Stress-Deformation Modeling with SIGMA/W

An Engineering Methodology

July 2013 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

SIGMA/W is a finite element software product that can be used to perform stress and deformation analyses of earth structures. Its comprehensive formulation makes it possible to analyze both simple and highly complex problems. For example, you can perform a simple linear elastic deformation analysis or a highly sophisticated nonlinear elastic-plastic effective stress analysis. When coupled with other GEO-SLOPE software products, it can also model the pore-water pressure generation and dissipation in a soil structure in response to external loads using either a fully coupled or un-coupled formulation. SIGMA/W has application in the analysis and design for geotechnical, civil, and mining engineering projects.

1.1 Applications

SIGMA/W can be used to compute stress-deformation with or without the changes in pore-water pressures that arise from stress state changes. In addition, it is possible to model soil structure interaction using beam or bar elements. The following are some typical cases that can be analyzed using SIGMA/W.

Deformation analysis

The most common application of SIGMA/W is to compute deformations caused by earthworks such as foundations, embankments, excavations and tunnels. Figure 1-1 shows a typical case of a fluid-filled tank on the ground surface. This figure presents the deformation at an exaggerated scale as a deformed mesh. Figure 1-2 shows the associated change in vertical stress in the ground caused by the applied load.

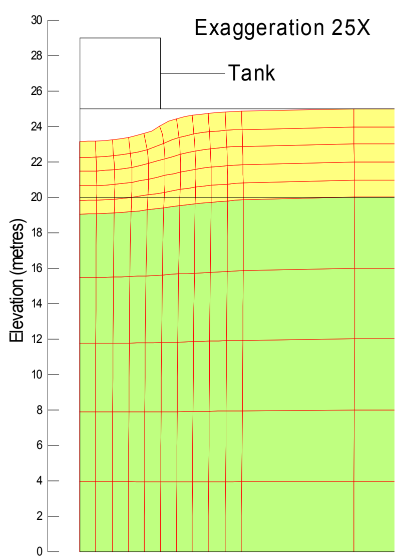


Figure 1-1 Deformation under a fluid-filled tank

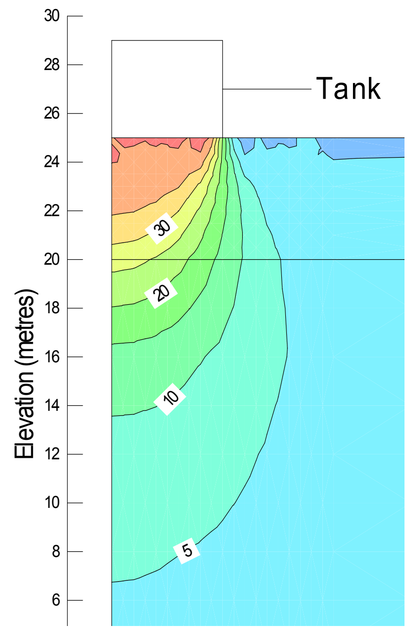


Figure 1-2 Vertical stress change under a fluid-filled tank

Staged construction

Soil regions can be activated or deactivated at different stages in an project file, making it possible to simulate the process over time. GeoStudio uses the concept of multiple analyses in a single project file. By automatically linking these analyses together on a common time continuum, you can simulate construction sequences. You can also apply boundary conditions that move with time. Figure 1-3 shows the sequencing of excavation and lowering seepage face location. Figure 1-5 shows the closure around a tunnel in anisotropic linear-elastic material.

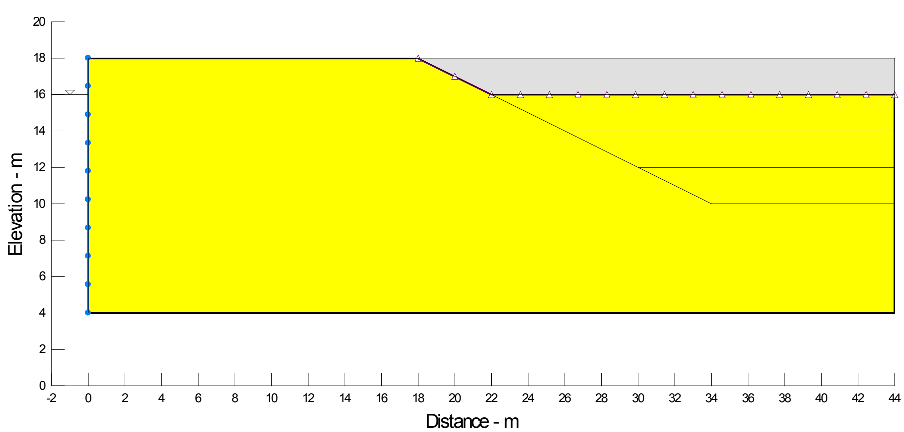


Figure 1-3 First excavation and lowered hydraulic BC

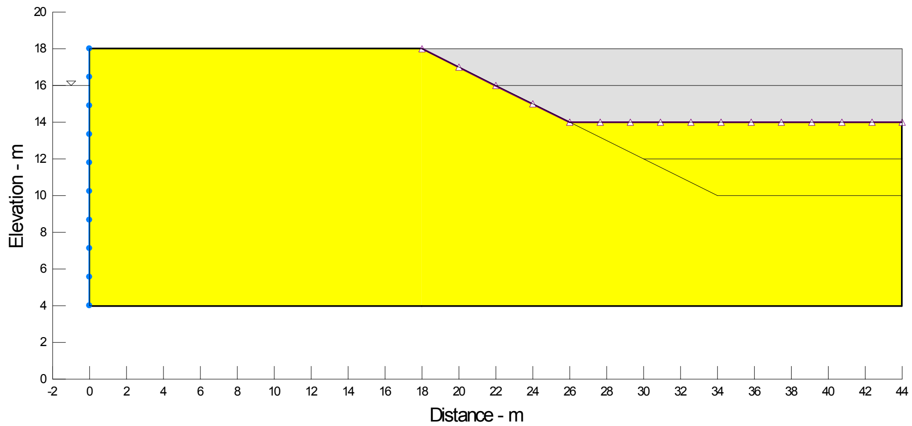


Figure 1-4 Second excavation and lowered hydraulic BC

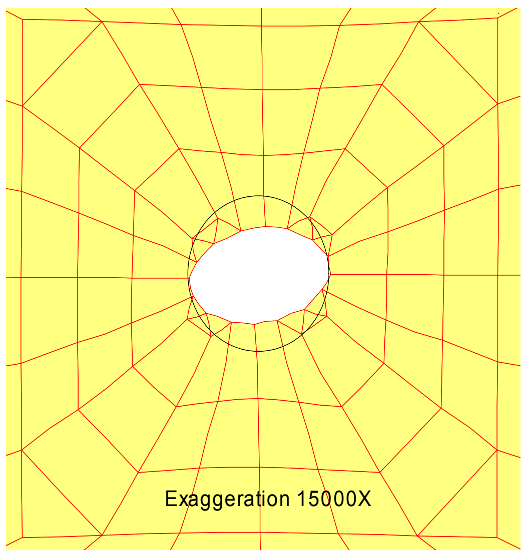


Figure 1-5 Closure around a tunnel

Excess pore-water pressures

The effect of excess pore-water pressures generated during fill placement is often a major consideration in slope stability during construction. SIGMA/W can be used to estimate these types of pore-water pressures. Excess pore-water pressures computed using SIGMA/W may be imported into SLOPE/W for slope stability analysis. Figure 1-6 shows the excess pore-water pressures in the foundation beneath an embankment immediately after construction. Note the use of infinite elements for modeling the right and left boundaries.

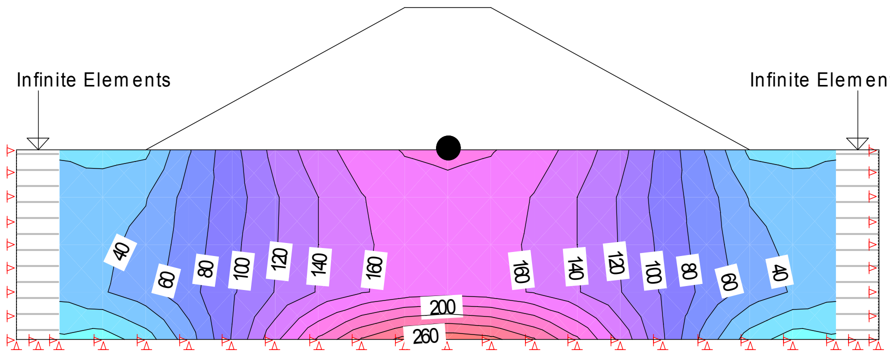


Figure 1-6 Excess pore-water pressures caused by fill placement

Soil-structure interactions

SIGMA/W can accommodate soil-structure interaction problems by including structural elements in two-dimensional plain strain analyses. These elements can be beam elements which have flexural stiffness, or bar elements which have only axial stiffness with no flexural stiffness. These structural elements are particularly useful when analyzing cases such as sheet-pile walls. Figure 1-7 illustrates the deformation for such a case, and Figure 1-8 shows the associated moment distribution in the sheet-pile wall.

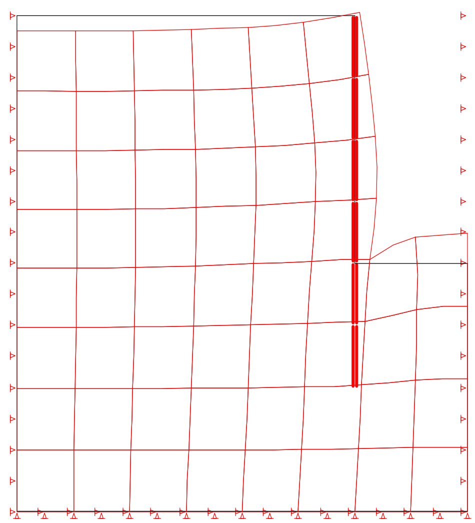


Figure 1-7 Sheet pile wall example

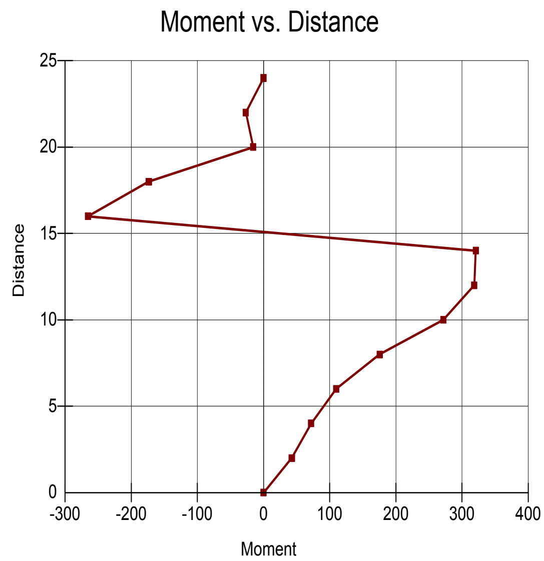


Figure 1-8 Moment distribution in a sheet pile wall

Consolidation analyses

SIGMA/W can be used together with SEEP/W to perform a fully-coupled consolidation analysis. When these two integrated products are run simultaneously, SIGMA/W calculates the deformations resulting from pore-water pressure changes while SEEP/W calculates transient pore-water pressure changes. This procedure is used to simulate the consolidation process in both saturated and unsaturated soils.

A fully-coupled analysis is required to correctly model the pore-water pressure response to an applied load. In certain cases, the pore-water pressure increase under an applied load can be greater than the applied load. This phenomenon is known as the Mendel-Cryer effect. Figure 1-9 shows a SIGMA/W analysis of a saturated triaxial sample with an applied lateral load of 100 kPa. The initial pore-water pressure before loading is zero. Figure 1-10 shows the pore-water pressure response with time at the center of the sample. The pore-water pressure rises to about 110 kPa, (110% of the applied load), before it gradually decreases.

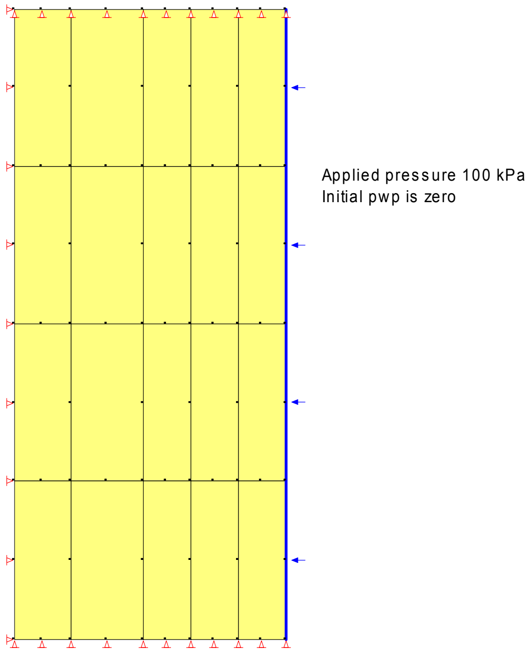


Figure 1-9 Tri-axial sample with applied lateral load

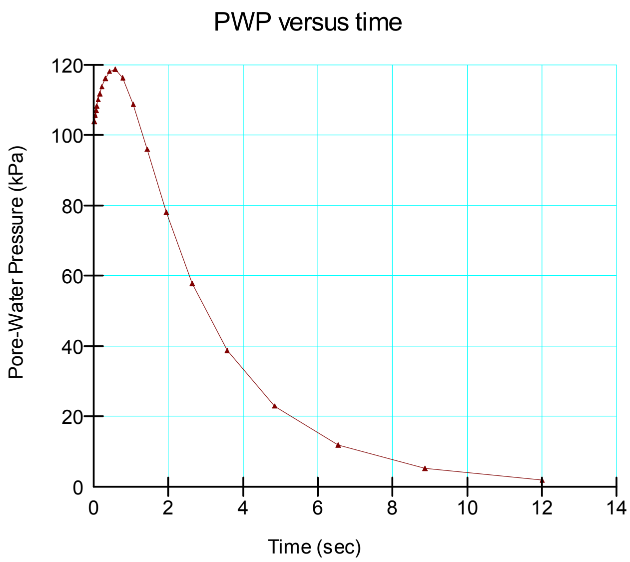


Figure 1-10 Pore-water pressure change with time

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

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

Numerical modeling as a field of practice is relatively new in geotechnical engineering and, consequently, there is a lack of understanding about what numerical modeling is, how modeling should be approached and what to expect from it. A good understanding of these basic issues is fundamental to conducting effective modeling. Basic questions such as, What is the main objective of the analysis?, What is the main engineering question that needs to 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. It should be noted now that some of the examples in this chapter pertain to stress-deformation analysis while others relate to seepage analysis. While the actual type of analyses may differ, the concepts illustrated apply to all engineering analyses.

2.2 What is a numerical model?

A numerical model is a mathematical simulation of a real physical process. SIGMA/W is a numerical model that can mathematically simulate the real physical process of ground volume change in response to self or external loading. Numerical modeling is purely mathematical and in this sense is very different than scaled physical modeling in the laboratory or full-scaled field modeling.

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

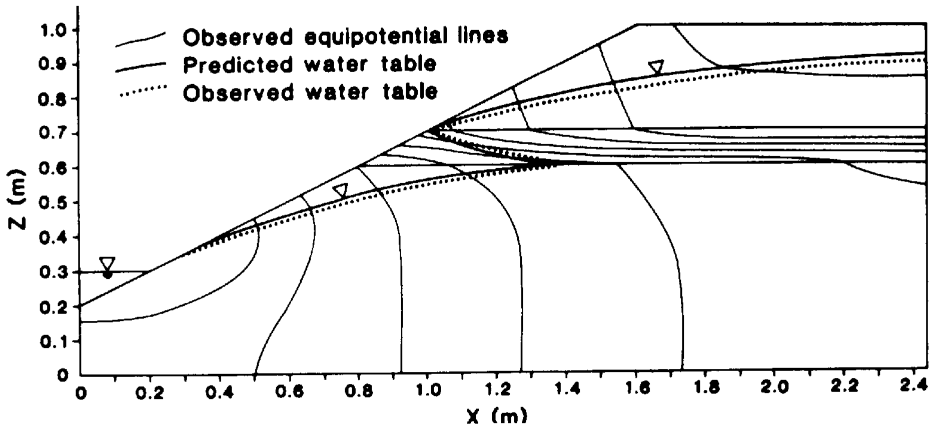


Figure 2-1 Rulon's laboratory scaled model results

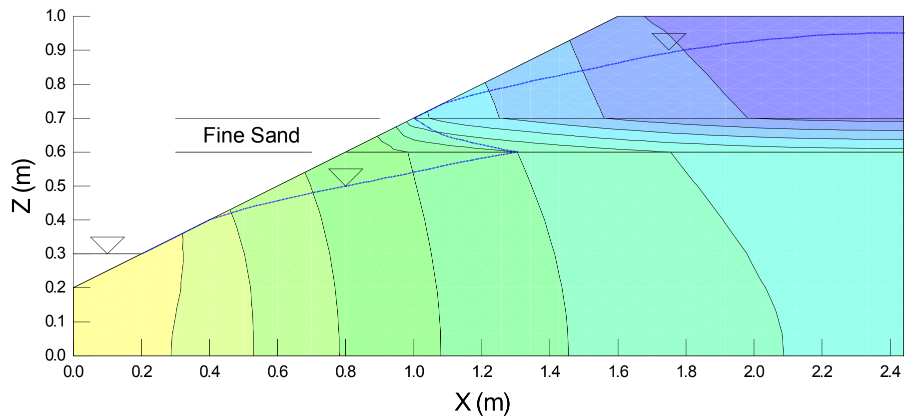


Figure 2-2 SEEP/W analysis of Rulon's laboratory model

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

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

- Numerical models can be set up very quickly relative to physical models. Physical models may take months to construct while a numerical model can be constructed in minutes, hours or days.
- A physical model is usually limited to a narrow set of conditions. A numerical model can be used to investigate a wide variety of different scenarios.

- Numerical models have no difficulty accounting for gravity. Gravity cannot be scaled, which is a limitation with laboratory modeling. A centrifuge is often required to overcome this limitation.
- With numerical modeling, there is no danger of physical harm to personnel. Physical modeling sometimes involves heavy equipment and worker safety is consequently a concern.
- Numerical modeling provides information and results at any location within the cross-section. Physical modeling only provides external visual responses and data at discrete instrumented points.
- Numerical models can accommodate a wide variety of boundary conditions, whereas physical models are often limited in the types of boundary conditions possible.

It would be wrong to think that numerical models do not have limitations. Associated with seepage flow there may also be temperature changes, volume changes and perhaps chemical changes. Including all these processes in the same formulation is not possible, as the mathematics involved simply become too complex. In addition, it is not possible to mathematically describe a constitutive relationship, due to its complexity. Some of these difficulties can and will be overcome with greater and faster computer processing power.

It is important to understand that numerical modeling products like SIGMA/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. SIGMA/W is formulated only for small strain cases and not for post failure deformation. In many cases, the constitutive equations used in this model are not defined for post failure stress conditions, so it is pointless to begin an analysis hoping to investigate those states. The important point to remember is that the mathematical formulations implemented in SIGMA/W result in a very powerful and versatile means of simulating real physical processes.

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

2.3 Modeling in geotechnical engineering

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

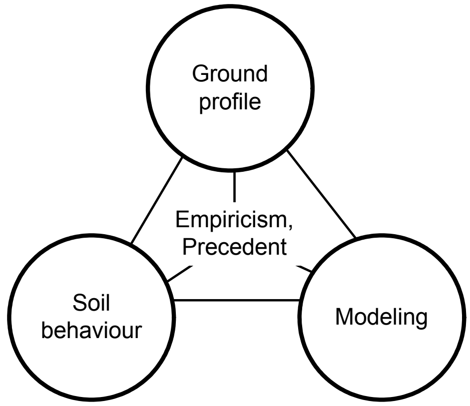


Figure 2-3 The Burland triangle (after Burland 1996)

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

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

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

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

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

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

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.

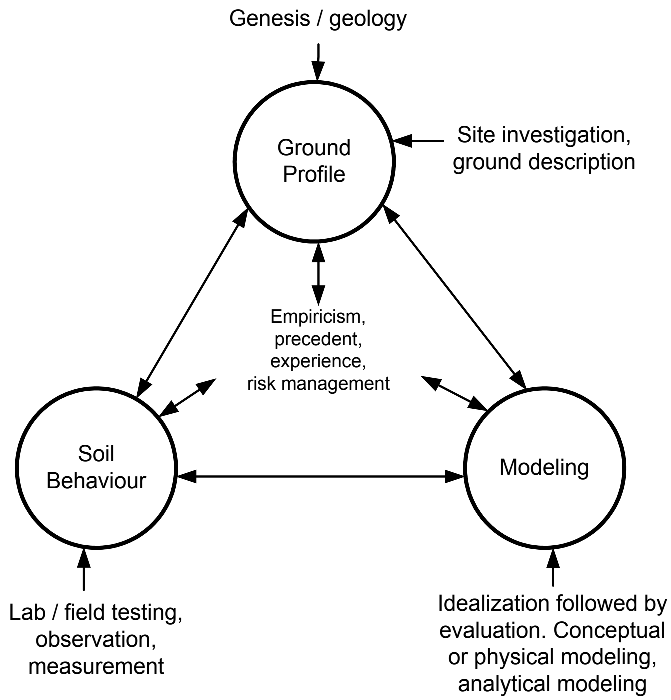


Figure 2-4 The enhanced Burland triangle (after Anon. 1999)

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 deformation under a footing, 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 extent of deformation under a footing, for example, is in large part controlled by the stiffness of the soil, which is itself controlled by the amount of confining stress. While it is realistic to obtain accurate stress distributions beneath the footing, we should realize that the accuracy of the deformation is only as good as our knowledge and confidence in our stiffness parameters.

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

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

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

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

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

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.

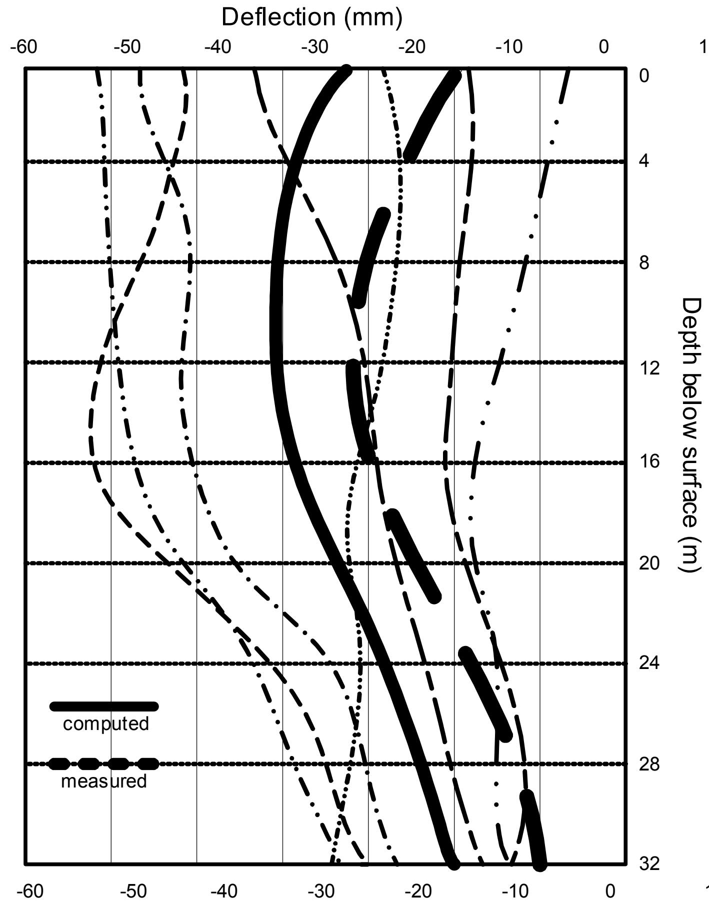


Figure 2-5 Comparison of predicted and measured lateral movements of a shoring wall (after Carter et al, 2000)

Compare alternatives

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

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

The actual computed values are not of significance in the context of this discussion. It is an example of how a model such as SEEP/W can be used to quickly compare alternatives. Secondly, this type of analysis can be done with a rough estimate of the conductivity, since in this case the pressure distributions will be unaffected by the conductivity assumed. There would be no value in carefully defining the conductivity to compare the base pressure distributions.

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

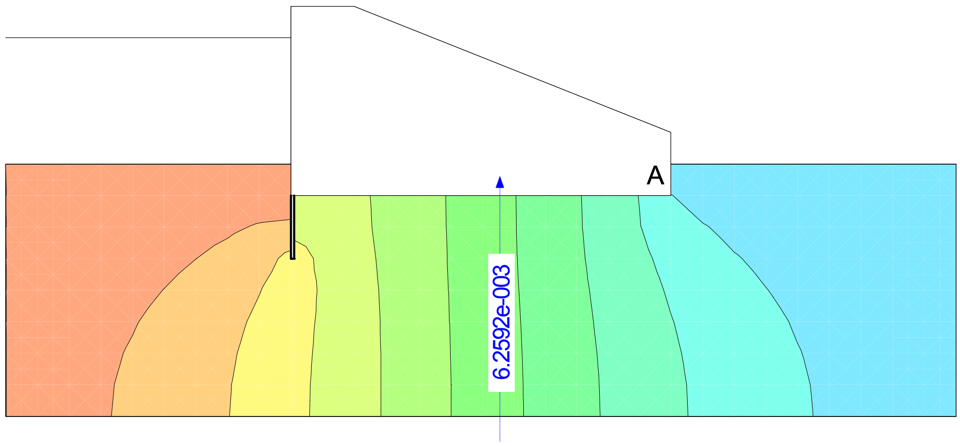


Figure 2-6 Seepage analysis with a cutoff

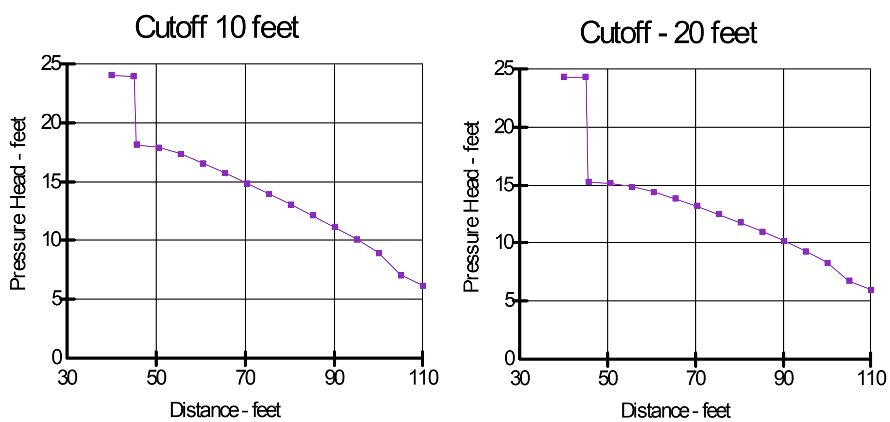


Figure 2-7 Uplift pressure distributions along base of structure

Identify governing parameters

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

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

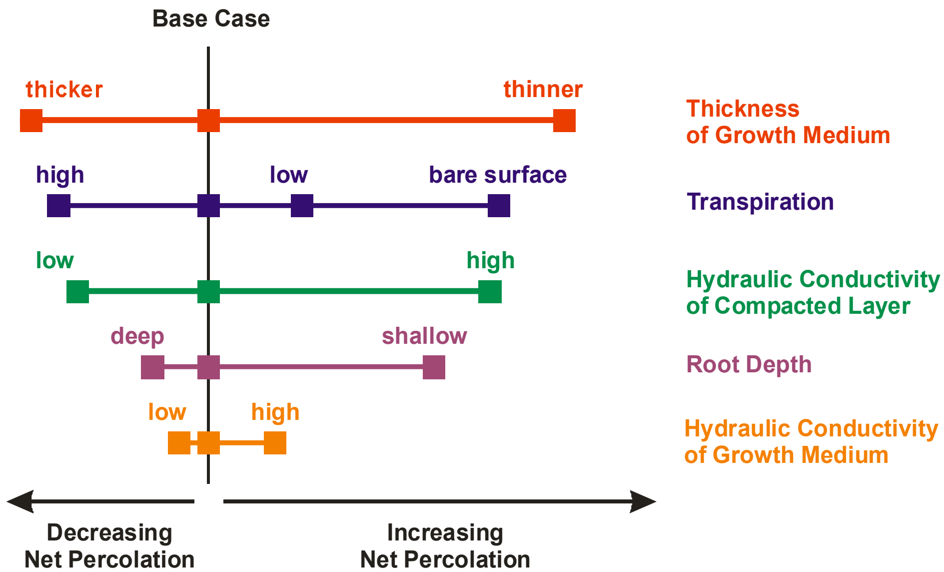


Figure 2-8 Example of a tornado plot (O'Kane, 2004)

Discover physical process - train our thinking

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

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

For unsaturated flow, it is necessary to define a *hydraulic conductivity function*: a function that describes how the hydraulic conductivity varies with changes in suction (negative pore-water pressure = suction). Chapter 4, Material Properties, 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 2-10 are sufficient. At low suctions (i.e., near saturation), the coarse material has a higher hydraulic conductivity than the fine material, which is intuitive. At high suctions, the coarse material has the lower conductivity, which often appears counterintuitive.

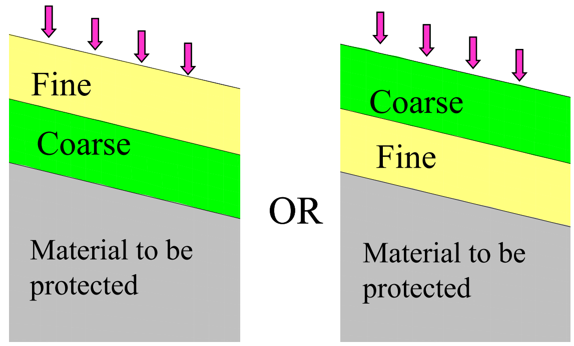


Figure 2-9 Two possible earth cover configurations

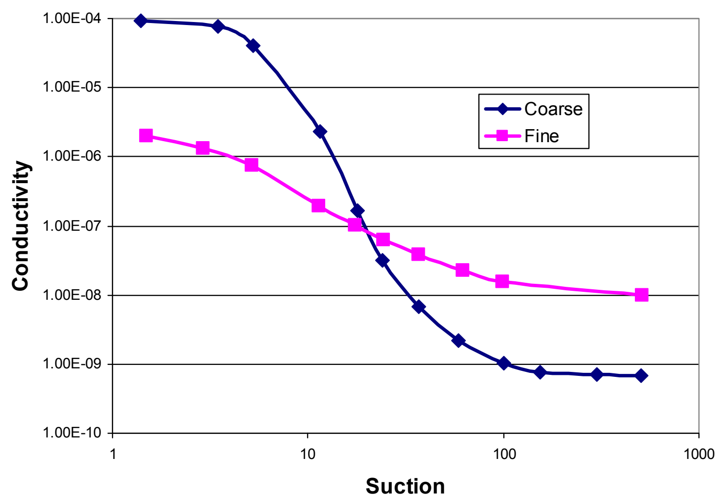


Figure 2-10 Hydraulic conductivity functions

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

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

A closer examination of the conductivity functions provides a logical explanation. The software is correct and provides the correct response given the input parameters. Consider the functions in Figure 2-13. When the infiltration rate is large, the negative water pressures or suctions will be small. As a result, the conductivity of the coarse material is higher than the finer material. If the infiltration rates become small, the suctions will increase (water pressure becomes more negative) and the unsaturated conductivity of the finer material becomes higher than the coarse material. Consequently, under low infiltration rates it is easier for the water to flow through the fine, upper layer soil than through the lower more coarse soil.

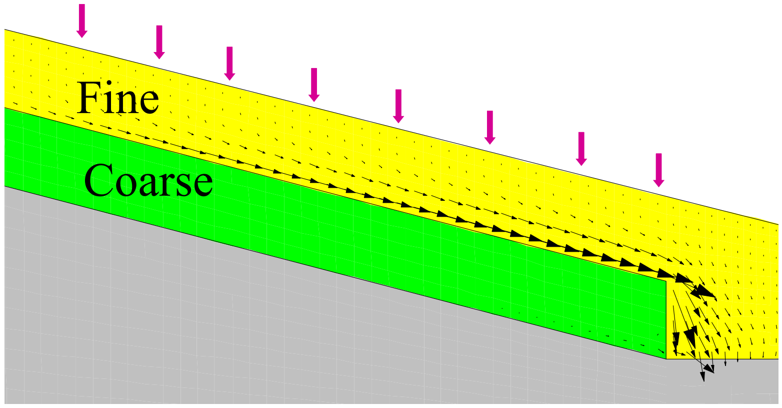


Figure 2-11 Flow diversion under low infiltration

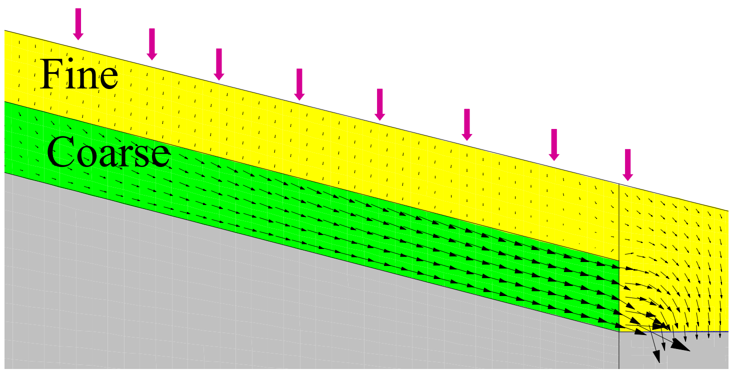


Figure 2-12 Flow diversion under high infiltration

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

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

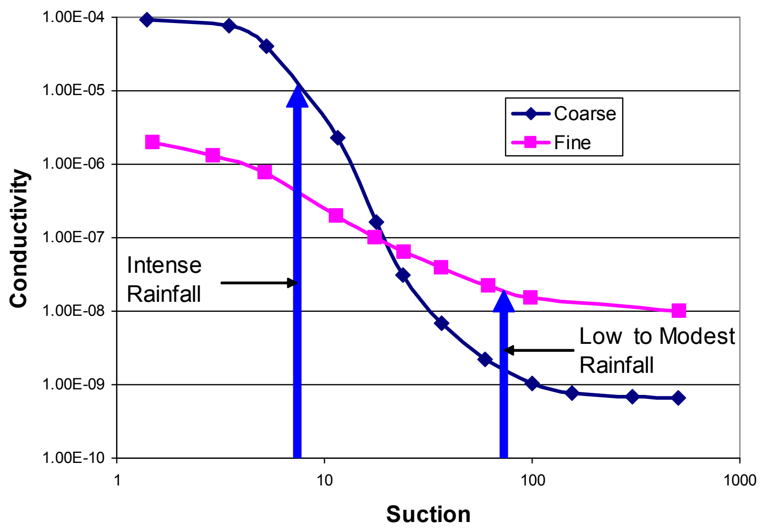


Figure 2-13 Conductivities under low and intense infiltration

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

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

2.5 How to model

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

Make a guess

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

Good modeling practice starts with some planning. If at all possible, you should form a mental picture of what you think the results will look like. Stated another way, we should make a rough guess at the solution before starting to use the software. Figure 2-14 shows a very quick hand calculation of stresses beneath a submerged, horizontal surface.

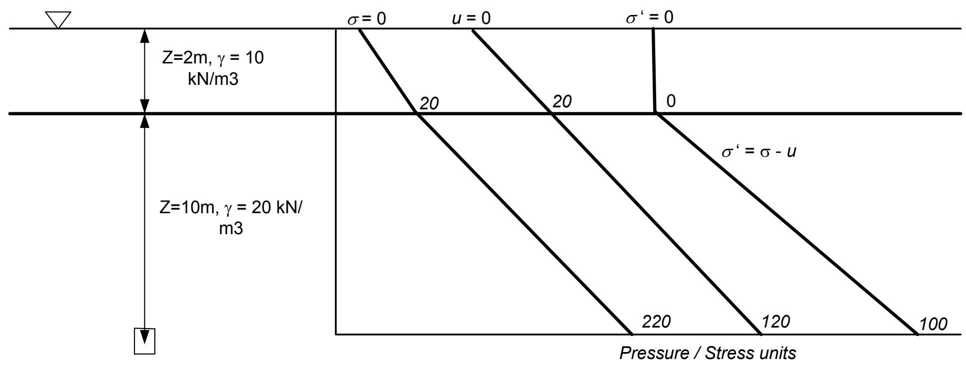


Figure 2-14 Hand calculation of in situ stresses

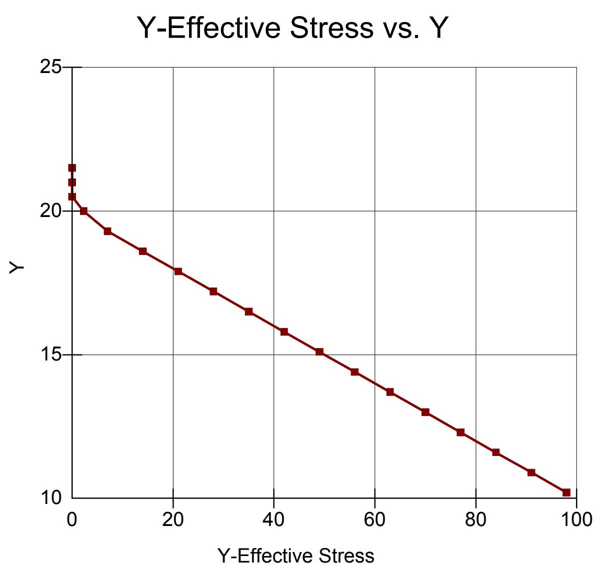


Figure 2-15 SIGMA/W computed effective stress profile

The hand calculation together with the SIGMA/W output can now be used to judge the validity of the computed results. If there is no resemblance between what is expected and what is computed with SIGMA/W then either the preliminary mental picture of the situation was not right or something has been inappropriately specified in the numerical model. Perhaps the pressure boundary condition was not specified on the ground surface in addition to the definition of the water table. The water table controls pore-water pressures in the soil, but does not add a weight load at the ground surface. Both must be specified and this error would be obvious by comparing expectations to computed results. Any differences ultimately need 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 of 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 stress distribution or is it deformation? If your main objective is to determine the stress distribution, there is no need to spend a lot of time on establishing an advanced soil model, a more simple linear-elastic model is adequate. If on the other hand your main objective is to estimate deformation, then a greater effort is needed in determining the appropriate stress-strain model.

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

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

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

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

Simplify geometry

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

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

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

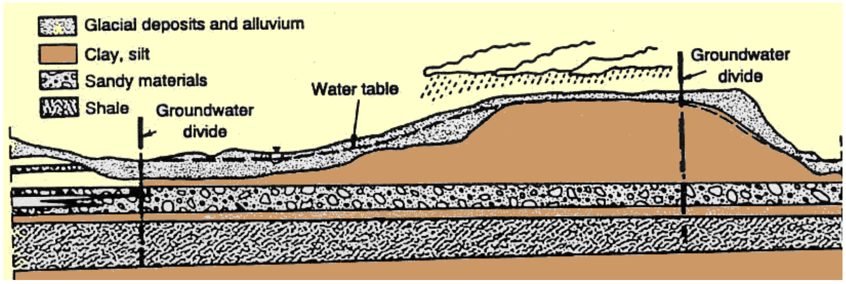


Figure 2-16 Example of a stratigraphic cross section (from National Research Report 1990)

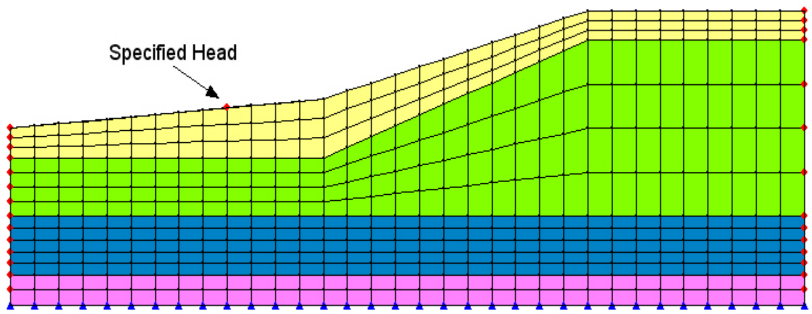


Figure 2-17 Finite element model of stratigraphic section

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

Start simple

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

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

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

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

Do numerical experiments

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

Consider the following example taken from a seepage analysis. You are uncertain about the results from a flux section or the meaning of a computed boundary flux. To help satisfy this lack of understanding, you could do a numerical experiment on a simple 1D case as shown in Figure 2-18. The total head difference is 1 m and the conductivity is 1 m/day. The gradient under steady state conditions is the head difference divided by the length, making the gradient 0.1. The resulting total flow through the system is the cross sectional area times the gradient which should be 0.3 m^3/day. The flux section that goes through the entire section confirms this result. There are flux sections through Elements 16 and 18. The flow through each element is 0.1 m^3/day, which is correct since each element represents one-third of the area.

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

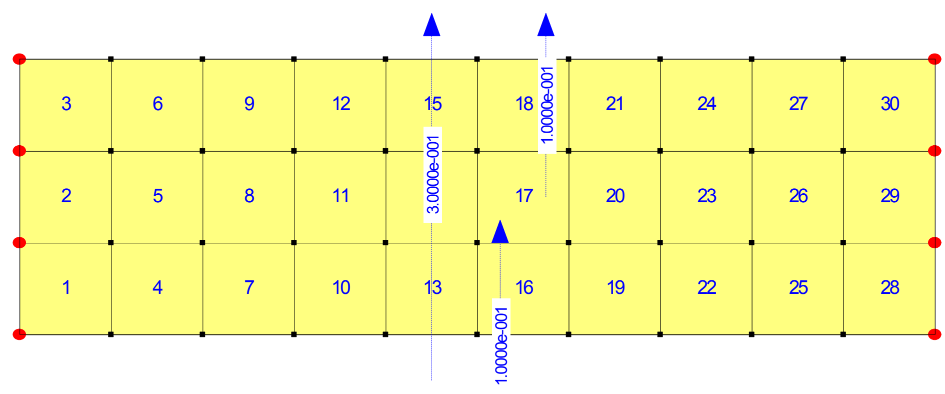


Figure 2-18 Horizontal flow through three element section

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

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

Model only essential components

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

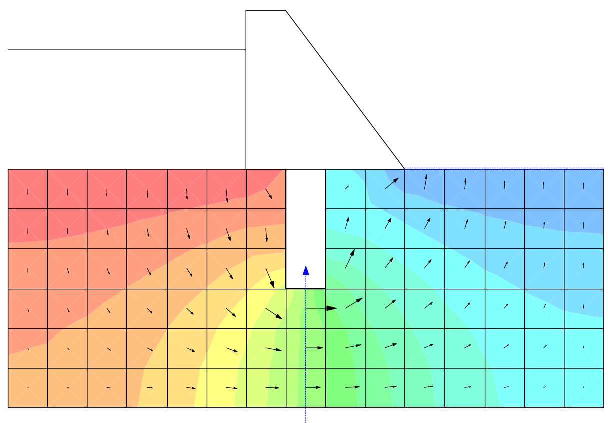


Figure 2-19 Simple flow beneath a cutoff

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

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

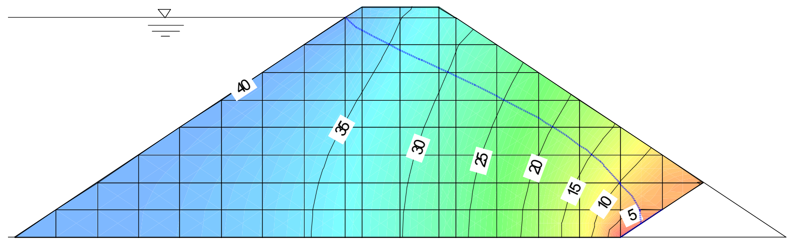


Figure 2-20 Flow through a dam with coarse toe drain

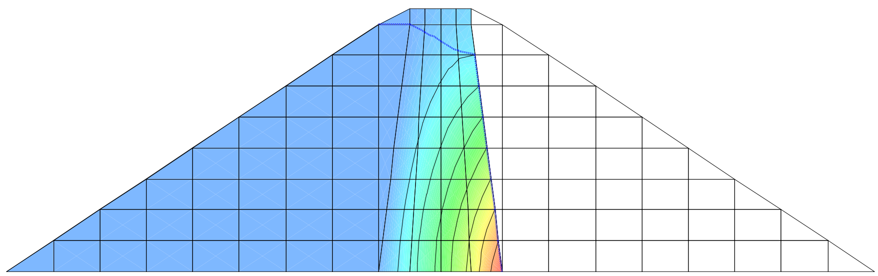


Figure 2-21 Head loss through dam core with downstream shell

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

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

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

Start with estimated material properties

In the early stages of a numerical modeling project it is often good practice to start with estimates of material properties. In SIGMA/W this means always start with a linear-elastic soil model. It will converge quickly because of its linearity, it will let you refine the geometry if needed, and it will let you establish the load step sequence you need. Simple estimates of material properties and simple property functions are more than adequate for gaining an understanding of the stress 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 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 SIGMA/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 when defining the model. Is the intended property function being applied to the correct soil? Or, are the initial conditions as you assumed?

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

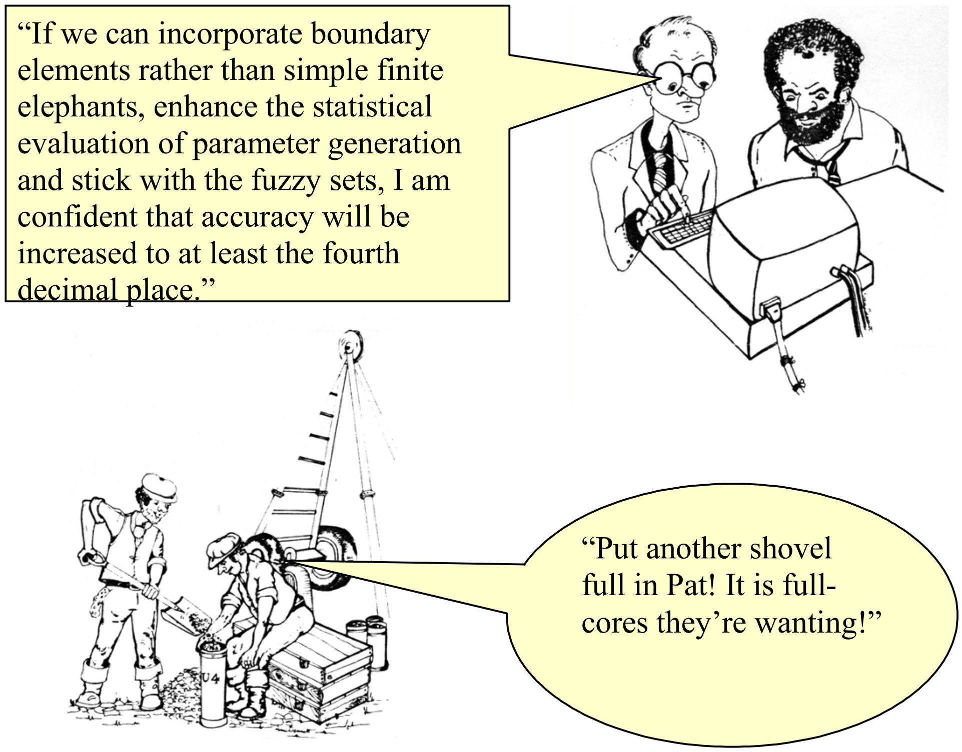
Inspecting and spot checking your results is an important and vital component in numerical modeling. It greatly helps to increase your confidence in a solution that is understandable and definable.

Evaluate results in the context of expected results

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

Remember the real world

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



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

2.6 How not to model

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

For novice modelers; the initial reaction when faced with incomprehensible results is that something must be wrong with the software. It must be a limitation of the software that the solution is inappropriate or completely senseless. It is important to remember that the software is very powerful; it can keep track of millions of pieces of information and do repetitive computations which are far beyond the capability of the human mind. Without the software it would not be possible to make these types of analyses. The software by itself is extremely powerful numerically speaking, but essentially unintelligent. Conversely, the human mind has the capability of logic and reasoning, but has significant limitations retaining large amounts of digital data. It is the combination of the human mind together with the capability of a computer that makes numerical modeling so immensely powerful. 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 advantages of the power the software can offer.

2.7 Closing remarks

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

When the numerical analysis software tool, SIGMA/W, 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 SIGMA/W: Fundamentals and Practical Modeling Considerations

3.1 Introduction

SIGMA/W is a powerful finite element method tool that can be used to model a wide range of stress-strain problems. In its appearance and usability it is much like other products in GeoStudio. For example, geometry and finite element meshes or soil regions common to seepage or slope analysis can be used directly in SIGMA/W. However, there are certain fundamental differences between SIGMA/W and the other models. Foremost of these differences is that SIGMA/W uses an incremental load formulation. This means that you enter a step change condition (e.g. an applied surface load) and it computes step changes in parameters such as strain and pore-water pressure. It does not compute the actual end condition unless you have established in-situ conditions prior to loading.

All of the GeoStudio Engineering books have a chapter focusing on Modeling Tips. The Modeling Tips chapters in the other books generally cover items specific to enhancing the usefulness of the program from a model set up and data interpretation perspective. However, due to SIGMA/W's uniqueness, this book has this chapter titled Fundamentals and Practical Modeling Considerations and it is placed near the front of the book in hopes it is read and understood prior to reading the rest of the book. This entire chapter will be most useful to novice users. More advanced users will find the first part a good review, but not entirely necessary. All users should ensure they read and understand the second part of this chapter.

This chapter has two main parts: the first few sections deal with an overview of fundamental concepts related to stress, volume change and strength of materials. A detailed theoretical discussion of various soil strength models is given in the Theory chapter. This discussion will focus more on ensuring there is a basic understanding of important parameters affecting volume change and soil strength so that the reader appreciates at a deeper level why SIGMA/W was formulated a certain way and what it is formulated to do.

The second several sections of this chapter will focus on how the fundamental concepts have been put into the model and how the model can be used in various unique and creative ways to solve real problems.

A key concept that will be developed and stressed throughout this chapter is that SIGMA/W is a very powerful tool for investigating the *serviceability* of engineered soil systems. It is not a tool for predicting *stability* of these systems.

Serviceability deals with obtaining confidence that a design will function as intended. The numerical soil models available in SIGMA/W can help show that engineered systems exposed to various applied external or internal loads will not fail.

Stability, on the other hand, deals with quantification of a factor of safety for an engineered system. However, in order to obtain a factor of safety it is necessary to have solved the numerical models to some point past failure so that the stress condition after failure can be compared to the allowable stress condition at failure. The problem with this is that the soil models in a small strain stress-deformation analysis are not defined at stress conditions that exceed failure. Therefore, a safety factor cannot be computed.

It is important to realize this, so that you can begin the modeling and design process by looking for an engineered system that works! If you generate results that indicate failure, two things should come to mind: 1) that the results are likely meaningless and, 2) that you are reminded the true objective is to design a working system, not one that fails. In other words, model to design a system that is *serviceable*.

3.2 Density and unit weight

For many applications in geotechnical engineering, a measurement of the in-situ density of a soil is required in order to determine the forces exerted by its' self weight. By definition, the density is the mass of soil in a sample divided by the volume of soil in the sample. In equation form:

$$\rho = \frac{m}{V}$$

where:

ρ	=	the bulk density of soil,
m	=	the mass of soil, and
V	=	the volume of soil.

Some caution should be exercised when working with the term "density." Density relates to mass per unit volume, which should not be confused with "weight density" denoted by the symbol, γ. The weight density is also called the unit weight and it has units of gravitational force (N, lbf) per unit volume. Since SIGMA/W is formulated to deal with displacement in response to forces, it makes use of the unit weight form of density.

The bulk, or total density of a soil refers to the mass of bulk soil which includes soil particles, water and air in a given volume. The bulk density will therefore change throughout a domain if there is a change in water content throughout the domain. There are other useful ways of quantifying density: the dry density, saturated density and buoyant density.

Dry density is the mass of dry soil per unit volume and is commonly referred to when dealing with compaction of earthworks. The dry density is related to the bulk density of the soil according to:

$$\rho_d = \frac{1}{1+w}\rho$$

where:

w	=	the water content by weight (in decimal form).

The saturated density, ρ_{sat}, describes the bulk density of a soil when all voids are filled with water and the submerged density is simply the saturated density minus the density of water as follows:

$$\rho' = \rho_{sat} - \rho_w$$

where:

ρ '	=	the submerged density, and
ρ_w	=	the density of water.

As with the unit weight of the bulk soil (including water weight), the submerged unit weight can be a useful parameter to use in SIGMA/W. The submerged unit weight is simply the total unit weight minus the unit weight of water as follows:

$$\gamma' = \gamma - \gamma_w$$

where:

γ_w = the unit weight of water.

Further discussion on the use of submerged unit weight is given in the Analysis Types chapter discussion on in-situ analysis, and in the chapter on Pore-water Pressures.

3.3 Compaction and density

Soil is used as a “fill” material in many engineering projects and in order to obtain satisfactory serviceability of the project, compaction (densification) of the fill is often required. Compaction of a loose material has several advantages from a design perspective, as it can be used to increase shear strength, decrease compressibility, and decrease permeability. In order to know how much compaction effort is required in order to achieve the desired soil property, it is useful to conduct a laboratory compaction test on a representative soil sample from the site. The objective of the test is to determine, for a given compactive effort, the optimum moisture content that will result in the maximum dry density. The laboratory test used most often is the Standard Proctor Test, introduced by R.R. Proctor in 1933.

Compaction involves application of mechanical energy in order to reduce the air voids in a soil with little or no reduction in water content. At low moisture contents, the soil particles are surrounded by thin films of water which tend to keep the grains separated when compacted. When more water is added, the soil grains move together more easily which results in a reduction of the air voids and an increase in bulk (total) density. At some point, making the soil too wet prior to applying the compactive effort will result in a reduction in density. The water content that corresponds with the maximum dry density is termed the optimum water content. Figure 3-1 shows the water contents and maximum dry densities for a single soil type subjected to different compactive efforts. Inset in the figure are illustrations of the orientation of soil fabric at different states. Generally, soils are quantified based on whether they are compacted dry of optimum water content, at the optimum water content, or wet of optimum water content. The usefulness of this classification in terms of SIGMA/W will be discussed shortly.

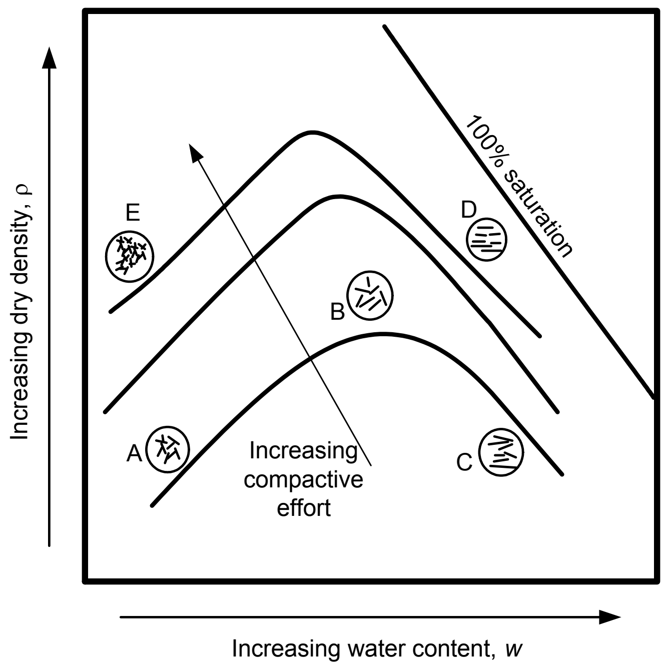


Figure 3-1 Dry density, water content and compactive effort (after Holtz and Kovacs, 1981)

Notice that adding more effort does not result in elimination of all air voids in a soil. It does result in reducing the required optimum water content and in increasing the maximum dry density. This type of a test comparison is very useful for helping assess the economic benefits of applying less water or with more compaction effort or visa versa.

Compaction and strength

Knowing the compactive effort and water content applied to an engineered soil can lead to a better understanding of its anticipated behavior under loading. For clay type soils, it is recognized that for a fixed compactive effort, as more water is added, the soil structure becomes increasingly oriented (refer to points A, B and C in Figure 3-1). That is, the clay minerals become less flocculated and more dispersed. Dry of optimum, the structure or soil fabric is considered flocculated, while wet of optimum, it is considered dispersed, or oriented. As the compactive effort is increased, the amount of orientation is increased (refer to points C and D in the figure).

A full discussion of the effects on soil properties due to changing water content and compactive effort is beyond the scope of this book. In general, however, it is important to understand that soils compacted dry of optimum are likely to have higher strengths than those compacted wet of optimum. In addition, hydraulic permeability at a constant compactive effort decreases with increasing water content and reaches a minimum at about the optimum water content. If compactive effort is increased, the permeability decreases because the void ratio decreases.

Understanding the anticipated behavior based on these observations can help to make the model set up and definition easier and the interpretation of output data more meaningful. For example, if you are solving a consolidation analysis and you are getting delayed response in dissipation of pore-water pressures within an engineered structure, you may want to adjust the applied unit weight (a function of total density) and make a corresponding change to the hydraulic permeability function applied in SEEP/W. This type of fine tuning soil properties is not usually recommended. However, you should understand from this discussion that there is a connection between how things are actually built in the field and what the soil properties can be.

3.4 Plasticity

SIGMA/W does not require, as input, any of the Atterberg limit parameters. However, as they are fairly simple to ascertain, it is useful to discuss their significance from a practical perspective. If you know the soil plastic and liquid limits, then you can anticipate the soil's response and better understand what you are asking the model to do.

The Atterberg Limits were developed in 1911 as a practical means of describing the plastic behavior of clays. While there are several classifications, the most relevant to this discussion are the Plastic Limit, the Liquid Limit and the Liquidity Index.

The liquid and plastic limits are both water content by weight values expressed in decimal form. The plastic limit, PL, is the water content at which point the soil starts to behave plastically when loaded. The liquid limit, LL, is the water content when the soil just starts to behave as a viscous fluid when loaded. The liquidity index is defined as:

$$LI = \frac{w - PL}{LL - PL}$$

where:

w = the natural water content of the soil.

If the LI is between 0 and 1, the soil will behave plastically, and if it is greater than 1, the soil will behave like a viscous fluid when sheared. Figure 3-2 compares anticipated stress-strain behavior for a range of liquid and plastic limits. It is clear that as the plastic limit is approached, there is the potential for larger strains in response to small changes in shear stress. The figure clearly shows that for soils approaching or exceeding the liquid limit, the amount of strain in response to small load changes can be infinite.

This is important to understand. If you are asking the model to compute strain for a condition where the soil is at a state between its plastic and liquid limits, then there is chance for large strains. The SIGMA/W formulation is a small strain formulation, and often it cannot converge on a reasonable solution when strains become too large. In some cases, a solution can be reached, but it can have no meaning. It is up to you to understand the behavior you expect from the soil and to model a stress condition that will lead to a meaningful response. If you want a structure to be useful in its application, then you must design (and model) it for conditions that are within is range of serviceability.

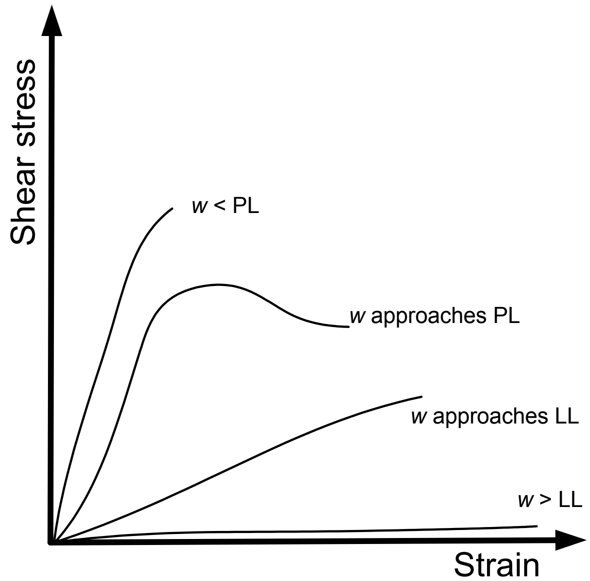


Figure 3-2 Stress-strain and Atterberg Limits

3.5 Volume change

Effective and total stress

It is absolutely critical from this point onwards, to have a clear understanding of effective and total stresses and the difference between the two. Incorrect use of these parameters will result in incorrect modeled output.

The total stress in a soil is dependent on the total unit weight of the soil above it plus any applied surface loads (such as free water). Additionally, the total stress at a given depth in the soil is different in the horizontal and vertical directions. However, for now, we will consider only the stresses in the vertical direction.

Consider the element of soil illustrated in Figure 3-3 that sits 10m below surface where the surface is covered by 2m of water. The total vertical stress in the soil element at this point is equal to the weight of everything above it divided by the unit cross-sectional area (assumed to be 1m^2). In equation form:

$$\sigma_v = \sum_i^n \gamma_i z_i$$

where:

z	=	the thickness of each material layer,
γ	=	the total unit weight of each material, and
i,n	=	the individual and total number of materials.

So, in this case, the total stress equals (2m*10 kN/m^3) + (10m*20 kN/m3), or 220 kN/m^2 = 220 kPa.

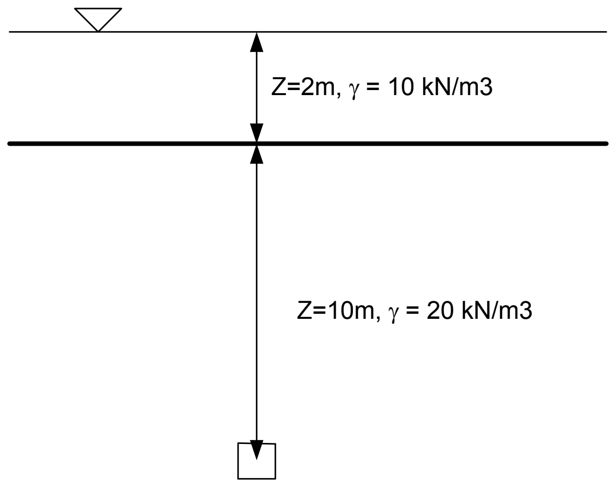


Figure 3-3 Illustration of total and effective stress

The effective stress is different than the total stress because the presence of water in between the soil grains provides a force that can either tend to push the particles apart or pull them closer together; as would be the case if the water in the soil was in a capillary state. In equation form, the effective stress is:

$$\sigma_v^{'} = \sigma_v - u$$

where:

u = the pore-water pressure.

Based on the example illustrated in Figure 3-3, the vertical effective stress in the soil element would be computed as the total stress, 220 kPa, minus the pore-water pressure (= 12m*10 kN/m^3 = 120 kPa) = 100 kPa.

The effective stress is less than the total stress which is, in effect, stating that the water in the soil is taking up some of the weight (body) load of the soil and thereby reducing the stress in the soil.

Compressibility and consolidation

Compressibility and consolidation deal with how the soil changes in response to a change in its loading. When a load is first applied on a soil, it will compress, or change volume due to:

- deformation of the soil grains,
- compression of the air and water in the voids, and
- squeezing of water and air from the voids.

In general, we neglect the deformation of soil grains and the compressibility of air and water because they are small compared with the third point.

In a coarse material, the squeezing of water from the voids can occur quite quickly which results in a rapid compression of the soil under loading. However, in a fine material, the rate of compression is directly a function of the hydraulic permeability of the soil and, therefore, the compression is time dependent. This phenomenon is called consolidation.

Consider Figure 3-4 where a submerged 1m^2 footing with a 50 kN load is applied to the previously illustrated soil element. Two important points can be made using this illustration.

When the load is first applied, there is a change in total stress on the soil of 50 kN/m^2 or 50 kPa. Therefore, the total stress in the soil element is now 270 kPa. When the load is first applied, there is no time for the finer grained soil particles to shift around in order to assume the new load so the pore-water carries the load and its pressure increases by an amount equal to the applied new stress. Therefore, the pore-water pressure at the soil element increases by 50 kPa and the new effective stress is computed to be 270 kPa minus 170 kPa = 100 kPa; which is the same as the starting effective stress. If there is no change in effective stress, there is no change in soil volume.

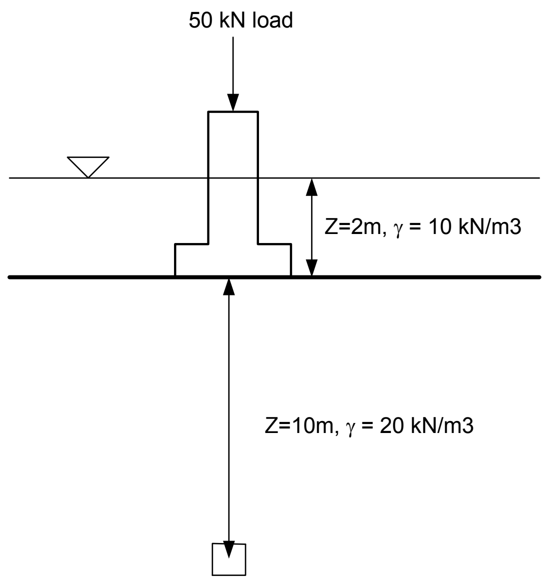


Figure 3-4 Surface load applied to example

Because this is a fine grained soil, we know that with time, the excess pore-water pressure will dissipate back to the pre-load value. When this happens, the effective stress will be computed as the new total stress minus the original pore-water pressure, or 270 kPa – 120 kPa = 150 kPa. The effective stress has increased which means that the soil must change volume or consolidate.

No change in effective stress means no volume change. An increase in effective stress means consolidation or shrinkage. A decrease in effective stress means swelling or heave.

Sometimes it is useful to plot individual effective stress parameters at the start and end of a load step. Consider Figure 3-5 which shows water pressure response to the loading discussed in Figure 3-4. The new total stress load here is considered to be applied instantly which results in an equal increase in pore-water pressure at the start of the consolidation phase.

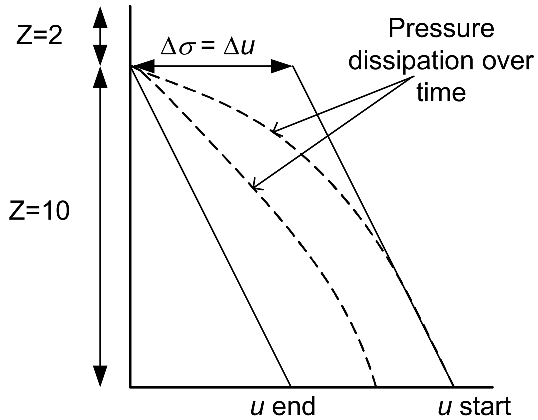


Figure 3-5 Dissipation of pressure over time

We can use this same approach and simultaneously consider the calculation of all parameters for the two cases described above. Figure 3-6 shows the calculation of all stresses and pressures for the two examples as a function of depth to the soil element. The top of the figure shows the calculation of effective stress for the in-situ case (prior to the load step) while the bottom of the figure shows what the stresses are just after the footing load is applied and after consolidation has occurred.

There are several key points to understand from this figure:

- In order to get the actual stresses after consolidation, the in-situ stresses must be known ahead of time.
- The effective stress at the ground surface is zero at all times.
- The step load from the footing instantly increases the pore water pressure because the soil has not had time to dissipate the pressure. The pressure increase is equal to the newly applied load.
- There is no change in total stress over time after the loading.
- There is no initial change in effective stress at the time the load is applied.
- As the pore-water pressure dissipates, the effective stress increases towards its new value.

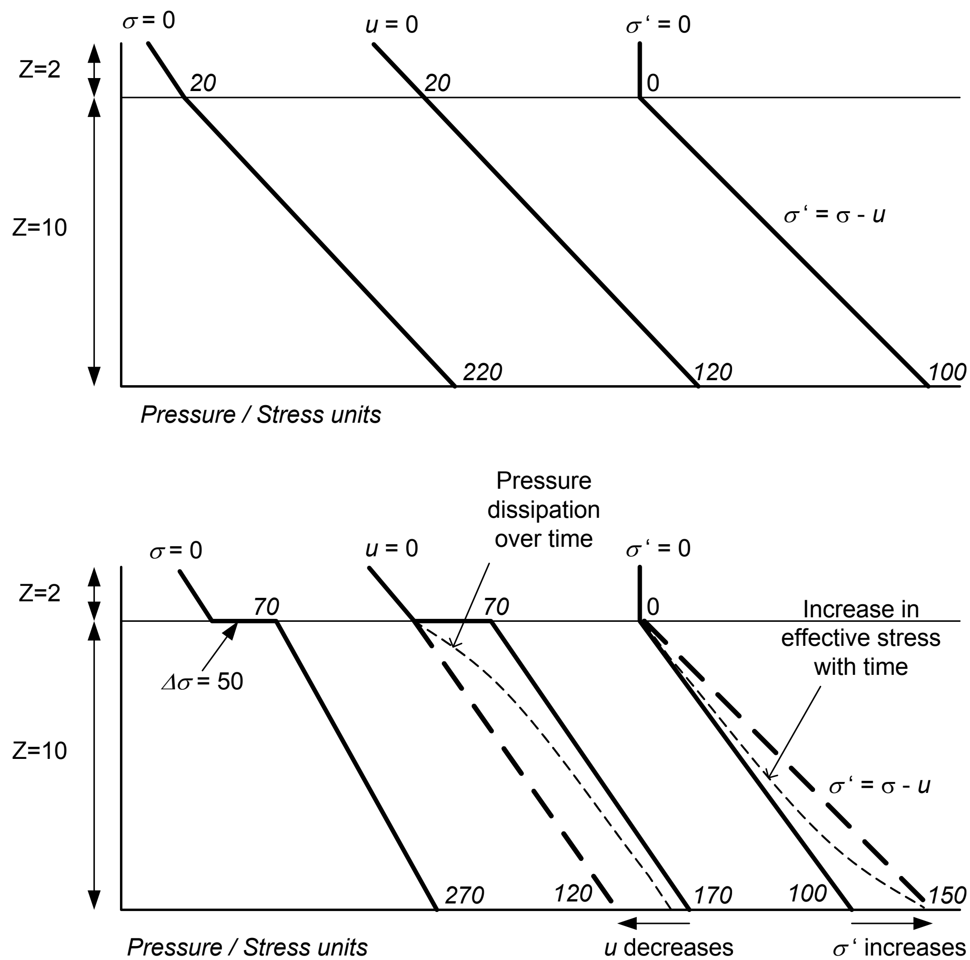


Figure 3-6 Stress and pressure profiles for two example cases

A concept that should be getting more evident at this point is that we can not generally compute the actual stresses in the ground at any given point in time simply based on the applied loads at that time. We must consider how we have changed the external conditions at any given time and then compute the change in the ground's response. For modeling purposes, we can, and should, think of our effective stress equation as:

$$\Delta\sigma_v^{'} = \Delta\sigma_v - \Delta u$$

where the delta symbol indicates a step change in each parameter. From our earlier examples, the change in total stress, $\Delta\sigma$, equals 50 kPa and the change in pore-water pressure, Δu, is also 50 kPa. Therefore, the change in effective stress just after loading is equal to zero, and, there is no volume change.

This concept of changes in loads, or incremental loading, is discussed in more detail later in this chapter. It is a critical concept to understand.

Zero change in pore-water pressure

What happens if there is no pore-water pressure change during a change in total stress? If this is the case then Δu = 0 and our effective stress equation becomes:

$$\Delta\sigma_v^{'} = \Delta\sigma_v - 0$$
$$or$$
$$\Delta\sigma_v^{'} = \Delta\sigma_v$$

In this case, the change in effective stress is equal to the total stress. By always thinking of the stresses in the soil in terms of effective stress, our rule about volume change only occurring with a change in effective stress always applies. It will get confusing if you start thinking of volume change in response to changes in total stress some of the time, and in terms of effective stress at other times. Always understand what the pore-water pressure changes are (or are not), and think in terms of effective stress.

3.6 Strength parameters

Mohr stress circle

The Mohr circle can be used to state of stress at a point in equilibrium. Consider the soil element in Figure 3-7 with some horizontal, vertical, shear, and normal forces acting upon it.

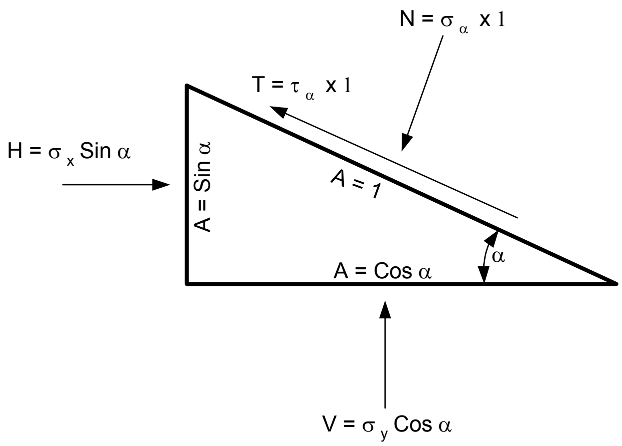


Figure 3-7 Forces acting on a soil element

If the forces in on each face are broken into horizontal and vertical components and converted to stresses by dividing them by the cross sectional area upon which they act, two equations in terms of σ, τ and α are developed. If these equations are each set to zero, they can be squared and added together. The end result is a single equation for a circle with a radius of:

$$\frac{(\sigma_x - \sigma_y)}{2}$$

and with its center at:

$$\left[\frac{(\sigma_x + \sigma_y)}{2}, 0\right]$$

The vertical and horizontal planes in the soil element have no shear force acting on them so they are, by definition, principal planes. Therefore, the forces acting on them are principal forces. An important point to remember here is that principal stresses act on planes where the shear strength is zero. Another point to note is that the stress with the largest magnitude is called the major principal stress (denoted σ_1), and the lower value stress is the minor principal stress (denoted σ_3).

If the information from the summation of forces equations can be used to construct a circle on the shear-normal stress plane as shown in Figure 3-8. Using this circle, the applied normal and shear stress along any potential failure plane in a soil element can be determined. In this case, the shear and normal stress point is shown for a plane inclined at an angle, α, from horizontal.

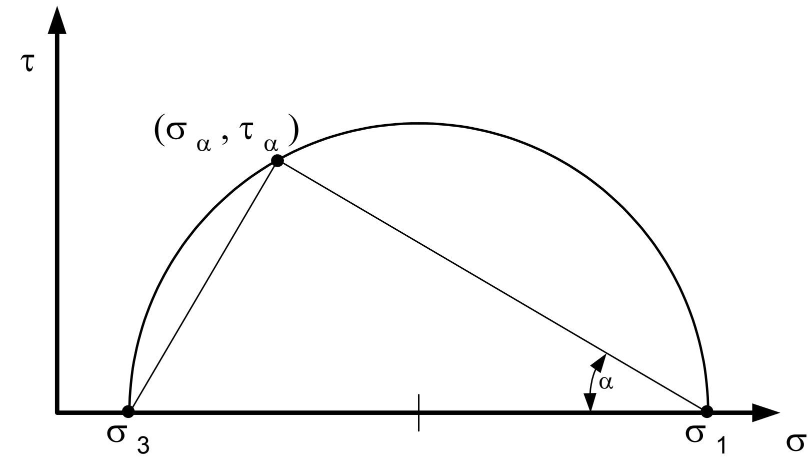


Figure 3-8 Mohr circle of stress

Mohr Coulomb failure criteria

The Mohr stress circle can be assembled for a series of shear tests on a similar soil using different applied normal loads and then combined with a frictional and cohesive strength theory developed by Coulomb to arrive at the Mohr Coulomb failure criteria. A more detailed discussion of this criterion is available in most geotechnical engineering text books so it is omitted from this book. The purpose of introducing it now is to help in our subsequent discussion of effective and total strength parameters and how they are obtained and what they mean.

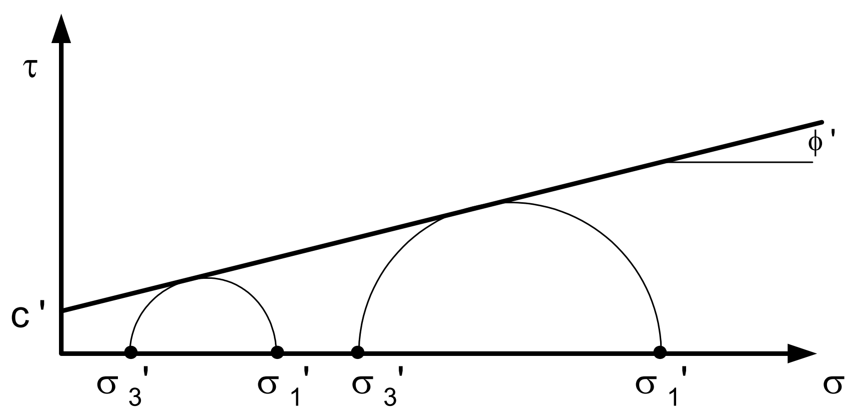


Figure 3-9 Mohr Coulomb "effective" failure parameters

Figure 3-9 shows a Mohr Coulomb failure envelope for a clay type soil. Two Mohr circles are plotted in this figure and a line is drawn tangent to the two circles such that it also intersects the vertical shear stress axis. The point of intersection of the tangent line provides the effective cohesion parameter and the angle of inclination of the tangent line provides the effective friction angle. All of the parameters illustrated in the figure have the "apostrophe" symbol, which indicates that they are effective stress based values and not total stress based values. They are effective stress based because the pore-water pressure in the sample was measured during the testing and subtracted from the total stress values applied in the test.

Once the data from the shear tests has been graphed, as shown in Figure 3-9, it is possible to create the equation that represents the data. We see that the tangent line is a straight line with a y-intercept. Therefore, the equation of the line, termed the Mohr Coulomb failure criteria, is given by:

$$\tau_f = c' + \sigma'_f (\tan \phi')$$

where the subscript, f, denotes the shear and normal stress condition in the soil element at failure.

Triaxial testing

Entire textbooks can be written on the subject of performing and interpreting data from the various types of triaxial tests on various types of soil. This type of a discussion is far too advanced in this context. It is important, however, to distinguish between two main types of test that can be carried out in a triaxial cell.

Undrained tests

Undrained tests can be performed on undisturbed as well as disturbed samples of clay and silt, and disturbed samples of sand or gravel. The sample is allowed to consolidate under a desired confining pressure and then sheared in axial compression.

In an un-drained test, the pore-water in the sample is not allowed dissipate, so the pressure builds up in response to changes in total stress. The changes in stress result from application of an applied strain to the top of the sample. So, for example, if the applied strain results in a principal major deviator stress of 50 kPa, the water pressure in the soil should also increase by an equal amount.

The undrained test can be performed with or without pore-water pressure measurements. If pore-water pressure is not measured, the results must be expressed in terms of total stress. The field conditions that a soil consolidates under are usually quite different than the conditions in the laboratory so the results of this test, when expressed in terms of total stress, can only be applied in a limited way to the field.

The results of a total stress triaxial test would look like those illustrated in Figure 3-10.

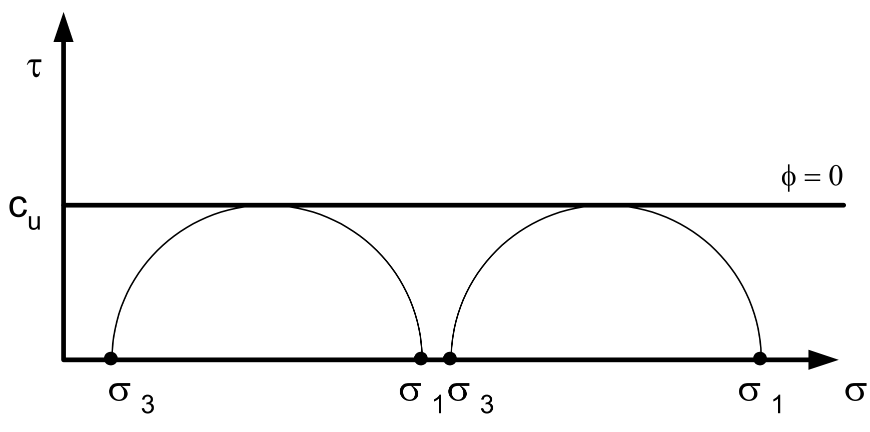


Figure 3-10 Mohr circles for undrained, total stress tests

If the pore-water pressure is measured during loading, the rate of applied strain must be slow enough to allow the pressures to equalize and be measured. With the measurement of pore-water pressure, the results can be expressed in terms of effective stress and the values of c' and Φ' can then be applied to a wider range of field applications. The results of an undrained effective stress triaxial test were illustrated previously in Figure 3-9.

The undrained test results are useful when a soil structure has had a long time to come to equilibrium and then a sudden change in total stress is applied with limited drainage. This may be the case for application of fill or rapid drawdown of a reservoir.

Drained tests

A drained test can be performed on all types of samples, whether disturbed or not. It is prepared in the same manner as an undrained test sample, but during the loading stage, the water is allowed to drain further. The applied rate of strain must be slow enough to prevent the development of excess pore-water pressures; in other words, it should remain at the state it was in after initial consolidation prior to loading.

At completion of testing, the Mohr circle failure envelope will look very similar to that shown for the undrained effective stress analysis, which makes sense because if the soil is permitted to drain, then the change in pore-water pressure will be zero and the change in effective stress will equal the change in total stress.

The drained test results are quite useful to use when the long term seepage dependent consolidation is an issue. However, it is not an easy test to conduct because it is important to ensure that pressures are not building up in the sample. In addition, for low permeability soils, it can take a long time to allow drainage which means a very slow rate of applied strain is required.

Soil stiffness – Young's modulus

Regardless of how the testing is carried out, the objective is to determine the strength parameters (described above) and also the stress-strain behavior. The stress-strain behavior is the actual constitutive model used in the solution of the partial differential equations. The strength parameters are only useful for telling if any given soil element has exceeded its yield point. Further discussion on the usefulness of strength parameters in stress-deformation modeling is included in the section on stability and serviceability below.

Consider Figure 3-11, which shows the stress-strain behavior for three confining loads during a strength test. As the confining stress increases, so does the slope of the deformation curve. It is the slope of these curves that relates the amount of strain in response to an applied stress. In equation form, the slope is given by:

$$E_i = \frac{\Delta\sigma}{\Delta\varepsilon}$$

or:

$$\Delta\varepsilon = E_i \Delta\sigma$$

which is better known as Hooke's Law.

The slope, E, is known as Young's Modulus and it is NOT constant for any given soil. This modulus, which represents the stiffness of the soil, is dependent on the effective confining stress, and it can change throughout the entire problem domain depending on the stresses throughout the domain. So, the parameter we are solving in SIGMA/W is strain, and in order to know strain we must know the stress and the stiffness. However, the stiffness is itself a function of stress. This circular logic is what makes the solution a non-linear process, and it is why we have to iterate to reach a converged result.

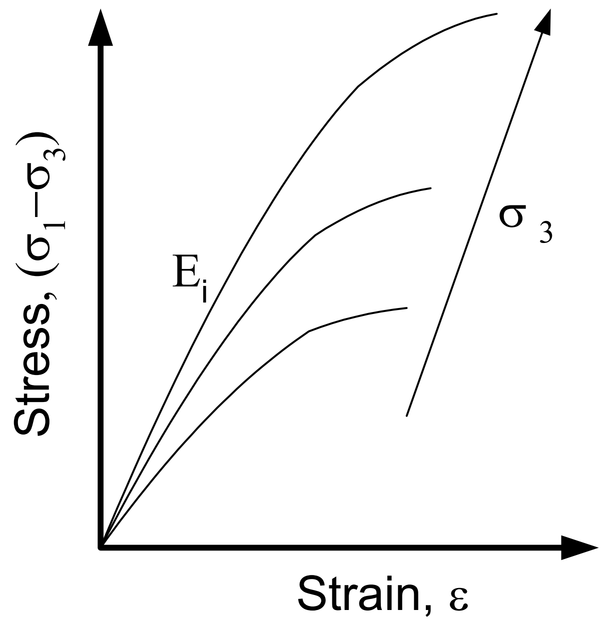


Figure 3-11 Stress-strain behavior for various confining loads during testing

3.7 Total stress, effective stress and pore-water pressures

As we have been discussing in the previous few sections, the presence of water under pressure can affect both how we obtain material properties and how they are used in the solution. Following common geotechnical practice, material properties can be specified in SIGMA/W using either effective stress parameters for analyses of drained soils, or total stress parameters for undrained soils. If you want to model a situation using effective stress parameters with pore-water pressure change, then you must do a fully coupled consolidation type analysis where pore-water pressures are computed simultaneously using SEEP/W. An uncoupled consolidation analysis does not compute pore-water pressure changes due to changes in total stress, because the pore-water pressure changes are computed independent of total stress changes in an external program such as SEEP/W, SIGMA/W, VADOSE/W or QUAKE/W.

When you specify the material properties as “effective stress” parameters, then SIGMA/W will take the existing total stress, subtract the specified pore-water pressure and arrive at an effective stress. This effective stress is then the value used to obtain the soil stiffness in a non-linear soil model. The solver then computes the change in total stress due to loading and it ADDS back on the original pore-water pressures in order to report a new effective stress at the end of the load step. So, in every case where “effective stress” parameters are specified and the analysis is not fully coupled, the change in effective stress will always equal the change in total stress.

In every case where “effective stress” parameters are specified and the analysis is not fully coupled, the change in effective stress will always equal the change in total stress.

In a load-deformation analysis, materials with properties specified using effective stress parameters and materials specified using total stress properties can be intermixed. This is also true in a fully coupled consolidation analysis however in this case if a material is a total stress material or an effective stress material with no pore-water pressure change, its pore-water pressures will remain at the initial condition value throughout the analysis. In other words, any water that drains into these regions will not cause a pressure change in these regions.

3.8 Incremental formulation

SIGMA/W is formulated for incremental analysis. For each time step, incremental displacements are calculated for the incremental applied load. These incremental values are then added to the values from the previous time step and the accumulated values are reported in the output files. In equation form:

$$\sigma_{step2} = \sigma_{step1} + \Delta\sigma_{new}$$

The incremental formulation has several advantages and, if used creatively, can be an effective way to establish unique loading or stress conditions. At this point it is important to remember that "time" is not a parameter of relevance for all analysis types except consolidation – which relies on time dependent dissipation of pore-water pressures. Because "time" is not relevant, the incremental formulation is not always sensitive to the order of loading. What is important is that the load steps can be added or subtracted such that at the end of the loading sequence, the ground stresses reflect the desired conditions.

Consider for example that you want to set up some initial conditions such that the total horizontal stress at a depth of 10 m is 200 kPa , while the total vertical stress is only 100 kPa. Assume, for now, that the unit weight of the soil is 20 kN/m^3. It is not common in reality to have the horizontal stress be greater than the vertical stress, but this example makes a good point of illustrating how some creative thinking using incremental load steps can work to your advantage.

- Step 1: Set Poisson's ratio to be 0.49 which results in the horizontal and vertical stresses at a depth of 10 m being equal to 200 kPa.
- Step 2: Change the unit weight to a value of -10 kN/m^3 and give Poisson's ratio a value of zero. Solve a load / deformation analysis using the results of step 1 as the initial conditions for step 2. The result of this load step is to subtract 100 kPa from the vertical stresses and 0 kPa from the horizontal stress.

In both cases, the end result is a horizontal stress of 200 kPa and a vertical stress of 100 kPa.

Obviously, if you had a choice, you would choose the one step approach to get the desired result. The point of this example is that the incremental formulation allows to you add and subtract loads, such that you end up with the ground stress conditions you think exist in the field.

Body forces – unit weight

The body force is the self weight of the soil and it must be included in an analysis if actual stresses in the ground are desired, or if a non-linear soil model requires that the actual stresses are known so that the correct soil stiffness values can be obtained. The body force can be left out of a linear-elastic soil model analysis, however, you must realize that any stresses reported are only incremental stresses due to the added load, and not the load plus the weight of the soil itself.

Full details of how the body loads are computed are given in the Theory chapter. For now, it is only necessary to realize that the body load applied in the model is simply a function of the unit weight specified and the size of the finite element. In other words, a finite element shape one meter square with the assumed one meter unit thickness would have one cubic meter volume. The body load applied at each node in the element would then have a proportion of the unit weight multiplied by the elemental volume.

Using the incremental approach, the body load (unit weight) is generally only applied when an element is included for the first time during an analysis. However, if there is a numerical advantage to apply the body load over several load steps, then it is possible to set this as an option.

If the body forces were applied in an in-situ analysis, and then the results from the in-situ analysis are used as the starting condition for a different analysis, you do not have to turn off the body forces in the new analysis. The software is aware that the body loads should only be considered the first time a soil becomes active. This was not the case in previous versions of the software and it was a stumbling block that many users faced when the body loads were continually added resulting in abnormally high stresses.

3.9 Serviceability versus stability

As stated in the introduction to this chapter, SIGMA/W is a very powerful tool for investigating the *serviceability* of engineered soil systems. It is not a tool for predicting *stability* of these systems.

Serviceability deals with obtaining confidence that a design will function as intended. Serviceability can tell you how a soil will move towards an unstable state (e.g. going excessively plastic), but it does not tell you, or quantify for you, what that state of instability is. Stability, on the other hand, deals with quantification of a factor of safety for an engineered system. It deals with applied loads and resisting forces. It says nothing about how a soil moves from one state to the other. A limit equilibrium tool such as SLOPE/W should be used where factors of safety are the primary concern.

In this chapter we have discussed some fundamental soil properties and how they can be measured, computed and interpreted in the model. One of the more important concepts introduced was that the strength properties of a soil are a measurement of the failure shear strength under different principal stresses. This has significant implications in a numerical stress-deformation model, because it is very likely that there are several failure planes throughout the geometry, and that they are at various orientations to the principal stresses. Given the highly varied combination, and add the fact that there is no way to quantify a soil strength once it has failed, how can a quantification of stability be made?

In a limit equilibrium analysis, the potential failure plane is specified. With this information, it is possible to determine what the shear strength is along that plane given principle stresses acting at that point. The “mobilized” shear strength can be divided by the “resisting” shear strength to determine a stability factor greater or less than 1.0. This cannot be done in stress-deformation analysis.

3.10 Modeling progression

A finite element analysis consists of two steps. The first step is to model the problem, while the second step is to formulate and solve the associated finite element equations. Modeling involves designing the mesh, defining the material properties, choosing the appropriate constitutive soil model, and defining the boundary conditions. SIGMA/W can formulate and solve the finite element equations. The modeling, however, must be done by the user; that is, the user must design an acceptable mesh, select the applicable soil properties, and control the boundary conditions. SIGMA/W cannot make judgments; this is your responsibility as the user.

Good modeling techniques require practice and experience. To assist you with this part of the analysis, this chapter presents some general modeling guidelines. The information presented is not an exhaustive statement on the "how-to" of modeling, but instead provides suggestions on how to model various conditions and how to outline conditions that may lead to difficulties.

One of the most important rules to follow in finite element modeling is to progress from the simple to the complex. When you include all the possible complexities at the start of an analysis, it often becomes difficult to interpret the results, particularly when the results are unrealistic. Moving from the simple to the complex makes it easier to pinpoint the reasons for unrealistic results. It is therefore good practice to first define a simplified version of the problem and then add complexity in stages.

In finite element modeling, it is also important that the results obtained are of a form similar to results obtained from simple hand calculations. It is easier to make this judgment if you start with a simplified version of the problem.

For example, a good method of moving from a simple analysis to a complex analysis is to start with a linear-elastic model. These results provide a reference with which to compare more complex nonlinear results, and they are also a means of checking that the boundary conditions have been applied correctly. Once you are satisfied that this part of the analysis is correct, you can move onto the nonlinear analysis with more confidence.

3.11 Units

Any set of units can be used in a SIGMA/W analysis. However, the units must be used consistently throughout the analysis. For example, if the geometry is defined in meters, then deformation is in meters.

Units must be selected for length, force and unit weight. The unit weight of water is set when the units of length are selected. Table 3-1 shows examples of consistent sets of units.

Table 3-1 Examples of consistent sets of units

Property	Units	Metric	Imperial
Geometry	L	meters	feet
Unit Weight of Water	F/L^3	kN/m^3	pcf
Soil Unit Weight	F/L^3	kN/m^3	pcf
Cohesion	F/L^2	kPa	pcf
Pressure	F/L^2	kPa	psf
Force	F	kN	lbs.
E (modulus)	F/L^2	kPa	psf

The unit weight of water must be entered correctly for whichever system of units you may have chosen. This unit weight is used in SIGMA/W to convert hydraulic heads into pressures. It is also used to calculate a limiting confining stress in some nonlinear constitutive models. For example, if the unit weight of water is kN/m^3, the minimum principal stress is limited to 1 kPa in the Elastic-Plastic Model when the internal friction angle is greater than 0. If the unit weight is 62.4 pcf, this limiting value is approximately 20 psf. For more information about limiting confining stress, see Elastic-Plastic Model in the Material Properties chapter.

4 Geometry and Meshing

4.1 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 4-1 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 4-2.

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.

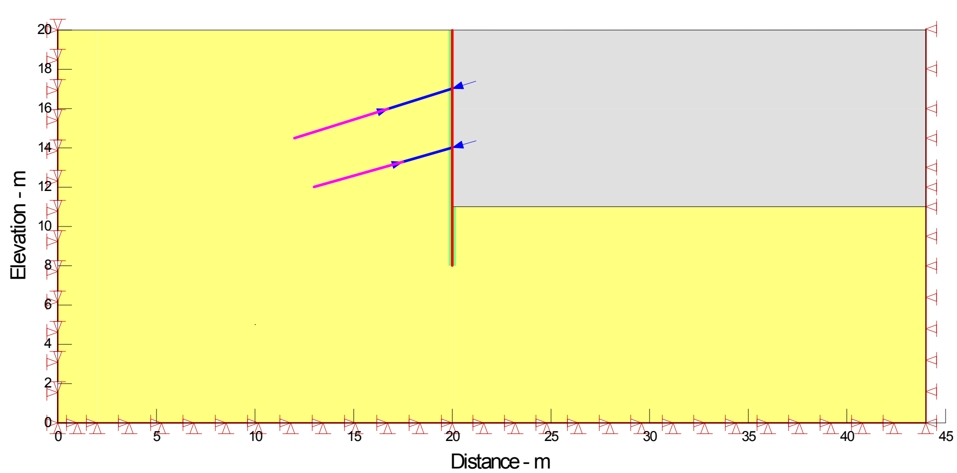


Figure 4-1 Fully defined geometry for soil excavation model

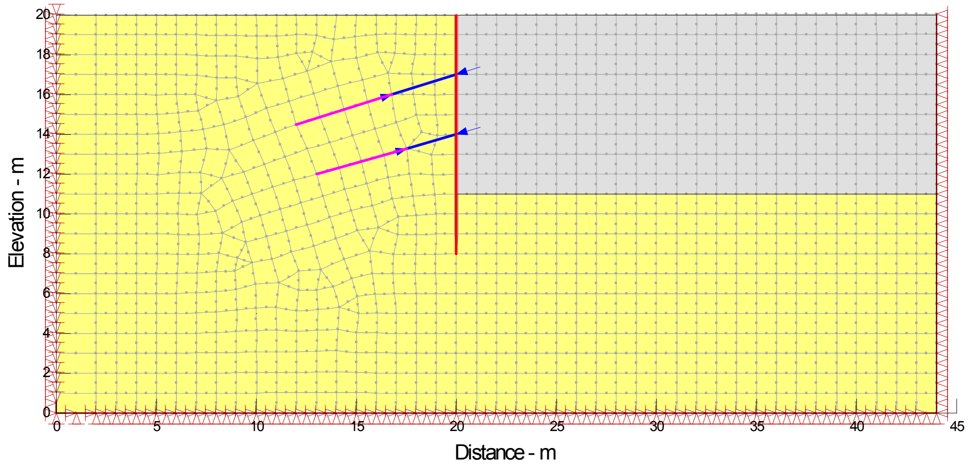


Figure 4-2 Default element discretization for model using 1m global element size constraint

4.2 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.

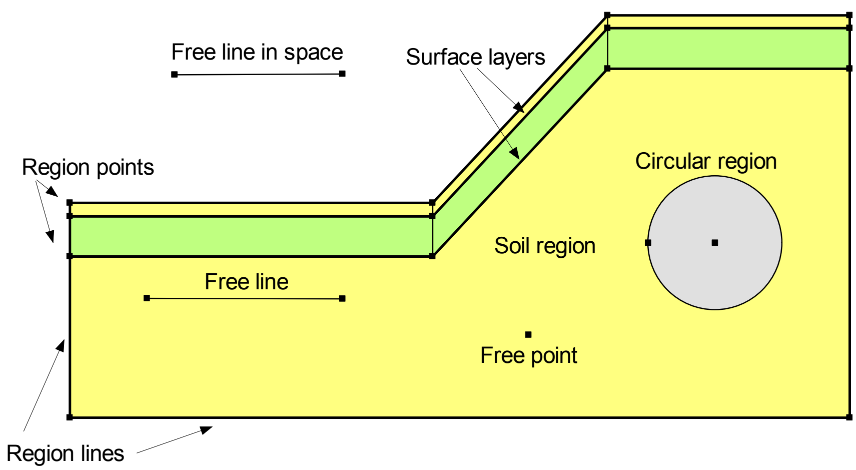


Figure 4-3 Available geometry objects

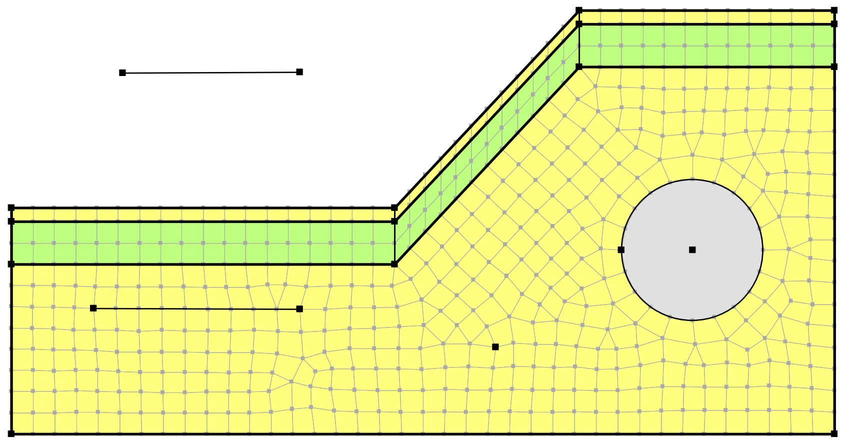


Figure 4-4 Mesh pattern for example model

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 utilization 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-side shapes like quadrilaterals or triangles or a free form, multi-sided polygon. Figure 4-5 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 4-6 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.

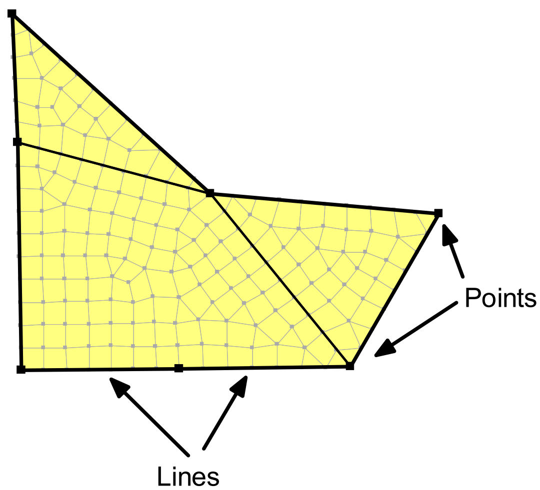


Figure 4-5 Illustration of a region's lines and points

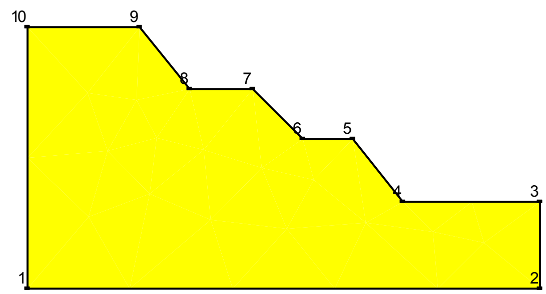


Figure 4-6 A multi-side polygonal region

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.

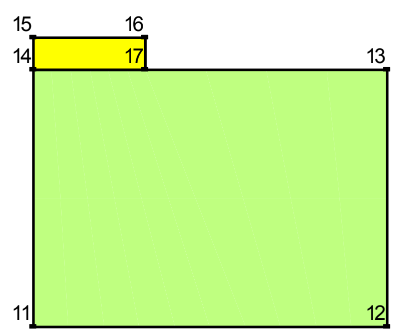


Figure 4-7 Regions of different size

Points are also required in order to join regions of different sizes and to control the meshing for specific purposes. Figure 4-7 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…

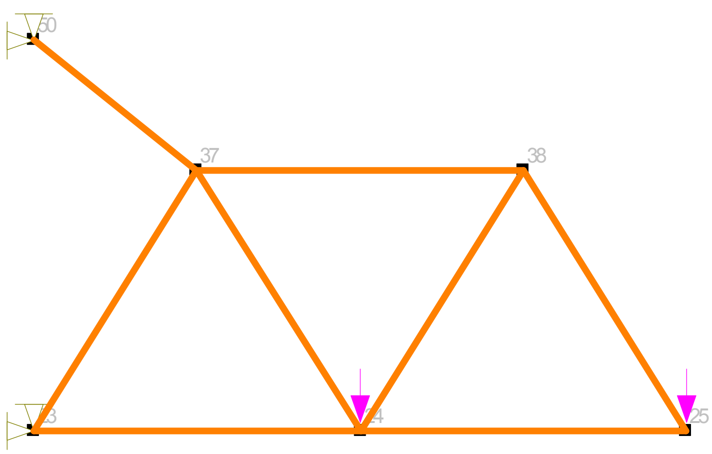


Figure 4-8 Truss loading model

In Figure 4-8 structural bar properties have been applied to a pattern of free lines in order to determine displacements under loading for the truss system.

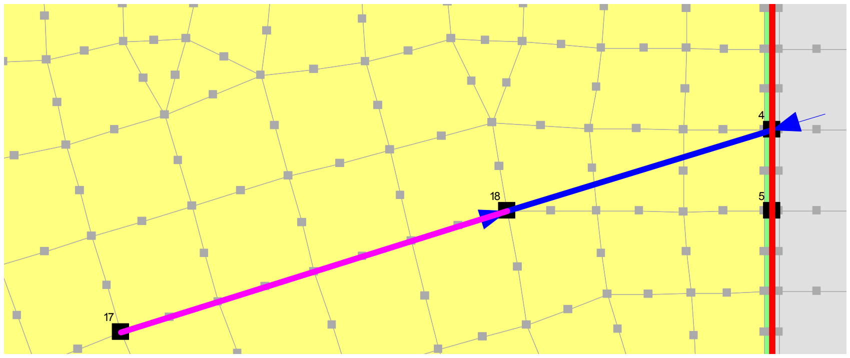


Figure 4-9 Anchors on lines (line is partially meshed)

Figure 4-9 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.

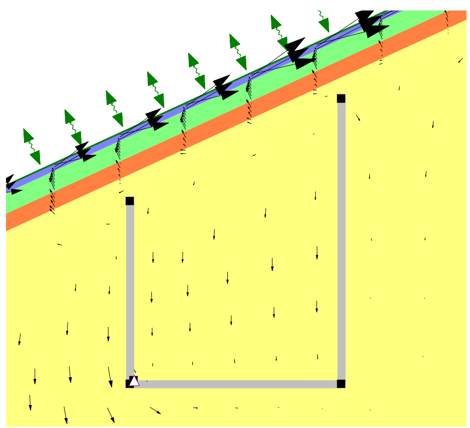


Figure 4-10 Lines with material model "none" (no flow) assigned to simulate lysimeter collection basin in waste beneath an engineered soil cover

Figure 4-10 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.

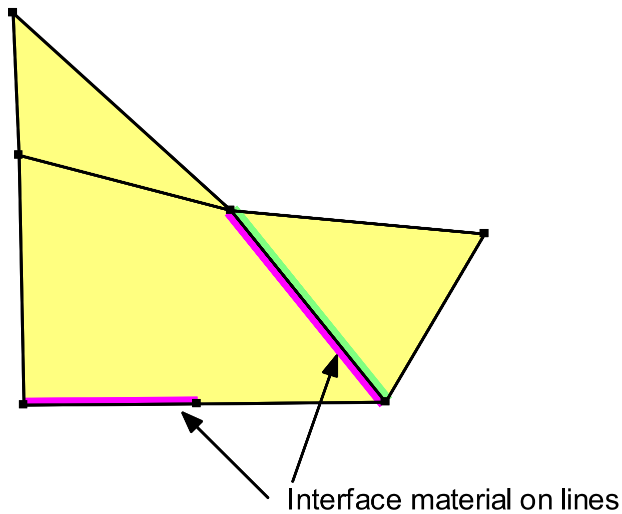


Figure 4-11 Illustration of "interface elements" on geometry lines

The discussion now will focus on how to apply and 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 4-12. 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 the 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.

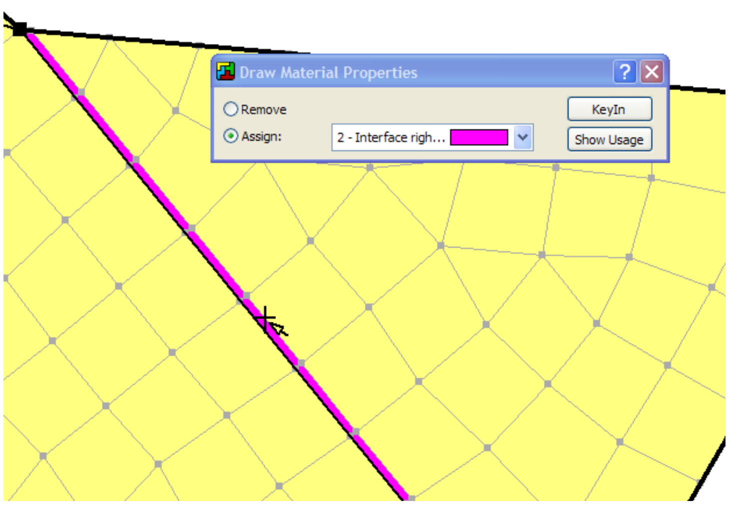


Figure 4-12 Using the Draw Line Material command to assign an interface model to a line

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 4-13.

The actual thickness of the interface elements may or may not have physical meaning but it depends on 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.

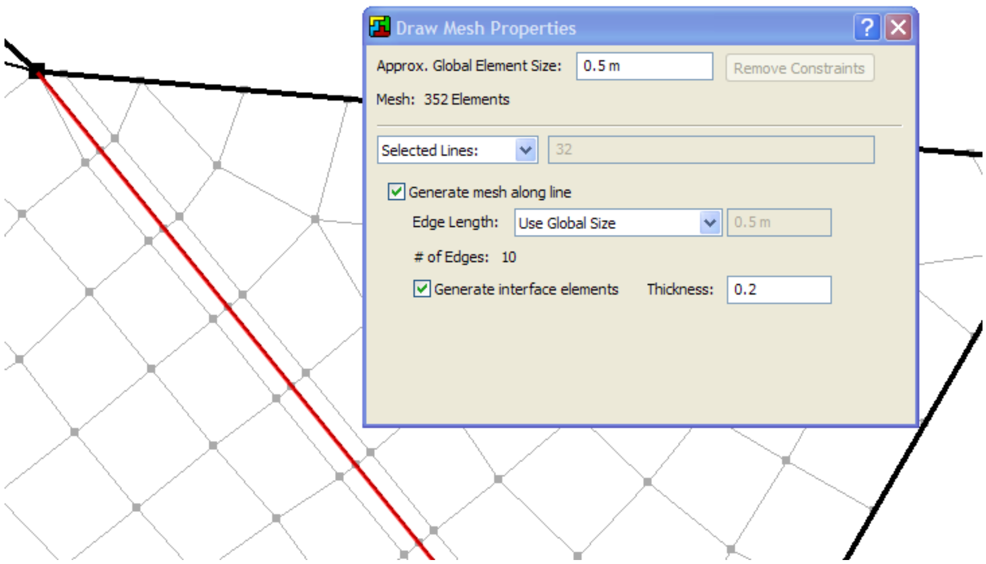


Figure 4-13 Using Draw Mesh Properties to create interface elements on selected line

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 4-14 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.

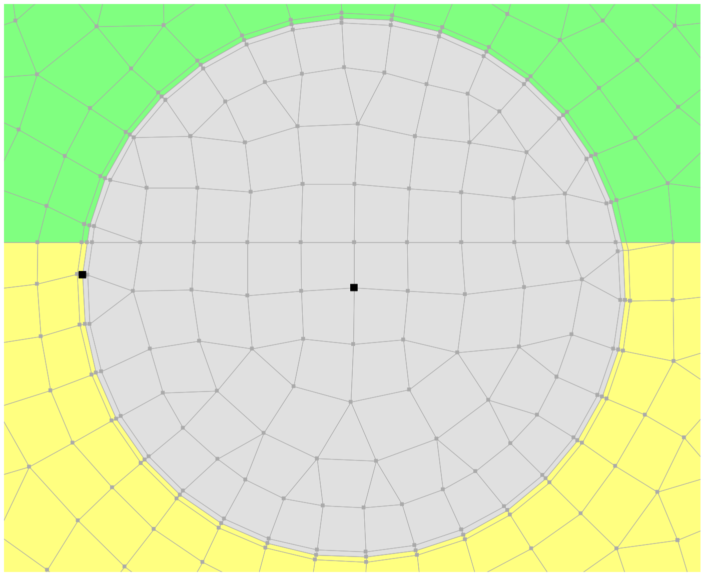


Figure 4-14 Circular region with defined center point, radius point and interface elements

4.3 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, a default mesh is generated for the soil regions 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.

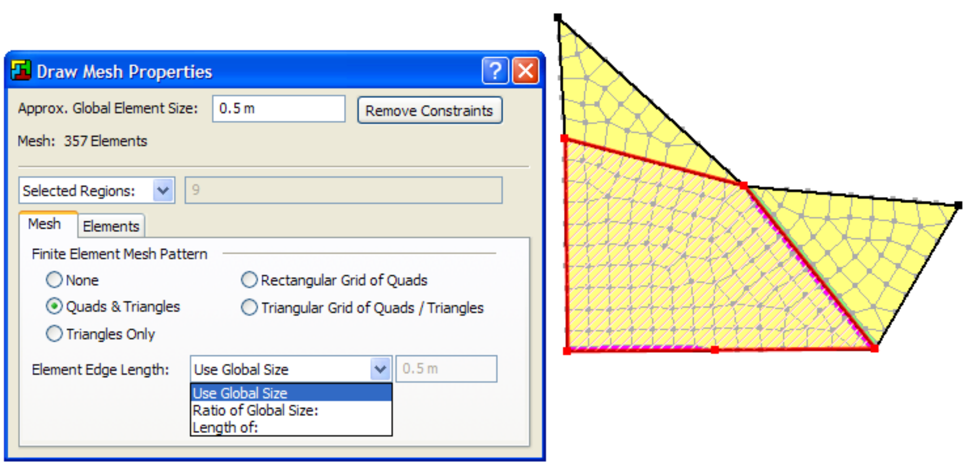


Figure 4-15 Draw Mesh Properties options

Structured mesh

Figure 4-16 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 4-16 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 4-17. In our opinion, this mesh option should be the first one you choose as it will meet your needs in most cases.

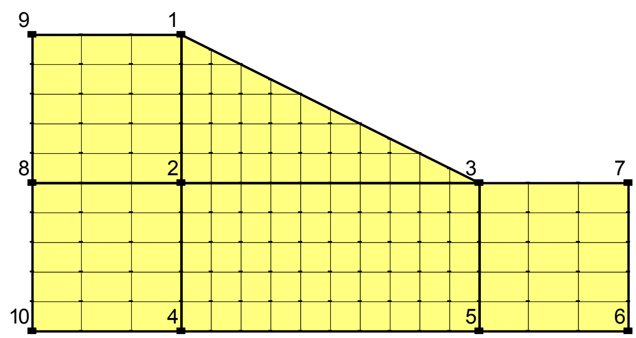


Figure 4-16 Structured mesh

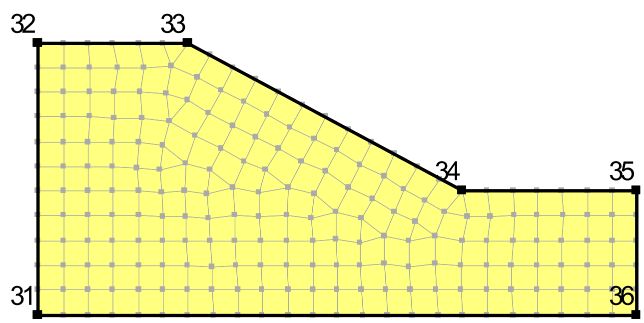


Figure 4-17 Mixed quad and triangle unstructured mesh

Unstructured triangular mesh

The diagram in Figure 4-18 shows the same section as in Figure 4-16, 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.

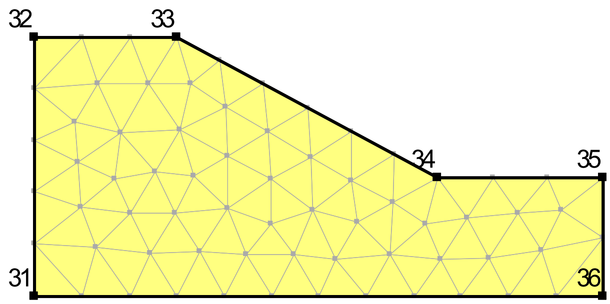


Figure 4-18 Unstructured triangular mesh

Triangular grid regions

GeoStudio has a special structured pattern for triangular regions called a triangular grid of quads/triangles. The next figure (Figure 4-19) 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.

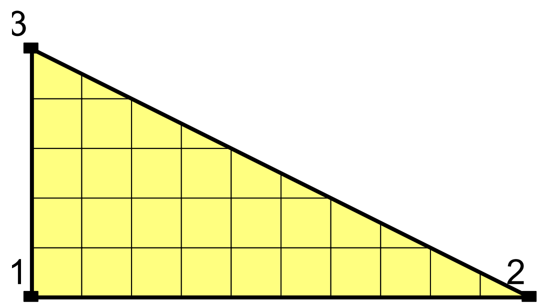


Figure 4-19 Triangular grid region

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 4-19, 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 4-20 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.

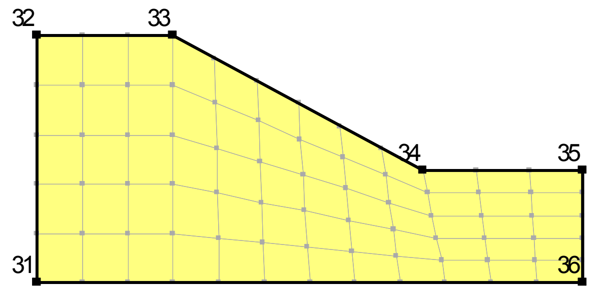


Figure 4-20 Rectangular grid of quad elements

4.4 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 4-21 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 layer has been created it will exist in all the other modules. Consequently, if a surface mesh 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.

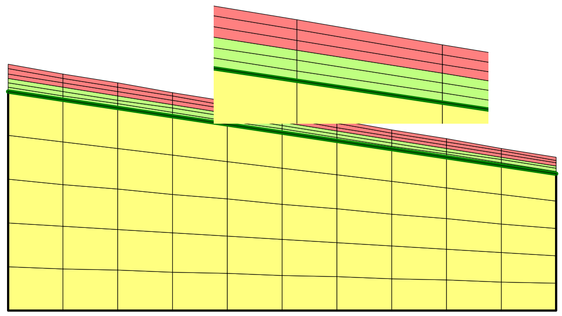


Figure 4-21 Illustration of a surface layer

Surface layers have special viewing options. Consider the two meshes illustrated in Figure 4-22. 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 4-23 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 4-23 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.

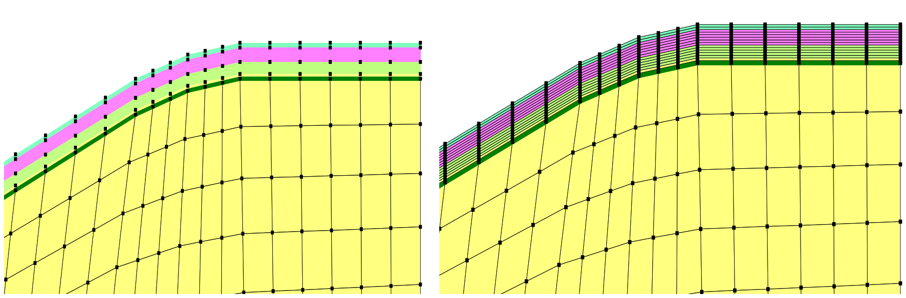


Figure 4-22 Surface layer mesh with details off (left) and on (right)

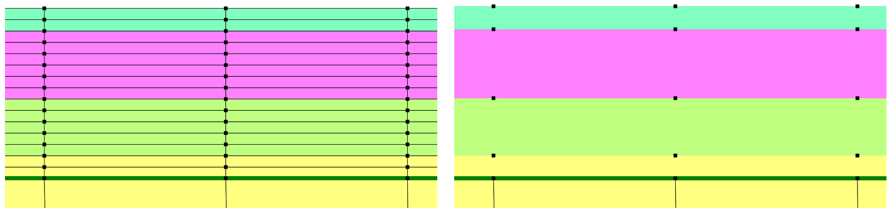


Figure 4-23 Close up of surface details on and off (note inter-layer nodes still visible in figure on right)

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 4-24 and Figure 4-25. 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.

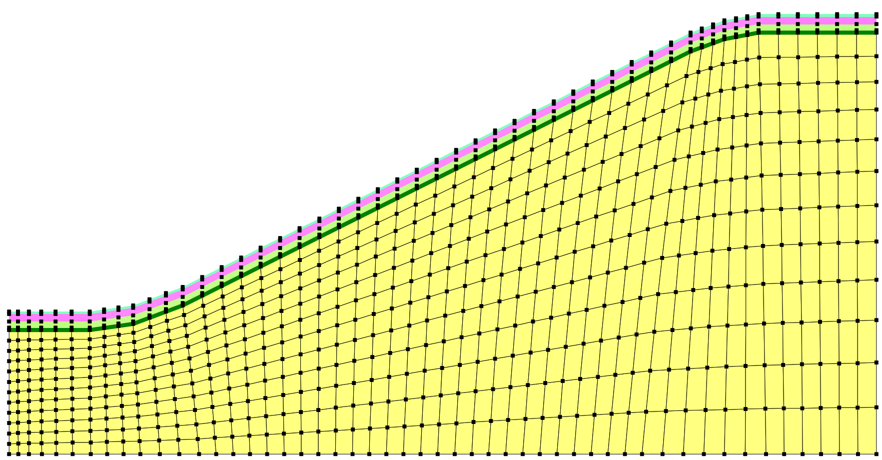


Figure 4-24 Mesh showing rounded surface slope breaks

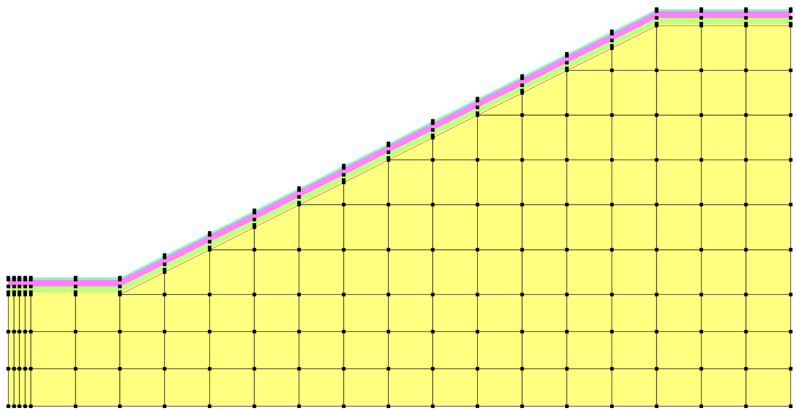


Figure 4-25 Mesh showing angular surface slope breaks

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 4-26 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.

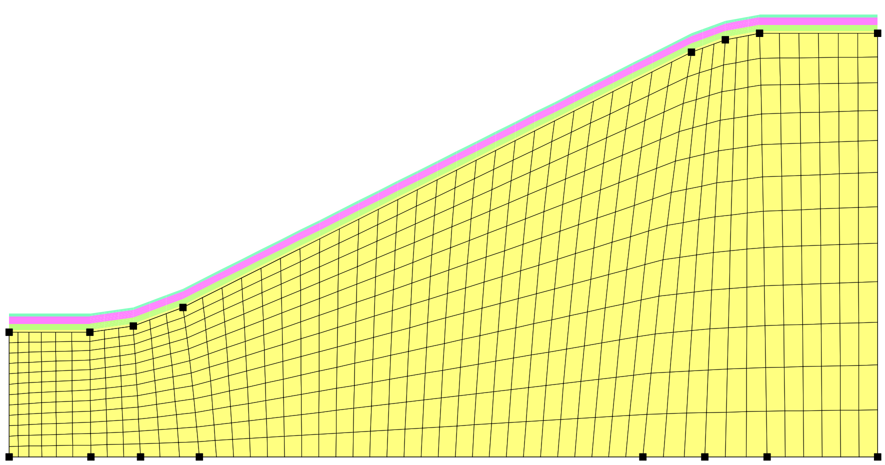


Figure 4-26 Region mesh with region corner points viewed and surface details not viewed

4.5 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 4-27. 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.

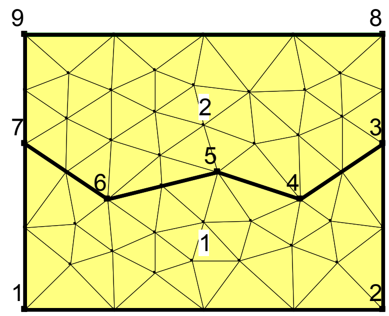


Figure 4-27 Regions joined along jagged line

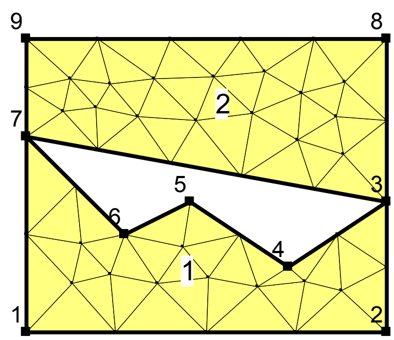


Figure 4-28 Adjoining regions with an open space

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 4-28. 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.

4.6 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.

4.7 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.

4.8 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 4-29 and Figure 4-30 show the nodes, represented as black dots.

The nodes are required and used for the following purposes:

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

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

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

Field variable distribution

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

For a linear distribution of the primary unknown, nodes are required only at the element corners. The two nodes (points) along an edge are sufficient to form a linear equation. Figure 4-29 illustrates this situation. Elements with nodes existing at the corners are referred to as first-order elements.

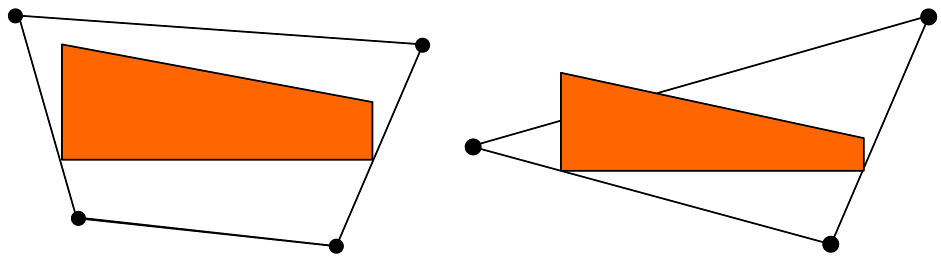


Figure 4-29 Primary field variable distribution in first-order elements

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 4-30. 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.

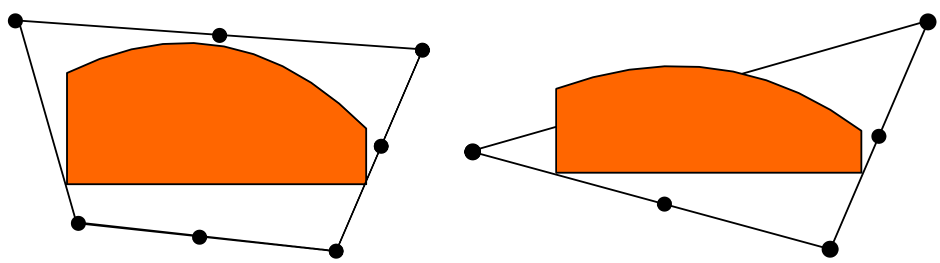


Figure 4-30 Primary field variable distribution in higher-order elements

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 4-31. 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 4-31. Common nodes are required to connect the elements as shown in Figure 4-32. Node 11, for example, is common to Elements 5, 6, 8 and 9.

Mixing elements of a different order can also create incompatibility. Figure 4-33 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 4-31 can also potentially exist at the region level. At the region level, region points need to be common to adjoining regions to ensure compatibility.

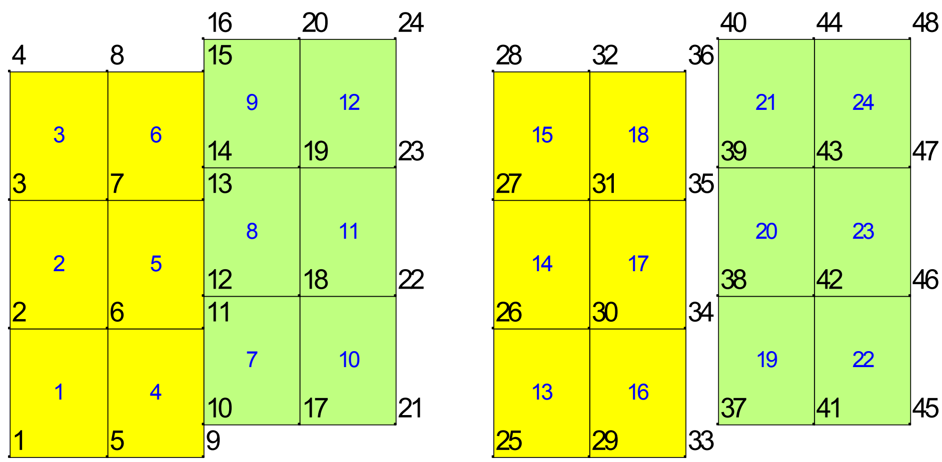


Figure 4-31 Disconnected elements – lack of compatibility

Figure 4-32 Connected elements – compatibility satisfied

Figure 4-33 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 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.

4.9 General guidelines for meshing

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

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

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

- Use as few elements as possible at the start of an analysis. Seldom is it necessary to use more than 1000 elements to verify concepts and get a first approximate solution.
- All elements should be visible to the naked eye when the mesh is printed at a zoom factor of 100 % and when the horizontal and vertical scales are the same. The exception to this guideline is 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 4-34. 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 4-34. 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.

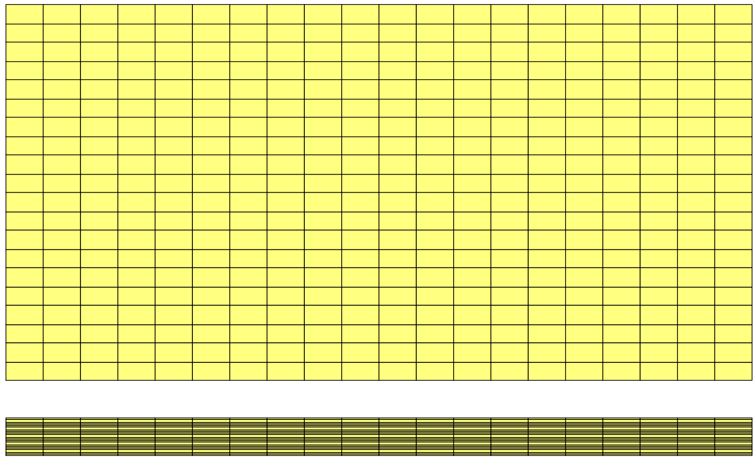


Figure 4-34 Mesh at an exaggerated scale (upper) and at a true scale (lower)

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 4-35 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 4-35 is adequate to analyze the seepage though the core.

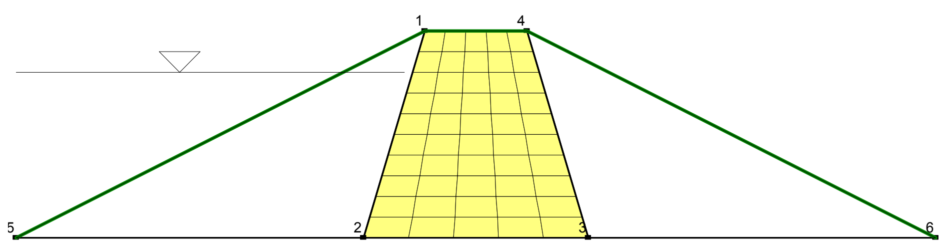


Figure 4-35 Modeling core of zoned dam

Figure 4-36 shows the total head contours (equipotential lines) in the core. From this the seepage quantities through the core can be computed.

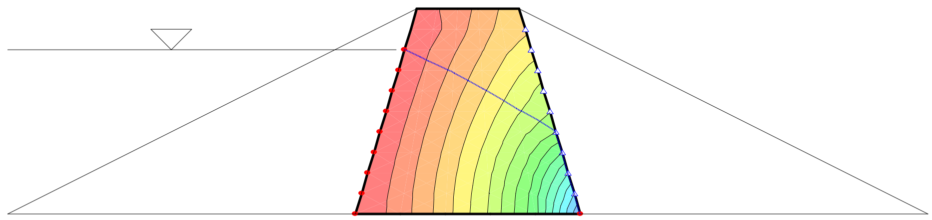


Figure 4-36 Equipotential lines in core of dam

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 4-37. 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 4-37 the rock shells have no conductivity function assigned to the material.

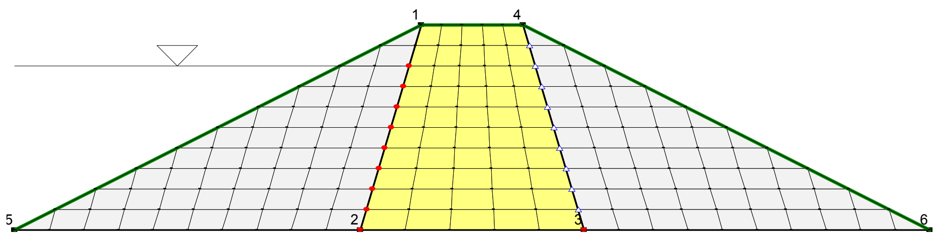


Figure 4-37 Mesh with null elements in shells of dam

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 4-35, the core can be analyzed in isolation. This would not be possible in physical modeling. The dam with a toe drain in Figure 4-38 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.

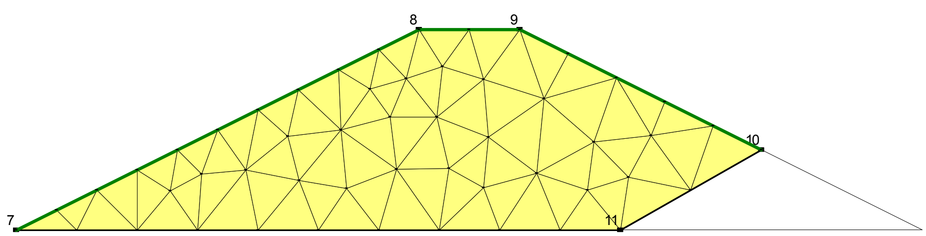


Figure 4-38 Dam with a toe drain

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.

5 Material Models and Properties

SIGMA/W includes six different soil constitutive models plus a new option for you to create a user add-in constitutive model. For each of these models, the behavior will be different depending on whether you are assigning the model to a total stress, effective stress with no pressure change, or effective stress with pore-water pressure change category.

To an inexperienced user, it may be difficult to decide which model to select for a particular application. While the ultimate choice is your responsibility, there are some points that you should consider; primarily, the material stiffness, tolerable displacement, and stability.

Consider the case of a heavy industrial structure founded on highly over-consolidated soil. Settlement is often the main design criterion, and the settlement must be fairly small. To ensure that the settlement is small, the applied loads are kept low relative to the ultimate capacity of the soil. The load-displacement response therefore is likely linear elastic along the lower initial portion of the stress-strain curve. A simple linear elastic analysis is adequate, and little would be gained by using a nonlinear analysis.

In the case of the construction of an embankment, considerable yielding and deformation can perhaps be tolerated without affecting the serviceability of the structure. A nonlinear analysis is required to obtain a realistic estimate of the potential displacements; thus, a simple linear-elastic analysis could considerably underestimate the displacements.

Placing fill for an embankment on soft soil can generate excess pore pressures to the point where the stability is affected. In such a case, and especially if you are interested in how pore-water pressures are being dissipated, it may be necessary to use one of the more sophisticated effective stress models, such as the Cam-clay model, together with a coupled or uncoupled consolidation analysis.

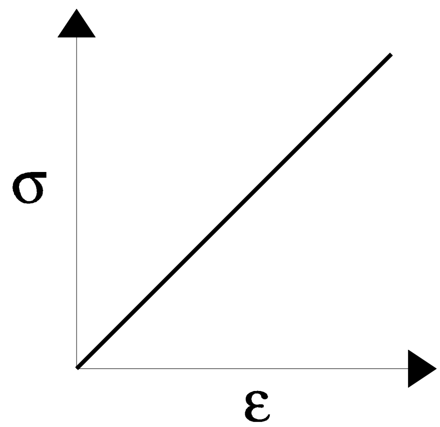
It is important to remember that each soil model is not necessarily applicable to all soil conditions. For example, the soft clay (Modified Cam-clay) model is best suited for use with slightly over-consolidated soils, not heavily over-consolidated soils. A linear-elastic model can give more realistic results for heavily over-consolidated soils than the Cam-clay models.

In summary, some thought must be given to selecting a soil model that is consistent with the soil conditions and the objective of the analysis.

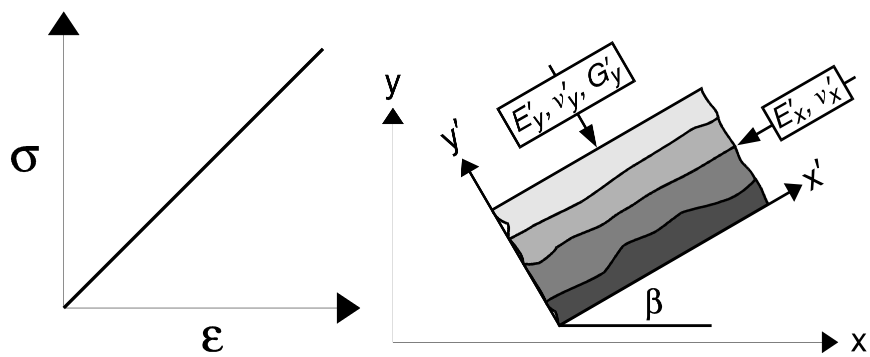
5.1 Constitutive models overview

SIGMA/W is formulated for several elastic and elasto-plastic constitutive soil models. All models may be applied to two-dimensional plane strain and axisymmetric problems. The following models are supported:

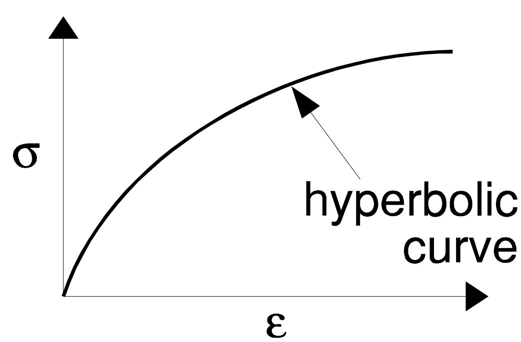
- Linear-elastic



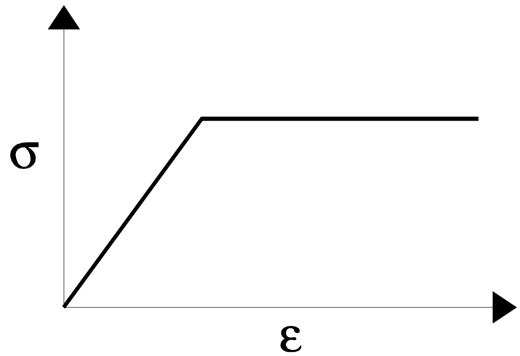
- Anisotropic Linear-elastic



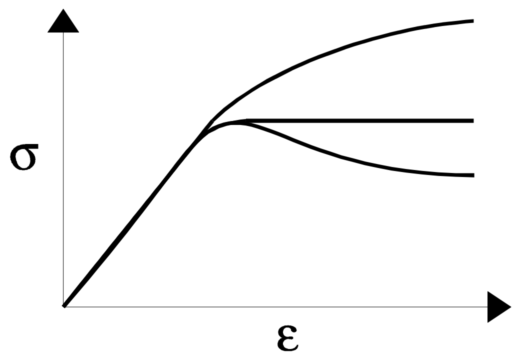
- Hyperbolic E-B



- Elastic-plastic (Mohr-Coulomb or Tresca)



- Soft Clay - Modified Cam-clay (Critical State)



Each of these is discussed in greater detail below.

5.2 Linear-elastic model

The simplest SIGMA/W soil model is the linear elastic model for which stresses are directly proportional to the strains. The proportionality constants are Young's Modulus, *E*, and Poisson's Ratio, ν. The stresses and strain are related by the equation:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \end{Bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 \\ \nu & 1-\nu & \nu & 0 \\ \nu & \nu & 1-\nu & 0 \\ 0 & 0 & 0 & \dfrac{1-2\nu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \end{Bmatrix}$$

For a two-dimensional plane strain analysis, ε_z is zero.

It is noteworthy that when ν approaches 0.5, the term $(1-2\nu)/2$ approaches zero and the term $(1-\nu)$ approaches ν. This means that the stresses and strains are directly related by a constant, which is representative of pure volumetric strain. Furthermore, the term $E/\left[(1+\nu)(1-2\nu)\right]$ tends towards infinity as $(1-2v)$ approaches zero. Physically, this means that the volumetric strain tends towards zero as Poisson's ratio, ν, approaches 0.5.

For computational purposes, ν can never be 0.5. Even values greater than 0.49 can cause numerical problems. Consequently, SIGMA/W limits the maximum value for Poisson's ratio, ν, to 0.49.

Table 5-1 Linear-elastic properties

Edit Box Label	Property
E Modulus	Young's Modulus
Poisson's Ratio	constant value
Cohesion	constant value
Friction angle	degrees

Data for the linear-elastic model includes the cohesion and friction angle. This information is not used in the solution, but it is used in the Contour program to help illustrate regions of the soil where the computed stresses have exceeded the yield strength. Recall that for a linear-elastic model, there is no yield value defined and computed strains may be very unrealistic. Using cohesion and friction angle along with the Mohr-Coulomb failure criterion will enable the computed shear stress to be compared visually with the theoretical yield stresses.

5.3 Anisotropic elastic model

Natural ground deposits are often stratified and inclined. Therefore, it is desirable to consider the possibility of having different stiffness values in two orthogonal directions. Consider the case illustrated in Figure 5-1. The soil strata are cross-anisotropic in the local orthogonal directions x' and y' with x', making an angle β with the global x-axis. The sign convention for β can be described as follows: when counter-clockwise from the x-axis, β is positive. The anisotropic elastic parameters in the local directions are defined by the following parameters:

- in the x′-direction: $E_{x'}, \nu_{x'}$
- in the y′-direction: $E_{y'}$
- coupling between x′ and y′: $G_{xy'}, \nu_{yx'}$

The parameter $\nu_{xy'}$ is the Poisson's Ratio of horizontal (x') strain to vertical (y') strain caused by a stress change in the y'-direction. These parameters must satisfy the following restrictions (Pickering, 1970):

$$E_{x'}, E_{y'}, \text{and } G_{xy'} > 0,$$
$$-1 < \nu_{x'} < 1, \quad \text{and,}$$
$$1-\nu_{x'} > 2\left(E_{x'}/E_{y'}\right)\nu_{yx'}$$

The constitutive matrix $[C']$ for anisotropic conditions is symmetric and can be defined as (Britto and Gunn , 1987):

$$[C'] = c_0 \begin{bmatrix} c_{11} & c_{12} & c_{13} & 0 \\ c_{21} & c_{22} & c_{23} & 0 \\ c_{31} & c_{32} & c_{33} & 0 \\ 0 & 0 & 0 & c_{44} \end{bmatrix}$$

where:

$$c_0 = \frac{E_y}{(1+\nu_{x'})\left(1-\nu_{x'}-2\left(E_{x'}/E_{y'}\right)\nu_{yx'}^2\right)}$$

$$c_{11} = \left(E_{x'}/E_{y'}\right)\left[1-\left(E_{x'}/E_{y'}\right)\nu_{yx'}^2\right]$$

$$c_{12} = \left(E_{x'}/E_{y'}\right)\nu_{yx'}\left(1+\nu_{x'}\right)$$

$$c_{13} = \left(E_{x'}/E_{y'}\right)\left[\nu_{x'}+\left(E_{x'}/E_{y'}\right)\nu_{yx'}^2\right]$$

$$c_{21} = c_{12}$$

$$c_{22} = \left(1-\nu_{x'}^2\right)$$

$$c_{23} = c_{12}$$

$$c_{31} = c_{13}$$

$$c_{32} = c_{23}$$

$$c_{33} = c_{11}$$

$$c_{44} = G_{xy'}/c_0$$

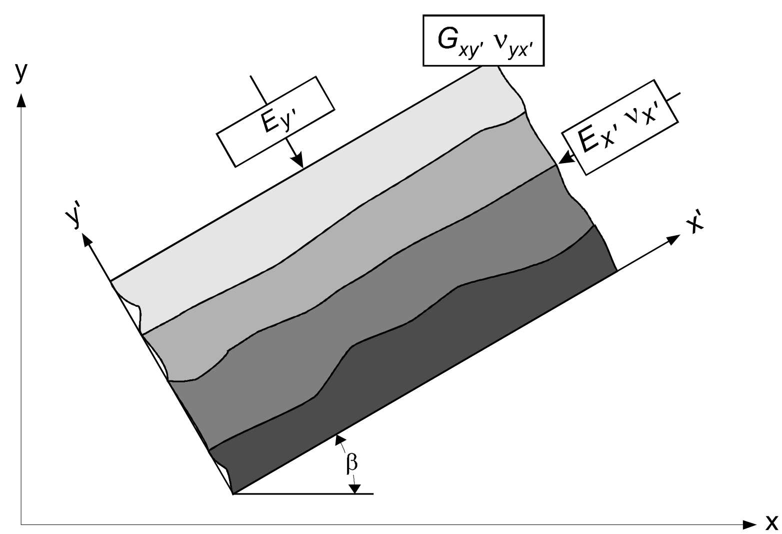


Figure 5-1 Anisotropic material properties

When the local and global axes do not coincide (when β is non-zero), it is necessary to transform the $[C']$ matrix from the local x'-y' coordinate system to the global x-y coordinate system. The transformation of $[C']$ to $[C]$ is given by the following equation (Zienkiewicz and Taylor, 1989):

$$[C]=[T][C'][T]^T$$

where:

$$[T]=\begin{bmatrix} \cos^2\beta & \sin^2\beta & 0 & -2\sin\beta\cos\beta \\ \sin^2\beta & \cos^2\beta & 0 & 2\sin\beta\cos\beta \\ 0 & 0 & 1 & 0 \\ \sin\beta\cos\beta & -\sin\beta\cos\beta & 0 & \cos^2\beta-\sin^2\beta \end{bmatrix}$$

Table 5-2 Anisotropic elastic properties

Edit Box Label	Property
E Modulus (1)	Young's Modulus in *x´*-direction, (E_x)
P. Ratio (1)	Poisson's ratio in *x´*-direction, ($\nu_{x'}$)
E Modulus (2)	Young's modulus in *y´*-direction, (E_y)
P. Ratio (2)	Poisson's ratio, ratio of *x´*-strain to *y´*-strain caused by *y´*-stress, ($\nu_{y'}$)
G Modulus (2)	Shear modulus, ($G_{xy'}$)
Angle	Inclination of the strata in degrees from *x*-axis, (β)

5.1 Hyperbolic E-B model

The hyperbolic model described by Duncan et al. (1980) is sometimes referred to as the hyperbolic E-B constitutive model because the bulk modulus (B) is considered to be constant during loading while the elastic modulus (E) varies according to a hyperbolic relationship (Duncan and Chang, 1970). The assumptions regarding the formulation result in non-linearity of the stress-strain response (O-A-B; Figure 5-2) and the volumetric response. The volumetric strain increment tends towards zero at failure as a result of the assumption of a constant bulk modulus, which is in-keeping with the concept of a critical state.

The hyperbolic E-B constitute model also allows for a unique definition of the unload-reload modulus. The stress-strain response during unloading-reloading (B-C; Figure 5-2) can therefore demonstrate a stiffer response as compared to the initial loading response (O; Figure 5-2), which is controlled by the specified initial elastic modulus and Poisson's ratio.

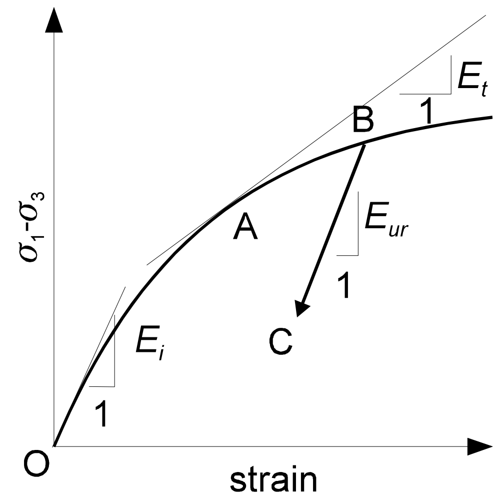


Figure 5-2 Non-Linear stress-strain behavior

Duncan and Chang (1970) derived the following expression for the tangent modulus, which is the instantaneous slope of a non-linear stress-strain curve:

$$E_t = \left[1 - \frac{R_f \ (\sigma_1 - \sigma_3)(1 - \sin\phi)}{2c(\cos\phi) + 2\sigma_3 \sin\phi} \right]^2 E_i$$

where c and ϕ are the strength parameters and σ_1 and σ_3 the major and minor principle stresses. The parameter R_f is the failure ratio, defined as:

$$R_f = \frac{(\sigma_1 - \sigma_3)_f}{(\sigma_1 - \sigma_3)_{ult}}$$

The deviator stress at failure $(\sigma_1 - \sigma_3)_f$ is less than the ultimate value $(\sigma_1 - \sigma_3)_{ult}$; consequently, the failure ratio R_f is less than 1.0 and is generally between 0.5 and 0.9 for most soils (Duncan et al., 1980). Mobilization of the full shear strength of the soil causes the tangent modulus to be given by:

$$E_t = \left[1 - R_f\,(1)\right]^2 E_i$$

which indicates that the minimum tangent modulus is controlled by the failure ratio R_f. For example, $E_t = 0.4(E_i)$ if $R_f = 0.8$. In addition, the minimum E_t is limited to 10% of P_a (atmospheric pressure).

A confining stress (or depth) dependency is introduced through the initial modulus, which is given by

$$E_i = K_L P_a \left(\frac{\sigma_3}{P_a}\right)^n$$

where K_L is the modulus number and n the modulus exponent; both of these parameters are dimensionless numbers. The atmospheric pressure P_a is introduced to make the relationship independent of the unit set. A typical value for the modulus exponent is about 0.5.

SIGMA/W has the equation for the initial modulus implemented as an estimation algorithm. The estimation algorithm includes an input for the earth pressure coefficient in order to calculate the initial confining stress as $\sigma_3 = K_0\sigma_1$. The major principle stress is assumed equal to the vertical stress (Figure 5-3). The use of a spline data point function is advantageous because: a) controls can be placed on the minimum and maximum modulus values; b) the function can be defined in terms of total stress or effective stresses and can take any shape (including a linear approximation); and c) the user can visualize the relationship before solving the analysis. The estimation routine in SIGMA/W includes a number of sample functions for a range of particle size distributions. The parameters used to define the sample functions are based on the compilations presented by Duncan and Chang (1970) and Duncan et al. (1980).

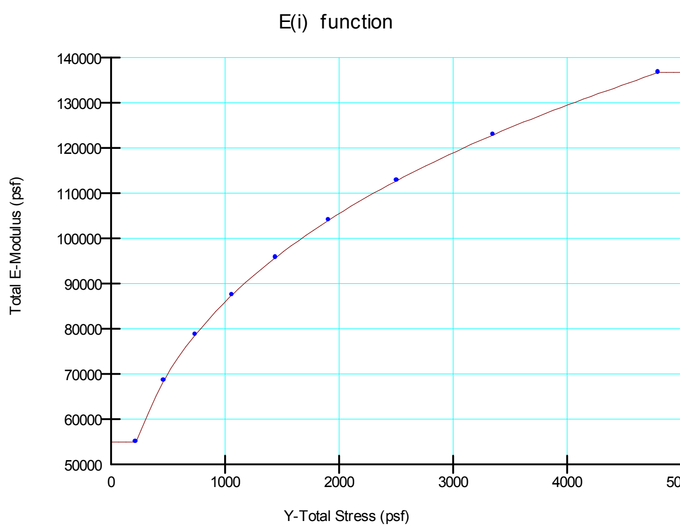


Figure 5-3 A Typical E_i function

The expression for the unload-reload modulus is given by:

$$E_{ur} = K_{ur} P_a \left(\frac{\sigma_3}{P_a} \right)^n = \frac{K_{ur}}{K_L} E_i$$

The unload-reload modulus is calculated internally by SIGMA/W in accordance with the initial stress-dependent modulus E_i and a user input ratio of unload-reload modulus number K_{ur} relative to the modulus number K_L. The value of K_{ur} is always larger than the value of $K_L$and may be 20% greater than K_L for stiff soils such as dense sands. For soft soils, like loose sands, K_{ur} could be three times as large as K_L (Duncan et al., 1980).

Non-linearity and stress-dependency is often reflected in the volume change response of the soil. The volume strain increments tend towards zero as a critical state is reached. A non-linear volume change response is accommodated by the hyperbolic E-B model by making the bulk modulus constant during loading. Duncan et al. (1980) describe a simple procedure for measuring the bulk modulus (B) of the soil, plotting it as a function of (initial) confining stress, and approximating the non-linear relationship by an equation of the same form used to describe the initial and unload-reload moduli. The relationship requires a uniquely defined bulk modulus number and bulk modulus exponent.

To reduce the input requirements of the hyperbolic model while retaining the non-linearity of the volume response, the bulk modulus is instead calculated internally according to the general 3D relationship between B, the initial modulus, and Poisson's ratio μ:

$$B = \frac{E_i}{3(1 - 2\mu)}$$

This assumption retains the confining stress-dependency of the bulk modulus.

Negative pore-water pressure

When the material properties are defined as effective parameters, it is possible in SIGMA/W to attach a Volumetric Water Content function to the soil such as shown in Figure 5-6. The first data point at the lowest suction is taken to be the water content at saturation θ_s. A water content equal to 5% of θ_s is taken to be the residual water content θ_r. The soil cohesion is then computed as,

$$C = c' + (u_a - u_w) \tan \phi' \frac{(\theta - \theta_r)}{(\theta_s - \theta_r)}$$

The parameter Ø′ is the effective soil friction angle and $(u_a - u_w)$ is the soil suction. This means that when the water content is at saturation ($\theta = \theta_s$), 100% of the suction contributes to the strength, and when the water content is at the residual value ($\theta = \theta_s$) the suction makes no contribution to the strength. The water content function is in essence used to apportion the suction contribution to the strength. This concept is based on a publication by Vanapalli et al. (1996).

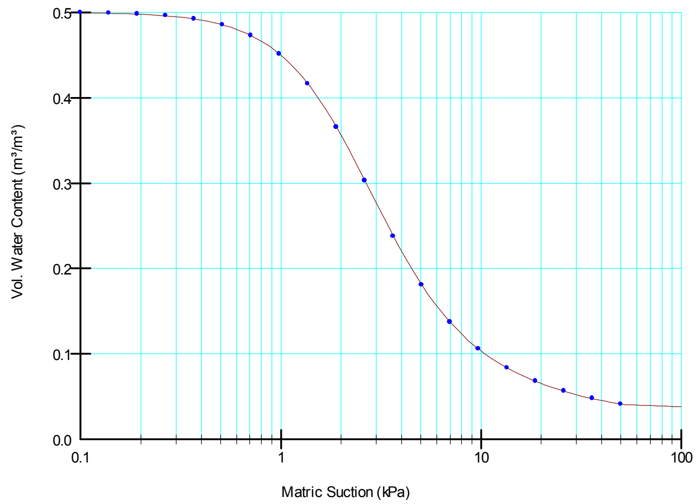


Figure 5-4 Volumetric water content function

Yield zones

From a theoretical standpoint, yield conditions cannot be defined when using a non-linear elastic model. In order to show high shear stress zones in a non-linear elastic material, SIGMA/W identifies such zones as "yielded" when the following criterion is satisfied:

Equation 5-1 $$\frac{\sigma_1 - \sigma_3}{2} - \frac{\sigma_1 + \sigma_3}{2}\sin\phi \geq R_f c\cos\phi$$

In Duncan and Chang's formulation of the hyperbolic model, R_f, the failure ratio, is used in the following manner:

$$(\sigma_1 - \sigma_3)_f = R_f(\sigma_1 - \sigma_3)_{ult}$$

The ultimate strength term, $(\sigma_1 - \sigma_3)_{ult}$, represents the asymptote which the hyperbolic stress-strain curve will approach at large strains. The $(\sigma_1 - \sigma_3)_f$ is the deviatoric stress at failure. From a Mohr's diagram, it can be seen that:

$$\frac{(\sigma_1 - \sigma_3)_{ult}}{2} - \frac{(\sigma_1 + \sigma_3)_{ult}}{2}\sin\phi \geq c\cos\phi$$

When multiplied by R_f and substituting in the second equation above, this equation can be written as follows:

$$\frac{(\sigma_1 - \sigma_3)_f}{2} - \frac{(\sigma_1 + \sigma_3)_f}{2}\sin\phi \geq R_f c\cos\phi$$

Comparing this equation with Equation 5-1, it is seen that the inequality given in Equation 5-1 provides an indicator as to how close the stress state is to the failure state.

Volumetric strains

SIGMA/W is based on an incremental formulation and within each incremental applied load, the soil is considered to behave in a linear-elastic manner and the relationship between stress and strain is assumed to be governed by Hook's Law, which in its simplest and conceptual form is:

$$\sigma = E\varepsilon$$

For a two-dimensional plane strain problem in matrix notation this relationship can be written as:

$$\{\sigma\} = [C]\{\varepsilon\}$$

where $[C]$ is the constitutive (element property) matrix and is given by:

$$[C] = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 \\ \nu & 1-\nu & \nu & 0 \\ \nu & \nu & 1-\nu & 0 \\ 0 & 0 & 0 & \dfrac{1-2\nu}{2} \end{bmatrix}$$

where:

E	=	Young's modulus
v	=	Poisson's ratio

The bulk modulus *B* by definition is the ratio of a change in mean stress divided by a change in volumetric strain. In equation form,

$$B = \frac{\Delta\sigma_{mean}}{\Delta v_{volumetric}}$$

From the theory of elasticity, *B* is related to *E* and *v* as:

$$B = \frac{E}{3(1-2\nu)}$$

We can write *v* in terms of *E* and *B* as:

$$\nu = 0.5 - \frac{E}{6B}$$

This makes it possible to account for the volumetric strain as the soil is stressed to failure. Numerically, this can be done by holding *B* constant and then varying *E* based on the tangent modulus E_t.

In SIGMA/W, B_i is computed from E_i (initial). As the soil is sheared and the tangent modulus E_t changes, Poisson's ratio *v* also changes. Since E_t tends towards a low value as the stress state reaches the shear strength, *v* tends towards 0.5, and as is known from the theory of elasticity, a value of *v* equal to 0.5 means the incremental volume change is zero due to an incremental change in stress.

Consequently, in the hyperbolic model as E_t tends to zero, Poisson's ratio v tends to 0.5, and any associate volumetric strain tends to zero. Stated another way, the volumetric strain in an elemental volume of soil will be zero when the applied shear stress reaches the shear strength. This is consistent with critical state soil mechanics concepts.

This implementation in SIGMA/W makes it possible to account for volumetric strains arising from both normal and shear stress changes when using the hyperbolic model.

5.2 Elastic-plastic model

The Elastic-Plastic model in SIGMA/W describes an elastic, perfectly-plastic relationship. A typical stress-strain curve for this model is shown in Figure 5-5. Stresses are directly proportional to strains until the yield point is reached. Beyond the yield point, the stress-strain curve is perfectly horizontal.

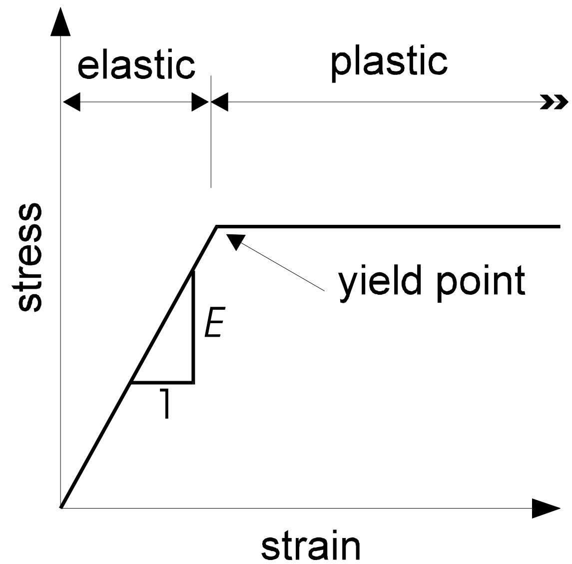


Figure 5-5 Elastic-perfectly plastic constitutive relationship

Plastic matrix

In SIGMA/W, soil plasticity is formulated using the theory of incremental plasticity (Hill, 1950). Once an elastic-plastic material begins to yield, an incremental strain can be divided into an elastic and a plastic component.

$$\{d\varepsilon\} = \{d\varepsilon^e\} + \{d\varepsilon^p\}, \quad or$$
$$\{d\varepsilon_e\} = \{d\varepsilon\} - \{d\varepsilon^p\}$$

Only elastic strain increments, dε^e, will cause stress changes. As a result, stress increments can be written as follows.

Equation 5-2
$$\{d\sigma\} = [C_e]\{d\varepsilon^e\}, \quad or$$
$$\{d\sigma\} = [C_e]\left(\{d\varepsilon\} - \{d\varepsilon^p\}\right)$$

A function which describes the locus of the yield point is called the yield function and is defined using the symbol, F. In the Elastic-Plastic model of SIGMA/W, the yield point depends only on the stress state. Consequently, the yield function can be written as follows in equation form.

$$F = F\left(\sigma_x, \sigma_y, \sigma_z, \tau_{xy}\right)$$

An incremental change in the yield function is given by:

$$dF = \frac{\partial F}{\partial \sigma_x} d\sigma_x + \frac{\partial F}{\partial \sigma_y} d\sigma_y + \frac{\partial F}{\partial \sigma_z} d\sigma_z + \frac{\partial F}{\partial \tau_{xy}} d\tau_{xy}$$

Alternatively, this equation can be written in the following matrix form.

$$dF = \left\langle \frac{\partial F}{\partial \sigma} \right\rangle \{d\sigma\}$$

The theory of incremental plasticity dictates that the yield function, F < 0, and, when the stress state is on the yield surface, dF is zero. This latter condition is termed the neutral loading condition, and, can be written mathematically as:

$$dF = \left\langle \frac{\partial F}{\partial \sigma} \right\rangle \{d\sigma\} = 0$$

The plastic strain is postulated to be:

Equation 5-3 $$\{d\varepsilon_p\} = \lambda \left\{ \frac{\partial G}{\partial \sigma} \right\}$$

where:

G	=	plastic potential function, and
λ	=	plastic scaling factor.

Substituting the plastic strain from Equation 5-3 into the incremental stress equation (Equation 5-2) gives:

$$\{d\sigma\} = [C_e]\{d\varepsilon\} - [C_e]\lambda \left\{ \frac{\partial G}{\partial \sigma} \right\}$$

Substituting the stress vector, $\{d\sigma\}$, into the neutral loading condition, the following expression for the plastic scaling factor, λ, can be derived.

$$\lambda = \frac{\left\langle \frac{\partial F}{\partial \sigma} \right\rangle [C_e]}{\left\langle \frac{\partial F}{\partial \sigma} \right\rangle [C_e] \left\{ \frac{\partial G}{\partial \sigma} \right\}} \{d\varepsilon\}$$

From the previous two equations, a relationship between stress increments and strain increments can be obtained.

$$\{d\sigma\} = \left([C_e] - [C_p] \right) \{d\varepsilon\}$$

where:

$$\left[C_p\right] = \frac{[C_e]\left\{\dfrac{\partial G}{\partial \sigma}\right\}\left\langle\dfrac{\partial F}{\partial \sigma}\right\rangle[C_e]}{\left\langle\dfrac{\partial F}{\partial \sigma}\right\rangle[C_e]\left\{\dfrac{\partial G}{\partial \sigma}\right\}}$$

To evaluate the plastic matrix, $[C_p]$, the yield function, F , and the plastic potential function, G , need to be specified.

Yield criterion

SIGMA/W uses the Mohr-Coulomb yield criterion as the yield function for the Elastic-Plastic model. The following equation provides a common form of the Mohr-Coulomb criterion expressed in terms of principal stresses.

$$F = \sqrt{J_2}\sin\left(\theta + \frac{\pi}{3}\right) - \sqrt{\frac{J_2}{3}}\cos\left(\theta + \frac{\pi}{3}\right)\sin\phi - \frac{I_1}{3}\sin\phi - c\cos\phi$$

The Mohr-Coulomb criterion can also be written in terms of the stress invariants I_1, I_2 and θ. The yield function, *F*, can then be written as follows (Chen and Zhang, 1991).

$$F = \sqrt{J_2}\sin\left(\theta + \frac{\pi}{3}\right) - \sqrt{\frac{J_2}{3}}\cos\left(\theta + \frac{\pi}{3}\right)\sin\phi - \frac{I_1}{3}\sin\phi - c\cos\phi$$

where:

$J_2 = \frac{1}{6}\left[\left(\sigma_x - \sigma_y\right)^2 + \left(\sigma_y - \sigma_z\right)^2 + \left(\sigma_z - \sigma_x\right)^2\right] + \tau_{xy}^2$ = the second deviatoric stress invariant,

$\theta = \frac{1}{3}\cos^{-1}\left(\frac{3\sqrt{3}}{2}\frac{J_3}{J_2^{3/2}}\right)$ = the lode angle,

$J_3 = \sigma_x^d\sigma_y^d\sigma_z^d - \sigma_z^d\tau_{xy}^2$ = the third deviatoric stress invariant,

$I_1 = \sigma_x + \sigma_y + \sigma_z$ = the first stress invariant,

ϕ = angle of internal friction, and

c = cohesion of the soil.

The deviatoric stress σ_i^d in the ith-direction can be defined as:

$$\sigma_i^d = \sigma_i - \frac{I_1}{3}$$

where i = x, y or z.

When the angel of internal friction, ϕ, is equal to zero, the Mohr-Coulomb yield criterion becomes the Tresca criterion (Smith and Griffiths, 1988):

$$F = \sqrt{J_2}\,\sin\left(\theta + \frac{\pi}{3}\right) - c$$

The plastic potential function, G, used in SIGMA/W has the same form as the yield function, F (i.e. $G = F$) except the internal friction angle, ϕ, is replaced by the dilation angle, ψ. Thus, the potential function is given by:

$$F = \sqrt{J_2}\,\sin\left(\theta + \frac{\pi}{3}\right) - \sqrt{\frac{J_2}{3}}\cos\left(\theta + \frac{\pi}{3}\right)\sin\phi - \frac{I_1}{3}\sin\phi - c\cos\phi$$

The derivatives of the yield function in terms of the stress invariants are computed using the chain rule of differentiation.

$$\left\langle \frac{\partial F}{\partial \sigma} \right\rangle = \frac{\partial F}{\partial I_1}\left\langle \frac{\partial I_1}{\partial \sigma} \right\rangle + \frac{\partial F}{\partial J_2}\left\langle \frac{\partial J_2}{\partial \sigma} \right\rangle + \frac{\partial F}{\partial \theta}\left\langle \frac{\partial \theta}{\partial \sigma} \right\rangle$$

Derivatives of the Mohr-Coulomb yield function, with respect to the stress invariants, can be written as follows:

Equation 5-4

$$\frac{\partial F}{\partial I_1} = -\frac{\sin\phi}{3}$$

$$\frac{\partial F}{\partial J_2} = \frac{1}{2\sqrt{J_2}}\left\{\sin\left(\theta + \frac{\pi}{3}\right) + \frac{1}{\sqrt{3}}\sin\phi\cos\left(\theta + \frac{\pi}{3}\right)\right\}$$

$$\frac{\partial F}{\partial \theta} = \sqrt{J_2}\cos\left(\theta + \frac{\pi}{3}\right) + \sqrt{\frac{J_2}{3}}\sin\phi\sin\left(\theta + \frac{\pi}{3}\right)$$

The derivatives of the stress invariants with respect to the stresses are:

$$\left\langle \frac{\partial I_1}{\partial \sigma} \right\rangle = \langle 1 \quad 1 \quad 1 \quad 0 \rangle$$

$$\left\langle \frac{\partial J_2}{\partial \sigma} \right\rangle = \left\langle \sigma_x^d \quad \sigma_y^d \quad \sigma_z^d \quad 2\tau_{xy} \right\rangle$$

$$\left\langle \frac{\partial \theta}{\partial \sigma} \right\rangle = -\frac{\sqrt{3}}{2J_2^{3/2}\sin 3\theta}\left(\left\langle \frac{\partial J_3}{\partial \sigma} \right\rangle - \frac{3J_3}{2J_2}\left\langle \frac{\partial J_2}{\partial \sigma} \right\rangle\right)$$

$$\left\langle \frac{\partial J_3}{\partial \sigma} \right\rangle = \left\langle \sigma_y^d\sigma_z^d + \frac{J_2}{3} \quad \sigma_x^d\sigma_z^d + \frac{J_2}{3} \quad \sigma_x^d\sigma_y^d + \frac{J_2}{3} - \tau_{xy}^2 \quad -2\sigma_z^d\tau_{xy} \right\rangle$$

Similarly, the derivatives of the potential function can be obtained by substituting ψ for ϕ in Equation 5-4.

The material properties required for this model are given in Table 5-3.

Table 5-3 Elastic-plastic material properties

Edit Box Text	Property
E Modulus	Initial linear-elastic stiffness of the soil
Poisson's Ratio	Constant value
Cohesion	Cohesive strength of the soil
Phi	Soil internal friction angle ϕ in degrees
Dilation Angle	Soil dilation angle ψ in degrees $(0 \le \psi \le \phi)$. If a value is not specified, the dilation angle is considered to be the same as the internal friction angle.

The soil stiffness modulus E can be specified as constant or as a function of the overburden stress the same as what was described earlier for the hyperbolic soil model.

Negative pore-water pressure

When the material properties are defined as effective parameters, it is possible in SIGMA/W to attach a Volumetric Water Content function to the soil such as shown in Figure 5-6. The first data point at the lowest suction is taken to be the water content at saturation θ_s. A water content equal to 5% of θ_s is taken to be the residual water content θ_r. The soil cohesion is then computed as,

$$C = c' + (u_a - u_w)\tan\phi' \frac{(\theta - \theta_r)}{(\theta_s - \theta_r)}$$

The parameter Ø′ is the effective soil friction angle and $(u_a - u_w)$ is the soil suction. This means that when the water content is at saturation $(\theta = \theta_s)$, 100% of the suction contributes to the strength, and when the water content is at the residual value $(\theta = \theta_s)$ the suction makes no contribution to the strength. The water content function is in essence used to apportion the suction contribution to the strength. This concept is based on a publication by Vanapalli et al. (1996).

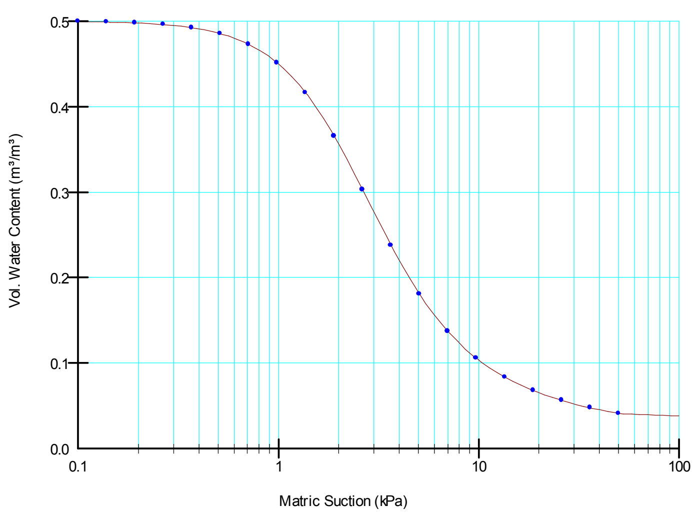


Figure 5-6 Volumetric water content function

5.3 Cam-clay model

The Cam-clay model is a critical state model as well as a hardening elastic-plastic model. Its formulation in SIGMA/W is based on presentations by Atkinson and Bransby (1978), and Britto and Gunn (1987). More detailed background and theory of the Cam-clay model can be found in these references. In this section, only the information required to explain the SIGMA/W formulation is presented. The Cam-clay model uses effective stress parameters. In the following discussion, effective stresses are denoted by a superscript (′).

Figure 5-7 (a) schematically shows volume change versus pressure plots for a soil comprising a normal consolidation line and an over-consolidation line. The over-consolidation line is also known as the swelling line. Consider a stress state on the over-consolidation line. An increase in applied stress will cause the stress state to move along the over-consolidation line towards the normal consolidation line. Once past the intersection of the two lines, any further stress increase will cause the stress state to move down the normal consolidation line.

When Figure 5-7 (a) is rotated counterclockwise through 90°, the over-consolidation and normal consolidation lines show the characteristic of an elastic-hardening plastic stress-strain curve, illustrated in Figure 5-7 (b). The over-consolidation line is analogous to the initial linear elastic portion, while the normal consolidation line is analogous to the hardening plastic portion of the stress-strain relationship.

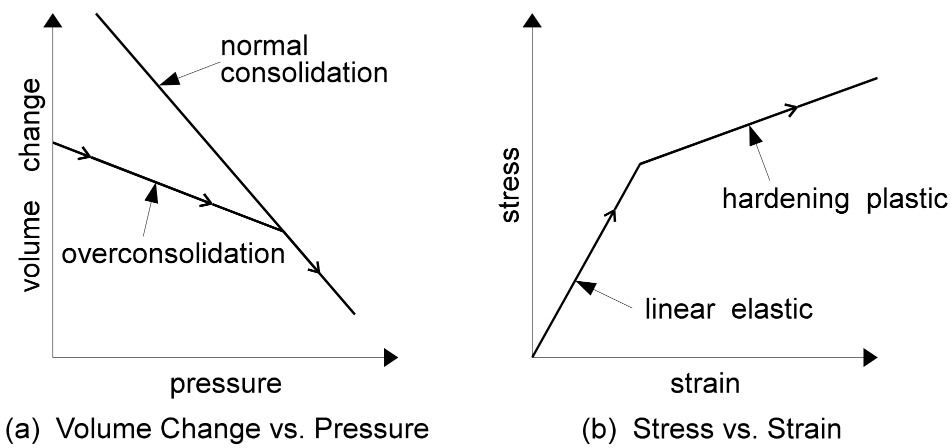


Figure 5-7 Analogy between volume-pressure and stress-strain relationships

Soil parameters

The Cam-clay model is an effective stress model which requires the following soil properties:

- M Slope of the critical state line in the $p'-q$ plane
- Γ Specific volume at the critical state when p' is 1.0 (or, ln (p') is 0)
- κ Slope of the isotropic over-consolidation (swelling) line
- λ Slope of the isotropic normal consolidation line
- v Specific volume

These five parameters are illustrated in Figure 5-8. The critical state line shown in the $p'-q$ plane is the locus of critical states projected onto that plane. The critical state line has a slope of, M , which is related to the angle of internal friction of the soil. For the case of triaxial compression, M can be expressed as:

$$\mathrm{M} = \frac{6\sin\phi'}{3-\sin\phi'}$$

By definition, the specific volume, v, is related to the void ratio, e, through the following expression:

$$v = 1 + e$$

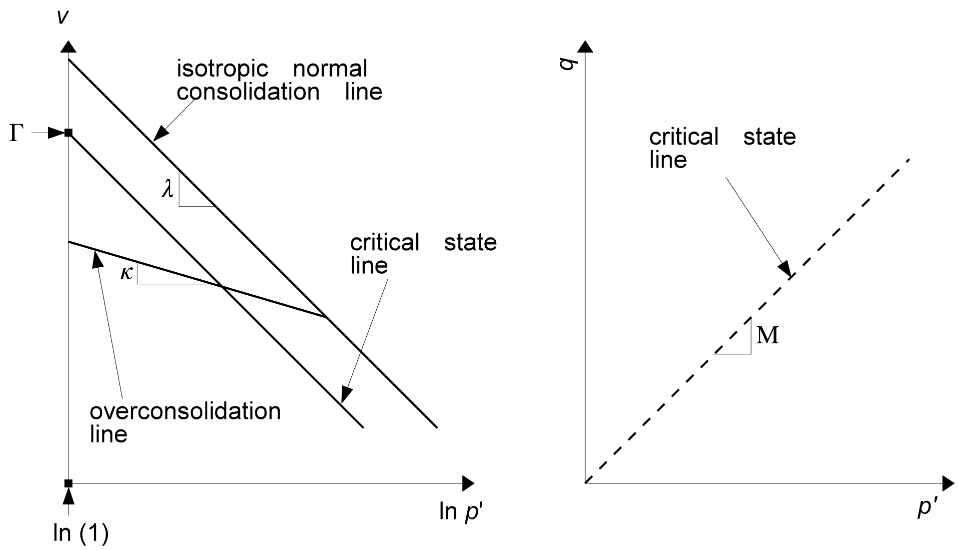


Figure 5-8 Definition of model parameters for Cam-clay

Yield function

The yield function for Cam-clay, as illustrated in Figure 5-9, can be expressed in terms of stress invariants p' and q as follows (Atkinson and Bransby, 1978):

$$F = \frac{q}{Mp'} + \ln\left(\frac{p'}{p'_x}\right) - 1$$

where:

p'_x = the peak mean stress and is the value of p' at the critical state line or t.

The peak mean stress, p'_x, is related to the pre-consolidation pressure, p'_c, by:

$$\ln p'_x = \ln p'_c - 1 \qquad \text{or}$$

$$p'_x = p'/2.71828$$

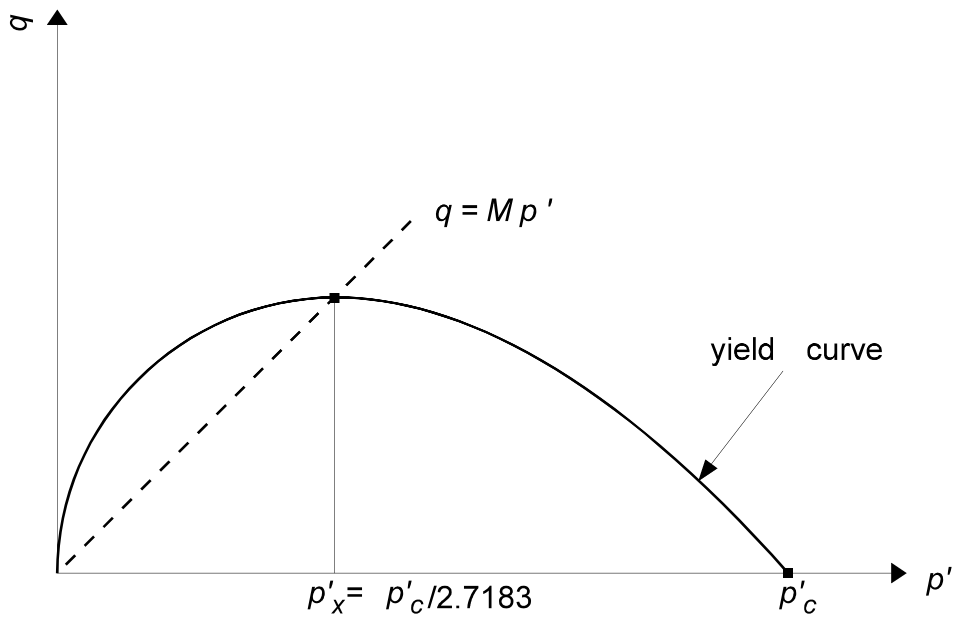


Figure 5-9 Yield curve for the Cam-clay model

The stress invariants p' and q are alternative forms of the first stress invariant, I_1', and the second deviatoric stress invariant, J_2, respectively.

$$p' = \frac{I_1'}{3}$$

and:

$$q = \sqrt{3J_2}$$

By definition, the first stress invariant, I_1', can be expressed in terms of effective stresses as:

$$I_1' = \frac{1}{3}\left(\sigma_x' + \sigma_y' + \sigma_z'\right)$$

and the second deviatoric stress invariant, J_2, can be written as:

$$J_2 = \frac{1}{6}\left[\left(\sigma_x - \sigma_y\right)^2 + \left(\sigma_y - \sigma_z\right)^2 + \left(\sigma_z - \sigma_x\right)^2\right] + \tau_{xy}^2$$

The second deviatoric stress invariant, J_2, is the same for either total or effective stresses.

Equations for the over-consolidation line and the critical state line can be used to calculate the peak mean stress, p'_x. As illustrated in Figure 5-9, the specific volume at critical state, v_x, for a particular over-consolidation line can be written as:

$$v_x = v + \kappa \ln p' - \kappa \ln p_x'$$

From the critical state line, the same specific volume, v_x, can also be calculated using:

$$v_x = \Gamma - \lambda \ln p'_x$$

Eliminating the specific volume, v_x, from these two equations gives the following expression for the peak mean stress, p'_x.

Equation 5-5 $$p'_x = \exp\left(\frac{\Gamma - v - \kappa \ln p'}{\lambda - \kappa}\right)$$

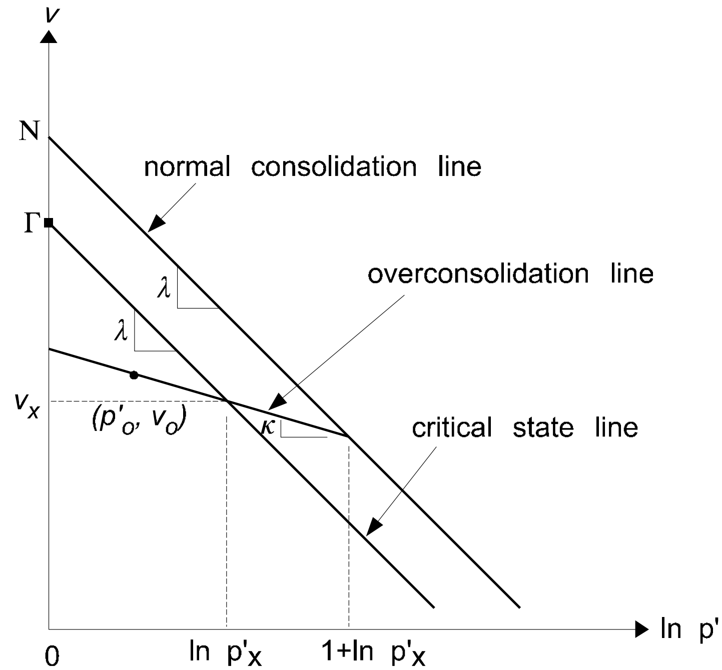


Figure 5-10 Definition of soil properties for Cam-clay model

Plastic matrix

The plastic matrix $[C_p]$ is developed using the yield function, F, in a manner similar to the development used for the Elastic-Plastic and Strain-Softening models. In the following discussion, subscripts e and p denote elastic and plastic state respectively.

As shown in Equation 5-5, the peak mean stress, p'_x, is a function of the specific volume, v. If it is postulated that a change in the peak mean stress, p'_x, can only be caused by a plastic change in the specific volume, an incremental change in the yield function, F, is given by:

Equation 5-6 $$dF = \left\langle \frac{\partial F}{\partial \sigma'} \right\rangle \{d\sigma'\} + \frac{\partial F}{\partial p'_x}\frac{\partial p'_x}{\partial v^p} dv^p = 0$$

From a given set of normal and over-consolidation curves, an incremental plastic change in specific volume can be expressed as:

$$dv^p = dv - dv^e = -(\lambda - \kappa)\frac{dp'}{p'}$$

The incremental plastic specific volume is related to the incremental plastic volumetric strain through:

$$dv^p = v_0 d\varepsilon_v^p$$

where:

dv^p	=	incremental plastic specific volume,
v_0	=	initial specific volume, and
$d\varepsilon_v$	=	incremental plastic volumetric strain.

Since only elastic strains can cause stresses to change, the incremental effective stress constitutive relationship can be written as follows:

$$\begin{aligned}\{d\sigma'\} &= [C'_e]\left(\{d\varepsilon\} - \{d\varepsilon_p\}\right) \\ &= [C'_e]\left(\{d\varepsilon\} - \lambda_p \left\{\frac{\partial G}{\partial \sigma'}\right\}\right)\end{aligned}$$

where:

$[C'_e]$	=	the effective elastic matrix,
G	=	the plastic potential function, and
λ_p	=	the (plastic) scaling factor.

Substituting $\{d\sigma'\}$, from the previous equation into Equation 5-6 gives:

Equation 5-7
$$\left\langle \frac{\partial F}{\partial \sigma'} \right\rangle [C'_e]\{d\varepsilon\} = \left\langle \frac{\partial F}{\partial \sigma'} \right\rangle [C'_e]\lambda_p \left\{\frac{\partial G}{\partial \sigma}\right\} - \frac{\partial F}{\partial p'_x}\frac{\partial p'_x}{\partial v^p} dv^p$$

The plastic increment in specific volume, dv^p, can be calculated from the original specific volume and the incremental plastic volumetric strain, $d\varepsilon_v^p$.

$$dv^p = v_o d\varepsilon_v^p = v_o \lambda_p \left(\frac{\partial G}{d\sigma'_x} + \frac{\partial G}{d\sigma'_y} + \frac{\partial G}{d\sigma'_z}\right)$$

Applying the associated flow rule (i.e. the plastic potential function, G, is the same as the yield function, F), the following expression is obtained for the plastic scaling factor, λ_p.

$$\lambda_p = \frac{\left\langle \frac{\partial F}{\partial \sigma'} \right\rangle [C'_e]}{\left\langle \frac{\partial F}{\partial \sigma'} \right\rangle [C'_e]\left\{\frac{\partial F}{\partial \sigma'}\right\} - \frac{\partial F}{\partial p'_x}\frac{\partial p'_x}{\partial v^p} v_0 F_m}\{d\varepsilon\}$$

where:

$$F_m = \frac{\partial G}{d\sigma'_x} + \frac{\partial G}{d\sigma'_y} + \frac{\partial G}{d\sigma'_z} = \frac{\partial F}{d\sigma'_x} + \frac{\partial F}{d\sigma'_y} + \frac{\partial F}{d\sigma'_z}$$

A relationship between incremental stress and incremental strain is obtained by substituting λ_p into Equation 5-7.

$$\{d\sigma'\} = \left(\left[C'_e\right] - \left[C'_p\right]\right)\{d\varepsilon\}$$

where:

$$\left[C'_p\right] = \frac{\left[C'_e\right]\left\{\frac{\partial F}{\partial \sigma'}\right\}\left\langle\frac{\partial F}{\partial \sigma'}\right\rangle\left[C'_e\right]}{\left\langle\frac{\partial F}{\partial \sigma'}\right\rangle\left[C'_e\right]\left\{\frac{\partial F}{\partial \sigma'}\right\} - \frac{\partial F}{\partial p'_x}\frac{\partial p'_x}{\partial v^p} v_0 F_m}$$

$[C'_p]$ is termed the plastic matrix.

Evaluation of the plastic matrix

To evaluate the plastic matrix, $\left[C'_p\right]$, for the Cam-clay model, it is necessary to compute the elastic matrix $\left[C'_e\right]$, the vector of derivatives of the yield function, $\left\{\frac{\partial F}{\partial \sigma'}\right\}$, the derivative terms $\frac{\partial F}{\partial p'_x}$, $\frac{\partial p'_x}{\partial v^p}$, v_0, and F_m, $\frac{\partial p'_x}{\partial v^p}$ and F_m, and the initial specific volume, v_0.

For given values of the effective stress, Young's modulus, E', and Poisson's ratio, v', the elastic matrix, $\left[C'_e\right]$, is the same as that for a linear elastic model. The Poisson's ratio, v', is a constant value input by the user, while the modulus, E', is calculated from the slope of the over-consolidation line, κ.

The slope of the over-consolidation line, κ, can be written as:

$$\kappa = -\frac{dv}{d\ln p'} = -p'\frac{dv}{dp'}$$

A change in volumetric strain is related to a change in specific volume through the following equation:

$$d\varepsilon_v = -\frac{dV}{V} = -\frac{de}{1+e} = -\frac{dv}{v}$$

where:

V	=	total volume
e	=	void ratio of the soil

Combining the previous two equations, the following expression for the incremental volumetric strain is obtained.

$$d\varepsilon_v = \left(\frac{\kappa}{vp'}\right)dp'$$

By definition, the effective bulk modulus, K', is given by:

$$K' = \frac{dp'}{d\varepsilon_v}$$

The bulk modulus, K', can also be expressed in terms of κ.

$$K' = \frac{vp'}{\kappa} = \frac{(1+e)p'}{\kappa}$$

In turn, the following expression is obtained for the effective stress modulus E'.

$$E' = 3K'(1-2\nu') = \frac{3p'}{\kappa}(1+e)(1-2\nu')$$

The vector of derivatives of yield function with respect to stresses is:

$$\left\{\frac{\partial F}{\partial \sigma'}\right\} = \left\langle \frac{\partial F}{\partial \sigma'_x} \quad \frac{\partial F}{\partial \sigma'_y} \quad \frac{\partial F}{\partial \sigma'_z} \quad \frac{\partial F}{\partial \tau'_{xy}} \right\rangle^T$$

These derivatives can be expressed as derivatives with respect to stress invariants I_1' and J_2 using the chain rule of differentiation.

$$\begin{aligned}\frac{\partial F}{\partial \sigma'_i} &= \frac{\partial F}{\partial p'}\frac{\partial p'}{\partial \sigma'_i} + \frac{\partial F}{\partial q}\frac{\partial q}{\partial \sigma'_i} \\ &= \frac{\partial F}{\partial I'_1}\frac{\partial I'_1}{\partial \sigma'_i} + \sqrt{3}\frac{\partial F}{\partial q}\frac{\partial J_2}{\partial \sigma'_i}\end{aligned}$$

where:

i = stress components x, y, z and xy

By definition, $q = \sqrt{3J_2}$. The derivatives of these stress invariants are given previously. The derivatives of the yield function F with respect to the stress invariants are expressed as follows:

$$\frac{\partial F}{\partial p'} = -\frac{q}{\mathrm{M}p'^2} + \frac{1}{p'}$$

$$\frac{\partial F}{\partial q} = \frac{1}{\mathrm{M}p'}$$

The remaining terms in the plastic matrix $\left[C_p\right]$, $\frac{\partial F}{\partial p'_x}$, $\frac{\partial p'_x}{\partial v^p}$, v_0, and F_{m}, and F_{m}, are described in the following equations:

$$\frac{\partial F}{\partial p'_x} = -\frac{1}{p'_x}$$

$$\frac{\partial p'_x}{\partial v^p} = \frac{p'_x}{\lambda - \kappa}$$

$$\begin{aligned}
F_m &= \frac{\partial F}{\partial \sigma'_x} + \frac{\partial F}{\partial \sigma'_y} + \frac{\partial F}{\partial \sigma'_z} \\
&= \frac{\partial F}{\partial p'}\left(\frac{\partial p'}{\partial \sigma'_x} + \frac{\partial p'}{\partial \sigma'_y} + \frac{\partial p'}{\partial \sigma'_z}\right) + \frac{\partial F}{\partial q}\left(\frac{\partial q}{\partial \sigma'_x} + \frac{\partial q}{\partial \sigma'_y} + \frac{\partial q}{\partial \sigma'_z}\right) \\
&= \frac{\partial F}{\partial p'}(1) + \frac{\partial F}{\partial q}(0) \\
&= -\frac{q}{\mathrm{M}p'^2} + \frac{1}{p'}
\end{aligned}$$

Initial condition

When using the Cam-Clay models it is useful to think of the yield surface in terms of the past, the present and the future. For a normally consolidated soil, the current state of stress is on the current yield surface. For an over-consolidated soil the current state of stress will be below the yield surface that was created sometime in the past. In either case future loads may cause the yield surface to expand; that is, there may be a new future yield surface.

Another important concept to remember is that the yield surface never shrinks in the Cam-Clay model – it can expand but not get smaller.

Key to using the Cam-Clay model is establishing the starting yield surface. This is done on the basis of the current or present insitu stresses together with a specified over consolidation ratio (OCR).

In equation form, the present in-situ stress state is

$$\{\sigma'_0\} = \begin{Bmatrix} \sigma'_{x0} \\ \sigma'_{y0} \\ \sigma'_{z0} \\ \tau_{xy0} \end{Bmatrix}$$

The maximum vertical stress that the soil has experienced in the past is usually determined from an odometer test. The ratio of the maximum vertical stress in the past, σ'_{vmax}, to that at present, σ'_v, is known as the over-consolidation ratio, OCR. In SIGMA/W, OCR is a user-specified parameter for each soil type. The relationship between the maximum vertical stress, σ'_{vm}, and maximum horizontal stress, σ'_{hm}, is approximated as follows:

$$K_0 = 1 - \sin\phi'$$
$$\sigma'_{hmax} = K_0 \sigma'_{vmax}$$

Assuming that the shear stress is zero, the maximum stress vector, $\{\sigma'_m\}$, can be written as:

$$\{\sigma'_m\} = \begin{Bmatrix} \sigma'_{xmax} \\ \sigma'_{ymax} \\ \sigma'_{zmax} \\ 0 \end{Bmatrix} = \begin{Bmatrix} \overline{\sigma'_{y0} K_0 OCR} \\ \sigma'_{y0} OCR \\ \overline{\sigma'_{y0} K_0 OCR} \\ 0 \end{Bmatrix}$$

The historic past maximum mean stress, p'$_m$, and maximum shear stress, q$_m$, are respectively:

$$p'_m = \frac{1}{3}\left(\sigma'_{xm} + \sigma'_{ym} + \sigma'_{zm}\right)$$

$$q'_{max} = \frac{1}{\sqrt{2}}\sqrt{\left(\sigma'_{xmax} - \sigma'_{ymax}\right)^2 + \left(\sigma'_{ymax} - \sigma'_{zmax}\right)^2 + \left(\sigma'_{zmax} - \sigma'_{xmax}\right)^2}$$

The pre-consolidation pressure, p'$_c$, occurs in the p'-q space where the shear stress, q, is zero. Therefore, from the yield function for Cam clay, the following expression can be obtained:

$$\ln\left(\frac{p'_c}{p'_x}\right) = 1 \quad \text{, or, } \ln p'_x = \ln p'_c - 1$$

Substituting the natural logarithm of the pre-consolidation pressure, ln p'$_x$, the yield function for Cam-clay can be rewritten as:

$$\frac{q}{\mathrm{M}p'} + \ln p' - \ln p'_c = 0$$

Therefore, the pre-consolidation pressure, p'$_c$, is given by:

$$p'_c = p'_{max} \exp\left(\frac{q_{max}}{M\, p'_{max}}\right)$$

The pre-consolidation pressure p'$_c$ together with a specified void ratio *e* is used to compute the specific volume parameters N and Γ using the following expressions:

$$v = 1 + e$$

From the normal consolidation line passing through the pre-consolidation pressure, p'$_c$, the specific volume at the critical state is given by:

$$v_c = N - \lambda \ln p'_c$$

$$\mathrm{N} = \Gamma + \left(\lambda - \kappa\right)$$

Also, from the over-consolidation line passing through the pre-consolidation pressure, p'$_c$, the initial specific volume, v$_0$, can be expressed as:

$$v_0 = v_c + \kappa \ln\left(\frac{p'_c}{p'_0}\right)$$

In this expression, the initial mean stress, p'_0 , can be calculated from the initial stresses using the equation:

$$p'_0 = \frac{1}{3}\left(\sigma'_{x0} + \sigma'_{y0} + \sigma'_{z0}\right)$$

This completes the computation for parameters defining a Cam-clay model in SIGMA/W.

Soil parameters

The Cam-clay models use the parameters; M, Γ, λ, and κ. These material properties can be obtained from some more common parameters such as ϕ' and the compression index C_c. This section presents some useful relationships when you are working with the Cam-clay models.

The slope of the critical state line, *M*, obtained from a triaxial compression test is related to the friction angle ϕ' through:

$$M = \frac{6\sin\phi'}{3 - \sin\phi'}$$

The compression versus vertical pressure characteristics of soil are usually obtained from a one-dimensional consolidation test, and it is common practice to obtain the compression index C_c from a plot of void ratio (*e*) versus $\log_{10}(p)$. The compression indices are related to the slope λ and κ by the relationship:

$$\lambda = \frac{C_c}{2.303}$$
$$\kappa = \frac{C_s}{2.303}$$

Alternatively, λ and κ can be obtained from a plot of void ratio versus $\ln(p)$ instead of void ratio versus $\log_{10}(p)$.

The specific volume v is equal to 1 plus the void ratio (*e*).

Table 5-4 summarizes the material properties required in the Cam-clay model.

Table 5-4 Cam clay / modified cam clay properties

Edit Box Label	Property
O. C. Ratio	Over-consolidation ratio
Poisson's Ratio	Constant value
Lambda	Slope of normal consolidation line
Kappa	Slope of over-consolidation (swelling) line)
Init. Void Ratio	Void ratio
Mu	Slope of critical state line

5.4 Modified Cam-clay model

The modified Cam-clay model is similar to the Cam-clay model except that the yield function is in the shape of an ellipse instead of a tear drop. The yield curve for the modified Cam-clay model is illustrated in Figure 5-11.

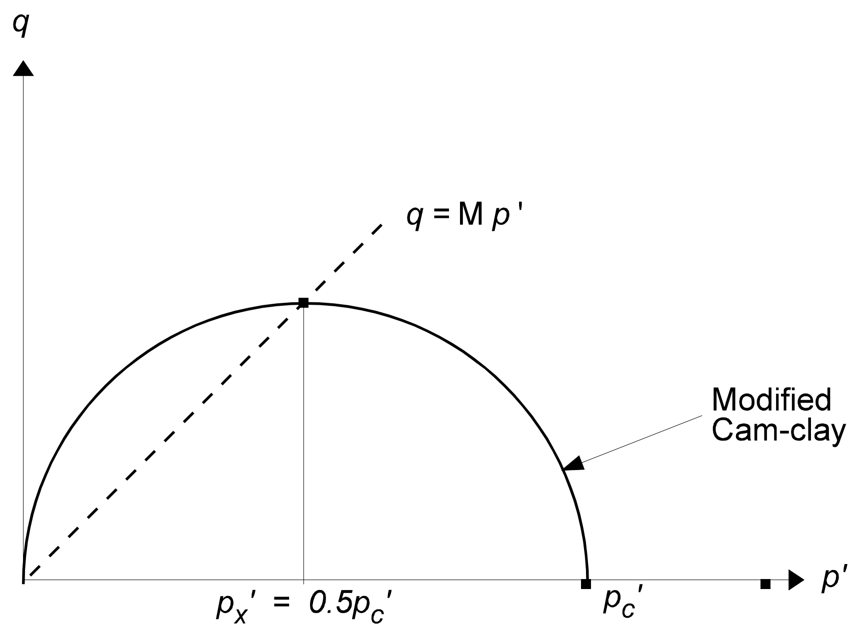


Figure 5-11 Yield function for the modified Cam-clay

The yield function for the Modified Cam-clay model is given by the following equation (Britto and Gunn, 1987):

$$q^2 = \mathrm{M}^2 p' p'_c - \mathrm{M}^2 p'^2$$

where:

p'_c = pre-consolidation pressure.

The parameters used to define the Modified Cam-clay model are: M, Γ, κ, and v. These parameters are the same as those used for the Cam-clay model as discussed in the previous section.

Similar to the Cam-clay model, SIGMA/W uses the peak mean stress, p'_x, to determine the size of the yield locus for the Modified Cam-clay model. The peak mean stress, p'_x, is the isotropic pressure when a soil reaches its critical state. At the critical state, the shear stress, q, is given by the following equation:

$$q = \mathrm{M}p'_x$$

Substituting this value of shear stress, q, into the modified yield function equation gives:

$$p'_c = 2p'_x$$

Therefore, the yield function, F, for the Modified Cam-clay model can be written as:

$$F = \frac{q^2}{p'} + \mathrm{M}^2 p' - 2\mathrm{M}^2 p'_x$$

The derivatives of the yield function, F, with respect to stress invariants p' and q are respectively:

$$\frac{\partial F}{\partial p} = \mathrm{M}^2 - \frac{q^2}{p'^2}$$

$$\frac{\partial F}{\partial q} = \frac{2q}{p'}$$

$$\frac{\partial F}{\partial q} = \frac{2q}{p'}$$

The terms $\left\{\frac{\partial p'}{\partial \sigma}\right\}$, $\left\{\frac{\partial q}{\partial \sigma}\right\}$, $\left[C_e\right]$, and $\frac{\partial p'_x}{\partial v^p}$ are the same as those used for the Cam-clay model. The remaining variables in the plastic matrix, $\left[C'_p\right]$, for modified Cam-clay are:

$$F_m = M^2 - \frac{q^2}{p'^2}$$

$$\frac{\partial F}{\partial p'_x} = -2M^2$$

$$F_m = M^2 - \frac{q^2}{p'^2}$$

Except for the differences noted in the following discussion, the procedure to define the initial condition of the Modified Cam-clay model is the same as that for the Cam-clay model described in the previous section. If a pre-consolidation pressure is not defined by the user, it is calculated using:

$$p'_c = \frac{1}{M^2 p'_{\max}}\left(q^2_{\max} + M^2 p^2_{\max}\right)$$

The interception of the isotropic normal consolidation line with the vertical axis (i.e. $\ln p' = 0$), N is given by:

$$N = \Gamma + (\lambda - \kappa)\ln 2$$

Initial Conditions

The procedure for establishing the initial conditions for the MCC are the same as for the original Cam-Clay discussed earlier.

5.5 Commentary on Cam-Clay Models

Both the Cam-clay and Modified Cam-Clay models are discussed here but only the Modified Cam-Clay (MCC) is made available in the latest Version of SIGMA/W. The original Cam-Clay model was available in earlier versions primary for historic reasons but since it is not used in practice it has been eliminated from the latest SIGMA/W version.

5.6 Slip surfaces

SIGMA/W can simulate a slip surface using quadrilateral elements with and Elastic-Plastic soil model. Generally, the friction angle ϕ is less than the value for the soil butting up against a wall for example. Usually, this means it is necessary to create a 'slip' material which represents the frictional characteristics between the soil and a structure.

5.7 User Add-in constitutive models

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.

5.8 E-modulus functions

The soil stiffness modulus *E* can be specified as a constant or as a function of the effective overburden stress.

Technical publications on the subject of soil stiffness reference *E* to the confining stress σ_3, the vertical or overburden stress σ_v or the mean stress $(\sigma_1 + \sigma_2 + \sigma_3)/3$. The difference between these is not all that great, especially in light of the accuracy with which *E* can be defined for field problems. The overburden stress can be readily computed mentally or by hand and therefore is a convenient reference point for spot checking results. It is for this arbitrary reason and for the sake of consistency, that SIGMA/W has been standardized on the overburden stress.

As was mentioned above in the Section on the hyperbolic model, the soil stiffness *E* is often proportional to the square root of the confining stress. Generally, the relationship is expressed as:

$$E = K\sigma^n$$

Where *K* is a soil property and *n* is an exponent.

As part of the development of the hyperbolic model at the University of California, Berkley in the 1970's, an extensive laboratory testing program was carried out to measure *K* and *n* for a wide range of soils. The results were presented in a University of California internal document by Duncan et al. (1980); (unfortunately this document is out of print). From all the tests, they prepared the table shown in Figure 5-12. They considered these to be conservative values suitable as estimates for preliminary or exploratory analyses.

These values were used as a guide to develop a series of sample functions for use in a SIGMA/W analysis. The next section briefly describes how to use these estimated functions.

Unified Classification	RC Stand. AASHTO	γ_m k/ft^3	ϕ_o deg	$\Delta\phi$ deg	C k/ft^2	k	n	R_f	K_b	m
GW, GP SW, SP	105	0.150	42	9	0	600	0.4	0.7	175	0.2
	100	0.145	39	7	0	450	0.4	0.7	125	0.2
	95	0.140	36	5	0	300	0.4	0.7	75	0.2
	90	0.135	33	3	0	200	0.4	0.7	50	0.2
SM	100	0.135	36	8	0	600	0.25	0.7	450	0.0
	95	0.130	34	6	0	450	0.25	0.7	350	0.0
	90	0.125	32	4	0	300	0.25	0.7	250	0.0
	85	0.120	30	2	0	150	0.25	0.7	150	0.0
SM-SC	100	0.135	33	0	0.5	400	0.6	0.7	200	0.5
	95	0.130	33	0	0.4	200	0.6	0.7	100	0.5
	90	0.125	33	0	0.3	150	0.6	0.7	75	0.5
	85	0.120	33	0	0.2	100	0.6	0.7	50	0.5
CL	100	0.135	30	0	0.4	150	0.45	0.7	140	0.2
	95	0.130	30	0	0.3	120	0.45	0.7	110	0.2
	90	0.125	30	0	0.2	90	0.45	0.7	80	0.2
	85	0.120	30	0	0.1	60	0.45	0.7	50	0.2

Figure 5-12 Soil stiffness properties presented by Duncan et al. (1980)

Sample functions

To estimate an *E*-modulus function, you need to specify a maximum depth, a *K*-modulus number, and exponent *n* and a K_o value. From these values, SIGMA/W computes data points for a function like the one in Figure 5-13.

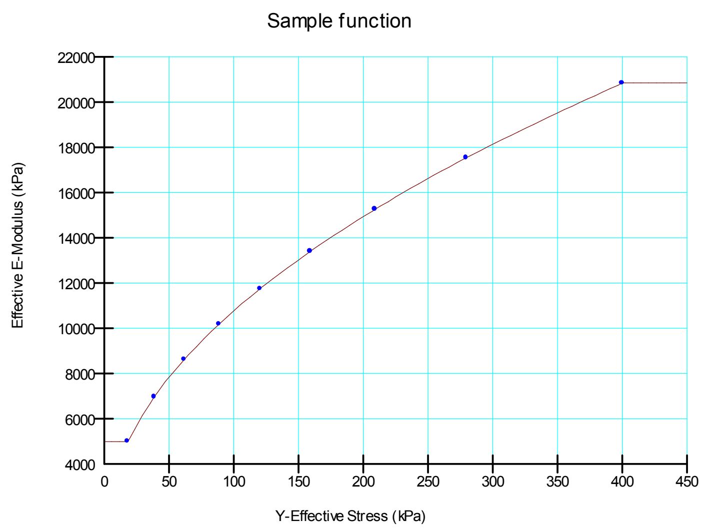


Figure 5-13 Sample *E*-modulus function

The specified depth value is used only to set up a stress range for the function. It is not related to any particular point in the problem geometry. The important issue is to establish the correct match between *E* and the overburden (vertical) stress.

Sample values for *K* and *n* are available from a list for:

- Sand/gravel
- Silty Sand
- Clayey Sand
- Low Plasticity Silty-Sandy Clay
- High Plasticity Clay

Of course any user defined values can also be used.

In addition, specific data points may be entered to create any unique function considered suitable for a particular site.

The E-modulus functions can be defined in terms of total or effective stresses.

The entire function can be shifted up or down by altering the y-intercept value.

Note that in Figure 5-13 the function has a short horizontal line at both ends. This is a graphic reminder that the *E*-modulus has a minimum and a maximum value.

The sample *K* and *n* values are intended to be used only for preliminary or exploratory analyses. These values must be confirmed and verified for site-specific use by the responsible engineering personal.

5.9 Undrained strength functions

When an analysis is done based on total stresses, it is often useful to vary the undrained strength C_u of the soil with depth. The strength variation can be defined as a function such as, for example, in Figure 5-14. The strength is defined as a function of the total vertical (overburden) stress. The function can be any data-point function of any form considered suitable for a particular site. The slight increase in strength at a low overburden stress (near the ground surface) in Figure 5-14 could perhaps represent a desiccated crust.

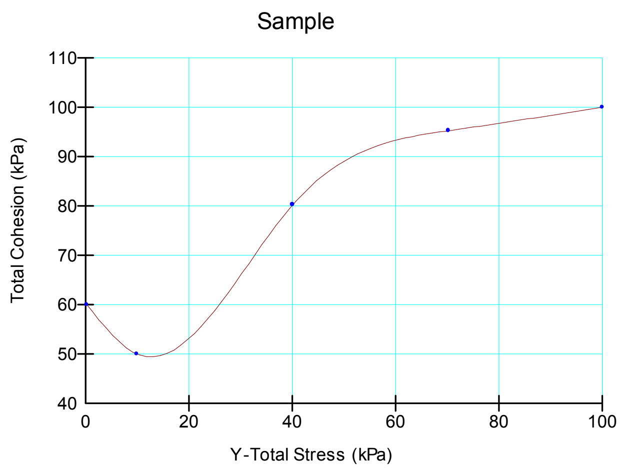


Figure 5-14 Sample undrained strength function

6 Boundary Conditions

This chapter discusses the various types of boundary conditions that can be applied in SIGMA/W analyses. There are many ways to apply forces and displacements to soil and structural elements in order to replicate real world loading conditions. Before any loading is applied, it is important to know what the model will do with the data you input. A discussion of the incremental finite element formulation has been on-going throughout the book (see Chapter 3 and the Theory chapter). It is critical to understand that a load should only be applied on one step in the model and then removed. Sometimes you must remove a load manually and sometimes SIGMA/W will turn them off automatically (such as during fill placement when body loads are only counted on the step the fill is activated).

This chapter shows what boundary condition options are available and how the model uses the data. The following chapter on Analysis Types discusses the differences between in-situ analysis and load-deformation analysis and when various boundary conditions may be used. Therefore, the present discussion makes limited reference to when different types of boundary conditions are applied.

6.1 Multiple boundary condition types

Multiple boundary condition types are implemented to support virtually all load-deformation modelling scenarios. In SIGMA/W, displacement, force or spring boundary conditions may be applied to points or nodes along lines, and stress and fluid pressure boundary conditions may be applied to element edges on lines.

6.2 Force or displacement

Fundamentally, there are only two types of boundary conditions that can be applied in a stress-deformation model: force or displacement. How these conditions are interpreted and applied is a different matter, because there are many options depending on what your objectives are.

In all stress-deformation models it is critical to “bound” the problem. This means that you must define some parts of the geometry as zero displacement boundary conditions. If there is no bounding to the problem, how can there be any reactionary force to the applied loading?

In general, it is common to bound the left and right sides of a problem along with the bottom edge. A typical example of this is given in Figure 6-1, where the bottom and side boundary edges are specified as zero x- and zero y-displacement. It is quite easy to see why there should be zero movement at the base of the model in both directions, but there can be some question about when to also set the vertical movement along the sides to zero. The answer to this latter question depends on what you are trying to model. If you are modeling a larger domain where you are saying the soil at the far sides is solidly attached to other soil even farther away, then both x- and y-displacement should be zero. If you are modeling a triaxial cell that is symmetric about the vertical axis, the axis of symmetry should be fixed in the horizontal direction, and the bottom should be free to move in the x-direction, but not y-direction as shown in Figure 6-2.

Figure 6-1 Bounding the model with zero displacement along edges

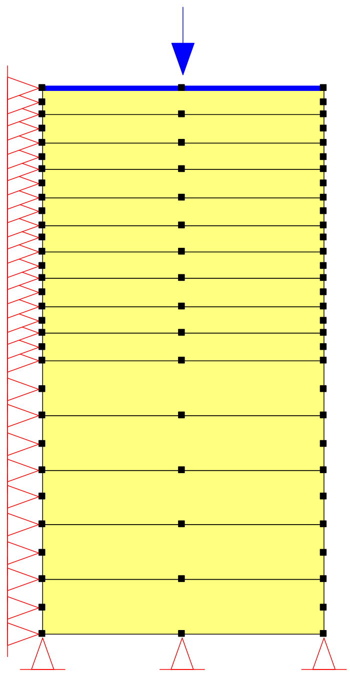


Figure 6-2 Symmetrical axis displacement boundary conditions

6.3 Body loads

Body forces are included in an analysis by specifying a non-zero unit weight for the soil. For each element, SIGMA/W computes the volume of the element, multiplies the volume by the specified unit load of the material, and applies the total element body load as forces at the nodes of the elements.

The unit weight of the soil is used to simulate gravity body loads. In SIGMA/W a graphical feedback lets you know if a body load is applied to an element. Body loads are displayed as a hatched pattern within each element as show below in Figure 6-3.

Body loads come into play when establishing the insitu stresses or when simulating the placement of fill in layers for example. They are only applied to the solution on the first load step that a soil material becomes active. They are specified as a property of each material.

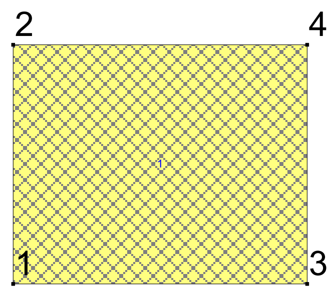


Figure 6-3 Hatch pattern to show body load applied to element

6.4 Boundary condition locations

In GeoStudio 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 6-5 and Figure 6-6 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.

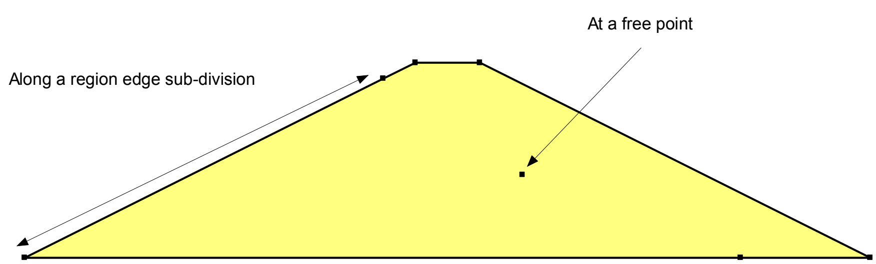


Figure 6-4 Desired BC locations

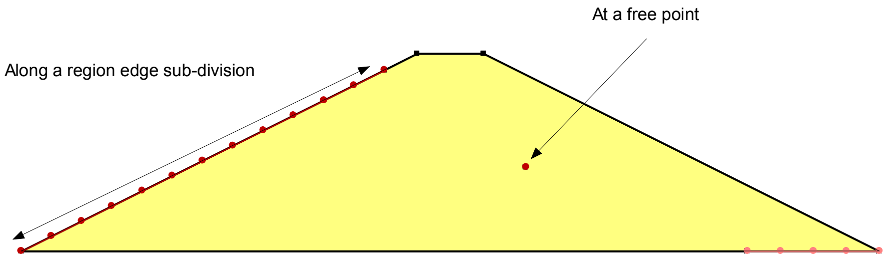


Figure 6-5 BC's attached to geometry

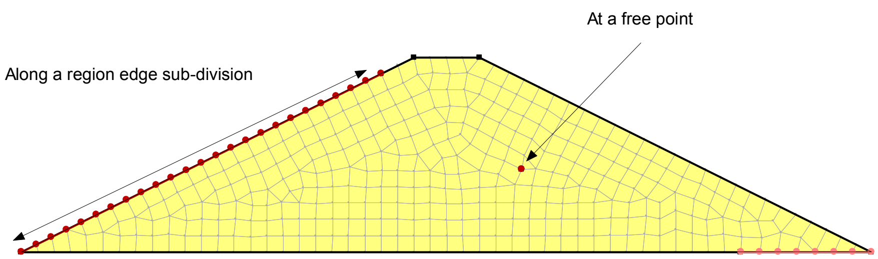


Figure 6-6 BC's with underlying mesh visible

6.5 Nodal boundary conditions

In SIGMA/W, boundary conditions are defined on specified geometry items such as region faces, region lines, free lines or free points. However, the boundary conditions are fundamentally distributed along either the nodes or element edges that exist underneath the geometry item. If you need to define a boundary condition to a specific location, it may be necessary to add additional region points to control the length of a region edge for an edge boundary condition or ensure that a node exists at a specific location for a nodal boundary condition. Some types of boundary conditions are appropriate only when applied to element edges or nodes while others can be assigned to either an edge or node to get the same desired effect. For example, a fluid pressure boundary condition must be drawn on an element edge.

The Draw Boundary Conditions command allows you to specify displacement, force, or spring boundary conditions which are applicable for nodal boundary conditions. Additionally, zero rotation boundary conditions can be specified on nodes that also belong to structural beam elements. All of these descriptions are unique to different types of real life conditions, but numerically, they all reduce to either forces or displacements.

The on-line help provides details of how to apply the boundary condition to the model. In general, however, node boundary conditions are defined in two stages. First, create the boundary conditions, such as the boundary type, action, color and boundary function number. Secondly, apply the boundary conditions to the specific geometry items that are to have these boundary conditions.

Table 6-1 lists the various nodal boundary conditions available in SIGMA/W and the shape of the symbol that will appear once they are specified. Boundary conditions can be added or removed from objects using the Draw Boundary Conditions command.

Table 6-1 Symbols used for each type of boundary condition

Type	Value	Symbol X-Direction	Symbol Y-Direction	Description
(none)	--------	--------	--------	none
Displacement	positive (+)	right	up	hollow arrow
Displacement	negative (-)	left	down	hollow arrow
Displacement	zero (0)			hollow triangle
Force	positive (+)	right	up	solid arrow
Force	negative (-)	left	down	solid arrow
Force	zero (0)	--------	--------	none
Spring	not applicable	left or right	up or down	springs
Spring	zero (0)	--------	--------	none
Rotation	zero (0)		--------	hollow circle

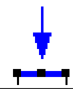
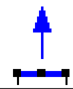
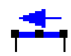
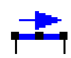
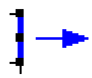
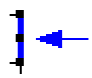
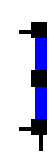
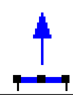
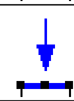
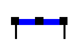
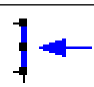
Boundary nodes have a negative *x*- or *y*-value point in the negative *x*- or *y*-direction, while boundary nodes that have an *x*- or *y*-value that is positive or equal to zero point in the positive *x*- or *y*-direction.

6.6 Edge boundary conditions (pressures or stresses)

In SIGMA/W, boundary conditions are defined on geometry items but are applied to the underlying nodes or element edges. If you need to define a boundary condition to a specific location, it may be necessary to add additional region points to control the length of a region edge for an edge boundary condition or ensure that a node exists at a specific location for a nodal boundary condition. Edge stress boundary conditions are defined in two stages. First, the boundary conditions are created, such as the stress boundary type, action, and boundary function number. Secondly, all geometry items that are to have these boundary conditions are selected. Boundary conditions can be removed through the Draw Boundary Condition command.

Table 6-2 shows the various types of edge boundary conditions that can be specified along with their associated symbol and data conventions. An easy way to distinguish between positive (+) and negative(-) normal stress is to remember that compressive stress is positive. For tangential stresses, going around the element in a counter-clockwise direction is positive. You can also check the direction of the arrow displayed at a normal/tangential pressure boundary.

Table 6-2 Symbols used for edge boundary conditions

Type	Value	Symbol	Description
(none)	--------	none	--------
Normal Stress	positive (+)		line along edge, arrow perpendicular towards edge
Normal Stress	negative (-)		line along edge, arrow perpendicular away from edge
Tangential Stress	positive (+)		line along edge along edge, arrow in counter-clockwise direction around an element
Tangential Stress	negative (-)		line along edge along edge, arrow in clockwise direction around an element
X-Stress	positive (+)		line along edge, arrow pointing right
X-Stress	negative (-)		line along edge, arrow pointing left
X-Stress	zero (0)		line along edge
Y-Stress	positive (+)		line along edge, arrow pointing up
Y-Stress	negative (-)		line along edge, arrow pointing down
Y-Stress	zero (0)		line along edge
Fluid Elevation	greater than min. edge *y*-coordinate		line along edge, arrow perpendicular towards edge
Fluid Elevation	less than min. edge *y*-coordinate		line along edge

Converting edge conditions to nodal equivalent forces

Stresses along the edge of an element are a form of nodal boundary forces. The specified stress multiplied by the length of the element edge gives a total force. The force is then proportionately divided among the nodes along the element edge.

As illustrated in Figure 6-7, the force computed from the specified stress is equally divided between the two corner nodes for a 4-noded element. For a higher-order 8-noded element, 1/6 of the force is assigned to each of the corner nodes and 4/6 is assigned to the middle intermediate node.

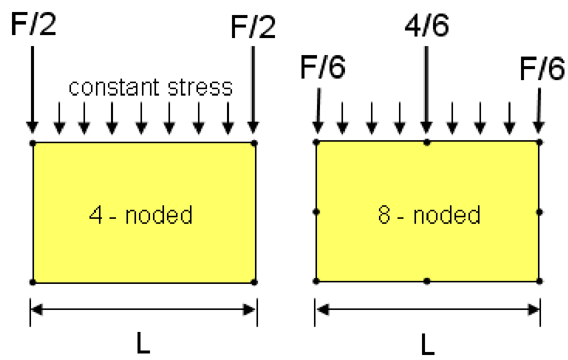


Figure 6-7 Pressure equivalent nodal forces

The nodal forces are actually computed numerically by integrating along the edge of each element (see the Theory chapter). This generalizes the scheme and makes it possible to consider a variable pressure distribution along the edge of the element.

It is useful to remember that specified boundary stresses cannot be used directly in a finite element analysis - they must always be converted to nodal forces. The solver will do this conversion for you.

Fluid pressure conditions

A fluid pressure boundary is a special type of normal pressure boundary. The pressure is defined by specifying a water surface elevation as illustrated in Figure 6-8. The pressure is computed from the distance between the specified water surface elevation and the y-coordinate of the boundary node. All the element edges between 2-3 and 3-4 are flagged as fluid pressure boundaries.

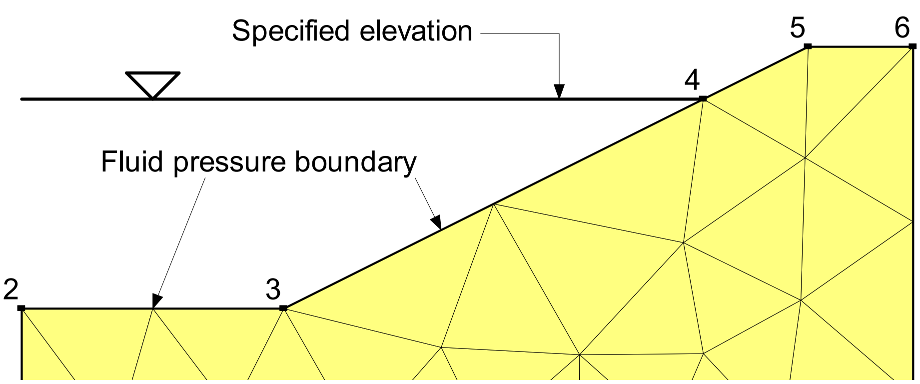


Figure 6-8 Illustration of fluid pressure boundary

When you apply the fluid pressure type to an element edge, SIGMA/W internally calculates equivalent force applied at each of the nodes defining the edge. The nodal pressure is obtained by subtracting the nodal y-coordinate from the fluid elevation and multiplying it by the fluid self-weight. SIGMA/W will only consider positive fluid pressures. Therefore, for the fluid pressure boundary condition to have an effect, the selected edge must have a y-coordinate less than the fluid elevation.

NOTE: Drawing a water table above the mesh does not generate a pressure boundary condition on the mesh. It allows pore-water pressures in the soil to be known, but it does not apply a total force (load) condition due to the weight of the water on the ground surface. You must specify both water table and fluid pressure if you need soil water pressures and surface water forces.

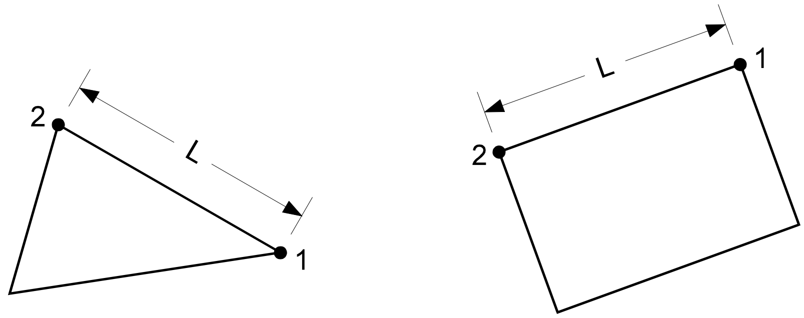
Defining a stress boundary at internal element edges

If you wish to define a stress boundary at an internal element edge, you can use the equivalent nodal force approach as described below. If you do not do this, you run the risk of having the stress value applied twice. As the program loops over all elements it will find an edge with a stress and convert this to a force. If it loops over a different element and finds the same edge boundary, it will apply it as a force again. By doing the force conversion yourself, you can ensure you apply the desired force.

Along an internal element edge, the equivalent nodal forces can be calculated by multiplying the applied pressure with the corresponding contributing areas and then applying those nodal forces as nodal boundary conditions. The contributing areas are dependent on the analysis type (either two-dimensional or axisymmetric) and on whether there is a mid-side node. Figure 6-9 and Figure 6-10 show the contributing areas computed for corner nodes, a_1 and a_2; and the mid-side node, a_4. For an axisymmetric analysis, it is important to note that the contributing areas are specified per unit radian when the element thickness is 1.0 and in units of 2π radians when the element thickness is 2π .

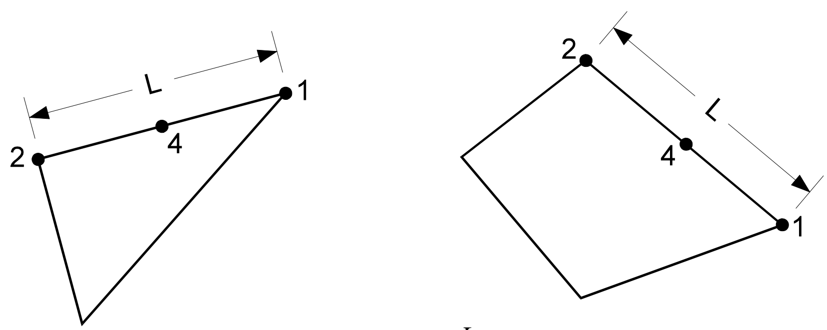
Region face boundary conditions

In GeoStudio a boundary condition can be applied to the "face" of a region. If you create a stress boundary condition and apply it to an edge, the nodal force is computed based on the stress value and the edge length adjacent to the nodes. If you apply the same stress boundary condition to the region face, then the stress is multiplied by the face area that surrounds a node and converted into a force at the nodes. If you assume the model has a unit thickness of 1 (which you can change) then the face boundary condition is a way to apply a body load that considers the volume of each element. You can use a body load face BC to simulate self weight of soil, or change in total stress due to water infiltration for example.



$$a_1 = a_2 = \frac{L}{2}$$

(a) No intermediate node

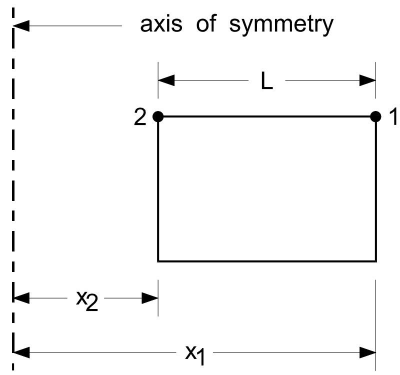


$$a_1 = a_2 = \frac{L}{6}$$

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

(b) With an intermediate node

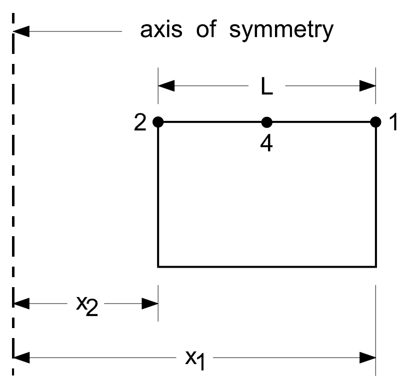
Figure 6-9 Contributing area for two dimensional elements of unit thickness



$a_1 = (2x_1 + x_2)L / 6$

$a_2 = (x_1 + 2x_2)L / 6$

(a) No intermediate node



$a_1 = Lx_1 / 6$

$a_2 = Lx_2 / 6$

$a_4 = L(x_1 + x_2) / 3$

(b) With an intermediate node

Figure 6-10 Contributing area for axisymmetric elements over 1 radian

6.7 Transient boundary conditions

A boundary condition function may be associated with each boundary condition instead of specifying a constant value. This feature is useful for specifying boundary conditions that vary with time or load step. In addition, transient boundary conditions may be cycled, allowing specification of transient boundary conditions that repeat themselves with some frequency.

SIGMA/W uses the incremental change in a function as the boundary condition for each load step. In the case of a force function, the incremental applied force is:

$$\Delta F = F_i - F_{i-1}$$

where i is the time step number.

SIGMA/W obtains F_i and F_{i-1} from the boundary function. The difference is applied as the boundary condition for the i-th time step.

As a result, it is necessary to start defining the function at Time 0. The time increment for the first step then is:

$$\Delta F_1 = F_1 - F_0$$

When you create a boundary function such as the one illustrated in Figure 6-11, you should think of the Y-value as a cumulative value. If you want 10 lbs force applied at load step 2, but at no other steps, then the function should look like the one below. For all steps past step 2, there is no change in load so no load is applied. If you had the function drop back to zero for all additional steps, then on the second step, the program would use a change in load of 0 – 100 = -100 lbs and you would undo what you tried to accomplish in the first step.

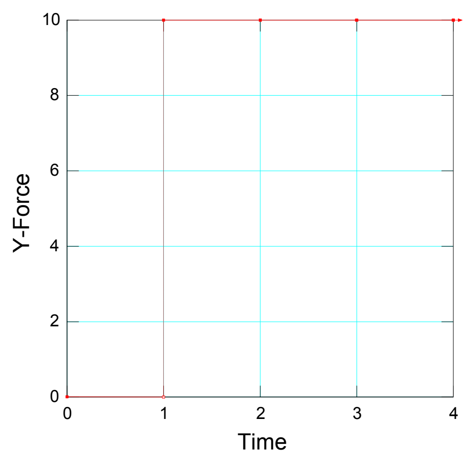


Figure 6-11 Step load of 10 on step 2 and zero on all other steps

7 Analysis Types

SIGMA/W is capable of solving different types of analyses and in two different views. The types of analyses that are defined within a project will depend on the objective(s) of the modelling study. The more common scenario is to define multiple analyses to investigate a single problem. For example, it may be necessary to: a) establish the initial stresses using an in-situ analysis; b) complete a load-deformation or coupled analysis to simulate fill placement; and c) complete another load-deformation or coupled analysis to simulate an excavation. The coupled (or uncoupled) consolidation analysis would be used if it was necessary to simulate the generation and dissipation of excess pore-water pressures in addition to deformations. The coupled stress-strain and seepage type of analysis is discussed in detail in a dedicated chapter.

7.1 Problem view

SIGMA/W can accommodate two-dimensional and axisymmetric analyses. A two-dimensional view analyzes a vertical cross-section with a unit thickness. The thickness can be specified when the region geometry is defined.

An axisymmetric view analyzes a problem that is symmetrical around a vertical axis (e.g., a right cylindrical tank or a laboratory tri-axial test). In SIGMA/W, the axis of symmetry is fixed at the vertical axis (coordinate r = 0). By default, the axisymmetric analysis is formulated for a sector of one radian.

7.2 Initial in-situ stresses

SIGMA/W has a special type of analysis called 'Insitu' that is formulated specifically for establishing the initial stresses. Most classes of problems will require initial stresses before proceeding with a load-deformation or coupled stress-strain and seepage type of analysis (e.g. simulation of a staged construction problem). The initial stresses are only the result of gravity and represent the equilibrium state of the undisturbed soil (or rock).

Initial stresses must be established if the following non-linear constitutive models are used:

1. elastic-plastic Mohr-Coulomb model (and slip surface model) – the initial stresses must be established because the model incorporates a failure criterion;
2. modified Cam clay – the initial stresses must be established because the model incorporates a failure criterion and the stiffness and overconsolidation properties are stress state dependent; and,
3. hyperbolic non-linear elastic – the initial stresses must be established when the total stiffness and strength parameters are specified as a function of the initial stresses.

The initial stresses are established by applying the self-weight of soil by means of a body load (described in the previous chapter). The specified unit weight multiplied by the element volume creates a downward nodal force. The value of Poisson's ratio (for each material) will control the development of horizontal stresses. Note that the stiffness properties (e.g. Young's Modulus) of the materials have no effect on the initial stresses.

The ratio of horizontal to vertical effective stresses in saturated soils having a history of one-dimensional deformation (e.g. sedimentation and possibly overconsolidation) is called the coefficient of earth pressure at rest:

$$K_0 = \frac{\sigma'_h}{\sigma'_v} = \frac{\nu'}{1-\nu'}$$

SIGMA/W limits the value of Poisson's ratio to 0.495; therefore, K_o is limited to approximately 0.98. Overconsolidated soils generally have K_o values that exceed this value; that is, the effective horizontal stresses exceed the vertical effective stresses. Initial stress states with $K_o > 0.98$ cannot be established using the Insitu type of analysis. The K_o procedure described subsequently must be used for this type of problem.

Insitu analysis should be completed with the left and right boundaries free to move in the vertical direction; that is, a boundary condition of x-displacement = 0 should be specified. The bottom boundary should be rigidly fixed; that is, both the x-displacement and y-displacement are set to 0 (Figure 7-1).

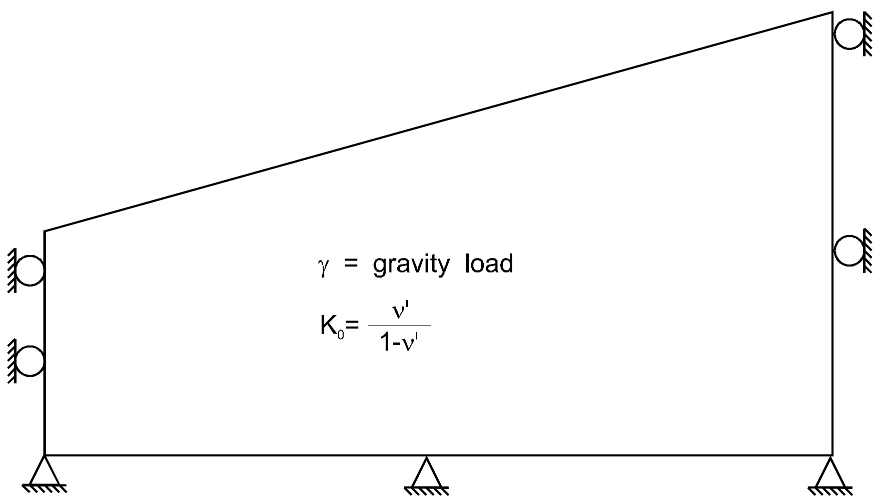


Figure 7-1 Conceptual illustration of the boundary conditions required for an in-situ analysis

K_o procedure

The K_o procedure can be used to establish the initial stresses for normally compressed or overconsolidated soils. The K_o procedure overrides the initial stresses established by the Insitu method for a particular region only when a value of K_o is specified in the material properties definition.

In general, the K_o procedure should only be used for problems involving: a) a horizontal ground surface; b) a horizontal phreatic surface (that is, hydrostatic pore-water pressures); and c) spatially continuous and horizontal stratigraphic units. Applying the K_o procedure to problems that violate these criteria can result is stress distributions that are not in equilibrium. The gravity loading Insitu type of analysis should be used for such problems; that is, do not specify a value for K_o.

Selecting the K_o procedure (by specifying a K_o for a material) causes the vertical and horizontal effective stresses to be calculated as:

$$\sigma'_v = \sum(\gamma_i h_i - u)$$

$$\sigma'_h = K_0 \sigma'_v$$

where the total vertical stress ($\gamma_i h_i$) is calculated based on the thickness h_i and unit weight γ_i of the overlying soils or water and u is the pore-water pressure.

Limitations

The value of K_o, and therefore Poisson's ratio, for normally compressed soils can be estimated from the effective friction angle according to the expression:

$$K_0 = 1 - \sin\phi'$$

For overconsolidated soil, numerous expressions have been proposed that link K_o to the overconsolidation ratio. These expressions generally produce an unlimited and exponential increase in K_o with increasing overconsolidation ratio. The user, however, must ensure that the input K_o (or Poisson's ratio) does not generate stress states that violate the Coulomb strength law; that is, initial stress states that sit outside the yield surface in illegal stress space.

Establishing the initial stresses with an appropriate value of K_o or Poisson's ratio is mandatory when using a non-linear soil model that incorporates a failure criterion. SIGMA/W includes two such models: a) the elastic-(perfectly) plastic Mohr-Coulomb model; and b) the modified Cam clay model. The linear elastic model will operate despite elements having illegal stress states; however, such modelling practices are not recommended.

According to earth pressure theory, the theoretical limitations on the earth pressure coefficient are:

$$\tan^2\left(45^\circ - \frac{\phi'}{2}\right) < K_0 < \tan^2\left(45^\circ + \frac{\phi'}{2}\right)$$

where the lower and upper bounds are the active and passive earth pressure coefficients, respectively.

Pore-water pressures

Initial pore-water pressures can be specified by a) drawing an initial water table; b) using the results of another finite element analysis (e.g. a SEEP/W or SIGMA/W analysis); or, c) using a spatial function. Options b) or c) should be used if the groundwater conditions differ significantly from the hydrostatic condition.

Initial water table

An initial water table requires the user to: a) specify the maximum negative pressure head (i.e. the capillary rise); and b) to Draw | Initial Water Table as a series of points that are automatically connected to form the water table. The ability to draw an initial water table is actually available to all analysis types.

The initial pore-water pressures are calculated by assuming a linear relationship between the pore-water pressure, unit weight of water γ_w, and the distance above or below the water table z_i:

$$u = \gamma_w z_i$$

Consequently, the pore-water pressure distribution is hydrostatic, increasing positively below the water table and negatively above the water table. The increase in negative pore-water pressures above the water table is terminated once the maximum negative pressure head is attained (Figure 7-2).

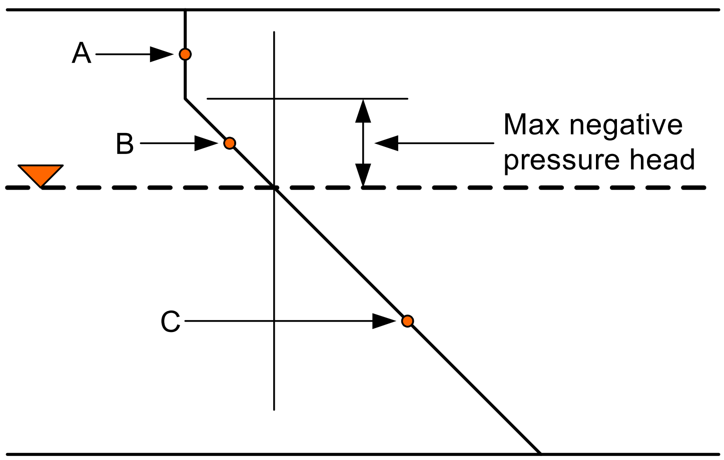


Figure 7-2 Calculation of pore water pressures using water tables

Spatial function

The spatial function alternative allows the user to specify the pressure head at discrete points within the problem domain. The pore-water pressures elsewhere in the problem domain are determined using linear or krigging interpolation. Figure 7-3 provides an example of a linearly interpolated pressure head distribution.

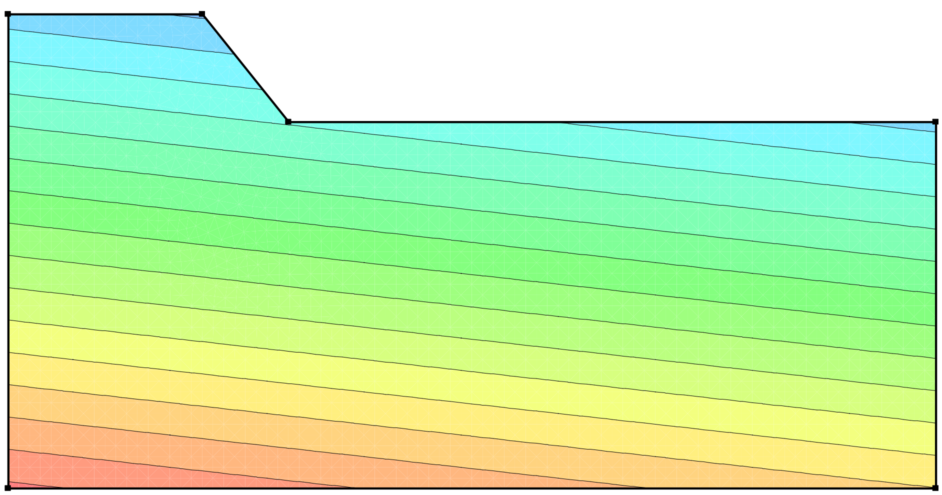


Figure 7-3 Spatial function assigned for initial pressures

Unsaturated Soils

The initial equilibrium stresses that exist within an unsaturated soil are more difficult to establish for two primary reasons. Firstly, the relationship between matric suction and effective stress is non-linear at higher suctions. The non-linear contribution of matric suction to effective stress is not yet accommodated by SIGMA/W because there is no well-established theoretical relationship. Constitutive approximations have been proposed and may be incorporated into SIGMA/W as part of a future release.

Secondly, there is no single and well accepted stress-strain constitutive relationship for unsaturated soil behavior. Fredlund and Rhardjo (1993) proposed the use of the stress-state variables net normal stress $(\sigma - u_a)$ and matric suction $(u_a - u_w)$, where u_a is the gauge air pressure (normally 0 kPa). As a result, Fredlund and Rhardjo (1993) express the coefficient of earth pressure K_o as a ratio of total, not effective, stress components:

$$K_0 = \frac{(\sigma_h - u_a)}{(\sigma_v - u_a)}$$

The theoretical relationship between K_o and Poisson's ratio (not shown) includes the stiffness properties and stress state variables of total vertical stress and matric suction. The expression indicates that K_o decreases with increasing matric suction for a given Poisson's ratio and total vertical stress (i.e. overburden stress). A value of $K_o = 0$ is predicted at high suctions and shallow depths, indicating a tendency for cracking (that is, the total horizontal stress equals zero). The K_o-Poisson's ratio relationship approaches that of a saturated soil at low suctions.

The Insitu analysis and K_o procedures make the assumption that the total (not effective) horizontal stresses are computed in accordance with the user specified Poisson's ratio or K_o, respectively, allowing the user the flexibility of controlling the initial total stress states. The effective stress states are then calculated assuming a linear relationship with matric suction. Consequently, calculating the earth pressure coefficient by hand for unsaturated regions of the problem domain from the SIGMA/W simulated results must be done using the total, not effective, stresses.

Example

The soil profile shown in Figure 7-4 permits the use of simple hand calculations to conceptually illustrate the calculation of effective stresses. The ground surface is horizontal and at an elevation of 10 m. The water table is 2 m above the ground surface. Unit weights of 20 kN/m^3 and 10 kN/m^3 were assumed for the soil and water, respectively. Poisson's ratio was specified as 0.334, which corresponds to $K_o = 0.5$.

The calculations of the effective and total stresses at the ground surface are:

$$u = \gamma_w z_i = 10(2) = 20 \text{ kPa}$$

$$\sigma_v' = \sum(\gamma_i h_i - u) = 10(2) - 20 = 0 \text{ kPa}$$

$$\sigma_h' = K_0 \sigma_v' = 0.5(0) = 0 \text{ kPa}$$

$$\sigma_v = \sigma_v' + u = 0 + 20 = 20 \text{ kPa}$$

$$\sigma_h = \sigma_h' + u = 0 + 20 = 20 \text{ kPa}$$

The calculations of the effective and total stresses at the bottom of the profile are:

$$u = \gamma_w z_i = 10(12) = 120 \text{ kPa}$$

$$\sigma_v' = \sum(\gamma_i h_i - u) = 10(2) + 20(10) - 120 = 100 \text{ kPa}$$

$$\sigma_h' = K_0 \sigma_v' = 0.5(100) = 50 \text{ kPa}$$

$$\sigma_v = \sigma_v' + u = 100 + 120 = 220 \text{ kPa}$$

$$\sigma_h = \sigma_h' + u = 50 + 120 = 170 \text{ kPa}$$

Figure 7-5 and Figure 7-6 show the simulated effective and total stress profiles, respectively (generated by Draw: Graph in CONTOUR mode). The simulated stresses are in agreement with the hand calculations.

This rather simple exercise is included to highlight an essential modelling practice: when possible, verify the simulated results by using hand calculations. This practice ensures proper definition of the problem.

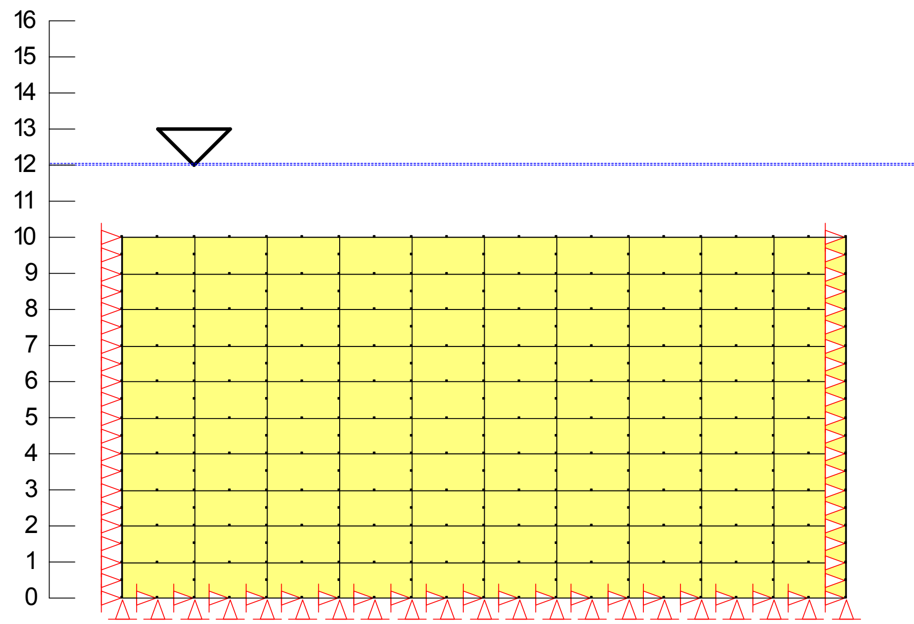


Figure 7-4 Insitu analysis example

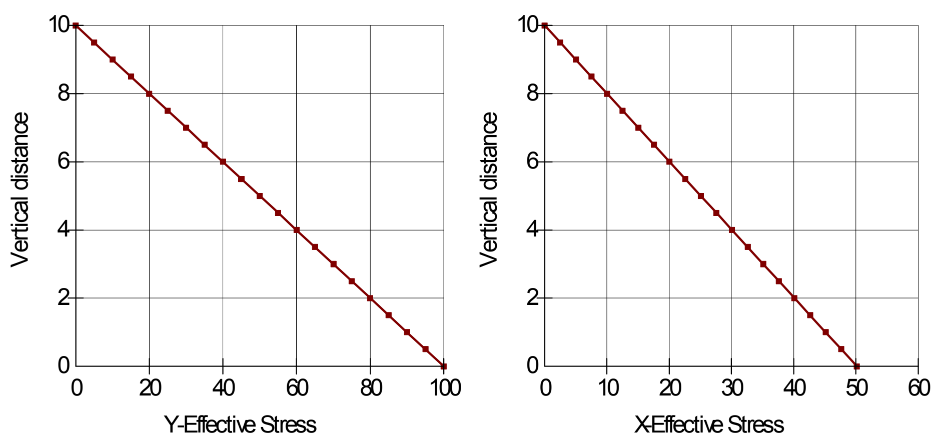


Figure 7-5 Effective stress profiles

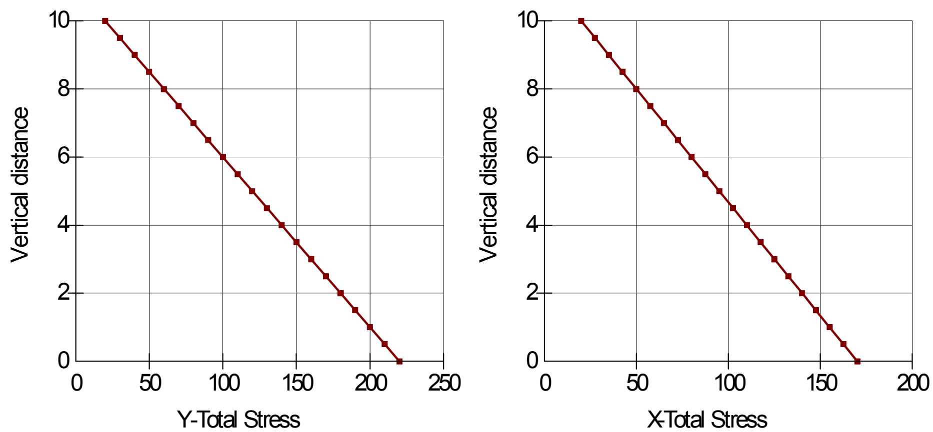


Figure 7-6 Total stress profiles

7.3 Specifying initial conditions

SIGMA/W adds the incremental stresses and displacements (strains) to the initial stress and strains if the user specifies initial conditions in the problem definition. The initial condition options are:

1. Total stresses conditions from a Parent Analysis or Other GeoStudio Analysis (e.g. SIGMA/W or QUAKE/W), with the option to exclude deformation and cumulative values from the previous analysis. (Note: Initial conditions that come from an analysis of the type Insitu are automatically excluded.)
2. Pore-water pressures from a Parent Analysis, Other GeoStudio Analysis (e.g. SIGMA/W, SEEP/W, VADOSE/W or QUAKE/W), Water Table, Spatial Function, or Activation Pore-water Pressure.

7.4 Load / deformation analysis

The Load/Deformation Analysis type is used whenever you want to apply loads and find the resulting stress changes and displacements, including simulation of fill placement and excavation construction procedures. In the case of a fill placement analysis, the weight of the fill is added to your model on the first load step that each fill layer is activated. In the case of an excavation analysis, SIGMA/W calculates the resultant forces associated with removing the excavated elements and applies these forces as negative values at the nodes along the excavation face. The magnitudes of the excavated forces are written to file so you can check them in the Contour program.

7.5 Dynamic deformation analysis

New in GeoStudio 7.1 is the ability to link SIGMA/W with output results from a QUAKE/W dynamic analysis or even another SIGMA/W analysis in which you need to compute the deformations due to nodal forces that are computed from incremental element stresses in a different analysis. For example, if you solve a QUAKE/W dynamic analysis the output results will include the new soil total stresses and pore-water pressures at each saved time step. SIGMA/W can read in the output from QUAKE/W at each time step, subtract it from the time before, and create an incremental force at each node. The force will then result in a deformation according to your choice of constitutive model.

The *Dynamic Deformation* analysis is fundamentally an elastic-plastic stress redistribution analysis. The dynamic stresses are redistributed for each time step that the QUAKE/W results are saved to file.

SIGMA/W computes an incremental load vector based on the stress difference between two time steps. The load vector is computed for each element from:

$$\{\Delta F\} = \int_v [B]^T \{\Delta\sigma\} dv$$

where $\{\Delta\sigma\} = \{\sigma_n\} - \{\sigma_{n-1}\}$ and n is the saved time step.

In words, the incremental load vector is the algebraic difference in the stress states between two successive time steps.

Each load step may produce some elastic strains and some plastic strains. It is the accumulation of the plastic strains and deformations that are a measure of the permanent deformations.

When you choose this analysis type you must select an already solved analysis that outputs stresses. The SIGMA/W model will then determine the necessary time stepping scheme such that an incremental load is applied for every change in saved output of the source file.

This type of analysis is applicable if the QUAKE/W dynamic inertial forces cause plastic strains in the soil which causes permanent deformations. If permanent deformations arise primarily from the generation of excess pore-pressures and the associated strength loss a stress redistribution analysis as described in the next section is likely more appropriate.

7.6 Stress redistribution

Concept and procedure

There are many situations where the current computed state of stress is higher than the soil strength; that is, the soil is over-stressed. Typically, this happens in a Linear-Elastic analysis where stresses are computed without consideration to the soil strength. Another common situation is where there is an increase in pore-pressures while the total stress remains constant like in the case of infiltration of water into the ground.

The situation of a change in pore-pressure is illustrated in Figure 7-7. Prior to the increase in pore-pressure the state of stress is underneath the strength envelope; after the pore-pressure increases there is some over stressing since part of the Mohr circle is above the strength envelope.

Alternatively, there could be some strength loss due to a change in the soil grain structure or due to perhaps a soil chemistry change. This also could lead to some overstressing as illustrated in Figure 7-8.

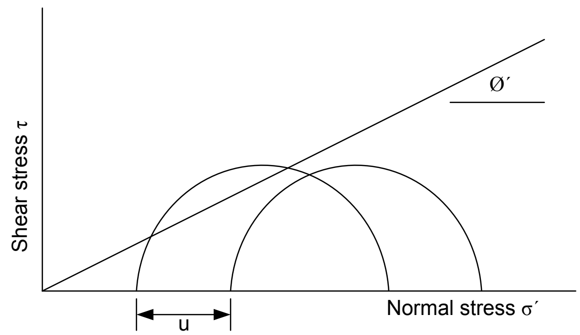


Figure 7-7 Over-stressing due to pore-pressure increase

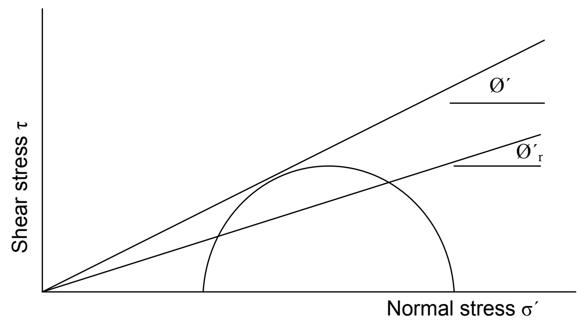


Figure 7-8 Over-stressing due to change in soil grain structure

Physically it is not possible to over-stress a soil. The excess stress will automatically be shed to neighbouring elements. Numerically, it is possible to have over-stressing as in a Linear-Elastic analysis for example.

SIGMA/W has a special routine (analysis type) for re-distributing stresses. This is done by using the Elastic-Plastic Material (constitutive) soil model. SIGMA/W computes a load vector for each element that represents the over-stressed portion within the element. This load vector is often referred to as the unbalanced load – a portion of the stress is unbalanced by the available shear resistance. The unbalanced load vector is applied as in any regular finite element analysis which creates deformations and stress changes. The procedure is iteratively repeated until the unbalanced load is zero or within some tolerable convergence criteria.

Strength reduction stability

A procedure known as the Strength Reduction method of stability analysis has come into geotechnical practice. The approach is to artificially weaken the soil in an elastic-plastic analysis until the slope "fails" Dawson et al., 1999; Griffiths and Lane, 1999). Numerically, this occurs when the soil strength is so low that it is no longer possible to obtain a converged solution.

The SIGMA/W finite element equations are in essence equations of equilibrium. Not being able to obtain a converged solution therefore infers the system is beyond the point of limiting equilibrium.

With this method the factor of safety is deemed to be the strength reduction factor which is defined as,

$$SRF = \left(\frac{\tan \phi'}{\tan \phi_f} \right) = \left(\frac{c'}{c_f} \right)$$

where $Ø_f$ and c_f are the soil parameters at 'failure".

The Stress Redistribution analysis type in SIGMA/W can be used to do a strength reduction stability analysis. The soil strength can be reduced using the SRF equation until it is no longer possible to obtain a solution. To be consistent with the published procedure, the strengths of all the soils should be reduced by the same factor, although this is not a requirement in SIGMA/W.

The Strength Reduction method has some limitations as discussed by Krahn (2007), and consequently a more preferred method is to use the SIGMA/W computed stresses directly in SLOPE/W together with conventional trial slip surfaces.

The basis for using SIGMA/W-computed stress for stability analysis is presented in the SLOPE/W documentation. Also, the SIGMA/W Illustrative Example called Stress Redistribution demonstrates and further discusses the procedure.

Permanent deformations

The redistribution of overstressing manifests itself in deformations. It is readily understandable that if there is going to some strength loss, there will likely be some movements or slumping. The Stress Redistribution type of analysis can consequently be used to estimate permanent deformation resulting from the strength loss.

Consider the case of earthquake shaking that leads to the generation of excess pore-pressure. The excess pore-pressures may cause some over-stressing as noted earlier which in turn can lead to some permanent deformations.

The SIGMA/W Stress Redistribution procedure is used, demonstrated and discussed in two QUAKE/W Illustrative Examples on the San Fernando Dam case histories.

Limiting equilibrium

The SIGMA/W finite element equations are fundamentally equations of equilibrium. The implication is that to obtain a solution to the equations, the structure must be stable. If the applied or driving forces on the structure are such that it is no longer stable, then it is not possible to obtain a solution to the finite element equations. In numerical terms this happens when it is not possible to obtain a converged solution.

In terms of a stability analysis, the factor of safety must be 1.0 or greater in order to obtain a converged solution.

It is for this reason that great care needs to be exercised when using the Stress Redistribution analysis. The soil strength should not be reduced arbitrarily without giving some though to equilibrium. It is always advisable and good modeling practice to reduce the strength gradually in steps if at all possible.

It is also advisable to do a strength reduction analysis in conjunction with a SLOPE/W stability analysis. The stability analysis can give you a guide as to when you are approaching the point of limiting equilibrium.

Strain Softening

SIGMA/W does not have a rigorously formulated strain-softening constitutive model. The Stress Redistribution algorithm can however be used to consider the effects of a strength loss. The simplification here is that the end point is known but the path to the end point is undefined as illustrated in Figure 7-9. The path to the reduced strength could have been any of the dashed lines or any other physically admissible path.

This approach is acceptable when the reduced strength has been created by some process other than straining like, for example, the sudden soil-grain structure collapse of very loose sands or due to some chemical change in the soil. If the strength loss is primarily due to the slow rearrangement of soil particles due to straining, then the simplified Stress Redistribution approach is at best an estimate of the actual straining softening, especially if large strains are required to reach the reduced (residual) strength.

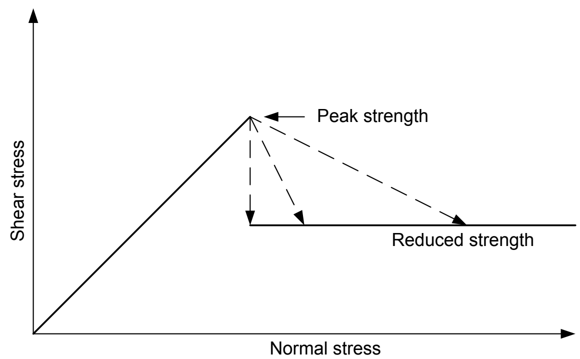


Figure 7-9 Path from peak strength to a reduced strength

A side effect of the Stress Redistribution analysis is that the reduced strength has to be specified; that is, the reduced strength does not develop as in a true strain softening constitutive model.

The Stress Redistribution approach is not entirely representative of actual conditions but for many field cases it can provide a reasonable estimate of the effects of the strength loss.

7.7 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 7-10 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.

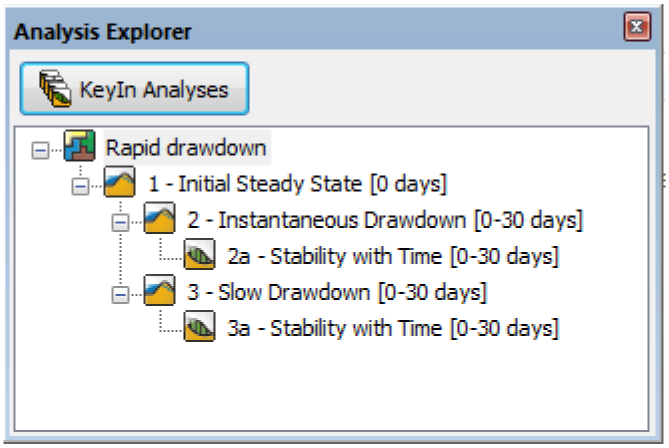


Figure 7-10 Example of an Analysis Tree in GeoStudio

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.

8 Coupled Stress-Pore Pressure Analysis

SIGMA/W is formulated to solve soil consolidation problems using a fully coupled or any of several un-coupled options. A fully coupled analysis requires that both the stress-deformation and seepage dissipation equations be solved simultaneously. The theoretical formulation for this rigorous method along with some practical limitations is discussed below. With the coupled analysis, it is no longer necessary to have SEEP/W as well as SIGMA/W. All hydraulic properties and boundary conditions can be developed and applied from within SIGMA/W.

When coupled, three equations are created for each node in the finite element mesh. Two are equilibrium (displacement) equations and the third is a continuity (flow) equation. Solving all three equations simultaneously gives both displacement and pore-water pressure changes.

Of perhaps more use in many real situations is the un-coupled consolidation formulation in SIGMA/W. In this analysis, the seepage analysis is solved independently of the volume change analysis. The incremental change in pore-water pressures from the seepage solutions are used at each load step in the stress-deformation calculation in order to determine the change in effective stresses. With the un-coupled analysis, the changes in pore-water pressures can be obtained from any other SIGMA/W, SEEP/W, VADOSE/W or QUAKE/W analysis and for any two or multiple time steps in those analyses.

Details of both types of consolidation analyses are discussed below along with several examples to illustrate the use and interpretation of this powerful option. The first part of this chapter discusses general theoretical issues related to consolidation and the fully coupled formulation. With this developed, it is more obvious what is happening when the formulation is un-coupled.

8.1 Constitutive equation for soil structure

The incremental strain-stress relationship for an unsaturated soil medium can be written as follows (Fredlund and Rahardjo, 1993):

$$\begin{Bmatrix} \Delta\varepsilon_x \\ \Delta\varepsilon_x \\ \Delta\varepsilon_x \\ \Delta\gamma_{xy} \\ \Delta\gamma_{yz} \\ \Delta\gamma_{zx} \end{Bmatrix} = \frac{1}{E}\begin{bmatrix} 1 & -\nu & -\nu & 0 & 0 & 0 \\ -\nu & 1 & -\nu & 0 & 0 & 0 \\ -\nu & -\nu & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 2(1+\nu) & 0 & 0 \\ 0 & 0 & 0 & 0 & 2(1+\nu) & 0 \\ 0 & 0 & 0 & 0 & 0 & 2(1+\nu) \end{bmatrix}\begin{Bmatrix} \Delta\left(\sigma_x - u_a\right) \\ \Delta\left(\sigma_y - u_a\right) \\ \Delta\left(\sigma_z - u_a\right) \\ \Delta\tau_{xy} \\ \Delta\tau_{yz} \\ \Delta\tau_{zx} \end{Bmatrix}$$
$$+\frac{1}{H}\begin{bmatrix} 1 & & & & & \\ & 1 & & & & \\ & & 1 & & & \\ & & & 0 & & \\ & & & & 0 & \\ & & & & & 0 \end{bmatrix}\begin{Bmatrix} \Delta\left(u_a - u_w\right) \\ \Delta\left(u_a - u_w\right) \\ \Delta\left(u_a - u_w\right) \\ \Delta\left(u_a - u_w\right) \\ \Delta\left(u_a - u_w\right) \\ \Delta\left(u_a - u_w\right) \end{Bmatrix}$$

where:

ε = normal strain,

γ = engineering shear strain,

σ = normal stress,

τ = shear stress,

u_a = pore-air pressure,

u_w = pore-water pressure,

E = elastic modulus for soil structure,

H = unsaturated soil modulus for soil structure with respect to matrix suction ($u_a - u_w$), and

ν = Poisson's ratio.

This equation is similar in form to the constitutive equation presented by Biot (1941).

This relationship can be re-written in the following stress-strain form for a two-dimensional space.

$$\begin{Bmatrix} \Delta(\sigma_x - u_a) \\ \Delta(\sigma_y - u_a) \\ \Delta(\sigma_z - u_a) \\ \Delta\tau_{xy} \end{Bmatrix} = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & 0 & 0 & 0 \\ & 1 & 0 & 0 \\ & & 1 & 0 \\ & & & \dfrac{1-2\nu}{2(1+\nu)} \end{bmatrix} \begin{Bmatrix} \Delta\left(\varepsilon_x - \dfrac{u_a - u_w}{H}\right) \\ \Delta\left(\varepsilon_y - \dfrac{u_a - u_w}{H}\right) \\ \Delta\left(\varepsilon_z - \dfrac{u_a - u_w}{H}\right) \\ \Delta\gamma_{xy} \end{Bmatrix}$$

Alternatively, this incremental stress-strain relationship can be written as:

Equation 8-1 $\{\Delta\sigma\} = [D]\{\Delta\varepsilon\} - [D]\{m_H\}(u_a - u_w) + \{\Delta u_a\}$

where:

[D] = drained constitutive matrix

$\{m_H\}^T = \left\langle \frac{1}{H} \frac{1}{H} \frac{1}{H} 0 \right\rangle$

If it can be further assumed that air pressure remains atmospheric at all times, Equation 8-1 becomes:

$$\{\Delta\sigma\} = [D]\{\Delta\varepsilon\} + [D]\{m_H\}u_w$$

On the other hand, for a soil element which is fully saturated, the total stress on the soil structure is given by:

$$\{\Delta\sigma\} = \{D\}\{\Delta\varepsilon\} + \{m\}\Delta u_w$$

where:

{m} = unit isotropic tensor, < 1 1 1 0 >.

Comparing these last two equations, it can be seen that, when the soil is fully saturated (i.e. S = 100%):

$$[D]\{m_H\} = \{m\}$$

For a linearly elastic material, this condition is satisfied when:

$$H = \left(\frac{E}{1-2v} \right)$$

8.2 Flow equation for water phase

The two-dimensional flow of pore-water through an elemental volume of soil based on Darcy's law and expressed in terms of pore-pressure is described by the following equation:

$$\frac{\partial}{\partial x}\left(k_x \frac{1}{\gamma_w} \frac{\partial u_w}{\partial x} \right) + \frac{\partial}{\partial y}\left(k_y \left(\frac{1}{\gamma_w} \frac{\partial u_w}{\partial y} + 1 \right) \right) + Q = \frac{\partial \theta_w}{\partial t}$$

where:

k_x, k_y	=	the hydraulic conductivity in x and y direction, respectively,
u_w	=	pore-water pressure,
γ_w	=	the unit weight of water,
Q	=	a sink or source specified boundary flow
θ_w	=	the volumetric water content, and
t	=	time.

The unity (1) term represents the flow due to gravity as opposed to flow due to a pore-pressure difference. This term is not carried through the formulation here. It is however accounted for in the end.

The volumetric water content for an elastic material is given by the following expressions (Dakshanamurthy et al. , 1984):

Equation 8-2 $$\theta_w = \frac{\beta}{3} \varepsilon_v - \omega u_w$$

and:

Equation 8-3
$$\beta = \frac{E}{H} \frac{1}{(1-2\nu)} = \frac{3K_B}{H} \quad \text{and,}$$
$$\omega = \frac{1}{R} - \frac{3\beta}{H}$$

where:

K_B	=	bulk modulus,
R	=	a modulus relating the change in volumetric water content with change in matric suction.

Since a soil-water characteristic curve is a graph showing the change of volumetric water content corresponding a change in matrix suction, $(u_a - u_w)$, the parameter R can be obtained from the inverse of the slope of the soil-water characteristic curve.

Assuming material properties remain unchanged during an increment, Equation 8-2 can be written in the following incremental form.

Equation 8-4 $\Delta\theta_w = \beta\Delta\varepsilon_v - \omega\Delta u_w$

At full saturation, the change in volumetric water content, $\Delta\theta_w$, is equal to the change in volumetric strain, $\Delta\varepsilon_v$. This condition is satisfied in Equation 8-3 when ω is equal to zero.

8.3 Finite element formulation for coupled analysis

In a coupled consolidation analysis, both equilibrium and flow equations are solved simultaneously.

In SIGMA/W, the finite element equilibrium equations are formulated using the principle of virtual work, which states that for a system in equilibrium, the total internal virtual work is equal to the external virtual work. In the simple case when only external point loads {F} are applied, the virtual work equation can be written as:

$$\int \left\{\varepsilon^*\right\}^T \left\{\Delta\sigma\right\} dV = \int \left\{\delta^*\right\}^T \left\{F\right\} dV$$

where:

$\{\delta^*\}$	=	virtual displacements,
$\{\varepsilon^*\}$	=	virtual strains, and
$\{\sigma\}$	=	internal stresses.

Substituting Equation 8-4 into this equation and applying numerical integration, it can be shown that the finite element equations that SIGMA/W solves are given by:

$$\sum[B]^T[D][B]\{\Delta\delta\} + \sum[B]^T[D]\{m_H\}\langle N\rangle\{\Delta u_w\} = \sum F\,,$$

$$[K] = [B]^T[D][B],$$

$$[L_d] = [B]^T[D]\{m_H\}\langle N\rangle \text{, and}$$

$$\{m_H\}^T = \left\langle \frac{1}{H} \quad \frac{1}{H} \quad \frac{1}{H} \quad 0 \right\rangle$$

where:

[B]	=	gradient matrix, (also called the strain matrix),
[D]	=	drained constitutive matrix,
[K]	=	stiffness matrix,
$[L_d]$	=	coupling matrix,
$\{\Delta\delta\}$	=	incremental displacement vector, and

Δ_{uw} = incremental pore-water pressure vector.

For a fully saturated soil, the coupling matrix, $[L_d]$, can be written as:

$$[L_d] = [B]^T \{m\}\langle N\rangle, \quad \text{with } \{m\}^T = \langle 1, 1, 1, 0\rangle$$

The flow equation can similarly be formulated for finite element analysis using the principle of virtual work in terms of pore-water pressure and volumetric strains. If virtual pore-water pressures, u^*_w, are applied to the Flow Equation and integrated over the volume, the following virtual work equation can be obtained.

$$\int u_w^* \left[\frac{k_x}{\gamma_w} \frac{\partial u_w^*}{\partial x^2} + \frac{k_y}{\gamma_w} \frac{\partial^2 u_w}{\partial y^2} + \frac{\partial \theta_w}{\partial t} \right] dV = 0$$

Applying integration by parts to this equation gives:

$$-\int \left[\frac{k_x}{\gamma_w} \frac{\partial u_w^*}{\partial x} \frac{\partial u_w}{\partial x} + \frac{k_y}{\gamma_w} \frac{\partial u_w^*}{\partial y} \frac{\partial u_w}{\partial y} \right] dV + \int u_w^* \frac{\partial \theta_w}{\partial t} dV = \int u_w^* v_n dA$$

where:

V_n = boundary flux.

Substituting in the expression for the volumetric water content, θ_w, gives:

$$-\int \left[\frac{k_x}{\gamma_w} \frac{\partial u_w^*}{\partial x} \frac{\partial u_w}{\partial x} + \frac{k_y}{\gamma_w} \frac{\partial u_w^*}{\partial y} \frac{\partial u_w}{\partial y} \right] dV + \int u_w^* \frac{\partial \left(\beta \varepsilon_v - \omega u_w\right)}{\partial t} dV = \int u_w^* v_n dA$$

Using finite element approximations, this equation can be written as:

$$-\int \frac{1}{\gamma_w} [B]^T [K_w][B]\{u_w\} dV - \int \langle N\rangle^T \langle N\rangle \left\{ \frac{\partial(\omega u_w)}{\partial t} \right\} +$$

$$\int \langle N\rangle^T \{m\}^T [B] \left\{ \frac{\partial(\beta\delta)}{\partial t} \right\} dV \quad = \int \langle N\rangle^T v_n \, dV$$

where:

$$[K_f] = \int [B]^T [K_w][B] dV,$$

$$[M_N] = \langle N\rangle^T \langle N\rangle, \text{ and}$$

$$[L_f] = \int \langle N\rangle^T \{m\}^T [B] dV$$

with:

[B] = gradient matrix,

$[K_w]$ = hydraulic conductivity matrix,

$[K_f]$	=	element stiffness matrix,
$\langle N \rangle$	=	row vector of shape functions,
$[M_N]$	=	mass matrix,
$[L_f]$	=	coupling matrix for flow,
$\{m\}^T$	=	isotropic unit tensor, $< 1\ 1\ 1\ 0 >$, and
δ	=	nodal displacement.

Integrating this equation from time t to time t + Δt gives:

$$-\int_t^{t+\Delta t} \frac{1}{\gamma_w}\left[K_f\right]\{u_w\}dt - \int_t^{t+\Delta t}\left[M_N\right]\left\{\frac{\partial(\omega u_w)}{\partial t}\right\}dt + \int_t^{t+\Delta t}\left[L_f\right]\left\{\frac{\partial(\beta\delta)}{\partial t}\right\}dt$$
$$= \int_t^{t+\Delta t}\langle N\rangle^T v_n dAdt$$

Applying the time differencing technique using θ as the time stepping factor to this equation, the following finite element equation is obtained.

$$-\frac{\Delta t}{\gamma_w}\left(\theta\left[K_f\right]\{u_w\}\big|_{t+\Delta t} + (1-\theta)\left[K_f\right]\{u_w\}\big|_t\right) - \left[M_N\right](\omega u_w)\big|_t^{t+\Delta t} +$$
$$\left[L_f\right]\{\beta\delta\}\big|_t^{t+\Delta t} = \Delta t\int\langle N\rangle^T\left(\theta v_n\big|_{t+\Delta t} + (1-\theta)v_n\big|_t\right)dA$$

When the backward (fully implicit) time-stepping scheme is used (by setting $\theta = 1$) and assuming that ω and β remain constant within a time increment, this equation becomes:

$$-\frac{\Delta t}{\gamma_w}\left[K_f\right]\{u_w\}\big|_{t+\Delta t} - \omega\left[M_N\right]\{\Delta u_w\} + \beta\left[L_f\right]\{\Delta\delta\} = \Delta t\,\{Q\}\big|_{t+\Delta t}$$

where:

$\{Q\}$	=	the flow at boundary nodes.

In order to obtain an equation involving an incremental pore-water pressure only, the first term, $\frac{\Delta t}{\gamma_w}\left[K_f\right]\{u_w\}\big|_t$, is added to both sides of the equation. The resultant equation describing the flow of pore-water is:

$$\beta\left[L_f\right]\{\Delta\delta\} - \left(\frac{\Delta t}{\gamma_w}\left[K_f\right] + \omega\left[M_N\right]\right)\{\Delta u_w\} = \Delta t\left(\{Q\}\big|_{t+\Delta t} + \frac{1}{\gamma_w}\left[K_f\right]\{u_w\}\big|_t\right)$$

A coupled consolidation analysis for saturated/unsaturated soils is thus formulated using incremental displacement and incremental pore-water pressure as field variables.

In summary, the coupled equations for finite element analysis are rewritten in the following form.

$\left[K\right]\{\Delta\delta\} + \left[L_d\right]\{\Delta u_w\} = \{\Delta F\}$, and

$$\beta\left[L_f\right]\{\Delta\delta\}-\left(\frac{\Delta t}{\gamma_w}\left[K_f\right]+\omega\left[M_N\right]\right)\{\Delta u_w\}=\Delta t\left(\{Q\}\big|_{t+\Delta t}+\left[K_f\right]\{y\}+\frac{1}{\gamma_w}\left[K_f\right]\{u_w\}\big|_t\right)$$

where:

$$[K]=\Sigma[B]^T[D][B]$$

$$[L_d]=\Sigma[B]^T[D]\{m_H\}\langle N\rangle$$

$$\{m_H\}=\left\langle \frac{1}{H} \quad \frac{1}{H} \quad \frac{1}{H} \quad 0 \right\rangle$$

$$\left[K_f\right]=\Sigma[B]^T[K_w][B]$$

$$[M_N]=\Sigma\langle N\rangle^T\langle N\rangle$$

$$\left[L_f\right]=\Sigma\langle N\rangle^T\{m\}[B]$$

Note that the gravitational term $[K_f]\{y\}$ now has been added to the flow equation.

Also, as given by Equation 8-3:

$$\beta=\frac{E}{H}\frac{1}{(1-2v)}=\frac{3K_B}{H}$$

$$\omega=\frac{1}{R}-\frac{3\beta}{H}$$

In order for these equations to model the fully saturated case, the following conditions must be satisfied:

β = 1,

ω = 0, and

$[L_f]$ = $[L_d]^T$

8.4 Computed material parameters

In SIGMA/W 2007, *H* and *R* are compute from the specified *E*-modulus and Poisson's ratio ν.

E and *H* are related by the equation,

$$H = \left(\frac{E}{1-2\nu} \right)$$

This relationship is fundamentally only correct for saturated conditions. The relationship is much more complex for unsaturated conditions as shown in a paper by Vu and Fredlund (2006). In reality the relationship is a three-dimensional constitutive surface. SIGMA/W has as yet not reached this level of sophistication. One of the included Illustrative Examples named "Heave due to infiltration" suggests that the current SIGMA/W implementation is adequate for practical field problems.

Also, at saturation with β equals 1.0 and ω = 0.0,

$$\frac{1}{R} = \frac{3(1-2\nu)}{E} = \frac{1}{K_B}$$

When ν =1/3,

$$\frac{1}{R} = \frac{1}{E} = \frac{1}{K_B} \text{ or } R = E = K_B$$

As with the *H*-modulus, SIGMA/W computes *R* from the specified *E*-modulus.

Under saturated conditions, the coefficient of volume compressibility m_v is equivalent to the slope of the Volumetric Water Content function where the pore-pressure is positive. By definition, the slope is also equal to 1/*R* and consequently m_v = 1/*R* = 1/*E* when ν = 1/3.

The coefficient of volume change m_v is specified and defined in conjunction with a Volumetric Water Content function. However, in a SIGMA/W coupled analysis, m_v is computed from E. This is required to enforce the above relationships between *R*, *E, H* and ν for saturated conditions.

8.5 Pore-pressure response ratio

The bulk modulus of water can be less than the bulk modulus of the soil skeleton. This means that even under saturated conditions (positive pore-pressure) the pore-pressure response to an applied load will be less than 100 percent. Numerically in the coupled formulation this is represented by R being less than E even when the pore-pressure is positive.

This can be achieved in SIGMA/W by specify a Load Response Ratio. By default the ratio is 1.0. Recall that when Poisson's ratio is 1/3, R is computed as being equal to E. If the Load Response Ratio is 0.9, for example, R is equal to 0.9E if Poisson's ratio is 1/3.

The Load Response Ratio is analogous to the total stress pore-pressure parameter B. In a one-dimensional constrained loading case the Load Response Ratio is equal to B if Poisson's ratio is 1/3. A specified Load Response Ratio of 0.9, for example, will result in a 90% pore-pressure response due to an applied surface load.

8.6 Coupled analysis data

Material properties

To do a fully coupled analysis you must select the following:

- Analysis Type: Coupled Stress/PWP
- Material Category: Effective Parameters w/PWP Change
- Material Model: Linear Elastic, Elastic-Plastic, or Modified Cam-Clay (or some Add-In model)

Other material models like the non-linear hyperbolic model, for example, cannot be used in a coupled analysis.

In addition it is necessary to define a:

- Volumetric Water Content function
- Hydraulic Conductivity function

Boundary conditions

In a coupled analysis it is mandatory to defined and apply **both** displacement and hydraulic boundary conditions. Remember that, two sets of equations are solved simultaneously and boundary conditions are required for the displacement **and** the hydraulic equations.

8.7 Saturated-only coupled analysis

Sometimes a coupled analysis can be numerically greatly simplified by allowing pore-pressure to change with time only in the saturated zones. This is an option with coupled analyses.

In a Saturated-Only coupled analysis, SIGMA/W converts initial negative pore-pressures into a fixed hydraulic boundary condition. The effect of this is that negative pore-pressures do not change with time. Stated another way, unsaturated zones remain unsaturated and do not change with time. There may be some volume change in the unsaturated zones due to elastic compression or over-stressing but the size of the unsaturated zone will not change

With this option, SIGMA/W solves the coupled equations but the initial negative pore-pressures become hydraulic boundary conditions which fixes the negative pore-pressures.

This is not an unreasonable approach when the issue is considered in light of the storage capacity of unsaturated soil versus saturated soil. The Volumetric Water Content function is representative of the soil's ability to store and/or release water. When the soil is saturated, the slope of the Water Content function is m_v, as discussed earlier. When the soil is unsaturated, the slope of the Water Content function is much steeper than m_v inferring that the soil has a much higher storage capacity. Viewed another way, the small amount of water that can be squeezed out a saturated soil can easily be stored in the adjoining unsaturated soil with its relatively large storage capacity.

Coupled analyses within unsaturated zones can sometimes cause considerable numerical convergence difficulties, especially when the Volumetric Water Content function is very steep like for sandy soils. Instead of struggling with the numerical complexity it is often advantageous to use the Saturated-Only option. A good close approximation is usually better than one with poor convergence.

8.8 Uncoupled analysis

A coupled analysis in essence involves solving the SIGMA/W equilibrium equations and seepage continuity equations simultaneously. Sometimes instead of solving the two sets of equations both at the same time, it is numerically advantageous to solve the SEEP/W transient flow equations first and then use the SEEP/W results in the coupled equations as known hydraulic boundary conditions. The change in pore-pressures are calculated first and then the related volume change is computed for the previously computed pore-pressure changes. This is known as an uncoupled analysis.

The following is a simplistic pictorial view of the coupled equations, where [L] is the coupling matrix, Δd is an incremental displacement and Δu is an incremental change in pore-pressure.

$$\left[\begin{array}{cc} [K]_s & [L] \\ \hline [L]^t & [K]_w \end{array}\right]\left\{\begin{array}{c} \Delta d \\ \hline \Delta u \end{array}\right\}=\left\{\begin{array}{c} F \\ \hline Q \end{array}\right\}$$

The top half represents the SIGMA/W equilibrium equation and the bottom half represents the seepage continuity equation. In an uncoupled analysis Δu is known from some other analysis, like SEEP/W (or VADOSE/W) for example, and becomes a known boundary condition in the above equation. The solution then gives the volume change resulting from the previously computed Δu.

An uncoupled analysis can be done with the Volume Change analysis type in SIGMA/W.

When the Volume Change analysis type is selected you have two options with which to specify the pore-pressure change.

- One-step initial and final pore-pressure: with this option you need to identify the result files that have the initial and final pore-pressures. SIGMA/W subtracts the two values to get Δu.
- Multiple step transient analysis: with this option you need to select the start time in a transient analysis. SIGMA/W looks at the related time sequenced and subtracts the pore-pressures from too successive steps to get Δu and then computes the volume change for each step.

The results from a coupled versus an uncoupled analysis are often quite similar which can be used to great advantage in highly complex situations.

Often it is highly recommendable to do a seepage analysis first to gain a full understanding of the flow regime. It helps with getting the correct hydraulic boundary conditions, appropriate material properties and a suitable time stepping sequence. Once this part of the analysis is well understood, the results can be used in an uncoupled volume change analysis or the gained knowledge can be used to set up a coupled analysis. It is a procedure whereby the analysis can be done in two more manageable parts rather than doing everything in one more complicated step.

The Illustrative Example, Settlement Due To Pumping Well, demonstrates the procedures in an uncouple Volume Change analysis.

An uncoupled analysis cannot capture what is known as the Mendel-Cryer effect discussed further on in this Chapter. In spite of this, the ultimate settlements from a coupled and uncoupled analysis are often not that dissimilar.

The uncoupled approach offers some advantages over the coupled analysis in that the pore-pressure change can come from a SEEP/W, VADOSE/W or QUAKE/W analysis. This greatly expands the range and types of problems that can be analyzed.

In an uncoupled consolidation analysis you should not apply any other loading in the analysis. For example, you should not add a surface load that will cause a change in pore-water pressure because it will not be accounted for correctly. The pore-water pressures are supplied by the seepage solution. Applying any load that results in a change in total stress may lead to a meaningless solution.

8.9 Time stepping sequence

In a SIGMA/W coupled analysis it is necessary to define an appropriate time stepping sequence exactly like in a transient SEEP/W analysis. All the guidelines for creating a suitable time-steeping discussed in the SEEP/W documentation, also apply here. This is another reason why it is very useful to first do a transient SEEP/W analysis before proceeding onto a coupled analysis.

There are no firm rules for establishing an appropriate time stepping sequence. There however some responses to consider which can help to at least get started in the right direction.

Like in a consolidation test, the initial time steps need to be fairly small and then increase in size with time exponentially.

Time step sizes can be too small just as well as be too large. Time steps that are too small will lead to numerical noise which can confuse the result interpretation. Moreover, it takes too much and unnecessary computing time. Time steps that are too large may misrepresent the actual transient process.

A very rough guide to setting a starting Δt is to divide m_v by K_{sat}. This will put you at least in the right order of magnitude.

In the end there is no alterative but to try various time step size sequences and adjust the sequences until there is a nice gradual movement in the solution towards the long term steady-state condition.

Another useful modeling practice is to do a steady-sate SEEP/W analysis early on in the project to have a clear mental picture as to the transient end point.

8.10 Changes in Conductivity

When a soil consolidates, the hydraulic conductibility (or permeability) decreases as the soil grain structure becomes more compact. This behavior can be accommodated in a SIGMA/W coupled analysis with a K-modifier function such as presented in Figure 8-1

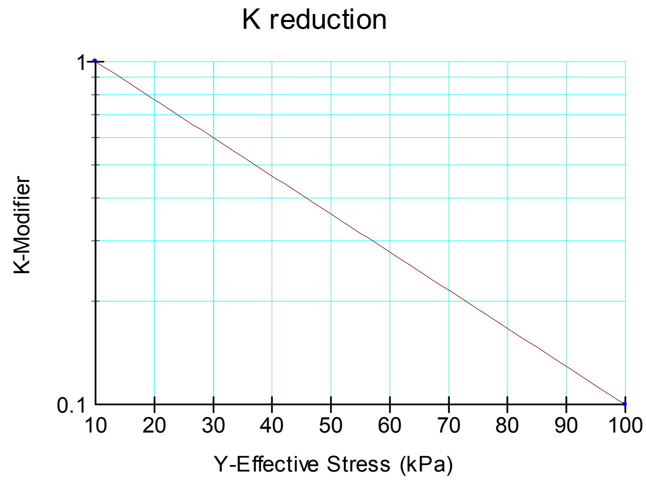


Figure 8-1 Sample K-modifier function

Basically, the K_{sat} from the specified Hydraulic Conductivity function is modified by a factor depending on the vertical effective stress state. In the above figure, K diminishes by a factor of 10 as the effective stress increases from 10 to 100 kPa.

Such a relationship can be obtained from a conventional odometer test. The conductivity K can be computed from the equation:

$K = \dfrac{c_v \, \gamma_w \, a_v}{(1+e_o)}$ where c_v is the coefficient of consolidation, also obtained from the odometer test.

The coefficient of compressibility a_v is,

$$a_v = \frac{e_1 - e_2}{\sigma_2' - \sigma_1'} = \frac{\Delta e}{\Delta \sigma'}$$

The parameter a_v is an incremental change in void ratio for an incremental change in effective stress. The computed *K* corresponds with the average effective stress in the stress increment. This information can be used to create the *K*-modifier function.

The procedure is to set the *K*-modifier function to 1.0 at the initial or starting approximate average effective stress within the soil layer. For all other vertical effective stresses, K_{sat} will be modified according to the specified *K*-modifier function, and *K* will be modified as the soil consolidates and the effective stresses increase.

SIGMA/W references the K-modifier to the vertical effective stress instead of the void ratio. There are several reasons for this but the main reason is that it is relatively straight forward to hand-calculate σ'_v from the overburden making it a convenient reference point for spot checking the results.

The Illustrative Examples Chapter includes an example on the sequential placement of mine tailings which makes use of a K-modifier function.

8.11 Submerged Deposition

The coupled formulation can be used to simulate the deposition of a saturated material (e.g. hydraulically deposited tailings). The activation of a material of the type Effective Drained w/ PWP Change will result in the generation of excess pore-water pressures due to self-weight. In order to maintain the water level at the top of the newly activated material – that is, prevent the development of excess pore-water pressures at the top of the material – a hydraulic boundary condition must be applied. The hydraulic boundary condition represents the location of the water level at the time of deposition. In addition, the staged construction functionality of SIGMA/W can be used to simulate multiple deposits of saturated materials. There are a number of Detailed Example Files that illustrate the use of the coupled formulation for problems of this type.

8.12 Validation

It has been known for a long time that when a saturated sphere is compressed, the pore-pressure response inside the sphere will be greater than the applied pressure. This is known as the Mendel-Cryer effect. A rigorous test for any coupled formulation is to demonstrate that the solution will exhibit this effect.

Figure 8-2 shows a SIGMA/W setup of this case. The applied pressure is 100 kPa and the result is as shown in Figure 8-3. The pressure at the center of the sphere increases at first and dissipate back to the pre-load conditions. Along with this increase in effective stress (e.g., dissipation of excess pressure) comes shrinkage of the sphere as shown magnified 10x in Figure 8-2.

The response closely matches hand-calculated values from closed-form solutions as discussed in more detail in the included Illustrative Example named Cryers Ball. This confirms that the SIGMA/W coupled formulation and implementation correctly reflects the theory.

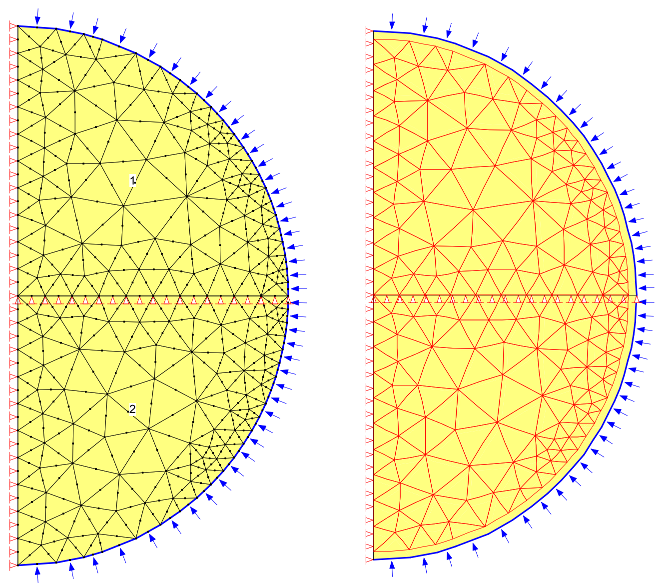


Figure 8-2 Mendel-Cryer sphere and deformed sphere (symmetric about left axis)

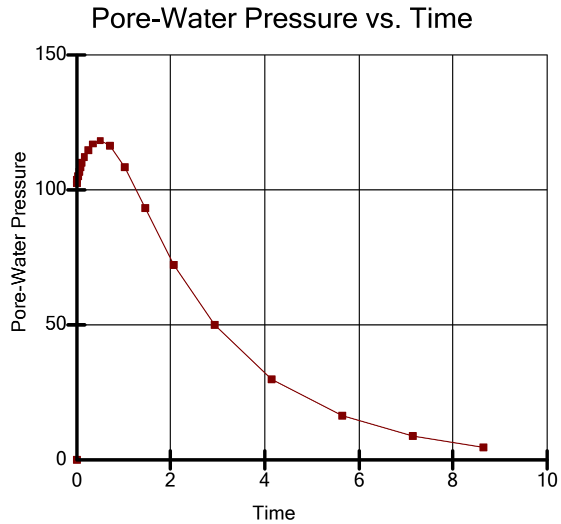


Figure 8-3 Mendel-Cryer effect: pressure increase before dissipation

8.13 Coupled analysis examples

Many of the Illustrative Examples described in a subsequent chapter and included with the software are about coupled analyses. Studying these examples together with the related documentation is good way to learn more about the details of this powerful feature in SIGMA/W.

9 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)

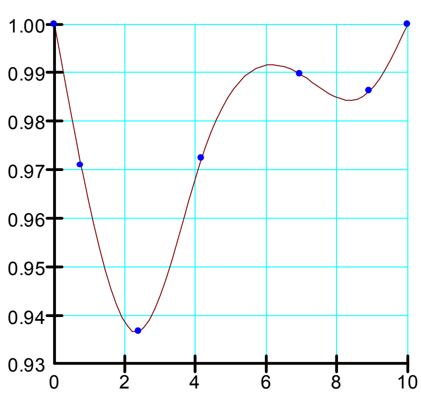
The type of function you choose to use will depend on what your needs are.

In many cases a required function 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.

9.1 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.



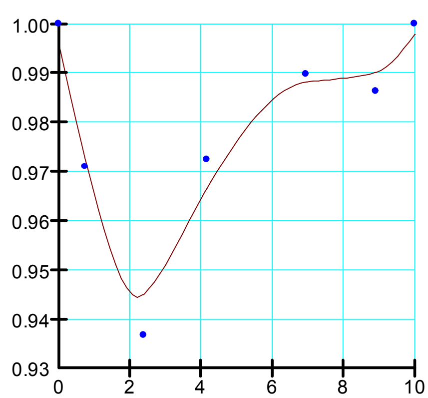


Figure 9-1 Spline functions with different settings

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.

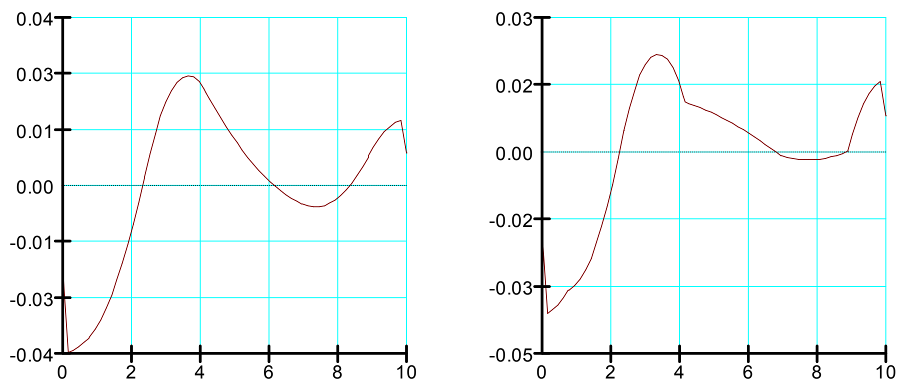


Figure 9-2 Slope of spline functions

9.2 *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.

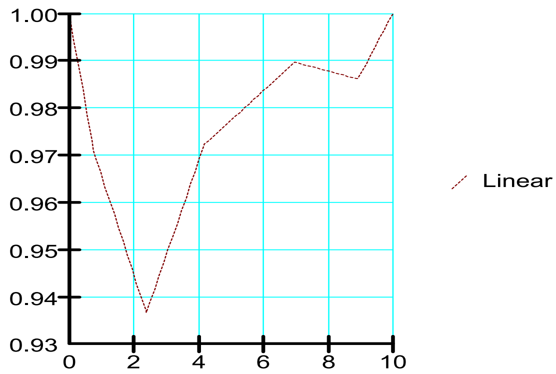


Figure 9-3 Linear fit spline

9.3 *Step functions*

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

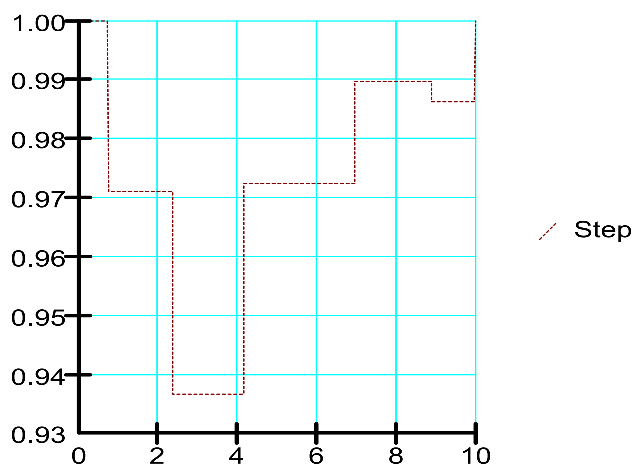


Figure 9-4 Step function

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.

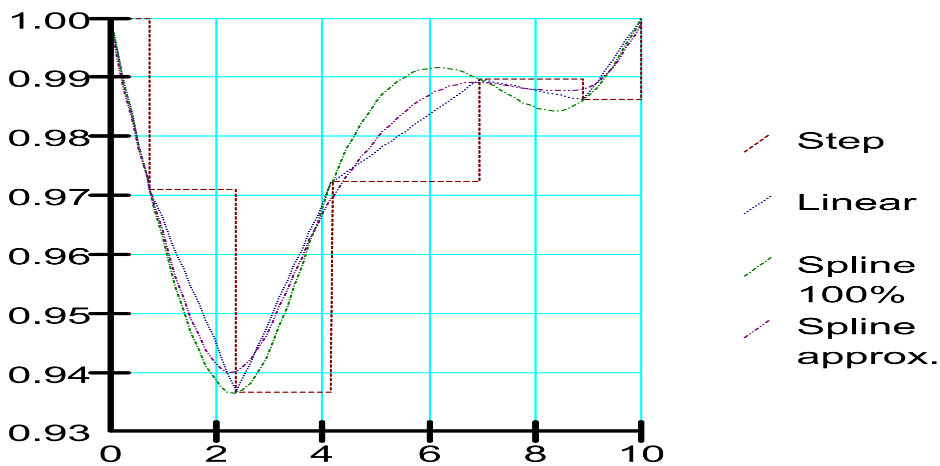


Figure 9-5 Comparison of all data point functions

9.4 Closed form curve fits for water content functions

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

9.5 Add-in functions

New to GeoStudio 2007 is the ability for the user to create add-in function. At any point in GeoStudio where you must enter a function, you have the option to bypass Geo-Slope default functions that relate "Y" to an "X" value in lieu of your own function that can relate "Y" to any number of parameters - either internal parameters passed to your function by the solver, or external parameters you obtain from another process.

Add in functions can be very simple or very complex. Consider the simple example shown below in which the returned "Y" value of the function is the sinusoid of the passed in "X" value.

```
// sample function takes the x value of the function type and returns the sin(x)

public class MySineCurve

{
	// this is the main function calculator
	public double Calculate( double x )
	{
		// declare the variable to return to the solver
		double y;
```

```
                // calculate the value to return
                y = Math.Sin(x);

                // return the function Y value
                return(y);
        }
}
```

The add-in function will actually be displayed in GeoStudio if it is based on available data. If data is unavailable until the solving time, no function will show. In the case of the above code, the displayed function is a sinusoid over a range of zero to 10 as seen in Figure 9-6.

By default, all add in functions are passed the pre-assigned "X" parameter based on the function type. For example, if your function is for water content, then the passed in X value will always be suction. It is up to you if you use this passed in X value or if you base the returned value on other data.

The possibilities for use of add in functions are endless. Because you can choose the computer language you write your function in (eg, C-sharp, .NET, Visual Basic, C++, Fortran) you can virtually ask your function to do anything you desire. It can simply return a "Y" value to the solver or it can run an entire other process on your computer, read or write a report based on current solving data, or even launch a second Geo-Studio analysis.

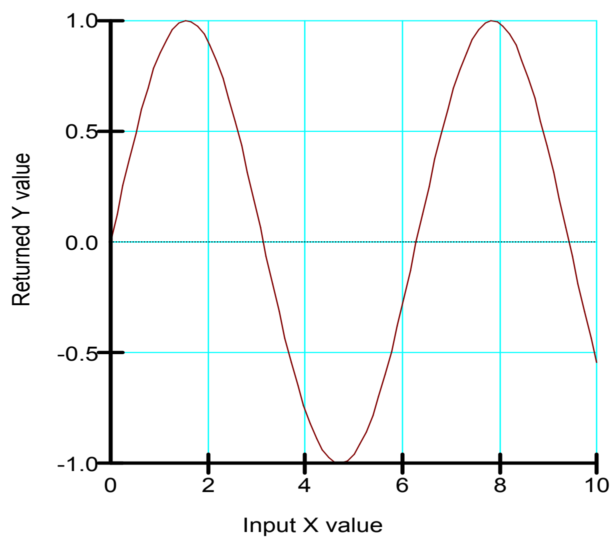


Figure 9-6 Simple user written function

Within your function you can ask the solver to pass you any live data it has in memory. So, for example, you could ask for the current solver time, the current air velocity, air content or density. Your function could keep track of data at the current and previous time step and you could account for hysteretic effects.

Your function is created and a copy is applied at every node or gauss point it is assigned to. This means it can "know" its position and you can create a function that returns a parameter based on some randomness or standard deviation from a normal value.

If you want to be very crafty, you can have your function read tabular data from a previously solved GeoStudio file so that, for example, you can base your ground strength in a SIGMA/W analysis on temperatures previously computed in a TEMP/W analysis.

You can have several functions inside the same library and a list of available functions and the required input will appear in GeoStudio dialogue boxes asking for user data.

As a final thought, once you create your add in function, you can compile it as a library file and pass it around to colleagues or sell it on e-Bay.

Again, the possibilities are unlimited.

For full details on Add In functions, please consult the Add In Developers Kit (SDK) available on the GEO-SLOPE website at www.geo-slope.com/downloads

9.6 Spatial functions

A spatial function in SIGMA/W can be used to establish starting pressure 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 pore-water pressures which would be applied in the seepage part of the analysis.

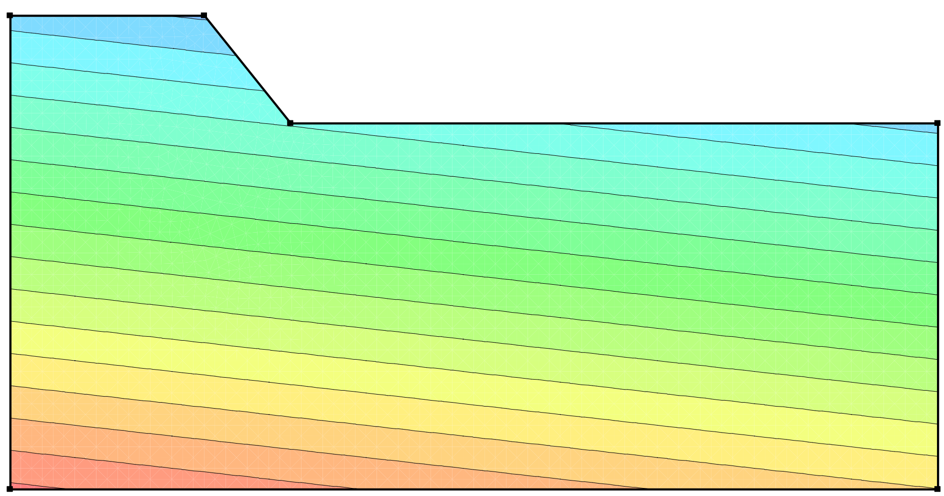


Figure 9-7 Example of spatial function assigned to model

10 Structural Elements

10.1 Introduction

This chapter presents the formulation used in SIGMA/W for structural elements. In SIGMA/W, a structural element can be either a bar element, which is capable of resisting axial force only, or a beam element, which is capable of resisting both bending and axial force. Structural elements can be used only in a two-dimensional plane strain load-deformation analysis.

10.2 Beam elements

In SIGMA/W, a beam element is formulated using the conventional (Bernoulli) beam theory (Hinton and Owen, 1979). This beam element requires the slope as well as the lateral displacement to be continuous within the element. Consequently, cubic Hermitian shape functions are used. Each node associated with a beam element is given a rotational degree-of-freedom in addition to the two displacement degrees-of-freedom.

Interpolating functions for a beam element

Figure 10-1 shows that for a one-dimensional element with two nodes at $r=-1$ and $r=1$, the lateral displacement w can be expressed in terms of the nodal lateral displacements, a, using the following:

$$w = N_1 a_1 + \bar{N}_1 \left(\frac{da}{dr}\right)_1 + N_2 a_2 + \bar{N}_2 \left(\frac{da}{dr}\right)_2$$

where:

$N_i, \bar{N}_i$ = the Hermitian interpolating functions evaluated at node i

The n-th derivatives in the local r and global x'-coordinate systems are related by:

Equation 10-1 $$\left(\frac{d^n w}{dr^n}\right) = \left(\frac{l}{2}\right)^n \left(\frac{d^n w}{dx'^n}\right)$$

where:

l = the length of the beam element.

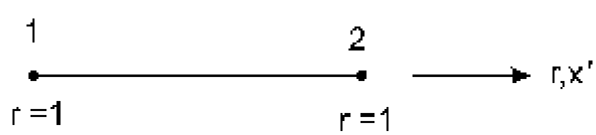


Figure 10-1 A 2-noded beam element

The Hermitian interpolation functions are tabulated as follows:

Table 10-1 Interpolating functions for beams

Interpolating Function
$N_1 = (2+r)(1-r)^2/4$
$\overline{N}_1 = (1+r)(1-r)^2/4$
$N_2 = (2-r)(1+r)^2/4$
$\overline{N}_2 = -(1-r)(1+r)^2/4$

Stiffness matrix for a beam element

For a beam of flexural rigidity EI and length l, the strain energy Π_b due to bending is given by:

$$\Pi_b = \int_0^l \frac{EI}{2}\left(\frac{d^2 w}{dx'^2}\right)dx'$$

where:

w = lateral displacement of the beam

x' = distance along the beam

A beam element in SIGMA/W is applied to a line object which can be concurrent with either 3-noded or 2-noded element edges. A 3-noded beam element is treated as a combination of two 2-noded elements. A 2-noded element is formulated using cubic Hermitian interpolation functions. For this element the lateral displacement can be expressed in terms of the Hermitian interpolating functions as follows:

$$w^e = N_1 w_1^e + \overline{N}_1 \left(\frac{dw}{dr}\right)_1^e + N_2 w_2^e + \overline{N}_2 \left(\frac{dw}{dr}\right)_2^e$$

The superscript e denotes that the variable is evaluated within an element.

Substituting in Equation 10-1, the previous equation can be written as:

$$w^e = N_1 w_1^e + \overline{N}_1 \frac{l^e}{2}\left(\frac{dw}{dx'}\right)_1^e + N_2 w_2^e + \overline{N}_2 \frac{l^e}{2}\left(\frac{dw}{dx'}\right)_2^e$$

where:

l^e = length of the beam element.

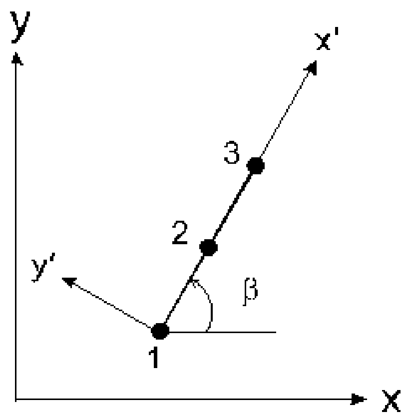


Figure 10-2 Local and global coordinates for a structural element

Again using Equation 10-1 and neglecting axial displacements, the curvature at any point within the beam element can be expressed as:

$$\left(\frac{d^2 w}{dx'^2}\right)^e = \frac{4}{l^{e2}}\left\langle 0 \quad \frac{d^2 N_1}{dr^2} \quad \frac{d^2 \overline{N}_1}{dr^2}\frac{l^e}{2} \quad 0 \quad \frac{d^2 N_2}{dr^2} \quad \frac{d^2 \overline{N}_2}{dr^2}\frac{l^e}{2}\right\rangle \begin{Bmatrix} u_1^e \\ w_1^e \\ (dw/dx')_1^e \\ u_2^e \\ w_2^e \\ (dw/dx')_2^e \end{Bmatrix}$$

$$= [B]\{a\}$$

where:

[B] = the strain matrix for the beam element,

{a} = vector of nodal field variables consisting of nodal displacements *u*, *w* and nodal rotation *dw/dx,*

u = nodal axial displacement, and

x = local coordinate $(-1 \le r \le 1)$.

The strain energy due to bending in a beam element now can be written as:

$$\Pi_b^e = \int_0^{l^e} \{a\}^T [B]^T C [B]\{a\} dx = \int_{-1}^{1} \{a\}^T [B]^T C [B]\{a\} \frac{l^e}{2} dr$$

where:

C = EI, the flexural rigidity of the beam.

The stiffness matrix for the beam element $[K]^e$ is given by:

$$[K]^e = \frac{l^e}{2}\int_{-1}^{1}[B]^T C[B]dr$$

The formulation is for a coordinate system (x', y') with the x'-axis coinciding with the longitudinal axis of the beam. This integral is evaluated numerically in SIGMA/W. Before the stiffness matrix can be assembled into the global finite element equation , it needs to be rotated into the global Cartesian coordinate system through the following transformation:

$$[K]^e_g = [T]^T [K]^e [T]$$

For a beam element inclining at angle β to the global x-axis, the transformation matrix is:

$$[T] = \begin{bmatrix} \cos\beta & \sin\beta & 0 & 0 & 0 & 0 \\ -\sin\beta & \cos\beta & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos\beta & \sin\beta & 0 \\ 0 & 0 & 0 & -\sin\beta & \cos\beta & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

In order to define a beam in SIGMA/W it is necessary to specify the E modulus, the cross-sectional area, the moment of inertia and the activation step. If the moment of inertia is set to zero, a beam will behave as a bar element with axial forces only.

10.3 Beam examples

The Illustrative Examples Chapter includes several examples on the verification and use of beam elements. The illustration in Figure 10-3 is one of them.

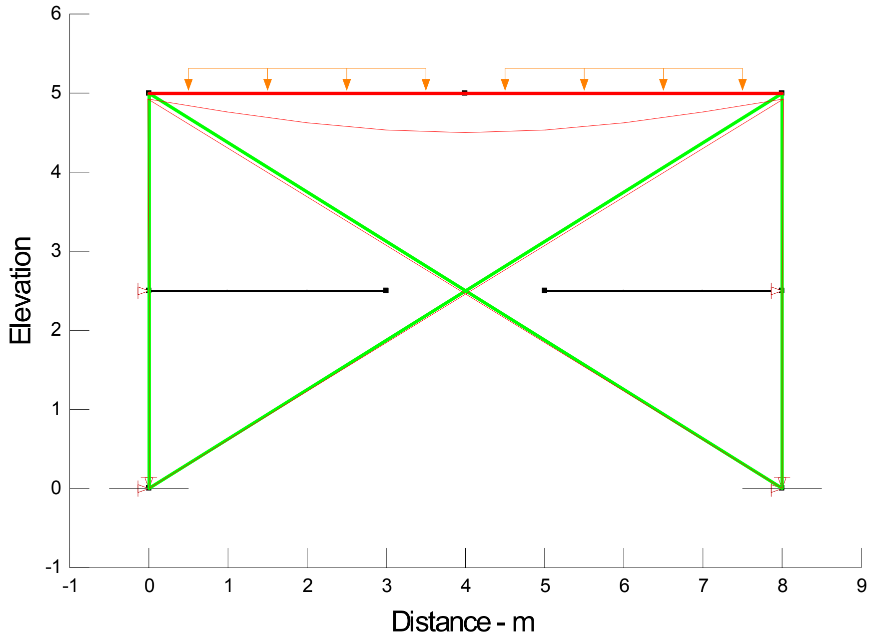


Figure 10-3 Example of beams and bars in SIGMA/W

10.4 Bar elements

In SIGMA/W, a structural element defined with a zero flexural rigidity (*EI*) is considered to be a bar element. A bar element is capable of resisting axial forces only. Thus, for the nodes of a bar element, rotational degree-of-freedom is not required. A bar element is also drawn along a line object but does not need to be concurrent with a soil element edge. It can cross over elements and only have connectivity at region points.

Interpolating functions for a bar element

Interpolating functions similar to those used for two-dimensional solid elements are used in formulating a bar element. The interpolating functions in terms of local coordinate $r \ (-1 \le r \le 1)$ are listed in the following table.

Table 10-2 Interpolating functions for bars

Function	Include in function if node 3 is present
$N_1 = \frac{1}{2}(1-r)$	$-\frac{1}{2}(1-r^2)$
$N_2 = \frac{1}{2}(1+r)$	$-\frac{1}{2}(1-r^2)$
$N_3 = (1-r^2)$	—

Stiffness matrix for a bar element

For a bar of axial rigidity *EA* and length *l*, the strain energy Π_a due to axial deformation is given by:

$$\Pi_a = \int_0^l \frac{EA}{2}\left(\frac{du}{dx'}\right)^2 dx'$$

where:

u	=	axial displacement along the beam, and
x'	=	distance along the beam.

In a bar element, axial displacements can be expressed in terms of the nodal displacement using the interpolating functions:

$$u^e = \sum_{i=1}^{n} N_i u_i^e$$

where:

n	=	2 or 3, the number of nodes used in the bar element.

The superscript e denotes that the variable is evaluated within an element.

The derivative e can also be expressed in terms of the derivatives of the interpolating functions as follows:

$$\begin{aligned}
\left(\frac{du}{dx'}\right)^e &= \sum_{i=1}^{n} \frac{dN_i}{dx'} u_i^e \\
&= \left\rangle \frac{dN_1}{dx'} \quad \frac{dN_2}{dx'} \right\langle \begin{Bmatrix} u_1^e \\ u_2^e \end{Bmatrix} \quad \text{, for a 2-noded element} \\
&= \left\rangle \frac{dN_1}{dx'} \quad \frac{dN_2}{dx'} \quad \frac{dN_3}{dx'} \right\langle \begin{Bmatrix} u_1^e \\ u_2^e \\ u_3^e \end{Bmatrix} \quad \text{, for a 3-noded element}
\end{aligned}$$

where:

{n}	=	vector of nodal axial displacements, and
n	=	2 or 3, the number of nodes used in the bar element.

The derivative of the interpolating function with respect to x' at node *i* can be evaluated using:

$$\left(\frac{dN_i}{dx'}\right)^e = \left(\frac{dr}{dx'}\right)\left(\frac{dN_i}{dr}\right) = J^{-1}\left(\frac{dN_i}{dr}\right)$$

where:

r	=	local coordinate $(-1 \le r \le 1)$
J	=	Jacobian operator = dr/dx'

After substituting, the expression for the derivative du/dx' becomes:

$$\left(\frac{du}{dx'}\right)^e = J^{-1} \left\rangle \frac{dN_1}{dr} \quad \frac{dN_1}{dr} \right\langle \begin{Bmatrix} u_1^e \\ u_2^e \end{Bmatrix} \quad \text{, for a 2-noded element}$$

$$= J^{-1} \left\rangle \frac{dN_1}{dr} \quad \frac{dN_2}{dr} \quad \frac{dN_3}{dr} \right\langle \begin{Bmatrix} u_1^e \\ u_2^e \\ u_3^e \end{Bmatrix} \quad \text{, for a 3-noded element}$$

$$= J^{-1} \left[B\right]\left\{a\right\}$$

where:

$[B]$	=	strain matrix for a bar element
$\{a\}$	=	vector of nodal axial displacements

The following expression for the strain energy due to axial deformation for a bar element is obtained:

$$\Pi_a^e = \int_0^{l^e} \left\{a\right\}^T \left[B\right]^T C\left[B\right]\left\{a\right\} dx = \int_{-1}^{1} \left\{a\right\}^T \left[B\right]^T C\left[B\right]\left\{a\right\} J^{-1} dr$$

where:

C	=	*EA*, the axial stiffness of the bar element.

The stiffness matrix for the bar element $\left[K\right]^e$ is given by:

$$\left[K\right]^e = \int_{-1}^{1} \left[B\right]^T C\left[B\right] J^{-1} dr$$

The formulation is for a coordinate system $\left(x', y'\right)$ with *x'* coinciding with the longitudinal axis of the bar. This integral is evaluated numerically in SIGMA/W. Before this stiffness matrix can be assembled into the global finite element equation, it needs to be rotated into the global Cartesian coordinate system in a procedure similar to that for a beam element.

Bar elements can be set to become active at different load steps. If the active step is defined as zero, the bar element is considered to be an active element from step 1. If the active step is nonzero, the bar element is considered to be an active element from that active step.

Enter tension axial force in a bar element as a negative value and compression force as a positive value. The reaction forces at the two end nodes will be shown as inward arrows for tension axial force and outward arrows for compression axial force those are the nodal forces that will be actually applied on the rest of the mesh at the step defined. A pre-axial force can be applied before activating a bar element.

10.5 Bar example

Figure 10-4 shows an example of how bar elements can be used to analyze a truss or frame. The details of this example are presented in the Illustrative Examples Chapter.

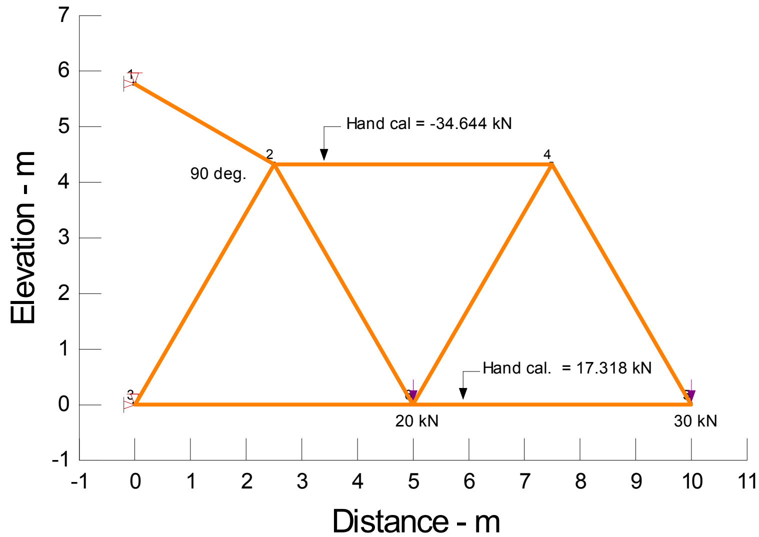


Figure 10-4 Bar elements is a truss

11 Staged Construction Sequencing

SIGMA/W can be used to simulate construction sequences or stages. In finite element terminology this means adding or removing elements from the mesh. In SIGMA/W, the terminology is activating or deactivating regions. The mesh remains the same for all analyses but regions can activate or deactivate to simulate, for example, the placement of fill or the removal of soil to create an excavation.

This chapter gives an overview of the steps involved in simulating, for example, the construction of an embankment by placing the fill in lift or the removal of the ground to create an excavation.

11.1 Basic concepts

SIGMA/W can string together a series of analyses. All the analyses can be sequential by assigning each analysis a duration in time to form a continuous time line. The start time of an analysis is the ending time of the previous analysis.

Regions are activated or deactivated for each analysis, by assigning a material to a region or removing the material from a region.

Boundary conditions are unique to each analysis.

Structural elements are added by assigning beam or bar properties to a line or are removed by taking away the beam and bar properties at any particular stage.

It is useful at all times to be mindful of the fact that the mesh and geometry is the same for all analyses, but the properties and boundary conditions are unique to the analysis.

11.2 Fill placement example

Consider the case of constructing an embankment by placing the fill in a series of lifts. The following diagram (Figure 11-1) shows a typical case. The fill will be placed in eight lifts. Plus the intention is to start with a known state of stress in the foundation before placing any fill. So a total of nine analyses are required as shown by the tree view in Figure 11-2.

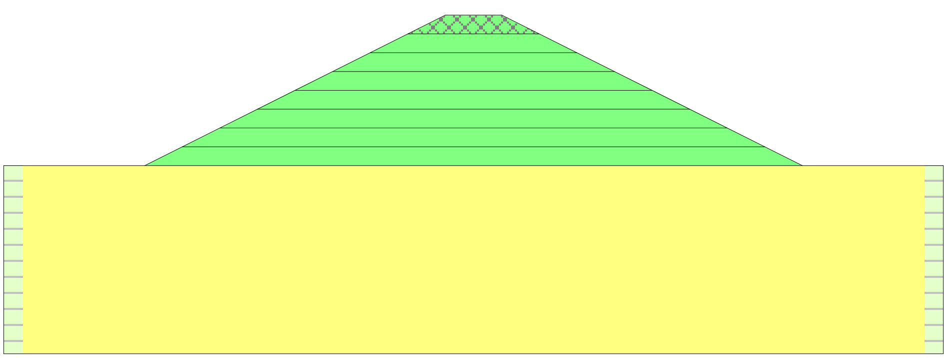


Figure 11-1 Illustration of fill placement by lifts

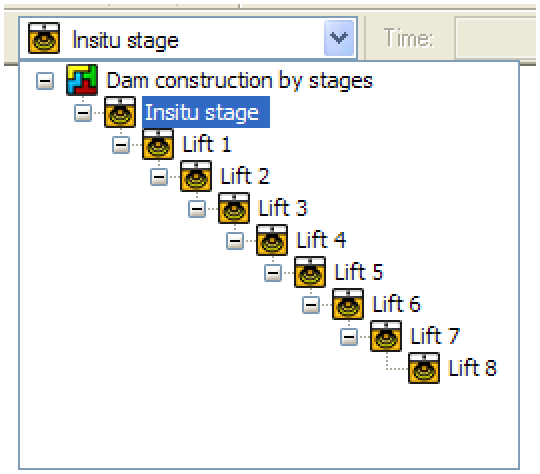


Figure 11-2 Analysis tree view

The first analysis or stage is to establish the insitu stress state. This is followed by eight analyses where each Lift-analysis represents the placement of one metre of fill.

This is a total stress analysis so no "real" time is involved as far the computations are concerned but each analysis is given one unit of time so that the results can be plot across all the analyses.

In SIGMA/W terminology, the "Parent" of Lift 1 is the Insitu stage or conversely the "Child" of the Insitu stage is Lift 1. The algorithmic significance of this is that the "Child" can always get or inherit its initial conditions from the "Parent". So in the context of the SIGMA/W incremental formulation, each analysis is an increment and the increments get added to the initial conditions.

The following three figures show the configuration for the first three analyses.

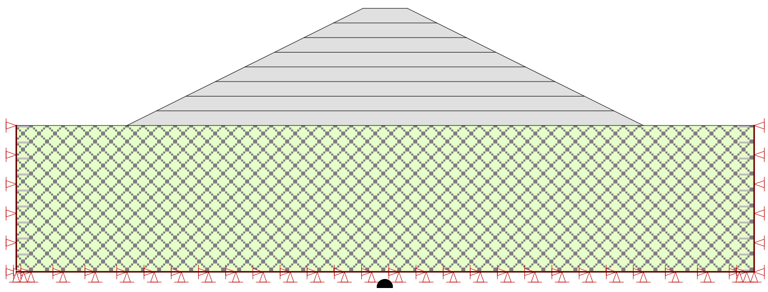


Figure 11-3 Configuration for the Insitu stage

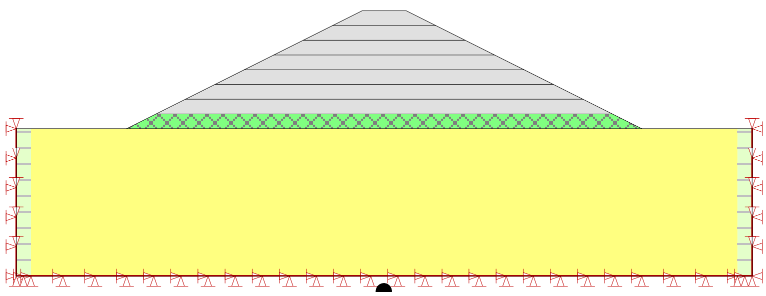


Figure 11-4 Configuration for Lift 1

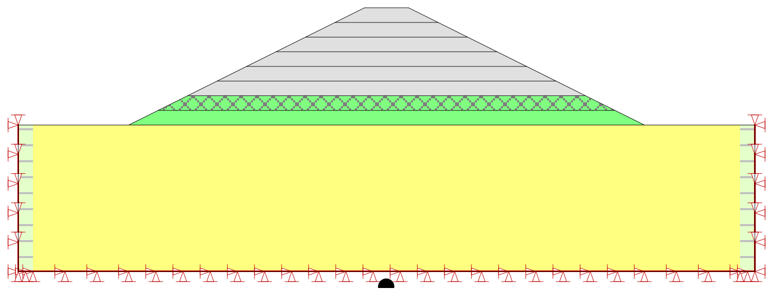


Figure 11-5 Configuration for Lift 2

Note the cross-hatching of the foundation for the Insitu stage. This signifies that a body load (self weight) is used in this analysis to compute the state of stress in the ground. For Lift 1, the cross hatching is missing from the foundation and shows up in the first fill placement lift. For Lift 2, the body load shifts to the second layer of fill. Figure 11-1 shows the 8th and last lift at the top. Generally, the body load is always applied to a region the first time the region becomes active.

Another important point here is that each lift needs to be a separate geometric region.

By assigning each analysis a time duration and making the time line continuous, the results can be view simultaneously for multiple analyses as illustrated in Figure 11-6.

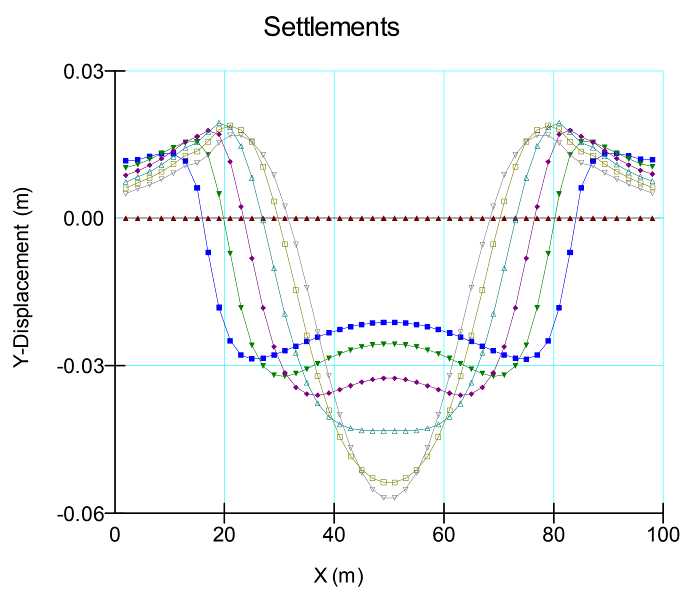


Figure 11-6 Settlement profiles along original ground surface

SIGMA/W can optionally compensate for the settlement caused by self compression of newly added fill elements. For each lift, SIGMA/W adds the current displacement to the element nodal coordinates to compute the element weight and volume, which in turn are used to compute the gravitational nodal forces to simulate the fill placement. Adding the displacements gives the element additional mass which represents the additional fill placed to compensate for settlement during construction. Fundamentally, this infers that sufficient fill will be place to compensate for any settlement during construction and the intent is to build the embankment to a specified design elevation.

The “Adjust fill elevation with foundation settlement” option must be used cautiously. It should only be used as a possible refinement after realistic results have been obtained without using this option.

Further details on this example are presented in The Illustrative Examples Chapter.

11.3 Excavation

The procedure to simulate an excavating process is very similar to fill placement except that regions are deactivated by removing the material from the region. Figure 11-7 and Figure 11-8 illustrate the excavation of layers.

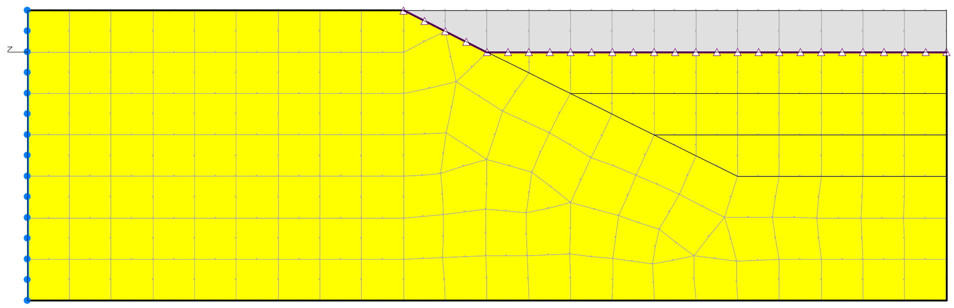


Figure 11-7 Excavation of first layer

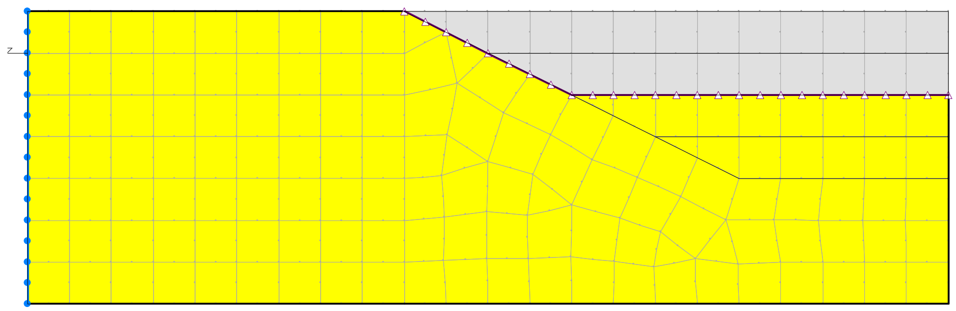


Figure 11-8 Excavation of second layer

This example is about excavating below the original water table and so the hydraulic boundary conditions must change as the excavation proceeds. Note how the hydraulic boundary condition follows the excavation face in the above two diagrams. This further illustrates that the boundary conditions can be different for each analysis.

As already noted above, excavation regions work in a similar way to fill regions in that they can be turned off at various load steps to simulate the process of digging an excavation from the top down (or in the case of a tunnel, from the center to the liner). When a region is removed, a force that pulls outward on the remaining soil must be applied. Unlike the fill region where the forces are known by the weight of the soil being added, an excavation region uses the existing total stresses at the element edges adjacent to the elements being removed. In this way, the actual total stresses acting in the x- and y-directions can be used to arrive at a nodal force load for the element edge nodes.

NOTE: An initial stress file is required when performing an excavation analysis. Without any prior stress conditions, it would not be possible to compute the unloading forces caused by removal of stress. In some cases where excavations are adjacent to structural components, it is better to manually specify the excavation forces using a boundary condition instead of element stresses. This is discussed in the Berlin Wall example.

11.4 Braced excavations

Dealing with structural elements is similar to placing fill or digging an excavation. Structural elements can be added or removed for any analysis. Figure 11-9 shows the left half about the symmetric axis of a braced excavation.

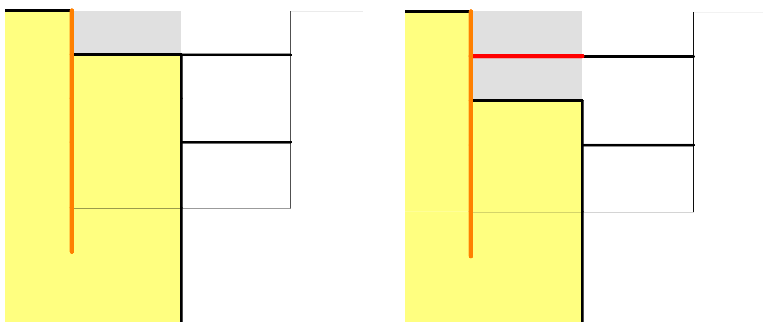


Figure 11-9 Two stages in the construction of a braced excavation

The diagram on the left shows the stage with the sheet pile wall in place and the removal of the first dig. The diagram on the right shows the first upper brace in place and the removal of the second dig.

11.5 Closing remarks

The ability to so conveniently simulate sequential construction stages is a very powerful SIGMA/W feature. Some careful planning is however advisable. Remember that the geometry and mesh must be the same for all analyses. Boundary conditions, material properties and structural components can change for each analysis.

Most of the examples presented in the Illustrative Examples Chapter have multiple analyses. Further details about the possibilities and procedures can be discovered by studying the examples.

12 Numerical Issues

This chapter discusses several numerical issues related to solving the partial differential equation for stress-deformation. It deals with non-linearity of soil constitutive models and how they are dealt with. The chapter also discusses some features in the SIGMA/W program that can help you assess convergence of your solution such as graphing convergence data during the solve process or stopping and restarting an analysis in order to make adjustments without having to re-start an entire solve process.

12.1 Nonlinear analysis

SIGMA/W is capable of performing analysis involving nonlinear material properties. The finite element equilibrium is derived for a linear static case. For nonlinear material properties, the global stiff matrix, [K], is no longer linear and iterations are required to achieve an acceptable solution. Additional information on the procedure for nonlinear analysis can be found in finite element text-books such as Bathe (1982), and Zienkiewicz and Taylor (1991). This section details some general aspects of nonlinear incremental analyses relating to convergence criteria and using SIGMA/W to estimate limiting loads.

In general, the basic problem in a nonlinear analysis is to find the state of equilibrium of a body corresponding to the externally applied loads. If these loads are a function of time, at time *t* the nonlinear equilibrium problem can be formulated as a solution of the following equation:

Equation 12-1 $$\left\{ {}^{t}\Psi \right\} = \left\{ {}^{t}R\left(a\right) \right\} - \left\{ {}^{t}F \right\} = 0$$

where:

Ψ	=	nodal unbalanced loads,
F	=	externally applied nodal loads,
R	=	nodal loads due to element stresses, and,
a	=	nodal displacements.

It is assumed that the solution will start from an equilibrium (or, at least, near equilibrium) state such that (Ψ^{t-1}) is approximately zero. The vector $\left\{R\right\} = \int\left[B\right]\left\{\sigma\right\}dv$ is calculated based on the internal element stresses as described previously in this chapter. In SIGMA/W, external applied loads are described using continuous spline functions, which describe the total load being applied on the system versus time. Therefore, for a particular time increment t, $\{{}^{t}F\}$ is an incremental load obtained by the difference between the value of the load functions evaluated at time t and at time t=1.

SIGMA/W solves Equation 12-1 iteratively using the Newton Raphson technique which approximates this equation as:

Equation 12-2 $$\left\{\Psi^{i+1}\right\} = \left\{\Psi^{i}\right\} + \left[\frac{\partial \Psi^{i}}{\partial a}\right]\left\{\delta a^{i}\right\}$$

where i is the iteration counter and the superscript t has been omitted for clarity. To start the iteration process for time step t, displacements at the end of the previous time step (i. e. , time t-1) are used as the initial estimate:

$${}^{t}a^{1} = {}^{t-1}a$$

If the derivative term in Equation 12-2 is written as:

$$\left[\frac{\partial \Psi}{\partial a}\right] = \left[\frac{\partial R}{\partial a}\right] = \left[K_T\right]$$

where:

$[K_T]$ = the tangential stiffness matrix.

The iterative correction applied to the nodal displacements can be calculated as:

$$\left[K_T{}^i\right]\left\{\delta a^i\right\} = -\{\Psi^i\}$$

or:

$$\left\{\delta a^i\right\} = -\left[K_T^i\right]^{-1}\left\{\Psi^i\right\}$$

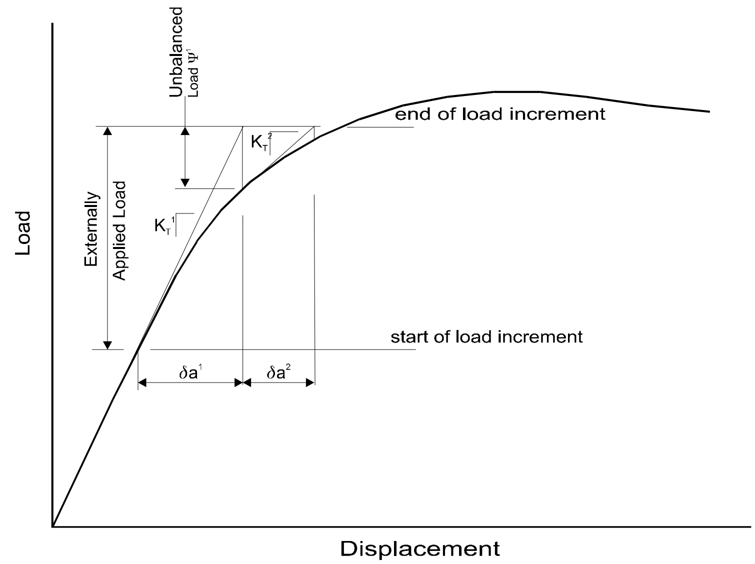
A series of successive iterations gives the following result:

$$\left\{{}^t a^{i+1}\right\} = \left\{{}^{t-1}a\right\} + \sum_{k=1}^{i}\left\{\delta a^i\right\}$$

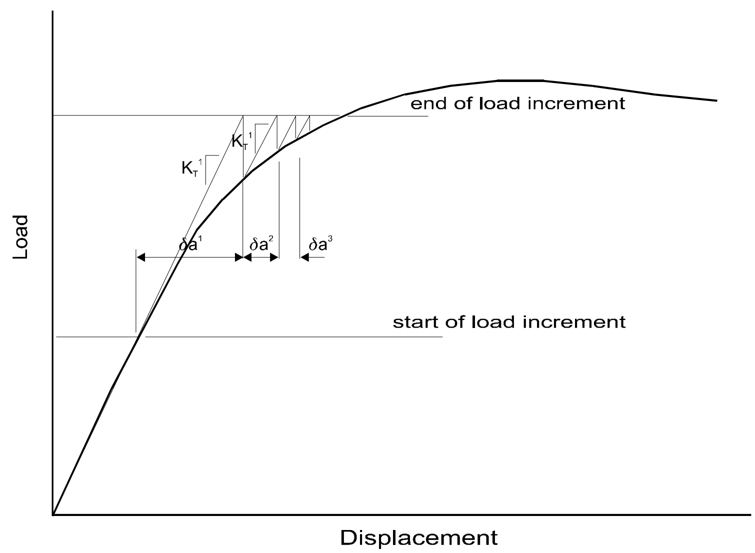
In the Newton Raphson method, the stiffness matrix is tangential to the load-displacement curve and is updated every iteration. This process is illustrated in Figure 12-1(a). For an analysis involving strain-softening materials, this process is modified such that the stiffness matrix is not updated, but the stiffness matrix used in the first iteration is retained throughout the time step, as illustrated in Figure 12-1(b). In order to achieve stability in SIGMA/W, the initial modulus is always calculated based on elastic behavior.

It should be noted that the load term is the externally applied load for the first iteration in a load step; and it is the unbalanced load for subsequent iterations.

Incremental displacements resulting from increment load $\{{}^tF\}$ at time t are obtained upon solving Equation 12-1. These incremental displacements are added to the displacements at the beginning of the time increment. In the displacement output files, SIGMA/W reports the total displacements at each node since the beginning of an analysis at Time Step 1.



a) Iteration Scheme Using Tangential Modulus



b) Iteration Scheme Using Constant (Initial) Modulus

Figure 12-1 Iteration schemes for nonlinear analysis

12.2 Convergence

In all non-linear analyses, it is necessary to use iterative techniques to compute acceptable solutions. Non-linear analyses exist when a soil property is dependent on the computed results. For example, the soil stiffness modulus E is dependent on the stress state in the ground, but the stresses are dependent on the soil stiffness. This means that the analysis has to be done many times until there is a reasonable match between the soil properties and the computed stresses. When the analysis has produced an acceptable match, the solution is deemed to have converged. As the user, you must decide what constitutes convergence.

SIGMA/W, like all the other GeoStudio finite element products, uses the comparison of the primary unknowns from two successive iterations to control the iterative convergence procedure. In SIGMA/W the primary unknown is the computed nodal displacement.

The Non-Linear formulation is an incremental displacement formulation. This means that each time-step or load-step computes an incremental displacement and the final displacement is the algebraic summation of all the incremental displacements. So if there are 50 time steps there will be 50 incremental displacements and the final displacement will be the summation of the 50 increments.

With this formulation, SIGMA/W compares the displacements at each node from two successive iterations. A solution is deemed to have converged when the displacements from two successive iterations are within the specified tolerance.

From a mathematically perspective, this means making a comparison of two floating numbers. GeoStudio uses two criteria to make this comparison. They are:

- Significant figures or digits
- 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 unknowns from the finite element solution. Specifying a criterion of two significant digits means that when the displacements 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 carry with them some numerical noise, or stated another way, digits that have no physical meaning. So when comparing floating point numbers it is necessary to filter out the meaningless digits.

GeoStudio does this with a user specified minimum-difference value. If the difference between two successive displacements at a node is less than this minimum specified value, the two displacements are deemed to be the same, or again the two displacements are said to have converged.

In SIGMA/W, a minimum difference specified as 0.001 means a minimum difference in displacement of 0.001 m or 1 mm; a value set to 0.005 means a displacement difference of 0.005 m or 5 mm and so forth.

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 very small and has no physical meaning in terms of a SIGMA/W analysis. The two numbers are consequently deemed to be the same or converged.

Maximum number of iterations

In any iterative scheme it is necessary to ultimately limit the number of iterations if it is not possible to meet the convergence criteria. The maximum number of iterations is a user-specified value.

Viewing convergence data

For each time step that the results are saved, it is possible to make a graph of the number of unconverged nodes versus the iteration count. The objective is to go through the iterative process until all nodes have converged, or until there are no unconverged nodes. In Figure 12-2, nine iterations were required for all nodes to meet the specified convergence criteria.

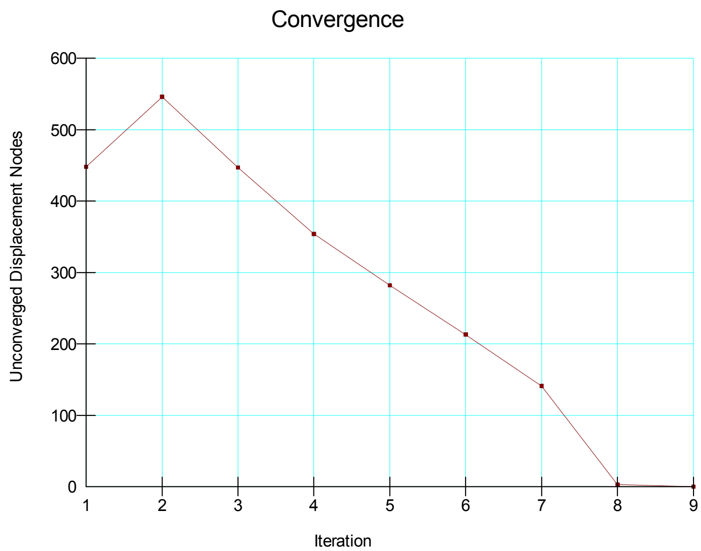


Figure 12-2 Unconverged nodes versus iteration count

When the displacement convergence criterion is satisfied, the analysis is considered to have converged (or achieved convergence) and SIGMA/W will stop the iteration process and go to the next time (load) step.

12.3 Limiting loads – use incremental loading

The SIGMA/W finite element equations are a set of equilibrium equations. The driving (applied) forces are in equilibrium with the resisting forces that arise from the soil strength. The load that can be applied is therefore limited by the ultimate resistance of the system. In the terminology of SLOPE/W (another GEO-SLOPE product), the limiting load is the load that results in a factor of safety equal to 1.0.

The modeling consequence of applying loads in excess of the limiting load is usually numerical instability; that is, a lack of convergence and widely varying oscillation in the solution. Unfortunately, the limiting load is not always known. Lack of convergence after a series of successive load steps has been applied may be a hint that you have reached the limiting load.

In certain cases, it may be of interest to carry the analysis very close to the point of limiting equilibrium. A technique that can be used is to specify the displacement instead of the applied load. Specifying displacement instead of force is likely mandatory for the strain-softening model if you wish to carry on an analysis until the ultimate strength of your material is mobilized. When the applied load approaches the ultimate value, you may experience difficulties with convergence unless you use “strain-controlled” loading.

It is also important to remember that SIGMA/W is not formulated for large displacements or large strains. Therefore, carrying an analysis well past the point of limiting equilibrium may produce unrealistic results.

12.4 Tension zones

The effect of tension zones is handled indirectly in SIGMA/W. Consider the case of a steep slope with tension in the crest area. If the material has tensile strength, energy will be stored in the system, preventing the soil from deforming to the point where cracks develop. However, if the soil has no tensile capacity, cracks will form and the equivalent stored energy will contribute to additional deformation. To prevent the soil from storing energy representative of the tensile strength, the soil can be made very soft by assigning it a low *E* modulus.

Generally, the effects of tension can be reduced by using an E-modulus function where E becomes small as the stress level becomes fairly low. It is necessary however to not make E too small since this can create convergence difficulties. A converged solution with less than theoretically ideal material properties is better than a poorly converged solution with ideal properties.

12.5 Equation solvers (direct or parallel direct)

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

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

Select the parallel direct equation solver option if you have a larger mesh. The parallel solver will save the matrices in a compressed format to eliminate all zero values 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.

13 Visualization of Results

13.1 Introduction

An attractive feature of SIGMA/W is the varied yet flexible ways that the results that can be viewed, evaluated and visualized. This is important for any finite element analysis due to the large amount of data involved. Much of the data can only be interpreted through graphical visualization using contours and graphs. And yet at the same time it is often necessary to look at details by examining the digital data at a particular node or at a Gauss integration point in an element.

This chapter describes and summaries the capabilities available in SIGMA/W for viewing, evaluating and visualizing the results.

13.2 Load steps

Some SIGMA/W analyses such as in-situ or single step un-coupled consolidation are solved in one load step. Others, however, such as sequential fill placement, require many small loading steps. The Contour program is capable of letting you view data from individual load steps or multiple load steps. If you have a single load step data set active, then you can visualize data such as the deformed mesh or contours of pressures, stresses etc. If you have multiple load step data active in Contour, then you can not view a contour of pressures, but you can then graph how any parameters change over the loading sequence. In either case, the data you are viewing is the actual condition in the soil at the time (or load step) you view it. Unlike the solver, which computes incremental changes between each load step, the Contour program presents the end result of adding all the incremental changes up. The option to view only incremental differences is discussed next.

13.3 Types of data available in Contour

All computed parameters except displacement, boundary force, and pore-water pressure are computed as values at element Gauss points. The Gauss values are then projected from the Gauss points to the nodes in order to obtain nodal values for viewing and graphing. A discussion on the implications of projecting Gauss point data to nodes is provided in the following section. Keep in mind that if nodal values appear inconsistent, you can always view the actual computed Gauss point data to verify any concerns.

Some of the computed parameters are obtained directly from the problem data files, while others are calculated from the values in the output files. If you used an Add In function or constitutive model, it now can make use of a feature that will write your own data to the results files for later graphing or contouring.

The following three tables show all the types of post processor computed stress and strain data available for visualization as well as how they are computed. Parameters computed directly by the solver are not included in these tables, even though they are accessible for visualization.

Table 13-1 Force and displacement parameters computed in Contour

Parameter	Equation
XY-Displacement (δ_{xy})	$\delta_{xy} = \sqrt{\delta_x^2 + \delta_y^2}$
XY-Force (F_{xy})	$F_{xy} = \sqrt{F_x^2 + F_y^2}$

Table 13-2 Strain parameters computed in Contour

Parameter	Equation
Maximum Strain $(\varepsilon_{\max})$	$\varepsilon_{\max} = \frac{(\varepsilon_y + \varepsilon_x)}{2} + \sqrt{\left[\frac{(\varepsilon_y - \varepsilon_x)}{2}\right]^2 + \left(\frac{\gamma_{xy}}{2}\right)^2}$
Minimum Strain $(\gamma_{\min})$	$\varepsilon_{\min} = \frac{(\varepsilon_y + \varepsilon_x)}{2} - \sqrt{\left[\frac{(\varepsilon_y - \varepsilon_x)}{2}\right]^2 + \left(\frac{\gamma_{xy}}{2}\right)^2}$
Maximum Shear Strain $(\gamma_{\max})$	$\gamma_{\max} = \sqrt{\left[\frac{(\varepsilon_y - \varepsilon_x)}{2}\right]^2 + \left(\frac{\gamma_{xy}}{2}\right)^2}$
Volumetric Strain (ε_v)	$\varepsilon_v = \varepsilon_x + \varepsilon_y + \varepsilon_z$
Deviatoric Strain (q_ε)	$q_\varepsilon = \frac{1}{\sqrt{2}}\sqrt{\begin{matrix}(\varepsilon_x - \varepsilon_y)^2 + (\varepsilon_y - \varepsilon_z)^2 + (\varepsilon_z - \varepsilon_x)^2 \\ + \frac{3}{2}\tau_{xy}^2\end{matrix}}$

Table 13-3 Stress parameters computed in Contour

Parameter	Equation
XY-Displacement $\left(\delta_{xy}\right)$	$\delta_{xy} = \sqrt{\delta_x^2 + \delta_y^2}$
XY-Force $\left(F_{xy}\right)$	$F_{xy} = \sqrt{F_x^2 + F_y^2}$
Maximum Total Stress $\left(\sigma_{\max}\right)$	$\sigma_{\max} = \dfrac{\left(\sigma_y + \sigma_x\right)}{2} + \sqrt{\left[\dfrac{\left(\sigma_y - \sigma_x\right)}{2}\right]^2 + \tau_{xy}^2}$
Minimum Total Stress $\left(\sigma_{\min}\right)$	$\sigma_{\min} = \dfrac{\left(\sigma_y + \sigma_x\right)}{2} - \sqrt{\left[\dfrac{\left(\sigma_y - \sigma_x\right)}{2}\right]^2 + \tau_{xy}^2}$
Mean Total Stress (*p*)	$p = \dfrac{\left(\sigma_x + \sigma_y + \sigma_z\right)}{3}$
Normal Total Stress $\left(\sigma_n\right)$	$\sigma_n = \dfrac{\sigma_x + \sigma_y}{2} + \dfrac{(\sigma_x - \sigma_y)}{2}\cos 2\theta + \tau_{xy}\sin 2\theta$
Tangential Total Stress $\left(\sigma_t\right)$	$\sigma_t = \dfrac{(\sigma_x - \sigma_y)}{2}\sin 2\theta - \tau_{xy}\cos 2\theta$
X-Effective Stress $\left(\sigma'_t\right)$	$\sigma_x' = \left(\sigma_x - u\right)$
Y-Effective Stress $\left(\sigma'_y\right)$	$\sigma_y' = \left(\sigma_y - u\right)$
Z-Effective Stress $\left(\sigma'_z\right)$	$\sigma_z' = \left(\sigma_z - u\right)$
Maximum Effective Stress $\left(\sigma'_{\max}\right)$	$\sigma_{\max}' = \sigma_{\max} - u$
Minimum Effective Stress $\left(\sigma'_{\min}\right)$	$\sigma_{\min}' = \sigma_{\min} - u$
Mean Effective Stress $\left(p'\right)$	$p' = \dfrac{\left(\sigma_x + \sigma_y + \sigma_z\right)}{3} - u$
Normal Effective Stress $\left(\sigma'_n\right)$	$\sigma_n' = \dfrac{\sigma_x' + \sigma_y'}{2} + \dfrac{(\sigma_x' - \sigma_y')}{2}\cos 2\theta + \tau_{xy}\sin 2\theta$
Tangential Effective Stress $\left(\sigma'_t\right)$	$\sigma_t' = \dfrac{(\sigma_x' - \sigma_y')}{2}\sin 2\theta - \tau_{xy}\cos 2\theta$

Maximum Shear Stress $\left(\tau_{xy-\max}\right)$	$\tau_{xy-\max} = \sqrt{\left[\dfrac{\left(\sigma_y - \sigma_x\right)}{2}\right]^2 + \tau_{xy}^2} = \sigma_1 - \sigma_3$
Deviatoric Stress (q)	$q = \dfrac{1}{\sqrt{2}}\sqrt{\begin{aligned}&\left(\sigma_x - \sigma_y\right)^2 + \left(\sigma_y - \sigma_z\right)^2 + \left(\sigma_z - \sigma_x\right)^2 \\ &+6\tau_{xy}^2\end{aligned}}$

13.4 Node and element information

In order to understand what type of information can be viewed as results output, it helps a bit to know how the data is obtained. So, to recap, you set up the problem geometry, define material properties, and apply boundary conditions of either known head (or pressure) or flux. The solver assembles the soil property and geometry information for every Gauss point in every element and applies it to the flow equation that is written for every node. Therefore, at each node we have applied boundary data, interpolated soil property data and geometry data. The solver then computes the unknown value in the equation for each node – the unknown value being either head or flux. It is the Gauss point data that is used to set up the nodal equations, so the Gauss point data written to the output file is the actual data used in the solver.

In GeoStudio, all output data for nodes and gauss points anywhere in the model is accessible using the View Results Information command. With the command selected, you can click the mouse on any single node to view the output at the node. You can also hold down the shift key to multi-select many points. If you click beside a node and within the element itself, you will get the Gauss point data at that location. You can multi-select Gauss point to see a table of data.

Figure 13-1 is an illustration of the type of information that can be viewed for each node in the finite element mesh. The data types are grouped by category with each category containing various related data options. 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.

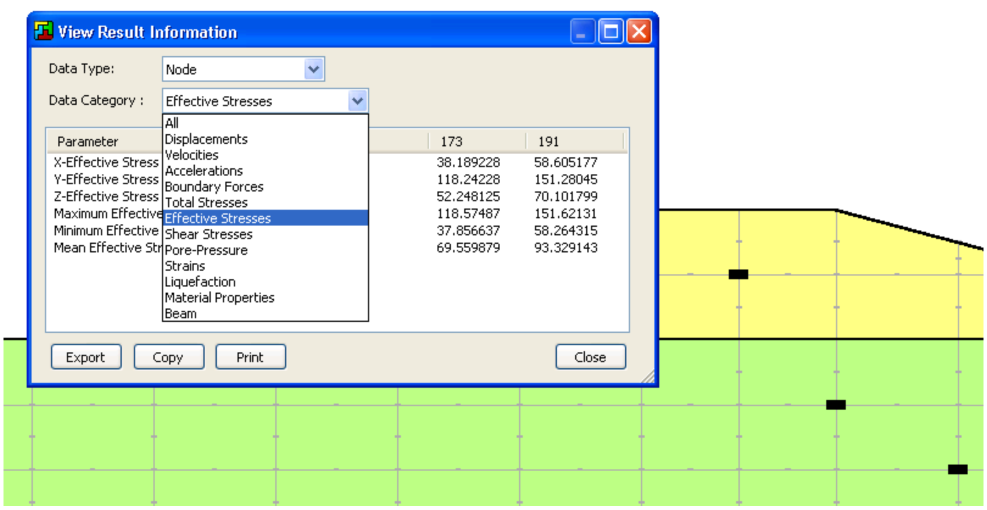


Figure 13-1 Visualization of node information

Figure 13-2 is the corresponding Gauss point information for the Gauss point located just below and to the right of the node 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, this Gauss area is equal to one quarter of the total area of the element.

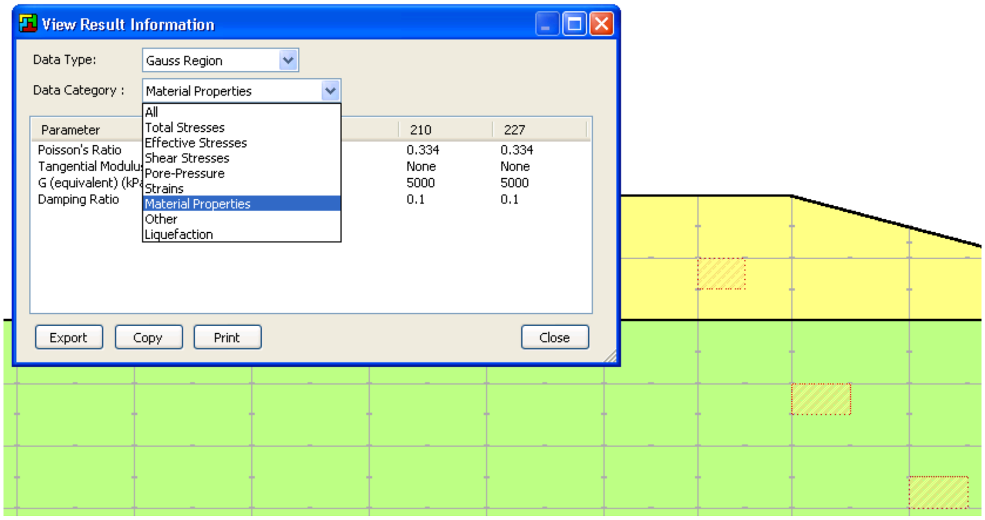


Figure 13-2 Visualization of element Gauss point information

13.5 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 13-3 shows a combination of all three graph data objects in a single dam cross section.

The advantage of using this type of data selection is that the location and type of data used in any graph can be named and saved. Each time you return to the graphing command, you can choose from your saved list of graphs and you do not have to re-define them. Even if you change the mesh, the model will know the new nodes nearest to your graph selections and it will draw the graph using the most recent solution.

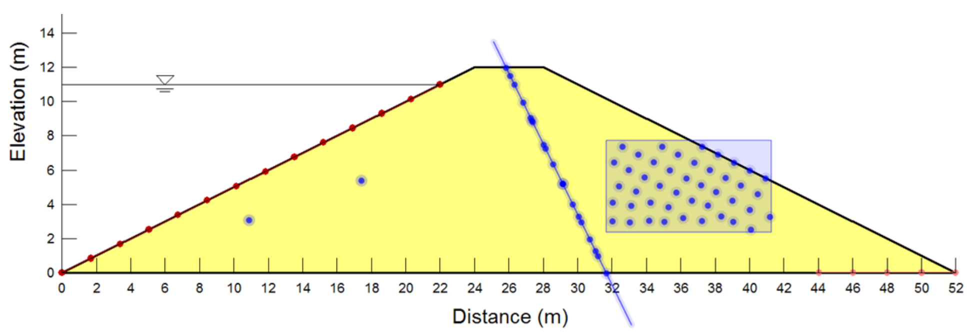


Figure 13-3 Graph data selection options (points, lines, planes)

The graphing options in SIGMA/W are very powerful. In most cases, any type of data you need to access and graph can be done right inside the program. In rare cases, you can get tabular access to the raw graph data and paste it directly into an external program such as Excel for additional graphing or manipulation.

At selected nodes, the Draw Graph command allows you to plot a graph containing any of the computed nodal parameter values listed in Table 13-4.

Table 13-4 Data available for graphing

XY-Displacement	X-Boundary Force
X-Displacement	Y-Displacement
Y-Boundary Force	XY-Boundary Force
X-Total Stress	Y-Total Stress
Z-Total Stress	Maximum Total Stress
Minimum Total Stress	Mean Total Stress (*p*)
Normal Total Stress	Tangential Total Stress
X-Effective Stress	Y-Effective Stress
Z-Effective Stress	Maximum Effective Stress
Minimum Effective Stress	Mean Effective Stress (*p'*)
Normal Effective Stress	Tangential Effective Stress
X-Y Shear Stress	Maximum Shear Stress
Deviatoric Stress (*q*)	Pore-Water Pressure
X-Strain	Y-Strain
Z-Strain	X-Y Shear Strain
Maximum Strain	Minimum Strain
Maximum Shear Strain	Volumetric Strain
Deviatoric Strain	Poisson's Ratio
Undrained Shear Strength	Void Ratio
Tangential Modulus (E)	
Moment (beam element)	Rotation (beam element)
Axial Force (beam / bar element)	Shear Force (beam element)

The parameters in this table are the *dependent variables* of the graph. Any of the dependent variables can be plotted versus the following *independent variables*: nodal *x*-coordinates, nodal *y*-coordinates, or distance between nodes (starting at the first selected node).

The following *independent variables* are also available if more than one time increment is being viewed: time, X-Displacement, Y-Displacement, X-Strain, Y-Strain, X-Y Shear Strain.

The independent graph variable that you choose affects how the selected nodes and time steps are used in the graph:

If the graph independent variable is x-coordinate, y-coordinate, or distance, then the parameter value at each selected node is plotted versus the nodal coordinate or the distance between nodes. Each selected time step is plotted as a separate line on the graph.

If the graph independent variable is time, then the parameter value at each selected node is plotted versus the elapsed time for each of the selected time steps. Each selected node is plotted as a separate line on the graph.

Sum(Y) versus average(X) graphs

A new graphing option that is quite powerful in some cases is the option to graph the sum of all selected nodal dependent variables versus the average of all selected independent variables. Consider for example that you want to create a load-deformation graph for designing a footing. In the past, you would have to extract force and displacement data for each node at the base of the footing and then take the data into Excel to sum the forces and average the displacement. This can be done automatically now as illustrated in Figure 13-4. In this figure, all the nodes beneath the footing are selected and then the graphing routine sums the forces and plots them against the average displacement.

This graphing option can be used for many scenarios throughout the GeoStudio analysis tools.

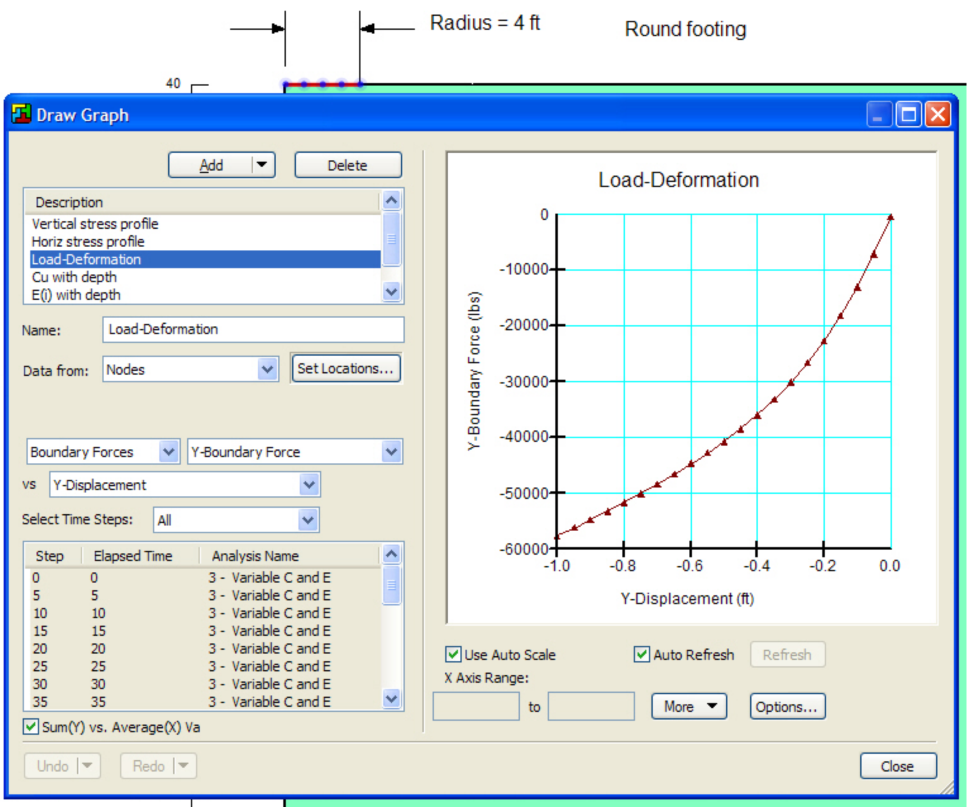


Figure 13-4 Load-displacement graph for footing design

13.6 "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 13-1 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.

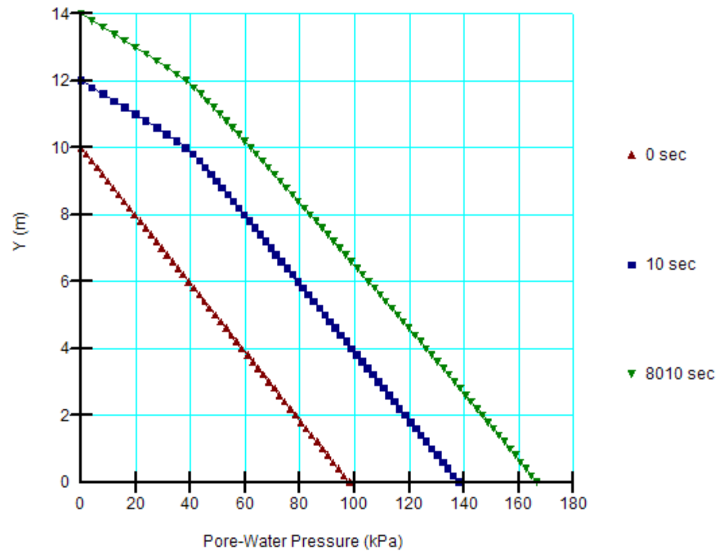


Figure 13-5 Graph showing how missing data is excluded and not printed as zero

13.7 Isolines

You can use the Draw Isolines command to choose which parameter you want the water table calculation based on. It can be pore water pressure, displacements or stresses. You can also choose to draw an isoline contour of any other parameter at an instance in time or over multiple times. If you draw an isoline at multiple time steps then you cannot also view contour shading as they are only valid for any instance in time. The isolines are a way to track a single value of a parameter as it changes over time, such as a water table.

13.8 Mohr circles

A complete picture of the stress state at a point can be viewed with a Mohr circle diagram such as in Figure 13-6.

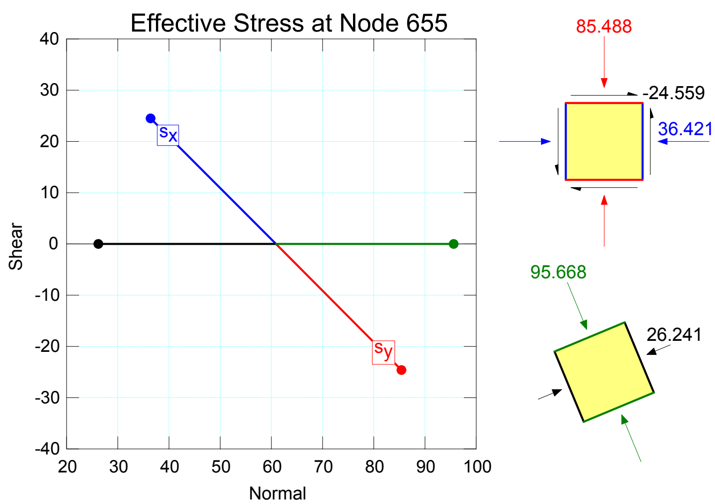


Figure 13-6 Mohr circle of stresses in an element

Mohr circles can be viewed at nodes or Gauss regions in an element. Node stresses are an average of the stresses in the adjoining Gauss regions.

Mohr circles can be in terms of total stresses or effective stresses.

Strain Mohr circles can be viewed as well as stress Mohr circles.

13.9 Animation

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.

13.10 Viewing displacements

Displacements can be displayed and viewed at any time for which was saved. The displacements can be viewed as a deformed mesh (Figure 13-7) or as displacement vectors Figure 13-8.

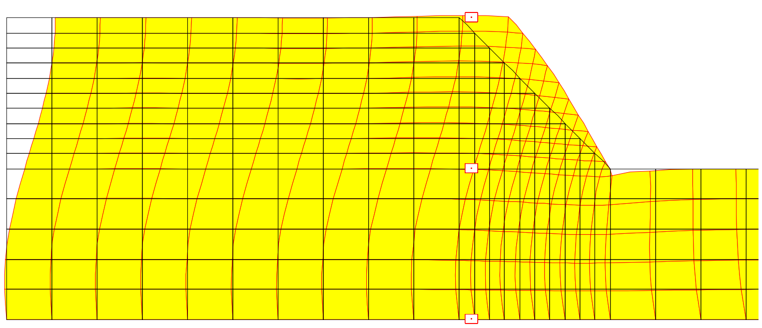


Figure 13-7 A deformed mesh at a 50x exaggeration

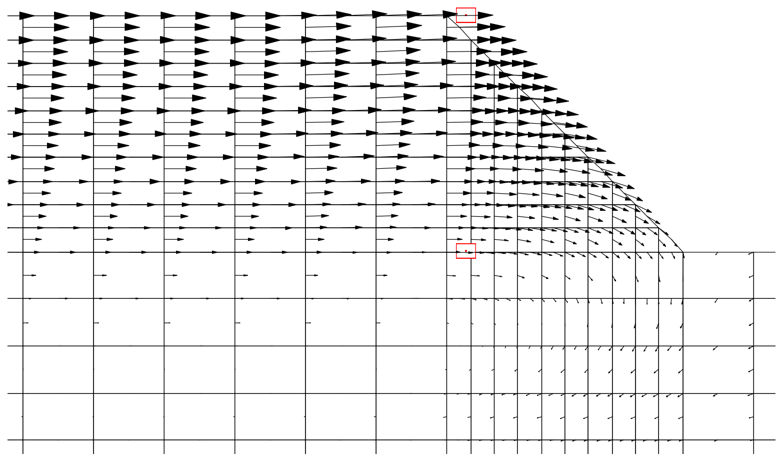


Figure 13-8 Displacement vectors

Generally, the deformed mesh presents a clearer picture of displacements than the displacement vectors. The vectors are useful in selected areas, but not for the entire mesh. Since the vectors are proportioned to displacement the vectors are often too long in one area and too small in another area or vise versa.

Displacements can be viewed only for a single time step. A sequence of the displacements can be viewed as discussed in the previous section on animation.

Most displacements cannot be drawn un-scaled, since they would be too small to be visible. Specifying a magnification value allows you to control the scale at which the displacement is 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 displacement will be drawn. You can control the displacement length either by specifying a magnification value or by specifying a maximum length value. The magnification value is computed according to the following general relationship:

$$\text{Magnification}=\frac{\text{Max.Length} * \text{Eng. Scale}}{\text{Max. Deformation}} * \text{Unit Conversion}$$

If you are viewing displacements across various analyses, you have the option to fix either the magnification factor or vector length. This will let you control the re-scaling of displacement which may be un-desirable in some cases.

13.11 Projecting Gauss point values to nodes

SIGMA/W contours parameter values calculated at nodes. Since displacement and boundary force are computed and stored at the nodes, these parameters can be contoured directly. All other parameters are stored at the element Gauss points, however, and must therefore be projected to the nodes for contouring purposes.

In triangular elements, the Gauss point values are projected on the basis of a plane that passes through the three Gauss points. For one-point integration, the value at the Gauss point is also taken to be the value at the nodes (i.e., the Gauss point value is constant within the element).

In quadrilateral elements, the Gauss point values are projected using the interpolating functions described in the appendix. In equation form:

$$x = \langle N \rangle \{X\}$$

where:

x	=	projected value outside the Gauss points at a local coordinate greater than 1. 0
$<N>$	=	matrix of interpolating functions
$\{X\}$	=	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 13-9 is an example of the local coordinates at the element corner nodes when projecting outwards from the four Gauss points in the element. The value of 1.7320 is the reciprocal of the Gauss point coordinate 0.57735.

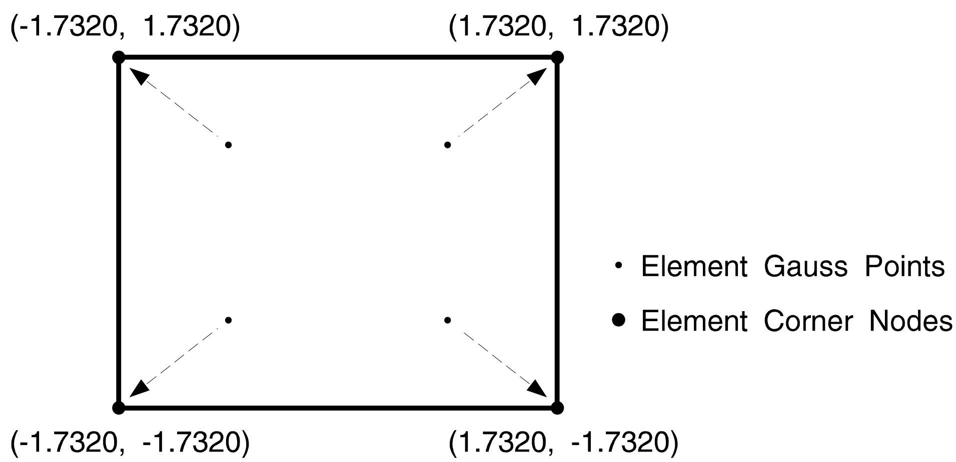


Figure 13-9 Local coordinates at the corner nodes of an element with four integration points

This projection technique can result in some over-shoot at the corner nodes when variation in the Gauss point values is large. For example, consider that we wish to contour stress in a highly deformed element which, consequently, has large variations in stresses at the Gauss points. Projecting such large variation in stresses can result in unrealistic stress at the nodes.

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.

13.12 Contouring data

There is a long list of computed values that can be contoured, and it generally includes any of the information that can be viewed at a node as illustrated above. Figure 13-10, for example, is a contour plot of excess (final minus initial) pore-pressures in the foundation.

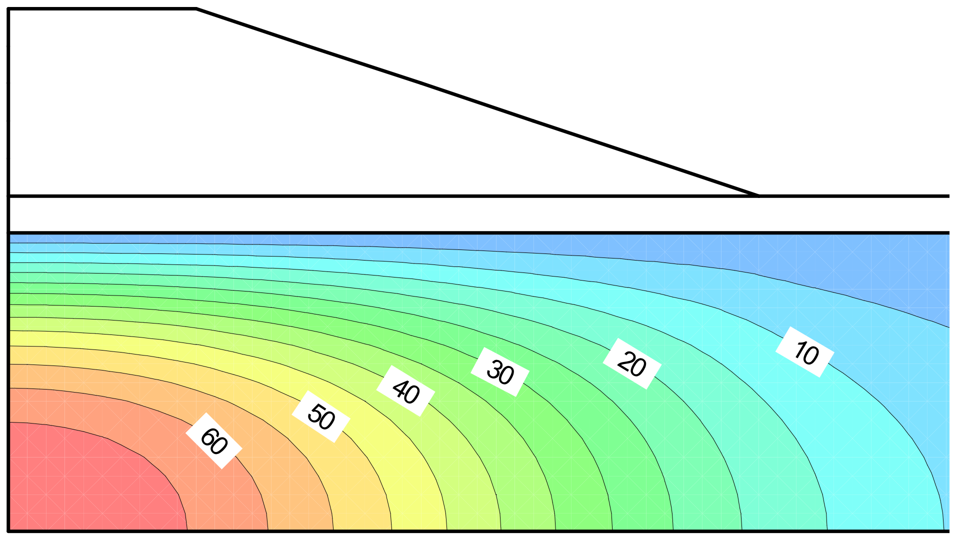


Figure 13-10 Contours of excess pore-pressure

The contour dialog box, illustrated in Figure 13-11, shows the data range for the selected parameter. In this example, the pore-pressure range is 0.0 to 69.8324 kPa. From this, SIGMA/W computes a starting contour value, a contour interval and the number of contours. These are default values. Seldom do these default contour parameters give a nice picture. The problem usually is some numeric noise in the data at the extremities of the data range. Usually the contour parameters need some adjustment to produce meaningful and publishable contour plots. The default values are good for a quick glance at the results, but not adequate for presentation purposes.

It usually takes several iterations to obtain a contour plot that is meaningful and presents the intended message. The contour coloring and shading scheme is described in the software on-line help. Once the contour is created, it can be named and saved so that its available whenever you need to review it again.

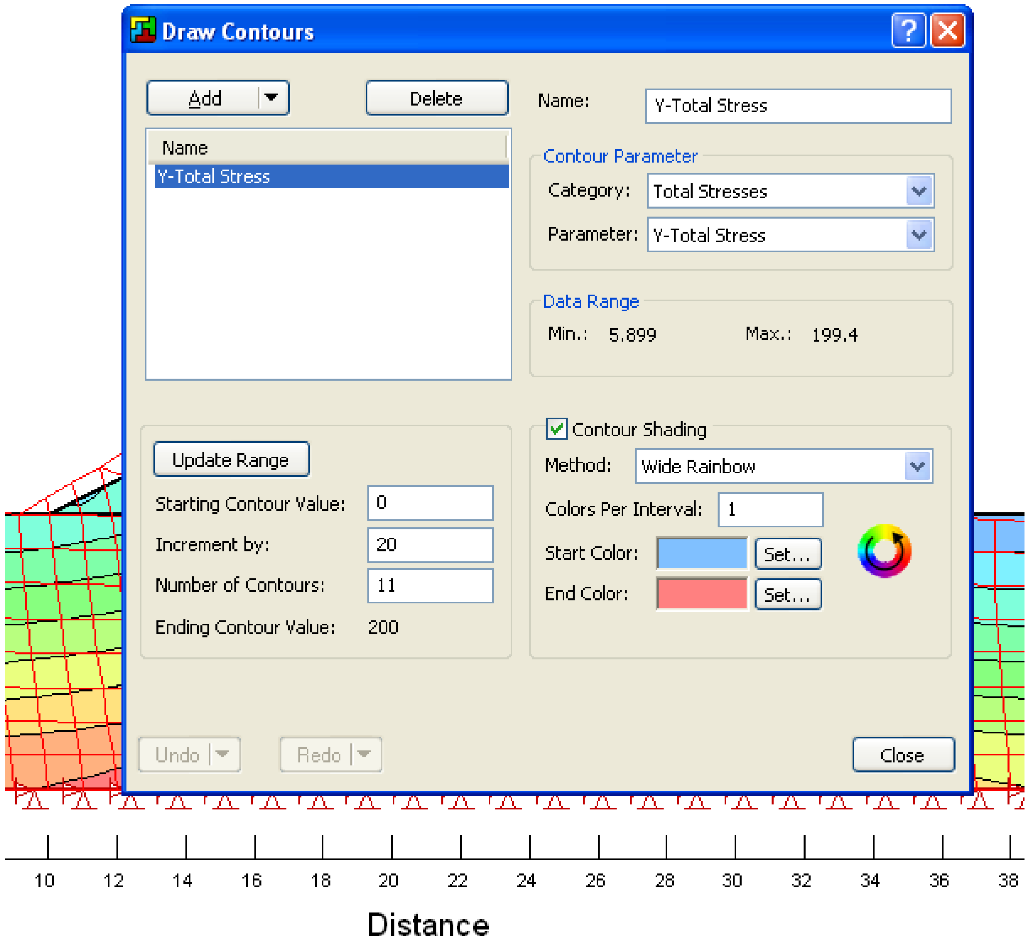


Figure 13-11 Contour dialog box

Some of the minor numerical irregularities can be ignored by simply selecting a contour range that excludes these values or by contour data specific to certain element regions as discussed next.

14 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.

15 Theory

15.1 Introduction

This chapter presents the methods, equations, procedures, and techniques used in the formulation and development of the SIGMA/W SOLVE function. It is of value to be familiar with this information when using the software. An understanding of these concepts will be of great benefit in applying the software, resolving difficulties, and judging the acceptability of the results.

The development of the finite element equations for stress/deformation analysis using potential energy, weighted residuals, or variational methods is well documented in standard textbooks, and consequently is not duplicated in this User's Guide. (See Bathe, 1982, Smith and Griffiths, 1988, Segerlind, 1984 and Zienkiewicz and Taylor, 1989 for further information on the development of finite element equations).

SIGMA/W is formulated for either two-dimensional plane strain or axisymmetric problems using small displacement, small strain theory. Conforming to conventional geotechnical engineering practice, the "compression is positive" sign convention is used. In this chapter, the bracket sets < >, { }, and [ ] are used to denote a row vector, a column vector and a matrix, respectively.

15.2 Finite element equations

The finite element equation used in the SIGMA/W formulation for a given time increment is:

$$\int_v [B]^T [C][B] dv \ \{a\} = b\int_v < N>^T dv + p\int_A < N>^T dA + \{F_n\}$$

where:

$[B]$	=	strain-displacement matrix,
$[C]$	=	constitutive matrix,
$\{a\}$	=	column vector of nodal incremental x- and y-displacements,
$< N>$	=	row vector of interpolating functions,
A	=	area along the boundary of an element,
v	=	volume of an element,
b	=	unit body force intensity,
p	=	incremental surface pressure, and
$\{F_n\}$	=	concentrated nodal incremental loads.

Summation of this equation over all elements is implied. It should be noted the SIGMA/W is formulated for incremental analysis. For each time step, incremental displacements are calculated for the incremental applied load. These incremental values are then added to the values from the previous time step. The accumulated values are reported in the output files. Using this incremental approach, the unit body force is only applied when an element is included for the first time during an analysis.

For a two-dimensional plane strain analysis, SIGMA/W considers all elements to be of unit thickness. For constant element thickness, t, the above equation can be written as:

Equation 15-1
$$t\int_A [B]^T [C][B] dA \; \{a\} \; = \; bt\int_A < N>^T dA \; + \; pt\int_L < N>^T dL$$

However, in an axisymmetric analysis, the equivalent element thickness is the circumferential distance about the axis of symmetry. Although the complete circumferential distance is 2π radians times the radial distance, R, SIGMA/W is formulated for one radian (unity). Consequently, the equivalent thickness is R and the finite element equation for the axisymmetric case becomes:

$$\int_A \left([B]^T [C][B]R\right) dA \; \{a\} \; = \; b\int_A \left(R< N>^T\right) dA + p\int_L \left(R<N>^T\right) dL + \{F_n\}$$

Unlike the thickness, t, in a two-dimensional analysis, this radial distance, R, is not a constant within an element. Consequently, R needs to be evaluated inside the integral.

In an abbreviated form, the finite element equation is:

Equation 15-2
$$[K]\{a\} = \{F\} = \{F_b\} + \{F_s\} + \{F_n\}$$

where:

[K] = element characteristic (or stiffness) matrix,

$= t\int_A \left([B]^T [C][B]\right) dA$ (for plane strain), or

$= \int_A \left([B]^T [C][B]R\right) dA$ (for axisymmetric)

{a} = nodal incremental displacements,

{F} = applied nodal incremental force which is made up of the following:

$\{F_b\}$ = incremental body forces,

$\{F_s\}$ = force due to surface boundary incremental pressures,

$= pt\int_L \left(< N>^T\right) dL,$ for two-dimensional analysis, or

$= p\int_L \left(R< N>^T\right) dL$ for axisymmetric analysis, and

$\{F_n\}$ = concentrated nodal incremental forces.

SIGMA/W solves this finite element equation for each time step to obtain incremental displacements and calculates the resultant incremental stresses and strains. It then sums all these increments since the first time step and reports the summed values in the output files.

Strain-displacement matrix

SIGMA/W uses engineering shear strain in defining the strain vector:

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \end{Bmatrix}$$

The field variable of a stress/deformation problem is displacement which is related to the strain vector through:

Equation 15-3 $$\{\varepsilon\} = [B]\begin{Bmatrix} u \\ v \end{Bmatrix}$$

where:

[B] = strain matrix,

u, v = nodal displacement in x- and y-directions, respectively.

SIGMA/W is restricted to performing infinitesimal strain analyses. For a two-dimensional plane strain problem, ε_Z is zero and the strain matrix is defined as:

$$[B] = \begin{bmatrix} \dfrac{\partial N_1}{\partial x} & 0 & \ldots & \dfrac{\partial N_8}{\partial x} & 0 \\ 0 & \dfrac{\partial N_1}{\partial y} & \ldots & 0 & \dfrac{\partial N_8}{\partial y} \\ 0 & 0 & \ldots & 0 & 0 \\ \dfrac{\partial N_1}{\partial x} & \dfrac{\partial N_1}{\partial y} & \ldots & \dfrac{\partial N_8}{\partial x} & \dfrac{\partial N_8}{\partial y} \end{bmatrix}$$

For an axisymmetric problem, the strain matrix can be written as:

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \dfrac{\partial u}{\partial x} \\ \dfrac{\partial v}{\partial y} \\ \dfrac{u}{r} \\ \dfrac{\partial u}{\partial y} + \dfrac{\partial v}{\partial x} \end{Bmatrix}$$

The associated strain matrix $[B]$ is then:

$$[B]=\begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & \ldots & \frac{\partial N_8}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial y} & \ldots & 0 & \frac{\partial N_8}{\partial y} \\ \frac{N_1}{R} & 0 & \ldots & \frac{N_8}{R} & 0 \\ \frac{\partial N_1}{\partial x} & \frac{\partial N_1}{\partial y} & \ldots & \frac{\partial N_8}{\partial x} & \frac{\partial N_8}{\partial y} \end{bmatrix}$$

The derivatives of the interpolating functions are given in the Derivatives of Interpolating Functions section in this chapter.

Elastic constitutive relationship

Stresses are related to strains as follows, within the theory of elasticity:

$$\{\sigma\}=[C]\{\varepsilon\}$$

where $[C]$ is the constitutive (element property) matrix and is given by:

$$[C]=\frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1-\nu & \nu & \nu & 0 \\ \nu & 1-\nu & \nu & 0 \\ \nu & \nu & 1-\nu & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix}$$

where:

E	=	Young's modulus
v	=	Poisson's ratio

The $[C]$ matrix is the same for both the two-dimensional plane strain and the axisymmetric cases.

Body forces

SIGMA/W can model body forces applied in both the vertical and the horizontal directions. These forces are applied to all elements when they first become active. The body force in the vertical direction, b_v, is due to gravity acting on an element. For a given material, the unit body force intensity in the vertical direction is given by its unit weight, γ_s, which is in turn related to its mass density, ρ:

$$\gamma_s = \rho g$$

where g is the gravitational constant. When the unit weight γ_s is non-zero, SIGMA/W evaluates the integral $\gamma_s \int_v \left(< N>^T\right) dv$ by numerical integration and applies a vertically downward (negative) force at each node of the element.

Similarly, when the unit body force intensity in the horizontal direction, b_h, is nonzero, nodal forces in the horizontal direction are computed using $b_h \int_v \left(< N>^T\right) dv$.

Forces due to boundary stresses

The term $p\int_A \left(< N>^T\right) dA$ in Equation 15-2 represents the nodal forces caused by externally applied pressure along the boundary of the element. SIGMA/W evaluates this integral using numerical integration in the SOLVE function.

Three loading types are available in SIGMA/W; namely, normal and tangential pressure; x- and y-stress; and fluid pressure. The procedure used to derive the equivalent nodal forces for each of the pressure boundary type is described below.

Consider the element edge subjected to normal and tangential pressure loading as shown in Figure 15-1. In order to calculate the equivalent nodal forces acting along that edge, the elemental loads must be resolved into x- and y- components. The normal and tangential pressure, pn and pt, acting on an elemental length, dS of the loaded edge, result in elemental forces, dPx and dPy, in the x- and y-direction. These forces can be written as follows:

$$dP_x = (p_t \cos\alpha - p_n \sin\alpha) dS.t = (p_t dx - p_n dy)t$$
$$dP_y = (p_t \sin\alpha + p_n \cos\alpha) dS.t = (p_n dx + p_t dy)t$$

where t is the thickness of the element.

The total force can be obtained by integrating along the element edge (i.e., along the local coordinate r). The differentials dx and dy can be expressed in terms of r:

$$dx = \frac{\partial x}{\partial r} dr$$
$$dy = \frac{\partial y}{\partial r} dr.$$

Substituting these differentials into the incremental force equation and applying the finite element approximation, the following equations can be derived for the equivalent nodal forces at node i along an element edge:

$$P_{xi} = \int_S N_i t (p_t \frac{\partial x}{\partial r} - p_n \frac{\partial y}{\partial r}) dr$$
$$P_{yi} = \int_S N_i t (p_n \frac{\partial x}{\partial r} + p_t \frac{\partial y}{\partial r}) dr.$$

The integration is carried out numerically in SIGMA/W SOLVE using Gaussian quadrature. The number of integration points used corresponds to the number of nodes along this element edge.

Resolution of elemental forces will not be necessary when x- and y-stresses being applied along an edge of the element. The equations for the equivalent nodal forces become:

$$P_{xi} = \int_S N_i t (p_x \frac{\partial x}{\partial r}) dr$$

$$P_{yi} = \int_S N_i t (p_y \frac{\partial y}{\partial r}) dr.$$

The fluid pressure boundary is as a particular case of normal and tangential case in which no tangential pressure is applied. During numerical integration, fluid pressure is evaluated at each integration point, say the i-th integration point, using:

$$p_{ni} = \gamma_f (y_f - y_i), \qquad (y_f - y_i) > 0$$

where:

γ_f	=	the self-weight of the fluid,
y_f	=	the elevation of the fluid, and,
y_i	=	the y-coordinate at the integration point.

The fluid pressure is only computed when the fluid elevation exceeds the y-coordinate of an integration point.

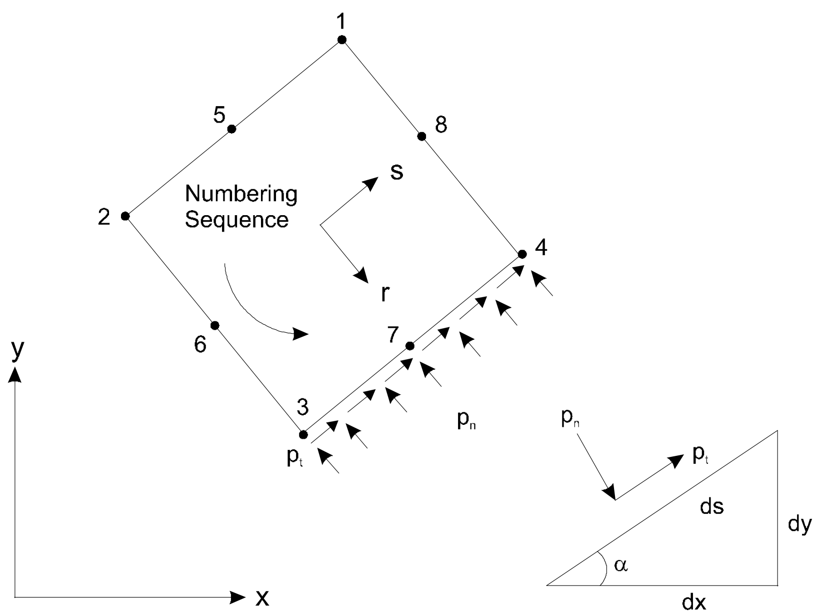


Figure 15-1 Normal and tangential pressure along an element edge

Nodal forces

In SIGMA/W, nodal forces are included in the finite element formulation, Equation 15-2, under two conditions. The nodal force can either be specified as boundary conditions or SIGMA/W can calculate them internally when elements are first excavated.

For each element, nodal forces are computed using:

$$\{F\} = \int_v [B]^T \{\sigma\} dv$$

where:

$\{\sigma\}$	=	the vector of element stresses,
$[B]^T$	=	the transpose of the strain-displacement matrix, and
dv	=	elemental volume.

The resultant nodal forces are accumulated at each node. To simulate the removal of soils, as in an excavation, the signs on the nodal forces are reversed before these forces are incorporated into the finite element equation.

In the displacement output files, SIGMA/W reports the nodal forces at all nodes where displacement or spring boundary conditions are specified. These nodal forces are calculated in the manner previously described. However, there is now no change of signs. The nodal forces represent boundary forces which may be used, for example, to estimate the loads supported by struts in a braced excavation.

15.3 Numerical integration

SIGMA/W uses Gauss-Legendre numerical integration (also termed quadrature) to form the element characteristic (or stiffness) matrix [K]. The variables are first evaluated at specific points within an element. These points are called integration points or Gauss points. These values are then summed for all the Gauss points within an element. This mathematical procedure is as described in the following.

To carry out numerical integration, SIGMA/W replaces the following integral from Equation 15-1:

$$\int_A [B]^T [C][B] dA$$

with the following equation:

$$\sum_{j=1}^{n} \left[B_j\right]^T \left[C_j\right]\left[B_j\right] \det\left|J_j\right| W_{1j} W_{1j}$$

where:

j	=	integration point,
n	=	total number of integration points or integration order,
$\det\left\|J_j\right\|$	=	determinant of the Jacobian matrix, and
W_{1j} , W_{2j}	=	weighting factors

Table 15-1 to Table 15-4 show the numbering scheme and location of integration points used in SIGMA/W for various element types.

Table 15-1 Sample point locations and weightings for four point quadrilateral element

Point	r	s	w_1	w_2
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 15-2 Sample point locations and weightings for nine point quadrilateral element

Point	r	s	w_1	w_2
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 15-3 Sample point location and weighting for one point triangular element

Point	r	s	w_1	w_2
1	0.33333	0.33333	1.0	0.5

Table 15-4 Sample point locations and weightings for three point triangular element

Point	r	s	w_1	w_2
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

The appropriate integration order is dependent on the number of secondary nodes. When secondary nodes are present, the interpolating functions are non-linear and consequently a higher integration order is required. Table 15-5 gives the acceptable integration order for various element types.

Table 15-5 Acceptable element integration orders

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

Under certain conditions, it is acceptable to use four-point integration for quadrilateral elements which have secondary nodes. This procedure is called reduced integration and is described in Bathe (1982), and Zienkiewicz and Taylor (1989). For example, reduced integration may yield more accurate results in nearly incompressible elasticity and in elements under flexure. In addition, selective use of reduced integration can greatly reduce the required number of computations.

It is also possible to use higher order (three-point and nine-point) integration with elements that have no secondary nodes. However, in this case, the benefits of using higher order integration are marginal, particularly for quadrilateral elements. Nine-point integration for quadrilateral elements 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. Using one-point integration implies that the material properties and strains are constant within the element. This can lead to poor performance of the element, particularly if the element is in a region of large stress gradients. 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 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.

15.4 Assembly and solving of global equations

The element matrix for each element in the discretized finite element mesh can be formed and assembled into a global system of simultaneous equations. The finite element solution requires the solving of the system of simultaneous equations.

SIGMA/W stores the coefficients of the global system equations using a Compressed Row Storage scheme (Barrett et. al, 1994). This is a general scheme that makes no assumptions about the sparsity structure of the matrix. Instead of a full matrix with many zero elements, the Compressed-Row Storage scheme stores the matrix with 3 vectors: one for the non-zero elements, one for the column index and one for the row pointers. As a result, it provides significant saving in storage memory particularly in very large finite element meshes.

SIGMA/W utilizes a preconditioned Bi-Conjugate Gradient (BiCG) iterative solver in solving the system equations. The BiCG solver is adopted from IML++ (Iterative Methods Library) made available freely by the National Institute of Standards and Technology, Oak Ridge National Laboratory, University of Tennessee, Knoxville, U. S. A. The BiCG solver works effectively with the compressed row storage scheme and is suitable for both symmetric and non-symmetric system of equations.

The equation solver can accommodate missing elements in the array. This feature makes it possible to add and delete elements from a finite element mesh without renumbering the nodes and elements. Unlike the Gauss elimination solver used in previous versions of SIGMA/W, sorting the node number vertically or horizontally will have no effect to the computational time and accuracy of the solution with this improved iterative solver.

As a result of the iterative solver, the solution of the system of simultaneous equations is to a large degree dependent on the convergence tolerance and the number of iterations. In most cases, the default tolerance and number of iterations will be adequate for a solution.

The Gauss Elimination Skyline Direct Solver is also included in the program. The computational speed of a direct solver is depending on the bandwidth of the finite element mesh. The direct solver is fast for simple problems, but it can be slower than the iterative solver in complex problems with large bandwidths. You may like to try both the direct and iterative solvers for a few iterations first before solving a large problem with many iterations and time steps.

15.5 Element stresses

SIGMA/W computes the stresses and strains at each integration point within each element once the nodal displacements have been obtained. Strains are computed from nodal displacements using Equation 15-3.

Stresses are computed at each Gauss point using the constitutive matrix [C] in the following manner:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \end{Bmatrix} = [C] \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \end{Bmatrix}$$

16 Appendix A: Interpolating Functions

16.1 Coordinate systems

The global coordinate system used in the formulation of SIGMA/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 16-1. Presented as well in Figure 16-1 is the local element node numbering system. The local coordinates for each of the nodes are given in Table 16-1.

Table 16-1 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	–	–

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

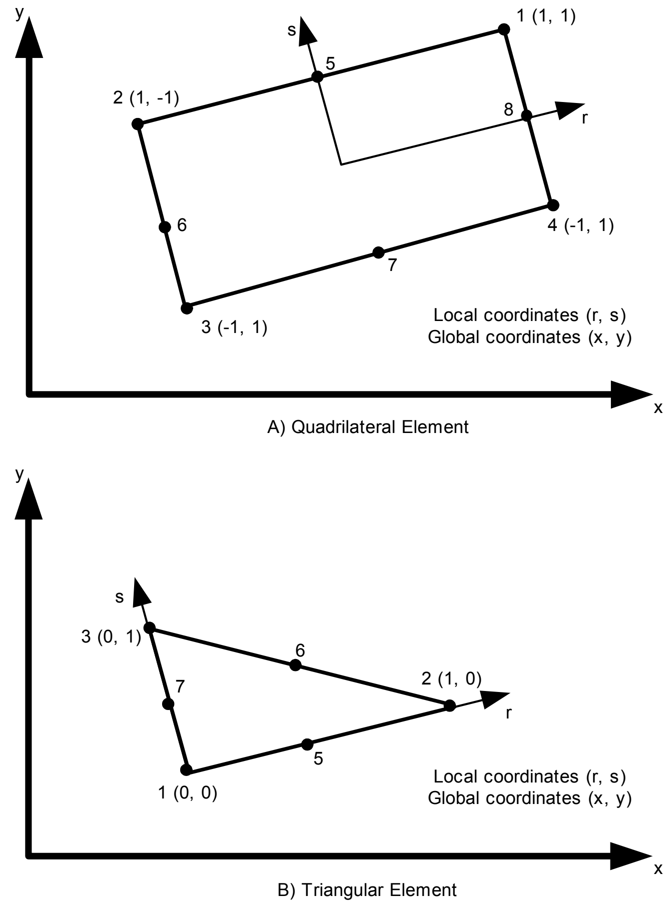


Figure 16-1 Global and local coordinate system

To formulate a finite element analysis it is necessary to adopt a model for the distribution of the field variable within each finite element. The field variable in a stress/deformation analysis is nodal displacement.

SIGMA/W assumes that the displacement distribution within the element follows the interpolating functions described previously in this chapter. This means that the displacement distribution is linear when secondary nodes are not used, and the displacement distribution is quadratic when the secondary nodes are used.

The displacement distribution model at any given location inside a finite element is given by the following set of equations:

Equation 16-1 $u = \langle N \rangle \{U\}$

Equation 16-2 $v = \langle N \rangle \{V\}$

where:

u	=	x-displacement at the given location
v	=	y-displacement at the given location
$\{U\}$	=	x-displacement at the nodes of the element

$\{V\}$ = y-displacement at the nodes of the element

$\langle N \rangle$ = interpolation functions evaluated at the given point

16.2 Derivatives of interpolating functions

The fundamental constitutive relationship used in the formulation of SIGMA/W relates stress, σ, to strain, ε, using the stiffness , E, of the material. In equation form:

$$\sigma = E\varepsilon$$

In a two-dimensional plane strain problem, there are three basic strain components: longitudinal strain in the x-direction, ε_x, longitudinal strain in the y-direction, ε_y, and shear strain in the x-y plane, γ_{xy}. SIGMA/W is formulated for small displacement, small strain problems. The strain components are related to x- and y displacements, u and v, as follows:

$$\varepsilon_x = \frac{\partial u}{\partial x}$$

$$\varepsilon_y = \frac{\partial v}{\partial y}$$

$$\gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}$$

At any point within a finite element, displacements u and v are related to the nodal displacement vectors $\{U\}$ and $\{V\}$ by Equation 16-1 and Equation 16-2. Strains, when expressed in terms of nodal displacements, can be written as follows:

$$\varepsilon_x = \frac{\partial u}{\partial x} = \left\langle \frac{\partial N}{\partial x} \right\rangle \{U\}$$

$$\varepsilon_y = \frac{\partial v}{\partial y} = \left\langle \frac{\partial N}{\partial y} \right\rangle \{V\}$$

$$\gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = \left\langle \frac{\partial N}{\partial y} \right\rangle \{U\} + \left\langle \frac{\partial N}{\partial x} \right\rangle \{V\}$$

The above equation shows that, in order to calculate strains, it is necessary to differentiate the interpolating functions with respect to x and y. The derivatives of the interpolating functions in the local and global coordinate systems are given by the chain rule:

Equation 16-3 $$\begin{Bmatrix} \left\langle \dfrac{\partial N}{\partial r} \right\rangle \\ \left\langle \dfrac{\partial N}{\partial s} \right\rangle \end{Bmatrix} = \begin{bmatrix} \dfrac{\partial x}{\partial r} & \dfrac{\partial y}{\partial r} \\ \dfrac{\partial x}{\partial s} & \dfrac{\partial y}{\partial s} \end{bmatrix} \begin{Bmatrix} \left\langle \dfrac{\partial N}{\partial x} \right\rangle \\ \left\langle \dfrac{\partial N}{\partial s} \right\rangle \end{Bmatrix}$$

in which:

Equation 16-4 $$\begin{bmatrix} \dfrac{\partial x}{\partial r} & \dfrac{\partial y}{\partial r} \\ \dfrac{\partial x}{\partial s} & \dfrac{\partial y}{\partial s} \end{bmatrix} = [J],$$ the Jacobian matrix.

Thus, the derivative of the interpolation functions, with respect to x and y can be determined by inverting Equation 16-3:

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

where $[J]^{-1}$ is the inverse of the Jacobian.

The Jacobian matrix can be obtained by substituting Equation 16-1 and Equation 16-2 into Equation 16-4.

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

As can be seen from Equation 16-3 and Equation 16-4, derivatives of the interpolating functions are also required for calculating strains and the Jacobian.

The derivatives of the interpolation functions with respect to r and s used by SIGMA/W for quadrilateral and triangular elements are given in Table 16-2 and Table 16-3, respectively.

Derivatives of the mapping functions with respect to r and s for one-directional and two-directional infinite elements are given in Table 16-4.

The following notation has been used in the preceding tables to represent the derivative of a given function N_i with respect to variable r:

$$N_i{,}r = \frac{\partial N_i}{\partial r} \quad .$$

Table 16-2 Interpolation function derivatives for quadrilateral elements

Derivative of Function			Include in derivative if node is present			
			5	6	7	8
N_1,r	=	¼(1+s)	-½(N_v,r)	—	—	-½(N_8,r)
N_2,r	=	-¼(1+s)	-½(N_5,r)	-½(N_6,r)	—	—
N_3,r	=	-¼(1-s)	—	-½(N_6,r)	-½(N_7,r)	—
N_4,r	=	¼(1-s)	—	—	-½(N_7,r)	-½(N_8,r)
N_5,r	=	-½(2r+2sr)	—	—	—	—
N_6,r	=	-½(1-s2)	—	—	—	—
N_7,r	=	-½(2r-2sr)	—	—	—	—
N_8,r	=	½(1-s2)	—	—	—	—
N_1,s	=	¼(1+r)	- ½(N_5,s)	—	—	-½(N_8,s)
N_2,s	=	¼(1-r)	-½(N_5,s)	-½(N_6,s)	—	—
N_3,s	=	-¼(1-r)	—	-½(N_6,s)	-½(N_7,s)	—
N_4,s	=	-¼(1+r)	—	—	-½(N_7,s)	-½(N_8,s)
N_5,s	=	½(1-r2)	—	—	—	—
N_6,s	=	-½(2s-2sr)	—	—	—	—
N_7,s	=	-½(1-r2)	—	—	—	—
N_8,s	=	-½(2s+2sr)	—	—	—	—

Table 16-3 Interpolation function derivatives for triangular elements

Derivative of Function			Include in derivative if node is present		
			5	6	7
N_1,r	=	-1. 0	$-½(N_5,r)$	—	—
N_2,r	=	1. 0	$-½(N_5,r)$	$-½(N_6,r)$	—
N_3,r	=	0. 0	—	$-½(N_6,r)$	$-½(N_7,r)$
N_5,r	=	(4-8r-4s)	—	—	—
N_6,r	=	4s	—	—	—
N_7,r	=	-4s	—	—	—
N_1,s	=	-1. 0	$-½(N_5,s)$	—	—
N_2,s	=	0. 0	$-½(N_5,s)$	$-½(N_6,s)$	—
N_3,s	=	1. 0	—	$-½(N_6,s)$	$-½(N_7,s)$
N_5,s	=	-4r	—	—	—
N_6,s	=	4r	—	—	—
N_7,s	=	(4-4r-8s)	—	—	—

Table 16-4 Mapping function derivatives for infinite elements

Mapping Function Derivatives					
1-D Infinite Elements			**2-D Infinite Elements**		
$M_{1,r}$	=	0	$M_{1,r}$	=	0
$M_{2,r}$	=	$(-2-s+s^2)/(1-r)^2$	$M_{2,r}$	=	0
$M_{3,r}$	=	$(-2+s+s^2)/(1-r)^2$	$M_{3,r}$	=	$-4(2+s)/(1-r)^2(1-s)$
$M_{4,r}$	=	0	$M_{4,r}$	=	0
M_5,r	=	$(1+s)/(1-r)^2$	$M_{5,r}$	=	0
$M_{6,r}$	=	$2(1-s2)/(1-r)^2$	$M_{6,r}$	=	$2(1+s)/(1-r)^2(1-s)$
$M_{7,r}$	=	$(1-s)/ (1-r)^2$	$M_{7,r}$	=	$4/(1-r)^2(1-s)$
$M_{8,r}$	=	0	$M_{8,r}$	=	0
$M_{1,s}$	=	0	$M_{1,s}$	=	0
$M_{2,s}$	=	$(-r+2s)/(1-r)$	$M_{2,s}$	=	0
$M_{3,s}$	=	$(r+2s)/(1-r)$	$M_{3,s}$	=	$-4(2+r)/(1-r)\ (1-s)^2$
$M_{4,s}$	=	0	$M_{4,s}$	=	0
$M_{5,s}$	=	$½(1+r)/(1-r)$	$M_{5,s}$	=	0
$M_{6,s}$	=	$-4s/(1-r)$	$M_{6,s}$	=	$4/(1-r)\ (1-s)^2$
$M_{7,s}$	=	$-½(1+r)/(1-r)$	$M_{7,s}$	=	$2(1+r)/(1-r)(1-s)^2$
$M_{8,s}$	=	0	$M_{8,s}$	=	0

17 References

Atkinson, J.H. and Bransby, P.L., 1978. The Mechanics of Soils: An Introduction to Critical State Soil Mechanics. McGraw-Hill.

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

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

Biot, M.A. , 1941. General theory of three-dimensional consolidation. Journal of Applied Physics, Volume 12, February, pp. 155-164.

Britto, A.M. and Gunn, M.J., 1987. Critical State Soil Mechanics via Finite Elements. John Wiley & Sons, Inc.

Chan, D. H.-K., 1986. Finite Element Analysis of Strain Softening Material. Ph.D. Thesis, Civil Engineering Department, University of Alberta, Edmonton, Alberta, Canada.

Chen, W.F. and Zhang, H., 1991. Structural Plasticity: Theory, Problems, and CAE Software. Springer-Verlag.

Dakshanamurthy, V., Fredlund, D.G, and Rahardjo, H., 1984. Coupled three-dimensional consolidation theory of unsaturated porous media. Proceedings of the Fifth International Conference on Expansive Soils, Adelaide, Australia, pp. 99 -103.

Dawson, E.M., Roth, W.H. and Drescher, A. 1999. Slope Stability Analysis by Strength Reduction, geotechnique, 49(6), pp. 835-840.

Duncan, J.M. and Chang, C.Y., 1970. Nonlinear analysis of stress and strain in soils. Journal of the Soil Mechanics and Foundations Division, ASCE, vol. 96, no. SM5, pp. 1629-1654.

Duncan, J.M., Byrne, P., Wong, K.S., and Mabry, P., 1980. Strength, Stress-Strain and Bulk Modulus Parameters for Finite Element Analyses of Stresses and Movements in Soil Masses. Report No. UBC/GT/80-01. Department of Civil Engineering, University of California, Berkeley, California (out of print).

Fredlund, D.G. and Morgenstern, N.R., 1976. Constitutive relations for volume change in unsaturated soil. Canadian Geotechnical Journal, Vol. 13, No. 3, pp.261 276

Fredlund, D.G. and Rahardjo, H., 1993. Soil Mechanics for Unsaturated Soils. John Wiley & Sons, Inc.

Gibson, R.E., Knight, K. and Taylor, P.W., 1963. A critical experiment to examine theories of three-dimensional consolidation. Proceedings from the European Conference on Soil Mechanics and Foundation Engineering, Weisbaden, Germany.

Griffiths, D.V. and Lane, P.A. 1999. Slope Stability Analysis by Finite Elements, Geotechnique, 49(3), pp. 387-403.

Henkel, D.J., 1960. The shear strength of saturated remoulded Clay. Proceedings of the ASCE Research Conference on Shear Strength of Cohesive Soil, Boulder, Colorado, USA, pp.533-544.

Hill, R., 1950. The Mathematical Theory of Plasticity. Oxford University Press.

Hinton, E. and Owen, D.R.J., 1979 An Introduction to Finite Element Computations, Pineridge Press Ltd., U.K.

Krahn, J. 2007. Limit Equilibrium, Strength Summation and Strength Reduction Methods for Assessing Stability, Proceedings, 1st Canada-U.S. Rock Mechanics Symposium, Vancouver, B.C., Canada, May 28-30.

Lambe, T.W. and Whitman, R.V., 1969. Soil Mechanics. John Wiley and Sons, Inc.

Lambe, T.W. and Whitman, R.V., 1979. Soil Mechanics, SI Version. John Wiley and Sons, Inc.

Lancaster, P. and Salkauskas, K., 1986. Curve and Surface Fitting: An Introduction. Academic Press. pp.101-106

Pickering, D.J., 1970. Anisotropic Elastic Properties for Soils. Geotechnique, Vol. 20, No.3, pp.271-276.

Salkauskas, K., 1974. C′ splines for interpolation of rapidly varying data. Rocky Mountain Journal of Mathematics, Vol. 14, No. 1, pp.239-250.

Segerlind, L.J., 1984. Applied Finite Element Analysis, 2nd Ed. John Wiley and Sons, Inc.

Smith, G.D., 1971. Numerical Solution of Partial Differential Equations, Oxford University Press, Oxford.

Smith, I.M. and Griffiths, D.V., 1988. Programming the Finite Element Method, 2nd Ed. John Wiley and Sons, Inc.

Terzaghi, K., 1943. Theoretical Soil Mechanics. John Wiley & Sons, Inc.

Winterkorn, H.F. and Fang, H.-Y., 1975. Foundation Engineering Handbook. Van Nostrand Reinhold Company, p.429.

Vanapalli, S.K., Fredlund D.G., Pufahl, D.E. and Clifton, A.W., 1996. Model for the Prediction of Shear Strength with respect to Soil Suction. Canadian Geotechnical Journal, Vol. 33, pp. 379-392.

Wroth, C.P. (1975). In-situ measurement of initial stresses and deformation characteristics, Proceedings of the Specialty Conference in In-situ Measurement of Soil Properties, ASCE, Raleigh, N.C., pp. 181-230.

Zienkiewicz, O.C. and Taylor, R.L., 1989. The Finite Element Method, 4th Ed., Vol. 1. McGraw-Hill.

18 Index

Examples of consistent sets of units

Property	Units	Metric	Imperial
Geometry	L	meters	feet
Unit Weight of Water	F/L^3	kN/m^3	pcf
Soil Unit Weight	F/L^3	kN/m^3	pcf
Cohesion	F/L^2	kPa	pcf
Pressure	F/L^2	kPa	psf
Force	F	kN	lbs
E (modulus)	F/L^2	kPa	psf