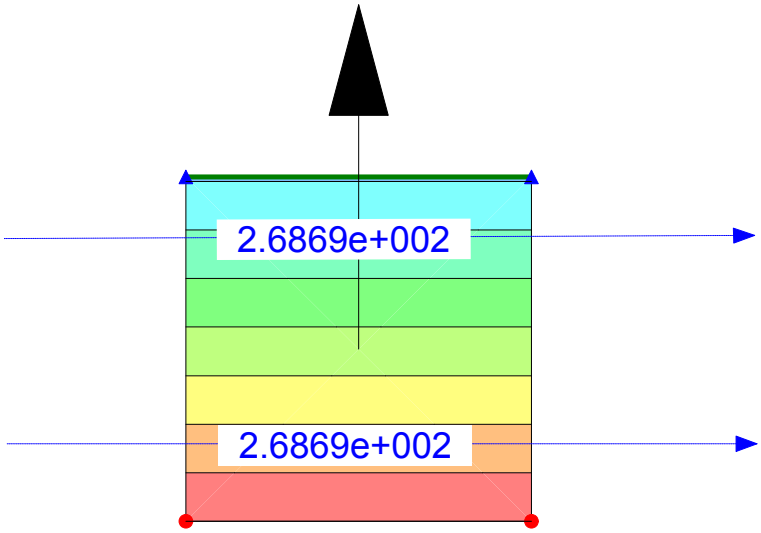


Figure 10-1 Test to check flux section locations

10.4 Source or sink flux values

Question: What is the best way to compute a total heat sink 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 -500 kJ per time unit (each node contributing a total nodal flux of -125 kJ). This example also shows that with no other place for the heat to flow except the sink, the flux into the base equals the flux out of the sink nodes.

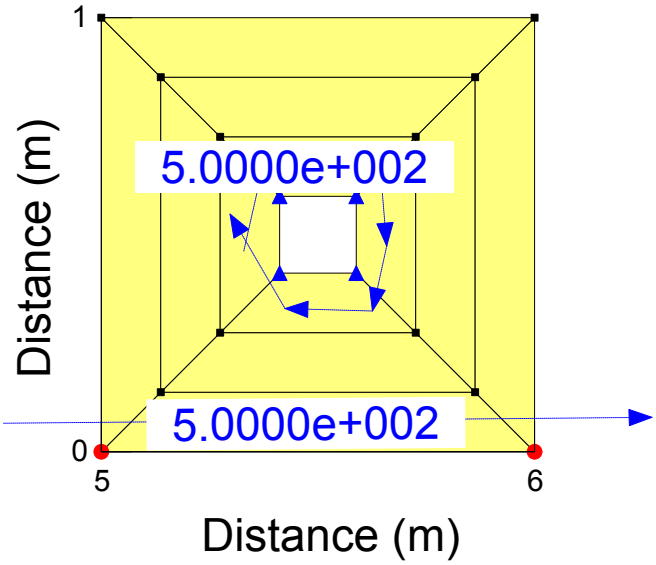


Figure 10-2 Test to see best method for observing sink flux

10.5 Unit energy flux versus total energy 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 -500 kJ / time / meter edge length has been applied to the top of the element. The top is a heat sink face which will let the energy out. The flux sections drawn in the element confirm that the total edge flux of -500 kJ / time has been converted by the solver into two equal total nodal fluxes of -250 kJ / time each. For such a simple mesh, it is also possible to use the View Node information option and click on each node to see the computed total energy flux at each node. The sum of the individual total nodal fluxes is the total energy flux across the element edge.

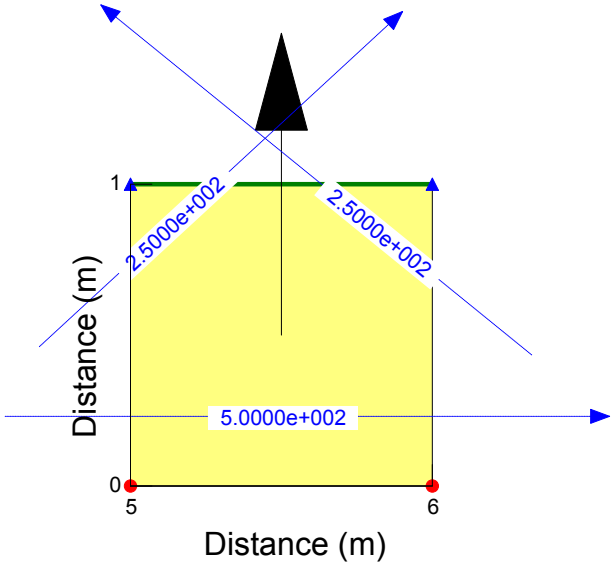


Figure 10-3 Test to compare unit flux and total flux

10.6 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 10-4 shows the nodes representing the location and boundary condition of freeze pipes in a shaft freeze project. The objective is to know how much heat is removed by all pipes over time. One option is to draw a flux section around each pipe 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 heat storage within the element. The most accurate way is to plot cumulative heat flow at each boundary node.

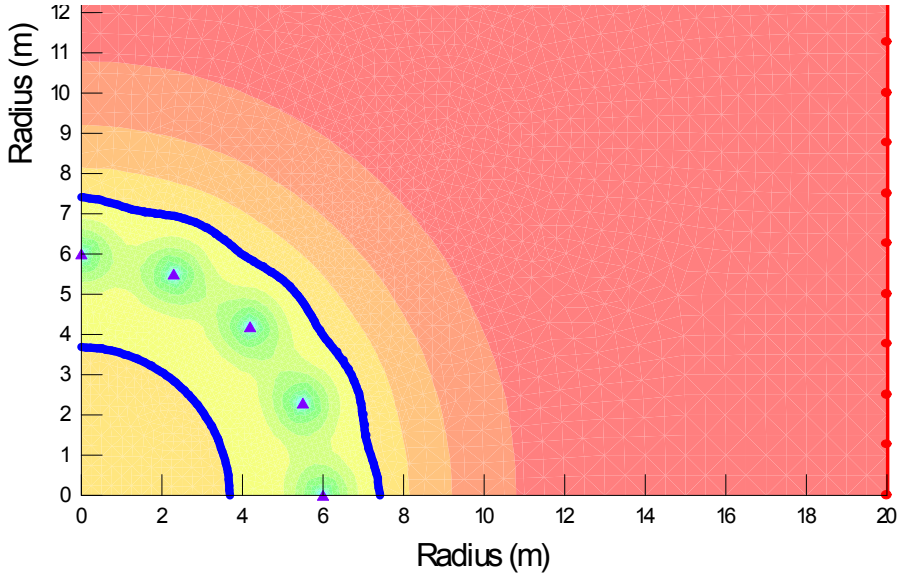


Figure 10-4 Graphing nodes selected on upstream dam face

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.

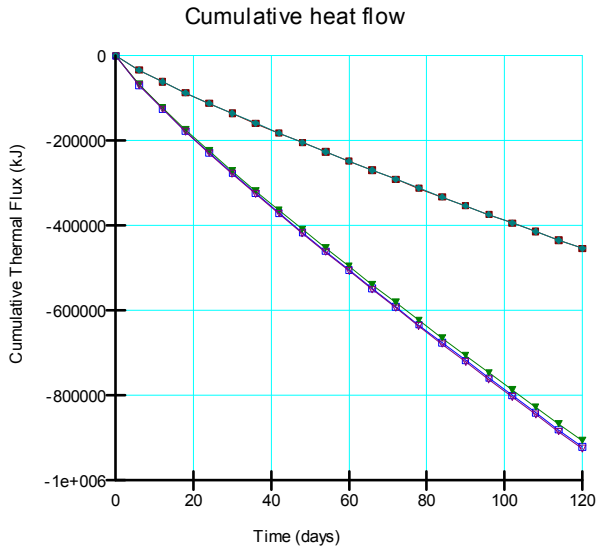


Figure 10-5 Individual nodal heat flow at freeze pipes

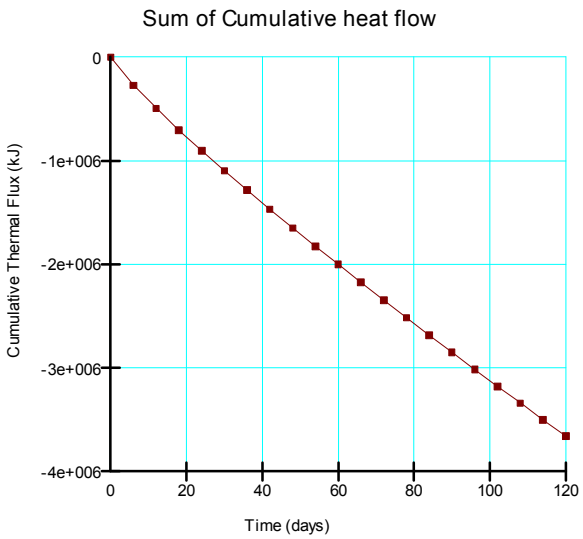


Figure 10-6 Sum of all nodal heat flows at freeze pipes

11 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.

12 Theory

This chapter describes the theoretical engineering basis for the TEMP/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.

12.1 Flow law

The principal mechanism for heat flow in soils in most engineering applications involving freezing and thawing is conduction. (see Harlan and Nixon, 1978) Conduction is the flow of heat by the passage of energy from one soil particle to another or through soil pore fluids. An option for convective heat transfer is also available in TEMP/W and is discussed towards the end of this chapter.

TEMP/W assumes that heat flux due to conduction is governed by the following equation:

Equation 12-1 $$q = -k\frac{\partial T}{\partial x}$$

where:

q = the heat flux,

k = the thermal conductivity,

T = the temperature, and

x = distance.

Equation 12-1 shows that heat flow due to conduction is directly dependent on the thermal conductivity of the soil medium and temperature gradient. The negative sign indicates that the temperature decreases in the direction of increasing x; that is, the heat flows in the direction from high temperature to low temperature.

12.2 Governing equations

The governing differential equation used in the formulation of TEMP/W is:

$$\frac{\partial}{\partial x}\left(k_x\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k_y\frac{\partial T}{\partial y}\right)+Q=\lambda\frac{\partial T}{\partial t}$$

where:

T = temperature,

kx = thermal conductivity in the x-direction,

ky = thermal conductivity in the y-direction,

Q = applied boundary flux,

λ = capacity for heat storage, and

t = time.

This equation states that the difference between the heat flux entering and leaving an elemental volume of soil at a point in time is equal to the change in the stored heat energy.

Under steady-state conditions, the flux entering and leaving an elemental volume is the same at all times. The right side of the equation vanishes and the equation reduces to:

$$\frac{\partial}{\partial x}\left(k_x \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k_y \frac{\partial T}{\partial y}\right) + Q = 0$$

The capacity to store heat is composed of two parts. The first part is the volumetric heat capacity of the material (either frozen or unfrozen) and the second part is the latent heat associated with the phase change. In equation form:

$$\lambda = c + L\frac{\partial w_u}{\partial T}$$

where:

c = volumetric heat capacity (material property),

L = latent heat of water,

Θu = total unfrozen volumetric water content,

T = temperature, and

λ = capacity for heat storage.

The volumetric heat capacity c is the slope of the energy curve in the frozen and unfrozen zones while the term $L\frac{\partial w_u}{\partial T}$ represents the rate of change of the latent heat component.

When the unfrozen water content function of a soil is defined, the total unfrozen volumetric water content can be expressed as:

$$w_u = W_u\, w$$

where:

Θr = unfrozen water content ($0 \le \Theta r \le 1$), and

Θ = volumetric water content of the soil.

Substituting for Θu for λ in the main thermal equation leads to the complete differential equation:

Equation 12-2 $$\frac{\partial}{\partial x}\left(k_x \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k_y \frac{\partial T}{\partial y}\right) + Q = \left(c + Lw\frac{\partial W_u}{\partial T}\right)\frac{\partial T}{\partial t}$$

12.3 Finite element heat flow equations

Applying the Galerkin method of weighed residual to the governing differential equation, the finite element for two-dimensional thermal equation can be derived as:

Equation 12-3

$$\tau\int_A \left([B]^T\ [C]\ [B]\right) dA\ \{T\} + \tau\int_A \left(\lambda <N>^T\ <N>\right) dA\ \{T\},t$$
$$= q\tau\int_L \left(<N>^T\right) dL$$

where:

[B]	=	the gradient matrix,
[C]	=	the element thermal conductivity matrix,
{T}	=	the vector of nodal temperatures,
<N>	=	the vector of interpolating function,
q	=	the unit flux across the edge of an element,
τ	=	the thickness of an element,
t	=	time,
λ	=	the capacity for heat storage,
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 radii, R about the symmetric axis. The complete circumferential distance is 2π radian times R, since TEMP/W is formulated for one radian, the equivalent thickness is R. Therefore, the finite element equation for the axisymmetric case is:

$$\int_A \left([B]^T\ [C]\ [B]R\right) dA\ \{T\} + \int_A \left(\lambda <N>^T\ <N>R\right) dA\ \{T\},t$$
$$= q\int_L \left(<N>^T R\right) dL$$

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 thermal equation can be expressed as:

Equation 12-4 $$[K]\{T\} + [M]\{T\},t = \{Q\}$$

where:

[K]	=	the element characteristic matrix,
[M]	=	the element heat matrix, and
{Q}	=	the element applied flux vector.

Equation 12-4 is the general finite element equation for a transient thermal analysis. For a steady-state analysis, the head is not a function of time and consequently the term $\{T\},t$ vanishes, reducing the finite element equation to:

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

which is the abbreviated finite element form of the fundamental flow equation.

12.4 Temporal integration

The finite element solution for a transient analysis is a function of time as indicated by the $\{T\},t$ 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):

$$\left(\omega\Delta t\left[K\right]+\left[M\right]\right)\{T_1\}=$$
$$\Delta t\left(\left(1-\omega\right)\{Q_0\}+\omega\{Q_1\}\right)+\left(\left[M\right]-\left(1-\omega\right)\Delta t\left[K\right]\right)\{T_0\}$$

where:

t = the time increment,

ω = a ratio between 0 and 1,

T_1 = the temperature at end of time increment,

T_0 = the temperature 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.

TEMP/W uses the Backward Difference Method, a method that sets ω to 1.0, the finite element equation is then simplified to:

Equation 12-5 $$\left(\Delta t\left[K\right]+\left[M\right]\right)\{T_1\}=\Delta t\{Q_1\}+\left[M\right]\{T_0\}$$

As indicated by Equation 12-5, in order to solve for the new temperature at the end of the time increment, it is necessary to know the temperature at the start of the increment. Stated in general terms, the initial conditions must be known in order to perform a transient analysis.

12.5 Numerical integration

TEMP/W uses Gaussian numerical integration to evaluate the element characteristic matrix $[K]$ and the heat 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 12-3):

$$[K]=\tau\int_A\left([B]^T\ [C]\ [B]\right)dA$$

can be replaced by:

$$[K]=\tau\sum_{j=1}^{n}\ [B_j]^T\ [C_j]\ \ \det|J_j|\ W_{1j}W_{2j}$$

where:

j = an integration point,

n = the number of integration points,

$[C_j]$ = the element thermal 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} = a weighting factor.

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 TEMP/W.

Table 12-1 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 12-2 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 12-3 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 12-4 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 12-5 gives the acceptable integration orders.

Table 12-5 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 unfrozen zones where the thermal 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 a frozen zone where the thermal conductivity varies sharply with changes in temperature. 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 thermal conductivity varies sharply within an element, it is best to use quadrilateral elements with secondary nodes together with nine-point integration.

12.6 Thermal conductivity matrix

The general form of the TEMP/W element thermal conductivity matrix is:

$$[C] = \begin{bmatrix} C_{11} & C_{12} \\ C_{21} & C_{22} \end{bmatrix}$$

where:

$$C_{11} = kx\cos^2\alpha + ky\sin^2\alpha$$

$$C_{22} = kx\sin^2\alpha + ky\cos^2\alpha$$

$$C_{12} = kx \sin\alpha \cos\alpha + ky \sin\alpha \cos\alpha$$

$$C_{21} = C_{12}$$

The parameters k_x, k_y and α are defined in Figure 12-1.

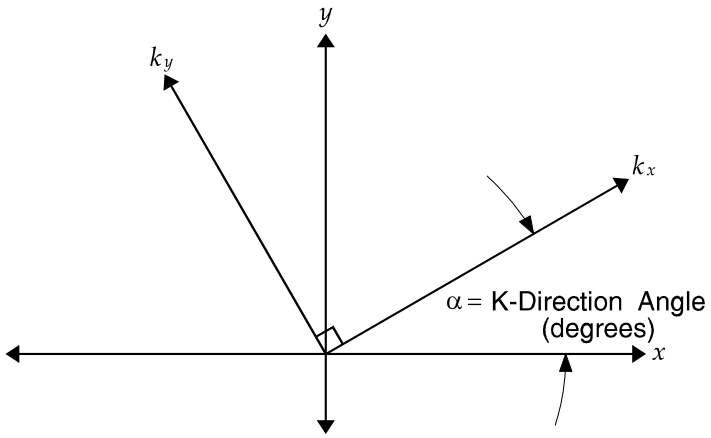


Figure 12-1 Definition of thermal conductivity matrix parameters

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 thermal conductivity function. Parameter k_y is then computed as k_x multiplied by k_{Ratio}. In equation form:

$$k_y = k_x \times k_{Ratio}$$

12.7 Heat storage matrix

As first presented in Equation 12-3 the element heat (or storage) matrix for a two-dimensional analysis is defined as:

$$[M] = \tau \int_A \left(\lambda < N >^T < N >\right) dA$$

Similar to the element characteristic matrix, the heat storage matrix is also evaluated by numerical integration as shown below:

$$[M] = \tau \sum_{j=1}^{n} \lambda_j <\mathrm{N}>^T <\mathrm{N}> \det|J_j| W_{1j} W_{2j}$$

TEMP/W uses a consistent formulation, as opposed to a lumped formulation, to establish the capacitance matrix. The two methods are presented and discussed by Segerlind (1984). The adoption of the consistent

formulation is necessitated by the generality of TEMP/W which allows the elements to have all, some, or none of the secondary nodes.

The capacitance matrix is formed by numerical integration as discussed in this chapter. This has the advantage of making it possible for the parameter λ to vary throughout the element. At each integration point, TEMP/W obtains a value of λ based on the latent heat of water, the volumetric water content, the volumetric heat capacity, and the slope of the unfrozen water content function.

TEMP/W computes the slope of the unfrozen water content function from a straight line between the previously-computed temperature and the newly-computed temperature at a Gauss point, as illustrated in Figure 12-2.

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 temperature values are nearly identical. In this case, TEMP/W computes the slope by calculating the derivative of the function at the average of the two temperature values.

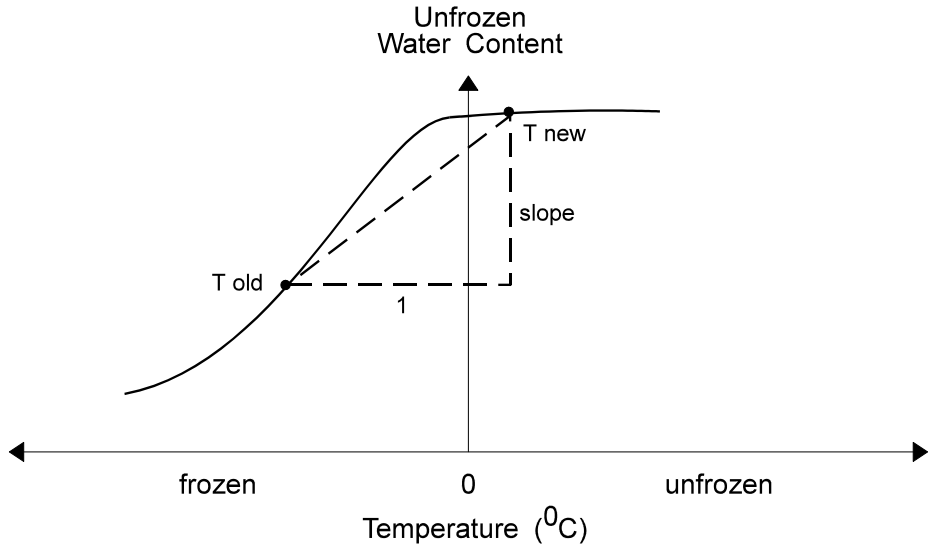


Figure 12-2 Computation of heat storage term

12.8 Flux boundary vector

The nodal flux boundary vector $\{Q\}$ for a two-dimensional analysis is defined as:

$$\{Q\} = q\tau \int_L \left(<N>^T\right) dL$$

or for an axisymmetric analysis as:

$$\{Q\} = q \int_L \left(<N>^T R\right) dL$$

and for a plan view analysis as:

$$\{Q\} = q \int_A \left(<N>^T < N >\right) dA$$

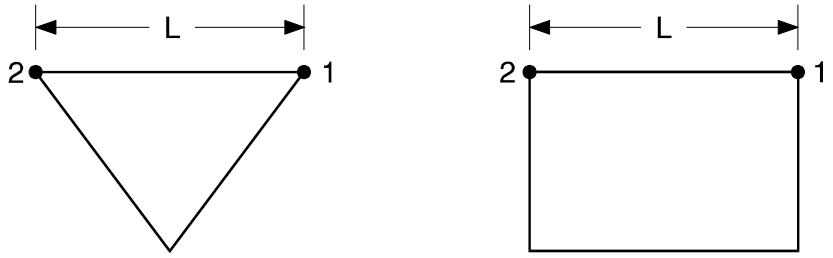
where:

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 12-3 and Figure 12-4. However, for plan view analysis, the contributing area of a node is computed by numerical integration in the same way as forming the heat 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.

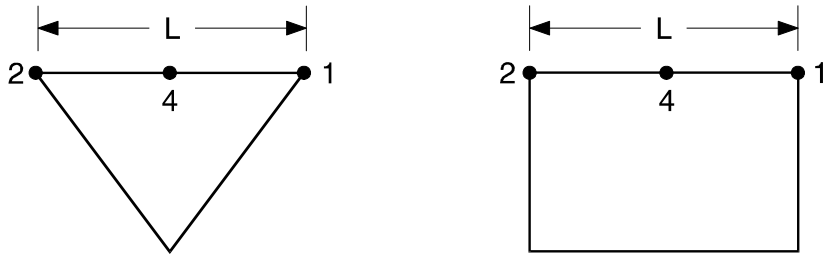
Two types of flux boundaries may be specified in TEMP/W namely: a nodal flux boundary (Q) and a unit flux boundary (q). A nodal flux boundary (Q) can be specified directly on the boundary nodes. A unit flux boundary (q) must be specified along the boundary edges of the elements, except for a plan view 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 or top view section in 2D) 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

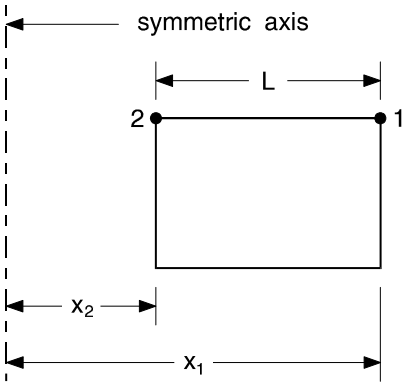


$$a_1 = a_2 = \frac{L}{6}$$

$$a_4 = \frac{4L}{6}$$

(b) With an intermediate node

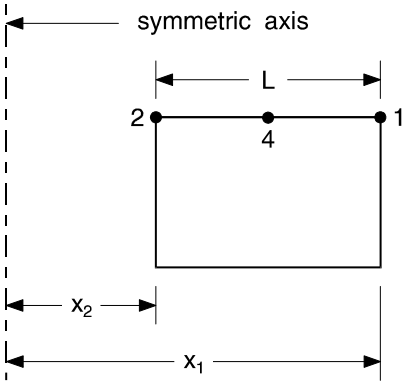
Figure 12-3 Contributing area for section view elements with unit thickness



$$a_2 = \frac{L}{6}(2x_2 + x_1)$$

$$a_1 = \frac{L}{6}(x_2 + 2x_1)$$

(a) No intermediate node



$$a_2 = \frac{L}{6}(x_2)$$

$$a_4 = \frac{2L}{6}(x_2 + x_1)$$

$$a_1 = \frac{L}{6}(x_1)$$

(b) With an intermediate node

Figure 12-4 Contributing area for axisymmetric elements over unit Radian thickness

12.9 Convective heat transfer with SEEP/W or AIR/W

TEMP/W has the ability to integrate with SEEP/W (AIR/W) in order to take into account the convective heat transfer that occurs due to flowing water. In order to include convective heat transfer effects, TEMP/W must obtain the water and air content and water and air velocity at every Gauss point for every time step.

With this information, the partial differential equation for heat flow (Equation 12-2) is modified to:

$$(\rho_s c_{ps} + L\theta_w \frac{\partial \theta_u}{\partial T})\frac{\partial T}{\partial t} = \frac{\partial}{\partial y}\left[K_t \frac{\partial T}{\partial y} \right] + c_{pa} \frac{\partial (\dot{m}_a T)}{\partial y} + \rho_w c_{pw} \frac{\partial (q_w T)}{\partial y} + Q$$

Where

$\rho_s c_{ps}$ = volumetric heat capacity of soil,

$c_{pa/w}$ = mass specific heat of air or water,

$\dot{m}_a$ = mass flow rate of air,

$\frac{\partial \theta_u}{\partial T}$ = the slope of the unfrozen water content function,

q_w = the specific discharge (Darcy velocity) of water, and

L = latent heat of water.

13 Appendix A: Interpolating Functions

13.1 Coordinate systems

The global coordinate system used in the formulation of TEMP/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 13-1. Presented as well in Figure 13-1 is the local element node numbering system. The local coordinates for each of the nodes are given in Table 13-1.

Table 13-1 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	½	0
	6	½	½
	7	0	½
	8	–	–

TEMP/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. TEMP/W uses the same functions for relating the coordinate systems as for describing the variation of the field variable (temperature) within the element. The elements are consequently isoperimetric elements.

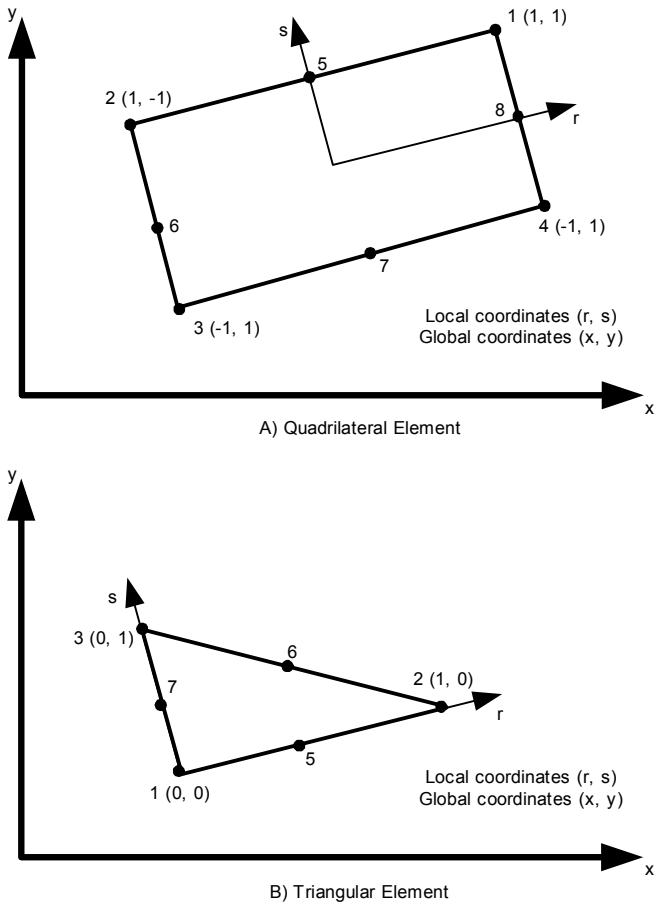


Figure 13-1 Global and local coordinate system

The x and y coordinates anywhere in the element are related to the local coordinates and to the x y coordinates of the nodes by the following equations:

Equation 13-1
$$x = \langle N \rangle \{X\}$$
$$y = \langle N \rangle \{Y\}$$

where:

<N> = 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 13-1.

13.2 Interpolating functions

TEMP/W uses a general set of interpolating functions presented by Bathe (1982, pp. 200, 230). 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 13-2 and Table 13-3, 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 13-2 Interpolation functions for quadrilateral elements

Function	Include in function if node is present			
	5	6	7	8
N1 = ¼(1+r)(1+s)	-½N5	—	—	-½N8
N2 = ¼(1-r)(1+s)	-½N5	-½N6	—	—
N3 = ¼(1-r)(1-s)	—	-½N6	-½N7	—
N4 = ¼(1+r)(1-s)	—	—	-½N7	-½N8
N5 = ½(1-r2)(1+s)	—	—	—	—
N6 = ½(1-s2)(1-r)	—	—	—	—
N7 = ½(1-r2)(1-s)	—	—	—	—
N8 = ½(1-s2)(1+r)	—	—	—	—

Table 13-3 Interpolation functions for triangular elements

Function	Include in function if node is present		
	5	6	7
N1= 1-r-s	-½N5	—	-½N7
N2 = r	-½N5	-½N6	—
N3 = s	—	-½N6	-½N7
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 thermal analysis is the temperature (T), it is necessary to adopt a model for the distribution of T within the element.

TEMP/W assumes that the temperature distribution within the element follows the adopted interpolating functions. This means that the temperature distribution is linear when the secondary nodes are missing, and the temperature distribution is nonlinear when the secondary nodes are present.

In equation form the temperature distribution model is:

Equation 13-2 $$t = \langle N \rangle \{T\}$$

where:

t = the temperature at any local coordinate,

<N> = a vector of interpolation function, and

{T} = a vector of temperatures at the nodes.

Interpolation function derivatives

The constitutive relationship for a conductive thermal analysis is Darcy's type flow 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 TEMP/W to compute the gradient.

From the adopted temperature distribution model, the temperature at any point within the element in terms of the nodal temperatures is obtained from Equation 13-2.

The gradients in the x and y directions are then known by:

Equation 13-3
$$i_x = \frac{\partial t}{\partial x} = \frac{\partial \langle N \rangle}{\partial x}\{T\}$$
$$i_y = \frac{\partial t}{\partial y} = \frac{\partial \langle N \rangle}{\partial y}\{T\}$$

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:

$$\begin{Bmatrix} \frac{\partial \langle N \rangle}{\partial r} \\ \frac{\partial \langle N \rangle}{\partial s} \end{Bmatrix} = [J] \begin{Bmatrix} \frac{\partial \langle N \rangle}{\partial x} \\ \frac{\partial \langle N \rangle}{\partial y} \end{Bmatrix}$$

where [J] is the Jacobian matrix and is defined as:

Equation 13-4 $$[J] = \begin{bmatrix} \frac{\partial x}{\partial r} & \frac{\partial y}{\partial r} \\ \frac{\partial x}{\partial s} & \frac{\partial y}{\partial s} \end{bmatrix}$$

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:

$$[B]=\begin{Bmatrix}\dfrac{\partial\langle N\rangle}{\partial x}\\ \dfrac{\partial\langle N\rangle}{\partial y}\end{Bmatrix}=[J]^{-1}\begin{Bmatrix}\dfrac{\partial\langle N\rangle}{\partial r}\\ \dfrac{\partial\langle N\rangle}{\partial s}\end{Bmatrix}$$

Recalling Equation 13-1 and taking their derivative with respect to s and r as follows:

$$\frac{\partial x}{\partial r}=\frac{\partial\langle N\rangle}{\partial r}\{X\}$$
$$\frac{\partial x}{\partial s}=\frac{\partial\langle N\rangle}{\partial s}\{X\}$$
$$\frac{\partial y}{\partial r}=\frac{\partial\langle N\rangle}{\partial r}\{Y\}$$
$$\frac{\partial x}{\partial s}=\frac{\partial\langle N\rangle}{\partial s}\{Y\}$$

then substituting these values into Equation 13-4, the Jacobian matrix becomes:

$$[J]=\begin{bmatrix}\dfrac{\partial(N_1,N_2\ldots N_8)}{\partial r}\\ \dfrac{\partial(N_1,N_2\ldots N_8)}{\partial s}\end{bmatrix}\begin{bmatrix}X_1 & Y_1\\ X_2 & Y_2\\ \vdots & \vdots\\ X_8 & Y_8\end{bmatrix}$$

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 13-3

The derivatives of the interpolation functions with respect to r and s used by TEMP/W for quadrilateral and triangular elements are given in Table 13-4 and

Table 13-5 respectively.

Table 13-4 Interpolation function derivatives for quadrilateral elements

Derivative of Function	Include in derivative if node is present			
	5	6	7	8
N1,r = ¼(1+s)	-½(N5,r)	—	—	`
N2,r = -¼(1+s)	-½(N5,r)	-½(N6,r)	—	—
N3,r = -¼(1-s)	—	-½(N6,r)	-½(N7,r)	—
N4,r = ¼(1-s)	—	—	-½(N7,r)	-½(N8,r)
N5,r = -½(2r+2sr)	—	—	—	—
N6,r = -½(1-s2)	—	—	—	—
N7,r = -½(2r-2sr)	—	—	—	—
N8,r = ½(1-s2)	—	—	—	—
N1,s = ¼(1+r)	- ½(N5,s)	—	—	-½(N8,s)
N2,s = ¼(1-r)	-½(N5,s)	-½(N6,s)	—	—
N3,s = -¼(1-r)	—	-½(N6,s)	-½(N7,s)	—
N4,s = -¼(1+r)	—	—	-½(N7,s)	-½(N8,s)
N5,s = ½(1-r2)	—	—	—	—
N6,s = -½(2s-2sr)	—	—	—	—
N7,s = -½(1-r2)	—	—	—	—
N8,s = -½(2s+2sr)	—	—	—	—

Table 13-5 Interpolation function derivatives for triangular elements

Derivative of Function	Include in derivative if node is present		
	5	6	7
N1,r = -1.0	-½(N5,r)	—	—
N2,r = 1.0	-½(N5,r)	-½(N6,r)	—
N3,r = 0.0	—	-½(N6,r)	-½(N7,r)
N5,r = (4-8r-4s)	—	—	—
N6,r = 4s	—	—	—
N7,r = -4s	—	—	—
N1,s = -1.0	-½(N5,s)	—	—
N2,s = 0.0	-½(N5,s)	-½(N6,s)	—
N3,s = 1.0	—	-½(N6,s)	-½(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

Bathe, K-J., 1982. Finite Element Procedures in Engineering Analysis. Prentice-Hall.

Bettess, P., 1992. Infinite Elements. Penshaw Press.

Black, P. and Allen Tice, 1989. comparison of Soil Freezing Curve and Soil Water Curve Data for Windsor Sandy Loam. Water Resources Research, Vol. 25, No. 10. pp. 2205-2210.

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, Sloan, S.W. 2000. Computing and Computer Modeling in Geotechnical Engineering, Invited Paper, Conference Proceedings, GeoEng2000, Melbourne, Australia.

Carslaw, H.S, and Jaeger, J.C., 1947. Conduction of Heat in Solids. Clarendon Press, Oxford

Coutts, R.J and Konrad, J.M., 1994. Finite Element Modeling of Transient Non-Linear Heat Flow Using the Node State Method. Intl. Ground Freezing Conf., France, November.

Edlefsen, N.E. & Anderson, A.B.C., 1943. Thermodynamics of soil moisture. Hilgardia 15: 2: pp. 31-298.

Farouki, O.T. 1981. Thermal Properties of Soils. CRREL Monograph 81-1. U.S. Army Corps of Engineers, Cold Regions Research and Engineering Laboratory, Hanover, New Hampshire.

Flerchinger, G.N. and K.E. Saxton. 1989. Simultaneous heat and water model of a freezing snow-residue-soil system I. Theory and development. Trans. of ASAE, 32(2), 565-571.

Goodrich, L.E., and Gold, L.W., 1981. Ground Thermal Analysis: Permafrost Engineering Design and Construction. Edited by Johnston, G.H. Ch. 4. John Wiley & Sons.

Harlan, R.L., and Nixon, J.F., 1978. Ground Thermal Regime. Geotechnical Engineering For Cold Regions, Edited by O.B. Andersland and D.M. Anderson. McGraw-Hill Inc.

Hwang, C.T., Murray, D.W., and Brooker, E.W, 1972. A Thermal Analysis for Structures on Permafrost. Canadian Geotechnical Journal, Vol. 9, No. 1.

Johansen, O., 1975. Thermal Conductivity of Soils. Ph.D. thesis. Trondheim, Norway. (CRREL Draft Translation 637, 1977), ADA044002.

Johnston, G.H., Ladanyi, B., Morgenstern, N.R., and Penner, E., 1981. Engineering Characteristics of Frozen and Thawing Soils. Permafrost Engineering Design and Construction. Edited by Johnston, G.H. John Wiley & Sons.

Liang, X., D. P. Lettenmaier, E. F. Wood, and S. J. Burges, A Simple hydrologically Based Model of Land Surface Water and Energy Fluxes for GSMs, J. Geophys. Res., 99(D7), 14,415-14,428, 1994.

Lancaster, P., and Salkauskas, K., 1986. Curve and Surface Fitting: An Introduction. Academic Press.

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.

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 Modeling 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, University of Saskatchewan, Saskatoon SK, Canada.

Newman, G. P. and G. W. Wilson (1997) “Heat and Mass Transfer In Unsaturated Soils during Freezing.” Canadian Geotechnical Journal, 34: 63-70.

Newman, G. and D. Maishman. Underground Freezing at a High Grade Uranium Mine. Ground Freezing 2000: Frost Action in Soils Proceedings of the 9th International Symposium, Louvain-La-Neuve, Belgium, 11-13 September 2000.

O’Kane Consultants Inc. 2004. Personal communication.

Nixon, J.F., and McRoberts, E.C., 1973. A Study of Some Factors Affecting the Thawing of Frozen Soils. Canadian Geotechnical Journal, Vol. 10.

Salkauskas, K., 1974. C' Splines For Interpolation of Rapidly Varying Data. Rocky Mountain Journal of Mathematics, Vol. 14, No. 1.

Segerlind, L.J., 1984. Applied Finite Element Analysis. John Wiley and Sons.

Wilson, G.W., 1990. Soil Evaporative Fluxes for Geotechnical Engineering Problems. Ph.D. Thesis, University of Saskatchewan, Saskatoon, Canada.

Index

adaptive time stepping 113
Analysis Types.....93
Artificial ground freezing application 83
Axisymmetric.....98
Boundary condition locations 73
Boundary Conditions 71
Boundary functions.....88
buried piping 4
Burland.....15
chilled pipelines 6
chilled structures 5
Closed form curve fits.....106
Compare alternatives 18
Contours.....124
Convective cooling of surfaces.....7
Convective heat transfer 150
Convective heat transfer surfaces ..82
Convergence 108
Element and mesh compatibility....49
Element fundamentals.....48
Element nodes.....48
Energy flow vectors 126
engineering units.....131
Equation solvers 112
estimated material properties 28
Field variable distribution 48
Finite element heat flow equations 141
Flow law.....140
Flow paths.....127
Flux boundary vector 147
Flux section application 129
Flux section location 133
Flux section theory.....127
Flux sections 127
Gauss integration 111
General guidelines for meshing 52
Governing equations 140
gradients and velocities.....125
Graphing 118
Ground surface temperatures 77
heat capacity.....63
Heat flow boundary conditions.....74

Heat storage matrix 146
How not to model.....29
How to model.....23
Illustrative Examples 139
Initial conditions 94
Interface Elements 36
Interface model 58
Interpolating functions 155
Interpolating Functions 153
Joining regions 45
Make a guess.....23
Material Properties.....57
Meshing.....31
Meshing for transient analyses.....46
Modeling Tips and Tricks.....131
Modifier functions 90
Nodal and unit heat flux versus time 90
Node and element information.....116
numerical experiments 26
Numerical integration 51, 143
Numerical Issues.....108
Numerical Modeling – What, Why and How 11
Plan view.....100
Pyrex tile cooling application 85
Quantitative predictions 17
Region face boundary conditions...74
roads and airstrips 1
Secondary variables 52
Simplify geometry 24
snow pack.....79
soil stabilization 2
Source or sink flux values.....133
Sources and sinks.....77
Spatial functions.....106
Start simple 26
Steady state 93
Structured mesh 40
Surface mesh.....42
Temperature boundary conditions .74
Temperature versus time.....89
Temporal integration.....143
Theory 140
Thermal conductivity matrix.....145
Thermal functions 58
Thermosyphon boundary conditions 86

Time stepping.....95, 113
train our thinking.....20
Transient 93
Triangular regions.....41
Unfrozen water content.....58
Unit energy flux versus total energy flux? 134
Unstructured mesh 40
Visualization of Results 116
Why model.....16