CRISP2D

GEOTECHNICAL FINITE ELEMENT ANALYSIS SOFTWARE

TECHNICAL REFEFRENCE MANUAL

For use with CRISP version 5

By
Rick Woods
and Amir Rahim

The CRISP Consortium Limited

www.mycrisp.com

This manual originally written by Rick Woods (Civil Engineering lecturer at Surrey University and Director of the CRISP Consortium Ltd) and Amir Rahim (Director of the CRISP Consortium Ltd and Director of CRISP Geotech, Inc)

The CRISP Consortium Ltd is registered in England and Wales. Company No. 3222642. Address: 82 Stoke Fields, Guildford GU1 4LT, UK
CRISP Geotech Inc is registered in the USA, Federal Tax No. 20-2780915. Address: 12739 Westella Dr, Houston, TX 77077, USA

Contents

INTRODUCTION

1

1.1 Program Capabilities

CRISP is a 2D finite element program. The finite element engine is capable of solving 2D and 3D problems, but the CRISP Windows interface is currently restricted to 2D plane strain and axi-symmetric problems. A short summary of the facilities available in CRISP are as follows:

1.1.1 Types of Analysis:

Undrained, drained or fully coupled (Biot) consolidation analysis of three- dimensional or two-dimensional plane strain or axisymmetric (with axisymmetric loading) solid bodies.

1.1.2 Constitutive Models:

- Homogeneous anisotropic linear elastic, non-homogeneous isotropic linear elastic (properties varying with depth),
- Elastic-perfectly plastic with Tresca, Von Mises, Mohr-Coulomb (Associative and Non-Associative flow), or Drucker-Prager yield criteria.
- Elastic-plastic Mohr-Coulomb hardening model, with hardeing parameters based on cohesion and/or friction angle.
- Critical state based models -Cam clay, modified Cam clay, Schofield's (incorporating a Mohr-Coulomb failure surface), and the Three Surface Kinematic Hardening model
- Duncan and Chang's Hyperbolic model

1.1.3 Element Types:

Linear strain triangle, cubic strain triangle, linear strain quadrilateral and the linear strain hexagonal (brick) element - all with additional excess pore pressure degrees of freedom for consolidation analysis. Beam, bar (tie) and interface (slip) elements.

1.1.4 Non-linear Techniques:

A Modified Newton Raphson iterative scheme is available as well as the Incremental (tangent stiffness) scheme. Implicit time stepping with $\theta = 1$ for consolidation analysis.

1.1.5 Updated Lagrangian analysis

This is to account for large deformation. Two options are available: simple update of geometry coordinates at the end of each increment, or the more advanced option of accounting for stress change due to change of geometry.

1.1.6 Boundary Conditions:

Prescribed incremental displacements or excess pore pressures on element sides. Nodal loads or pressure (normal and shear) loading on element sides. Automatic calculation of loads simulating excavation or construction when elements are removed or added.

1.1.7 Miscellaneous:

Parametric capablility to compare results across different (related) analyses, stop-restart facility to permit continuation from a previous run using disk file storage. Mesh data checked for errors prior to analysis using a separate program. Selected increments of an analysis written to disk for subsequent post- processing.

1.2 Historical Development

CRISP was written and developed by research workers in the Cambridge University Engineering Department Soil Mechanics Group, starting in 1975. Mark Zytynski wrote the first version of the program and was responsible for the original system design (Zytynski, 1976). He adopted in his work many conclusions from earlier research workers who had implemented critical state models in finite element programs (Simpson 1973; Thompson 1976; Naylor 1975). Mike Gunn (since 1977) and Arul Britto (since 1980) have been responsible for a considerable number of enhancements and modifications to CRISP. Some enhancements to the program has originated from the work of other members of the Soil Mechanics Group (John Carter, Nimal Seneviratne and Scott Sloan).

The program was originally called “MZSOL” and in 1976 it was renamed “CRISTINA”. The 1980 version of the program was called “CRISTINA 1980” and early in 1981 it was renamed “CRISP” (Critical State Program). Following several different releases for mainframe use in 1982, 1984 and 1988 (used mainly by universities and research organisations), a version of the program specifically for use on Intel-based PC computers was released in 1990. Known as CRISP-90, this PC version offered the same computational facilities as the 1988 release, but with a semi-graphical user interface and data generation capabilities. Extensions to this interface (principally the post-processing) led to further versions being released in 1992, ‘93 and ‘94.

CRISP was launched in 1995, with a completely new user interface written under Microsoft Windows. The program capabilities (element types, constitutive models, etc.) were the same as for CRISP-90.

1.3 Program Structure

Most finite element analysis "engines" (i.e. the parts in between the pre- and post-processing modules) comprise just one computer program. The CRISP package, however, uses two completely separate programs: the "Geometry Program" (GP) and the "Main Program" (MP). Essentially, the Geometry Program deals with mesh processing and checking whilst the Main Program solves the finite element equations. Although this process may take slightly longer than with more conventional programs, CRISP has been consciously designed in this way to help the user avoid some of the more common pitfalls in finite element analysis. The input data, which you must supply to CRISP, can be divided into the following categories:

- Information describing the finite element mesh (i.e. the co-ordinates of nodal points associated with each finite element)
- Material properties and in situ stresses for each finite element
- Boundary conditions and loading sequences for the analysis (i.e. imposed displacements, applied loads, geometry changes, etc.)

1.4 Program Limitations

See Section 8.1 for further details about large displacement analysis capability

CRISP is a finite element program which is able to perform drained, undrained and time dependent analysis of static problems (not dynamic) under monotonic loading/unloading conditions. It is not suitable for stress cycling in its present form, nor is it capable of handling partially saturated conditions.

Plane strain, axisymmetric and three-dimensional analysis can be carried out. Axisymmetric analyses are limited to loading that is also axisymmetric, hence torsion loading cannot be modelled using the axisymmetric option.

For example, CRISP in its present form could not be used to model the driving of a pile as this is a dynamic event. However, it could model the behaviour of a pile which was already installed. Furthermore, axisymmetric analysis could be used if the pile loading was purely axial, otherwise full 3D would be required.

See Section 8.2 for further details about applying stress corrections

CRISP uses the incremental (tangent stiffness) approach without any stress corrections when critical state based models are employed. This means that if the number of increments is insufficient the response will drift away from the true solution. For elastic-perfectly plastic models, however, when yielding occurs the stress state is corrected back to the yield surface at the end of every increment. The un-balanced load arising from this stress correction is re-applied in the subsequent increment. Additionally, at the end of every increment the strains are sub-divided into smaller steps and the stress state is re-evaluated more accurately. Ten steps are used for each increment. Therefore any analysis which only uses elastic-perfectly plastic (or wholly elastic) models requires fewer increments than a similar CSSM analysis.

If both CSSM models and elastic-perfectly plastic models are employed in the same analysis, the number of increments required will be governed by the CSSM behaviour, i.e. you need to use the same number of increments as for an analysis with only CSSM models.

It is entirely your responsibility to decide whether or not the models available in CRISP are adequate to represent the soil behaviour encountered in a real field situation. Some known deficiencies of the available models are as follows:

- Small-strain behaviour may not be adequately modelled. Very large stiffnesses are reported (Jardine et al, 1984; Powrie, 1986) at small strain levels and also at load reversal. The Three

Surface Kimematic Hardening model (3-SKH) has been introduced to allow the simulation of large stiffness at small strain levels.

- The constitutive models currently available in CRISP are not suitable for analysing partially saturated soils.

1.5 Use Of Engineering Judgement

Never take it for granted that any results produced from a finite element program are implicitly correct. Unless supported by an independent source (theoretical analysis, experimental observation, field data, results from other finite element programs) CRISP results should be treated with caution.

It is very difficult to assess the results of a CRISP analysis if you have not used the program for this particular type of problem before and virtually impossible if you have not used CRISP at all. Assuming that you have covered the general background material (see Section 1.6) the next step is to identify and consult published references on the application of CRISP to related problems. In this respect, users should find the CRISP Publications Directory particularly helpful. If others have used CRISP to carry out similar analyses and have compared the results with field data or laboratory model tests (or simply used them in a parametric study) then this will support the appropriateness of your intended application. Tips on suitable finite element meshes, number of load increments, application of boundary conditions, etc. can all be extremely useful. If, in contrast, it is difficult to find any reference to your particular type of problem it could indicate that you are trying to use CRISP to do something for which it was never intended.

Next, you should question the suitability (or otherwise) of the constitutive models available in CRISP for representing the soil in your problem - whether it is naturally occurring in the field, or has been remoulded for use in laboratory experiments. Review the literature to see how others have represented the same type of soil taking note of any difficulties they encountered. Care is needed to ensure that the soil parameters reported are relevant to the situation you are contemplating. Different types of structures on the same soil can lead to completely different behaviour. For example, construction of an embankment is likely to cause the soil to behave in an entirely different manner to excavation behind an embedded retaining wall, even in the same soil type. Soil behaviour can be (and often is) stress path dependent. The strength and stiffness parameters for a plane strain analysis may be quite inappropriate for an axisymmetric analysis of the same soil. Carry out single element tests to replicate the stress paths likely to be followed in different areas around the structure, and compare them with laboratory test data. If the soil representation is inadequate, the results will be unsatisfactory regardless of however carefully you carry out the analysis.

THE MAIN SOURCE OF ERROR IN A FINITE ELEMENT ANALYSIS IS INADEQUATE CHECKING OF DATA.

1.6 Further Sources Of Information

Before attempting to use CRISP, it is advisable that you acquire some knowledge of Critical State Soil Mechanics and of the Finite Element (Displacement) Method. Both these topics are covered in the book **Critical State Soil Mechanics via Finite Elements** (Britto and Gunn, 1987) which may be helpful. Further information on Critical State Soil Mechanics can be found in a number of texts (Schofield and Wroth, 1968; Atkinson and Bransby, 1978; Bolton, 1979; Atkinson 1981 and 1993; Muir-Wood, 1991). A large number of introductory texts are now available which describe the finite element method as applied to problems of linear elasticity (Fagan, 1992; Cook, 1995). Techniques for non-linear finite element analysis are described in texts by Zienkiewicz (1977) and Desai and Abel (1972). The programming of the Finite Element Method is covered in texts by Hinton and Owen (1977) which covers linear problems and Owen and Hinton (1980) which covers non-linear problems.

All papers, technical notes, etc. on the use of CRISP have been compiled into a single volume known as the CRISP Publications Directory which is supplied with CRISP. All users of CRISP should find this an invaluable source of reference material.

1.7 How To Use This Manual

The CRISP Technical Reference Guide has been devised to assist you, the user, in finding relevant information on setting up and analysing a problem as conveniently as possible. The material covered has been divided up into logical sections:

Chapter 2 provides a description of the basic principles and conventions which must be followed when setting up an analysis with CRISP.

Chapter 3 discusses how to set up a finite element mesh and gives some advice on the decisions which a new and inexperienced user will have to make.

Chapter 4 focuses on constitutive models and selecting the parameters which go into them.

Chapter 5 details all of the initial conditions which have to be specified, in terms of in situ stresses and boundary conditions for displacement and loading.

Chapter 6 describes the information required to define the finite element analysis itself - applied loading, excavation and filling operations, installation of structural members, change in drainage conditions, pore pressure equalisation, etc.

Chapter 7 explains the output which is produced by CRISP, and assists you in the interpretation of an analysis. Particular emphasis is given to those parts of the output which provide a measure of the accuracy and reliability of the analysis.

Chapter 8 is a collection of topics of special interest - program features which may not be commonly required, or the theoretical details of items discussed earlier on in the manual.

The References are for those authors specifically cited in the text, they are not a comprehensive listing of known applications of CRISP.

Appendix A provides listings of the input data file structures for the Geometry and Main Programs respectively, and will be familiar to users of earlier versions of CRISP.

Appendix B comprises an extended listing of some of the data records given in Appendix A.

Appendix C gives a complete list of error and warning messages from the Geometry and Main Programs.

Appendix D lists and describes all of the files created by/ used by CRISP.

BASIC PRINCIPLES

2

2.1 Co-ordinate System

It is recommended that you adopt the co-ordinate system shown in Figure 2-1 with the y axis pointing upwards, and the x axis pointing to the right If the x axis points to the left then the program will calculate element areas and stiffness as negative quantities. This is because CRISP expects element node numbers to be listed in an anticlockwise sense. In principle it is possible to use a co-ordinate system with the x axis pointing to the left but then it would be necessary to list element node numbers in a clockwise sense and a different sign convention for shear stresses would be needed.

You may rotate the co-ordinate system if desired (i.e. so that the y axis no longer points vertically upwards) but it should be noted that the following input options for the program will not work in the normal fashion:

- Specification of material self weight loads relevant to excavation, construction and gravity increase.
- Elastic properties varying linearly with depth.
- Axisymmetric analysis.

When the axisymmetric analysis option is selected it is assumed that the y axis is the axis of symmetry (i.e. the x axis is the radial direction).

In 3D analysis the z axis points outward from the plane of the paper to complete a right handed co-ordinate system. The z co-ordinate of a node is specified only for 3D analysis. Interchanging or rotation of this set of axes is not allowed for the 3D analysis.

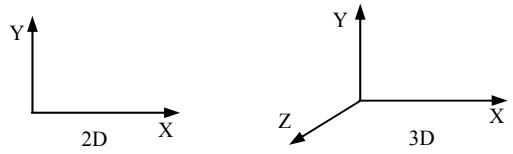


Figure 2-1 *Co-ordinate Axes for 2D and 3D*

2.2 Stress System

CRISP provides for 3 different stress systems (Figure 2-2 to

Figure 2-4) :

- Plane strain (2D)
- Axisymmetry (2D)
- General (3D) (Cannot be modelled with CRISP Windows interface)

Many geotechnical problems approximate to plane strain (strip footings, long excavations, highway embankments, etc.) and this particular stress system is probably used the most frequently, Figure 2-2. With respect to the out-of-plane stress, in the field, before civil engineering construction, it is reasonable to assume that horizontal stress is the same in all directions (i.e. $\sigma_z = \sigma_x$). During the analysis, σ_z can vary as it chooses, and in general $\sigma_z \neq \sigma_x$. τ_{xy} is the only shear stress of interest, as both τ_{yz} and τ_{zx} are both implied as zero.

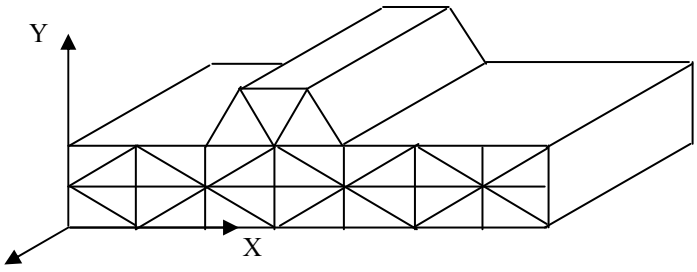
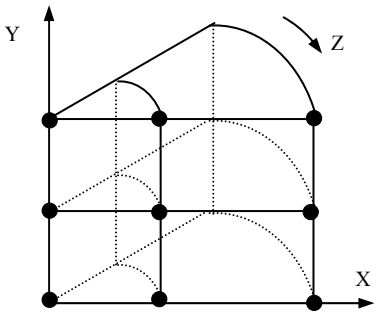


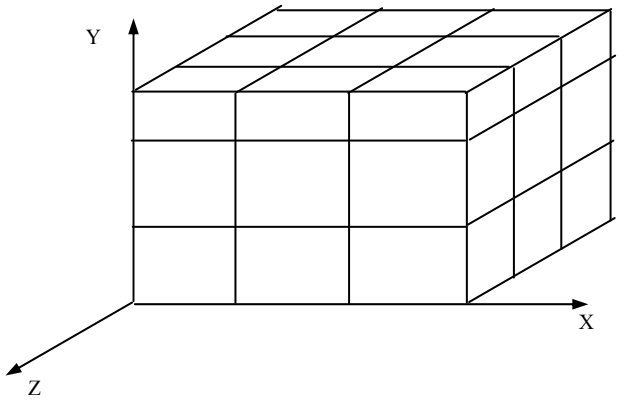
Figure 2-2 Plane Strain Stress System

Axisymmetry is confined to a narrower range of problems - circular shafts, single piles, cylindrical soil samples etc., Figure 2-3. Nonetheless, it is a useful pseudo-3D stress system. In axisymmetry, classical texts denote σ_x, σ_y, and σ_z as σ_r (radial), σ_z (vertical), and σ_θ (circumferential) respectively. In CRISP, σ_x, σ_y and σ_z are retained, so you need to be clear which direction is being referred to. In field problems, $\sigma_x = \sigma_z$ ($\sigma r = \sigma_\theta$) to begin with, but need not be equal once loading etc. commences. τ_{xy} (τ_{rz}) is the only shear stress of interest.



Finally, the completely general stress system may be required if truly 3D problems have to be analysed,

. In virtually all field problems, $\sigma_x = \sigma_z$ and $\tau_{xy} = \tau_{yz} = \tau_{zx} = 0$ at the initial stage; thereafter, changes may occur as dictated by the problem at hand. It is normally the geometry which demands three-dimensionality in geotechnical problems, rather than the original stress state.



2.3 Units

Refer to Section 7.4.1 in the User Guide for details about how to change the base system of units

You can choose any appropriate unit of length for specifying the co-ordinates of nodal points. It is important, however, that the units chosen to describe material properties, stresses and loads in CRISP are consistent. In a drained or undrained analysis you can only select the units for two quantities independently - the units for describing all other items are then automatically determined. Since the unit of length is always determined by the co-ordinate data you have one choice remaining and this can most simply be regarded as relating to the units of force that are to be used. For example, if length and force units are chosen to be metres (m) and kilo-Newtons (kN) respectively then stresses and elastic moduli must be in kN/m^2 and unit weights must be in kN/m^3.

Table 2-1 *Consistent Set of Units*

Units

Unit Name	Consistent Sets			
length	m	mm	mm	Ft
force	kN	N	mN (milli Newton)	Lbf
time	sec	sec	sec	Hr
angle	deg	deg	deg	Deg
acceleration	m/s²	mm/s²	mm/s²	ft/hr²
pressure, stress	kN/m²	kN/m²	mN/mm²	Lbf/ft²
unit weight	kN/m^3	kN/m^3	kN/m^3	Lbf/ft^3
permeability	m/s	mm/s	mm/s	ft/hr

When a consolidation analysis is performed suitable units of time must also be chosen and the units chosen for permeability imply certain units for time steps (e.g. if permeability has units of metres/year then time steps will be in years).

You should note that all data files created by CRISP will always be in base units of m, KN, seconds and degrees.

2.4 Sign Conventions

2.4.1 Load And Displacement

Loads and displacements are referenced to the global x and y (and, if appropriate, z) directions. A force with a positive x component is one which is acting in the positive x direction (i.e. from left to right); a negative y displacement is one acting in the opposite direction to positive y (i.e. downwards).

2.4.2 Stress And Pressure

Stress, following the standard geotechnical convention, is positive when compressive. One would expect to see compressive internal stresses in all soil elements throughout an analysis, so negative stresses should be investigated very carefully - unless they are in steel, concrete or rock material zones. The linear elastic models are capable of sustaining tensile stresses, as may the elastic-perfectly plastic models, depending on the specified cohesion value.

Applied normal stresses are taken as positive if they act on an element edge in towards the element centre. Applied shear stresses are taken as positive if they act on element edges in an anticlockwise direction about the element centre. Figure 2-5 illustrates these definitions.

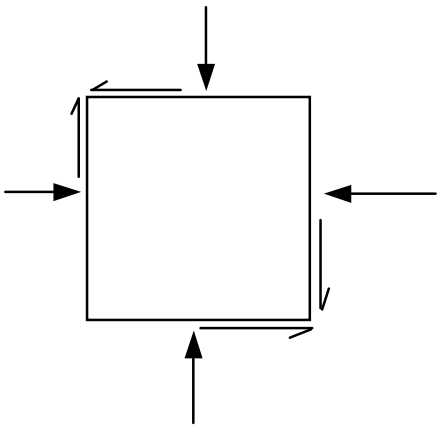


Figure 2-5 *Conventions for Positive Normal and Shear Stresses*

2.4.3 Strain

Normal strains are considered positive if they are compressive - i.e. tending to cause a reduction in volume. In contrast to stress, it is not unusual to find negative (tensile) strains in soil, such as the minor principal strain in an undrained triaxial test.

Shear strains are positive if the angle between orthogonal fibres at the origin of the element of soil are increasing, Figure 2-6.

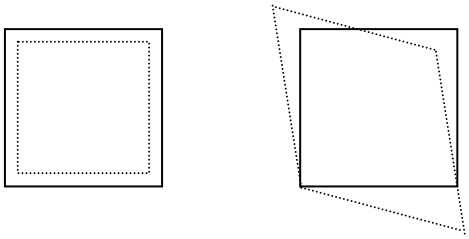


Figure 2-6 *Conventions for Positive Normal and Shear Strains*

2.5 Effective v Total Stress

One of the principal features of any dedicated geotechnical finite element package should be its ability to work explicitly in terms of effective stress and pore pressure. Soils are at least 2-phase materials (solids and water), and many constitutive models are developed within an effective stress framework (e.g.Cam clay). General purpose FE codes will normally work in total stresses, and whilst it is possible to use appropriate material properties and perform a geotechnical analysis, pore pressure changes will not be explicitly calculated.

CRISP expects to work in effective stresses. In situ stresses are defined thus, as are strength and stiffness parameters. If you wish to work in total stress terms, this is also possible (except for CSSM models, and any consolidation analysis) and more detailed instructions are given in later chapters.

2.6 Drained v Undrained

Drained conditions prevail when the rate of loading is much *slower* than the rate of drainage. Drained analysis may also be of interest to the engineer if he/she wishes to find the final state of deformation at which no further drainage takes place. For drained analysis, initial pore pressures do not change, as excess pore pressures have adequate time to dissipate and do not build up during loading. Volume change, however, may (and often does) occur. To summarise:

$$\Delta u = 0$$
$$\Delta \mathrm{V} \neq 0 \qquad \textit{Eqn. 2.6-1}$$

Undrained conditions prevail when the rate of loading is much *faster* than the rate of drainage. Initial pore pressures will change, as excess pore pressures do not have adequate time to dissipate and will build up during loading. Undrained analysis could also be viewed as the initial instant at which load is applied. Volume change, however, cannot occur. To summarise:

$$\Delta V = 0$$
$$\Delta u \neq 0 \qquad \textit{Eqn. 2.6-2}$$

Following undrained loading, a period of time-dependent pore pressure dissipation (or equalisation) may take place, if drainage paths exist. This is called consolidation if positive excess pore pressures are dissipating and volume is reducing, or swelling if negative excess pore pressures (suctions) are being satisfied and volume is increasing as water is taken in.

For many practical problems, it is accurate enough to assume that real behaviour corresponds to one or other of these limits (i.e. drained or undrained). However, an important class of problems exist for which loading and drainage rates are comparable. These are known as coupled loading-consolidation problems, and it is possible for excess pore pressures to be generated simultaneously with drainage and volume change (positive or negative). Total mean stress p may also vary during loading (unlike the classic 1D diffusion theory of Terzaghi, which stipulates mean total stress constant during consolidation). To summarise:

$$\Delta u \neq 0$$
$$\Delta V \neq 0 \qquad \textit{Eqn. 2.6-3}$$
$$\Delta p \neq 0$$

2.7 Material Zones

In principal, every single element in a finite element mesh could have different material properties, indeed, this is one of the main strengths of the method. In practice, ground cross-sections will be divided into a number of different strata or horizons - usually with horizontal boundaries, but often inclined - and it will be convenient to group elements together as lying in one or other of these real material regions (or zones).

CRISP permits the mixing not only of different soils, but also of soil with rock, concrete and other structural materials - with appropriate interface behaviour. An individual soil layer can be drained, undrained or partially drained (see 2.6 above) and there is no restriction as to how these may be mixed.

2.8 Analysis Identification

Each analysis may involve several different data files. These are all given the same filename as the CRISP Data file (the .SCD file) created by the Pre-Processor, but with various extensions that identify the origin/ purpose of the file. A listing of all these can be found in Appendix D, along with a brief description as to their purpose.

FINITE ELEMENT MESH

3

3.1 General

The finite element method (FEM) is an approximate method, and the choice of mesh can have a critical effect on the accuracy of the solution. In this chapter, we describe the element types which are available in CRISP, and offer some advice on how to design a reasonable mesh with them. One can be fairly precise in specifying the attributes of a particular element, but can only give general comments on how to create suitable FE meshes. There are simply too many different types of geotechnical problem to offer a comprehensive guide in a manual such as this. You are strongly advised to consult appropriate published examples and/or discuss your problem with an experienced analyst. In time, you will gain experience as you carry out more and more CRISP analyses, and will be able to make these judgements with increasing confidence.

3.2 Mesh Boundaries

3.2.1 Boundary Location

The outer boundaries of your mesh should be placed sufficiently far away from the region subjected to the greatest load changes so as not to influence the results. An appreciation of likely stress gradients is helpful, such as the "pressure bulbs" seen underneath a footing. If you can place the boundaries beyond regions where stress changes are less than 5% (ideally 1%), then you are unlikely to introduce significant errors into the analysis from this source. For example, consider the analysis of a retaining wall. If H is the height of the retaining wall, then, as a general rule, the vertical boundaries should be placed at a distance of at least 4 to 5H from the wall. Similarly for the base of the mesh. However, if a hard stratum is encountered at a depth less than this, then the base of the mesh can be fixed coincident with the hard layer. This assumes that the displacements which may occur in the hard layer (due to the construction activities above) are negligible.

In setting up the mesh, make use of any existing axes of symmetry, (see Figure 3-1) in order to reduce computational time and resources.

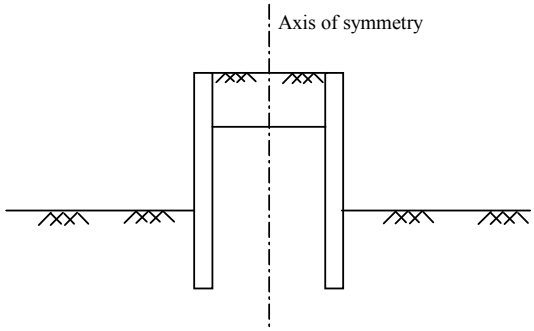


Figure 3-1 *Exploiting Axes of Symmetry*

3.2.2 Boundary Roughness

If the outer boundaries are placed far enough away so as not to influence the results, it should not matter whether the boundaries are treated as rough or smooth. It is good practice to vary the fixity on the mesh boundaries, as a means of assessing the adequacy of mesh boundary distance. The specification of boundary fixity is described more fully in Section 6.4.

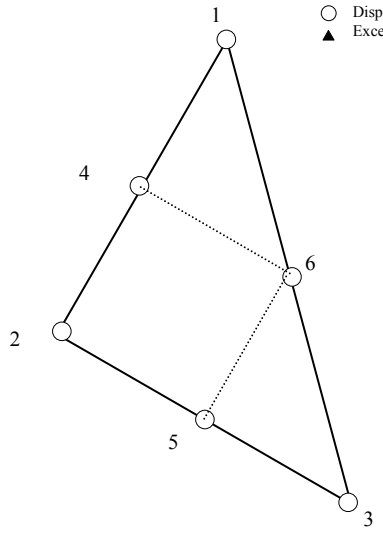
3.3 Element Types

The variation of displacements, strains and (where appropriate) pore pressures are summarised in Table 3-1. All the elements are standard displacement finite elements, are illustrated in Fig. 3.8, and are described in most texts on the finite element method (e.g. Zienkiewicz, 1977).

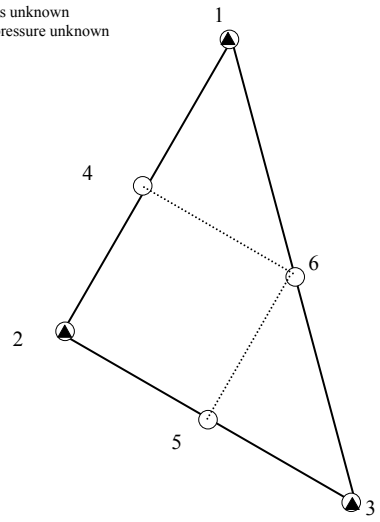
Table 3-1 *Available Element Types*

Name	Crisp Element ID No.	Displacement	Strain	Excess Pore Pressure	Integration Points
Linear Strain, 3 node Bar	1	Quadratic	Linear	N/A	5
Linear Strain, 2 node Bar	14	Quadratic	Linear	N/A	5
Linear Strain Triangle (LST)	2	Quadratic	Linear	N/A	7
Linear Strain Triangle with excess pore pressure d.o.f. (LSTp)	3	Quadratic	Linear	Linear	7
Linear Strain Quadrilateral (LSQ)	4	Quadratic	Linear	N/A	3x3
Linear Strain Quadrilateral with excess pore pressure d.o.f. (LSQp)	5	Quadratic	Linear	Linear	3x3
Cubic Strain Triangle (CuST)	6	Quartic	Cubic	N/A	16

Name	Crisp Element ID No.	Displacement	Strain	Excess Pore Pressure	Integration Points
Cubic Strain Triangle with excess pore pressure d.o.f. (CuSTp)	7	Quartic	Cubic	Cubic	16
Linear Strain Hexagonal (brick) LSH	8	Quadratic	Linear	N/A	3x3x3
Linear Strain Hexagonal with excess pore pressure d.o.f. (LSHp)	9	Quadratic	Linear	Linear	3x3x3
Linear Strain, 2 node Beam	15	Quadratic	Linear	N/A	5
Linear Strain, 3 node Beam	12	Quadratic	Linear	N/A	5
Linear Strain (Total Stress) interface element	13	Quadratic	Linear	N/A	5
Linear Strain Effective Stress interface element	20	Quadratic	Linear	Accounts for excess pp from consolidation element on one side	5
Linear Strain 3D interface element (fits between 2 brick elements)	21	Quadratic	Linear	N/A	3x3

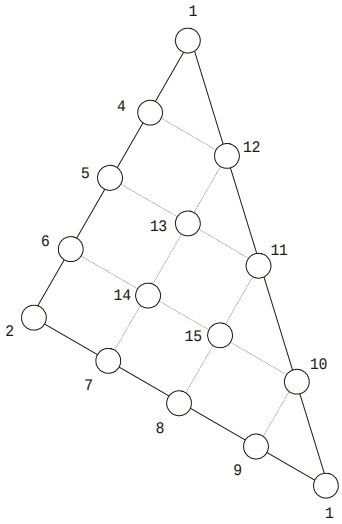


(i) LST (element type 2)
6 nodes, 12 d.o.f
(for drained/undrained analysis and zones)

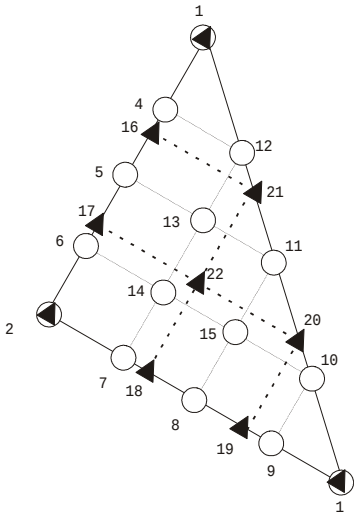


(ii) LST (element type 3)
6 nodes, 15 d.o.f
(for consolidation analysis and zones)

Figure 3-2(a) *Finite Element Types*

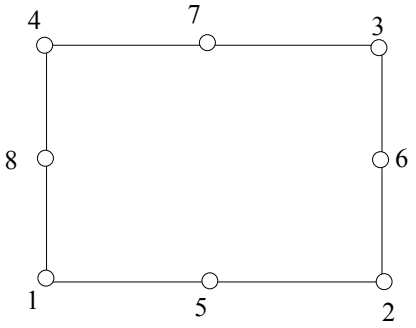


(i) CuST (element type 6)
15 nodes, 30 d.o.f
(for drained/undrained analysis and zones)

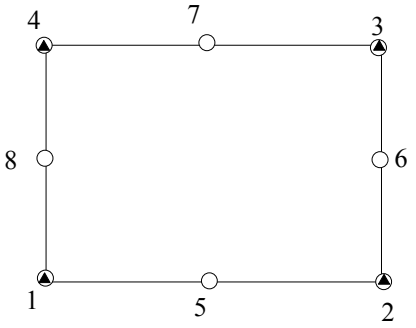


(i) CuSTp (element type 7)
22 nodes, 40 d.o.f
(for consolidation analysis and zones)

Figure 3-2(b) *Finite Element Types*

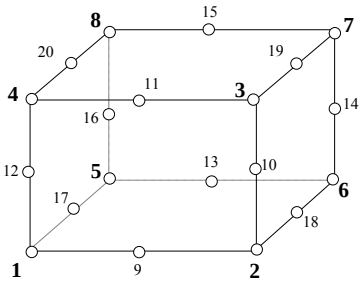


(i) LSQ (element type 4)
8 nodes, 16 d.o.f
(for drained/undrained analysis and zones)

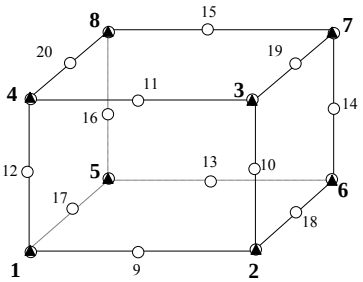


(ii) LSQp (element type 5)
8 nodes, 20 d.o.f
(for consolidation analysis and zones)

Figure 3-2(c) *Finite Element Types*

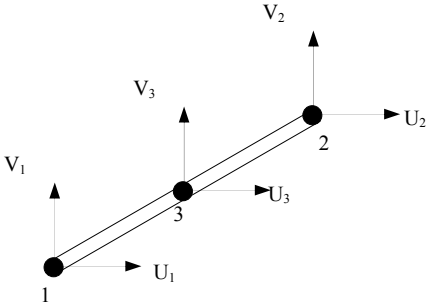


(i) LSB (element type 8)
20 nodes, 60 d.o.f
(for drained/undrained analysis and zones)

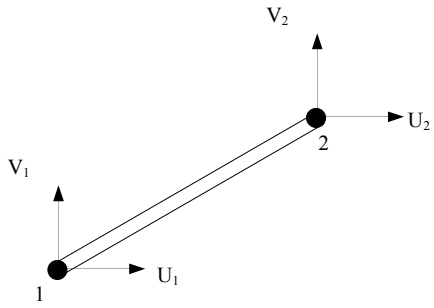


(i) LSBp (element type 9)
20 nodes, 68 d.o.f
(for consolidation analysis and zones)

Figure 3-2(d) *Finite Element Types*

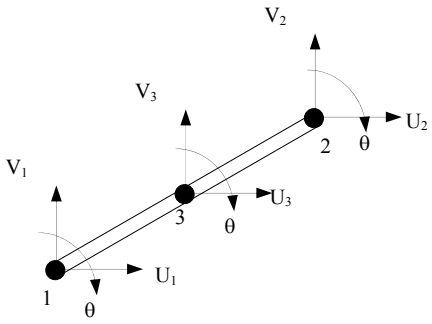


(i) 3 Node Bar (element type 1)
6 d.o.f, axial stress only

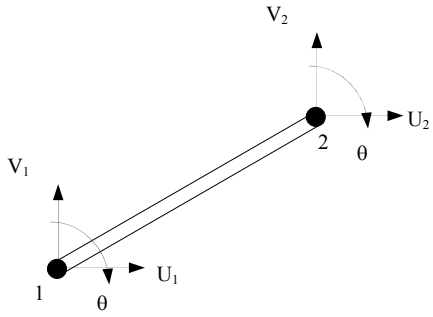


(ii) 2 Node Bar (element type 14)
4 d.o.f, axial stress only

Figure 3-2(e) *Finite Element Types*

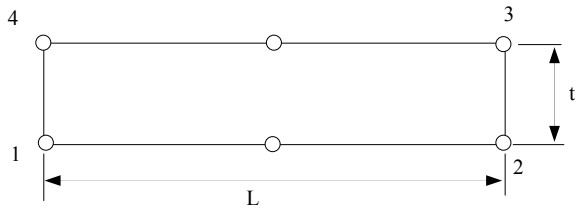


(i) 3 Node Beam (element type 15)
9 d.o.f, axial stress only



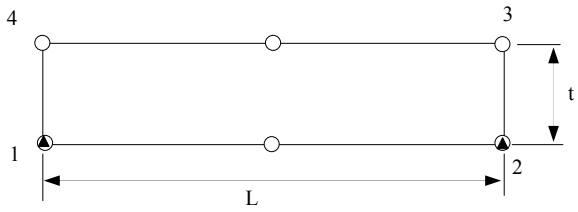
(ii) 2 Node Beam (element type 14)
6 d.o.f, axial stress only

Figure 3-2(f) Finite Element Types



Joint interface (slip) element (element type 13)

Figure 3-2(g) *Finite Element Types*



Consolidation "Effective Stress" Joint interface (slip) element
(element type 13). Notice excess pore pressure nodes on one side

Figure 3-2(h) *Finite Element Types*

Refer to Section 3.7 for details about permissible mixing of element types

Although CRISP allows you complete freedom in the choice of element type, experience tends to suggest that the particular types of element are applicable to particular problems as recommended below:

(i) For Plane Strain analysis:

For drained or undrained analysis use Linear Strain Triangular or Quadrilateral elements (ID 2 or 4), for coupled loading and consolidation analysis use type 3 or 5.

(ii) For Axi-symmetric analysis:

For drained analysis or consolidation analysis where collapse is not expected then Linear Strain Triangular or Quadrilateral elements (ID 2 to 5) will probably be adequate (i.e. the same as (i) above). For undrained analysis or situation where collapse is expected then the higher order Cubic Strain Triangular elements (ID 6 and 7) are recommended. Research has shown that, in axisymmetric analyses, the constraint of no volume change (which occurs in undrained situations) leads to finite element meshes "locking up" if low order finite elements such as the LST and LSQ are used (Sloan and Randolph, 1980).

(iii) Three-Dimensional analysis

For drained or undrained analysis use the Linear Strain Hexagonal 'brick' element (ID 8). For coupled-consolidation analysis use the equivalent hexagonal consolidation element (ID 9).

3.4 Number Of Elements

It is difficult to lay down hard and fast rules for the number of finite elements needed to analyse a particular problem. The following hints may assist inexperienced analysts:

(a) Avoid using too few elements - remember that in the case of the linear strain triangle, for example, stresses will vary linearly across the element.

(b) On minimum specification machines, avoid using too many elements - in most situations between 100 and 200 LSTs will be adequate as will between 30 and 50 CUSTs.

(c) The mesh should be finer (i.e. elements should be smaller) in regions where rapidly varying strains/stresses are to be expected (e.g. near loaded boundaries).

In general about 100 to 200 Linear Strain Trianglular or Quadrilateral (ID 2, 3, 4 or 5) elements would be sufficient if these are deployed sensibly giving a graded mesh. More elements may be needed if the problem analysed is complex (geometrically or in terms of stress-strain behaviour).

An attempt should be made to analyse the problem with 2 different meshes, one being finer than the other (i.e. with more elements or using a higher order element). A comparison of the results (characterised perhaps by a key displacement or stress value) would indicate whether a sufficient number of elements (d.o.f.s) are being used in the analysis. Theoretically, an increasingly finer mesh should produce results which tend towards the true solution, although this can only be checked where analytical solutions exist.

Whilst the actual number of elements used in the mesh is important, the critical point is how they are deployed. Regions which are subjected to loading or unloading are of principal interest, and you should concentrate a good percentage of the finer (smaller) elements around this region, plus around any expected stress concentrations, such as around the re-entrant corner of the mesh in Figure 3-3. Use slightly larger elements (making the transition gradual) as you move away from the region of principal interest. Somewhat large elements can be used at boundaries distant from loaded regions.

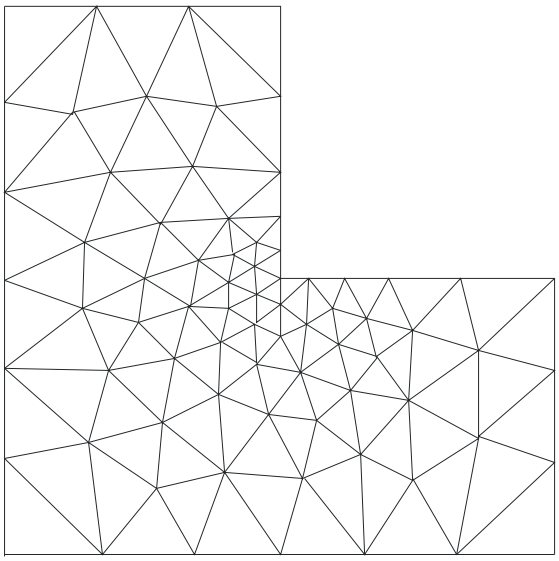


Figure 3-3 *A Well Graded Finite Element Mesh*

Rapid variation in stresses and strains and displacement takes place around loaded region. When preparing the mesh you need to ask whether the chosen type and size of the element are adequate to represent the expected variation in strain. An illustration is given figure below , which shows the effect of number of elements used in analysing a thick cylinder - a problem for which the analytical solution is available. *(NB: this example uses constant strain triangles (CSTs), which CRISP does not provide - the CST is a notoriously poor element, so this example is probably a little extreme.)*

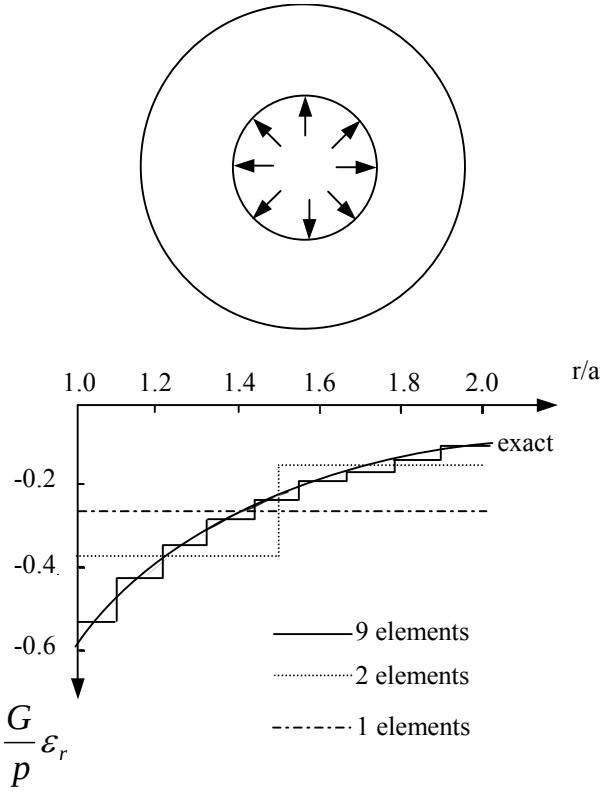


Figure 3-4 *Effect of Number of Elements*

3.5 Size And Shape Of Elements

The finite element method approximates the distribution of an unknown variable in a precise manner across the body concerned. These distributions are only reliably produced if the shapes of the elements are not excessively distorted. Ideally, the elements should all be as 'regular' as possible. The allowable limits of distortion are difficult to quantify, and depend very much on the variable distribution that the elements are representing. If the variable is nearly constant, then even large distortions will not produce significant errors; conversely, rapid changes in variable are most sensitive to element shape.

One measure of element distortion is aspect ratio, defined as the ratio of the longest side of an element to the shortest side. An alternative method of assessing element distortion is to consider the internal angles of the elements. You are recommended to consult the finite element literature for further details on this matter - Fagan (1992) is useful reading - and/or look to see what other CRISP users have done for similar problems.

3.6 Automatic Mesh Generation

See Section 8.1 in the User Guide for further details about automatic mesh generation

CRISP provides two means of automatically generating finite element meshes - the *unstructured mesh generator* and the *structured mesh generator*. Each of these operates on a super mesh concept in which you create regions (super elements) inside which finite elements will be created according to your specified grading requirements.

These generators are particularly useful when you want to build large or complex meshes.

3.7 Mixing Element Types

3.7.1 General

In many situations, the mesh will comprise just one element type, but the possibility exists of mixing different element types in CRISP. Element types can be mixed within (and only within) one of three different groups, see Table 3-2.

Table 3-2 Permissible Mixing of Element Types

Group	Element type	ID in Crisp
A	Beam, bar, LST, LSQ, and Slip (Joint) elements	1, 2, 3, 4, 5, 12, 13, 14, 15, 20
B	High order Cubic Strain Triangles	6, 7
C	3D Hex elements and 3D Slip (joint) element	8, 9, 21

All of the element types within a particular group can be present in the same mesh if desired. Element types from one group cannot be mixed with element types from any other group. For example you cannot mix element a high order Cubic Strain Triangle (group B) with a Beam element (group A). The basic requirement is to have the same number of nodes along common element edges.

3.7.2 Consolidation Analysis

As we shall see in Chapters 4 and 6, the type of element being used dictates some (not all) of the parameters which need to be specified in a material zone. For example, the consolidation elements LST, LSQ, CuST and LSHex (3, 5, 7 and 9) will expect permeability to be defined in material properties and non-zero time steps to be defined in the analysis increments. The non-consolidation elements (2, 4, 6 and 8) do not require this. Hence, legally mixed non-consolidation elements (e.g. 2 and 4) can all have the same material zone number in a mesh, but if compatible consolidation elements are also present, they must be used within a separate material zone (and vice versa).

Common examples of mixing include:

(a) Interbedded layers of free draining sand and consolidating clay.

(b) Impermeable inclusions (e.g. a diaphragm wall) in permeable soil undergoing time-dependent pore pressure changes.

In example (a) you could use element types 2 and 4 (or 6 or 8) for the sand, and types 3 and 5 (or 7 or 9) for the consolidating clay. The sand could be zone 1 and the clay zone 2 (unless there were

significant variations of strength etc. in either or both layers, when two or more zones would be needed for the layer(s) concerned).

In example (b), the clay would need element types 3 and 5 (or 7 or 9), whereas for the wall you have two choices. Either you can use the same (consolidation) element types and set permeability to a lower value (e.g. x 1/1000) than that of the clay or (more simply) use non-consolidation elements 2 and 4 (or 6 or 8) and have them as undrained.

In both examples, there are implications for drainage boundary conditions, and this is dealt with more fully in Section 6.5.

3.8 Elements With Curved Sides

The CRISP Geometry Program calculates the co-ordinates of nodes along sides and within element interiors by linear interpolation between vertex nodes, assuming that the elements are straight edged. However in some analyses (e.g. circular tunnels, buried pipes) it is more appropriate for the element sides to be curved to accurately model the physical geometry. As CRISP does not have the facility to calculate co-ordinates of nodes along element sides that are curved, you must specify directly the co-ordinates of the intermediate nodes.

3.9 Bar And Beam Elements

3.9.1 Bar Elements

The 2 node and 3 node bar elements provided in CRISP have displacement degrees of freedom (only) and are basically limited to carrying axial force. Both ends make pinned connections with other elements and thus cannot transmit moments.

Bar elements can be used to simulate props, struts, layers of geotextile reinforcement, etc.

The 2 node bar cannot be placed alongside triangular or quadrilateral elements as the requirement for nodes along common element edges to be compatible will not be met.

It is appropriate to use the 3 node bar to simulate a tie-back or similar structure where the bar must lie between triangular or quadrilateral elements.

If the 3 node bar is used as a strut, it is necessary to fix the mid-side node as well as the end nodes (see Section 6.4).

3.9.2 Beam Elements

The 2 and 3 node beam elements have rotational as well as displacement degrees of freedom at each node, and are therefore capable of transmitting moments as well as axial forces.

Beam elements can be used to simulate sheet pile walls in place of 2D continuum elements, thus avoiding excessively high aspect ratios.

As with the 2 node bar element, the 2 node beam element cannot be laid alongside continuum elements due to node incompatibility.

3.10 Interface Elements

3.10.1 General

The interface (or slip) element is a flat 8-noded element with two 'dummy' nodes midway along each of the narrow edges, see Figure 3-5. It can be used to simulate the interface between soil and structure whether it is smooth, or begins to slip after a limiting stress condition has been reached. Before attempting to use the slip element you should consider the possibility of using an 8 noded quadrilateral element to simulate a smooth interface (for example smooth end platens in a triaxial test). A very flat quadrilateral element with linear elastic properties and a very small G (shear modulus) has been found to give satisfactory results (Airey, 1987; Al-Tabbaa 1987), although care is required to avoid excessive aspect ratios.

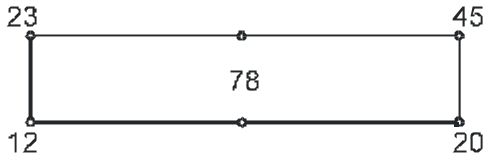


Figure 3-5 *Node Numbering of an Interface Element*

However if you require an interface behaviour which is rough (or adhesive) when the shear stress mobilised is less than the limiting value but which permits relative slip once this limiting stress is reached, then the slip element must be used.

3.10.2 Definitions And Use

The interface (slip) element is 3-noded along its long sides, and can only be used with other linear elements (LST, LSQ and 3 node bar and beam elements). The interface element is only available in 2D (not 3D) and has been validated only for plane strain analysis - its performance in axisymmetric analysis is, as yet, unproven.

Interface elements are treated as flat rectangular elements. They have 4 corner (vertex) nodes (each of whose node numbers are different). Even though it is an 8-noded element, the 2 mid-side nodes along the narrow dimension are dummy nodes (i.e. these nodes are not used in the analysis, even though these are assigned numbers). Interface elements should be placed such that they extend right through the mesh. You should not attempt to attach any other elements types (e.g. LST, LSQ, bar or beam elements) to either of its narrow sides. Any attempt to connect an interface element to any other element type along its width leads to lack of compatibility along its narrow side. The narrow sides can only be attached to another slip element, left free (exiting on a free boundary) or joining a fixed boundary as in Figure 3-6.

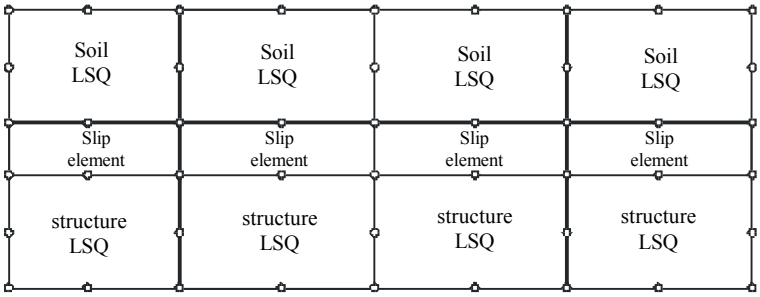


Figure 3-6 *Permissible use of Interface Elements*

In CRISP you create interface elements by selecting one or more 2D elements around which you wish to generate an interface. This interface layer is treated as a chain of interface elements connected end to end. For example in an embedded wall analysis, the interface elements should completely separate the wall from the soil, with a continuous layer stretching down the retained side, under the base, and up the excavated side of the wall, as in Figure 3-7.

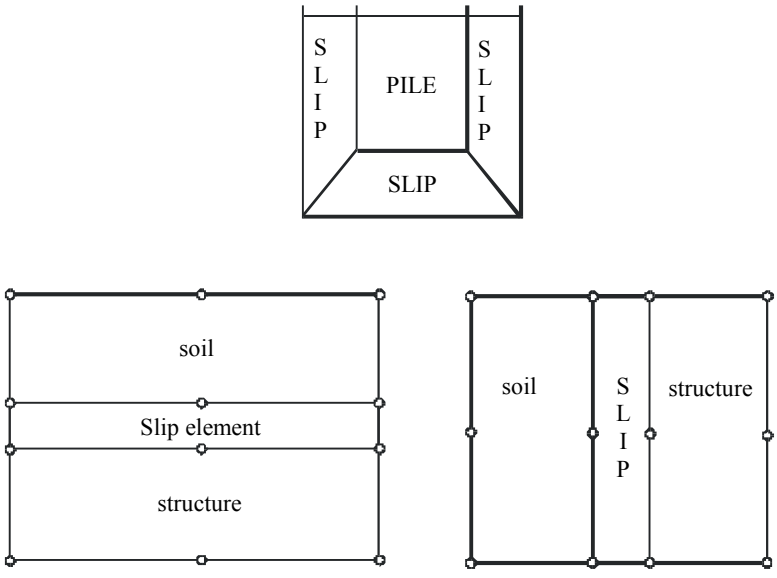


Figure 3-7 *Use of Interface Elements in Soil-Structure Interactions*

The slip element may be looked at as a one-dimensional element with 6 distinct nodes, 3 each in the direction of its length. Any node on one side along the length would have exactly the same coordinates as the node on the opposite side. But the slip element would have a thickness, t, which is specified in the material properties. The thickness is necessary for a proper formulation of the stiffness matrix.

3.11 Checking The Mesh

3.11.1 General

For validation purposes, the mesh checking features available in previous versions of CRISP are retained in CRISP. These are described in Sections 0 to 3.11.4 following.

See Sections 6.3 and 6.10 for details about the Graphics Window and the Cleanup option

The Graphics Window in CRISP provides the easiest interface for viewing and checking your mesh (see the User Guide). It is, however, also possible to use the text output file written by the CRISP Geometry Program. Provided you have retained this file (i.e. the cleanup option has not been actioned). This file has the extension .GPO and comprises general summary information, nodal co-ordinate data, and element connectivity information.

Note that this is currently the only way to check 3D mesh data in CRISP.

3.11.2 Summary Information

The output begins with the current version number of the Geometry Program, the date when it was last modified, the title of the analysis currently being carried out, and the amount of the storage array used.

The control parameters read in records B and C are echoed. The vertex node co-ordinates are printed out as the co-ordinates of each node are read. If an alternative element numbering scheme has been specified by the user, it is printed.

For the element numbering sequence specified by the user, CRISP calculates the maximum frontwidth for solution, the minimum core required to solve the equations and the additional store required for in-core solution and these are printed.

Finally the output from the Geometry Program is completed by the total number of nodes and degrees of freedom in the mesh being printed.

3.11.3 Nodal Data

The list of vertex nodes associated with each element is printed as they are read. This is followed by the listing of all mid-side displacement nodes numbered by the program. This is printed as each new node is numbered and its co-ordinates calculated. Displacement node co-ordinates along any curved sides are also read in and printed out.

The co-ordinates of additional excess pore pressure nodes are printed as they are calculated (if any are present). Finally, the Geometry Program reads excess pore pressure node co-ordinates along any curved sides and prints them out.

3.11.3.1 Two-dimensional analysis

The nodes can be divided into three categories, depending on the number of degrees of freedom

(1)	3 d.o.f	-	u, v, p	unknown
(2)	2 d.o.f	-	u, v	unknown
(3)	1 d.o.f	-	p	unknown

where u, v are the horizontal and vertical displacements, and p is the excess pore pressure

For a drained or undrained analysis (LST, LSQ, LSB) all nodes have 2 d.o.f and they belong to category (2). For consolidation analysis if LSTp elements have nodes belonging to categories (1) and (2). The vertex nodes are of category (1) and the nodes along element sides are of category (2).

For consolidation analysis using CuSTp elements, all three categories are present. The vertex nodes are of category (1). All other displacement nodes are of category (2) and the rest of the nodes belong to category (3). i.e. excess pore-pressure is the only unknown. In general for a consolidation analysis all vertex nodes are of category (1). The vertex nodes are numbered by the user. In the presence of nodes of category (2) and (3), all mid-side nodes are numbered, starting with a number that is one greater than the greatest existing vertex node number. Then all nodes of category (3) are numbered.

3.11.3.2 Three dimensional analysis

(1) 4 d.o.f. - u, v, w, p unknown

(2) 3 d.o.f. - u, v, w unknown

where u, v, w are the displacements in the x, y, and z directions respectively, and p is the excess pore pressure. The first category of nodes are the corner nodes (20 noded brick element) in a consolidation analysis. All the other nodes are of category (2).

3.11.4 Element-Nodal Links

A complete list of nodes associated with each element is printed out. The first column gives the element number, the second the element type number and the third the material zone number. The vertex nodes are printed in the anti-clockwise direction starting from the lower-left node. This is followed by the additional displacement nodes and finally if there are any additional excess pore pressure nodes these are also printed. The sequence of all these nodes for every element type are given in Figure 3-2.

CONSTITUTIVE MODELS AND PARAMETER SELECTION

4

4.1 General

Once the mesh has been designed and the different material zones created, the next stage is to assign specific properties to each zone. This is really a two-part process:

- Choose a constitutive model capable of reproducing those features of real behaviour thought to be most important
- Select appropriate parameters for the model, with respect to the intended application.

This is a very wide subject in its own right, and you are advised to consult the extensive literature on the subject. With a few possible exceptions (mentioned later) all the parameters should be regarded as being effective stress properties i.e. they either relate changes of strain to changes in effective stress, or they describe soil strength in terms of effective stresses acting in the soil skeleton.Linear and Non-Linear Elastic Models.

4.2 Linear and Non-Linear Elastic Models

This will present the material models based on the elastic constitutive relationships and the Duncan and Chang's hyperbolic model.

4.2.1 Homogeneous Anisotropic Linear Elastic

4.2.1.1 Basic Description

This is the standard elastic model which allows for cross (transverse) anisotropy. Fully isotropic behaviour is covered as a special case.

4.2.1.2 Stiffness Parameters

The stiffness parameters required are:

Table 4-1

E_h	Young's modulus in the horizontal direction
E_v	Young's modulus in the vertical direction
ν_{vh}	Poisson's ratio linking vertical and horizontal directions
ν_{hh}	Poisson's ratio linking both horizontal directions
G_{vh}	Shear modulus in v-h plane

4.2.1.3 Governing Equations

The anisotropic elastic properties relate strains to changes in stress via the following equations:

$$\varepsilon_x = \frac{1}{E_h}\sigma_x - \frac{\nu_{vh}}{E_v}\sigma_y - \frac{\nu_{hh}}{E_h}\sigma_z \qquad \textit{eqn. 4.2-1}$$

$$\varepsilon_y = -\frac{\nu_{hv}}{E_h}\sigma_x + \frac{1}{E_v}\sigma_y - \frac{\nu_{hv}}{E_h}\sigma_z \qquad \textit{eqn. 4.2-2}$$

$$\varepsilon_z = -\frac{\nu_{hh}}{E_h}\sigma_x - \frac{\nu_{vh}}{E_v}\sigma_y + \frac{1}{E_h}\sigma_z \qquad \textit{eqn. 4.2-3}$$

$$\gamma_{xy} = \frac{1}{G_{hv}}\tau_{xy} \qquad \textit{eqn. 4.2-4}$$

Note that suffixes h (for horizontal) and v (for vertical) have been adopted here to clarify the type of anisotropic properties which CRISP expects to be specified for soil. This is because soil deposits are often formed by a process of sedimentation in horizontal layers and the associated soil fabric and stress history lead to one set of properties for the x-z (or h) plane (E_h and ν_{hh}) and another set relating to the vertical direction (v or y) and the coupling between horizontal and vertical directions (E_v, ν_{hh}, ν_{hv} G_{hv}).

The significance of these properties can be deduced from the above equations. Basically, an increase in vertical stress leads to an increase in vertical strain $\Delta\sigma_y/E_v$ and a tensile strain $(\nu_{vh}/E_v)\Delta\sigma_y$ (in the absence of any changes in horizontal stresses). Hence ν_{vh} is the Poisson's ratio which gives the ratio of horizontal strain to vertical strain caused by a stress increment in the vertical direction, and a similar statement can be made as to the meaning of ν_{hv}.

Note however, that CRISP requires only the specification of ν_{vh} and not ν_{hv}. This is because energy/reversibility considerations for an elastic material lead to the relationship:

$$\frac{\nu_{hv}}{E_h} = \frac{\nu_{vh}}{E_v} \qquad \textit{eqn. 4.2-5}$$

4.2.2 Nonhomogeneous Isotropic Linear Elastic

4.2.2.1 Basic Description

This model is particularly suited to soil materials, where increasing confining pressure (with depth) often leads to a linearly increasing stiffness with depth. Poisson's ratio ν is not allowed to vary, so shear modulus G will be linked to Young's modulus through a constant ratio. This model is sometimes referred to as 'Gibson soil', due to the fact that some of its surprising properties were highlighted by Gibson in his 1974 Rankine Lecture.

4.2.2.2 Stiffness Parameters

The stiffness parameters required are:

Table 4-2

E_0	Young's modulus at the datum elevation
m	Rate of increase of Young's modulus with depth
y_0	Datum elevation at which $E = E_0$
ν	Poisson's ratio

4.2.2.3 Governing Equations

The elastic Young's modulus at an elevation y is given by the equation:

$$E = E_0 + m(y_0 - y) \qquad \textit{eqn. 4.2-6}$$

where all the terms are defined in the above table, and in Figure 4-1. The datum elevation should be fixed at some convenient level in the mesh (such as the ground surface, or the base of the mesh).

Shear modulus G will be forced to vary in the same way as E, through the standard relationship:

$$G = \frac{E}{2(1+\nu)} \qquad \textit{eqn. 4.2-7}$$

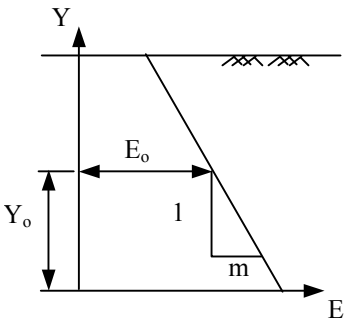


Figure 4-1 *Permitted Variation of Stiffness with Depth*

4.2.3 Duncan and Chang Hyperbolic model

The hyperbolic model was developed by Duncan and Chang (1970) to describe the non-linear stress dependent behaviour of soil. It is effective in representing the effects of confining pressure and deviator stress on stress-strain behaviour.

4.2.3.1 Stress-strain relationship

The incremental stress strain relationship for plane-strain conditions may be given by

$$\begin{Bmatrix} d\sigma_x \\ d\sigma_y \\ d\sigma_z \\ d\tau_{xy} \end{Bmatrix} = \frac{3B}{9B - E_t} \begin{bmatrix} (3B + E_t) & (3B - E_t) & (3B - E_t) & 0 \\ (3B - E_t) & (3B + E_t) & (3B - E_t) & 0 \\ (3B - E_t) & (3B - E_t) & (3B + E_t) & 0 \\ 0 & 0 & 0 & E_t \end{bmatrix} \begin{Bmatrix} d\varepsilon_x \\ d\varepsilon_y \\ 0 \\ d\gamma_{xy} \end{Bmatrix} \qquad \textit{eqn. 4.2-8}$$

where E_t is the tangent Young's modulus, and B is the bulk modulus. Procedures for relating these parameters to the stresses in soil are described below.

In a triaxial test, soil is known to be stiffer when the axial load is being reduced from its highest previous value (unloading) than when the axial load is being increased above its highest previous value (primary loading). The hyperbolic model allows assigning different values of Young's modulus for primary loading and unloading conditions.

4.2.3.2 Primary loading stiffness

During this loading the hyperbolic model approximates the stress-strain relationship with a hyperbola. This is derived from the relationship between the deviator stress (σ_1- σ_3) and the axial strain ε as follows:

$$(\sigma_1 - \sigma_3) = \frac{\varepsilon}{a + b\varepsilon} \qquad \textit{eqn. 4.2-9}$$

where $a = \frac{1}{E_i}$ and $b = \frac{1}{(\sigma_1 - \sigma_3)_{ult}}$

The initial tangent of the hyperbolic curve is expressed by the initial Young's modulus E_i . According to Duncan and Chang, E_i is dependent on the minor principal stress σ_3 as follows:

$$E_i = K.P_a\left(\frac{\sigma_3}{P_a}\right)^n \qquad \textit{eqn. 4.2-10}$$

where K is the Young's modulus coefficient, P_a is the atmospheric pressure, and n is the Young's modulus exponent. The value of P_a is 101.325 KN/m^2 at mean sea level.

The hyperbolic curve gradually reaches the ultimate deviator stress $(\sigma_1 - \sigma_3)_{ult}$ as can be seen in the Figure 4-2 below.

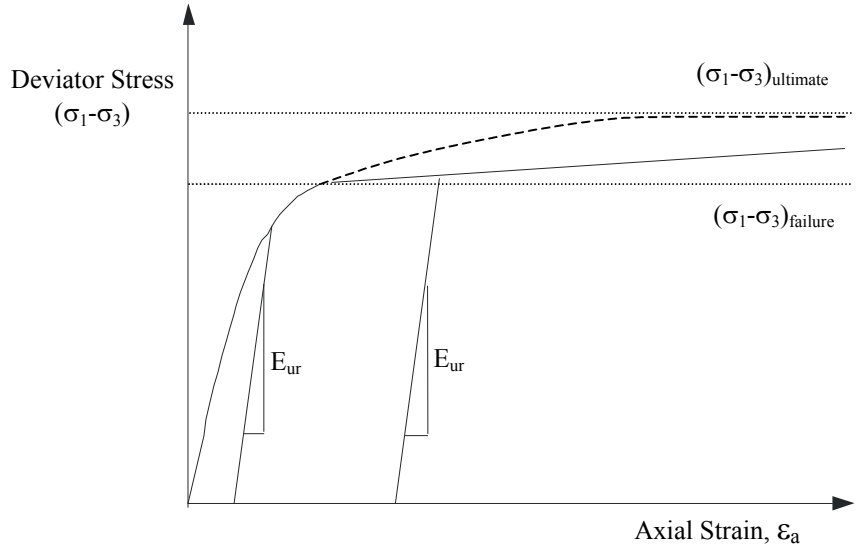


Figure 4-2 Stress-strain relationship for hyperbolic model

The deviator stress at failure is defined according to the Mohr-Coulomb failure criterion as follows:

$$(\sigma_1 - \sigma_3)_f = \frac{2c.Cos\phi + 2\sigma_3 Sin\phi}{1 - Sin\phi} \qquad \textit{eqn. 4.2-11}$$

The tangential Young's modulus at any point along the curve is expressed as:

$$E_t = \left(1 - R_f.SL\right)^2.E_i \qquad \textit{eqn. 4.2-12}$$

where SL is the stress level, defined by:

$$SL = \frac{(\sigma_1 - \sigma_3)}{(\sigma - \sigma_{31})_f} \qquad \textit{eqn. 4.2-13}$$

and R_f is the failure ratio defined as:

$$R_f = \frac{(\sigma_1 - \sigma_3)_f}{(\sigma_1 - \sigma_3)_{ult}} \qquad \textit{eqn. 4.2-14}$$

Equation 12 is used to determine the value of the tangent Young's modulus for values of the stress level SL less than unity. For values of the stress level equal to or greater than unity (ie stress in failure zone), the tangential Young's modulus E_t is set to a very small value E_f which is defined as:

$$E_f = 0.01 \times K.P_a \times (0.05)^n \qquad \textit{eqn. 4.2-15}$$

4.2.3.3 Unload-reload stiffness

Upon unloading, soils exhibit a stiffer behaviour. The hyperbolic model allows the use of a Young's modulus value which is higher than the tangential Young's modulus E_t .The value of this unload-reload Young's modulus is given by:

$$E_{ur} = K_{ur}.P_a\left(\frac{\sigma_3}{P_a}\right)^n \qquad \textit{eqn. 4.2-16}$$

where K_{ur} is the unload-reload coefficient, P_a ,is the atmospheric pressure and n is the Young's modulus exponent

4.2.3.4 Stress state for 2D

The stress level defined in equation 6 is valid for triaxial stress conditions. For the 2D case, a more complex criterion is required for the unload-reload stress state. Duncan used an unload-reload condition using a dimensionless quantity SS for the stress state. This is defined as:

$$SS = SL\sqrt[4]{\frac{\sigma_3 + \dfrac{c}{\tan\phi}}{P_a}} \qquad \textit{eqn. 4.2-17}$$

where SL is the stress level, defined by equation 6. This takes into account soils with cohesion intercept c in the strength envelope, and with an angle of internal friction ϕ.

A state of stress is defined as being "unload-reload" if the stress state SS is less than the maximum previous stress state SS_{max}

A stress state is defined as being "unload-reload" if the stress level is less than the critical stress level SL_{crt} , where

$$SL_{crt} = \frac{SS_{\max}}{\sqrt[4]{\frac{\sigma_3 + \frac{c}{\tan\phi}}{P_a}}} \qquad \textit{eqn. 4.2-18}$$

4.2.3.5 Stiffness Parameters

K	Modulus number K
Ku	Modulus number Ku
ν	Poisson's ratio v
C	Cohesion C (KN/m^2)
ϕ	Friction Angle
P_a	Atmospheric pressure (Default is 101.3 KN/m^2)
R_f	Failure Ratio
n	Exponent for ratio of minor principal stress over atmospheric pressure

4.3 Elastic-Perfectly Plastic models

Perfect plasticity or rigid plasticity implies that a yield surface is fixed in space. For a one dimensional uniaxial stress-strain test, the stress-strain relationship is shown in Figure 4-3.

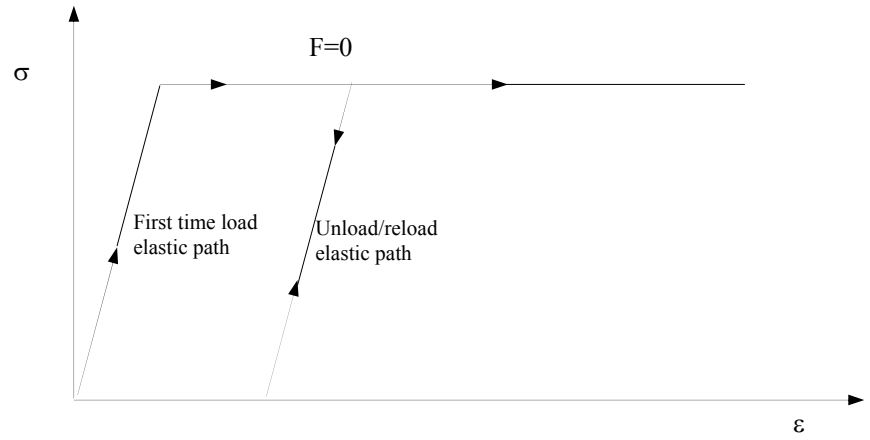


Figure 4-3 Perfect Plasticity

Rigid plasticity assumes that the yield function depends only upon the stress, so that the yield function is:

$f(\sigma)=0$

The elastic-perfectly plastic models provide a means of imposing limiting shear stress (i.e. soil strength) on what would otherwise be a wholly elastic response. As the name implies, there is no hardening or softening of the yield surface during plastic yielding - the shape and size of the yield surface remain constant. Several different yield criteria are provided, suitable either for effective or total stress analyses. Nonhomogeneity of elastic stiffness and strength is allowed, with both quantities being permitted to vary linearly with depth (at independent rates). Anisotropy of stiffness and strength, however, are not permitted - full isotropy is enforced.

A full description of these elastic-perfectly plastic models can be found in Owen and Hinton (1980), but be aware that these authors use a tension positive sign convention for normal stress.

4.3.1 Stiffness Parameters

The stiffness parameters required are:

Table 4-3

E_0	Young's modulus at the datum elevation
m_E	Rate of increase of Young's modulus with depth
y_0	Datum elevation at which $E = E_0$ (and $c = c_0$ - see Table 4-4)
ν	Poisson's ratio

4.3.2 Strength Parameters

The strength parameters required are:

Table 4-4

C_0	Cohesion at the datum elevation
m_C	Rate or increase of cohesion with depth
ϕ	Angle of internal friction for Mohr Coulomb and Drucker-Prager models only

The shapes of the four possible yield surfaces in principal stress space are shown in figures below. Note that these are both yield and failure surfaces. c and ϕ have their normal soil mechanics meaning and can be selected as effective or total stress parameters as appropriate. ϕ is specified in degrees.

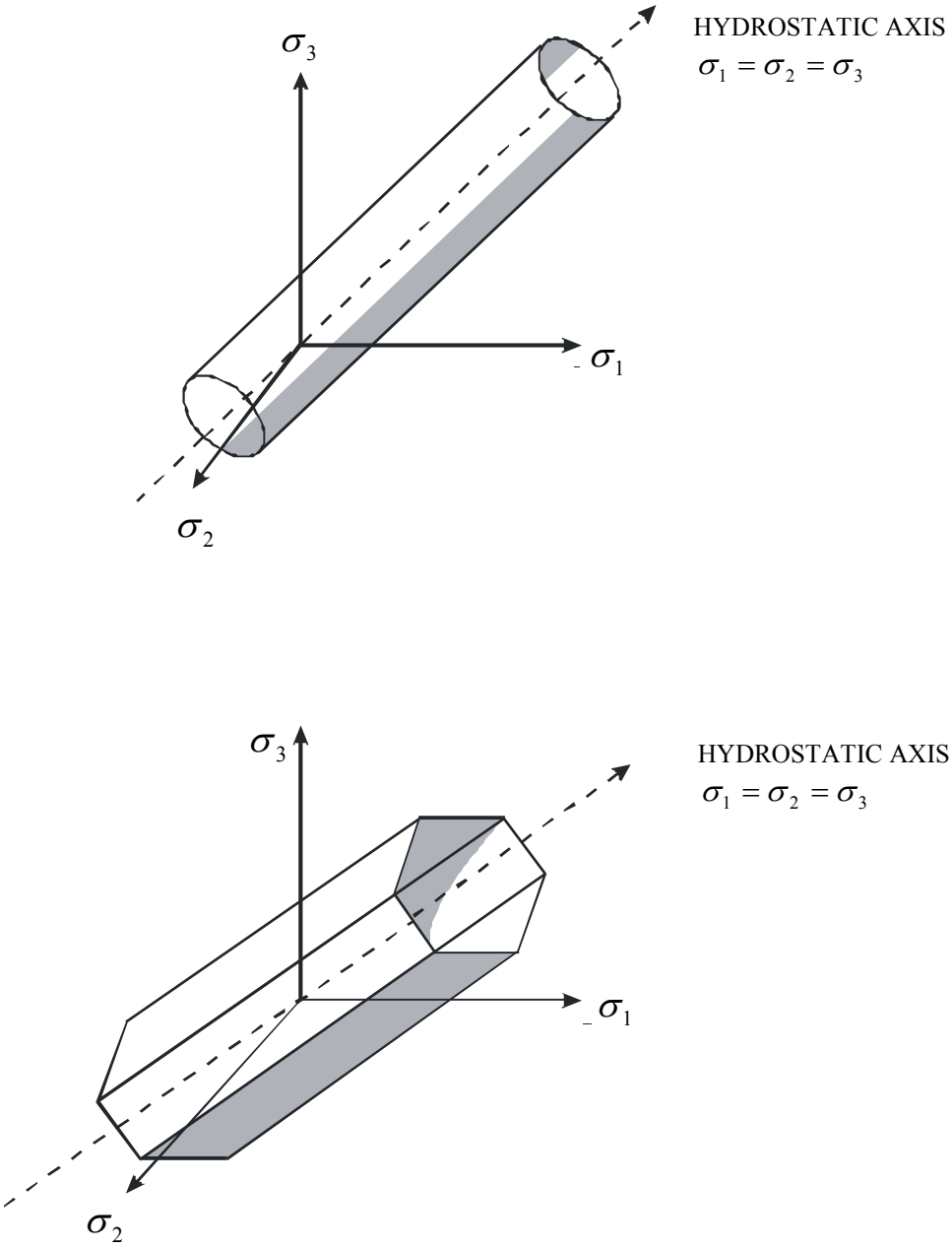


Figure 4-4 Von-Mises and Tresca Failure Surfaces (ϕ=0)

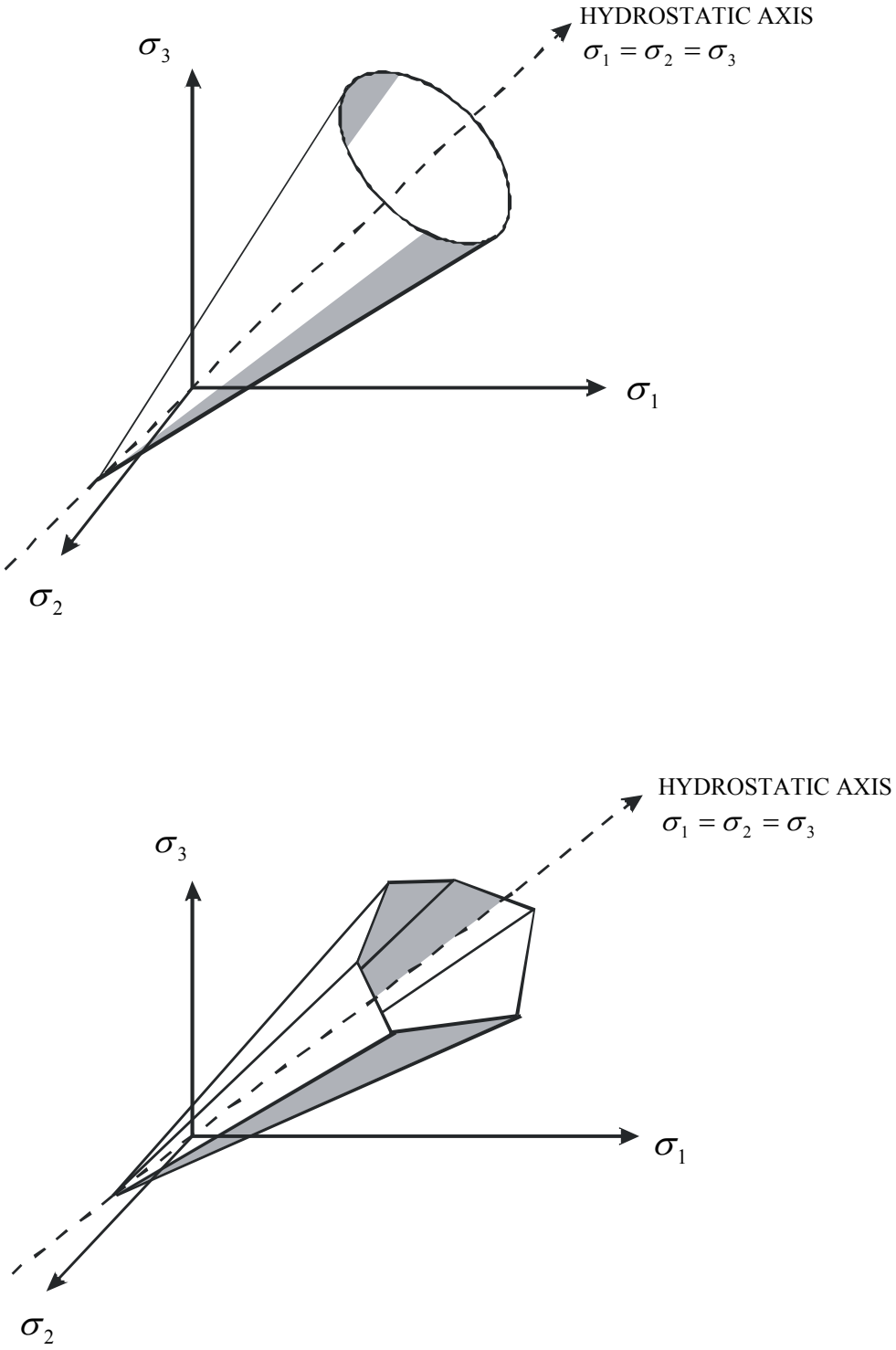


Figure 4-5 Drucker-Prager and Mohr-Coulomb Failure Surfaces

4.3.3 Governing Equations

The parameters E_o, C_o, m_E, m_C allow for a linear variation of both Young's modulus and shear strength with depth (see figure below)

$$E = E_o + m_E (y_o - y)$$

$$c = c_o + m_c (y_o - y)$$

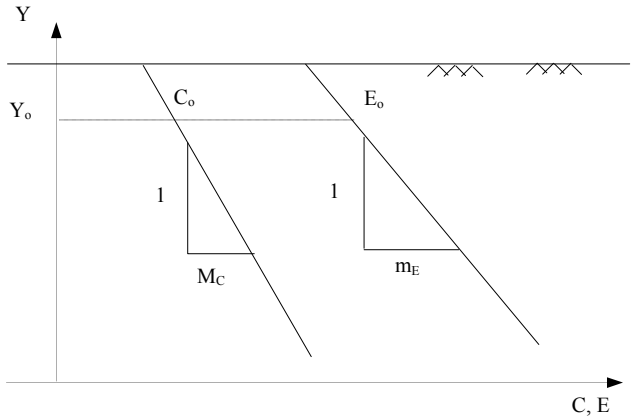


Figure 4-6 Permitted variation of stiffness and strength with depth

4.3.4 New Elastic-Perfectly Plastic Mohr Coulomb model

A new elastic-perfectly plastic Mohr Coulomb model has been introduced. This model has the following features:

- Non-associated flow, thus allowing for better prediction of dilation
- improved stress return algorithm especially around corners of the Mohr Coulomb surface and the apex of the yield surface.

This model is very much similar to the elastic-plastic hardening model described below, but without the hardening parameters. For a fuller understanding of the governing equations see section 4.4.

The material parameters for this models are as follows

Table 4-5

E_0	Young's modulus at the datum elevation
ν	Poisson's ratio
C_0	Cohesion at the datum elevation
ϕ	Angle of internal friction for Mohr Coulomb and Drucker-Prager models only
y_0	Datum elevation at which $E = E_0$ (and $c = c_0$ - see Table 4-4)
ψ	Dilatancy Angle
m_e	Rate or increase of Young's modulus with depth
m_C	Rate or increase of cohesion with depth
ISYM	Default = 0. Non zero value would force use of Symmetric solver. See note below.

4.3.4.1 Note on Frontal Solver and Non-Associative flow:

If the material being analysed has a dilatancy angle which is not the same as the friction angle, the non-symmetric frontal solver would be invoked in CRISP. This is necessary as the stiffness formulation would produce a non-symmetric stiffness. See section 4.4 below. It is possible to force the use of the conventional symmetric frontal solver by specifying ISYM=1. This will instruct the program to use a symmetric matrix based on the elastic properties in the stiffness assembly stage and make the necessary correction in the stress evaluation stage accounting for non-associative flow.

For most applications, it is recommended that ISYM is left as the default value (ie 0) .

4.4 New Elastic -Plastic Strain Hardening model with non-associative flow based on Mohr-Coulomb

This model was originally developed for Alexander Gibb Ltd. The Crisp Consortium is indebted to Alexander Gibb for making the model available for Crisp users. The original model is based on the paper by P.A.Vermeer and R. de Borst (Heron vol 29 1984, no.3)

The model is based on the classical Mohr Coulomb yield criterion, with the addition of the following features:

- Non-Associative plastic flow accounting for a more accurate prediction of volumetric strains
- Cohesion hardening,
- Friction hardening
- Dilation, allowing angle of dilatency to increase to an ultimate value at peak stress

Cohesion hardening, friction hardening and dilation are controlled by the amount of accumulated plastic strain as well as user defined limiting parameters

4.4.1 Material Parameters

Table 4-6

E	Young's Modulus
ν	Poisson's Ratio
C_i	Initial cohesion
C_f	Final (limiting) cohesion
ϕ_i	Initial friction angle
ϕ_f	Final (peak) friction angle
ψ_i	Initial dilation angle
ψ_r	Final dilation angle
h_c	Cohesion hardening parameter
ε_f	Effective plastic strain required to mobilise the peak friction angle

4.4.2 Governing Constitutive Relationships

For an elasto-plastic material, the incremental stress is related to the incremental strain thus

$$\{d\sigma\} = [D^{ep}]\{d\varepsilon\} \qquad \textit{Eq. 4-1}$$

We now have to derive the form of the elasto-plastic stiffness D^{ep}. Starting with the additivity postulate which states that the strain consists of elastic and plastic parts thus,

$$\{\varepsilon\} = \{\varepsilon^e\} + \{\varepsilon^p\} \qquad \textit{Eq. 4-2}$$

In incremental form, this becomes,

$$\{d\varepsilon\} = \{d\varepsilon^e\} + \{d\varepsilon^p\} \qquad \textit{Eq. 4-3}$$

The stresses are related to the elastic components of strain through the elastic matrix D^e, thus

$$\{d\sigma\} = [D^e]\{d\varepsilon^e\} \qquad \textit{Eq. 4-4}$$

or

$$\{d\varepsilon^e\} = [D^e]^{-1}\{d\sigma\} \qquad \textit{Eq. 4-5}$$

in which the D matrix is defined according to Hook's law, thus

$$[D^e] = \frac{E}{(1+\upsilon)(1-2\upsilon)}\begin{bmatrix} 1-\upsilon & \upsilon & \upsilon & 0 \\ \upsilon & 1-\upsilon & \upsilon & 0 \\ \upsilon & \upsilon & 1-\upsilon & 0 \\ 0 & 0 & 0 & \left(\frac{1-2\upsilon}{2}\right) \end{bmatrix} \qquad \textit{Eq. 4-6}$$

Substituting (3) into (4) gives

$$\{d\sigma\} = [D^e]\{d\varepsilon - d\varepsilon^p\} \qquad \textit{Eq. 4-7}$$

The plastic strain increments are related to the flow rate, thus

$$d\varepsilon^p = d\lambda\left\{\frac{\partial Q}{\partial \sigma}\right\} \qquad \textit{Eq. 4-8}$$

where Q is a plastic potential function which is equal to the yield function when using an associated flow rule. For non-associated flow rule, the plastic potential function Q, would be geometrically similar to the yield function, but with the friction angle phi replaced by the dilation angle psi.

The yield function for this elasto-plastic hardening model is defined as

$$F(\{\sigma\},c(\kappa),\phi(\kappa)) = \frac{1}{2}(\sigma_1 - \sigma_3) + \frac{1}{2}(\sigma_1 + \sigma_3)\sin(\phi(\kappa)) - c(\kappa).\cos(\phi(\kappa)) \quad \textit{Eq. 4-9}$$

F<0 inside yield surface, therefore elastic

F>= 0 on or outside yield surface, therefore plastic

The mobilised cohesion, c(k) and mobilised friction angle, ϕ(k), depend on the hardening parameter, k.

We shall derive the elasto-plastic matrix for the general non-associated case in which Q ≠ F.

It is a basic assumption that during plastic yield the stress remains on the yield surface, thus:

$$dF(\{\sigma\},c(\kappa),\phi(\kappa)) = 0 \quad \textit{Eq. 4-10}$$

differentiating equation (10) by the chain rule we get,

$$dF = \left\{\frac{\partial F}{\partial \sigma}\right\}^T \{d\sigma\} + \frac{\partial F}{\partial c(\kappa)} dc(\kappa) + \frac{\partial F}{\partial \phi(k)} d\phi(\kappa) = 0 \quad \textit{Eq. 4-11}$$

Equation (11) is known as the consistency equation.

Substituting equation (5) and (8) into (3), we obtain

$$\{d\varepsilon\} = \left[D^e\right]^{-1}\{d\sigma\} + d\lambda\left\{\frac{\partial Q}{\partial \sigma}\right\} \quad \textit{Eq. 4-12}$$

The expansion of the yield surface is related to the hardening parameter k, which is related to the effective plastic strain thus,

$$\kappa = \bar{\varepsilon}_p = \sum d\overline{\varepsilon_p} \quad \textit{Eq. 4-13}$$

where

$$d\overline{\varepsilon_p} = d\lambda\sqrt{\frac{2}{3}\left\{\frac{\partial Q}{\partial \sigma}\right\}^T\left\{\frac{\partial Q}{\partial \sigma}\right\}} \quad \textit{Eq. 4-14}$$

The consistency equation (11) now becomes

$$0 = \left\{\frac{\partial F}{\partial \sigma}\right\}^T \{d\sigma\} + \frac{\partial F}{\partial c(\bar{\varepsilon}_p)} . \frac{\partial c(\bar{\varepsilon}_p)}{\partial \bar{\varepsilon}_p} d\bar{\varepsilon}_p + \frac{\partial F}{\partial \phi(\bar{\varepsilon}_p)} . \frac{\partial \phi(\bar{\varepsilon}_p)}{\partial \bar{\varepsilon}_p} d\bar{\varepsilon}_p \qquad \textit{Eq. 4-15}$$

Making the substitution

$$-Hd\lambda = \frac{\partial F}{\partial c(\bar{\varepsilon}_p)} \frac{\partial c(\bar{\varepsilon}_p)}{\partial \bar{\varepsilon}_p} d\bar{\varepsilon}_p + \frac{\partial F}{\partial \phi(\bar{\varepsilon}_p)} \frac{\partial \phi(\bar{\varepsilon}_p)}{\partial \bar{\varepsilon}_p} d\bar{\varepsilon}_p \qquad \textit{Eq. 4-16}$$

where H is a hardening parameter, equation (15) now becomes

$$0 = \left\{\frac{\partial F}{\partial \sigma}\right\}^T \{d\sigma\} - Hd\lambda \qquad \textit{Eq. 4-17}$$

$$H = -\left(\frac{\partial F}{\partial c(\bar{\varepsilon}_p)} \frac{\partial c(\bar{\varepsilon}_p)}{\partial \varepsilon_p} + \frac{\partial F}{\partial \phi(\bar{\varepsilon}_p)} \frac{\partial \phi(\bar{\varepsilon}_p)}{\partial \varepsilon_p} \right) \sqrt{\frac{2}{3} \left\{\frac{\partial Q}{\partial \sigma}\right\}^T \left\{\frac{\partial Q}{\partial \sigma}\right\}} \qquad \textit{Eq. 4-18}$$

Pre-multiplying equation (12) by $\left\{\frac{\partial F}{\partial \sigma}\right\}^T \left[D^e\right]$ and noting that $\left[D^e\right]\left[D^e\right]^{-1}$ is the identity matrix, gives

$$\left\{\frac{\partial F}{\partial \sigma}\right\}^T \left[D^e\right]\{d\varepsilon\} = \left\{\frac{\partial F}{\partial \sigma}\right\}^T \{d\sigma\} + \left\{\frac{\partial F}{\partial \sigma}\right\}^T \left[D^e\right] d\lambda . \left\{\frac{\partial Q}{\partial \sigma}\right\}^T \qquad \textit{Eq. 4-19}$$

Substituting for $\left\{\frac{\partial F}{\partial \sigma}\right\}^T d\sigma$ as in equation (17) leads to

$$d\lambda = \frac{\left\{\frac{\partial F}{\partial \sigma}\right\}^T \left[D^e\right]\{d\varepsilon\}}{H + \left\{\frac{\partial F}{\partial \sigma}\right\}^T \left[D^e\right]\left\{\frac{\partial Q}{\partial \sigma}\right\}} \qquad \textit{Eq. 4-20}$$

Re-arranging (12) leads to

$$\{d\sigma\} = \left[D^e\right]\{d\varepsilon\} - d\lambda \left[D^e\right]\left\{\frac{\partial Q}{\partial \sigma}\right\} \qquad \textit{Eq. 4-21}$$

Substituting for dλ into (8) gives

$$\{d\varepsilon^p\} = \frac{[D^e]\left\{\frac{\partial Q}{\partial \sigma}\right\}\left\{\frac{\partial F}{\partial \sigma}\right\}^T \{d\varepsilon\}}{H + \left\{\frac{\partial F}{\partial \sigma}\right\}^T [D^e]\left\{\frac{\partial Q}{\partial \sigma}\right\}} \qquad \textit{Eq. 4-22}$$

Substituting the above into (7) gives

$$\{d\sigma\} = [D^e]\{d\varepsilon\} - \frac{[D^e]\left\{\frac{\partial Q}{\partial \sigma}\right\}\left\{\frac{\partial F}{\partial \sigma}\right\}^T [D^e]\{d\varepsilon\}}{H + \left\{\frac{\partial F}{\partial \sigma}\right\}^T [D^e]\left\{\frac{\partial Q}{\partial \sigma}\right\}} \qquad \textit{Eq. 4-23}$$

in which

$$[D^{ep}] = [D^e] - \frac{[D^e]\left\{\frac{\partial Q}{\partial \sigma}\right\}\left\{\frac{\partial F}{\partial \sigma}\right\}^T [D^e]}{H + \left\{\frac{\partial F}{\partial \sigma}\right\}^T [D^e]\left\{\frac{\partial Q}{\partial \sigma}\right\}} \qquad \textit{Eq. 4-24}$$

Both the yield function and the plastic potential function could be expressed in terms of the stress invariants, I1, J2 and theta. The yield function is given by

$$F(\{\sigma\}, c(\kappa), \phi(\kappa)) = \left(\frac{I_1}{3}\right)\sin(\phi(\kappa)) + \sqrt{J_2}\left[\cos(\theta) - \frac{\sin(\theta)\sin(\phi(\kappa))}{\sqrt{3}}\right] - c(\kappa).\cos(\phi(\kappa)) = 0$$

Eq. 4-25

The plastic potential function is given by:

$$q(\{\sigma\}, \psi(\kappa)) = \left(\frac{I_1}{3}\right)\sin(\psi(\kappa)) + \sqrt{J_2}\left[\cos(\theta) - \frac{\sin(\theta)\sin(\psi(\kappa))}{\sqrt{3}}\right] \qquad \textit{Eq. 4-26}$$

4.4.3 Hardening Formulation

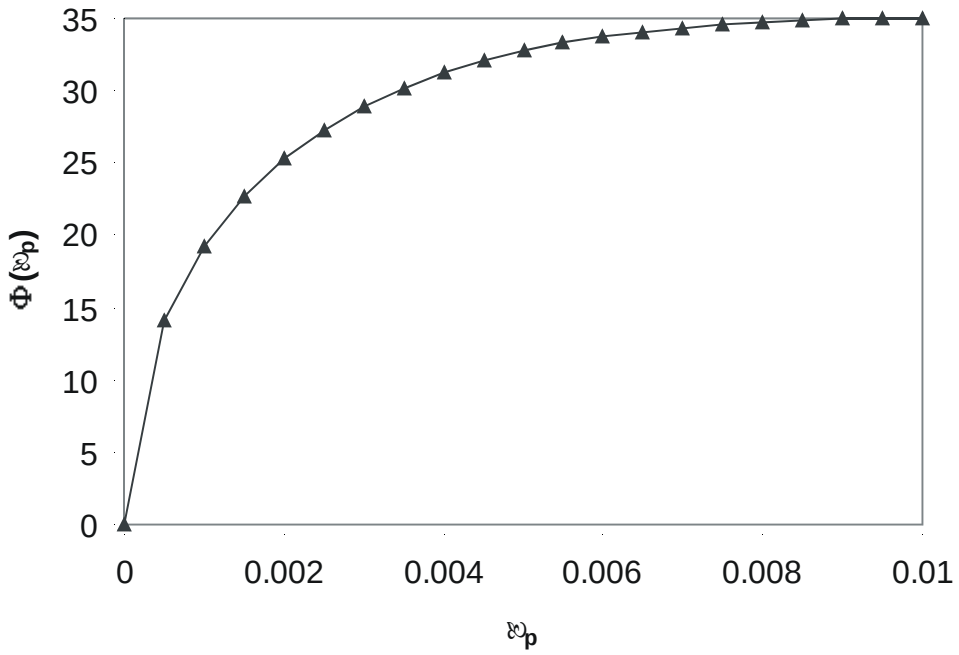
These hardening formulas were presented by Vermeer and de Borst (1984).

4.4.3.1 Friction Hardening

The mobilised friction angle is given by:

$$\phi(\overline{\varepsilon_p}) = \arcsin\left(2\frac{\sqrt{\overline{\varepsilon_p}\,\overline{\varepsilon_f}}}{\overline{\varepsilon_p} + \overline{\varepsilon_f}} \sin(\phi_f) \right) \qquad \textit{Eq4-27}$$

where $\overline{\varepsilon}_f$ is the effective plastic strain required to mobilise the peak friction angle ϕ_f. This relationship is shown in the figure below.



Variation of ϕ with effective plastic strain for limiting values of ϕ_f=35 and ε_f=0.01

Figure 4-7

4.4.3.2 Cohesion hardening

This is assumed to be linear with respect to the effective plastic strain as follows:

$$c(\overline{\varepsilon_p}) = c_i + h_c \overline{\varepsilon_p} \qquad \textit{Eq4-28}$$

in which c_i is the initial cohesion and h_c is a constant cohesion hardening parameter. In order to limit the increase of $c(\overline{\varepsilon}_p)$ a maximum cut-off cohesion, c_f, is specified such that

$$c(\overline{\varepsilon_p}) \le c_f$$

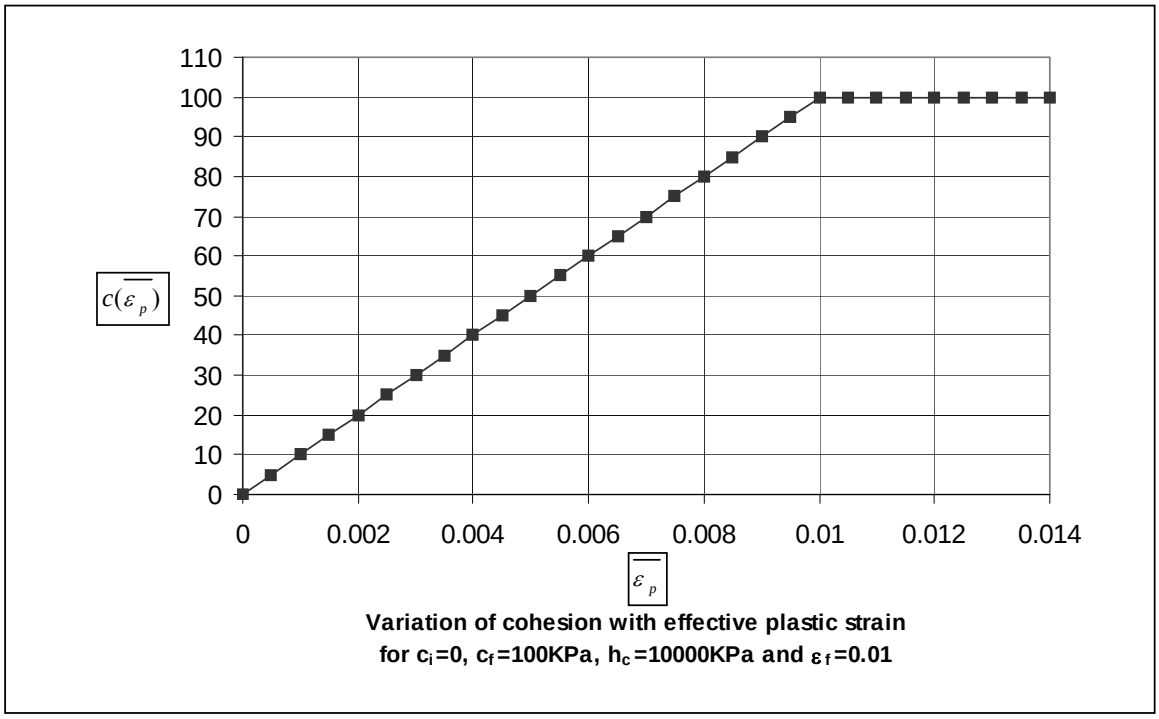


Figure 4-8

4.4.3.3 Dilation

Laboratory tests have shown that dilatancy angle ψ increases gradually during shearing up to a limit value ψ_f at peak stress state. The relationship between the mobilised dilation angle $\psi(\kappa)$ and the mobilised friction angle $\phi(\kappa)$ is given by,

$$\sin(\psi(\kappa)) = \sin(\psi_i) + (\sin(\psi_f) - \sin(\psi_i))\left[\frac{\sin(\phi(\kappa)) - \sin(\phi_i)}{\sin(\phi_f) - \sin(\phi_i)}\right] \qquad \textit{Eq4-29}$$

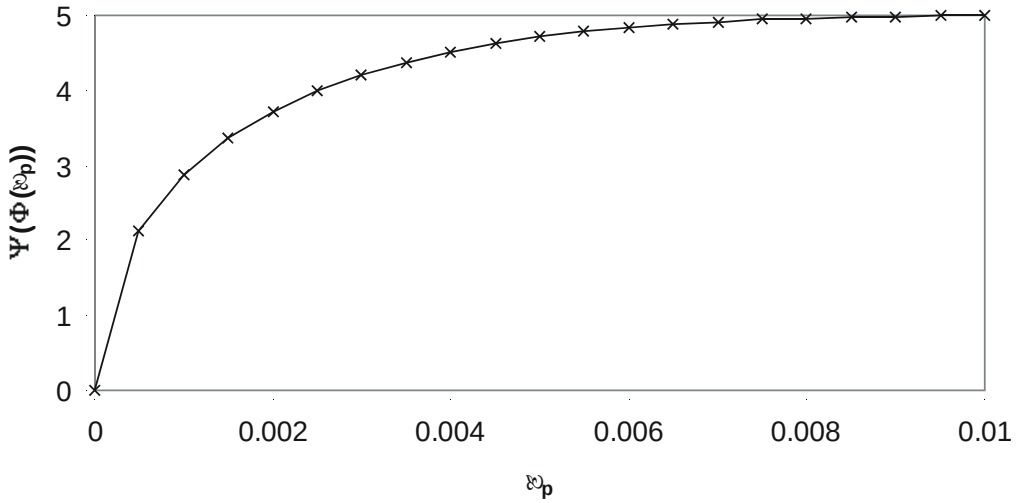
where ψ_i is the initial dilation angle, ϕ_i is the initial friction angle, ψ_f is the final dilation angle and ϕ_f is the final friction angle.
From equation (27) we have

$$\phi(\kappa) = \phi(\overline{\varepsilon_p}) = \arcsin\left(2\frac{\sqrt{\overline{\varepsilon_p}\,\overline{\varepsilon_f}}}{\overline{\varepsilon_p} + \overline{\varepsilon_f}}\sin(\phi_f)\right)$$

Therefore, the mobilised dilation angle could be expressed with the relationship:

$$\sin(\psi(\kappa)) = \sin(\psi_i) + (\sin(\psi_f) - \sin(\psi_i)) \left[\frac{\left(2\frac{\sqrt{\varepsilon_p \varepsilon_f}}{\varepsilon_p + \varepsilon_f} \sin(\phi_f) \right) - \sin(\phi_i)}{\sin(\phi_f) - \sin(\phi_i)} \right] \qquad \text{Eq4-30}$$

This is shown in the figure below.



Variation of dilatancy angle with effective plastic strain with the limits ϕ_f =35, ψ_f=5 and ε_f=0.01

Figure 4-9

4.5 Critical State Family

4.5.1 Basic Description

CRISP provides the Cam clay (Schofield & Wroth, 1968), modified Cam clay (Roscoe & Burland, 1968), and Schofield (Schofield, 1980) models. All of these models are specific formulations within the framework of critical state soil mechanics (CSSM).

The yield loci of the critical state family implemented in CRISP are depicted in Figure 4-10 to Figure 4-12 below:

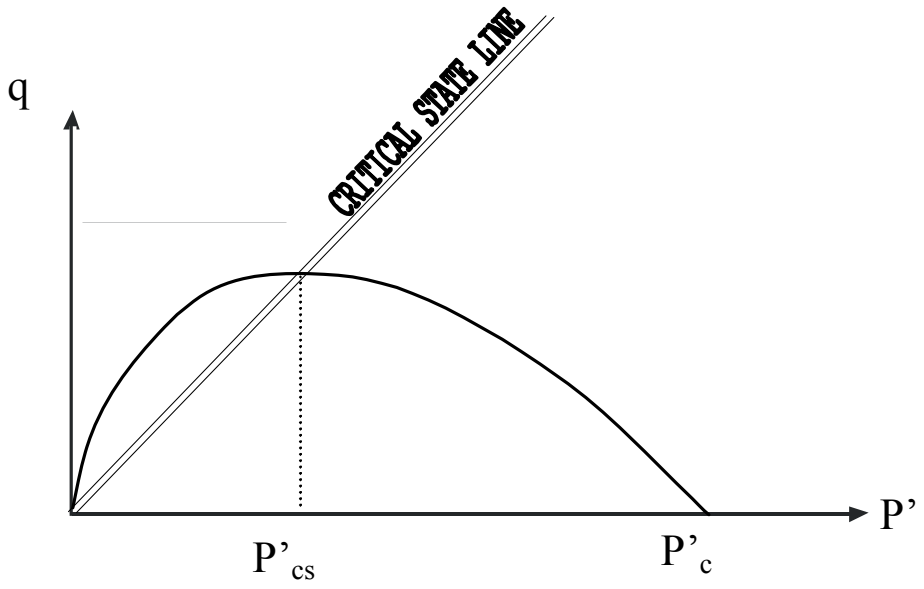


Figure 4-10 *Cam Clay Yield Locus*

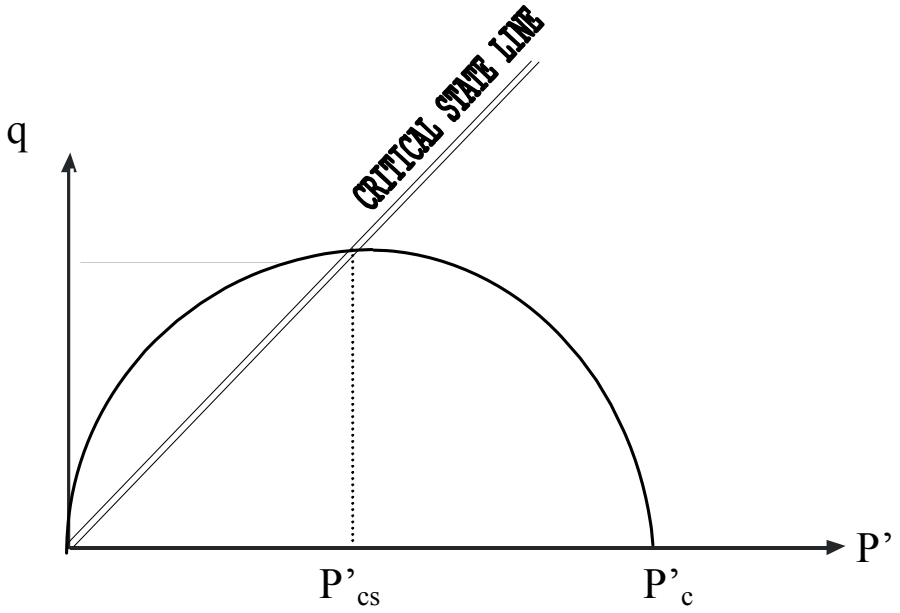


Figure 4-11 *Modified Cam clay Yield Locus*

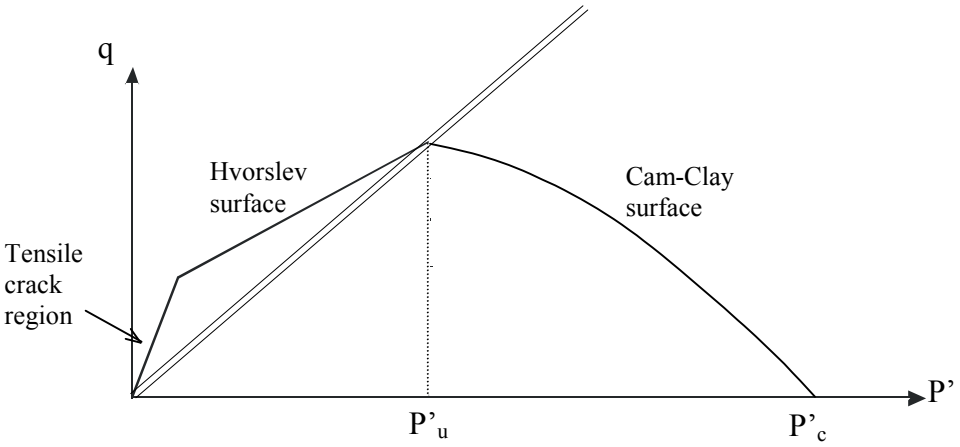


Figure 4-12 *Schofield Model Yield Locus*

4.5.2 CSSM Parameters

The parameters required for all of the CSSM models are:

Table 4-7

Symbol	Description
λ	Slope of isotropic compression line in v : ln(p')
κ	Slope of unload-reload lines in v : ln(p')
e_{cs}	Reference void ratio on critical state line when p' = 1
M	Slope of critical state line in q : p'

λ, κ and M have the normal meanings (see Schofield & Wroth, 1968; Atkinson and Bransby, 1978).

e_{cs} is the value of void ratio on the critical state line for p' = 1 (using units of pressure consistent with other stresses input to CRISP). In the terminology of critical state soil mechanics : $e_{cs} = \Gamma - 1$

Additional parameters required for the Schofield model are as follows:

Table 4-8

Symbol	Description
H	Slope of Hvorslev surface in q : p' space
S	Slope of "no-tension" cut-off in q : p' space

4.5.3 Elastic Parameters

In the classical development of the Cam clay (Schofield & Wroth, 1968) and modified Cam clay (Roscoe & Burland, 1968) models, it is assumed that there are no elastic shear strains. In terms of the conventional elastic parameters this means that the shear modulus G is infinite, i.e. Poisson's ratio = -1. Since it is not possible to implement this assumption in a displacement based finite element program (some stiffness terms are indeterminate and the program will attempt to divide by zero) it is necessary to specify an extra elastic property which allows non-zero elastic shear strains to be calculated. CRISP interprets the number entered into the data according to its value. If the value is less than 0.5 then it is assumed that the number refers to Poisson's ratio, whereas a larger number is taken to be a value of shear modulus. It is straightforward to show (by differentiating the equation of a κ-line) that the elastic bulk modulus for effective stress changes is given by:

$$K' = \frac{(1+e)\,p'}{\kappa} \qquad \textit{eqn. 4.5-1}$$

It has been shown (Zytynski et al, 1978) that the assumption of a constant Poisson's ratio together with this variable effective bulk modulus leads to elastic behaviour which is non-conservative i.e. it is possible to extract work from a closed cycle in stress space. The use of a constant value of shear modulus, G, is preferable from a theoretical standpoint since this problem does not arise. However a constant value of Poisson's ratio is more often assumed (typically a value of about 0.3) for the following reasons.

(i) The Cam clay and Modified Cam clay models do not give very good results in situations where there are cyclic loads (whatever elastic properties are assumed). In practice a build up of pore pressure is often seen, whereas the models predict no change in pore pressures for stress cycles within the yield

locus. If the effect of stress cycling is important in a problem, it is probably worthwhile considering the incorporation of a new constitutive model in the program.

(ii) Prediction of triaxial test results is usually better if a constant value of Poisson's ratio is used.

(iii) Attempts to measure G experimentally show that it is dependent on stress level (however it should be noted that the variation observed is not the same as for the bulk modulus - there is a strong correlation with the size of yield locus, which is determined by p'_c)

(iv) It is appropriate to assume that values of the critical state parameters are the same over the whole problem domain. In these circumstances a constant value of Poisson's ratio is to be preferred to a constant value of G bearing in mind the points made in (iii).

In conclusion it is worth pointing out that the main strength of theCam clay models is in the calculation of plastic strains during yielding as opposed to the elastic strains which are calculated for overconsolidated behaviour. The choice of a value of Poisson's ratio is normally based on an experimentally observed relationship between K_0 and OCR (see Section 5.1.5 for a discussion of how this is done).

4.5.4 Governing Equations

4.5.4.1 Cam clay

The yield locus forCam clay is shown in Figure 4-10, and is defined by the equation:

$$q = Mp' \ln\left(\frac{p'}{p'_c}\right) \qquad \textit{eqn. 4.5-2}$$

The yield locus for modified Cam clay is shown in , and is defined by the equation:

$$q^2 = M^2 p'(p'_c - p') \qquad \textit{eqn. 4.5-3}$$

The yield locus for the Schofield model is shown in Figure 4-12. The Roscoe surface is the same as that for Cam clay. The Hvorslev surface is defined by:

$$q = (M - H)\, p'_u \left(\frac{p'}{p'_u}\right)^{\frac{\kappa}{\lambda}} + Hp' \qquad \textit{eqn. 4.5-4}$$

The "no tension" cut-off is simply a straight line at a gradient of S in q : p' space.

4.5.5 DETERMINATION OF CRITICAL STATE SOIL PARAMETERS USING LABORATORY TESTS

Original notes by Professor Mike Gunn, South Bank University, London, UK

Produced by the CRISP Consortium Ltd

4.5.6 PARAMETERS TO BE DETERMINED

$$\left.\begin{matrix} M \\ \Gamma \\ \lambda \end{matrix}\right\} \text{Define critical states} \begin{cases} q = Mp' \\ v = \Gamma - \lambda \ln p' \end{cases}$$

κ elastic volumetric compressibility

($v = v_\kappa - \kappa \ln p'$)

\- one other elastic parameter

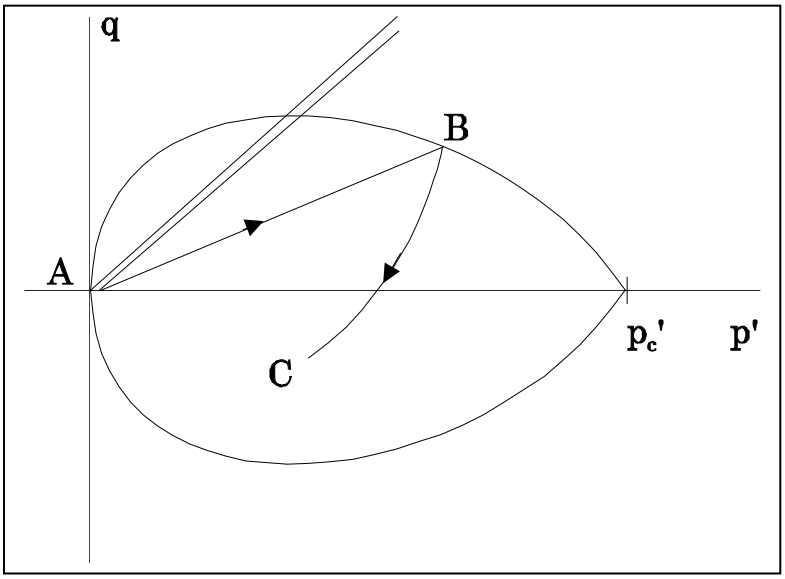
p_c' size of Cam-clay yield locus

Note:

Whereas M, Γ, λ, κ are assumed (in Critical State theory) to be soil properties which do not change, p_c' depends on the past loading of the soil.

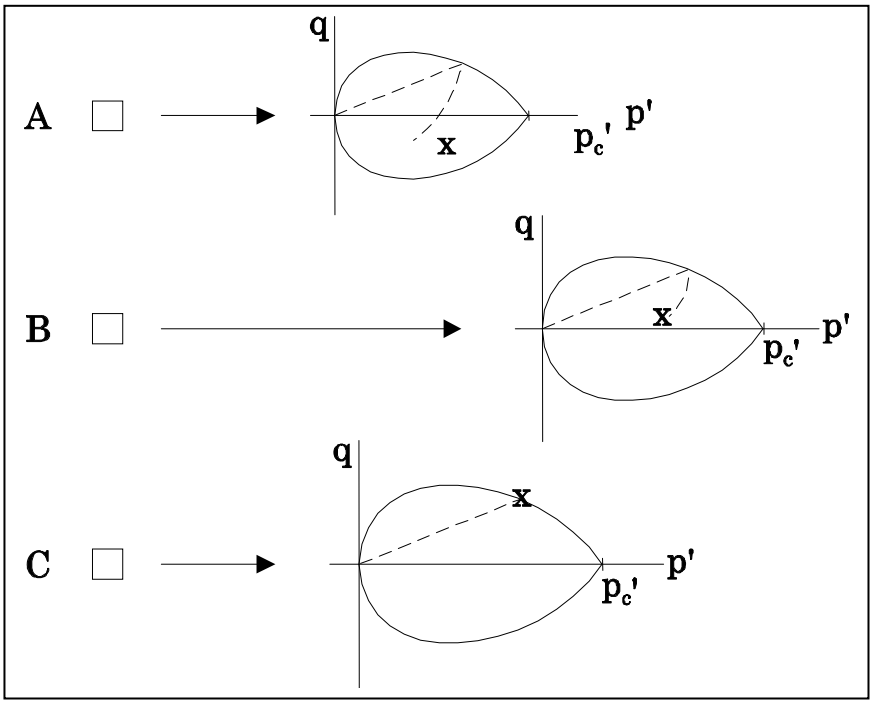
In a soil deposit p_c' will vary with depth.

A typical element of soil in the field follows a stress path as shown below:



AB virgin compression

BC unloading (due to erosion or water table changes) to produce over consolidated state.



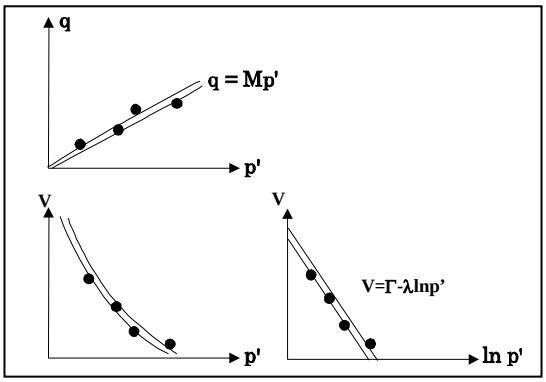
In each case ‘x’ indicates the current effective stresses.

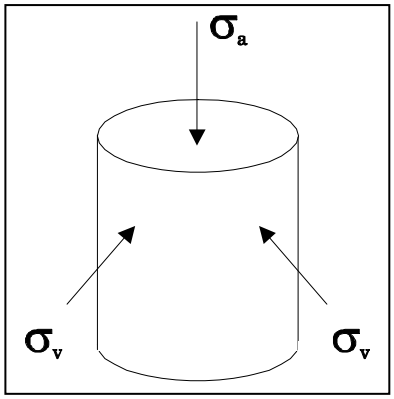
The distance between ‘x’ and the yield surface in each case determines (together with the stress path direction when the soil is loaded) the amount of elastic straining before plastic yielding starts.

Hence it is important to obtain accurate estimates of pc’ if we are going to use CSSM to predict strains and ground movements.

4.5.7 TRIAXIAL TESTS

If we plot the end points of triaxial tests in (p’, q) and (ln p’, v) plots then we can determine M, Γ and λ.



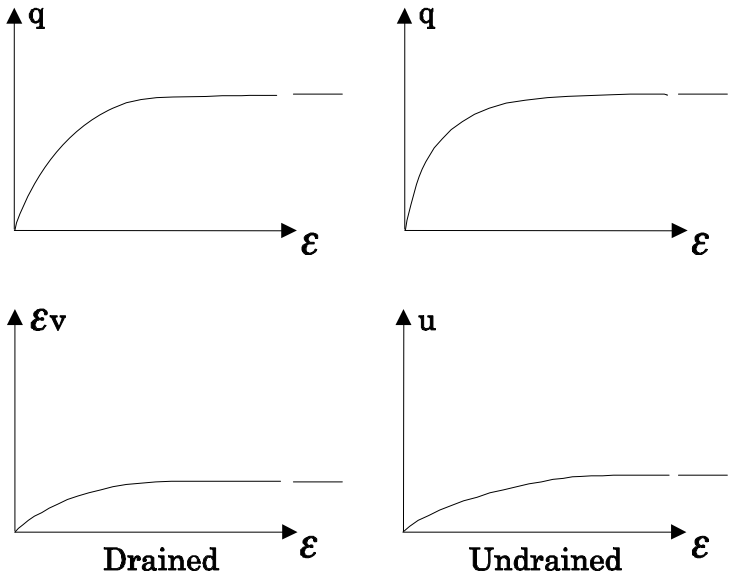


The tests can be drained or undrained (slow, with pore pressure measurement).

Note

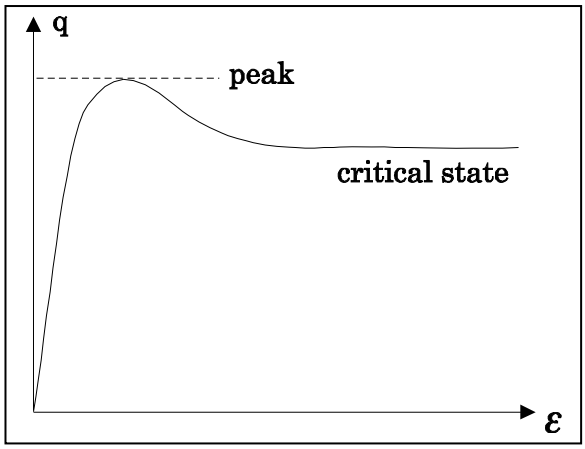
the test samples must be consolidated to a known effective pressure at the start of the tests.

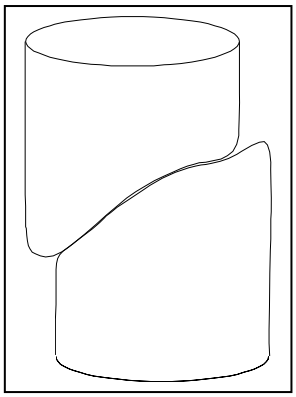
the tests must be continued to large strains to ensure that samples are close to the critical state (in which the samples undergo continuing shear with no change in stress or volume).



Some samples

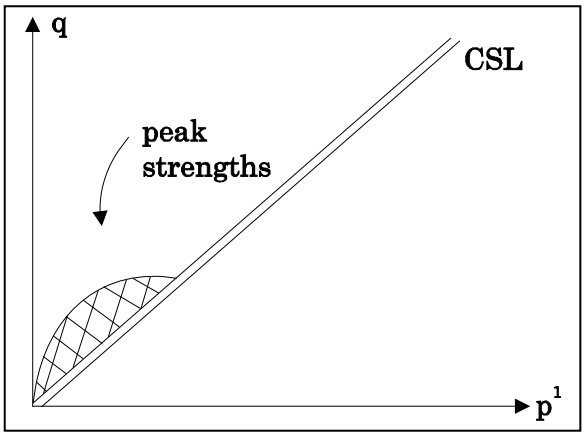
(particularly heavily over-consolidated ones) fail before the critical state is reached.



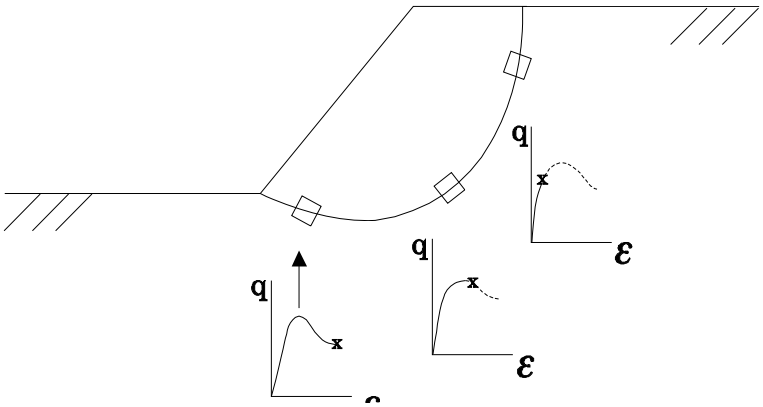


If we could conduct very careful strain - controlled tests on these samples we would see the critical state at a lower stress than the peak.

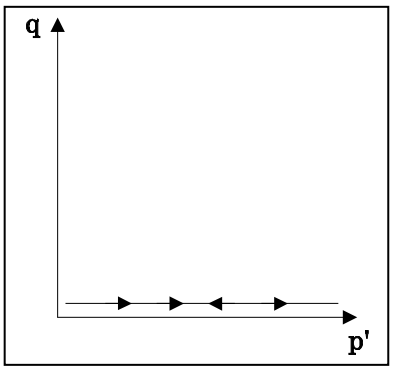
Peak strengths correspond to points which plot above the Critical State Line in a (p', q) plot.

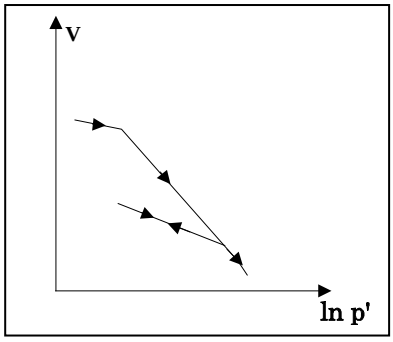


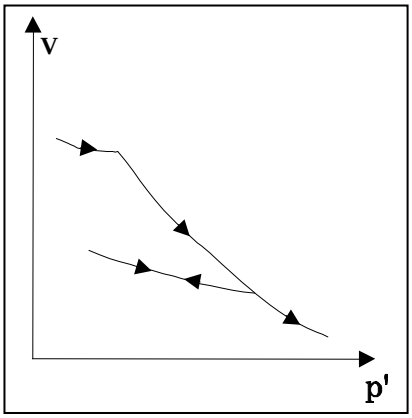
It is unwise to rely on peak strengths in design as the average mobilised strength will always be less than the peak.



We can also find λ and κ by drained isotropic loading and unloading in the triaxial apparatus.







4.5.8 STRESS PATH TESTS

Different types of test (triaxial compression, triaxial extension) may give different values of M.

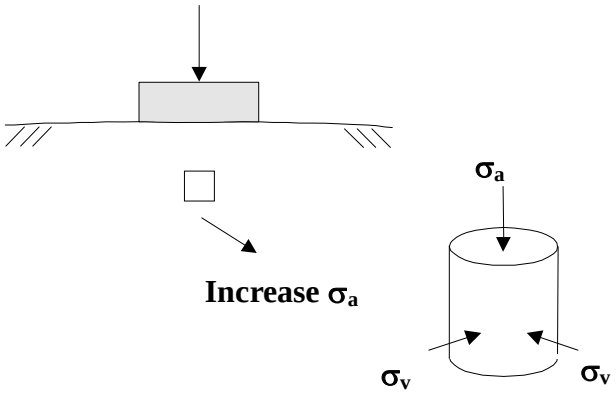
λ (and particularly κ) may vary with stress level / degree of unloading.

For these reasons we usually:

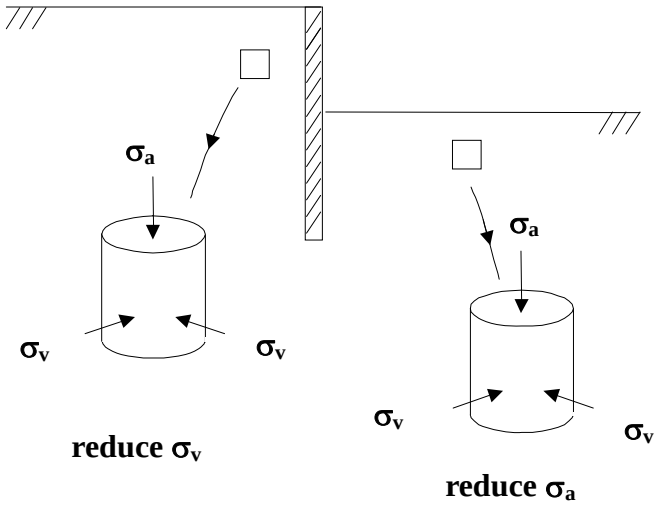
reconsolidate the soil sample to its field effective stresses

apply a stress path in the triaxial test corresponding to the expected stress path during construction in the field.

Footings and slabs



Embedded retaining walls



4.5.9 OEDOMETER TESTS

According to Cam-clay theory, one-dimensional loading also gives a "λ - line" in a (V, ln p') plot.

Also, it is easy to show that:

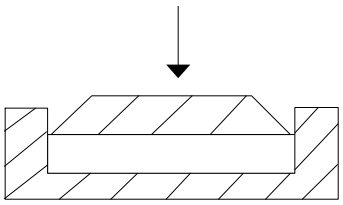
$$\lambda = \frac{C_c}{2.3}$$

$$\kappa = \frac{C_s}{2.3}$$

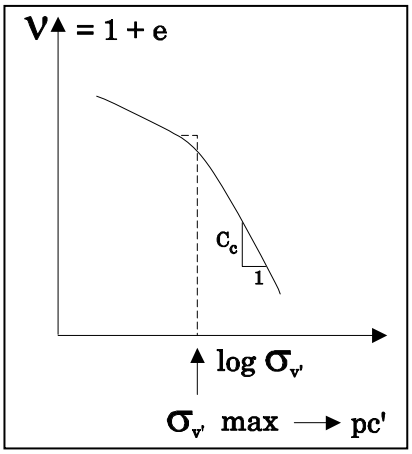
where C_c and C_c are obtained by the standard interpretation of the oedometer test.

(NB: ln (10) = 2.303)

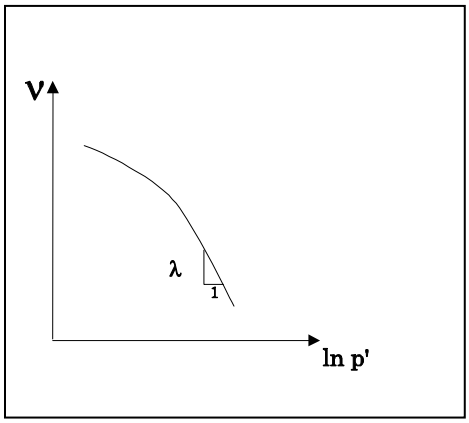
schematic of apparatus



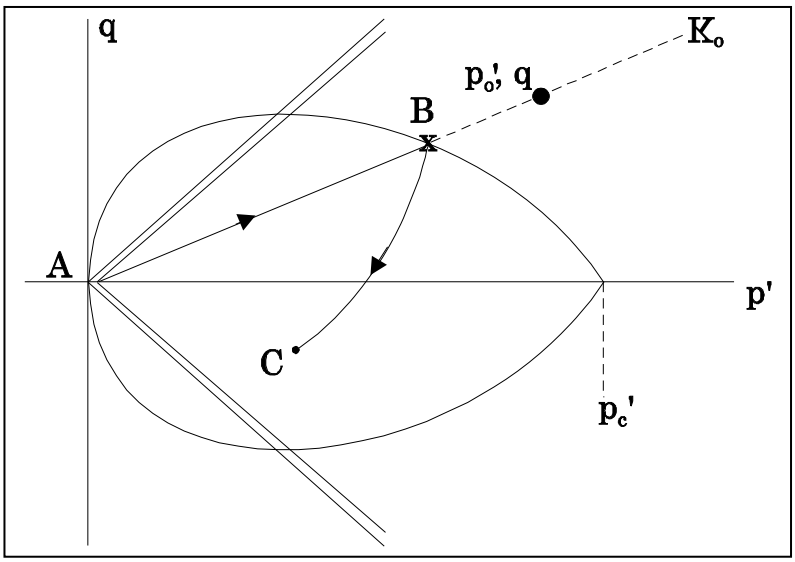
"conventional" plot e, log(σ'_v)



CSSM plot v, ln p'



4.5.10 Size of Yield Surface



To get size of current yield surface:

assume K_o for normally consolidated state $K_o = 1-\sin\phi'$

where $K_o = \sigma'_h / \sigma_v'$

$$p_o' = \left(\frac{1+2K_o}{3}\right)\sigma_{v\,max}'$$

$q_o = (1 - K_o)\,\sigma_v'_{\,max}$

rearrange equation for yield surface:

$q = Mp' \ln\left(\frac{p_c'}{p'}\right)$

$$\Rightarrow p_c' = p' \exp\left(\frac{q}{Mp'}\right)$$

substitute $p' = p_o$

and $q = q_o$

to get p_c'

4.5.11 INDEX TESTS

These are done to establish the moisture contents corresponding to the Plastic Limit (PL) and Liquid Limit (LL) of the soil.

If we assume that the strength of the soil at the Plastic Limit is 100 times the strength at the Liquid Limit, then:

$$\lambda = \frac{V_L - V_p}{\ln 100} = \frac{(w_L - w_p)G_s}{\ln 100}$$

or

$$\lambda = \frac{PI \times G_s}{160}$$

where PI is % Plasticity Index.

4.6 Three Surface Kinematic Hardening Model

The traditional elasto-plastic hardening models incorporate isotropic hardening. This is represented by the yield function:

$f(\sigma,h)=0$

where h is a hardening parameter which is dependent on the amount of plastic strain. In a multi-dimensional stress space, the centre of the yield surface does not change during the loading process, hence the term isotropic.

For a proper description of cyclic loading it is necessary that the yield surface not only expands, but also translates, resulting in kinematic hardening. Thus the yield function becomes:

$f(\sigma-a,h)= 0$

Where the parameter a refers to the current centre of the yield surface.

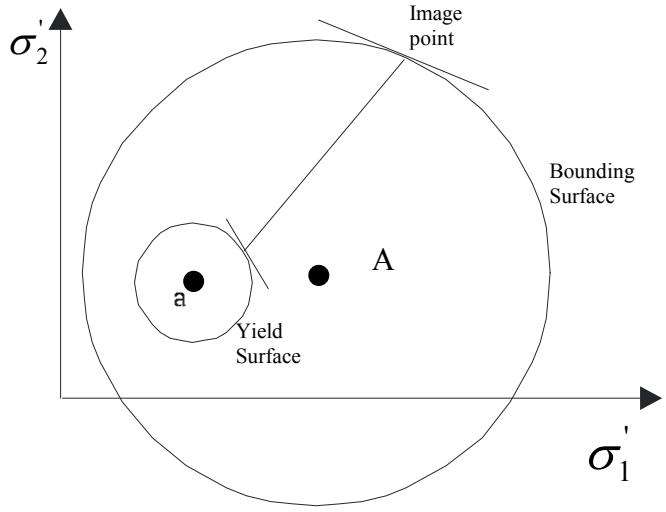


Figure 4-13 Kinematic hardening and bounding surface plasticity

The bounding surface always encompasses the yield surface The yield function for the bounding surface is assumed to depend on an image stress tensor Σ, which is related to a corresponding point on the yield surface, thus,

$F(\Sigma-A,h)=0$

The Three Surface Kinematic Hardening model developed by researchers at City University (*see Stallebrass, S.E and Taylor, R.N 1997 Geotechnique 47, No. 2, pp235-253*) accounts for Kinematic hardening which is a feature of cyclilc loading. In addition, an extra third surface was introduced in order to take into account the effect of recent loading history on the stiffness of overconsolidated soils. The three nested surfaces consist of an innermost surface representing the elasto-plastic yield surface. This is followed by the history surface which determines whether the initial response of the soil is

dilatent or compressive . These two surfaces are in turn is encompassed by a Cam-Clay based bounding surface as in figure (4). Thus, the model accounts for the following:

- High shear stiffness observed at very small strains of the order of 0.04%
- Effect of stress history on stiffness of overconsolidated soils
- Changes in stiffness due to cyclic loading according to a kinematic hardening criterion

The model requires eight parameters. Five parameters are similar to those used by the Modified Cam-Clay model. The other three, which are used to express the relationship between the three surfaces, are determined from a small number of well-controlled stress path tests.

In this model, the stiffness is initially heavily dependant on the recent stress history. The effect of recent stress history decreases as the soil is loaded, becoming negligible after a certain change in stress.

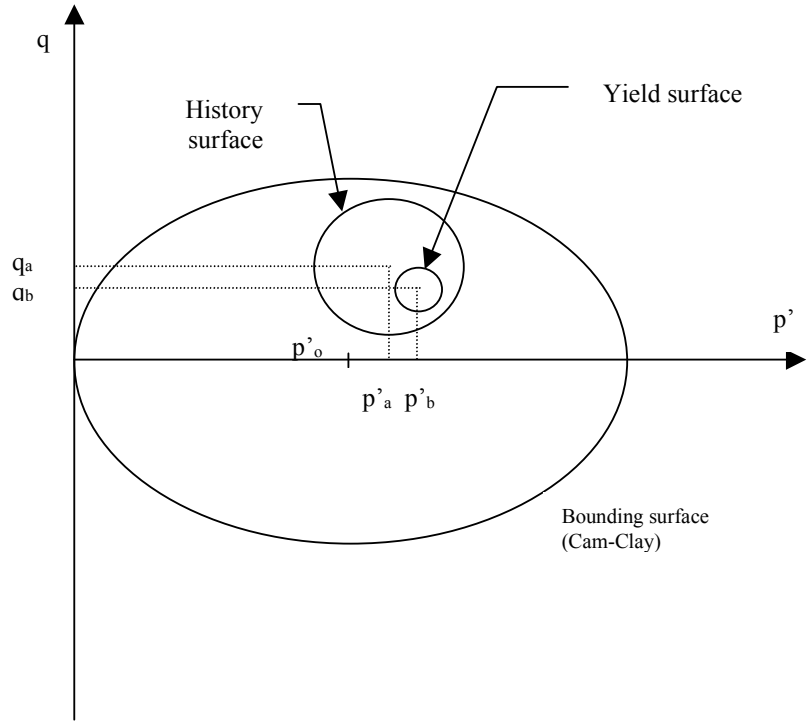


Figure 4-14 Three Surface Kinematic Hardening model developed at City University (UK)

The model is described here in the triaxial (axisymmetric) stress space for simplicity. However, in Crisp the generalised stress invariants (in 3D stress space) are implemented.

Bounding surface (Cam-Clay)

$$(p' - p_o')^2 + \frac{q'^2}{M^2} = p'^2_o \qquad \textit{eqn. 4.6-1}$$

History surface

$$(p' - p_a')^2 + \frac{(q' - q'_a)^2}{M^2} = T^2 p'_o{}^2 \qquad \textit{eqn. 4.6-2}$$

where p_a' *and* q'_a are the centre of the history surface

The Yield surface is represented by

$$(p' - p_b')^2 + \frac{(q' - q'_b)^2}{M^2} = T^2 S^2 p'_o{}^2 \qquad \textit{eqn. 4.6-3}$$

where p_b' *and* q'_b are the centre of the yield surface.

The dimensions of the three surfaces are linked by the two fixed ratios T and S. S is the ratio of the size of the yield surface to the history surface and T is the ratio of the size of the history surface to the bounding surface

If the stress state is within the yield surface, deformation is governed by the familiar isotropic elastic constitutive law, otherwise the stress-strain behaviour is elasto-plastic with associated flow rule on all surfaces and a hardening rule which extends the standard Modified Cam-Clay hardening rule, linking the expansion and contraction of all three surfaces to changes in volumetric strain. The ratios T and S are the extra parameters required by the 3-SKH model in addition to a third parameter needed for the specially formulated hardening rule.

$$\begin{bmatrix} \delta\varepsilon_v^e \\ \delta s_v^e \end{bmatrix} = \begin{bmatrix} k^*/p' & 0 \\ 0 & 1/3G_{ec}{}^1 \end{bmatrix} \begin{bmatrix} \delta p' \\ \delta q \end{bmatrix} \qquad \textit{eqn. 4.6-4}$$

$$\begin{bmatrix} \delta\varepsilon_v^e \\ \delta s_v^e \end{bmatrix} = \frac{1}{h_o} \begin{bmatrix} (p' - p'_b)^2 & (p' - p'_b)^{\frac{(q-q_b)}{M^2}} \\ (p' - p'_b)^{\frac{(q-q_b)}{M^2}} & \left(\frac{(q-q_b)}{M^2}\right)^2 \end{bmatrix} \times \begin{bmatrix} \delta p' \\ \delta q \end{bmatrix} \qquad \textit{eqn. 4.6-5}$$

$$h_o = \frac{(p' - p_b')}{\lambda^* - \kappa^*} \left(p'(p' - p_b') + q\frac{(q-q_b)}{M^2} \right) \qquad \textit{eqn. 4.6-6}$$

Equations (5) and (6) reduce to the Modified Cam-Clay constitutive equations when all the surfaces are in contact. However, the hardening modulus ho, as defined in equation (6) cannot be used without modification as it predicts infinite strains at a number of points on the kinematic surfaces

κ^*	Slope of Swelling line
λ^*	Slope of N.C line
e_{cs}	Void's ratio at critical state
M	Slope of C.S line
G_{ec}	Shear Modulus (KN/m^2)
T	Ratio of extent of Stress History Surface
S	Ratio of extent of Yield Surface
ψ	Exponent of Hardening Modulus
n	Shear Modulus exponent (=0 for Constant G)
m	Shear Modulus exponent (=0 for Constant G)

Eight soil parameters are requires to define the model. Five of these parameters have their origin in the Modified Cam-Clay model. The normal compression line is defined in lnv-lnp' space. The parameters defining the elastic deformations are κ^* and G_{ec} . An estimate of the value of κ^* can be obtained by plotting K'_c/p' against p'/p'_m . for isotropic swelling from the start of constant p' shearing following a complete stress reversal. The parameters n and m are optional and are meant to allow variation of shear modulus according to a power function as follows:

$$G_{ec} = G_{eco} \times P'^{n} \times \left(\frac{P'}{P'_c} \right)^m$$

where G_{eco} is the initial shear modulus and P'_c is size of the cam-clay bounding surface.

4.7 Other Material Properties

4.7.1 Bulk Modulus Of Water

K_w is the bulk modulus of water. When an undrained response is required (in a non consolidation analysis), K_w is normally set to a value between 50 and 500 times K' - the effective stress bulk modulus for the soil skeleton. When a drained analysis is required K_w is set equal to zero. The theoretical basis for this is explained in more detail in Section 8.5.

It is possible to work in terms of total stress, using $\nu_u = 0.49$ and $K_w = 0$, but pore pressures will not be calculated explicitly. The use of effective stress parameters (ν' for CSSM models, E' for elastic) and $K_w >> 0$ is more flexible, and has the advantage that pore pressures are calculated directly.

4.7.2 Bulk Unit Weight

γ_{bulk} is the bulk unit weight of the soil. This value is used by CRISP when:

(a) Calculating implicit loads caused by excavation (element removal) or construction (element addition) sequences.

(b) The gravity acceleration field is increased (or decreased) during an analysis (e.g. when modelling a geotechnical centrifuge test).

The unit for γ_{bulk} is weight/unit volume (Permeability

For coupled-consolidation analyses, it is necessary to define the coefficient of permeability in both the horizontal and vertical directions, k_x and k_y. The values input at the beginning will remain constant throughout the analysis, and are not adjusted by the program. However, it is generally accepted that changes in void ratio will cause changes ability, and if you think this could be important in your analysis then you may wish to co odifying the program code accordingly (see, for example, Almeida, 1984).

The unit for k_x and k_y is length/time (see Table 2-1).

4.8 Models For Structural Elements

The constitutive models described above can be used in conjunction with any of the 2D or 3D elements in CRISP. This section deals with the material properties which must be defined for structural elements (so-called because they are generally used to represent structural components, or interfaces between soil and structure).

4.8.1 Bar Elements

The material properties required for a bar element are:

Table 4-9

Symbol	Description
E	Young's modulus
ν	Poisson's ratio
A	Cross sectional area of bar

E and A should be selected in order to give the desired axial stiffness (= EA/L).

4.8.2 Beam Elements

The material properties required for a beam element are:

Table 4-8

Symbol	Description
E	Young's modulus
ν	Poisson's ratio
A	Cross sectional area
I	Second moment of area of cross section

E, A and I should be selected in order to give both the required axial stiffness and flexural rigidity. (Further details can be found in most structural analysis text books, in chapters dealing with the stiffness method.)

4.8.3 Interface Elements

The material properties required for an interface element are:

Table 4-9

Symbol	Description
C	Cohesion
ϕ	Angle of friction (in degrees)
K_n	Normal stiffness
K_s	Shear stiffness
K_s	Residual shear stiffness
t	Thickness of slip element (usually $10 < L/t < 100$), where L = length of the slip element

The slip element should be assigned normal and shear stiffnesses which are consistent with the continuum material(s) either side of it. The k_n value should be calculated from:

$$k_n = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \qquad \textit{eqn. 4.8-1}$$

$$k_s = G \qquad \textit{eqn. 4.8-2}$$

where E and ν are relevant to the continuum. For example, if $E = 1 \times 10^4$ and $\nu = 0.25$, then $k_n = 1.2 \times 10^4$ and $k_s = 4 \times 10^3$.

In this way, the slip element behaves like the continuum, until limiting stress conditions are reached. If σ_n and s are the normal and shear stresses calculated for the slip element, then the limiting shear stress τ is calculated as:

$$\tau = c + \sigma_n \tan\phi \qquad \textit{eqn. 4.8-3}$$

If s is less than τ , then the behaviour is elastic, i.e. the slip elements bind the soil to the structure. However is $s > \tau$ then the shear modulus is set equal to the limiting value of k_{sres} (which is usually about 100 to 1000 times lower than the k_s value). In all subsequent analysis this permits relative slip between the soil and the structure. If the calculated sign (the normal stress in the slip element) is negative (i.e. tensile) then it is assumed that the soil and structure are moving apart. Then the normal

and shear stiffness values are reduced by a very large factor (10^5). This in effect reduces the stiffness (resistance to deformation) and produces a very stretchable interface element.

In this development work, a new effective stress interface element has been introduced in Crisp. This allows the interfacing of consolidating material with drained material. The effective normal stress is then used as the controlling parameter in the Mohr-Coulomb criterion used for the interface element. In addition the existing 'total stress' interface element has been modified to allow improved simulation of stiffness during cyclic shear loading as well as during re-bonding. Finally, a new 3D interface element has been introduced. This has two local shear stresses and one local normal stress.

4.8.4 Basic Theory of 2D Interface element

Goodman's interface element is currently used in CRISP to allow slip to occur between dissimilar materials or materials having a large difference in their stiffnesses. An example of this is the settlement of soil behind a concrete wall where the soil moves relative to the concrete. It is possible to use ordinary refined isoparametric elements with appropriate choice of stiffness, but the interface element allows the use of slender elements without the necessity to refine the mesh. The shear stress along the interface element is limited by the user specified interface shear strength. Mohr-Coulomb yield criterion is used to check whether the shear stress exceeded the maximum shear strength. If so, the residual shear modulus is then used in the calculation of element stiffness. Checks are made on whether the element has gone into tension. If so, both the normal stiffness and the shear stiffness are multiplied by a factor of 1/10000 so that the element can simulate separation.

The nodal coordinates for the 2D interface element are the same for the two rows of nodes along the length dimension of the element. The element thickness is specified as part of the material properties and this is used in the calculation of stiffness. The element geometry is shown below. In Crisp, the mid-side nodes on the shorter sides of the element are dummy nodes, and are there only to satisfy the geometry program. There are two degrees of freedom in the x and y direction for each node, except the dummy nodes which have no degrees of freedom.

The interface element is characterised by the normal and shear stresses. The normal stress, σ, and the shear stress τ, are related by the constitutive equation to the normal and tangential strain as follows:

$$\begin{Bmatrix} \Delta\sigma \\ \Delta\tau \end{Bmatrix} = [D] \begin{Bmatrix} \Delta\varepsilon \\ \Delta\gamma \end{Bmatrix} \qquad \textit{Eq. 4-31}$$

For plane strain condition, the [D] matrix takes the form:

$$[D] = \begin{bmatrix} k_n & 0 \\ 0 & k_s \end{bmatrix} \qquad \textit{Eq. 4-32}$$

where k_n and k_s are the elastic normal and shear stiffnesses.

The element stiffness matrix is evaluated using local coordinates system, thus:

$$[K_e] = \int_{-1}^{1} [B]^T [D][B] J \times t.d\xi \qquad \textit{Eq. 4-33}$$

where B is the strain displacement matrix, D is defined above, |J| is the determinant of the Jacobian matrix and t is the thickness of the interface element.

Once the displacements are found, incremental strains are calculated using equation 8, then incremental stresses are found using equation 1. The normal stress σ is then updated, and the limiting shear stress is evaluated using Mohr Coulomb yield criterion as follows:

$$\tau_{limit} = C + \sigma \tan(\phi) \qquad \textit{Eq. 4-34}$$

where C and ϕ are the cohesion and angle of internal friction respectively.

If τ is found to have exceeded τ_{limit} then, the shear stiffness K_s is replaced by the residual shear stiffness K_{sres} . In addition, a further check is made on the normal stress. If σ is found to be negative which indicates that tension has occurred, then both the normal and shear stiffnesses are set to a very small value by multiplying K_s and K_n by a factor of 1/10000.

4.8.5 2D 'effective stress' interface element

This is the same as that of the traditional 'total stress' interface element, except that the new element allows for the presence of two pore pressure nodes on one side. These nodes may have three degrees of freedom in x and y direction as well as a pore pressure degree of freedom.

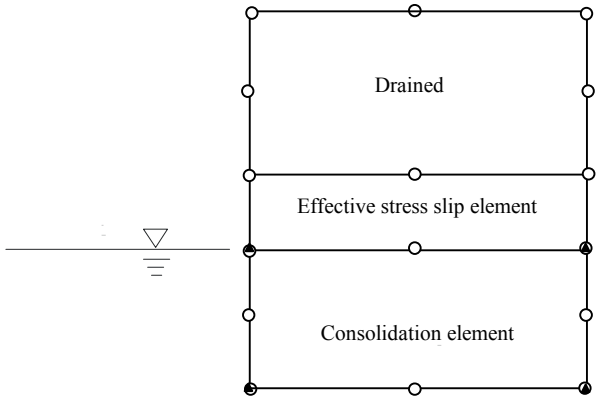


Figure 4-15 ***Geometry of 2D 'Effective Stress' interface element***

Stiffness assembly is the same as for the usual 'total stress' interface element. The new changes were restricted to the stress evaluation stage. Here, the pore pressure is calculated at each Gauss point as follows:

$$\Delta\bar{u} = \overline{N}\Delta\mathbf{b} \qquad \textit{eqn. 4.8-4}$$

where $\overline{N}$ represents the shape functions of the two pore pressure nodes on one side of the interface element, and $\Delta\mathbf{b}$ is a vector of excess pore pressure for these two nodes. The excess pore pressure variables belong to the neighbouring consolidation element, therefore a search is made to identify the adjacent consolidation element before this step is carried out.

The normal and shear stresses are calculated as usual without the pore pressure. The effective normal stress is used in the Mohr-Coulomb yield criterion to check whether the shear stress has exceeded the shear strength. Pore pressure is then added to the normal stresses if interface element is in compression so that correct normal forces can be calculated.

INITIAL CONDITIONS 5

5.1 In Situ Stresses

5.1.1 General

In situ stresses are of vital importance in geotechnical finite element work, for two reasons. Firstly, in an elasto-plastic analysis the stiffness matrix of a finite element will depend on the stress state within the element. In general the stress state will vary across an element and the stiffness terms are calculated by integrating expressions dependent on these varying stresses over the volume of each element. CRISP integrates these expressions numerically by 'sampling' the stresses at particular points within the element and then using standard numerical integration rules.

Secondly, in any analysis involving excavation (whether elastic or elasto-plastic) the self weight loading has to be specified, and this must be in equilibrium with the in situ stresses and applied loads. As will be demonstrated later, equilibrium is of great importance.

The purpose of the in situ stress input data in CRISP is to enable the program to calculate the stresses (and, for the CSSM family of models, the size of the yield locus specified by p'_c) before the analysis starts.

In situ stresses are only allowed to vary in the vertical direction - there is no provision for a horizontal variation. The only way to obtain horizontally varying in situ stresses would be to use a previous CRISP analysis to simulate the events leading to the current stress state (making use of the stop/restart option - see Section 8.3). For example, if you wish to start off with a sloping ground surface, you could commence with a level surface, simulate the excavation, and then allow the ground to come into long term equilibrium with the excavated geometry. However, if this is the case, it could be argued that the previous analysis might just as well be incorporated in the main run.

Earlier versions of CRISP had an option to set in situ stresses to zero everywhere in the mesh, but this has been phased out with the release of CRISP. It was only useful for the most trivial of analyses, since most soil mechanics problems require the specification of non-zero in situ stresses. If you do need to set stresses to zero this can, of course, be done as a special case of a linearly varying stress distribution.

5.1.2 Values Required

The in situ stress parameters required by CRISP (at all integration points) are as follows:

Table 5-1 *In Situ Stress Parameters*

Symbol	Description
σ'_x	Effective horizontal stress in x direction (r - radial)
σ'_y	Effective horizontal stress in y direction (z - vertical)
σ'_z	Effective horizontal stress in z direction (θ - circumferential)
τ_{xy}	Shear stress in xy plane for plane strain (rz in axisymmetry)
τ_{yz}	Shear stress in yz plane (3D analysis only)
τ_{zx}	Shear stress in zx plane (3D analysis only)
u	Initial pore pressure (usually the static head)
$\bar{U}$	Excess Pore Water Pressure
p'_c	Isotropic preconsolidation pressure (the value of p' on the current yield locus when q = 0) - for CSSM models only

You do not, however, need to specify these values for every integration point - they are, instead, defined at horizontal elevations throughout the mesh as described in the next section (5.1.3).

5.1.3 Defining In Situ Stresses

In CRISP, in situ stresses can be automatically calculated for reference elevations using the In Situ Stress Converter (see Section 6.8 in the User Guide). This uses the methods described below to automatically generate stresses and (if appropriate) CSSM parameters, prompting you for the required parameters (see Table 5-1).

5.1.3.1 In Situ Stresses at Reference Elevations

For the purposes of specifying in situ stresses, the mesh is divided into a number of horizontal layers. For most problems the in situ stresses do not vary in the horizontal direction and it is assumed that the stresses vary only with depth. Therefore you specify a set of reference elevations (differentiating the layers) along a vertical section, together with the stresses at these elevations. The in situ stresses at the integration points are interpolated from the stresses specified at these elevations, Figure 5-1. (For the locations of these integration points, see Figure 8-1).

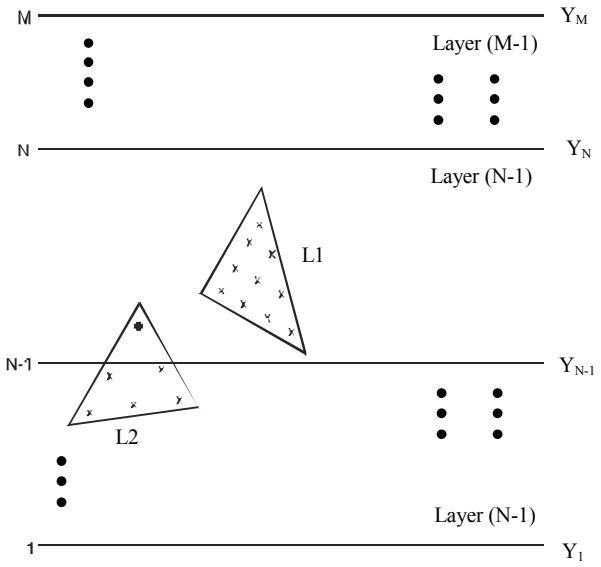


Figure 5-1 *Interpolating In Situ Stresses*

If all the in situ stress values vary linearly throughout the mesh, then it is sufficient to define these values at just 2 elevations (at the top and bottom of the mesh). These two elevations then form a single in situ layer. If there are N in situ reference elevations then there are N-1 in situ layers. The parameters have the same value at a given depth across the mesh, i.e. they only vary in the vertical (y) direction.

This scheme even allows for a discontinuity in a particular stress value (e.g. the effective horizontal stress can be different either side of an interface between two different layers). Then two reference elevations can be specified with the same y co-ordinate but with different values of the horizontal effective stress. It should be noted that some stress parameters (example σ'_y, u) are expected to be continuous and hence should have the same value at both these reference elevations.

The in situ reference elevations should cover the whole of the mesh. If there are N in situ reference elevations then no part of the mesh which requires in situ stresses to be assigned to it should lie outside the range of y co-ordinates specified by the elevations 1 and N. The only exception to this is any part of the mesh which is removed prior to the beginning of the analysis; these can lie outside the range of the in situ reference elevations without causing errors.

Within CRISP the values of σ'_x σ'_y σ'_z τ_{xy} u and p'_c are linearly interpolated at each integration point (for all the elements) from the two in situ reference elevations which lie on either side of it (see Figure 5-1). Using the y co-ordinate of the integration point a search is made to find the corresponding in situ layer. The interpolated values are then entered in the relevant arrays.

5.1.4 Linear Variation

A linear variation of in situ stress is obtained by defining reference elevation parameters for just the top and the bottom of the mesh.

Some examples of linear distributions that can be captured in this way are given below:

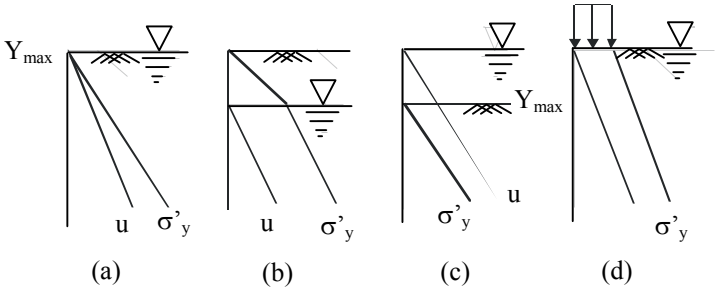


Figure 5-2 *Linearly Varying In Situ Stress Distributions*

Figure 5-2(a) to (d) illustrates the following situations:

(a) Ground water level is at ground surface.

(b) Ground water level is below ground surface.

(c) Submerged, with ground surface *h* units underwater.

(d) Surcharge acting on surface, ground water level at ground surface.

5.1.4.1 Excess Pore Water Pressure at Reference Elevations

When performing finite element analysis for an undrained problem, it is often assumed that the in-situ total pore pressure distribution varies linearly with depth and is hydrostatic (ie $U=d.\gamma_w$). In some geotechnical problems, this variation is non-linear as can be seen in the diagram below. Typical in-situ examples are: young clays, hydraulic fills and underdrained deposits. This change in the distribution of pore pressure occurs over a period of time and would necessitate a coupled consolidation analysis. The total pore pressure distribution curve would start up with the hydrostatic line and would eventually reach the final pore pressure taking the shape of isochrones. The points A1 and A2 are the points at which the pore pressure remains fixed throughout.

In order to carry out this analysis, the user must have the in-situ total pore pressure distribution corresponding to any of the dotted curves in Figure 5-3. The excess pore pressure can then be found as follows:

$\bar{U}$ *=in-situ total pore pressure - hydrostatic pore pressure*

For this example, the excess pore pressure is negative.

CRISP analysis could be carried out to find the settlements corresponding to the final state of pore pressure distribution . The user would need to provide the following parameters:

- Suitable material parameters including values for soil permeability and a suitable bulk unit weight.
- Pore pressure fixities for points A1 and A2. In this example the pore pressure fixity for point A1 is zero.
- In-situ stresses, including total pore pressure distribution, corresponding to the hydrostatic state
- The initial excess pore pressure as defined above. Bearing in mind that these must be negative for underdrained examples such as this.

- Suitable time period and time steps leading to the final pore pressure line

The in-situ total pore pressure values are inserted in the field PWP in the in-situ stresses box. The initial excess pore pressure values are entered in the field $\bar{U}$ in the in-situ stresses box.

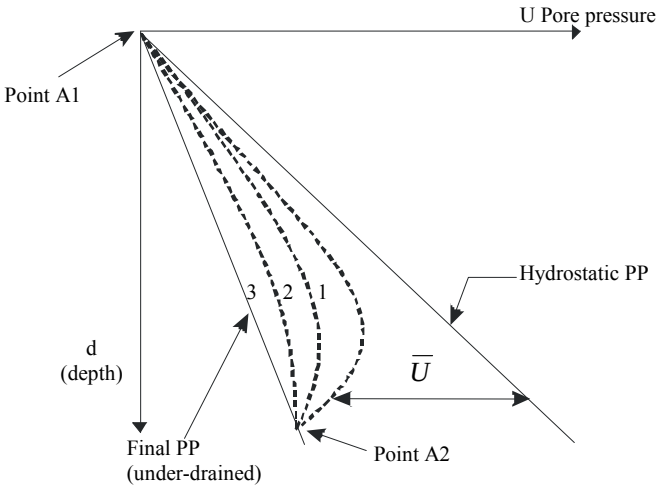


Figure 5-3 Pore pressure distribution for various stages for an underdrained soil.

5.1.5 The In Situ Stress Converter

the In Situ Stress Converter provides a utility that will automatically perform the calculations required to derive the in situ stresses given the following parameters listed in

Table 5-2 Parameters Required by the In Situ Stress Converter

Symbol	Description
γ_{bulk}	Bulk density of soil
γ_{water}	Density of water
K_0	In situ earth pressure coefficient
$Y_{water\ table}$	Y co-ordinate of water table
Overburden Pressure	Maximum pressure previously applied to the soil's surface
λ	Slope of the normal consolidation line (CSSM only)
κ	Slope of the swelling line (CSSM only)
M	The frictional constant (CSSM only)
e_{cs}	Void ratio on the CSL at p' = 1 (CSSM only)

For CSSM models, the procedure used to calculate p'_c and e is as follows:

1. Calculate K_{nc} according to eq. 5.1-1 and 5.1-5.

2. Calculate p'_c according to 5.1-7 through 5.1-11 for (Cam-Clay) or 5.1-13 for (Modified Cam-Clay)

3. Calculate e according to 5.1-13 through 5.1-15

5.1.6 CSSM Models

For CSSM models it is important to try to establish the in situ stress state as accurately as possible. This is because the displacements predicted by an analysis are quite sensitive to the relative amounts of elastic to plastic straining that take place.

5.1.6.1 Horizontal effective stress σ'_x

In specifying the in situ stress parameters shown in Table 5-1, the greatest difficulty lies in the determination of σ'_x (and hence σ'_z). There are basically three approaches to determining the in situ stresses:

Method 1

An analysis is performed (either using CRISP or by hand) in which a soil column is subjected to the stress history which is believed has been applied to the soil deposit in practice. This approach has the merit of being theoretically consistent with subsequent analysis but it suffers from the disadvantage that Cam clay (and to a lesser extent modified Cam clay) are not very successful in predicting values of K_{nc}. (This is because the one dimensional compression is a very constrained situation in which the magnitude of elastic strains has a significant influence on the value of K_{nc} that is predicted).

Method 2

The second approach is to use a rather more empirical method based on the data accumulated by Wroth (1975). In Wroth's method the value of K_{nc} is taken as:

$$K_{nc} = 1 - \sin\phi' \qquad \textit{Eq. 5.1-1}$$

following Jaky (1944). There is evidence that this approximation is sufficiently accurate for most engineering purposes according to the experimental data that is available (Wroth, 1972). Wroth proposes two relationships between K_0, K_{nc} and OCR ($OCR = \sigma'_{vmax}/\sigma'_v$):

$$K_0 = OCR\, K_{nc} - \frac{\nu'}{1-\nu'}(OCR - 1) \qquad \textit{Eq. 5.1-2}$$

and

$$m\left(\frac{3(1-K_{nc})}{1+2K_{nc}} - \frac{3(1-K_0)}{1+2K_0}\right) = \ln\left(\frac{OCR(1+2K_{nc})}{1+2K_0}\right) \qquad \textit{Eq. 5.1-3}$$

Equation 5.1-2 is obtained by considering elastic unloading from the normally consolidated state and gives a good fit to the existing data for a number of soils up to an OCR of about 5. The values of ν' necessary to fit the observed data were determined by Wroth to be in the range 0.254 to 0.371 for eight different soils. Equation 5.1-3 was proposed as being valid up to higher values of OCR and was obtained from the observation that an unloading plot of q/p' versus ln(p') is a straight line relationship. An empirical constant, m, has been shown by Wroth to be linearly related to Plasticity Index (PI) for a number of soils.

In situ stresses can be calculated using Wroth's second approach (5.1-3) described above. Plasticity index can be used to obtain the empirical constant m:

$$m = 0.0022875PI + 1.22 \qquad \textit{Eq. 5.1-4}$$

and K_{nc} can be calculated from 5.1-1 and 5.1.-5

$$\sin\phi' = \left(\frac{3\,\mathrm{M}}{6+\mathrm{M}}\right) \qquad \textit{Eq. 5.1-5}$$

Method 3

Finally, you could use the empirical relationship:

$$\mathrm{K}_0 = \mathrm{K}_{nc}.OCR^{\phi} \qquad \textit{Eq. 5.1-6}$$

where ϕ is in radians.

5.1.6.2 Preconsolidation Pressure p'c

With knowledge of the past stress history of the ground or the laboratory sample you can determine the variation of OCR with depth. Using an empirical approach or other data you have to establish the relationship between OCR and K_0 as mentioned above. Knowing K_0 you can calculate the variation of σ_x. Using the OCR value you can calculate the σ'_{vmax} (= σ'_{ymax}).

$$\sigma'_{y\max} = OCR.\sigma'_y \qquad \textit{Eq. 5-7}$$

where,

OCR	Overconsolidation ratio
K_o	Coefficient of in situ earth pressure
K_{nc}	Coefficient of in situ earth pressure in the normally consolidated state
σ'_y	Effective vertical stress
σ'_{ymax}	Maximum previous vertical effective stress

Assuming that the soil would be in a normally consolidated state when it was subjected to the maximum vertical pressure:

$$\sigma'_{x\max} = \mathrm{K}_{nc}\sigma'_{y\max} \qquad \textit{Eq. 5.1-8}$$

$$p'_{\max} = \frac{2\sigma'_{x\max} + \sigma'_{y\max}}{3} = (2\mathrm{K}_{nc} + 1)\left(\frac{\sigma'_{y\max}}{3}\right) \qquad \textit{Eq. 5.1-9}$$

$$q_{\max} = \sigma'_{y\max} - \sigma'_{x\max} = (1 - \mathrm{K}_{nc})\sigma'_{y\max} \qquad \textit{Eq. 5.1-10}$$

Substituting these values of p' and q in the expression for theCam clay (or Modified Cam clay) yield locus you can obtain the p'_c value for that depth.

for Cam clay:

$$p'_c = p'_{\max} . \exp\left(\frac{-q_{\max}}{M p'_{\max}} \right)$$ *Eq. 5.1-11*

for Modified Cam clay:

$$p'_c = \left(\frac{\left(\frac{q_{\max}}{M} \right)^2}{p'_{\max}} \right) + p'_{\max}$$ *Eq. 5.1-12*

where M is the gradient of the critical state line.

Using 5.1-8 to 5.1-12 you can derive a relationship between p'_c and σ'_y which can be used to calculate p'_c and σ'_y. This is repeated for any other depth (and in situ reference elevation). If the variation of any of these parameters is non-linear (usually σ'_x) then you have to approximate the variation as piecewise linear over a number of in situ layers.

5.1.6.3 Initial Void Ratio

The initial void ratio can be calculated at any given depth once p' and q and p'_c are known. Using the following CSSM parameters e_{cs} (=Γ-1), κ and λ the void ratio is calculated in the following manner.

First assume that, in e : ln p' space, the intersection of the CSL and the swelling line passing through the in situ stress state is given by e_a and p'_a respectively (point A in Figure 5-4). Furthermore, assume that e and p' represent the initial void ratio and effective mean normal stress respectively (point P in Figure 5-4). Then for the CSL:

$$e_a = e_{cs} - \lambda . \ln(p'_a)$$ *Eq. 5-13*

where

$$p'_a = \frac{p'_c}{2} \quad \text{(for modified Cam clay)}$$

$$p'_a = \frac{p'_c}{2.718} \text{ (for Cam clay)}$$

Then along the line passing through the initial stress state (point P),

$$e - e_a = \kappa . \ln\left[\frac{p'_a}{p'} \right]$$ *Eq. 5-14*

giving:

$$e = \mathrm{e}_{\mathrm{cs}} - (\lambda - \kappa) . \ln p'_a - \kappa . \ln p'$$ *Eq. 5-15*

Note that the initial void ratio depends only on the initial p' and is independent of the initial value of q.

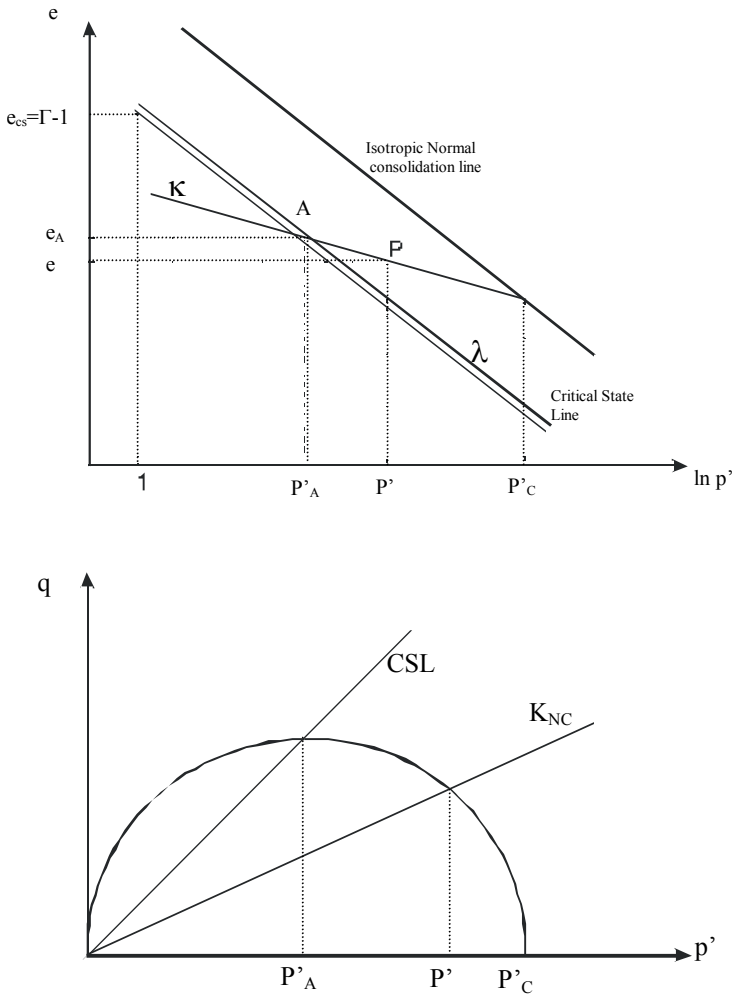


Figure 5-4 *Calculation of Initial Void Ratio*

5.1.6.4 In situ stress history

Unlike the elastic models, the CSSM models require the specification of non-zero stress state as a starting point; in other words they require a stress history. The normal effective stresses must be such that the initial p' is greater than zero at any integration point. The initial p'_c value (size of the initial yield locus) also cannot be zero at any integration point. This would indicate a yield locus of zero size. Note that these apply to values at integration points.

You can specify a linearly increasing variation of stresses (and hence p') with depth. It is also possible to specify zero stresses at the ground surface, together with a zero value of p'_c. This is because none of the integration points in any of the element types provided actually lie on the element sides. All of them lie within the element, although some do lie quite close to the side (e.g. integration points 10 to 15 in the cubic strain triangle, Figure 8-1). Integration points close to the ground surface will have very small values of p' and p'_c if the soil is in a normally consolidated state. This results in a very small stiffness near the ground surface which may give rise to large deformations.

This very low stiffness may cause problems, depending on whether or not loads are applied to the ground surface, perhaps causing shear failure in the analysis (watch out for p' becoming negative, which always indicates trouble). In the analysis of an embankment on a soft foundation, the integration points nearest the surface will probably become stressed in such a way that the yield ratio will increase at an unacceptable rate (see Sections 6.1.3 and 7.7.3). This can be avoided by using a

5.1.7 Position Of The Water Table

In general, the position of the water table is assumed to be ground level. However it need not be so. The only requirement is that the chosen ground water table should coincide with a horizontal line which lies along element boundaries. For this reason, CRISP cannot be used to model phreatic surfaces.

Consider the example given in Figure 5-5. Lines 1-1, 2-2, 3-3 form element boundaries (i.e.. there are nodes along this line and element sides coincide with this line). These are 'natural' boundaries between layers of elements. The table cannot be along level 4-4 because it does not form a 'natural' boundary. This is for the simple reason that there is no way of separating the elements within the zone 1-1-2-2 into 2 zones which are on either sides of line 4-4. The presence of water table along 4-4 makes it impossible to specify different set of material properties to the soil below and above the water table. This is the only reason why the water table should coincide with a horizontal line which forms a natural separation of different layers.

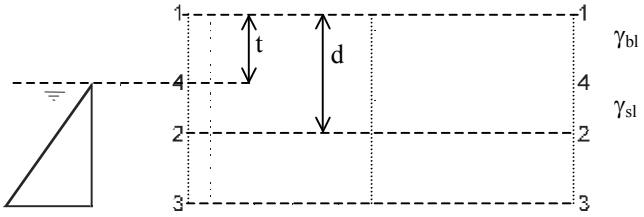


Figure 5-5 *Position of Water Table*

For the zone above the water table, the unit weight for the material should be set to the dry unit weight. For the zone below the water table, the unit weight for the material should be set to the bulk (saturated) unit weight.

The in situ vertical stresses specified should take into account the difference in the unit weight of the soil above and below the water table. At a depth d:

$$\sigma'_y = \gamma_{bl} t + \gamma_{sl}(d - t)$$
$$u_0 = (d - t)\gamma_w \qquad \textit{Eq. 5-16}$$

where t is the thickness of the zone above the water table.

5.1.8 Centrifuge Test Analyses

If this is an analysis of a centrifuge test and if the starting point of the analysis is after equilibrium has been attained at a particular gravity level (say) n g. then the stresses at a depth d are given by :

$$\sigma'_v = nd\gamma_{sub}$$
$$= nd(\gamma_{bulk} - \gamma_w)$$
$$\sigma'_x = \sigma'_z \qquad \textit{Eq. 5-17}$$
$$= \mathrm{K}_0 \sigma'_v$$
$$u_0 = nd\gamma_w$$

where

γ_w	Unit weight of water
γ_{sub}	Submerged unit weight of soil
γ_{bulk}	Bulk unit weight of soil
σ'_x	Effective horizontal stress
σ'_y	Effective vertical stress
N	Gravity level in centrifuge (for field problems n = 1)
K_0	Coefficient of earth pressure

The gravity factor Fg should be set to n, and the dimensions of the mesh should be set to centrifuge model size.

If you are carrying out the analysis at prototype scale, the size of the mesh will be n x the centrifuge model size. In this case set Fg = 1.

5.2 Boundary Conditions

5.2.1 Displacements

Any displacement fixities (i.e. restraints along the mesh boundary) only need to be specified once, either at the in situ stage (in the presence of in situ stresses) or in the first increment block. Once specified these restraints (displacement fixities, zero prescribed displacements) remain effective during the rest of the analysis. Therefore these need not be re-specified for each and every increment block - further specification will lead to accumulated values.

It should be noted that only the displacement fixities are specified and not the pore pressure boundary condition, at the in situ stage. **Pore pressure boundary conditions are not specified at the in situ stage**.

5.2.2 Excess Pore Pressures

These may not be specified at the in situ stage, for reasons which are explained in Section 6.5.

5.2.3 Applied Loads

You have to specify the external loading (pressure loading along the boundary) and self weight loading (due to body forces) that is in equilibrium with the in situ stresses. The zero displacement boundary condition has to be specified along the boundary that is restrained. In specifying these conditions you must consider the entire boundary of the mesh and ensure that along any part of the boundary which is loaded (i.e. not free of stress) either the pressure loading or the restraint condition is specified.

The nodal loads equal to the *in situ stresses* are calculated and compared with the nodal loads due to the *external loading* (including self-weight loads). These must be equal at all nodes except for the ones along any restrained boundary. This is known as the equilibrium check. Any difference between these two loads are known as the error in equilibrium (printed under the heading of out-of-balance loads). It should be noted that these external loads described above are not the same as the loads

applied later during the course of the analysis. These are the loads which maintain equilibrium at the in situ stage.

THE EQUILIBRIUM CONDITIONS MUST BE SATISFIED BEFORE THE COMMENCEMENT OF THE ANALYSIS.

Similar to the 2D case where the element *edges* are restrained, for the 3Dcase element *faces* can be fixed in a particular direction. However at present there is no option in CRISP to calculate the nodal loads equivalent to the pressure applied on an element face, for the 3D case. Therefore you have to calculate the equivalent nodal loads and input it directly as nodal loads (using option (a) in record H2 - see Appendices A and B). When calculating the nodal loads it should be remembered that the intermediate nodes along the element side also carry part of this applied load.

5.3 Equilibrium

5.3.1 General

Throughout this manual, the following terms will appear frequently and is important that you understand their exact meaning and in what context they are being used:

(a) Equilibrium error

(b) Equilibrium check

(c) Out-of-balance load

5.3.2 Vertical Equilibrium

At all in situ reference elevations the following condition should be satisfied.

$$\sigma_y = \sigma'_y + u_0$$
$$= Fg.\gamma_{bulk}.(y_N - y_{GS}) + surch\arg e \qquad \textit{Eq. 5-18}$$

where

Y_N	y co-ordinate of the reference elevation
Y_{GS}	y co-ordinate at the ground surface
γ_{bulk}	bulk unit weight
Fg	gravity field factor - 0 for laboratory test analysis - 1 for field situations (full-scale) - n for centrifuge test (gravity level = ng)

If the above equation is not satisfied, it will lead to an error in equilibrium (printed under the heading PERCENTAGE ERROR IN EQUILIBRIUM) at in situ stage. These, in general, should be negligible in any analysis, regardless of the constitutive model used.

If, on the other hand, the equilibrium equation is not satisfied and if equilibrium errors are still present, then check that:

- All the fixities along the entire mesh boundary (see Section 5.2).
- The boundary stresses applied at the in situ stage are consistent with the in situ stresses.
- The surcharge pressure in the equilibrium equation above has been specified correctly
- There are no gaps in the mesh.

5.3.3 Equilibrium Check

This is a check carried out 1) at the in situ stage and 2) at the end of every increment of the analysis. This is a check carried out by CRISP to ensure that externally applied loads are in equilibrium with the internal stresses.

The results of these checks are printed at the in situ stage and at the end of every increment under the heading 'EQUILIBRIUM CHECK'.

For comparison purposes the externally applied loads and the internal (element) stresses are converted (using numerical integration) into equivalent point loads for all the nodes in the mesh. There is one set for the external loads {Pt} and one set for the internal stresses {Psn}. The largest value in the x and y directions are picked up from {Pt} These are the values printed under the heading 'MAXIMUM EXTERNAL LOAD' in the Main Program output. The difference:

$$\{Perr\} = \{Pt\} - \{Psn\} \qquad \textit{Eq. 5-19}$$

is calculated for all the nodes. If equilibrium is satisfied (as it should be) then {Perr} should be nearly zero at all the nodes. However if you find that these out-of-balance loads are not zero then it would indicate data errors. You must check the in situ data input to find the mistake.

$$EQerr = \frac{\{Perr\}\max}{\{Pt\}\max \text{ x } 100} \qquad \textit{Eq. 5-20}$$

is expressed as a % and printed under the heading

PERCENTAGE ERROR IN EQUILIBRIUM.It should be less than 1% at the in situ stage (but usually much smaller). This level of equilibrium must be maintained throughout the analysis for the results to pass the first test of acceptability (only true for analysis using wholly elastic or CSSM models - NOT elastic-perfectly plastic).

Warning If you do not have equilibrium at the in situ stage, then there is definitely something wrong with the data - DO NOT PROCEED WITH THE ANALYSIS. Element types, constitutive models, stiffness contrasts, problems with computer precision, etc. have no bearing on equilibrium at the in situ stage.

Your analysis MUST be in equilibrium at its onset.

5.3.4 Sufficiency Of Equilibrium Check

You should be quite clear that the equilibrium check in CRISP does NOT provide sufficient verification of the analysis. It is a necessary condition, but passing the equilibrium check does not automatically confer acceptability on the results.

If you look at the Examples Manual, you will notice that the equilibrium checks have one thing in common, equilibrium is satisfied to less than 1%, thus meeting the above acceptability criteria. However, considering Example 3 (undrained triaxial compression test using modified Cam clay) you will notice that the results are incorrect when as few as 6 increments are used. For this example, the values of p', q and u calculated at failure are 10-20% in error, despite equilibrium being satisfied. Approximately 50 increments were needed in this example before accurate answers were obtained.

Warning Under NO circumstances should you conclude from Example 3 that 50 increments are adequate, or that 6 increments will result in errors of 10-20% in any type of analysis which involves CSSM models.

FINITE ELEMENT ANALYSIS 6

6.1 Increment Blocks

6.1.1 General

CRISP allows you to sub-divide the analysis into one or more increment blocks, with each increment block comprising one or more increments. This facility is provided for two reasons:

- If the loads for each analysis increment had to be specified separately there would be a very large amount of data input needed for most problems. Much of this information would be repeated many times (e.g. which element sides were being loaded, which nodes had non-zero prescribed displacements, etc.) - such increments can be grouped together as an increment block.

- When performing an excavation (or construction) analysis CRISP calculates the implied loadings caused by the removal (or addition) of specified elements. When the material behaviour is non-linear, these implied loadings will often be too large to be applied in a single increment. The use of an increment block spreads these implied loads over several increments. (Note that this procedure introduces an extra approximation in the modelling of excavations: the stiffness of an element is removed entirely in the first increment of a block whereas the loads are spread over all increments in the block.)

6.1.2 Number Of Blocks

The number of increment blocks required will depend on the analysis. In general, a block will be required whenever there is a distinct operation involving:

- Element removal or addition.

- Application of boundary loads.

- Imposition of prescribed displacements (zero and non-zero).

- Changes to boundary drainage conditions (consolidation analyses only), either separately or in combination.

In some analyses as few as one block is sufficient, in others, several tens of blocks might be needed. When modelling construction activities it is convenient to use a separate block for each new phase of activity (e.g. wall installation, excavation stage, prop removal, long term equalisation of pore pressures, etc.).

6.1.3 Number Of Increments

CRISP uses the incremental method which, in contrast to iterative methods, tends to give a response which continually drifts away from the true response where the stress strain behaviour is non-linear. Consequently, when a non-linear and/or consolidation analysis is performed it is necessary to divide the loading and/or time span of the increment block into a number of increments. For example, if a total stress of 20 kPa is applied to part of the finite element mesh boundary, it might be divided into ten equal increments of 2 kPa each of which is applied in turn. The total number of increments that are necessary will vary from problem to problem but in general about fifty increments would be required in a drained or undrained analysis using any of the CSSM models. CRISP calculates the incremental displacements for each increment using a tangent stiffness approach i.e. the current stiffness properties are based on the stress at the start of each increment.

While it is desirable to use as many increments as possible to obtain accurate results the escalating computer costs that this entails will mean that compromise between accuracy and cost is inevitable. If using CSSM models the recommended way to determine whether enough increments have been used is to examine the values of yield ratio (YR) at each integration point (see Chapter 7). A change in size of yield locus of less than +/- 2% (i.e. from 1.02 to 0.98, depending on whether hardening or softening) should indicate sufficient accuracy. If this increases to +/- 5% (1.05 to 0.95), more increments should be used (i.e. size of load steps reduced). More extensive discussion can be found in Section 7.8.

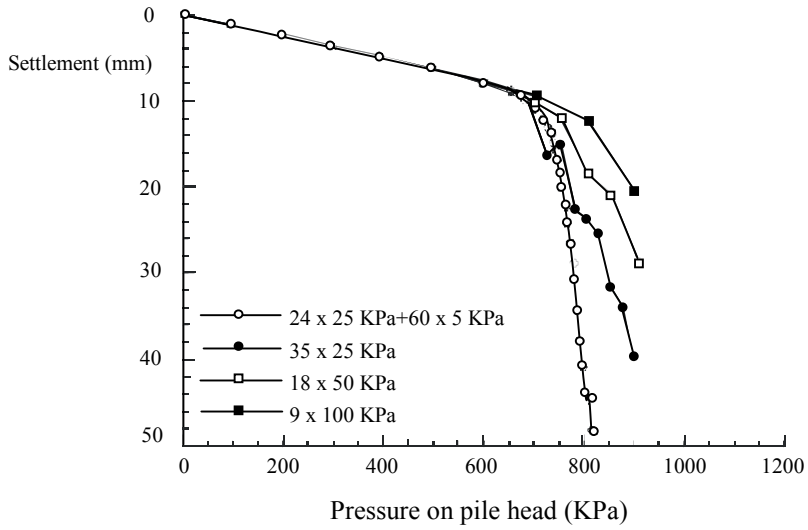
Any analysis which uses purely linear elastic models does not require more than one increment. However if you mix linear elastic models with other soil models then it is the latter which will govern the required number of increments.

If you are analysing a problem for which you have no prior experience with CRISP, you are should compare the results of at least two analyses with different numbers of increments (for the same finite element mesh). The Double Increments feature allows you to easily apply this recommendation.

For example, if the analysis is that of the settlement under a flexible footing then plot the variation of central settlement (or average settlement) against loading for both analyses. If these are these significantly different then discard the analysis which used fewer increments and repeat the analysis with twice the number of increments again. Repeat the comparison of the results and increase the number of increments if necessary.

When two successive analyses produce sufficiently similar results, then you will have established the required number of increments. Note the similarity with assessing the adequacy of a given mesh refinement. Figure 6-1 shows the effect of using more increments with smaller load steps.

If a unique solution exists to the problem you are analysing, then using more increments should converge towards this solution.



Influence of number of increments on load/settlement curve
CRISP – Undrained analysis, K'_o=0.61

Figure 6-1 *Effect of Number of increments*

6.1.4 Time Steps

Time steps are only required in a coupled (consolidation) analysis. The size of the steps (Δt) you select for a given increment should be based on a consideration of the following factors:

(i) Excess pore pressures are assumed to vary linearly with time during each increment.

(ii) In a non-linear analysis the increments of effective stress must not be too large (i.e. The same criteria apply as for a drained or undrained analysis).

(iii) It is advisable to use the same number of time increments in each log cycle of time. Thus, for linear elastic analysis the same number of time increments would be used in carrying the analysis forward from one day to ten days as from ten days to hundred days. Not less than three time steps should be used per log cycle of time (for a log base of ten).

According to this, a suitable scheme might be:

Table 6-1 *Example Time Stepping Schemes*

Inc No.	Δt	ΣΔt
1	1	1
2	1	2
3	3	5
4	5	10
5	10	20
6	30	50
7	50	100
8	100	200
9	300	500
10	500	1000

This scheme would be modified slightly near the start and end of an analysis, as described in (iv) and (v) below.

(iv) If a very small time increment is used near the start of the analysis then the finite element equations will be ill-conditioned (because division by zero will be attempted inside the program).

(v) When a change in pore pressure is applied at a boundary, the associated time step should be large enough to allow the effect of consolidation to be experienced by those nodes in the mesh with excess pore pressure d.o.f. that are immediately adjacent to the boundary. If this is not done then the solution will produce excess pore pressures which oscillate both in time and space.

The application of (v) will often mean that the true undrained response will not be captured in the solution, though the following procedure usually leads to satisfactory results:

- Apply loads in the first increment (or first few increments for a nonlinear analysis) but do not introduce any excess pore pressure boundary conditions.
- Introduce the excess pore pressure boundary conditions in the increment(s) following the application of the loads, in a separate increment block of their own.

6.1.5 Increment Factors

The applied loading, self weight loads and prescribed displacements/excess pore pressures are specified for the entire increment block, and are applicable to that particular increment block only. The loading and any non-zero prescribed displacement for an individual increment will be a proportion (or ratio) of that of the whole increment block.

There is no restriction on how these loadings and non-zero prescribed displacements are divided among the increments in an increment block. They can be equally divided between all the increments (the default), or distributed unevenly between the increments as desired In the latter case, a separate list of increment factors (or ratios) will be required. (This may be useful in an analysis to enforce

large load increments when the problem is in the elastic state, and smaller load increments as plastic yielding begins to takes place).

It should be noted that, in a given increment block, the same ratios apply to the pressure loading, the gravity loading, the prescribed displacements (and excess pore pressures), and any implied loadings due to removal or addition of elements.

The sum of the individual ratios used in a block must be equal to 1. However, some of these ratios can take zero values (as illustrated in the example below).

6.1.6 Time Increment Factors

In a coupled consolidation analysis, a total time step must be specified for the entire block. The default would then be for this time step to be equally divided among all the increments in the increment block. However, you can specify directly the time steps for each increment.

Warning No time-step for any consolidation increment may ever be zero.

6.1.7 Output Control

You can control (reduce) the amount of text output produced by the Main Program for:

- Nodal displacements
- General stresses
- Elasto plastic model parameters
- Cumulative strains
- Out-of-balance loads

This is done in two stages. First, specify the range of element numbers, vertex node numbers, and midside node numbers for which text output is required. Then, specify what level of information is wanted:

Table 6-2 Levels of Text Output

Elements	Nodes
none	none
centroids only	vertex nodes only
all integration points	all nodes

This specification is handled via the Output Options and the Output Range dialogue boxes in the CRISP Pre-Processor (see Section 7.3 in the User Guide).

6.1.8 Storage For Post-Processing

In order to use the CRISP Post-Processor to produce plots of the results at selected stages of the analysis, the results at the end of selected increments are written to a disk file. You must specify for which increments you wish output to be recorded. This is dealt with via the Output Options dialogue box (see Section 7.3 in the User Guide).

6.2 Changing Geometry

6.2.1 General

CRISP allows soil constructions or excavations to be modelled in an analysis via the addition or removal of elements as the analysis proceeds. All the elements that appear at any stage in the analysis must have been included in the input data for the Geometry Program. Any number of finite elements can be removed from the parent finite element mesh to form the primary mesh before the analysis is started. (These elements are added later at appropriate stages to simulate construction).

You cannot add any element which has a stress history, hence elements which are added to represent a construction event cannot have CSSM material types assigned to them. Such elements will have to be modelled as an elastic or elastic-perfectly plastic material. If using the latter with the Mohr-Coulomb or Drucker-Prager yield criteria, then specify a small value for cohesion. This prevents the initial zero stress state being on the tip of the yield surface and causing the plastic multiplier ($d\lambda$) to be negative (see Section 7.8.3.1).

CSSM models require the stress history of the soil to be known in order that the yield locus p'_c can be specified. Since p'_c is input at the in situ stage, any element removed to form the primary mesh cannot therefore be from the CSSM family of soil models.

6.2.2 Excavation - Element Removal

Any element (regardless of material properties) can be removed during the course of the analysis to simulate excavation. Its stiffness is removed from the mesh right at the beginning of the increment block concerned, whereas the effect of unloading (self weight plus any surcharge) is spread over all increments in the block.

6.2.3 Filling - Element Addition

Any element which is going to be added (for example, to simulate an embankment construction, as in the example problem 2 in Appendix B) must be removed before the analysis commences. Such elements are not lost or deleted, they are simply inactive until called for by the program.

Elements which will be added can only be assigned wholly elastic or elastic-perfectly plastic material properties. You cannot add elements which have been assigned CSSM model parameters because any element which is being added has zero stresses, hence p' is zero implying that the soil stiffness is zero. This will cause a divide by zero DBOS failure if you try to run CRISP.

6.2.4 Installation - Element Swapping

In the analysis of a diaphragm wall (for example), it may be desirable to start with the original undisturbed ground, and model the excavation of soil followed by the installation of concrete. To do

this in CRISP you must specify two sets of elements occupying the same region (denoted by R, Figure 6-2). This is termed "overlaying" or "super imposing", although only one set of elements is going to reside in region R at any given time during the analysis. They are superimposed in the "parent" mesh, but the second set of elements (in this case, the concrete) is removed at the outset to form the "primary" mesh, consisting of soil only.

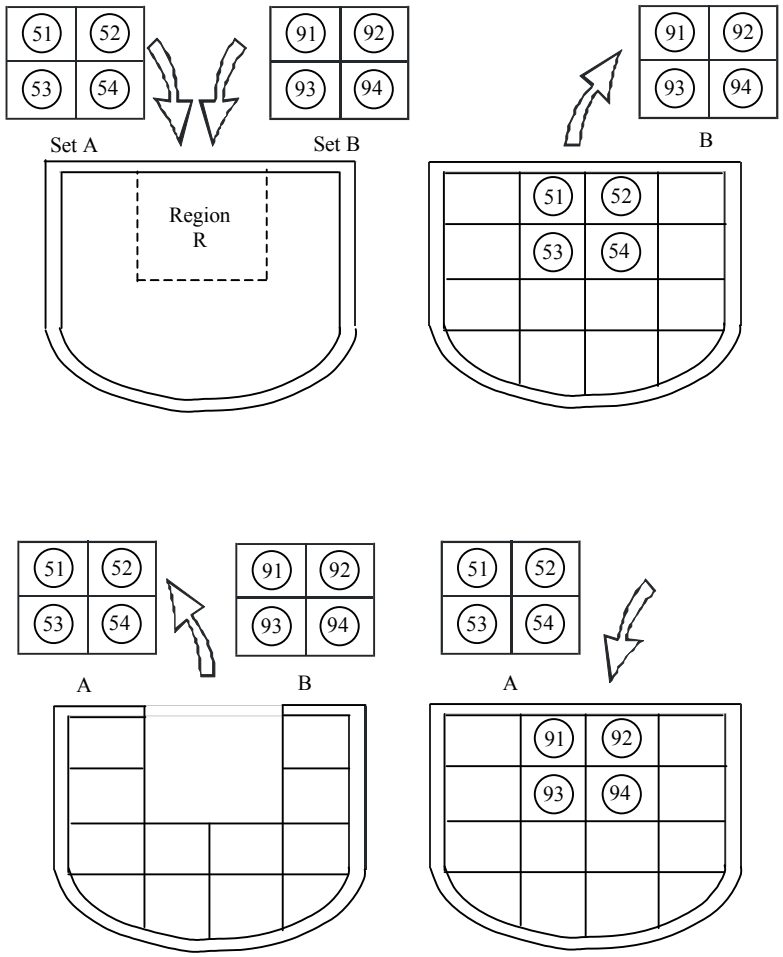


Figure 6-2 *Reinstating Removed Elements*

When a region of the mesh is selected for super imposing, the super imposed elements need bear no resemblance to the elements being overlain. The only requirement is that along the *boundaries* of the super imposed region, all elements in all layers must share common nodes and common edges. Internal to these regions, nodes and edges should not be common, i.e. they should be numbered differently. So in Figure 6-2 above, elements 51 and 52 each share two common nodes and one common edge with elements 91 and 92 and elements 53 and 54 each share three common nodes and two common edges with elements 93 and 94 respectively.

6.2.5 Reinstating Removed Elements

In its present form, CRISP does not permit the reinstatement of elements which have been removed (to simulate excavation) earlier in the analysis. The absence of any element from the mesh is indicated by making the 'program' number for that element negative. However the relevant entries in

the array which stores the stresses still retain their values. This means that the 'stresses' are retained for the removed element. These are not zeroed, and consequently will not have the right stresses when they are placed back in the mesh.

Putting back elements (simulating back-fill) which have been removed is, however, possible in one of two ways:

(i) By using 'Super-imposed' elements

This is precisely the same procedure as described in 6.2.4 above, except that the second set of elements represents the back-fill. It is not possible to model compaction-induced stresses in the fill with the present version of CRISP, which may be a serious deficiency in some applications.

(ii) By changing the program

This is beyond the scope of this guide. Details may be found in the various references listed in Section 1.6.

6.3 Applied Loads

6.3.1 General

You should note the significance of specifying incremental loads in the input data. The total loads acting at any particular time are the accumulation of all the previous incremental loads. Thus if part of the mesh is at first loaded and then subsequently unloaded, it will be necessary to specify negative incremental loads. Total loads and total fixities remain in force from one increment block to the next if no action is taken to remove them.

The following example is intended to clarify these points in the context of a consolidation analysis:

(a) Part of the boundary of a soil mass is loaded with a load of ten units (applied in ten equal increments).

(b) Consolidation takes place for a period of time (over ten increments).

(c) The load is removed from boundary of the soil mass in five equal increments.

(d) Consolidation takes place with no total external loads acting.

Table 6-2

Increment number	Incremental load applied	Total load acting
1	1	1
2	1	2
3	1	3
4	1	4
5	1	5
6	1	6
7	1	7

Increment number	Incremental load applied	Total load acting
8	1	8
9	1	9
10	1	10
11	0	10
12	0	10
...	...	...
21	-2	8
22	-2	6
23	-2	4
24	-2	2
25	-2	0
26	0	0
27	0	0
28	0	0
29	0	0
30	0	0

Another way of representing this sequence of loading into input data would be to make increments 1 to 10 the first increment block with an incremental load of 10 units and 10 load factors equal to 0.1. The second block (increments 11 to 20) would have no incremental loads and the third block (increments 21 to 25) would have an incremental load of -10 with 5 load factors equal to 0.2.

6.3.2 Normal And Shear Stresses

Normal and shear stress distributions (or pressure loadings) must be specified over an entire element edge. They are not available for 3D analysis (where everything is done through point loads).

The components of σ and τ are defined at all nodes along an element side - 3 nodes for linear strain elements and 5 nodes for cubic strain elements. A 3-noded edge could cope with constant, linearly and quadratically varying distributions as required (see Figure 6-3). The sign conventions to follow are defined in Section 2.4.2.

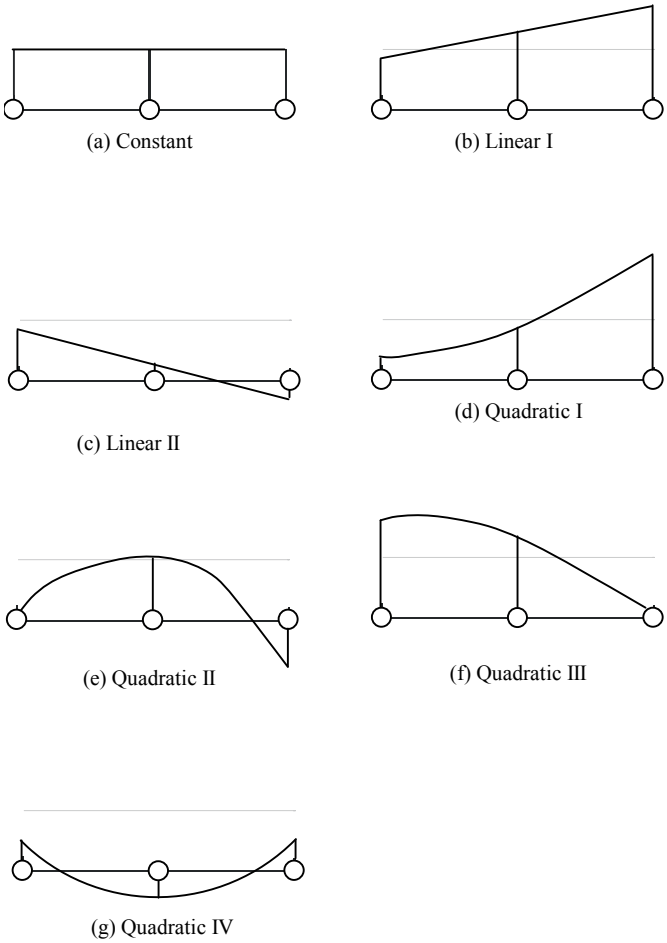


Figure 6-3 Examples of Allowable Variations in Boundary Tractions (Shear and Normal Stress)

6.3.3 Point Loads

Point loads are specified acting at nodes, and are available in both 2D and 3D analyses. In plane strain and axisymmetry, just two components (x and y) are needed to describe the applied force vector. In 3D, three components (x, y and z) must be provided. (Remember the sign conventions - see 2.4.2)

Although finite element programs have to deal with point loads at the level of equation solution, and will convert σ and τ distributions to statically equivalent forces, you are advised against doing this conversion manually, unless you really understand the theory of consistent nodal loads in FEM. (They are counter-intuitive for elements of higher order than constant strain.)

6.3.4 Gravity Changes

See Section 6.9 in the User Guide for how to set the gravity increment

In any given increment block, gravity can be increased/reduced by a factor Δgrav. For example, if you have started off with 1g and wish to increase to 3.5g, the Δgrav would be 2.5. Δgrav will usually be zero in field problems or laboratory tests.

When modelling a centrifuge test, it may be necessary to replicate the increasing material self weight caused by centrifugal acceleration (see Li, 1990 for an analysis of some problems associated with this method).

6.4 Displacement Fixities

6.4.1 General

Most displacement fixities will tend to be specified at in situ stage and then remain operative throughout the analysis. Boundaries along the sides and base of the mesh typically come into this category - they must be specified at the outset, or the mesh may be kinematically indeterminate (i.e. insufficiently restrained).

Displacement fixities are defined for a whole increment block and are divided over each increment according to that block's increment factors (see Section 6.1.5).

Fixities are applied, altered and released via Fixities dialogue box in the Pre-Processor (see Section 6.9.4 in the User Guide).

6.4.2 Imposing Fixities

During an analysis, it may be necessary to fix element sides which were not originally restrained. Examples of this include the remote ends of installed props, and element edges on the vertical centreline boundary of a fill embankment. This can be done at any time.

Note that boundaries can only be smooth if they are aligned with the vertical and horizontal directions. Rough boundaries can be created in any direction.

6.4.3 Releasing Fixities

Once a node is fixed, it remains fixed throughout the analysis until any further changes are imposed on it. Any node which has been fixed previously can be released so that it is completely unrestrained thereafter.

6.4.4 Prescribed Displacements

In addition to fixing an element side against any movement, it is possible to specify a prescribed displacement in the x and/or y direction. In fact, fixity is simply a special case of this - where the prescribed displacement is zero.

Typical applications might be strain-controlled triaxial tests, simulating the penetration of a rough-rigid surface footing, etc. The displacement is specified at each edge node (as with pressures), allowing for non-uniform distributions.

Note that by specifying a positive fixity you are actioning a move in the positive x or y direction, and vis. vis. for negative prescribed displacements.

6.5 Excess Pore Pressure Fixities

6.5.1 General

Section 6.9.4 in the User Guide describes how to apply pore pressure fixities

Similar to displacement fixities, excess pore pressure fixities are in general specified along the mesh boundaries. An exception is when it is necessary to specify the excess pore pressures along an interface which separates a draining layer from a consolidating layer. Excess p

ore-pressure boundary conditions are not specified in a drained or an undrained analysis, only in a consolidation analysis.

The boundary of a finite element mesh in a consolidation analysis can be either:

(a) Impermeable (undrained)

(b) Drained (to atmosphere)

(c) Drained (against a non-zero pressure head)

In (a) you do not need to specify any excess pore pressure boundary conditions. All boundaries are automatically assumed to be impermeable, unless specified otherwise. In cases (b) and (c) care is needed in specifying the pore pressure fixities.

If no further pore pressure changes are to take place along a given boundary at any stage of the analysis, then the incremental excess pore pressure should be fixed at zero. This means that the total pore pressure is held at the value prior to the specification of the excess pore pressure fixity (because change = 0).

If the total pore pressure along a boundary is to remain at a specific value, then this can be specified directly using the Total Pore Water Pressure option. An example of this would be along the top draining surface of an Oedometer test sample, after application of the load. When it is necessary to specify excess pore pressures along a boundary on which the current pore pressure distribution is not known, then fix the absolute excess pore pressure.

If these pore pressure fixities are used in an increment block comprising more than one increment, incremental excess pore pressure fixities are spread over all the increments exactly in the same manner as displacement fixities (see Section 6.4). However absolute excess pore pressure fixities are enforced in the very first increment of the increment block, and no further change in pore pressure takes place along these boundaries in the remaining increments in that block.

6.5.2 Incremental Excess Pore Pressures

An incremental excess pore pressure fixity operates in the same manner as a displacement fixity. The excess pore pressure is given an incremental change specified by the user. Note that this has the effect of changing the total pore pressure by the same amount. If no further pore-pressure changes are prescribed to the nodes in question at a later stage of the analysis, no further changes take place at these nodes. By specifying the excess pore pressure fixity you force these nodes to behave as a sink or a source of water. If the excess pore pressure in the adjacent soil is greater than at the node with prescribed pore pressure it behaves as a drain (or a sink). If the excess pore pressure is smaller than the value specified at the node then it behaves like a source. Therefore the excess pore pressure (and

hence the total pore pressure) is maintained in the rest of the analysis until it is overridden by a subsequent specification by the user.

6.5.3 Absolute Excess Pore Pressures

absolute excess pore pressure fixities should be used when you cannot tell beforehand what the excess pore pressure (or total pore-pressure) is going to be at the node(s) concerned. If u represents the total pore pressure at which a particular node is to be maintained, and if u_o is the in situ (hydrostatic) value of the pore pressure at that node, then:

$$total\ p.p. = in\ situ\ p.p. + excess\ p.p$$

Eqn. 6.5-1

$$u = u_0 + u_e$$

$$\text{i.e.}\quad u_e = u - u_0$$

You then specifie u_e as an absolute excess pore pressure fixity, and not u. As this particular aspect of CRISP has been known to cause confusion in the past, three different examples will be considered here.

6.5.4 Illustrative Examples

6.5.4.1 Example 1

Consider an excavation of a trench in a saturated clay, Figure 6-4. The trench is excavated in layers of 2 metres, and assuming the unit weight of water is 10 kN/m^3 the in situ pore pressures at nodes 1, 2 and 3 are 0, 20 and 40 kPa respectively. We assume that (free) surface water is available to dissipate the suction set up along the excavated face, due to removal of soil.

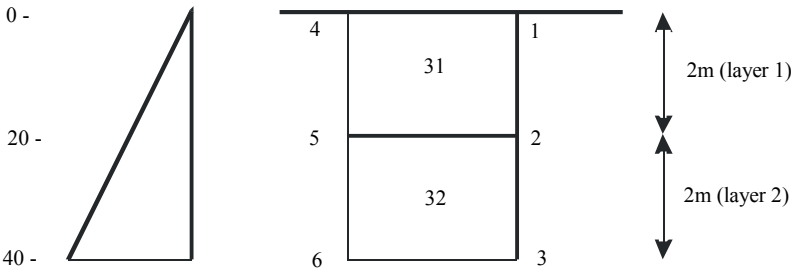


Figure 6-4

After excavating the first layer, the total pore pressure at node 2 = 0. Therefore the excess pore pressure = 0 - 20 = -20 kPa. Similarly after two layers have been excavated the excess pore pressure along the base (at a depth of 4 metres) and at nodes 6 and 3 is given by 0 - 40 = -40 kPa. This particular feature often causes confusion to users. The most common mistake is to (incorrectly) fix the absolute excess pore pressure to **0.**

6.5.4.2 Example 2

Consider an oedometer (1D compression) test analysis. A vertical stress of 100 kPa is applied to the top platen, which has the effect of generating an excess pore pressure of 100 kPa everywhere in the sample. If we wanted drainage to take place from the top and bottom of the sample, we would specify

an incremental change in excess pore pressure of -100 kPa, so that the resulting excess pore pressure on both of these boundaries is 0. We were able to do this only because we knew that the excess pore pressure before implementing the pore pressure fixity was 100 kPa.

Alternatively absolute excess pore pressure fixities could have been used . Note that it is the u_e value which is specified and not the u value,

$$\begin{aligned} u_e &= u - u_0 \\ &= 0 - 0 \\ &= 0 \end{aligned}$$

Therefore a value of $u_e = 0$ is specified as an absolute excess pore pressure fixity. For this particular example both fixity options have the same effect.

6.5.4.3 Example 3

The next example is of a footing resting on a layer of soil, to which a load is applied. This will generate excess pore pressures around the footing, of varying magnitude. If however the ground surface is overlain by a layer of sand which acts as a drain then the excess pore pressure will be zero on the draining surface. Therefore you need to impose a pore pressure fixity condition which gives a zero total pore pressure value along the ground surface. However we do not know how the pore pressure is currently distributed underneath the footing. The total pore pressure fixity is provided exactly to cater for this kind of situation. You don't need to know what the current distribution of the excess pore pressure is along the element side on which you want to impose a known pore pressure. You only need to know the total pore pressure (u) at which you want to maintain the surface in question.

6.5.5 Incremental v Absolute

Earlier the adjective 'absolute (abs)' was used to mean 'cumulative'. The change that takes place in the excess pore pressure in an increment is known as the incremental excess pore pressure (Δu_e). The changes over a number of increments (say n) are summed to give the cumulative value which is referred to as the absolute excess pore pressure:

Eqn. 6.5-2

$$u_e = \sum_{i=1}^{n} \Delta u_e$$

In the case of an incremental excess pore pressure fixity, the node undergoes a change in (excess) pore pressure equal to Δu_e. In the case of an absolute excess pore pressure fixity, the total pore pressure is held at u (corresponding to an excess pore pressure of u_e) in the rest of the analysis. Here again CRISP calculates the necessary incremental value of excess pore pressure required to cause this, taking into consideration the current value of the pore pressure. In the foregoing description no mention was made of the current value of the pore pressure at the node. However the program takes into account the current value of the excess pore pressure, when absolute excess pore pressure fixities are specified.

6.5.6 Releasing Fixities

Any element side that has had excess pore pressures fixed previously (incremental or absolute) can be released (i.e. the pore pressures are free to change, and the element side becomes impermeable).

6.5.7 Relationship To Applied Loads

You should not apply external loads and impose pore pressure boundary conditions in the same increment. It is better to apply loads in one increment, thus generating the excess pore pressures in the mesh. Then, in the next increment, excess pore pressure boundary conditions for any drainage boundaries can be applied.

If at all possible do not combine these two opposing features in the same increment. This is because you are trying to change the pore pressure at a given node (or an element side/face) by two different sources:

(1) Application of a load

(2) Imposition of the excess pore pressure

6.5.8 Time Steps

It is very difficult to give general guidance on appropriate time steps for coupled-consolidation analyses. This really is one area where it will pay to consult the literature (e.g. the CRISP Publications Directory) to see what other analysts have done. Suitable time steps can vary from seconds for laboratory test simulations to days for field constructions. You have to make an initial choice and then examine the excess pore pressures which are generated. If they look random (e.g. u_e fluctuates wildly from node to node within the mesh) then try time steps of 0.1 x and 10 x the original value.

If no improvement is found carry out 2 more analysis with time steps 0.01 x and 100 x. In most cases the problem will be resolved when you change the time step. The pore-pressure boundary condition is imposed in the subsequent increment.

Some additional guidance may be found in Britto and Gunn, 1987.

INTERPRETING OUTPUT 7

7.1 Overview

As with the Geometry Program, output from the Main Program is much more meaningful when displayed graphically. Indeed, the sheer volume of data from a FE analysis is overwhelming and patterns of behaviour can only be discerned when results are post-processed (contour plots, displacement plots, status plots and graphing tools are supplied in the Post-Processor). Nonetheless some useful things can be obtained from text output.

Output from the CRISP Main program consists of the following two components:

- General information produced before any calculations are carried out
- Results for each specified increment of the finite element analysis.

The general information includes analysis identification, storage requirements, control parameters, stress system, in situ stresses, and other summary information. This is only available in the .MPO file.

The output at the end of each increment can be divided into five parts:

1. Nodal displacements and (in consolidation analyses) excess pore pressures
2. General stresses
3. Cumulative strains
4. Parameters for CSSM and elastic-perfectly plastic models.
5. Out-of-balance loads.

This is available in both the Post-Processor (the .PPS file) and (if specifically requested) in the .MPO file. The amount of output in each of these files is controllable - see Section 6.1.7 in this volume and Section 7.3.1 in the User Guide.

See Section 6.14 in the User Guide for details about printed output and the report Generator

Printed output from the Pre and Post-Processors is channelled via the Report Generator which allows you to format the data as you wish. This section describes the Main Program output organised in the order that the information appears in the .MPO file.

7.2 General Analysis Information

7.2.1 Storage Requirements

This is the first item of information printed for the user, and is not normally of interest unless the analysis stops prematurely. Then it might tell you that insufficient storage has been provided in the Main Program, and what must be done to remedy the situation.

The store requirements for the Main Program can be divided into three parts. (i) for the principal arrays (ii) minimum core for solving equations (iii) buffer for storing coefficients of eliminated equations. All this store is allocated in a single one-dimensional array of fixed size, called G. The amount of store for each of these categories is calculated and printed (see Section 8.12 for more information). If the allocation for G is insufficient, the required size is printed and the run is terminated.

After allocating store for category (i) and (ii) the remaining store in G is allocated to (iii), the buffer region. If the size of the buffer is sufficient to store all the coefficients of eliminated equations then the solution is carried out in-core, and the following message is printed:

EQUATIONS ARE SOLVED IN-CORE

otherwise the following message is printed:

EQUATIONS ARE SOLVED OUT-OF-CORE

The values of the various parameters set in the Geometry Program and pertinent to the current analysis are printed next. A list of these is given in Section 8.12.

7.2.2 Control Parameters

The control parameters for the analysis and for the text output are printed. This is followed by the list of increments for which results are written to disk file for subsequent post-processing.

The type of analysis will be confirmed as being either as plane strain, axisymmetric, or three-dimensional.

7.2.3 Material Properties

The input material properties are printed in tabular form, for each material zone defined in the mesh.

7.2.4 Initial Conditions

If any elements have been removed from the parent mesh to form the primary mesh at the beginning of the analysis, then a list of these elements is printed out.

Details of the in situ stress option are printed. If you have specified the in situ stresses at reference elevations or directly at integration points, these are printed. If the in situ stresses are set to zero then a message to that effect is printed.

If the stresses have been specified at in situ reference elevations then, by interpolation, the in situ stresses are calculated at the integration points of all the elements in the primary mesh. The in situ stresses σ'_x σ'_y σ'_x τ_{xy} and u the pore pressure (plus τ_{yz} and τ_{zx} for 3D) at all integration points are printed. For CSSM models p', q, p'_c and e (the void ratio) are also given.

Note: for any finite element run which is a continuation of a previous run (using stop-restart) all the information about the in situ stresses given above would not be printed.

In the presence of in situ stresses, you have to specify an external loading which is in equilibrium with the in situ stresses. External loading includes the pressure loading on element sides along the mesh boundary and any body forces. Instead of pressure loads one could specify the equivalent set of point loads (obligatory for 3D analysis). It is also necessary to specify the displacement boundary conditions (i.e. fixities) along the boundary where the mesh is restrained. This information is also printed.

Finally, an equilibrium check is carried out and the nodal loads due to the in situ stresses and external loading are printed out for all the nodes. For equilibrium the nodal loads must be equal; any difference is the error in equilibrium and this is also printed. This topic is dealt with more fully in Sections 5.3, 7.10 and 7.11.

7.2.5 Stress State Codes

See Section 6.10.3 in the User Guide for details about the Analysis Summary viewer

In order to assist the interpretation of results when using the CSSM family of models, the stress state of each integration point at the end of each increment is assigned a number to indicate its location within the current yield locus. The different codes and the regions they represent are printed out in a table in the Analysis Summary dialogue box. A full description is given later (see Section 7.7.5).

7.3 Increment Information

The information that is presented in this section s repeated for each increment of the analysis. Each increment forms a part of an increment block and output consists of the increment block number followed by the control parameters.

7.3.1 Element Alterations

If there are changes to the current mesh a list of element changes are printed under the heading "LIST OF ELEMENT ALTERATIONS".

Elements which appear in this list are added if they are absent from the current mesh. The elements are removed if they are present in the current mesh. (NB: current mesh is referred to as the group of elements which are active at the present time.)

7.3.2 Gravity Level And Time

The current increment number is printed followed by the incremental gravity level, the total gravity level, the time increment, and total time.

7.3.3 Equation Solution Requirements

Information relevant to the solution of equations is printed. This comprises (a) the maximum frontwidth for the solution (b) minimum store required to solve the equations (c) additional store required for in-core solution. As the mesh can change from one block to the next these details are re-calculated for each block.

7.3.4 Increment Factors

If the option to specify a separate list of increment ratios, output options and time steps has been selected, these are printed.

7.3.5 Applied Loading

External loading either in the form of nodal point loads or as pressure loading along element sides, are printed. For the case of pressure loading along an element side a second line gives the equivalent nodal point loads in x and y directions respectively. However the sequence of nodes may not be the same as specified in the input data for pressure loading. The nodes are always arranged to follow the anticlockwise order about the element centre.

7.3.6 Boundary Conditions

The prescribed boundary conditions (displacements and excess pore pressures) are printed. A combined list of applied loads and d.o.f. with prescribed displacements (or excess pore pressures) is printed out under the heading.

"PRESCRIBED BOUNDARY CONDITIONS"

(a) Two-dimensional analysis (maximum of 4 D.O.F. per node)

Table 7-1 Degree of Freedom Options (2D Analysis)

Degree of Freedom	Description
1	Displacement/load along x (or r) axis
2	Displacement/load along y (or z) axis
3	Excess pore pressure u_e.
5	Rotation for beam element θ.

(b) Three-dimensional analysis (maximum of 4 D.O.F. per node)

Table 7-2 Degree of Freedom Options (3D Analysis)

Degree of Freedom	Description
1	Displacement/load along x (or r) axis
2	Displacement/load along y (or z) axis
3	Displacement/load along z (or q) axis
4	Excess pore pressure ue.

The sequence in which each d.o.f. is output follows the order of elimination in the frontal solution.

7.4 Nodal Displacements

7.4.1 Incremental Displacements

Incremental displacements are the primary unknown determined by the finite element (displacement) method. They are fairly unambiguous and do not require much in the way of explanation here.

A list of incremental displacements at node points are printed:

DXI incremental displacement along x (or r) axis

DYI incremental displacement along y (or z) axis

DZI incremental displacement along z (or θ) axis (3D analysis only)

DUI incremental excess pore pressures (coupled consolidation analyses only)

7.4.2 Cumulative Displacements

In addition to incremental displacements, CRISP also calculates and prints out cumulative displacements:

Eqn. 7.4-1 $$\delta = \sum \delta_i$$

A list of cumulative (absolute) displacements at nodes are printed:

DXA absolute displacement along x (or r) axis

DYA absolute displacement along y (or z) axis

DZA absolute displacement along z (or q) axis (3D analysis only)

DUA absolute excess pore pressures (coupled consolidation analysis only)

NB: In some situations, the true displacement pattern may be obscured if you only plot cumulative displacements (for example in a curved flow field accompanying bearing capacity failure).

7.5 General Stresses

7.5.1 Direct Stresses

Current direct (Cartesian) effective stresses are calculated at integration points and are printed out, together with point coordinates. The stresses printed are:

SIG-X	□'x	horizontal effective stress
SIG-Y	σ'_y	vertical effective stress

SIG-Z	σ'_z	out-of-plane effective stress
TAU-XY	τ_{xy}	shear stress in xy plane

In addition, for 3D analyses:

TAU-YZ	□yz	shear stress in yz plane
TAU-ZX	τ_{zx}	shear stress in zx plane

Note, these are cumulative values, in that they are the original in situ stress values plus the changes which have occurred in all increments up to (and including) the current one:

Eqn. 7.5-1 $$\sigma_i = \sigma_0 + \sum \Delta\sigma_i$$

7.5.2 Pore Pressures

A list of current pore pressures at the integration points (along with the point coordinates) is calculated and printed:

Eqn. 7.5-2 $$u_i = u_0 \sum \Delta u_i$$

7.5.3 Principal Stresses

Principal effective stresses (in the xy plane) are calculated at the integration points and printed out:

SIG-1	σ'_1	major principal effective stress
SIG-3	σ'_3	minor principal effective stress

7.5.4 Principal Stress Directions

The angle between the x axis and the major principal stress direction (measured in the π plane) is computed and printed out (see Figure 7-1 for clarification):

TH-XY θ_{xy} principal stress direction

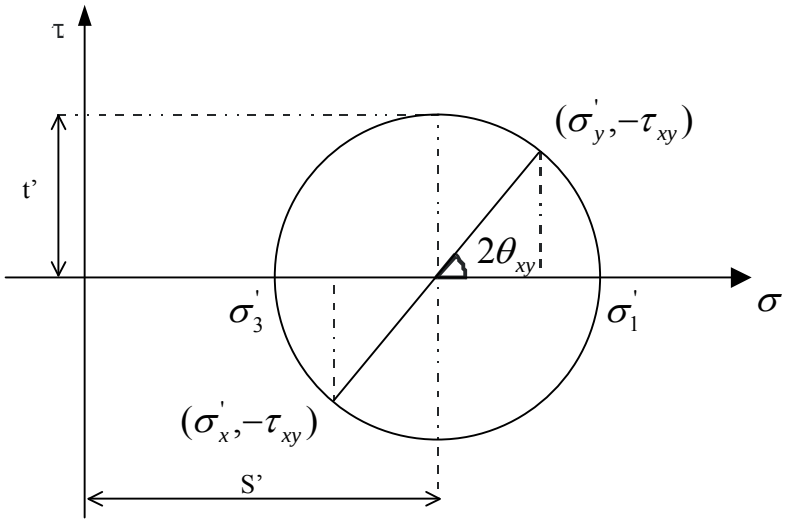


Figure 7-1 *Major Principal Stress Directions*

7.6 Cumulative Strains

7.6.1 Direct Strains

The current (cumulative) strains are calculated at integration points and are printed out along with the coordinates of the integration points. The strains are:

EPS-X	ε_x	normal strain in x direction
EPS-Y	ε_y	normal strain in y direction
EPS-Z	ε_z	normal strain in z direction
GAM-XY	γ_{xy}	engineer's shear strain in xy plane

In addition, for 3D analyses:

GAM-YZ	γ_{yz}	engineer's shear strain in yz plane
GAM-ZX	γ_{zx}	engineer's shear strain in zx plane

7.6.2 Strain Invariants

Some strain invariants are also calculated at integration points and are printed out:

GAM-MAX	γ_{max}	maximum engineer's shear strain
EPS	ε	deviatoric strain

7.6.3 Other Strain Parameters

Finally, a number of miscellaneous terms are calculated at integration points and are printed:

FI-MOB	ϕ_{mob}	mobilised angle of friction (actually calculated from stresses and not strains)
PSI	ψ	angle of dilation

7.7 CSSM Parameters

These parameters are calculated from the cumulative stresses described in Section 7.4, for CSSM models only. The parameters can be grouped under the headings of :

- Stress invariants
- Stress ratios
- Yield ratio
- Void ratio
- Stress state codes

7.7.1 Stress Invariants

The following invariants are calculated at each integration point in those elements assigned CSSM properties:

PE	p'	Mean normal effective stress
Q	q	Deviatoric stress
PT	p	Mean normal total stress
PC	p'_c	Previous maximum consolidation pressure
TH-3	θ	Third stress invariant

All of these parameters have their conventional CSSM meanings and definitions, with the exception of θ. This is a measure of the orientation of the effective stress state in principal stress space, and is analogous to the Lode angle. The angle is measured from the x axis in a counter-clockwise direction (see Figure 7-2). The x and y axes lie on the π plane (for the interpretation of this angle) and the z axis is the space diagonal.

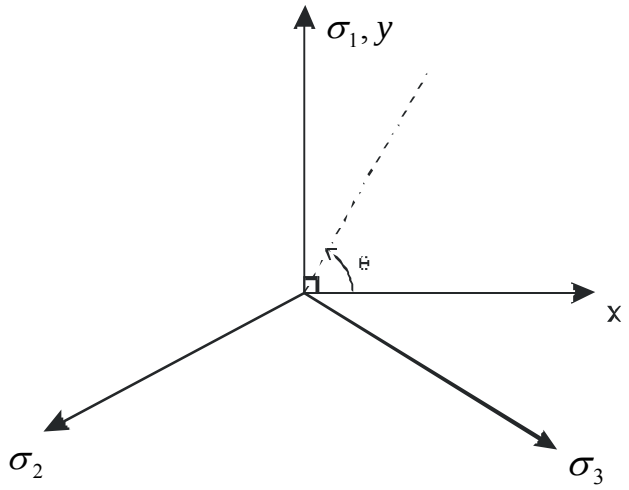


Figure 7-2 *Definition of Angle θ*

7.7.2 Stress Ratios

From the stress invariants, several stress ratios are calculated:

ETA	η	stress ratio (= q/p')
ETA/M	η/M	normalised stress ratio

These parameters have their conventional meanings, with η/M being a useful measure of how close to the critical state ratio the current stress state is.

7.7.3 Yield Ratio

Based on the current stress state and size of yield locus, CRISP calculates a parameter which estimates how much the yield locus has changed in size in the current increment.

The yield ratio provides a means of assessing whether or not the current increment size (load step) is small enough. For a point with stress state P the yield ratio is given by:

Eqn. 7.7-1

$$YR = \frac{p'_y}{p'_{co}}$$

Where

p'_y	The value of p' which defines a curve geometrically similar to the current yield locus at the end of this increment
p'_{co}	The value of p' defining the yield locus at the beginning of this increment

These terms are defined in Figure 7-3. The yield ratio for points with current stress states A2, B2 and C2 with corresponding stress states A1, B1 and C1 at the beginning of the increment are:

$$YR_A = \frac{p'_{yA}}{p'_{co}}$$

$$YR_B = YR_C = \frac{p'_{yA}}{p'_{co}}$$

Note that all three points have the same yield locus at the beginning of the increment. However whereas B1 is plastic, points A1 and C1 are elastic.

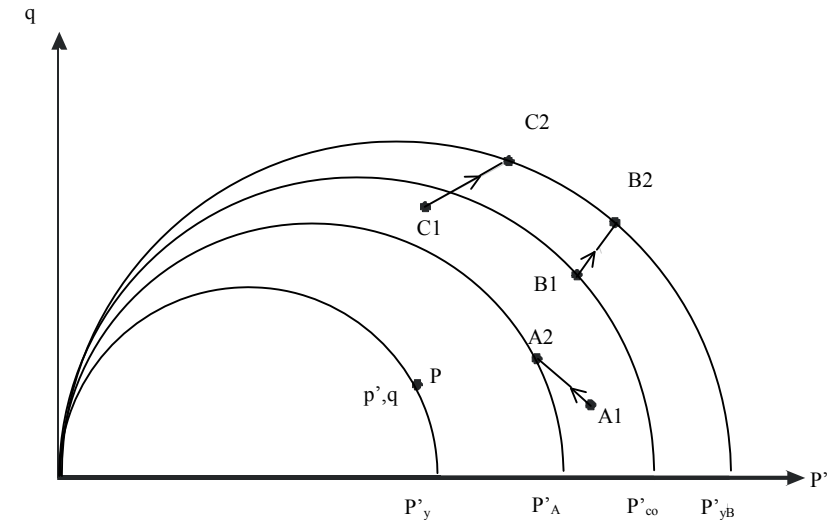


Figure 7-3 *Yield Ratio Definitions*

YR is calculated at each integration point for every element which has been assigned CSSM model properties. However, just looking at element centroids or a range of elements never fully reveals what is happening. Hence a summary table is automatically printed (you cannot suppress it) for each increment. It counts the number of integration points under different categories and also different ranges of YR values. This conveys a very good picture of what has happened in the increment, at a glance.

It should be stressed that this summary applies to a single increment only and does not categorise the behaviour in a cumulative manner. For example, it is not sufficient just to look at the summary for the final increment (for example in a 100 increment analysis) and find that YR for most integration points which are hardening is in the range 1 to 1.05. This simply indicates that the loading applied in the last increment was sufficiently small and it makes no comment about the previous 99 increments.

When plastic hardening is taking place, the value of YR gives the ratio of the size of yield locus following the increment to the size before the increment. Thus a value of 1.10 means that the yield locus has grown in size by 10%. Values of about 1.02 (0.98 if softening) are generally regarded as leading to sufficiently accurate calculations. If values greater than 1.05 (less than 0.95 if softening) are seen then the size of the load increments should be reduced. When one of the CSSM models is softening (i.e. yielding dry of critical) smaller increments (than the size suggested by the above discussion) may be necessary. Ideally most of the integration points which are yielding/hardening should have a yield ratio value close to 1.0 (less than 1.01).

The use (and limitations) of YR is illustrated with an example problem in the Examples Manual which is a strain-controlled undrained triaxial test with 3% axial strain applied in 6 equal increments. The results obtained are compared with an analysis of about 50 increments and also with the theoretical solution. As can be seen, the analysis with 6 increments overestimates the deviatoric stress q at yield. This example is used to illustrate that errors can occur if sufficient number of increments are not used. Because of the uniform nature of the triaxial test, the stress state at any given integration point is representative of the behaviour of the entire mesh. However in an analysis of a field problem the behaviour at a given integration point will be different from the behaviour from any other integration point.

From Table 7-3 it can be seen that in increment 1 YR is less than 1.0 (0.844). A value of less than 1.0 means that either the stress state is elastic or it is undergoing softening with the size of the yield locus reducing. In the case of elastic behaviour, YR indicates how close it is to the yield locus; a stress state close to the yield locus will therefore give a YR value of nearly 1.

In increment 2 the transition from elastic to elasto-plastic state occurs. The value of 1.125 indicates that the current yield locus is 12.5% larger than the previous one. The value is much higher than the recommended upper limit of 1.05 and way above the value of 1.01 which would be needed for accurate prediction.

Table 7-3 *Yield Ratios for Example Problem (Examples Manual)*

Increment No.	Yield Ratio
1	0.844
2	1.125
3	1.023
4	1.012
5	1.007
6	1.005

Increments 3 to 6 have YR values 1.023, 1.012, 1.007 and 1.005 which are more acceptable. The values for increment 3 and 4, however, are not sufficiently close to 1.0 to give an accurate predication. Therefore, especially in increments 2 to 4 the strain increment of 0.5% is much too high for an accurate predication of the theoretical response.

7.7.4 Void Ratio

E-STRS	e_σ	Void ratio calculated from stress state
E-STRN	e_ε	Void ratio calculated from stress state at the beginning of the current increment and the incremental strains in the current increment

7.7.5 Stress State Codes

CODE - A number to indicate the location of the current stress state within the yield locus (see Table 7-4 for a full list of codes and their definitions)

Table 7-4 *Stress state codes for critical state models*

Code	p'	q	Comments
0	$> p'_{cs}$	< M p'	elastic
1	$< p'_{cs}$	< M p'	elastic
2	$< p'_{cs}$	> M p'	elastic
3	$> p'_{cs}$	< M p'	hardening
4	$< p'_{cs}$	> M p'	softening
5	$< p'_{cs}$	> M p'	softening
6	$< p'_{cs}$	> M p'	yielding
7	$> p'_{cs}$	> M p'	hardening
8	$< p'_{cs}$	> M p'	hardening
9	negati ve		

Codes 5 and 6 are only applicable to model 6

Codes 7 and 8 are inadmissible and arise from numerical problems

figs following illustrate the position of the stress state in q : p' space represented by each of these status codes:

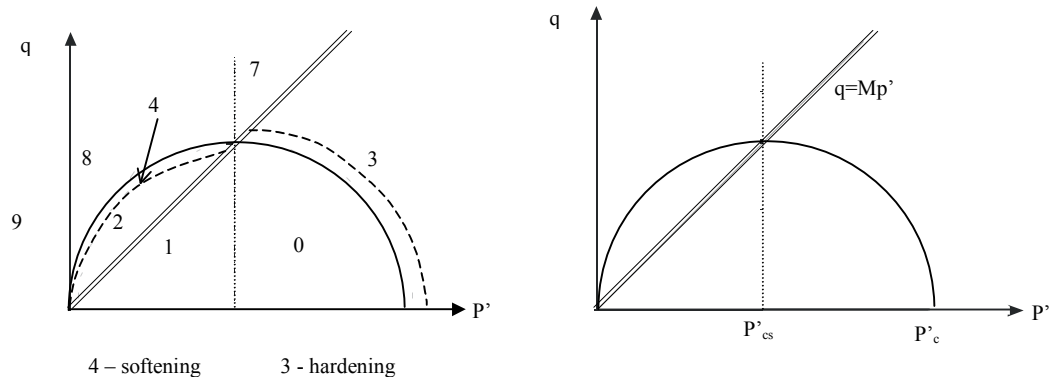


Figure 7-4(a) *Stress State Codes forCam clay and Modified Cam clay Soil Models*

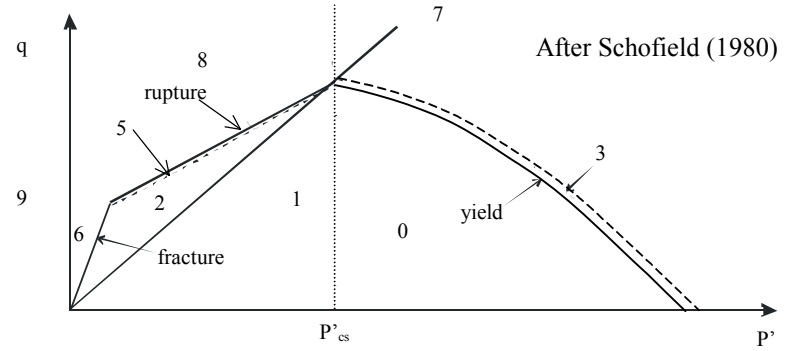


Figure 7-4(b) *Stress State Codes for the Schofield Soil Model*

7.7.6 Warning Messages

Wherever applicable, warning messages are printed for CSSM models (see also Appendix C).

7.7.6.1 Approaching the critical state

The form of the warning message is (for example):

WARNING **** ELEMENT 8 HAS INTEGRATION POINTS

1 2 0 0 0 6 0 APPROACHING CRITICAL STATE

This indicates that, out of seven integration points, the 1st, 2nd and 6th integration points are approaching the critical state (the actual numbers are the stress state codes).

7.7.6.2 Negative value for p'

The form of the warning message is (for example):

WARNING **** ELEMENT 11 INTEGRATION POINTS

0 9 0 0 9 0 9 HAVE NEGATIVE VALUES FOR p'

This indicates that, out of seven integration points, the 1st, 2nd and 6th integration points are approaching the critical state.

7.7.7 Analysis Summary

If there are CSSM models present in the analysis a summary of the CSSM behaviour is presented at this point. This is compiled from the various stress codes and gives an overview of all the regions modelled using CSSM models without distinguishing between the different models. It gives a count of the total number of integration points which have the different stress codes. A further subdivision in terms of ranges of yield ratios (YR) enables you to evaluate the size of the applied increment. Too many integration points with YR > 1.05 when hardening (stress code 3) would mean the applied load increment is too big for that particular increment.

7.8 Elastic-Perfectly Plastic Model Parameters

These parameters are calculated from the cumulative stresses described in Section 7.4, for elastic-perfectly plastic models only. Parameters can be grouped under the headings of

- Stress invariants
- Stress state codes

7.8.1 Stress Invariants

PE	p'	Effective mean normal stress
SBAR	$\sqrt{J_2}$	Second stress invariant
S*	s'	Average normal stress in xy plane
T*	t	Maximum shear stress in xy plane
TH-XY*	θ_{xy}	Angle between σ'_1 and axis with the largest normal stress
		(* only relevant for plane strain analysis)

The second stress invariant $\sqrt{J_2}$ is related to other more widely used invariant parameters thus:

Eqn. 7.8-1

$$\text{octahedral shear stress } t_{oct} = \left(\frac{2J_2}{3}\right)$$

$$\text{deviatoric stress } q = (3J_2)$$

The parameters s' and t are the plane strain invariants, corresponding to the axisymmetric pair p' and q:

Eqn. 7.8-2

$$s' = \frac{1}{2}(\sigma'_x + \sigma'_y) = \frac{1}{2}(\sigma'_1 + \sigma'_3)$$

$$t = \frac{1}{2}(\sigma'_x - \sigma'_y) = \frac{1}{2}(\sigma'_1 - \sigma'_3)$$

7.8.2 Stress State Codes

CODE - A number to indicate the location of the current stress state stress state (different from those for CSSM models)

Table 7-5 *Stress state codes for elastic-perfectly plastic models*

Code	Comments
0	Elastic
1	First time yielding (i.e. changeover from elastic to plastic in this increment)
2	Continued yielding (remains plastic in this increment)

Note: in drawing the Mohr's circle for the stress state in the xy plane no account is taken of the out-of-plane stress σ'_z. It is assumed that σ'_z is the intermediate stress which may not necessarily be true.

7.8.3 Warning Messages

There are 3 different messages which may be printed when using the elastic-perfectly plastic models:

7.8.3.1 DLAMB IS NEGATIVE

The form of the warning message is (for example):

WARNING *****AT ELEMENT 15 INTEGRATION POINT 3 DLAMB IS NEGATIVE

followed by the list of DLAMB ($d\lambda$) values (one for each of the sub-increments). The plastic multiplier $d\lambda$ is given by:

Eqn. 7.8-3

$$d\underline{\varepsilon}_p = d\lambda\left(\frac{dQ}{d\underline{\sigma}}\right) = d\lambda.\underline{a}$$

where $d\underline{\varepsilon}_p$ = plastic strain increment vector

$d\lambda$ = plastic multiplier (a scalar)

$\underline{a}$ = the flow vector

Q = the plastic potential

Theoretically, $d\lambda$ cannot take negative values. However, due to numerical problems (especially when too large an increment has been applied close to collapse) $d\lambda$ may take negative values. This can also happen when the stress state lies outside the yield surface. If this is the case, then the above message is printed and $d\lambda$ is set to zero.

This message at a few integration points should not be cause for concern. However, if such messages appear for whole regions or for many elements, with or without the following message:

*** TOO MANY INTEGRATION POINTS WITH DLAMDA NEGATIVE INCREASE SIZE OF ARRAY VLDL AND ALSO RESET NVL IN ROUTINE UPARAL (ROUTINE YIELD)

then the analysis may be invalid. The above message is printed if the stress state which has yielded and ventured outside the yield surface has been 'supposedly' corrected back to the yield surface, but still lies outside the yield surface. This can happen when you start an analysis with zero stresses using the Mohr-Coulomb model with C = 0. This means the whole region is on the tip of the yield surface and has presumably yielded. All these elements will be assigned plastic stiffness instead of elastic. To avoid this specify an arbitrarily small value for C (say 1 kPa) to ensure that initial zero stress state lies within the yield locus rather than on it.

The same problem can occur when non-zero initial stresses are specified which lie outside the yield locus for any of the elastic-perfectly plastic models. Ensure that the initial stress state does not violate the yield criterion. Otherwise this will lead to errors.

The occurrence of the message $d\lambda$ negative is rare in a well-behaved analysis. Array VLDL is set to a size of 200, which is more than adequate for any situation. If this limit is reached then probably something is wrong with the analysis.

7.8.3.2 UNLOADING TO ELASTIC STATE

The form of the warning message is (for example):

WARNING ***** ELEMENT 6 HAS INTEGRATION POINTS

0 0 3 0 5 6 0 UNLOADING TO ELASTIC STATE 0 0 0 11 0 0 0 0 0

Out of the 16 integration points (this element being a cubic strain triangle), integration points 3, 5, 6 and 11 appear to be unloading to elastic state.

7.8.3.3 ICOR IS SET EQUAL TO ZERO

The form of the warning message is (for example):

WARNING **** ICOR IS SET EQUAL TO ZERO IN THE PRESENCE

OF ELASTO-PLASTIC MODEL

For elastic-perfectly plastic models the stress state after yielding are normally corrected back to the yield surface, leading to out-of-balance loads which are significant (can be up to 5 - 10 % of the maximum applied load). This message is printed when you have specified that the out-of-balance loads should **not** be carried over and applied as additional loads in the subsequent increment. It is rather unusual that you should be doing this, and there are very few circumstances in which it could be justified. If you see this message, you have probably made a mistake in the input by not requesting stress corrections to be carried out.

7.9 Nodal Reactions

Reactions are printed for those nodes which have prescribed displacements (both zero at fixed boundaries and non-zero elsewhere), in the directions which have been prescribed. That is, reactions only exist for d.o.f. which have been prescribed.

This information is useful when you want to know the loads being implied through displacement controlled loading. For example, if a rough-rigid plate is simulated by giving a line of nodes on the soil surface a prescribed downward displacement, the reactions at these nodes must be summed to give the actual load on the 'plate' (be very careful how you sum these reactions in axisymmetry).

7.10 Out-Of-Balance Loads

For all nodes specified (see 6.1.7), this part of the output consists of:

- The incremental applied load
- The out-of-balance load
- The nodal loads equivalent to the element stresses and
- The total applied loads at the nodes.

In the rest of this section, we discuss the interpretation of out-of-balance loads with reference to the different type of constitutive models which might be used.

7.10.1 Linear Elastic And CSSM Models

CRISP uses the tangent stiffness (incremental) approach. It is assumed that you sub-divide the whole analysis into sufficiently small load-increments, so that the difference between the piecewise linear assumption and the true response is negligible. Since for wholly elastic and CSSM models there is no correction made to the stresses at the end of the increment, there should be no out-of-balance (or residual) loads. Large out-of-balance loads indicate that there is something wrong with the analysis. For these constitutive models it would be wrong to re-apply these loads to the increments which follow, as the errors would simply be propagated through the remaining increments.

7.10.2 Elastic-Perfectly Plastic Models

With the elastic-perfectly plastic models, when yielding occurs and the stress state moves outside of the yield surface (point 1 in Figure 7-5), the stresses are corrected back to the yield surface (point 2). This correction is denoted by $\Delta\sigma_{C1}$ for a single integration point. When converted into a set of equivalent nodal loads denoted by ΔPc_1 this represents the out-of-balance load. Out of the applied load ΔP_1 only (ΔP_1 - ΔP_{C1}) is satisfied by the internal stresses.

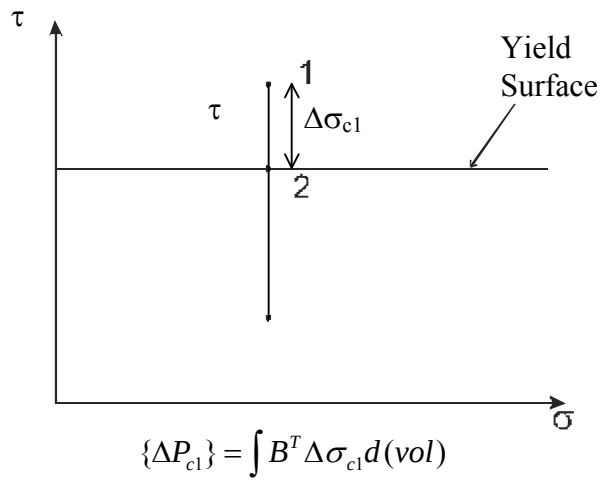


Figure 7-5 *Stress State Correction for Elastic-Perfectly Plastic Soil Models*

In a finite element program that uses an iterative technique, the load ΔPc_1 is re-applied and the resulting incremental displacements are added to the current displacements. If during the application of ΔPc_1 further yielding takes at some integration points then a second set of out-of-balance loads, ΔPc_2 are calculated and the above procedure is repeated until the resulting incremental displacements or ΔPc_n is less than a preset tolerance. This is represented in Figure 7-6.

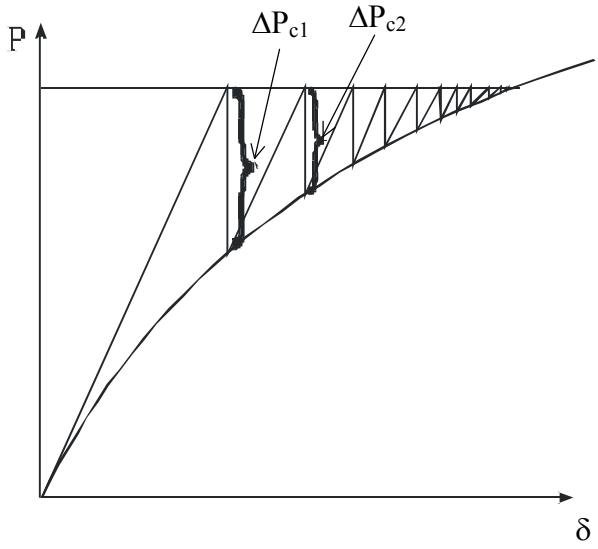


Figure 7-6 *Iterative Solution Techniques*

In CRISP, instead of calculating the incremental displacements due to the out-of-balance loads separately, the loads are carried over to the next increment and added to whatever loading is applied in that increment. Therefore the applied load increment in the next increment is $\Delta P_2 + \Delta Pc_1$, Figure 7-7.

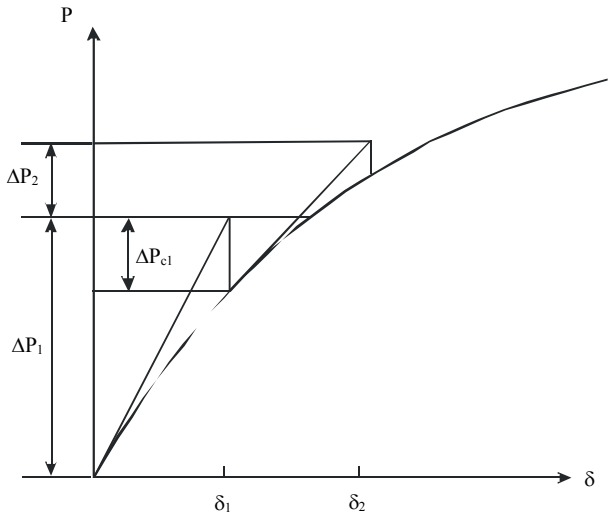


Figure 7-7 *Carry Over of Out-Of-Balance Loads*

At the end of each increment in an analysis which uses elastic-perfectly plastic models, the increase in strain is divided into ten equal steps. The stress state is re-evaluated at the end of each step, illustrated in Figure 7-8. The stress state calculated is closer to the true solution than in a CSSM analysis, so fewer increments are required.

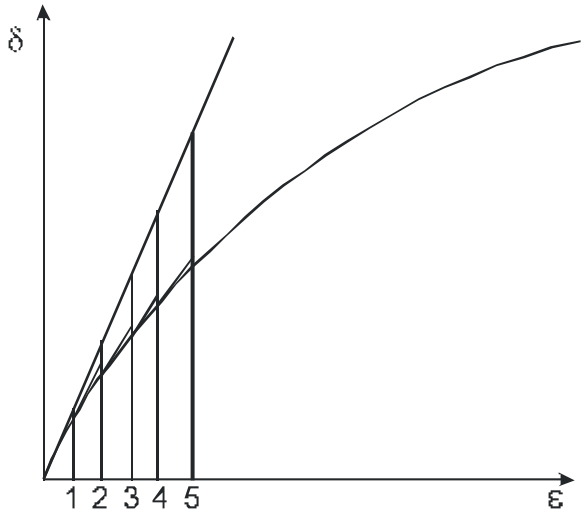


Figure 7-8 *Use of Sub-Increments for Elastic-Perfectly Plastic Soil Models*

If, at the end of an analysis, there are out-of-balance errors greater than about 1% you may have to add a few more increments than you had originally intended after the last increment (but without any further loading). This will cause the residual out-of-balance loads to be re-distributed ensuring that equilibrium is satisfied. The displacements at the end of the analysis will then be consistent with the applied loading. The stop-restart facility may be useful in this respect (see Section 8.3).

7.11 Equilibrium Errors

The output for each increment is conclude by printing out the equilibrium check which gives the percentage error in equilibrium (see Section 5.3 for definitions). Equilibrium errors in any analysis should be less than 5%, and preferably less than 1%. This section comprises a discussion of the causes and the significance of equilibrium errors and what you should do if they arise.

7.11.1 Errors Due To Plastic Yielding

Out-of-balance loads can be present during the analysis if elastic-perfectly plastic models are used and yielding is occurring. As explained above (see 7.10.2), these are legitimate out-of-balance loads, and should not give rise to any concern. The point that should be remembered is that the % out-of-balance should be kept to within reasonable limits (less than about 15-20%), when elastic-perfectly plastic models are used. This only applies to the actual analysis increments onwards and not to the in situ stage - there should be no out-of-balance load at the beginning under any circumstances.

Significant out-of-balance loads (resulting in equilibrium errors > 1%) during the course of an analysis (i.e. after the in situ stage) when elastic and/or CSSM models are used would indicate ‘numerical problems’ and no other (assuming that there were no equilibrium errors at in situ stage).

7.11.2 Errors Due To Ill-Conditioning

The other form of equilibrium error arises from numerical problems (ill-conditioning). Consider the following matrix equation:

Eqn. 7.11-1 $$P = \mathrm{K}u$$

where

P = column vector of loads

K = stiffness matrix

u = unknown displacements

After solving for the unknown nodal displacements u is multiplied by K and compared with P. The error or residual is calculated thus:

Eqn. 7.11-2 $$Perr = P - \mathrm{K}u$$

This difference can be expressed as a % of the largest applied load:

Eqn. 7.11-3 $$\%err = \frac{\left[P - \mathrm{K}u\right]}{P\max}$$

When the above matrix K is well conditioned then P - K u is negligible, as is the % error. However if the matrix K is ill-conditioned then the % error can be significant. This may happen in CRISP when very stiff structures are buried in soft soils (e.g. steel sheet piles in alluvium). This is unacceptable. The whole point of the equilibrium check (which is carried out at the end of every increment) is to prevent this type of error from creeping into a CRISP analysis.

Numerical problems in the presence of elastic-perfectly plastic models may be hard to detect. The out-of-balance loads due to ill-conditioning can be masked by the out-of-balance loads due to yielding and stress corrections. If you suspect this is taking place, then replace the elastic-perfectly plastic soil with a linear elastic model and re-run the analysis. If significant out-of-balance loads are still present, then you have a numerical problem.

If there are high stiffness contrasts in a soil-structure interaction analysis, this can lead to problems with computer precision (i.e. the fact that a finite number of digits are used to represent numbers). This can be investigated by reducing the stiffness of the structure and re-running the analysis.

It should be pointed out that not all soil-structure interaction problems automatically run into numerical difficulties with precision. However, you should be on the lookout against such an event.

7.11.3 Summary

In summary, there are two sources of out-of-balance loads:

(a) In an analysis which uses elastic-perfectly plastic models, yielding and the subsequent stress correction gives rise to out-of-balance loads. This is permitted and quite normal. At the beginning of an analysis where the initial state is elastic, and during the analysis until yielding takes place there should not be any out-of-balance loads. During elastic behaviour no stress correction is carried out, because there is no need.

(b) When very stiff structures are present in a soft soil, out-of-balance loads may result from numerical problems. Out-of-balance errors can also occur in a consolidation analysis where an inappropriate time step has been used. Any out-of -balance loads from these sources are unacceptable.

The equilibrium condition being met at in situ stage and throughout the analysis is a pre-requisite for a CRISP analysis to yield an acceptable solution. However, this simply means that numerical problems are absent. This condition alone does not guarantee that the result produced are acceptable. It is necessary, but not sufficient.

It is possible to carry out an analysis with a soil model which is inappropriate to the problem being analysed, insufficient number of increments, and an inadequate mesh - and still end up with an analysis which satisfies the equilibrium check all the way through. Therefore the compliance of the equilibrium check does NOT justify:

1. The appropriateness of the constitutive model.
2. The number of elements used and their manner of deployment.
3. The number of increments used.

7.12 Final Comments

CRISP (like any finite element code) produces a large amount of output, and engineering judgement has to be exercised in interpreting the results. Some authors recommend that you should be critical of finite element results until you have sufficient grounds for accepting them - this is quite the reverse of the normal attitude (i.e. believe computer output until proven wrong). Questions you should ask include:

1. Does the deformation pattern look reasonable?
2. Do the stress paths behave in an acceptable manner?
3. Are principal stress/strain directions consistent with applied loads?
4. Is the distribution of pore pressures sensible?
5. Is the distribution of the strains credible?

Plot the zones of yielding if you are using elastic-perfectly plastic models or the CSSM models. Zones of yielding should be concentrated around areas subjected to loading and/or unloading. If you find that plastic zones are developing far away from loaded areas, you should be suspicious of the results.

SPECIAL TOPICS

8

8.1 Large Displacement Analysis

The normal assumption made in linear elastic finite element programs and also in most finite element programs with non-linear material behaviour is that displacements are small. External loads and internal stresses are assumed to be in equilibrium in relation to the original (i.e. undeformed) geometry of the finite element mesh.

It is possible in CRISP to request that nodal co-ordinates to be updated after each increment of the analysis, by adding to the co-ordinates the displacements undergone by the nodes during the increment. The stiffness matrix of the continuum is then calculated with respect to these new co-ordinates during the next analysis increment. The intention of this process is that, at the end of the analysis, equilibrium will be satisfied in the final (deformed) configuration. Although this approach would seem to be intuitively more appropriate when there are significant deformations it should be noted that it does not constitute a rigorous treatment of the large strain/displacement behaviour for which new definitions of strains and stresses are required (e.g. Carter et al, 1977).

A new algorithm was intruduced to account for large deformation. This is based on the Updated Lagrangian method (*see Bathe K-J, Finite Element Procedures, 1996, Prentice Hall*). This algorithm accounts for changes in incremental stresses due to rotation as well as due to straining. It must be realised, however, that materials which exhibit severe deformation, such as consolidating slurry, would require a special constitutive material model which would simulate changes in stiffness due to large strain.

Various research workers have examined the influence of a large strain formulation on the load deformation response calculated by the finite element method using elastic-perfectly plastic models of soil behaviour. The general conclusion seems to be that the influence of large strain effects is not very significant for the range of material parameters associated with most soils. In most situations the inclusion of large strain effects leads to a stiffer load-deformation response near failure and some enhancement of the load carrying capacity of the soil. If a program user is mainly interested in the estimation of a collapse load using an elastic-perfectly plastic soil model then it is probably best to use the small displacement approach. Collapse loads can then be compared (and should correspond) with those obtained from a classical theory of plasticity approach.

8.2 Stress Corrections

Stress corrections have already been discussed in the context of yielding and equilibrium errors (Sections 6.1.3 and 7.11). It is recommended that you request stress corrections when elastic-perfectly plastic models are used, but not for other models. When elastic-perfectly plastic models are used, stresses at points which have yielded are corrected back to the yield surface, by adding any ‘unbalanced’ loads which arise from this procedure the next increment of loading. Corrected stresses

will not necessarily be in equilibrium with the applied loading. A detailed description of this technique may be found in Nayak and Zienkiewicz (1972) and Owen and Hinton (1980). Note that the way that the stresses are scaled back to the yield surface is arbitrary, but ensures that the yield criterion is never violated by an excessive amount. Physically, the stresses are corrected along a direction which is normal to the yield surface (shortest distance).

The New Mohr Coulomb models (both elastic-perfectly plastic and elastic-plastic hardening) incorporate a special stress return algorithm based on research by Pankaj and Bicanic (see Pankaj and Bicanic N. "On multivector stress returns in Mohr-Coulomb plasticity" Computational Plasticity, Part 1, Pienrdige Press, 1989). This stress return method uses is a backward Euler stress correction (See Ortiz M and Simo J.C " An analysis of a new class of integration for elasto-plastic constitutive relations. Int J. of Num. Meth. in Eng. Vol 23, 1986). This new algorithm is capable of handling stresses near the apex of the Mohr Coulomb surface.

The stress corrections have no effect on CSSM models. There is no correction to the stresses made at the end of the increment. When the stress state leaves one yield locus (either due to softening or hardening) it is always possible to find a new yield locus through the new stress state. Therefore in theory the stresses are in equilibrium at the end of the increment without having to correct them.

8.3 Iterative Solution Scheme

CRISP incorporates an iterative solution scheme based on the Modified Newton Raphson method. This method is applicable to all non-linear models except those based on Cam-Clay. This is because Cam-Clay models in Crisp do not carry out a stress correction when reaching the yield surface and would rely on suitable size of increments.

Use the iterative solution for the elasto-plastic models by selecting the appropriate parameters in Project Setup of CRISP pre-processor. A displacement convergence criterion is used to check for convergence. This criterion states that convergence is achieved when the square root of the sum of displacements in the current iteration over the square root of the sum of total displacements up to the current iteration is equal or less than a user specified tolerance, or:

$$\frac{\left\| \{\Delta u_i\} \right\|_2}{\left\| \{^{t+\Delta t}u\} \right\|_2} \leq Tolerance$$

A limit for the maximum number of iterations may also be specified by the user. If this limit is exceeded, the program will stop.

8.4 Stop-Restart Analysis

The STOP/RESTART facility enables you either to

- Break up a large analysis into a series of medium sized runs.
- Go back a few increments and then continue the analysis (with, for instance, a different load increment size)

This facility is particularly useful for reviewing the progress of an analysis, and making a change at some intermediate stage, without having to repeat the entire analysis.

The first run in a series of runs (i.e. a complete analysis) is called a Starting run. Any subsequent run in that analysis is a Restarting run.

Stop/Restart analyses are set-up in the Stop/Restart dialogue box (see Section 7.3.4 in the User Guide). This allows you to select the increments from which you think you may wish to run a Restart analysis. Note that all such increments need to be specified in the Starting run.

Tip If you set up a Parent/Child relationship (see Section 7.3.3 in the User Guide) between the Starting run and the Restart runs you can directly compare results between the various analyses in the Post-Processor.

Note that increment numbers are consecutive across Stop/Restart analyses. This means that each the first increment number of each Restart analysis is one greater than the increment number at the end of the Start analysis.

8.4.1 Soil-Structure Interaction Analysis

The Single Precision version of CRISP may be unsuitable for soil-structure interaction problems if the stiffness of the structure is several orders of magnitude higher than that of the soil. For example a very stiff (concrete) pile in soft clay or the metal end platen in a triaxial test. If sensible answers are not obtained (even after reducing the stiffness of the structure) the calculation needs to be carried out in double precision. Numerical problems may be evidenced by large equilibrium errors or by wildly varying pore-pressures in undrained/consolidation analyses.

It is possible to carry out a soil-structure interaction analysis using the single precision version of the Main Program. For example, it may not be necessary to specify the real actual elastic parameters of the stiff structure. A Young's modulus of approximately 100 times the equivalent Young's modulus of the soil should give an adequate contrast. Alternatively if using CSSM models for the soil, the bulk modulus of the structure (= $E/(3(1-2\nu'))$ specified should not exceed 100 times the effective bulk modulus of the soil (= $(1+e)p'/\kappa$). If there is considerable variation of the bulk modulus of soil from one region to another then choose an average, representative value for the soil.

Even after following these suggestions, you should watch out for problems with accuracy which manifest themselves in the form of large or gradually increasing equilibrium errors at the end of each increment. Anything more than 5% error in the equilibrium check would indicate numerical problems.

8.4.2 Undrained Analysis

In undrained analysis an equivalent bulk modulus of water is added to soil skeleton stiffness terms. In such an analysis if the results appear to be meaningless or if the pore pressures generated fluctuate wildly from point to point then the analysis should be repeated with a lower bulk modulus for water. In most cases the true undrained behaviour can still be captured with a low value of bulk modulus of water. However, too low a value would cause the behaviour to be partially drained. The analysis should then be repeated using a finer mesh with more elements.

The above numerical problem may also be evident from large equilibrium errors. If despite reducing the effective bulk modulus of water, the results appear to be meaningless then try running the analysis as a consolidation analysis. Replace all elements with the corresponding consolidation elements (LST replaced by LSTp, LSQ replaced LSQp, etc.). Re-run the Geometry Program to create a new link file, and then make the appropriate changes to the material properties data (specify non-zero permeabilities, and replace K_w with γ_w the unit weight of water. Then specify a small time step and run the Main Program. This should give the undrained response of the problem.

8.4.3 Consolidation Analysis

If there is any sign of numerical problem in the increment in which loading is applied then this is an indication of too small a time step. The analysis should be repeated with a larger time step for the load increment.

8.5 Undrained Analysis Using K_w

The effective stress law can be written in matrix notation:

Eqn. 8.5-1 $$\underline{\sigma} = \underline{\sigma'} + \underline{m} u_w$$

Here u_w is the pore water pressure and $\underline{m}$ is a vector indicating which stress terms participate in the effective stress relation. For example, if a fully three dimensional stress condition is considered:

$$\underline{\sigma} = [\sigma_x \ \sigma_y \ \sigma_z \ \tau_{xy} \ \tau_{yz} \ \tau_{zx}]^T$$

$$\underline{\sigma'} = [\sigma'_x \ \sigma'_y \ \sigma'_z \ \tau_{xy} \ \tau_{yz} \ \tau_{zx}]^T$$

and $\mathrm{m} = [1\,1\,1\,0\,0\,0]^T$

Suppose an element of soil undergoes an incremental total stress change $\Delta\underline{\sigma}$ which results in a change of pore pressure Δu_w and incremental strains $\Delta\underline{\varepsilon}$. Suppose also that incremental effective stresses are related to incremental strains by the relationship:

Eqn. 8.5-2 $$\Delta\underline{\sigma'} = D'\Delta\underline{\varepsilon}$$

D' may describe either an elastic or an elasto-plastic law. The assumption is now made that the volumetric strain experienced by the soil is due entirely to a change in volume of pore water. The volumetric strain experienced by the soil element can be written as $\underline{m}^T\Delta\underline{\varepsilon}$ and the volumetric strain experienced by the pore water is equal to $[(1 + e)/e]\underline{m}^T \Delta\underline{\varepsilon}$ where e is the current void ratio. Then the change in pore water pressure is given by:

Eqn. 8.5-3 $$\Delta u_w = \mathrm{K}_w \frac{(1+e)}{e} \underline{m}^T \Delta\underline{\varepsilon}$$

Combining this with the effective stress law and the incremental effective stress-strain relation the following equation is obtained:

Eqn. 8.5-4 $$\Delta\underline{\sigma} = D'\Delta\underline{\varepsilon} + m\mathrm{K}_w \frac{(1+e)}{e} \underline{m}^T \Delta\underline{\varepsilon}$$

CRISP uses this equation in the following way:

(i) CRISP expects that the material properties supplied relate to changes in effective stress.

(ii) When calculating the element stiffness matrices CRISP adds in the terms corresponding to the volumetric stiffness of the pore water.

(iii) Following the solution of the finite element equations CRISP calculates the changes in effective stresses and pore pressure separately.

In a drained analysis you set $K_w = 0$ and no changes in pore pressure are calculated. For elastic material behaviour the above procedure for an undrained analysis is equivalent to using a value of Poisson's ratio close to 0.5. However the above procedure has the advantage that the pore pressure changes are calculated explicitly and exactly the same technique is valid for an elasto-plastic material law. It is well known that in conventional linear elastic finite element analysis the use of a Poisson's ratio very close to 0.5 can lead to numerical ill conditioning of the finite element equations. The use of a value of K_w in the range suggested above is equivalent to the use of a value of Poisson's ratio in the range 0.49 to 0.499 and should give reasonably accurate results.

8.6 Numerical Integration

8.6.1 General

To obtain the stiffness matrix for a finite element, a matrix product has to be integrated over the volume of the element. For all but the simplest of elements, this integration would be very complex if attempted analytically, so it is common in finite element codes for the integration to be done numerically.

Numerical integration of a function within a given interval replaces the analytical expression by a summation of various terms that can be calculated within the interval. The terms comprise function values sampled at pre-determined points (known as integration points), multiplied by various weighting factors. Both the location of the integration points and the magnitude of the weighting factors are based on numerical integration rules - in CRISP (as with many FE codes) Gaussian integration rules are used. Further information on numerical integration can be found in any text book on numerical methods (e.g. Griffiths and Smith, 1990).

8.6.2 Integration Points

The finite elements in CRISP are all numerically integrated, and each has a fixed number of integration points (sometimes called Gauss points), depending on the element type. These points are arranged in a regular pattern around the axes of symmetry of the element, as shown in Fig 8.1.

Knowledge of the location of Gauss points can be helpful in the interpretation of program output. This is because quantities like element stresses and strains are actually calculated at these Gauss points in the first instance. They are known as secondary or derived quantities, as they are obtained from the primary unknowns (i.e. nodal displacements and excess pore pressures) once these have been computed.

8.6.3 Reduced Integration

In numerical integration, a certain number of points are required to integrate exactly a function of a given order (quadratic, cubic, etc.). For example, the linear strain quadrilateral (LSQ) assumes a quadratic variation in the displacement field, and needs 9 integration points in a 3 x 3 arrangement.

Some experts have argued that better results can be obtained in a FE analysis if fewer points (than the number needed for exact integration) are used. The reasoning behind this is as follows.

The FE method approximates a real system having infinite degrees of freedom (d.o.f.) with one which has finite d.o.f.. In force-displacement terms, this means that the FE model is too stiff and that displacements will tend to be underestimated. If a way could be found to reduce the stiffness of the FE model a little, then perhaps the calculated displacements would be closer to reality. This is essentially what reduced integration does. By using fewer integration points (e.g. 4 points in a 2 x 2 array for the LSQ) the stiffness of the element is underestimated (compared with the theoretically correct value). The hope is that the two different sources of error (finite d.o.f. and inexact integration) will cancel each other out, and certainly some excellent results have been achieved in this fashion. However, it is probably fair to say that reduced integration in commercial geotechnical analysis is not well established and should probably only be attempted by specialist users.

Reduced integration is not available in any of the elements provided with the standard release of CRISP, but it would not be difficult to implement You should refer to Appendix E of Britto and Gunn (1987) for instructions.

8.7 Non-Standard Constitutive Models

Whilst not totally comprehensive, the range of constitutive models provided in the standard release of CRISP is adequate for a large number of problems. However, CRISP is relatively amenable to the introduction of new constitutive models, with clear guidance having been provided in the Cambridge programmers documentation (e.g. Britto and Gunn, 1982; 1984 etc.). This facility has been exploited by many research workers, leading to a large number of customised versions of CRISP coming into existence over the years. A representative list of models implemented by U.K. workers is given in Table 8-1 below.

Table 8-1 *Some Constitutive Models Implemented in CRISP*

Constitutive Model	University	Reference
Nonlinear elastic (power law)	Surrey	Gunn (1992)
Naylor continuous plasticity critical state	Surrey	Hillier (1992)
Anisotropically consolidated Cam clay	Cardiff	Newson (1992)
Partially saturated	Sheffield	Ali (1993)
Simpson 'bricks on strings'	QMWC	Chandler (1995)

Although various customized versions of CRISP exist, their availability is variable and distribution is not undertaken by The CRISP Consortium Ltd. You should contact the university concerned to establish what can be obtained and how. CRISP User's Workshops, the Publications Directory, and the CRISP User's Newsletter are all potential sources of information on this.

In order to implement your own constitutive model you will need access to the source code for the Main Program and a good PC Fortran compiler. Detailed instructions on how to modify the code to accept a new model are given in Appendix D of Britto and Gunn (1987).

8.8 Frontal Solution Scheme

The frontal method of solving equations requires an efficient element numbering scheme. The element numbering scheme adopted by the user may not be (and often is not) the most efficient. Inefficient element numbering in any analysis of medium to large sized problems may prove to be

prohibitively expensive in terms of computional resources. No attempt is made in the Geometry Program to renumber the elements for efficient use of the frontal method, but the option to specify an alternative frontal sequence of the elements is allowed for in records H and I. If this alternate set of element numbers are specified by the user, then the elements are assembled in the sequence as specified in record I. If no alternative element numbering is provided then the elements are assembled in the same sequence as presented in records J. However the results output at the end of analysis (stresses at integration points for each element) will be printed in the ascending order of element numbering adopted by the user.

Alternatively, element renumbering can be carried out automatically with the front 'squashing' program, SQ. This takes the .GPD file and rewrites it with records H and I filled in with the optimum solution order for the elements.

It should be pointed out that the specification of a separate list of element sequence in record I (which is used as the sequence in which the element stiffness matrices are assembled within the Main Program) does not require any change in the element numbering you have adopted in the first place. It is important that the option provided via records H and I is made use of in any analysis where there are more than about 1000 d.o.f. in the mesh, to keep the computational cost down. Frontal optimising programs like SQ determine an element sequence which reduces the maximum frontwidth (f). The smaller the maximum frontwidth the less is the CPU time taken to solve the equations and the store required as seen below:

The time taken to solve equations

Eqn. 8.8-1 $= C \text{ x } f^2 \text{ x NDF}$

The amount of (backing) store required to store eliminated coefficients

Eqn. 8.8-2 $= (f + 4) \text{ x NDF}$

where:

f - maximum frontwidth

NDF - total number of variables (d.o.f.) in the mesh.

C - constant dependent on the type of computer being used.

ADVICE FOR NEW USERS 9

9.1 Basic Knowledge

Users are expected to be familiar with undergraduate level soil mechanics. An understanding of critical state soil mechanics is essential if using CSSM models in the analysis (it should be remembered that these are hypothetical models, and not real soils).

9.2 Introductory Analyses

If you are using the program for the first time, the best place to start is with the examples given in the Examples Manual of the CRISP documentation. This contains four different examples ranging from simple to moderately complex; between them they illustrate the use of virtually all of the features and options available in CRISP. You should familiarise yourself with the input data and the output results. The next few sections in this chapter give a few suggestions for other introductory analyses that you might wish to try.

9.2.1 Linear Elastic Analysis

Start with an elastic analysis for which a well known solution exists, e.g. settlement under a strip footing on an elastic medium. Compare the finite element results with the theoretical solution. If the results are significantly different then check whether you are using a sufficient number of increments.

9.2.2 Consolidation Analysis

Next, try simulating the Terzaghi one-dimensional consolidation analysis (see Section 9.3 of Britto & Gunn, 1987). Use a single element (8 noded, type 5). The first thing you will notice is that to analyse a time dependent problem you need to use a different element type to that for the drained/undrained type of analysis. The element used for a consolidation analysis has excess pore pressure as an additional variable at some of the nodes.

Use a small time step of 1 sec for the first increment block in which the loading is applied. Increment block 2 consists of 7 increments and you will notice that the time steps are increased gradually (see note in Chapter 4 of Britto & Gunn on how to choose these time steps). Also note the pore pressure boundary condition applied to the top surface. The application of 100 units of pressure results in an excess pore pressure of 100 units everywhere within the mesh. In this analysis the sample is assumed to drain only from the top surface. Hence the excess pore pressures should be 0 at the top. This is achieved by specifying either -100 as an incremental excess pore pressure fixity, or 0. as an absolute excess pore

Draw the theoretical settlement vs time plot and compare it with the CRISP results. Then, try re-running the consolidation analysis with about 20 increments in the second increment block. Compare the results with the previous analysis.

Plot the pore pressure distribution within the sample for a selected time factor (say 0.5). The 8 noded elements use a 3 x 3 integration scheme. The pore pressures are printed under 2 headings:

- Nodal displacements and excess pore pressures
- General stresses

Under the former heading, both incremental and cumulative values are printed. Only the excess pore pressure at the top and bottom can be found for this example because there are only nodes at the top and bottom of the mesh. The general stresses are available at each integration point or at element centroids. From the co-ordinates given establish the location of the integration points (see Fig. 8.1). Here you have values of pore pressures at 3 different depths within the mesh. How does this compare with the theoretical distribution of the pore pressures. Is it sufficient to use a single element to model the consolidation analysis?

Here is an indication of modelling the spatial variation of pore pressure by a linear variation. We have run into a problem where the number of elements used is not sufficient in the y direction (i.e. the direction of variation of the pore pressure). There is no variation of pore pressure along any horizontal line. Therefore it is sufficient to use only a single element in the x direction. (compare the results with Fig 3.21 given in Britto & Gunn, 1987).

Now try a mesh with more elements in the vertical direction (and graded in the same manner, smaller elements nearer the top drainage boundary) and see whether the results shown in Britto & Gunn could be improved upon. What conclusions do you come to in terms of:

a) Increasing the number of elements

b) Increasing the number of increments?

9.2.3 CSSM Analysis

Start with the simplest type of analysis, for example, a triaxial test (drained and undrained type). A single 8 noded quadrilateral element (type 4) is sufficient for this purpose. Try to reproduce the response of the undrained triaxial test example given in the Examples Manual where 2 cubic strain triangles are used. Try 2 different analyses with 6 and 50 increments but increasing the total strains from 3% to 5%. If you are familiar with critical state theory, calculate the theoretical end point (on the critical state line) of the analysis. Compare the values of q, p, u and e at the critical state with the results obtained from CRISP.

Try running the example of the undrained triaxial test with about 50 increments and compare the results with that shown in the Examples Manual. Now try running an analysis of a drained triaxial test. You need to make only one change to the Main Program input data and no changes to the Geometry Program data. This change is the effective bulk modulus of water, which becomes zero rather than 5.E5, making the response drained. If you have specified an initial pore pressure then it remains the same throughout the analysis, as no changes take place.

It is important to examine the printed output from the Main Program. This is the most important step in assessing the reliability of the results obtained. First of all check the output for any data errors that may have been picked up by CRISP. The program will carry on with the analysis when non-fatal errors are encountered, but it is unlikely that this is the analysis you wanted. CRISP only crashes due to fatal errors.

For CSSM analysis look at the following:

1. Messages concerning data errors. These messages are displayed on the screen and a copy is written to the .MPE (main program errors) file.
2. Equilibrium checks
3. Yield ratios
4. Warning messages
5. Inadmissible stress states

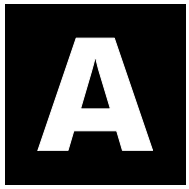
9.3 More Complex Problems

Now that you are familiar with the use of the CRISP program in terms of drained, undrained and time dependent (consolidation) analysis you can attempt a more difficult problem. At this stage, try tackling problems for which you have an independent solution.

Remember never to over-reach yourself in terms of attempting a complex analysis at the initial stages. Avoid any analysis which includes slip elements. These are only for experienced users. Use this element type only if absolutely necessary, you can get by in most cases without them.

INDEX

GEOMETRY AND MAIN PROGRAM INPUT DATA SUMMARY



This appendix provides a listing of all the variables that appear in the Geometry Program and Main Program input data files.

This is included for people who want to look at and/or modify the finite element program's runtime files.

It should be noted that the finite element code can deal with 3 dimensional analyses, so the 3D input specification is provided in this listing even though the Windows interface cannot currently handle 3D problems.

A.1 CRISP Data Files

The input data specification is divided into input for the CRISP Geometry Program and input for the Main Program, which are contained in two separate files on disk. A matching set of each of these files is required for an analysis to run.

The Geometry Program requires a file with the DOS extension .GPD (Geometry Program Data) and the Main Program requires a file with the extension .MPD (Main Program Data). The prefix to each of these files must be the same.

A.2 Geometry Program Input Data

Record A - Title

(one record only)

TITLE

TITLE Up to 80 alphanumeric characters

Record B - Job Number

(one record only)

LINK

LINK A code number set by the user.

Record C - Domain Information

(one record only)

NVTX	NEL	MXNDV	MXTYP	NDIM	IPLOT

NVTX Number of vertex nodes in the mesh.

NEL Number of elements in the mesh.

MXNDV Maximum number of vertex nodes in any element.

MXTYP Element type with most number of total nodes (per element) in mesh:

1 - 3-noded bar element with displacement unknowns (plane strain only).

2 - Linear strain triangle with displacement unknowns.

3 - Linear strain triangle with displacement and excess pore pressure unknowns (linear variation in pore pressure).

4 - Linear strain quadrilateral with displacement unknowns.

5 - Linear strain quadrilateral with displacement and excess pore pressure unknowns.

6 - Cubic strain triangle with displacement unknowns.

7 - Cubic strain triangle with displacement and excess pore pressure unknowns.

8 - Linear strain brick with displacement and excess pore pressure unknowns.

9 - Linear strain brick with displacement and excess pore pressure unknowns. (Linear variation in pore pressure, plane strain only).

12 - 3-noded beam element with displacement and rotation unknowns (plane strain only).

13 - 6-noded interface element with displacement unknowns (plane strain only).

14 - 2-noded bar element with displacement and rotation unknowns (plane strain only).

15 - 2-noded beam element with displacement and rotation unknowns (plane strain only).

NDIM Number of dimensions to problem:

2 - Two dimensional problem.

3 - Three dimensional problem.

IPLOT Plotting option parameter with the following possible values:

0 - No Plotting.

1 - Unnumbered mesh.

2 - Mesh and vertex node numbers.

3 - Mesh and side (edge) node numbers.

4 - Mesh and all node numbers.

5 - Mesh and element numbers.

6 - Mesh, vertex (corner) node numbers and element numbers.

7 - Mesh, side (edge) node numbers and element numbers.

8 - Mesh and all numbers.

Record D - Maxima Information

(one record only)

NUMAX	MUMAX

NUMAX Maximum value of user vertex node numbers.

MUMAX Maximum value of user element numbers.

Note: Use of 0 is only valid if user node and element numbers begin with 1 and there are no gaps in the numbering.

Record E - Programmer Debugging

(one record only)

ID1	ID2	ID3	...	ID10

Debugging options ID1 - ID10 can be set either to 0 (no printout) or 1 (give printout).

ID1 Print NCORR after exiting from routine CONECT which reads input element nodal connectivity (routine GPSUB).

ID2 Print ITAB after all side (edge) displacement node co-ordinates have been calculated (routine MIDSID).

ID3 Print IFR after all variables have been allocated store in FRONT (routine SFWZ).

ID4 Print NDEST after all variables have been allocated store in FRONT (routine SFWZ).

ID5 Print, NCORR, MREL, MRELVV after all vertex node co-ordinates and element nodal connectivities have been read (routine CONECT).

ID6 Print MFRN (Optimum frontal order of elements specified by the user) only relevant if IRNFR =1 (routine CONECT).

ID7 Print NCORR, MREL, MRELVV, NREL, NRELVV, LTYP, MAT, NQ after all nodes have been numbered (and co-ordinates calculated) (routine GPSUB).

ID8 Print contents of array G (only REAL part of array G is printed - routine MAIN2).

ID9 Not used.

ID10 Print NQ, NW (routine GPOUT).

Record F - Curved Sides

(one record only)

NSDZ	NSPZ	NDCUR	NPCUR

NSDZ Number of nodes along element sides (excluding end nodes) (Displacement nodes).

NSPZ Number of nodes along element sides (excluding end nodes) (Excess pore pressure nodes).

NDCUR Number of Curved sides (Displacement nodes).

NPCUR Number of Curved sides (Pore pressure nodes).

Note: These four parameters are only relevant if the element sides are curved and the user intends to specify the co-ordinates of nodes along these edges. Otherwise (default option) all four variables must be set to 0.

Record G - Nodes

(**NVTX** records)

N	X	Y	Z

N Vertex node number

X X co-ordinate of node.

Y Y co-ordinate of node.

Z Z co-ordinate of node (only specified for 3-D analysis).

Record H - Sequence Toggle

(one record only)

IRNFR

IRNFR Option to specify separate list of optimum frontal numbering of elements.

1 - Read separate list (see record I).

0 - Use sequence in which elements are read (see Record J).

Record I - Element Sequence

([NEL-1]/10 + 1 records - only present if IRNFR = 1)

MFRU (1)	MFRU (2)	...	MFRU (NEL)

MFRU (1)... MFRU (NEL) Optimum frontal numbering of elements - a list of element numbers in optimum solution order.

Note: There must be 10 values per record (except for the last record in this group).

Record J - Element Info.

(NEL records)

KEL	ITYP	IMAT	N1	N2	NV

KEL Element no.

ITYP Element type:

1 - 3-Noded bar (plane strain only).

2 - 6 Noded LST (2D).

3 - 6-Noded LSTp (2D consolidation).

4 - 8-Noded LSQ (2D).

5 - 8-Noded LSQp (2D consolidation).

6 - 15 Noded CuST (2D).

7 - 22 Noded CuSTp (2D consolidation).

8 - 20 Noded LSB (3D).

9 - 20 Noded LSBp (3D consolidation).

12 - 3 Noded beam element (plane strain only).

13 - 6 Noded interface (slip) element (plane strain only). *

14 - 2 Noded Bar (plane strain only).

15 - 2 Noded Beam (plane strain only).

16 8-noded quad (for seepage, special CRISP version)

17 4-noded quad (for seepage, special CRISP version)

18 4-noded quad, non-consolidation

19 8-noded quad, non-consolidation, reduced 2x2 integration

20 6-noded interface (slip) consolidation element with 2 pp nodes on one side

21 20 noded 3D interface (slip) element

Note: Elements with ITYP=9,16,17,17,18,19 and 21 are not available in the Windows interface of CRISP. They are only available in the finite element engine

IMAT Material zone number in the range 1 to 25.

N1, N2, ..., NV Vertex node numbers listed in anticlockwise order, where NV is the no of vertex nodes = MXNDV (based on element type MXTYP in Record C)

Note: If element types with different number of vertex nodes are mixed in a mesh then NV is the maximum number of vertex nodes in any element. Then all element entries must have NV nodal entries. For elements which have less than NV vertex nodes zeroes are added at the end of the record to make up the NV nodal entries.

***** For the interface element the nodes along the longer dimension should be input first.

Record K - Curved Sides

(NDCUR records - only present if NDCUR > 0)

MU	ND1	ND2	X1	Y1	Z1	...	XN	YN	ZN

MU Element number.

ND1, ND2 Nodes at either end of edge.

X1, Y1, ..., XN, YN Co-ordinates of intermediate (displacement) nodes along curved element side. There should be NSDZ displacement nodes (excluding end nodes).

Z1, ..., ZN For 3-D elements only.

Record L - Curved Sides

(NPCUR records - only present if NPCUR > 0)

MU	ND1	ND2	X1	Y1	Z1	...	XN	YN	ZN

MU Element number.

ND1, ND2 Nodes at either end of edge.

X1, Y1, ..., XN, YN Co-ordinates of intermediate (pore-pressure) nodes along curved edge. There should be NSPZ pore pressure nodes (excluding end nodes).

Z1, ..., ZN For 3-D elements only.

A.3 Main Program Input Data

Note: Records with underlined headings indicate that the record is either new or that there is a change in version 4 of CRISP

Record A - Title

(one record only)

TITLE

TITLE Up to 80 alphanumeric characters.

Record B - Job Number

(one record only)

LINK	IVERS

LINK A code number set by the user (must be the same as the one set in the corresponding GP file).

IVERS An optional number indicating the version number of CRISP. If more than 4, program will follow the new Material Models Interface

Record C1 - Analysis Set-up *v4

(one record only)

NPLAX	NMAT	NOIB	INC1	INC2	IPRIM	IUPD	ICOR	ISR	3SKH

NPLAX Stress system option:

0 - plane strain or 3D analysis.

1 - axisymmetric analysis.

NMAT Number of material zones.

NOIB Total number of increment blocks.

INC1 Increment number at start of analysis.

INC2 Increment number at finish of analysis.

IPRIM Number of elements to be removed to form primary mesh.

IUPD Geometry updating option:

0 - co-ordinates are not updated after each increment.

1 - co-ordinates are updated after each increment.

2 Updated Lagrangian algorithm will be used. This is a new feature for CRISP ver 4

ICOR Option to apply out of balance loads from one increment as correcting loads in the next increment:

0 - correcting loads are not applied.

1 - correcting loads are applied.

(ICOR is set to 1 only in the presence of elastic perfectly plastic materials).

2 iterative solution will be used, therefore must have record C6 (see below). This is a new feature for CRISP ver 4

ISR Option to stop and restart an analysis:

0 - stop/restart facility not being used.

1 - stop/restart facility to be used, saving only the last increment from the analysis.

2 - stop/restart facility to be used, saving all of the increments from the analysis.

3SKH Option to allow stress array VARINT to expand higher than the original limit (default original limit is 7 for 2D analysis, 9 for 3D analysis). This is needed for some models such as the 3-SKH which requires extra information for stress points (positions of yield surfaces, etc). A value entered for 3SKH will be added to the default value in order to expand the array.

Record C2 - MPO Output

(one record only)

INSOP	IBC	IRAC	NVOS	NVOF	NMOS	NMOF	NELOS	NELOF

INSOP In-situ stresses option:

0 - in situ stresses not printed.

1 - in situ stresses printed at centroids only.

2 - in situ stresses printed at all integration points.

IBC Boundary conditions output option:

0 - boundary conditions are not printed.

1 - boundary conditions are printed.

IRAC Reactions output option:

0 - reactions are not printed.

1 - reactions are printed.

NVOS Starting vertex node number for output *

NVOF Finishing vertex node number for output *

NMOS Starting midside vertex node number for output *

NMOF Finishing midside vertex node number for output *

NELOS Starting element number for output *

NELOF Finishing element number for output *

* This allows you to reduce the output and print out the results for nodes and elements which are within a specified range. This option is applied on the output code IOUT specified in record I and K2.

Record C3

(one record only)

PPS	ORS

PPS Total number of increments written for post processing.

ORS Total number of increments for stop-restart analysis.

Record C4

([PPS-1]/10 + 1 records - only present if PPS>0)

INCLST (1)	INCLST (2)	INCLST (...)	INCLST (PPS)

INCLST (1, ..., PPS) List of selected increments written to disk file for post processing.

Note: There must be 10 values per record (except for the last record in this group).

Record C5

([ORS-1]/10 + 1 records - only present if ORS>1)

INCLST (1)	INCLST (2)	INCLST (...)	INCLST (ORS)

INCLST (1, ..., ORS) List of selected increments written to disk file for stop-restart analysis. This should be a list of the final increment in each increment block chosen for stop-restart analysis.

Note: There must be 10 values per record (except for the last record in this group).

Record C6

This is present only if ICOR=2 in record C1 above and applies to CRISP version 4

TOLER	NITER

TOLER a limiting value for convergence used with the displacement convergence criterion of the iterative solution scheme.

NITER Maximum number of iteration allowed in each increment. If this limit is reached and convergence is not achieved, the program will stop.

Record D - Material Properties

IF IVERS=< 4, THE OLD FORMAT MAY STILL BE USED AS FOLLOWS:

(NMAT records)

For all element types with the exception of bar, beam and slip elements:

MAT	NTY	P (1)	P (2)	...	P (12)

MAT Material zone number - all elements given the same number in record J in Geometry Program will have the following properties:

NTY Soil model number:

1 - Elastic, isotropic/anisotropic.

2 - Elastic, linear variation with depth.

3 - Modified Cam Clay (MCC).

4 - Cam-Clay (CC).

5 - Elastic perfectly plastic (see parameter J below).

6 - Schofield soil model (SCHO) (requires 16 properties).

Property Number	Soil Model					
	1	**2**	**3**	**4**	**5**	**6**
P(1)	E_h	E_0	κ	κ	Eo	κ
P(2)	E_v	y_0	λ	λ	ν	λ
P(3)	ν_{hh}	m	ecs	Ecs	C	ecs
P(4)	ν_{vh}	ν	M	M	ϕ	M
P(5)	G_{hv}	0	G or ν'	G or ν'	y_0	G or ν'
P(6)	0	0	0	0	J	0
P(7)	0 for drained, K_w for undrained, γ_w for consolidation					
P(8)	γ_{bulk}					
P(9)	k_x for consolidation, 0 for drained or undrained					
P(10)	k_y for consolidation, 0 for drained or undrained					
P(11)	0	0	0	0	m_E	H
P(12)	0	0	0	0	m_C	S
P(13)	-	-	-	-	-	0
P(14)	-	-	-	-	-	0
P(15)	-	-	-	-	-	k_{xt}
P(16)	-	-	-	-	-	k_{yt}

Note: For the Schofield soil model, P(13) to P(16) are input on a second line.

The symbols above refer to the following model parameters:

γ_{bulk} Bulk unit weight of soil.

k_x, k_y Permeabilities in x and y directions.

γ_w Unit weight of water.

K_w Bulk modulus of water.

k_{xt}, k_{yt} Permeabilities in tensile crack region.

E_o Young's modulus at reference elevation.

m_E Rate of increase in Young's modulus with depth.

m_C Rate of increase in shear strength with depth.

H Slope of Hvorslev surface along constant volume section in p' : q space.

S Slope of tensile crack region in p' : q space.

J Elastic perfectly plastic yield criterion:

1 - Von Mises.

2 - Tresca.

3 - Drucker-Prager.

4 - Mohr-Coulomb.

κ Negative slope of the swelling and recompression lines in ln p', V plots.

λ Negative slope of the virgin compression line in ln p', V plots.

e_{cs} Voids ration on the critical state line for p'= 1.

M Slope of the critical state line in p' : q plots.

G Shear modulus of soil.

ν Poisson's ratio.

ν_{hh} Poisson's ratio for a stress increment in the vertical direction.

ν_{vh} Poisson's ratio for a stress increment in the horizontal direction.

G_{hv} Shear modulus of anisotropic soil.

For the bar, beam and interface elements the material property type number is 8, with the following material properties being specified.

Property Number	Soil Model 8		
	Bar	**Beam**	**Interface**
P(1)	E	E	C
P(2)	ν	ν	ϕ
P(3)	A	A	k_n
P(4)	0	I	k_s
P(5)	0	0	k_{sres}
P(6)	0	0	t
P(7)	0	0	0
P(8)	0	0	0
P(9)	0	0	0
P(10)	0	0	0
P(11)	0	0	0
P(12)	0	0	0

The symbols above refer to the following model parameters:

t Thickness or height of slip elements.

A Cross sectional area.

I Second moment of area of cross section.

k_n Normal stiffness.

k_s Shear stiffness.

$$k_n = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)}$$

$$k_s = G = \frac{E}{2(1+\nu)}$$

$$k_{sres} = G_{res}$$

IF IVERS>= 4 IN RECORD B (VERSION 4 OR ABOVE), THE NEW FORMAT WILL BE USED AS FOLLOWS:

(two records)

MAT	MATMOD

MAT Material zone number

MATMOD Material model number as indicated in the table below

P (1)	P (2)	...	P (12)

For more than 12 properties, a new line should be added

Property No.	Material Model									
	Elastic Isotropic. 101	Elastic Linear 102	Elastic Anisotropic 103	E.P von Mises 141	E.P Tresca 142	E.P Drucker-Prager 143	E.P Original Mohr Coulomb 144	E.P New Mohr Coulomb 145	Cam-Clay Original 161	Cam-Clay Modified 162
1	E	Eo	E_h	Eo	Eo	Eo	Eo	Eo	κ	κ
2	0	Yo	E_v	ν	ν	ν	ν	ν	λ	λ
3	ν	M	ν_{hh}	C	C	C	C	C	Ecs	Ecs
4	0	ν	ν_{vh}	0	0	ϕ	ϕ	ϕ	M	M
5	0	0	G_{vh}	Yo	Yo	Yo	Yo	Yo	G or ν'	G or ν'
6	0	0	0	0	0	0	0	ψ	0	0
7	0 for drained, K_w for undrained or γ_w for consolidation									
8	γ_{bulk}									
9	Kx	Kx	Kx	Kx	Kx	Kx	Kx	Kx	Kx	Kx
10	Ky	Ky	Ky	Ky	Ky	Ky	Ky	Ky	Ky	Ky
11	0	0	0	Me	Me	Me	Me	Me	0	0
12	0	0	0	Mc	Mc	Mc	Mc	Mc	0	0
13								ISYM(=0 for Unsymmetric solver, =1 for Symmetric solver using elastic D)		
14										
15										
16										

0BGeometry And Main Program Input Data Summary

Property No.	Soil Model								
	Cam-Clay Schofield .163	3-skh 164	Bar 181	Beam 182	Slip 191	Advanced MC 155	Duncan & Chang 121	Lade 157	Jardine-Gunn 123
1	κ	Slope of Swelling line κ	Young's Modulus E	E	C	Young's modulus E (KN/m2)	Modulus no K (dimensionless)	Modulus no.Ku (dimensionless)	Parameter a
2	λ	Slope of N.C line λ	Poisson's Ratio V	V	ϕ	Poisson's ratio v	Modulus no Ku (dimensionless)	Atmospheric pressure P_a (Default is 101.3 KN/m2)	Parameter n
3	Ecs	Ecs	Area of Cross Section A	A	Kn	Initial Cohesion Ci (KN/m2)	Poisson's ratio v	Poisson's ratio v	Poisson's Ratio v
4	M	Slope of C.S line M		I	Ks	Initial Friction angle ϕ_i (degrees)	Cohesion C (KN/m2)	Collapse Modulus C (Not needed for N.C)	Cohesion C (KN/m2)
5	G or v'	Shear Modulus G (KN/m2)			Ksres	Initial Dilation angle ψ_i (degrees)	Friction Angle ϕ	Collapse Constant p	Yo
6	0	0			t	Cohesion hard. h_c (dimensionless)	Atmospheric pressure P_a (Default is 101.3 KN/m2)	Yield Constant η_1	Mc
7	0 for drained, K_w for undrained or γ_w for consolidation								
8	γ_{bulk}								
9	K_x	$K_{x\,(m/s)}$				K_x	K_x	K_x	K_x
10	K_y	$K_{y\,(m/s)}$				K_y	K_y	K_y	K_y
11	H	Ratio of extent of Stress History Surface T (dimensionless)				Final (peak) friction angle ϕ_f (degrees)	Failure Ratio R_f (dimensionless)	Yield Exponent m	Critical Strain
12	S	Ratio of extent of Yield Surface S (dimensionless)				Final dilation angle ψ_f (degrees)	Exponent n (dimensionless)	Pl. Potent. Const R (=0 for N.C)	Spare
13	Kxt	Exponent of Hardening Modulus ψ (dimensionless)				Effective plastic strain needed to mobilise peak friction angle ε_f (dimensionless)		Pl. Potent. Const S	
14	Kyt	Shear Modulus exponent n (=0 for Constant G) (dimensionless)				Final (limiting) cohesion C_f (dimensionless)		Pl. Potent. Const t	
15		Shear Modulus exponent m (=0 for Constant G) (dimensiolness)				ISYM (=0 for unsymmetric solver,=1 for symmetric solver using elastic D matrix)		Work-Hard. Const α	
16								Work-Hard. Const β (=0 for N.C)	
								Work-Hard. Const P	
								Work-Hard.	

								Exponent l (=1 for N.C)	

Description of properties:

E	Young's modulus for isotropic material
E_h	Young's modulus in horizontal direction
E_v	Young's modulus in vertical direction
ν_{vh}	Poisson's ratio linking vertical and horizontal directions
ν_{hh}	Poisson's ratio linking both horizontal directions
G_{vh}	Shear modulus in the v-h plane
G	Shear modulus
kx,ky	Permeability coefficients in x and y directions
γ_{bulk}	Bulk unit weight of soil
γ_w	Unit weight of water
Eo	Young's modulus at datum elevation
Yo	Datum elevation at which E=Eo
ν	Poisson's ratio
m	Ratio of increase of Young's modulus with depth as in the formula $E = E_o + m(y_o - y)$
M_e	as above
c	Cohesion
M_c	Ratio of increase of cohesion with depth as in the formula $c = c_o + m_c(y_o - y)$
ϕ	Angle of friction
ψ	Angle of dilation
Ecs	Void's ratio at critical state
κ	Slope of Swelling line
λ	Slope of N.C line
M	Slope of C.S line
ν'	Poisson's ratio used in Cam-Clay models
H	Slope of Hvorslev surface in q:p' space for Schofield's Cam-Clay model
S	Slope of "no-tension" cut-off in q:p' space for Schofield's Cam-Clay model
Kxt	Permeability coefficient in x direction in tension zone for Schofield's Cam-Clay model
Kyt	As above in y direction

A	Cross sectional area of bar (tie) element
I	Second moment of area of beam element
Kn	Normal stiffness of joint interface (slip) element
Ks	Shear stiffness of joint interface (slip) element
Ksres	Residual shear stiffness of joint interface (slip) element
t	thickness of joint interface (slip) element

Some models require extra stress parameters which are stored in record C1 under 3SKH. This helps to expand array Varint.
These models are: 3-SKH model requires 11 extra stress parameters, Jardine-Gunn model requires 5 extra stress parameters

Record E - Removed Elements

([IPRIM-1]/10 + 1 records - only present if IPRIM > 0)

L(1)	L(2)	...	L(IPRIM)

L(1, ..., IPRIM) List of element numbers to be removed to form mesh at start of analysis.

Note: These elements are later "added" to simulate a construction event (e.g. embankment)

Furthermore, these elements do not have a stress history and therefore cannot be assigned CSSM model properties i.e. soil model number (NTY) cannot be 3, 4 or 6.

There must be 10 element numbers per record (except for the last record, which can have less than or equal to 10).

Record F - In Situ Stresses Toggle

(one record only)

INSIT	NNI

INSIT In situ stress option:

0 - Set in situ stresses to zero.

1 - Interpolate in situ stresses from a given set of reference points representing layers.

2 - Direct specification of in situ stresses at all integration points.

NNI* The number of in situ reference points (giving NNI-1 in situ layers)

***** These reference points are not the same as the nodes in the finite element mesh. No gaps are allowed in the reference point numbering and these should be input in ascending order in record G1. These serve as reference points for interpolation of in situ stresses.

Record G1 - In Situ Stresses

(NNI records - only present if INSIT = 1)

IN	YN	V(1)	V(2)	...	V(NVRS)

IN In situ reference point

YN Y co-ordinate of reference point

For 2-D analysis, NVRS= 7

For 3-D analysis, NVRS= 9.

	V(1)	V(2)	V(3)	V(4)	V(5)	V(6)	V(7)	V(8)	V(9)
2-D Analysis	σ'_x	σ'_y	σ'_z	τ_{xy}	u	0	p'_c	-	-
3-D Analysis	σ'_x	σ'_y	σ'_z	τ_{xy}	τ_{yz}	τ_{zx}	u	0	p'_c

Where:

u In situ pore pressure (the static head)

p'_c Specified as zero if soil model is not CSSM

Note: All stresses are effective stresses.

Records G1 must be input in the ascending order of the in situ reference points. No gaps are allowed in the reference point numbering.

Record G2 - In Situ Stresses

(Only present if INSIT = 2)

There are NEL sets of records in G2 and G3 - one set for each element.

IL

IL Element number.

Record G3 - In Situ Stresses

(NIP records - only present if INSIT = 2)

VAR(1) VAR(2) ... VAR(NVRS)

	VAR (1)	VAR (2)	VAR (3)	VAR (4)	VAR (5)	VAR (6)	VAR (7)	VAR (8)	VAR (9)
2-D Analysis	σ'_x	σ'_y	σ'_z	τ_{xy}	u	e	p'_c	-	-
3-D Analysis	σ'_x	σ'_y	σ'_z	τ_{xy}	τ_{yz}	τ_{zx}	u	e	p'_c

e Voids ratio.

p'_c Size of initial yield locus.

Note: For all soil models other than the cam clays and Schofield, e and p'_c must be set to zero.

Record H1 - In Situ Conditions

(one record only- omit if INSIT = 0)

NLODI	NFIXI	GRAVI

NLODI Number of edges with pressure loading (NLODI < 0) or number of nodes with point loads (NLODI >0)

Note: These must be in equilibrium with the in situ stresses.

NFIXI Number of edges which are restrained along boundary of the mesh. (i.e. displacement fixities)

GRAVI In situ gravity acceleration field- one of:

0 - For analysis which does not involve the effects of gravity (e.g. laboratory triaxial tests, consolidation tests).

1 - For analysis of field problems or prototypes.

n - For analysis of centrifuge tests (for example in analysing a test at 100g n is set to 100).

Record H2 - In Situ Loads

(NLODI records- omit if NLODI = 0 in record H1)

(a) NLODI > 0 (nodal point loads)

N	DFX	DFY	DFZ

N Node numbers

DFX Total in situ load in x direction

DFY Total in situ load in y direction

DFZ Total in situ load in z direction (only for 3D)

Sign Convention for point loads:

- Point loads are positive in the direction of the axes.

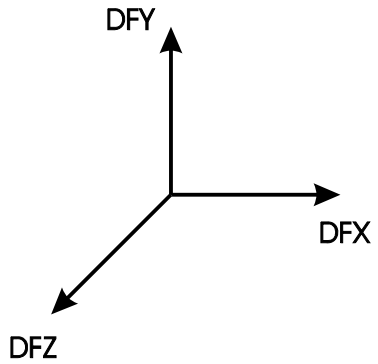


Figure A-1 Sign Convention For Point Loads

(b) NLODI < 0 (distributed pressure loading) - This option is only available for 2D analysis.

For linear strain triangles and linear strain quadrilaterals (LT = 2, 3, 4 or 5) record H2 is as follows:

L	N1	N2	T1	S1	T3	S3	T2	S2

For cubic strain triangles (LT = 6 or 7) record H2 is as follows:

L	N1	N2	T1	S1	T3	S3	T4	S4	T5	S5	T2	S2

Where:

L Element number.

N1, N2 Node numbers at either end of the loaded element side.

T1 Total in situ shear stress at N1.

S1 Total in situ normal stress at N1.

T3, T4, T5 Total in situ shear stress at edge nodes 3, 4 and 5.

S3, S4, S5 Total in situ normal stress at edge nodes 3,4 and 5.

T2 Total in situ normal stress at N2.

S2 Total in situ normal stress at N2.

Sign Convention for Stresses:

- Shear stresses which act in an anticlockwise direction about element centroid are positive.
- Compressive normal stresses are positive, i.e. pressures which cause compression in an element subjected to pressure, are positive.

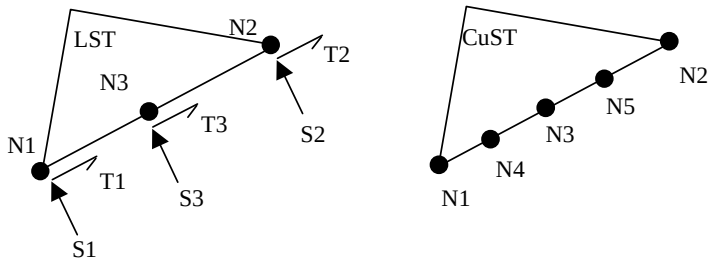


Figure A-2 Sign Convention For Stresses

Record H3 - In Situ Fixities

(NFIXI records - omit if NFIXI = 0 in record H1)

(a) 2-D Analysis (see Figure A-3(a)-(d))

For linear strain triangles and linear strain quadrilaterals (LT = 2, 3, 4 or 5), record H3 is as follows:

LNE	**ND1**	**ND2**	**IVAR**	**IFX**	**0**	**0**	**0**

For cubic strain triangles (LT = 6 or 7), record H2 is as follows:

LNE	**ND1**	**ND2**	**IVAR**	**IFX**	**0**	**0**	**0**	**0**	**0**

LNE Element number

ND1, ND2 Node numbers at either end of the restrained side.

IVAR The direction in which the side is restrained, according to:

1 - x direction.

2 - y direction.

IFX Fixity code = 1

(b) 3-D Analysis (see Figure A-3e-f)

For linear strain bricks (LT = 8 or 9), record H3 is as follows:

LNE	**ND1**	**ND2**	**ND3**	**ND4**	**IVAR**	**IFX**	**0**	**0**	**0**	**0**	**0**	**0**	**0**	**0**

LNE Element number.

ND1, ..., ND4 Node numbers on face in anti-clockwise order.

IVAR The direction in which the face is restrained, according to:

1 - x direction.

2 - y direction.

3 - z direction.

IFX Fixity code =1.

Note: Records H1, H2 and H3 are omitted if in situ stresses are all set to zero (i.e. INSIT = 0 in record F).

Note: For a re-started analysis using the STOP/RESTART facility records E to H3 are omitted from the input data. Also IPRIM is set to 0 in record C1.

Record I - Increment Block Information

(One only, but the group of records I to M is repeated for each increment block i.e. NOIB times).

IBNO	INCA	INCB	ICHEL	NLOD	ILDF	NFIX	NFXB	IOUT	IOPT	DTIME	ITMF	DGRAV	IMTFLG

IBNO Increment block number.

INCA Increment number at the start of the current increment block (INCA =>INC1 in record C1).

INCB Increment number at the end of the current increment block (INCB <= INC2 in record C1).

ICHEL Number of elements to be added/removed for the current increment block.

NLOD Number of incremental nodal loads (NLOD > 0) the or number of element sides with pressure loading (NLOD < 0).

ILDF Increment ratios, according to:

0 - The loading, prescribed displacements/ pore pressures are equally distributed over the INCB-INCA+1 increments.

1 - Read separate list of increment ratios for each increment (record K1).

NFIX Number of element sides (or faces) with prescribed value of the variable (degree of freedom).

NFXB Number of nodes with fixities or prescribed variables (only used with bar and beam elements).

IOUT Standard output for this increment block - a five digit number abcde, where:

Note: IOUT is set to zero if IOPT = 1.

a - Out-of-balance loads:

0 - No out of balance loads.

1 - Out of balance loads at vertex nodes.

2 - Out of balance loads at all nodes.

b - Extra parameters for Cam Clay and elasto plastic models only:

0 - No output.

1 - Parameters at element centroids.

2 - Parameters at all integration points.

c - Option for printing cumulative strains:

0 - No strains printed.

1 - Cumulative strains at element centroids.

2 - Cumulative strains at all integration points.

d - Option for printing general stresses:

0 - No stresses printed.

1 - Stresses at element centroids.

2 - Stresses at all integration points.

e - Option for nodal displacements:

0 - No displacements printed.

1 - Displacements at vertex nodes.

2 - Displacements at all nodes.

IOPT Output Option:

0 - Standard output given by IOUT for each increment in the increment block.

1 - Read separate list of output options for each increment (in record K2).

DTIME Time increment for consolidation analysis.

ITMF Time increment option:

0 - Time increment DTIME is divided equally between all the increments in the increment block.

1 - Read separate list of time steps for each increment (in record K3).

DGRAV Increment in gravity acceleration field:

= (Δn - change in numbers of gravities in a centrifuge test).

= 0 for any analysis other than that of a centrifuge test (in general).

Note: The number of increments in the increment block, NOINC (= INCB - INCA + 1), must not exceed 100.

IMTFLG If set to number i, it signifies that material zone i is to be redefined in the current load case, hence expect to read new material parameters for zone i in record K4 below

Record J - Removed Elements in Inc. Block

([ICHEL-1]/10 + 1 records - only present if ICHEL > 0)

L(1)	L(2)	...	L(ICHEL)

L(1), ..., L(ICHEL) List of elements numbers which are added/removed in this increment block. Note that while some elements are removed others can be added in the same increment block.

Note: There must be 10 element numbers per record (except for the last record which can have less than or equal to 10).

Record K1 - Incremental Load Ratios

([NOINC-1]/10 + 1 records - only present if NOINC > 0)

R(1)	R(2)	...	R(NOINC)

R(1), ..., R(NOINC) The ratio of incremental loads or incremental prescribed displacements to be applied in each increment.

Note: There must be 10 values per record (except for the last record in this group).

Record K2 - Incremental Output Options

([NOINC-1]/10 + 1 records - only present if IOPT = 1)

IOUT(1)	IOUT(2)	...	IOUT(NOINC)

IOUT(1), ..., IOUT(NOINC) The output options for each increment. Each value is a five digit code- abcde (see IOUT in record I).

Note: Leading zeroes can be omitted- e.g. 00011 can be input as 11.

Note: There must be 10 values per record (except for the last record in this group).

Record K3 - Incremental Time Steps

([NOINC-1]/10 + 1 records - only present if ITMF = 1)

DTM(1)	DTM(2)	...	DTM(NOINC)

DTM(1), ..., DTM(NOINC) The time steps for each increment (these are not ratios).

Note: There must be 10 element numbers per record (except for the last record, which can have less than or equal to 10).

Record K4 - Redefine material properties

Record K4, invoked only if IMTFLG above is set to i non-zero value (new for version 4)

If invoked, expect to read new material properties for i zone as in Record D exactly

Record L - Loads

(NLOD records - omit if NLOD = 0 in record I)

(a) NLOD > 0 (Nodal Point Loads)

N	DFX	DFY	DFZ

N Node number

DFX Increment of point load in x direction

DFY Increment of point load in y direction

DFZ Increment of point load in z direction (only for 3D analysis)

Sign Convention for point loads (see Figure A-1):

- Point loads are positive in the direction of the axes.

(b) NLODI < 0 (distributed pressure loading along element sides) - This option is only available for 2D analysis.

For linear strain triangles and linear strain quadrilaterals (LT = 2, 3, 4 or 5) record L is as follows:

L	N1	N2	T1	S1	T3	S3	T2	S2

For cubic strain triangles (LT = 6 or 7) record L is as follows:

L	N1	N2	T1	S1	T3	S3	T4	S4	T5	S5	T2	S2

Where:

L Element number.

N1, N2 Node numbers at either end of the loaded element side.

T1 Increment of shear stress at N1.

S1 Increment of normal stress at N1.

T3, T4, T5 Increment of shear stress at edge nodes 3, 4 and 5.

S3, S4, S5 Increment of normal stress at edge nodes 3, 4 and 5.

T2 Increment of normal stress at N2.

S2 Increment of normal stress at N2.

Sign Convention for Stresses (see Figure A-2):

- Shear stresses which act in an anticlockwise direction about element centroid are positive.
- Compressive normal stresses are positive, i.e. pressures which cause compression in an element subjected to pressure, are positive.

Record M - Edge Fixities

(NFIX records - omit if NFIX = 0 in record I)

(i) For 2D analysis (see Figure A-3a-d)

(a)	**LNE**	**ND1**	**ND2**	**IVAR**	**IFX**	**V1**	**V3**	**V2**	
(b)	**LNE**	**ND1**	**ND2**	**IVAR**	**IFX**	**V1**	**V3**	**V4**	**V5**	**V2**
(c)	**LNE**	**ND1**	**ND2**	**IVAR**	**IFX**	**V1**	**V2**	**0**	
(d)	**LNE**	**ND1**	**ND2**	**IVAR**	**IFX**	**V1**	**V3**	**V4**	**V2**	**0**

LNE Element number

ND1, ND2 Node numbers at either end of the restrained side

IVAR Variable which is prescribed or fixed:

1 - x-displacement.

2 - y-displacement.

3 - excess pore pressure.

IFX* Fixity code:

0 - Remove previously specified fixity.

1 - Increment value of variable.

2 - Absolute value of excess pore pressure.

3 - Total pore pressure.

V1, V2 Prescribed values at end nodes.

V3, V4, V5 Prescribed values at nodes along elements side (excluding end nodes).

(a) Displacement fixity:

Linear strain triangle - element type 2 and 3.

Linear strain quadrilateral - element type 4 and 5

IVAR = 1 or 2 IFX = 1

(b) Displacement fixity:

Cubic strain triangle - element type 6 and 7

IVAR = 1 or 2 IFX = 1

(c) Excess pore pressure fixity:

Linear strain triangle - element type 3

Linear strain quadrilateral - element type 5

IVAR = 3 IFX = 1, 2 or 3

(d) Excess pore pressure fixity:

Cubic strain triangle - element type 7

IVAR = 3 IFX = 1, 2 or 3

(ii) For 3D Analysis (see Figure A-3e-f) for the 20 node brick element only.

(e)	**LNE**	**ND1**	**ND2**	**IVAR**	**IFX**	**V1**	**V5**	**V2**	**V6**	**V3**	**V7**	**V4**	**V8**
(f)	**LNE**	**ND1**	**ND2**	**IVAR**	**IFX**	**V1**	**V2**	**V3**	**V4**	**0**	**0**	**0**	**0**

LNE Element number

ND1, ..., ND4 Nodes on element face in anticlockwise order

IVAR Variable that is prescribed or fixed:

1 - X-displacement.

2 - Y-displacement.

3 - Z-displacement.

4 - Excess pore pressure.

IFX* Fixity code:

0 - Remove previously specified fixity.

1 - Increment value of variable.

2 - Absolute value of excess pore pressure.

3 - Total pore pressure.

V1, V2, V3, V4 Prescribed values at corner nodes.

V5, V6, V7, V8 Prescribed values at nodes along element side.

(e) Displacement fixity

IVAR = 1, 2 or 3IFX = 1

(f) Excess pore pressure fixity

IVAR = 4 IFX = 1, 2 or 3

***** By re-specifying the record M, but with a fixity code of 0 (IFX = 0) previously specified displacement fixities can be released (i.e. the nodes are then free to move in the direction IVAR).

Record N - Node Fixity

(NFXB records - omit if NFXB = 0 in record I)

ND	IVAR	IFX	V

ND Node number.

IVAR Variable that is prescribed or fixed:

1 - X-displacement.

2 - Y-displacement.

3 - Excess pore pressure (for 2D analysis) or z displacement (for 3D analysis).

4 - Excess pore pressure (for 3D analysis).

5 - Rotation (for 2D plane strain analysis only).

IFX Fixity code:

1 - Incremental value of variable.

2 - Absolute value of excess pore pressure.

V Prescribed value of variable.

Note: This option is provided to specify prescribed values to a variable or restrain it at a node on an individual basis. This is provided to supplement the side fixities (record M) and caters for fixities that cannot be handled by side fixities. i.e. fixing an individual node or giving it a prescribed value. This record should only be used to fix a beam element (types 12 and 15) from rotating or to apply a prescribed rotation to a beam at a node. Can also be used to restrain the nodes of bar elements (types 1 and 14).

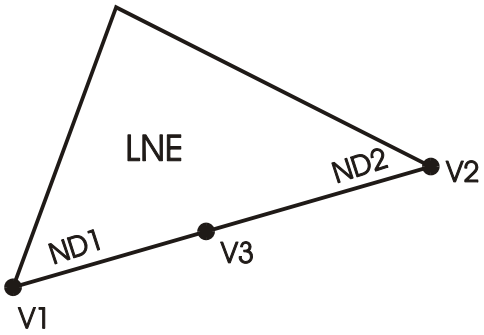


Figure A-3(a) Displacement Fixity Nodes (Element Types LT = 2, 3, 4, 5)

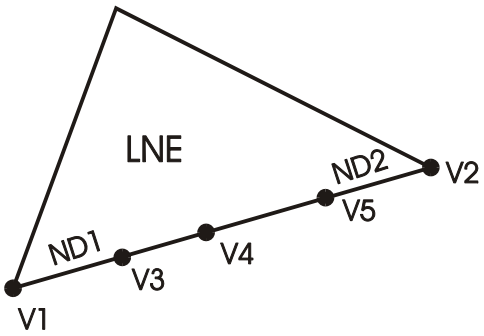


Figure A-3(b) Displacement Fixity Nodes (Element Types LT = 6, 7)

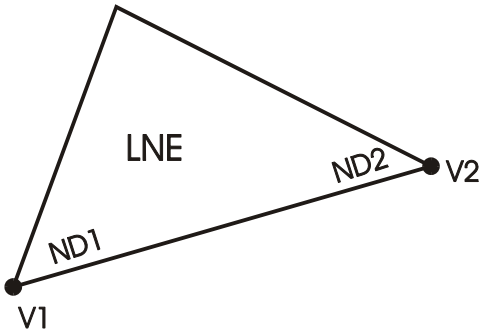


Figure A-3(c) Pore Water Pressure Fixity Nodes (Element Types LT = 3, 5)

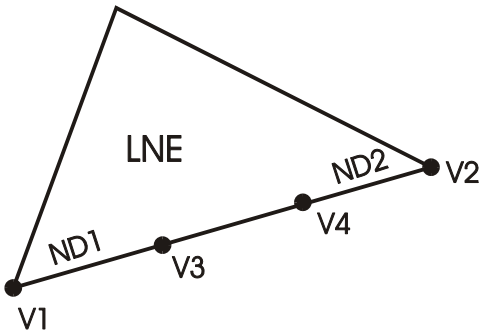


Figure A-3(d) Pore Water Pressure Fixity Nodes (Element Type LT = 7)

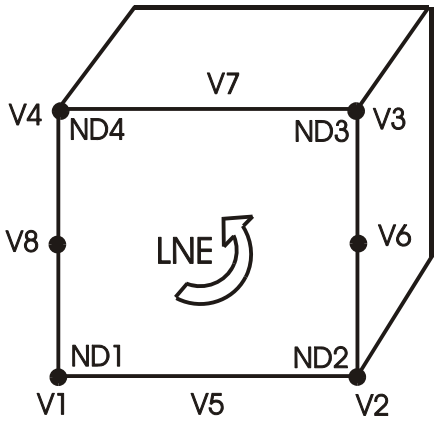


Figure A-3(e) Displacement Fixity Nodes (Element Types LT = 8, 9)

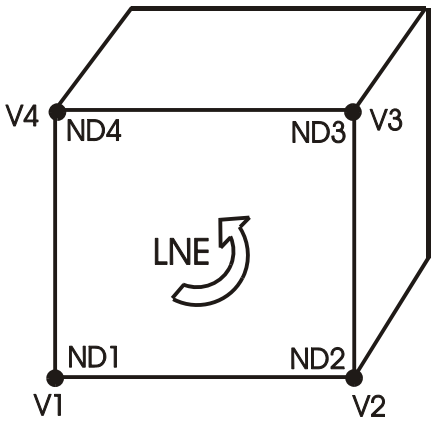


Figure A-3(f) Pore Water Pressure Fixity Nodes (Element Type LT = 9)

A.4 Data Summary

Geometry Program

Record Number	No of Records	Data
A	1	Title
B	1	Link
C	1	NVTX, NEL, MXNDV, MXTYP, NDIM, IPLOT
D	1	NUMAX, MUMAX
E	1	ID1, ID2, ID3, ..., ID10
F	1	NSDZ, NSPZ, NDCUR, NPCUR
G	NN	N, X, Y, Z
H	1	IFNFR
I	N	MFRU(1), MFRU(2), ..., MFRU (NEL)
J	NEL	KEL, ITYP, IMAT, N1, N2, ..., NV
K	NDCUR	MU, ND1, ND2, X1, Y1, ..., ..., XN, YN
L	NPCUR	MU, ND1, ND2, X1, Y1, ..., ..., XN, YN

Main Program

Record Number	No of Records	Data
A	1	Title
B	1	Link
C1	1	NPLAX, NMAT, NOIB, INC1, INC2, IPRIM, IUPD, ICOR, ISR
C2	1	INSOP, IBC, IRAC, NVOS, NVOF, NMOS, NMOF, NELOS, NELOF
C3	1	PPS, ORS
C4	1	INCLST(1), INCLST(2), ..., INCLST(PPS)
C5	1	INCLST(1), INCLST(2), ..., INCLST(ORS)
D	NMAT	MAT, NTY, P(1), P(2), ..., P(12)
E	NM	L(1), L(2), ..., L(IPRIM)
F	1	INSIT, NNI
G1	NNI	NI, YN, V(1), V(2), ..., V(NVRS)
G2	NEL	IL
G3	NIP	VAR(1), VAR(2), ..., VAR(NVRS)
H1	1	NLODI, NFIXI, GRAVI
H2	NLODI < 0	L, N1, N2, T1, S1, T3, S3, T2, S2; or L, N1, N2, T1, S1, T3, S3, T4, S4, T5, S5, T2, S2
	NLODI > 0	NDE, DFX, DFY, DFZ
H3	NFIXI (2D)	LNE, ND1, ND2, IVAR, IFX, 0, 0, 0; or LNE, ND1, ND2, IVAR, IFX, 0, 0, 0, 0, 0
	(3D)	LNE, ND1, ND2, ND3, ND4, IVAR, IFX, 0, 0, 0, 0, 0, 0, 0, 0
I	1	IBNO, INCA, INCB, ICHEL, NLOD, ILDF, NFIX, NFXB, IOUT, IPOT, DTIME, ITMF, DGRAV
J	NCH	L(1), L(2), ..., L(ICHEL)
K1	NS	R(1), R(2), ..., R(NOINC)
K2	NS	IOP(1), IOP(2), ..., IOP(NOINC)
K3	NS	DTM(1), DTM(2), ..., DTM(NOINC)
L	NLODI < 0	L, N1, N2, T1, S1, T3, S3, T2, S2; or L, N1, N2, T1, S1, T3, S3, T4, S4, T5, S5, T2, S2
	NLODI > 0	NDE, DFX, DFY, DFZ

Record Number	No of Records	Data
M	(2D)	LNE, ND1, ND2, ND3, ND4, IVAR, IFX, V1, V3, V2; or LNE, ND1, ND2, ND3, ND4, IVAR, IFX, V1, V3, V4, V5, V2; or LNE, ND1, ND2, ND3, ND4, IVAR, IFX, V1, V2, 0; or LNE, ND1, ND2, ND3, ND4, IVAR, IFX, V1, V3, V4, V2, 0
	(3D)	LNE, ND1, ND2, ND3, ND4, IVAR, IFX, V1, V5, V2, V6, V3, V7, V4, V8; or LNE, ND1, ND2, ND3, ND4, IVAR, IFX, V1, V2, V3, V4, 0, 0, 0, 0
N	NFXB	ND, IVAR, IFX, V

Note: Records I – N appear NOIB times. Once for each increment block.

$$NOINC = INCB - INCA + 1$$

Where,

$$NM = \frac{(IPRIM - 1)}{10} + 1$$

$$NCH = \frac{(ICHEL - 1)}{10} + 1$$

$$NS = \frac{(NOINC - 1)}{10} + 1$$

GEOMETRY AND MAIN PROGRAM INPUT DATA DESCRIPTION

B

B.1 Geometry Program Input Data

B.1.1 Record A - Title

The mesh TITLE should include sufficient information to distinguish it from other meshes.

Note that comment lines cannot be included before the TITLE record. However these can be included anywhere else in the Geometry Program data file. Comment lines are identified by C in the first column (as in Fortran).

B.1.2 Record B - Job Number

The Geometry Program stores basic information describing the finite element mesh on disk ; subsequently read by the Main Program. You will often set up several (different) finite element meshes, and then run the Main Program a number of times for each of these meshes. To help ensure that a Main Program run accesses the correct “link” file, the LINK number is stored on the link file by the Geometry Program and must be quoted correctly in the input for the Main Program. Hence LINK should be set to a different integer number for each finite element mesh that you specify.

B.1.3 Record C - Domain Information

NPLAX

NPLAX determines whether the analysis is plane strain (NPLAX=0) or axisymmetric (NPLAX=1).

NVTX

NVTX is the total number of vertex (i.e. corner) nodes in the finite element mesh. The Geometry Program automatically generates node numbers and co-ordinates for any nodes lying on element sides or within elements.

MXTYP

If different element types are present in the same mesh then MXTYP is set to the element type with the most nodes per element. Although it is possible to include more than one type of finite element in a mesh, often all elements will be of the same type. MXTYP can take any of the type numbers of the elements available in CRISP.

MXTYP	Element
1	Bar (3-noded)
2	LST
3	LST (Consolidation)
4	LSQ
5	LSQ (Consolidation)
6	CuST
7	CuST (Consolidation)
8	LSB
9	LSB (Consolidation)
12	Beam (3-noded)
13	Interface (6-noded)
14	Bar (2 noded)
15	Beam (2 noded)

For example, if element types 2 and 4 are mixed in a finite element mesh, the 8-noded quadrilateral (4 user defined vertex nodes and 4 computer defined midside nodes) element has more nodes and hence MXTYP is set to 4.

Permissible combinations of elements are given in Section 3.6.

B.1.4 Record D - Maxima Information

NUMAX, MUMAX

These parameters need to be specified as non-zero **only** if there are gaps in the vertex node numbering and element numbering, respectively. This information is necessary in order to allocate sizes to arrays which store the node numbers and element numbers, thus avoiding the imposition of an arbitrary size limit.

B.1.5 Record E - Programmer Debugging

ID1 to ID10

Normally, these flags will all be set to zero. They have been left in for the benefit of users/programmers who may want to change the Geometry Program and will thus need to check intermediate output.

B.1.6 Record F - Curved Sides

NSDZ, NSPZ, NDCUR, NPCUR

The program does not have the facility to calculate co-ordinates of nodes assuming that the element sides are curved - you must directly specify the co-ordinates of the intermediate nodes along curved element sides in Records K and L.

Displacement and pore-pressure nodes are dealt with separately. For element types 2, 4 and 6 (non-consolidation elements) and also element types 3 and 5 (no pore pressure nodes along sides) NSPZ and NPCUR must be set to zero. Record L is then omitted from input.

B.1.7 Record G - Nodes

Each element and each vertex node in the finite element mesh must be assigned a unique (integer) number. It is not necessary for either the node numbers or the element numbers to form a complete set of consecutive integers, i.e. there may be "gaps" in the numbering scheme adopted. This facility means that users may modify existing finite element meshes by removing elements without the need for renumbering the whole mesh.) The Geometry Program assigns numbers in the range 751 upwards to nodes on element sides and in element interiors (unless there are >750 vertex nodes in the mesh, when additional node numbering will commence at the next available number).

B.1.8 Records H and I

The frontal method of solving equations requires an efficient element numbering scheme. The element numbering scheme adopted by the user may not necessarily be the most efficient. Inefficient element numbering in a medium to large sized analysis may prove excessively time consuming. No attempt is made in the program to renumber the elements for efficient use of the frontal method. With element renumbering programs for the frontal method available, the option to specify an alternative frontal sequence of the elements is allowed for in records H and I. If this alternate set of element numbers are specified (IRNFR = 1) by the user then the elements are assembled in the sequence as specified in record I. If no alternative element numbering is provided (IRNFR = 0) then the elements are assembled in the same sequence as presented in records J. However the results output at the end of analysis (stresses at integration points for each element) will be printed in the ascending order of element numbering adopted by the user.

See also Section 8.12.

B.1.9 Record J - Element Information

IMAT

The user must assign a zone number (in the range 1 to 25) to each finite element. The zone number associates each element with a particular set of material properties (Record D of Main Program input). For example, if there are three zones of soil with different material properties, zones 1 and 2 may be modelled by Cam-clay with distinct material parameters whereas zone 3 may be modelled as linear elastic. (NB: "gaps" in zone numbering are not allowed.)

NV

The number of vertex nodes (NV) which must be specified for each element (after the element number, element type number and material zone number) is equal to the maximum number of vertex nodes in any element in the mesh. For example if linear strain triangles (LST, type 2, 6-noded) and the linear strain quadrilateral (LSQ, type 4, 8-noded) are mixed, then all relevant records must have 4 vertex node numbers. For the triangular elements, a trailing zero is added to complete the record.

The vertex (or corner) node numbers are specified in the anti-clockwise order starting from any node for the 2-D elements. The 20-noded brick element (LSB) has eight corner nodes which have to be specified. Any face is selected and the 4 nodes on that face are specified in the anticlockwise order (starting from any node). This is followed by the other 4 nodes which are specified such that they exactly match (or map onto) the first 4 nodes (this is illustrated in Fig. 3.8).

Interface Elements

When defining the element nodal links, the first 2 nodes specified should be along the length of the element. The 4 nodes should also be entered in the anticlockwise direction. For the example shown in Fig. 3.13, the following two options are correct:

	Element Number	Type	Material Zone	Nodes			
				1	2	3	4
Record J:	78	13	2	12	20	45	23
Record J:	78	13	2	45	23	12	20

However the option shown below is incorrect:

Record J:	78	13	2	20	45	23	12

because the first two nodes presented (20 and 45) lie along the narrow dimension of the element.

In this example the same co-ordinates could be assigned to nodes 12 and 23 (similarly for nodes 20 and 45). The thickness of the element **t** (i.e. the width along the narrow dimension) is input as part of the material properties. The material zone number specified for the slip elements should be different to that for the soil (obviously).

B.1.10 Records K and L - Curved Sides

The co-ordinates of nodes along element sides (for both displacement and excess pore pressure nodes) need to be specified if the element sides are curved (NSDZ $\neq$ 0, and NSPZ $\neq$ 0, in record F). However for element type 3 (LST/p) there are no excess pore pressure nodes along the element sides, so only record K needs to be specified. For element type 7 (CuST/p) both types of records (K and L) have to be specified.

The element number is followed by nodes ND1 and ND2 (which are at either end of the side) to identify the element side. Following this, the nodal co-ordinates of intermediate nodes along the element side are given in sequence from node ND1 to ND2 (note that the co-ordinates of nodes ND1 and ND2 are not specified).

B.2 Main Program Input Data

B.2.1 Record A - Title

The analysis TITLE should include sufficient information to distinguish it from other analyses.

Note that comment lines cannot be included before the TITLE record. However, these can be included anywhere else in the Main Program data file. Comment lines are identified by C in the first column (as in Fortran).

B.2.2 Record B - Job Number

The link number must be the same as that specified in the Geometry Program input data for the appropriate finite element mesh. (See Record B, Geometry Program input).

B.2.3 Record C1 - Analysis Set-up

NMAT

Must be equal to the number of different material zones specified in the geometry program.

INC1, INC2

It is essential that INC2 ≥ INC1 . If INC1 > 1, this analysis is a restart analysis and records E to H3 are omitted.

IPRIM

This is the number of finite elements that must be removed to form the primary finite element mesh, before the analysis is started.

IUPD

IUPD = 0 corresponds to the normal assumption made in linear elastic finite element programs, and also in most finite element programs with non-linear material behaviour. External loads and internal stresses are assumed to be in equilibrium in relation to the original (i.e. undeformed) geometry of the finite element mesh. This is usually known as the "small displacement" assumption.

IUPD = 1 is used when the nodal co-ordinates are to be updated after each increment of the analysis, by adding to the co-ordinates the displacements undergone by the nodes during the increment. The stiffness matrix of the continuum is then calculated with respect to these new co-ordinates during the next analysis increment. The intention of this process is that at the end of the analysis equilibrium will be satisfied in the final (deformed) configuration.

ICOR

ICOR = 1 when elastic-perfectly plastic models are used (i.e. property type 5) and ICOR = 0 for all other models. When elastic perfectly plastic models are used stresses at points which have yielded are corrected back to the yield surface. Corrected stresses will not necessarily be in equilibrium with the applied loading. If ICOR is set to 1, any "unbalanced" loads which arise from this procedure are added to the next increment of loading. See Section 8.2 for further details.

ICOR has no effect on CSSM models, as there is no correction to the stresses made at the end of the increment. When the stress state leaves one yield locus (either due to softening or hardening), it is always possible to find a new yield locus through the new stress state. Therefore in theory the stresses are in equilibrium at the end of the increment, without having to correct it.

NOIB

The analysis is sub-divided into one or more increment blocks. Each increment block consists of one or more increments. The use of increment block is adopted for two reasons: (1) removal of elements (excavation) and addition of elements (construction) can be carried out over a number of increments (i.e. an increment block) instead of a single increment, and (2) increments with repeated application of loading (or non-zero prescribed displacements) can be grouped together as an increment block (provided that boundary conditions do not change) thereby reducing the amount of data input.

IBNO

Consider an analysis which is split into two separate runs using the stop-restart facility (see 8.3). It should be remembered that IBNO always starts with 1 for each of these runs. Increment blocks are just a convenient way of grouping a number of increments.

For example if run 1 consists of 2 increment blocks each with 10 increments then assuming that the re-started run has a further 2 increment blocks (each with 10 increments) then the input data are as follows:

	NOIB	**INC1**	**INC2**
Run 1 / Record C1	2	1	20
Run2 / Record C1	2	21	40

	IBNO	**INCA**	**INCB**
Run 1 / Record I / Inc. block 1	1	1	10
Run 1 / Record I / Inc. block 2	2	11	20

	IBNO	INCA	INCB
Run 2 / Record I / Inc. block 1	1	21	30
Run 2 / Record I / Inc. block 2	2	31	40

ISR

CRISP can be stopped and restarted, allowing a lengthy analysis to be split into a number of shorter runs. The facility is particularly useful for reviewing and perhaps altering the size of the load increments without having to repeat the entire analysis. The input data for a starting run is exactly the same as for a normal run except that ISR (in record C1) is set to 1 or 2 rather than zero. When a run is restarted ISR is set equal to 1 or 2 and records E to G3 are omitted from the input data (in this case the details of the current stresses are read from the restart file).

A value of INCS > 1 in record C1 indicates that this is a restarted run. INCS must follow on in sequence from the previous analysis. When ISR = 1 it is only possible to restart the analysis from the last increment of a previous run. When ISR = 2 it is possible to restart from any previous increment. Mixing ISR = 1 and ISR = 2 in a series of runs is not permitted. The results from a previous run are always read from unit IR1 and the results from the current run are stored on unit IW2. The restart files for ISR = 2 will be large.

Option 1 : ISR = 2

The STOP/RESTART facility which uses two disk files enables one to either break up a large analysis into a series of medium sized runs or go back a few increments and then continue the analysis perhaps with different load increment size.

The first run in a series of runs (i.e. a complete analysis) is called a STARTing run. Any subsequent run in that analysis is a RE-STARTing run. For a STARTing run the input data is exactly the same as a single run consisting of a complete analysis except for ISR = 2 in record C1. For a RE-STARTing run records E to H3 of the input data are omitted.

The user can re-start an analysis from any point (increment) from a past run. From which point the analysis is to be continued is indicated by INCS in record C1. When INCS > 1, which implies a re-started run, all incremental results from increment 1 to (INCS-1) are read from unit IR1 and written to unit IW2, and the analysis is continued from increment INCS up to INCF. As each increment is completed the results are written to unit IW2. At the end of the run unit IW2 contains all the results from increment 1 to INCF and any subsequent run can be continued from any point between increment 1 and INCF.

Option 2 : ISR = 1

This option also makes use of disk files but only the results from the last increment are written to the disk file. When the analysis is re-started this results is read and the analysis is continued. With this option there is no possibility of going back and re-starting at an earlier stage and then the analysis can only be continued from the point left off.

B.2.4 Record C2 - MPO Output

INSOP, IBC, IRAC

These parameters operate independently, whereas other output control parameters operate in conjunction with IOUT (records I, K2). This selects the range of vertex node numbers, mid-side node numbers and element numbers for outputting the relevant information. For the nodes it is the displacements and for the elements it is the general stresses and additional parameters (for example Cam-clay parameters for the critical state models).

NELOS, NELOF, NMOS, NMOF

If you require the general stresses to be printed at all integration points for elements in a particular range, you should specify NELOS as the first element in that range and NELOF as the last element in that range. In addition you should specify d = 2 in the output code abcde for IOUT (e.g. 11121). Specifying zero values for both NELOS and NELOF means no information will be printed for that category of output, irrespective of the option set by IOUT. Similarly, if both NMOS and NMOF are set to zero, then even if option 2 is used for the nodal displacement (e = 2, e.g. IOUT = 00112) no displacements will be printed at the mid-side nodes. Therefore the rule used between these range of numbers and IOUT (in record I) is logical product.

B.2.5 Record C3, C4, C5

In order to use the CRISP post processor, the results at the end of selected increments are written to a disk file (filename .PPS). If a stop-restart analysis is to be carried out, ORS must be specified.

PPS specifies the total number of increments to be written to the disk file and record C4 gives the list of increment numbers. These increment numbers must be specified is ascending order. For example, in an analysis consisting of 100 increments if 4 increments (10, 20, 80 and 100) are to be written to the disk file.

ORS specifies the total number of increments to be written for stop-restart analysis and record C5 gives the list of increment numbers. These increment numbers must be specified in ascending order and each one should be the final increment number in an increment block for which a stop-restart analysis is required. For example, in an analysis consisting of 100 increments, with 20 increments in each of five increment blocks. If increment blocks 2 and 4 are chosen for stop-restart, then increments 40 and 80 will be written to the disk file:

Record C3	**PPS ORS**
	4 2
Record C4	**INCLST(1) ... INCLST(PPS)**
	10 20 80 100
Record C5	**INCLST(1) ... INCLST(PPS)**
	40 80

B.2.6 Record D - Material Properties

Detailed comments on the meanings and definitions of the constitutive model parameters listed in Appendix A can be found in the main body of the Technical Reference (see Chapter 4).

Note that, for structural elements, some of the "material properties" required are in fact sectional properties (e.g. A, I etc.).

B.2.7 Record E - Removed Elements

Record E simply comprises a list of numbers of elements which are to be removed before the start of the analysis, to create the primary mesh. The total number of such elements has been specified earlier as IPRIM (Record C1).

B.2.8 Records F, G1, G2 and G3

The purpose of record types F, G1, G2 and G3 is to enable the program to calculate the stresses (and for CSSM models the size of the yield locus specified by p'_c) before the analysis starts.

The parameters specified in records G1 and G3 are as follows:

σ'_x Effective horizontal stress in x direction, (r - radial)

σ'_y Effective vertical stress in y direction, (z - vertical)

σ'_z Effective out-of-plane stress in z direction, (q - circumferential)

τ'_{xy} Shear stress in xy plane of plane strain (rz plane in axisymmetry)

u Initial pore pressure (usually the static head)

$\mathbf{p'_c}$ Size of the initial yield locus (p' when q = 0).

Note that the initial voids ratio, e, is not specified in the input data. Instead simply specify a zero value for it (see records G1, G3). This is because the voids ratio is calculated within the program, using all the other information supplied.

In problems where the stresses do vary in the horizontal direction a separate option (option 2: INSIT = 2) is provided in specifying the stresses. In this option (see records G2 and G3) the user has to specify directly the in situ stresses at each integration point for all the elements (**including** the void ratio).

B.2.9 Record I - Increment Block Information

CRISP allows the user to describe a sequence of increments as an "increment block".

A previously fixed node may be released by using a fixity code of 0 (zero).

NLOD, NFIX, ILDF

The loading (NLOD), self weight loads (DGRAV) and prescribed displacements (and pore pressures) (NFIX) are specified for the entire increment block, and are applicable to that particular increment block only. The loading and any non-zero prescribed displacement for the individual increments are taken as ratios (< 1) of that for the increment block.

There is no restriction on how these loading and non-zero prescribed displacements are divided among the increments in an increment block. They are equally divided between all the increments if ILDF = 0 in record I. However if the user wants to distribute the loading (and non-zero prescribed displacements) unevenly between the increments then by setting ILDF = 1, a separate list of increment unit ratios are read in record K1. (This is generally useful in an analysis where large load increments can be applied when the problem is in the elastic state and smaller load increments as plastic yielding takes place).

DGRAV

DGRAV is used in problems in which the material's self-weight is increased during an analysis (e.g. in the "wind-up" stage of a centrifuge test increasing centrifugal acceleration can be regarded as having this effect). This is defined in terms of number of G's in a centrifuge test.

DGRAV should be set to zero, in general, if you are analysing field problems or laboratory tests.

B.2.10 Record K1 - Incremental Load Ratios

It should be noted that the same ratios R(I) etc. (record K1) apply to the pressure loading (NLOD - record L), the gravity loading (DGRAV - record I) and the prescribed displacements (and pore pressures) (NFIX - record M). These ratios also apply to any implied loadings due to removal or addition of elements in that increment block.

The sum of ratios R(I) must be equal to 1. However, some of these ratios can take zero values as illustrated in the example given below.

B.2.11 Record K3 - Incremental Time Steps

ITMF

In a consolidation analysis the time increment DTIME (> 0) is specified for the entire block. If ITMF = 0 in record I then DTIME is equally divided among all the increments in the increment block. However if ITMF = 1 then the user directly specifies (in record K3) the time increments for each increment. Unlike the increment ratios R(I) etc. (record K1) these are actual time steps for the increments and not ratios. None of these can be zero and for reasons of consistency DTIME in record R must be set equal to the sum of all the time steps in the increment block.

The use of records K1 and K3 is illustrated by an example. In a consolidation analysis of 100 secs total duration spread over 9 increments the load is gradually applied in 3 secs and the subsequent transient response is analysed.

(a) First, the example is used to illustrate the use of a single increment block

Table B-1 Record I

IBNO	INCA	INCB	ICHEL	NLOD	ILDF	...	DTIME	ITMF	DGRAV
1	1	9	0	5	1	...	100	1	0

Table B-2 Record K1

R(1)	R(2)	R(3)	R(4)	R(5)	R(6)	R(7)	R(8)	R(9)
0.33	0.33	0.34	0	0	0	0	0	0

Table B-3 Record K3

DTM1	DTM2	DTM3	DTM4	DTM5	DTM6	DTM7	DTM8	DTM9
1	1	1	2	5	10	10	20	50

(b) As an alternative the analysis could be split into two increment blocks. In the first increment block the loading is applied, whereas in the second consolidation takes place with no change in the load.

Table B-4 Record 1

IBNO	INCA	INCB	ICHEL	NLOD	ILDF	...	DTIME	ITMF	DGRAV
1	1	3	0	5	0	..	3	0	0
2	4	9	0	0	0	..	97	1	0

Table B-5 Record K1

Not present for either increment blocks (ILDF = 0 in record I)

Table B-6 Record K3

	DTM1	DTM2	DTM3	DTM4	DTM5	DTM6
Inc Block 2	2	5	10	10	20	50

NB: not present for first increment block (ITMF = 0 in record I)

Option (a) cannot be used in cases where you want to change the pore pressure boundary condition (for example to allow drainage to take place) immediately after applying the load. Under these circumstances one has to use the option (b). The appropriate pore pressure boundary condition can be specified at the beginning of next increment block.

B.2.12 Record M - Edge Fixities

IVAR, IFX

Note: IFX = 2 should **NOT** be used. IFX = 3 fulfils the same function but more simply. Details on IFX = 2 have been included to ensure backwards compatibility with older versions of CRISP.

Similar to the displacement fixity, pore pressure fixities are, in general, specified along the mesh boundary. An exception is when it is necessary to specify the pore pressures along an interface which separates a draining layer from a consolidating layer. Pore-pressure boundary conditions are not specified in a drained or an undrained analysis. These are only specified in a consolidation analysis.

The boundary of the finite element mesh in a consolidation analysis can either be

(a) impermeable (undrained) boundary

(b) or a drained boundary or a boundary with a known pressure head

In the former case you need not specify any pore pressure boundary conditions. All boundaries are automatically assumed to be impermeable, unless specified otherwise. Care is needed in specifying the pore pressure fixities in the case of the latter.

If no further pore pressure changes are to take place along a given boundary, at any stage of the analysis then fixity code (IFX) 1 is used with a zero value for the pore pressure. This means that the total pore pressure is held at the value prior to the specification of the pore pressure fixity.

If the total pore pressure along a boundary is to remain at a given value you subtract the in situ (normally hydrostatic) pore pressure from this value and then specify this with a IFX =3.. For example this will be the case along the top draining surface of an oedometer test after the application of the load. When it is necessary to specify a pore pressure boundary condition along a boundary whose current pore pressure distribution is not known then use IFX = 3.

Finally if these pore pressure fixities are used in an increment block consisting of more than one increment, the changes specified using IFX = 1 is spread over all the increments exactly in the same manner as the displacement fixity (see records K1). However IFX = 2 or 3 is enforced in the very first increment of the increment block and no further change in pore pressure takes place along these boundaries in the rest of the increments, within the incremental block.

IFX = 2 or 3

The incremental changes of excess pore-pressures along element sides are treated in exactly the same fashion as incremental displacements, using fixity code 1. Under certain circumstances the total pore-pressure needs to be fixed (for example along a drainage boundary or along a boundary where a known pressure head is applied). In older versions of CRISP, it was not possible to fix total pore pressure directly. This had to be done using IFX = 2 to fix an excess pore water pressure (see example below). It is, however, now possible to fix total pore pressure directly using IFX = 3.

If u represents the total pore pressure at which a particular node is to be maintained, and if u_o is the in situ (hydrostatic) value of the pore pressure at that node, then:

$$\text{total p.p.} = \text{in situ p.p.} + \text{excess p.p}$$

$$u = u_o + u_e$$

$$\text{i.e.} \quad u_e = u + u_o$$

You specify u_e in the input data with a fixity code 2, and not u. Alternatively, specify u with a fixity code 3. Both have identical effects and since what we are trying to fix is u, specifying u directly clearly saves the effort of calculation and reduces the risk of mistakes. IFX = 2 has been left in to ensure file compatibility with older versions of CRISP. The different methods are illustrated with an example: Consider an excavation of a trench in a saturated clay. The trench is excavated in layers of 2 metres; assuming unit weight of water is 10 kN/m^3 the in situ value of pore pressure at nodes 1, 2 and 3 are respectively 0, 20 and 40 kPa.

For nodes 2 and 5, after excavating the first layer total pore pressure = 0. Therefore the excess pore pressure = 0 - 20 = -20 kPa. It is similar after two layers have been excavated. At nodes 6 and 3, total pore pressure = 0 and the excess pore pressure along the base (at a depth of 4 metres) is given by 0 - 40 = -40 kPa. Using IFX = 2 often causes confusion to the users. The most common mistake is to incorrectly fix the excess pore pressure to 0 using a fixity code of 2. The fixity inputs are shown below using IFX = 2 and IFX = 3 for the above example (elements used are eight noded quadrilaterals of type 5).

Incr block 1

Element		N1	N2	IVAR	IFX	V1	V2
31	1	2	3	2	0.	-20.	

31	4	5	3	2	0.	-20.
31	5	2	3	2	-20.	-20.

Incr block 2

Element		N1	N2	IVAR	IFX	V1	V2
32	2	3	3	2	-20.*	-40.	
32	5	6	3	2	-20.*	-40.	
32	6	3	3	2	-40.	-40.	

* For IFX = 2, at nodes 2 and 5 the excess pore pressure values were fixed to -20 in increment block 1. However the fixities for these two nodes need to be specified again in increment block 2. This is because the excess pore-pressures along sides 2-3 and 5-6 of element [32] have to be specified due to excavation of layer 2 (i.e. element [32]).

Therefore the above example illustrates the correct way of dealing with excess pore pressure fixities. It is incorrect to specify 0 instead of -20. (*). Alternatively, IFX = 3 can be used, as shown below:

Incr block 1

Element		N1	N2	IVAR	IFX	V1	V2
31	1	2	3	3	0.	0.	
31	4	5	3	3	0.	0.	
31	5	2	3	3	0.	0.	

Incr block 2

Element		N1	N2	IVAR	IFX	V1	V2
32	2	3	3	3	0.	0.	
32	5	6	3	3	0.	0.	
32	6	3	3	3	0.	0.	

Therefore the above example illustrates the correct way of dealing with total pore pressure fixities. It has exactly the same effect as using excess pore pressure but as the reader can see, it is a simpler technique and is to be recommended. In both the above examples, the assumption is made that (free) surface water is available to dissipate the suction set up along the excavated face, due to removal of soil.

Earlier the adjective "absolute (abs)" was used with the meaning cumulative. The changes that take place in the excess pore pressure in an increment are known as the incremental excess pore pressures (Δu_e).

incremental excess pore pressure = Δu_e

The changes over a number of increments (say n) are summed to give the cumulative value which is referred to as the absolute excess pore pressure:

$$\text{absolute excess pore pressure } (u_e) = \sum_{i=1}^{n} \Delta u_e$$

Summarising, it could be said that when using fixity code 1 it is the Δu_e which is specified, when using fixity code 2 it is u_e which is specified and when using fixity code 3, it is u which is specified. In the case of the former the node undergoes a change in (excess) pore pressure equal to Δu_e. In the case of both the latter, the total pore pressure is held at u (corresponding to a excess pore pressure of u_e for fixity code 2) for the rest of the analysis. Here again the program calculates the necessary incremental value of excess pore pressure required to cause this, taking into consideration the current value of the pore pressure. In the foregoing description no mention was made of the current value of the pore pressure at the node. However the program takes into account the current value of the excess pore pressure, when a fixity code 1 is specified.

Any element side that has pore pressures fixed previously can be released (i.e. the pore pressures are free to change, and the element side becomes impermeable) by means of specifying a fixity code of 0.

EXPLANATIONS OF ERROR AND WARNING MESSAGES

C

This section contains a listing of every error message that may be output by either the CRISP Geometry Program or the Main Program.

C.1 How to Use This Chapter To Identify Errors

Errors from the CRISP programs come from one of two main sources:

- The Geometry Program
- The Main Program

Errors can be further identified as arising from within particular code subroutines within these two programs. This chapter is divided into two sections, Geometry Program errors and Main Program errors, each of which contain descriptions of the error messages listed by subroutine name in alphabetic order.

See Section 6.10.3 in the User Guide for further details about the Error Message Viewer

The description associated with each error message is also available in the form of on-line help. This may be invoked by double-clicking on any error message that appears in the Error Message Viewer dialogue box.

A third source of errors is also possible:

- DBOS errors

DBOS is the memory management system required by the finite element programs. Errors in the input data can sometimes cause crashes in the memory manager that are not trapped by the finite element program. The most common examples of this include unexpected divide by zero errors or errors related to the use of the single precision version when the double precision version is required.

DBOS errors are accompanied by a screen full of messages that describe where in the finite element program the error occurred. The most useful line of this information is the first line- this gives the name of the subroutine in which the error occurred. You should note this down and look up the related error messages in this chapter.

C.2 Rectifying Errors

A general rule of thumb that applies to many program errors as well as to DBOS errors is to re-run the analysis using smaller load and (if applicable) time steps, spread over more increments.

If you are getting errors at the in situ stage, check that you are not violating any basic rules (e.g. that you are not applying in situ pore pressure fixities) and that you have included the effect of any applied in situ loading in your specification of the in situ stresses at reference elevations. Also try re-calculating the in situ stresses using the in situ stress converter (although this won't help if you are applying non-vertical in situ loads).

If you are using a critical state soil model, try running the analysis using a simple elastic model just to be sure that you are applying boundary conditions correctly.

However, the first thing you should do is to note down the name of the subroutine in which the error occurred and follow the course of action recommended in the following sections.

C.3 Geometry Program Errors

C.3.1 CUREDG

(a) * ERROR *** EDGE CONTAINING NODES N1 N2 NOT FOUND**

Each element side is given a unique code IHASH (= 10000 * I1 + I2, I1 and I2 are the (Program) node numbers at either end; I1 < I2) when the program calculates the co-ordinates of the nodes along the side. This code is entered in the first column of array ITAB (LTAB, LDIM). When the user specifies the nodal co-ordinates along the curved element sides, as a first step the code for the element side is calculated and the above array is scanned to find it. If it is not found then the above message is printed. (probable user error in specifying the nodes N1 and N2).

(b) ELEMENT M1 DOES NOT CONTAIN NODES N1 N2

When the user specifies the co-ordinates of nodes along curved element sides he/she identifies the element side by the element number and the nodes at either end of the side. When either or both of these nodes (N1 and N2 are not found in element M1 this statement appears. (probable user error).

(c) PROGRAM TERMINATED IN ROUTINE CUREDG

If errors of category (a) and (b) are encountered then the program is stopped after the input data of nodal co-ordinates along curved sides have been read with message (c).

C.3.2 FFIN

(see error messages from Main Program)

C.3.3 MAIN2

INCREASE SIZE OF ARRAY BY N1 FOR GEOMETRY PROGRAM (ROUTINE MAIN2)

The array allocation G (10000) is insufficient for the geometry program. Array size of G must be increased by the specified amount in routine MAIN. Also reset LG to the new size of array G.

C.3.4 MAKENZ

****** ERROR - PROGRAM NODE N HAS INCONSISTENT SET OF D.O.F.**
ND1 ND2
D.O.F. LIST L(1), ..., L(6) (ROUTINE MAKENZ)

This means a possible program error in the absence of data errors. Make sure that the Geometry program input data are correct. Check the nodes associated with all elements in the mesh and also the element type numbers. If none of these then notify the author of the program with a copy of the input data and the output listing.

C.3.5 MIDPOR AND MIDSID

The following two statements appear when the estimated allocation for the additional nodes has been exceeded. The first statement is for the user node number and the second is for the program node numbers. These messages are unlikely to be printed, because the allocations for the additional nodes are more than adequate. If the messages do appear the entry for the corresponding element type (MXTYP in record C) in array NAD (in routine MAIN2) should be increased.

(a) INCREASE NO. OF ADDITIONAL NODES (ROUTINE MIDPOR or MIDSID)

(b) ***ERROR* MORE THAN NNE NODES IN MESH (ROUTINE MIDPOR or MIDSID).**

C.3.6 SEWZ

(See error messages from Main Program)

C.4 Main Program Errors

C.4.1 ATRANS

When a yield criterion number J (PR(6) in material properties - record D) other than in the range 1 to 4 is specified for the elastic perfectly-plastic models then the following message is printed.

******ERROR - INADMISSIBLE YIELD CRITERION NUMBER N (ROUTINE ATRANS)**

C.4.2 DCAM AND DMCAM

If the mean normal effective stress (p') becomes negative then to avoid numerical problems (due to negative stiffness) a p' value of one tenth of the p'_c value (size of the current yield locus) is used in the calculation of the D matrix (hence the element stiffness). Then the following message is printed and the analysis continued.

****** ELEMENT M INT. PT N PE IS NEGATIVE**
MAT ZONE NUMBER MZ (ROUTINE DCAM)

If this happens in the first increment then check the specified in situ stresses and ensure that nowhere is $p' \leq 0$.

C.4.3 DELP

If the Young's modulus (E) is calculated to be negative then the following message is printed and the analysis stopped.

****** ERROR - YOUNGS MODULUS IS NEGATIVE ----------**
IN ELEMENT M AT INT.POINT N FOR MAT ZONE NUMBER MZ (ROUTINE DELP).

Possible user error. Check the values of Eo, Yo, m (material properties 1, 5 and 11 respectively) so that the value of E is not negative anywhere in the mesh.

C.4.4 DETJCB

****** ERROR - JACOBIAN OF ELEMENT 1 INT. POINT 1 LE ZERO**
DJACB = ----- (ROUTINE DETJCB)
CODE TO INDICATE STAGE OF ANALYSIS = 4

CODE STAGE OF THE ANALYSIS

1. CALLED BY MSUB3/EQLOD/SELF CALCULATION OF IN SITU SELF WEIGHT LOADS

2. CALLED BY MSG/CHANGE/SELF LOADS DUE TO ELEMENT CHANGES
3. CALLED BY MSG/SEL1/SELF INCREMENTAL SELF WEIGHT LOADS
4. CALLED BY UPOUT/EQLOD/SELF WEIGHT LOADS FOR EQUILIBRIUM CHECK

When the determinant of the Jacobian matrix is negative this message is printed. If this message is printed well into the analysis (i.e. increment number > 1) and if you have used the option to update the geometry (i.e. by setting IUPD = 1 in record C1) then an element in the mesh has deformed into an unacceptable shape which makes mapping/transformation impossible. Try re-running the analysis using IUPD = 0 and smaller load steps.

C.4.5 DETMIN

(a) JACOBIAN IS NEGATIVE ----- OF ELEMENT M OF IP N (ROUTINE DETMIN)

The determinant of the Jacobian matrix of element M at integration point N is negative. this is followed by a code to indicate at what stage of the analysis this error occurred. This is mainly for the benefit of the programmer when testing the program. If the message is printed in the very first increment then check for the following errors in the input data to the geometry program. Otherwise see the explanation given for routine DETJCB.

1. Check that the nodes associated with element with 'program' number M are input in the correct sequence (anti-clockwise) in the geometry program input data.
2. Check that the node numbers of program element M are correct. i.e. ensure non are repeated by mistake.
3. Also check for nodes whose co-ordinates have not been specified. (i.e.. non-existent node numbers).
4. Also check the co-ordinates of the nodes associated with this element.

If none of the above then it is quite possible that the contents of array XYZ (the co-ordinates) may have got corrupted or have been over-written. Include WRITE statements and check the contents of the array XYZ. This can only happen if some program modifications have been carried out by the user. For other possibilities see below. The message also prints out the following lines to indicate at which stage of the program the error occurred. This is indicated by the stage number. This is illustrated by an example:

CODE TO INDICATE STAGE OF ANALYSIS - 1

1. CALLED BY INSIT/EQLIB/FORMB2 LOAD EQUIVALENT TO INSITU STRESSES
2. CALLED BY CHANGE/EQLIB/FORMB2 CALCULATION OF IMPLIED LOADINGS

3. CALLED BY FRONTZ/LSTIFF/FORMB2 CALCULATION OF STIFFNESS MATRIX
4. CALLED BY UPOUT/FORMB2 CALCULATION OF STRAINS OUTPUT STAGE

The following information indicates in which routine the array XYZ should be printed out.

Code	Routine	Statement
1	INSIT	Before the DO 150 J=1, NEL statement
2	MSG	At the beginning of the routine
3	FRONTZ	Before the DO 62 NE=1, NEL statement
4	UPOUT	At the beginning of the routine

This is to print all the contents of array XYZ once only.

The above error may occur when the XYZ array is written in double precision (for use with the double precision version of Main program) to the LINK file and inadvertently used with the single precision version of the Main program or vice versa.

C.4.6 DISTLD

(a) ** ERROR -- YOU HAVE PUT A PRESSURE LOAD ON ELEMENT M WHICH IS NOT PRESENT IN CURRENT MESH (ROUTINE DISTLD)**

Probable user error. The element M is not present at that particular stage of the analysis. Check the element number in the input data.

(a) ** ERROR - M ELEMENT DOES NOT CONTAIN NODES : N1 N2 (ROUTINE DISTLD)**

The nodes N1, N2 are used to identify the side, of element M which has a pressure loading. The message should not be printed since before entering this routine a check is carried out to find whether nodes N1 and N2 belong to element M. Therefore this message can only mean a program error.

C.4.7 DLIN

(see error message for routine DELP)

C.4.8 DSCHO

(see error message for routine DCAM)

C.4.9 EDGLD

(see error messages for subroutine DISTLD)

C.4.10 EQLBM

WARNING ** NO APPLIED LOADING - CHECK WHETHER ALL BOUNDARY CONDITIONS ARE DISPLACEMENTS (ROUTINE EQLBM)**

This is a precautionary message to draw the attention to the fact that no load of any significant magnitude has been applied in the current increment. Probably the analysis is displacement prescribed.

C.4.11 FFIN

All messages issued by this routine are self explanatory and are printed when data errors are encountered.

(a) * ERROR *** DATA RECORD --------- IS EMPTY**

Printed when a blank line is encountered in the input data.

(b) NA NUMBERS EXPECTED BUT NB ENCOUNTERED

Printed when the right number of values (NA) are not in the data record The data record appears to contain NB numbers.

(c) NUMBER IN POSITION L1 IS NOT AN INTEGER

When an INTEGER number is expected the number read is checked to see whether it is an INTEGER. If it is not an INTEGER then this message is printed. L1 is the position of the number in the data record. This error may probably due to an inadvertently placed '.' (dot).

(d) ILLEGAL SYNTAX FOR A NUMBER

Each input term is read character by character and the routine tries to interpret the number. When an inadmissible character is encountered as part of a number or a character is found in a inadmissible position (e.g. . (dot) after E) this message is printed. Other examples are. (dot) + - followed by blank.

(e) INCOMPLETE NUMBER NEAR END OF LINE

This message is printed when end of line is encountered after reading any one of the following characters . + -.

C.4.12 FIXX2

(a) ******ERROR : L TH FIX. ELEMENT M DOES NOT CONTAIN NODES : N1 N2 (ROUTINE FIXX2)

This is output when either or both of the nodes N1 N2 are not present in element M. This error is encountered when reading the list of fixities and it is the L'th fixity.

(b) INCREASE SIZE OF ARRAYS MF, TF AND DXYT TO MIN COMMON BLOCK FIX (ROUTINE FIXX2)

When more than 200 nodal fixities have been specified, this message is printed. The size of arrays MF, TF and DXYT have to be increased.

The message is printed when the number of nodal fixities exceeds 200 and it will also specify how much the array sizes have to be increased.

Note that even though the user specifies displacement and pore pressure fixities for element sides, within the program these are converted and stored as nodal fixities. The relevant arrays have been set up to cater for a maximum of 200 nodes with fixities.

The COMMON statement labelled FIX appears in the following routines and a change in size of the arrays means altering all these routines:

FIXX2	FIXX2B	FIXX3	FRONTZ	MAST2
MSG	MSUB2	MSUB3	RESTRN	SETUP
UPOUT				

The parameter MXFXT must be set to the new size of the arrays in routine MAXVAL.

C.4.13 FIXX2B

(See error message (b) for routine FIXX2)

C.4.14 FIXX3

(a) **** ERROR : FIXITY NF IN LIST.**
ELEMENT IL DOES NOT HAVE FACE WITH NODES N1 N2 N3 N4 (ROUTINE FIXX3)

This is output when the routine cannot find the element face with nodes N1 N2 N3 N4. This is the NF'th fixity in the list. Probable user error.

Make sure that the corner nodes have been input in the correct sequence (see record M in Appendix A).

If the message is printed for all the fixities specified by the user then check your co-ordinate system. It must be the right-handed system as shown in Figure A-1.

(b) (same as for routine FIXX2)

C.4.15 FOMFBM

****** ERROR - JACOBIAN OF ELEMENT M INT. POINT N LE ZERO (ROUTINE FOMFBM)**

Jacobian of element M at integration point N is negative. If this happens in the first increment check the node numbers and co-ordinates of element in the geometry input data.

C.4.16 FOMFBR

(see error message for routine FOMFBM)

C.4.17 FOMFSL

(see error message for routine FOMFBM)

C.4.18 FRONTZ

(a) PROBABLE SERIOUS ILL-CONDITIONING (ROUTINE FRONTZ)

This is followed by:

VAR NUMBER = NVAR NODE = N ELEMENT = M
CRIT = ---------- PIVOT = ----------

ELEMENT (PROG) NODES = list of nodes associated with element M. The contents of array NDEST.

This is followed by the contents of the front regions and the load vector when this error occurred. This is mainly for pinpointing the nature of the error. This is an indication of possible numerical problems, for example when a very stiff structure is lightly sprung to earth. If this analysis is that of a soil-structure interaction problem make sure that the stiffness parameters of the structure is not several orders higher than that for the soil. If possible reduce the stiffness of the structure (if the structure is modelled using the linear elastic model then reduce the E and G values). Otherwise use the double precision version of the Main program to analyse the problem.

(b) ERROR - ZERO PIVOT (ROUTINE FRONTZ) IN ELEMENT = M NODE = N VAR NO. = NVAR NPA = N2 NDIAG = N3

This is followed by the contents of the front region and the load vector when this error occurred. This is mainly for pinpointing the nature of the error.

This happens when the diagonal stiffness term of the unknown equation which is about to be eliminated is found to be equal to zero. The above message can be expected when the permeabilities have been incorrectly set to zero in the material property table in a consolidation analysis for element types 3, 5, 7 or 9 (especially if this happens in the very first increment). Also check that the time step has not been inadvertently set to zero in a consolidation analysis. (DTIME in record I).

However if this error is encountered in an increment other than the first then it would suggest that a pivot has become zero during Gaussian elimination. This is somewhat rare. Chances are that there are negative stiffness terms (can happen in Cam-clay models when p' becomes negative). When this happens re-run analysis with smaller load steps and the problem may disappear. Otherwise the results obtained are suspect.

C.4.19 INSIT

(a) *****ERROR - REFERENCE POINT NO. READ IS M EXPECTED IS N (ROUTINE INSIT)

This message appears when the in situ reference points are not input in ascending order or if there are gaps in the reference point numbering.

-No gaps are allowed in the in situ reference points. These reference points are not be confused with the nodes in the actual mesh. These reference points should be entered in ascending order in records G1. They also must be in the correct sequence and no gaps are allowed. Example 1, 2, 3, 4, 5. The sequence 1, 2, 4, 5 is not permitted because number 3 has been missed out. Otherwise check the y co-ordinates of the reference points. These should be entered either in the y co-ordinates increasing or decreasing sequence.

(b) *****ERROR - CHECK IN SITU REFERENCE POINT NUMBERS AND Y COORDINATES (ROUTINE INSIT)

If there are more than 10 errors of the type (a) then there is definitely something wrong with the in situ input data. This message is then issued and the program is stopped.

(c) WARNING ------ POINT OUTSIDE IN-SITU STRESS SPACE (ROUTINE INSIT)

This message is followed by the (program) element number and the integration point number. This happens when the in situ region (defined by a set of layers) does not cover the entire region of the primary mesh. Ensure that the in situ reference points span the entire mesh except for the part of the mesh removed at the beginning to simulate later construction.

1. If an attempt is made to divide by zero then it indicates a zero value for p' at some integration point.
2. If an attempt is made to calculate the logarithm of a negative or zero value then it means p'_c (the size of the yield locus) is zero at some integration point.

When this happens it means that the in situ stresses are incorrectly specified. Make sure that (i) the zero in situ stress option has not been specified (i.e. INSIT = 0 in record F). (ii) the in situ stresses specified are admissible. i.e. p' and p'c must be positive everywhere within the mesh. Note however that zero values of p' and p'c are permitted at the ground surface.

C.4.20 LCLSTR

(a) ****ERROR - CHECK NODE SEQUENCE FOR SLIP ELEMENT M (ROUTINE LCLSTR)

Slip element numbering must be anticlockwise with the first two vertex nodes being along one of the long edges. See Section 3.10 for further details.

(b) *****ERROR - CHECK NODE SEQUENCE AND NODE COORDINATES OF ELEMENT M (ROUTINE LCLSTR)

For one-dimensional elements check the nodes associated with element M and also the co-ordinates in the Geometry input data.

C.4.21 LODLST

INCREASE SIZE OF ARRAYS IN COMMON BLOCK PRSLD ALSO SET MXLD IN ROUTINE MAXVAL (ROUTINE LODLST)

The size of arrays LEDG, NDE1, NDE2 and PRESLD are set to 100. If more than 100 element sides are subjected to pressure loading in the input data then this message is printed.

The value of MXLD in routine MAXVAL must be set to the new (increased) size of these arrays. The above arrays occur in the following routines and they have to be changed:

EQLOD	LODINC	LODLST	MAST2	MSG
MSUB2	MSUB3	SETUP	UPOUT	

C.4.22 LSTFBM

******ERROR - ELEMENT M WITH MAT ZONE NUMBER MZ HAS INADMISSABLE MAT TYPE NUMBER MT (ROUTINE LSTFBM)**

The admissible material type number is 8. Probable user error.

C.4.23 LSTFBR

(See error message for routine LSTFBM)

C.4.24 LSTSLP

(See error message for routine LSTFBM)

C.4.25 LSTIFF

If an error message is generated in routine LSTIFF where an attempt is made to divide by zero, then in a consolidation analysis check that the 7th material property (PR(7,KM) which is the unit weight of water (γ_w) is not input as zero for material zones with element types 3, 5, 7 or 9 (especially if this happens in the very first increment).

C.4.26 MAKENZ

(see error message for routine MAKENZ in Section C.3.4)

C.4.27 MAST2

(a) ERROR ------LINK CODE MISMATCH LINK1 LINK2 (ROUTINE MAST2)

This message is printed when the link numbers specified in the geometry and main program input data are different. User error, specify the same link number in the main program input data.

(b) TO PROVIDE MINIMUM CORE TO SOLVE EQUATIONS INCREASE SIZE OF ARRAY G BY N1 IN MAST (ROUTINE MAST2)

N1 is the amount by which size of array G has to be increased in routine MAST. Also set LG equal to the new size of array G. Note that this only provides the minimum core and the equations are solved out-of-core.

C.4.28 MSG

(a) ** ERROR - TIME STEP SPECIFIED AS ZERO IN A CONSOLIDATION ANALYSIS. DTIME = ----- (ROUTINE MSG)**

In a consolidation analysis the time step has been specified as zero.

(b) NO OPTION TO CALCULATE NODAL LOADS FROM PRESSURE LOADING FOR 3-D PROBLEM (ROUTINE MSG)

At present there is no option to specify a pressure loading for 3-D problems where the equivalent nodal point loads are calculated to be within the program as available for the 2-D case. Therefore the user has to calculate the equivalent point loads by hand using the expressions given in Section 8.12 of the Technical Reference Guide and directly input these point loads.

(c) ERROR IN INCR BLOCK NUMBER NA NB (ROUTINE MSG)

The increment block numbers must be in sequence. The program has an internal counter and it expects the increment block number to be NB but in the data input it has the number NA (probable user error). If this happens in a restarted analysis then see Section 8.3 of the Technical Reference Guide for further explanation.

(d) ERROR IN INCR NO IN INPUT DATA = IA INCREMENT NO EXPECTED IS IB (ROUTINE MSG)

When reading the control parameters for the current increment block the first and last increments are read as INC1 and INC2. INC1 must be in sequence (if this is not the first increment block in the analysis, INC1 must have the value next to the last increment in the previous block) and if it is not equal to the counter within the program the above message is printed. The above message will also appear when INC2 < INC1.

(e) INCREASE SIZE OF ARRAYS, RINCC, DTM AND IOPT TO NH ALSO SET INCZ IN ROUTINE MSG

When the number of increments in a block exceeds the allocated 100 the above message is printed:

INCZ must be set equal to the actual number of increments (NH in this example) in addition to the array sizes being increased in routine MSG.

These changes should be carried out only if it is really necessary. Alternatively the number of increments can be split into a number of separate increment blocks with each having less than or equal to 100 increments.

C.4.29 MSUB1

(a) ERROR IN NO. OF INCREMENTS = N1 INCS = N2 INCF = N3 (ROUTINE MSUB1)

This message is printed when the increment at the finish of the analysis (N3) is less than the increment number at start of the analysis (N2).

(b) ***ERROR - STARTING INCREMENT NO. (INCS) = IA WHEN TOP/RESTART FACILITY IS NOT BEING USED (ISR) = IB (ROUTINE MSUB1)

When the stop/restart facility is not being used INCS must always be 1.

(c) ***ERROR - MAT ZONE NUMBER IN INPUT DATA IS MT1WHEREAS ZONE NO. EXPECTED IS MT2 (ROUTINE MSUB1)

No gaps are allowed in material zone numbers and these must be input in ascending order, i.e. material zone numbers in the input data (record D) must have consecutive numbers starting with 1.

(d) ****ERROR - THE SLOPE YOU HAVE SPECIFIED FOR THE HVORSLEV SURFACE ALONG CONSTANT VOLUME LINE (H) ------ IS GREATER THAN THE SLOPE OF THE CRITICAL STATE LINE ------ FOR MAT ZONE NUMBER MZ (ROUTINE MSUB1)

(Only for the Schofield soil model.)

The above message is self-explanatory. The slope of the Hvorslev surface (along a constant void ratio line) cannot be greater the slope of the critical state line.

(e) ***ERROR - AFTER ITZ ITERATIONS CANNOT FIND INTERSECTION POINT BETWEEN TENSILE CRACK REGION AND HVORSLEV REGION. CHECK YOUR PARAMETERS FOR MODEL NO. 6 - MAT ZONE NUMBER MZ (ROUTINE MSUB1).

(Only for the Schofield soil model.)

The demarcation point between the Hvorslev surface and the tensile crack region is calculated by iteration. If the point cannot be found after (say) 100 iterations then there is an error in the material properties.

C.4.30 MSUB2

(a) *** ERROR - INADMISSIBLE STOP/RESTART OPTION - ISR**

Only permissible values for the ISR are 0, 1 or 2. User error.

(b) *** ERROR IN STOP/RE-START FACILITY USING DISK FILE OPTION (ISR =1)**

THE RESULTS (OF LAST INCREMENT) STORED FROM PREVIOUS RUN IS OF INCREMENT INCA.

THE RESULTS EXPECTED IS THAT OF INCREMENT INCB.

(1) CHECK THAT THE CORRECT FILE WHICH STORES THE LAST INCREMENT FROM PREVIOUS RUN HAS BEEN SPECIFIED

(2) CHECK THE VALUE OF PARAMETER INC1 IN RECORD C1 OF MP INPUT DATA (ROUTINE MSUB2)

Again the message is self-explanatory. When using the option ISR = 1 the increment numbers are written along with the results. Check the file names and the increment numbers in the input data.

C.4.31 OUTBM

(see the error message for routine FOMFBM)

C.4.32 OUTBR

(see the error message for routine FOMFBM)

C.4.33 OUTSLP

(see the error message for routine FOMFBM)

C.4.34 REACT

INCREASE ARRAY SIZE OF R, NDENO AND NDIR TO NRZ IN ROUTINE REACT

When the number of reactions exceeds 500 this message is printed. Array size of R, NDENO and NDIR are increased and NCT is set equal to this new size (NRZ). These changes only need to be made in routine REACT.

C.4.35 SCOCAM

(see the error message for routine DCAM)

C.4.36 SELF2

(see the error message for routine DETJCB)

C.4.37 SFWZ

(a) NO ELEMENTS IN SOLUTION : (ROUTINE SFWZ)

When no elements are left in the mesh (possibly due to user error; elements removed incorrectly) this message appears.

(b) *****ERROR TOO MANY DEGREES OF FREEDOM IN FRONT (ROUTINE SFWZ)**

The allocation for maximum frontwidth is IFRZ which is et equal to 2000 in the geometry program. If the maximum frontwidth exceeds 2000 this message would have appeared in the geometry program. The allocation for the maximum frontwidth must be increased in the geometry program. No changes are necessary to the main program because IFRZ is passed on through the LINK file. It is unlikely that this message will be printed in the main program.

(b) PROGRAM ERROR - NO NODE ON END OF FRONT(ROUTINE SFWZ).

This message is unlikely to appear. When the FRONT shrinks due to variables being eliminated from the end of the FRONT the FRONT size is re-calculated. However when this FRONT size is not reduced to the correct value this message is printed.

(b) PROGRAM ERROR - LAST APPEARANCE NODE IS NOT IN FRONT (ROUTINE SFWZ)

This is also an unlikely error.

C.4.38 SHAPE

(a) UNKNOWN ELEMENT TYPE ** LT (ROUTINE SHAPE)**

When a request is made to calculate shape functions for inadmissible element types, this message is printed.

(b) ERROR * ELEMENT TYPE LT NOT IMPLEMENTED (ROUTINE SHAPE)**

This message is printed when request is made to calculate shape functions for element types which have not been implemented.

C.4.39 SHFNPP

(the messages are same as for routine SHAPE)

C.4.40 UPARAL

ALLOCATED STORE EXCEEDED; REQUIRED M ALLOCATED N (ROUTINE UPARAL)

Increase size of array G in routine MAST by the required amount. Also reset LG to the new size of array G.

C.4.41 UPOUT

WARNING ** THE NODAL CO-ORDINATES ARE UPDATED**

See Chapter 7 of the Technical Reference Guide, Interpreting Output.

C.4.42 VALFUN

***** ERROR UNKNOWN YIELD CRITERION TYPE KTN (ROUTINE VALFUN)**

The four available yield criterion for Elastic-perfectly plastic models are as follows:

1 - Von Mises

2 - Tresca

3 - Drucker-Prager (outscribing circle)

4 - Mohr-Coulomb

Any value for the criterion number other than 1 to 4 is inadmissible.

C.4.43 VARCAM

(a) PE = ZERO FOR ELEMENT = M INT PT = IP Q = ------ PCO = ------ (ROUTINE VARCAM)

p' has become zero during the course of the analysis.

(b) * ELEMENT M INT. POINT IP Q = ------ PE = ------ PY = ------ QT/(M*PE) = ------**

The above message is printed when the value η/M is more than 30. Attempting to calculate the exponential (to calculate the size of the yield locus) may result in an overflow problem. The value is restricted to 30. This may be because of a very small value of p'.

(c) ***ELEMENT M INT. PT IP PE IS NEGATIVE ------ MAT ZONE NUMBER M (ROUTINE VARCAM)

When too many of these errors are encountered then the results obtained are suspect. Re-run the analysis with smaller load steps and smaller time steps (in a consolidation analysis).

(d) Attempt to divide by zero (DBOS crash)

Probably p' = 0. Re-run the analysis with smaller load steps and smaller time steps (in a consolidation analysis).

(e) Attempt to calculate the logarithm of a negative or zero value (DBOS crash)

Probably p' has become zero during the course of the analysis. Re-run the analysis with smaller load steps and smaller time steps.

C.4.44 YIELD

***** TOO MANY INTEGRATION POINTS WITH DLAMDA NEGATIVE. INCREASE SIZE OF ARRAY VLDL AND ALSO RESET NVL IN ROUTINE UPARAL (ROUTINE YIELD)**

The allocation for array VLDL of 200 is more than adequate. The error may have been caused if the initial stress state lies outside the yield surface or on the yield surface. This can happen if using the Mohr-Coulomb or Drucker-Prager model with C = 0). Then the initial zero stress state would mean that the stress state lies on the tip of the yield surface. Re-run the analysis with a small value for C (say 1.).

FILE EXTENSIONS USED IN CRISP

D

This appendix provides a listing of all the filename extensions that appear in CRISP, as well as their meanings and an explanation of which program module creates them.

This is included for completeness and so that hard disk usage can be kept to a minimum by enabling the user to delete unneeded files.

D.1 File Extension Summary

The words in brackets refer to the program module that creates the extension. Pre is the Pre-Processor, Post is the Post-Processor and the others are self explanatory. Files followed by ***D** can be deleted once the analysis has been successfully run.

BAK **Ba**c**k**up created by Autosave of the .SCD file (Pre)

BMG File containing **b**ending **m**oment **g**raph information (Post)

CNT File containing **c**o**nt**our plot information (Post)

DGH File containing **d**uration **g**rap**h** information (Post)

FAM Parent/child (**fam**ily) information for parametric analysis (Pre)

GPD File created by CRISP containing all data for Geometry Program. **G**eometry **P**rogram **D**ATA (CRISP Front Squasher)

GPE **G**eometry **P**rogram **E**rror (CRISP Geometry Program) ***D**

GPO **G**eometry **P**rogram **O**utput (CRISP Geometry Program) ***D**

GPR Input file for Front Squasher. **G**eometry **P**rogram **R**ead (Analysis Launcher)

IGH File containing **i**nstance **g**rap**h** information (Post)

LDB File containing **l**inks for **d**ata**b**ase generation (Converter)

LIK **Li**n**k** file containing geometry input for Main Program (CRISP Geometry Program) This file is in binary format.

MAS **M**ain Program analysis **as**sessment (CRISP Main Program) ***D**

MDB **M**ain Program output **d**ata**b**ase (Converter)

MPD File created by CRISP containing input data for Main Program. **M**ain **P**rogram **D**ATA (CRISP Main Program)

MPD **M**ain **P**rogram **d**ata (Analysis Launcher)

MPE **M**ain **P**rogram **e**rror listing (CRISP Main Program) ***D**

MPO **M**ain **P**rogram ASCII text **o**utput file (CRISP Main Program) ***D**

MPW File containing **M**ain **P**rogram **w**arning listings (CRISP Main Program) ***D**

NRS **N**ew Stop-**R**e**s**tart file (CRISP Main Program)

NRD **N**ew Stop-**R**estart file for **d**ouble precision version of Main Program (CRISP Main Program)

ORS **O**ld Stop-**R**e**s**tart file (Analysis Launcher)

ORD **O**ld Stop-**R**estart file for **d**ouble precision version of Main Program (Analysis Launcher)

OVL File containing displacement plot **ov**er**l**ays (Post)

PPS Text file containing Main Program results for **P**ost-**P**rocessing (**S**AGE), used by the Converter (CRISP Main Program) ***D**

SBF **S**AGE **b**atch **f**ile contains a list of all analyses and their path statements to be run in the current session (Pre)

SCD File containing all Pre-Processor data (**CRISP d**ata) (Pre)

SCP File containing all **CRISP P**ost-Processing data. This is a copy of the Pre-Processor data for use in the Post-Processor (Pre)

SGR **S**A**G**E **R**eport Generator data file (Pre and Post) ***D**

SIE **S**age **I**mport **E**rror file (Pre) ***D**

SIL Copy of imported MPD file for error checking- **S**AGE **I**mport **L**og (Pre) ***D**

SMY Cam clay **s**u**m**mar**y** (CRISP Main Program)

SQE Front **Sq**uasher **E**rror (CRISP Front Squasher) ***D**

SQO Front **Sq**uasher **O**utput (CRISP Front Squasher) ***D**

STP File containing **st**atus **p**lots information (Post)

D.2 File Creation in the Pre-Processor

When a file is created in the Pre-Processor and then saved, two files are made. The **.SCD** which contains all the data from the Pre-Processor and the **.FAM** which has details of the family tree.

If Autosave is on, then the contents of the **.SCD** file will be saved in a **.BAK** file whenever the Autosave operates.

When the CRISP files are created, three new files are also generated: **.GPR**, the data file for the Geometry Program; **.MPD**, the data file for the Main Program and **.SCP**, a copy of the **.SCD** file, used to draw the mesh, etc. in the Post-Processor.

On running the analysis, CRISP first creates a **.SBF** file. This contains directory information about the file to be analysed.

If the data file is an imported CRISP 94 analysis, the **.MPD** file is copied to the **.SIL** file for error checking. If errors are found, they are reported in the **.SIE** file.

D.3 File Creation in the Geometry Program

The **.GPR** file is used as the input for the Front Squasher Program. This renumbers the elements in the mesh for optimum efficiency and generates the **.GPD** and **.SQO** files. The **.SQO** file contains essentially the same information as the **.GPO** file and is only used for trouble shooting if a crash occurs.

Once this has been done, the Geometry Program is run. This generates three new files: the **.GPO** file, contains the Geometry Program output; the **.LIK** file which contains all the geometrical information in a binary format for the use of the Main Program and in Post-Processing.

D.4 File Creation in the Main Program

The Main Program requires two input files in order to run. The **.MPD**, and **.LIK** files. The **.LIK** contains the geometry information for the analysis and the **.MPD**, all other data (for example, in situ stresses, output parameters, increment block information etc.). The Main Program creates several new files. The **.MPO** file contains text output of the results, the size of this file can be reduced by selecting less data output in the Output Options dialogue box in CRISP. The **.NRS** file has the output data for increments chosen for a Restart analysis (the **.NRD** file is the same but for the double precision version of the Main Program), this will be empty unless a Stop-Restart analysis has been run. The **.ORS** file is a copy of the results from the last increment in a **.NRS** file. This is used as the start point of a new Restart analysis (again, the **.ORD** file is the same thing for a double precision analysis). The **.MPE** and **.MPW** files are the Main Program error and warning files respectively and contain a list of the error messages and warning messages created by the Main Program. The **.MPE** will be empty if the Main Program has run successfully, similarly the **.MPW** will be empty if the Main Program gave no warnings. The **.MAS** file contains an assessment of the analysis and is shown at the end of a CRISP run as the Analysis Assessment in the Analysis Docker. The **.SMY** file shows the cam clay summary and again, this can be accessed through the Analysis Docker. The **.PPS** file contains the all the output data for each increment selected for Post-Processing output in CRISP.

D.5 File Creation in the Converter

The converter only creates two new files, the **.LDB** and the **.MDB**. The **.LDB** file contains the links for all the data output, in order to create the **.MDB** file. The **.MDB** file contains all the output from the analysis reformatted into a MS Access document.

D.6 File Creation in the Post-Processor

The Post-Processor only generates data for the different graph types that are created. **.BMG** contains bending moment graph information **.CNT** contains contour plot information, **.DGH** contains duration graphs information, **.IGH** contains instance graphs information, **.OVL** contains displacement plot information and **.STP** contains status plots information.