

4. USER'S GUIDE TO INPUT

4.1 Introduction (why more than one program?)

Most finite element analysis packages consist of one computer program. In order to perform an analysis using the CRISP package, however, it is necessary to submit at least two computer jobs which run two completely separate computer programs: the "Geometry Program" and the "Main Program" (see system description - Chapter 2). Although this process may take slightly longer than with the more conventional programs, CRISP has been consciously designed in this way to help the program user avoid some of the more common pitfalls in finite element analysis. The input data which the user must supply the program can be divided into the following categories.

- (1) information describing the finite element mesh, i.e. the co-ordinates of nodal points associated with each finite element,
- (2) material properties (and perhaps in situ stresses) associated with each finite element
- (3) boundary conditions to the analysis (i.e. imposed displacements and loads).

Experience shows that mistakes in the specification of the finite element mesh are often made by program users. These mistakes sometimes result in a mesh which is valid so far as the program is concerned but is simply not the mesh which the user intended. For this reason the CRISP program writes the information necessary to produce a plot of the finite element mesh together with element and node numbers to a Plot Data (P.D.) file. A separate program is then used to produce a plot of the mesh. This allows the program user to detect any errors in the geometric input data before embarking on a full analysis.

4.2 User's guide to Input (Geometry Program)

Record A

The title is usually set by the user to be descriptive of the subject of the finite element analysis. The title appears on the Mesh Plotting Program's plot of the finite element mesh as well as near the start of printed output from the Geometry Program. If different meshes are used to tackle the same problem then the titles should be different. e.g.

FOOTING ANALYSIS - MESH 1 - 100 LST ELEMENTS

and

FOOTING ANALYSIS - MESH 2 - 200 LST ELEMENTS

Record B

The Geometry program stores basic information describing the finite element mesh on a computer disk file (the "link" file) which is subsequently read by the Main Program (see system description - Section 2). A user of CRISP will often set up several (different) finite element meshes and run the Main program several times for each of these meshes. In order to ensure that a particular Main program 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 the user specifies.

Record CElement Types (MXTYP)

If different element types are present in the same mesh then MXTYP is set to the element type with the most number of nodes. Although it is possible to include more than one type of finite element in a mesh, normally all elements will be of the same type. The element type is defined by MXTYP which at present can take one of the eight values associated with the elements shown in the fig. 4.1:

<u>MXTYP</u>	<u>element</u>
1	Bar
2	LST
3	LST (Consolidation)
4	LSQ
5	LSQ (Consolidation)
6	CuST
7	CuST (Consolidation)
8	LSB
9	LSB (Consolidation)
12	Beam
13	Slip

If element types 2 and 4 are mixed in a finite element mesh, the 8-noded quadrilateral element has more nodes and hence MXTYP is set to 4.

Mixing element types

The element types that can be mixed in any mesh or analysis are listed under three different groups.

<u>Group</u>	<u>Element types</u>
A	1, 2, 3, 4, 12, 13
B	6, 7
C	8, 9

All element types within a particular group can be present in the same mesh.

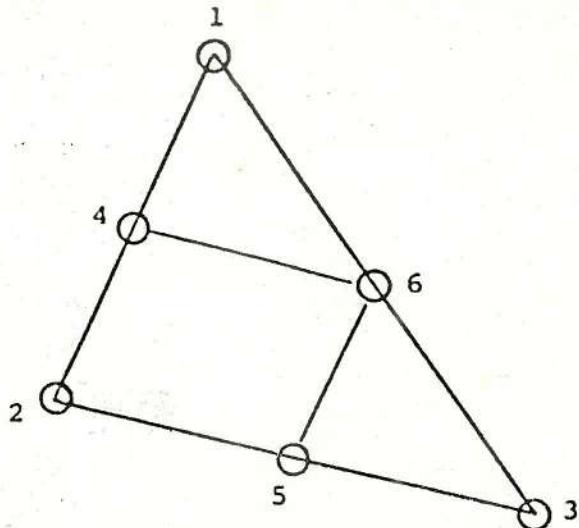
Element types from one group cannot be mixed with element types from a different group. For example one cannot mix element type 6 - CuST (group B) with element type 12 - Beam (group A).

Using bar element as a prop

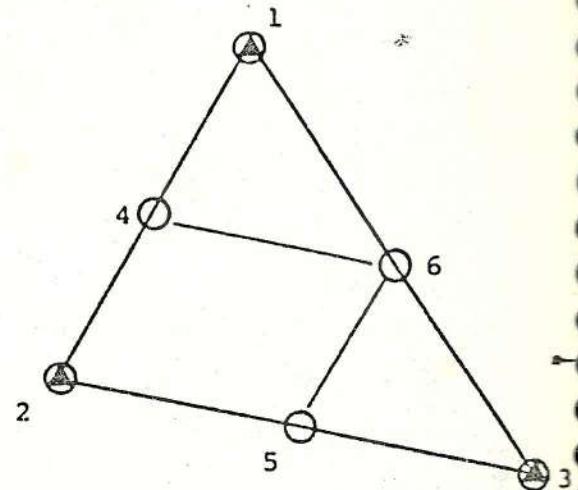
Bar element can be used to simulate a prop or a strut. However because of the presence of the mid-side node it is necessary to fix both the mid-side node and the end node (using NFXB in record I and record N of the main program) for this. If the mid-side node is not fixed it may result in a zero pivot error (see appendix E, routine FRONTZ).

○ u, v - displacements unknown
 ▲ p - pore-pressure unknown

4.6

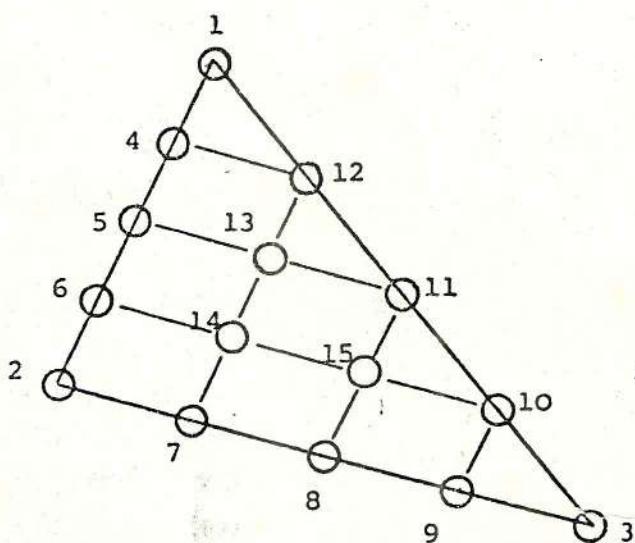


(a) LST (element type 2)
6 nodes, 12 d.o.f.



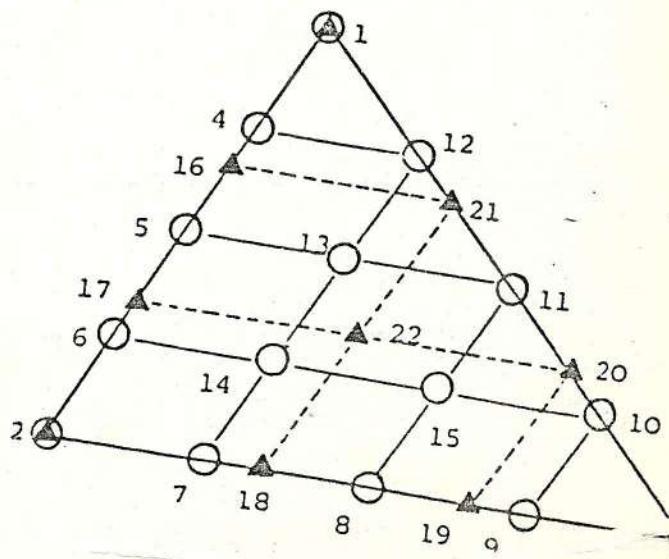
(b) LST (consolidation)
6 nodes, 15 d.o.f.
(element type 3)

(for drained/undrained analysis
and zones)



(c) CuST (element type 6)
15 nodes, 30 d.o.f.

(for drained/undrained analysis
and zones)

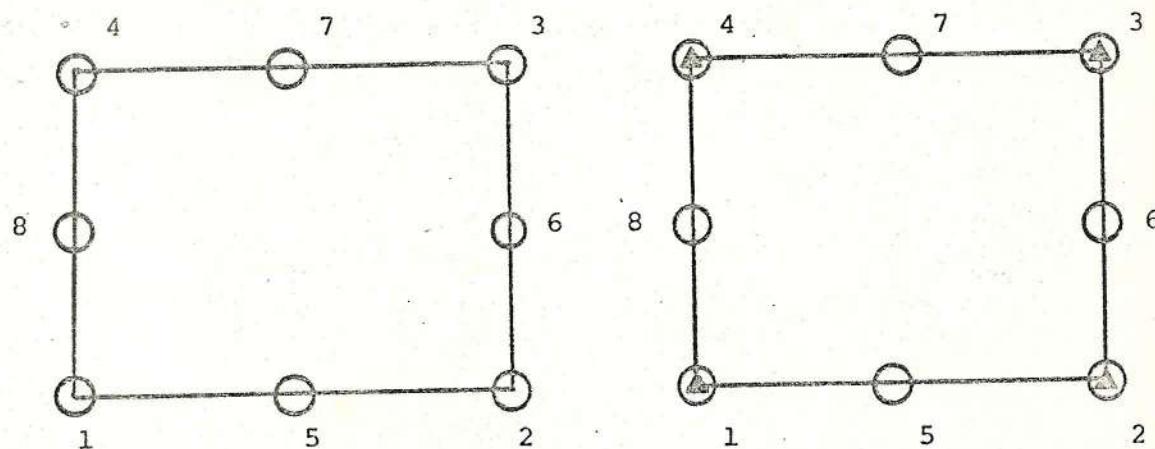


(d) CuST (consolidation)
(element type 7)
22 nodes, 40 d.o.f.

Fig. 4.1 Different types of elements

○ u, v, w - displacements unknown

▲ p - pore pressures unknown



(e) LSQ (element type 4)

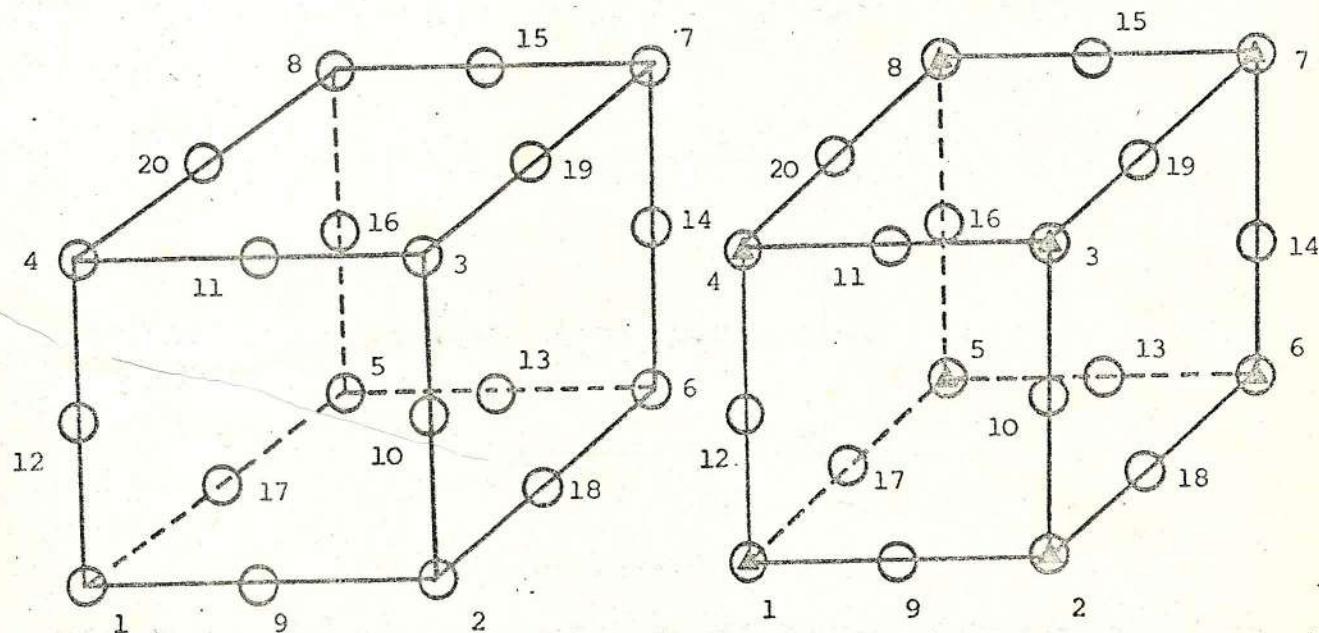
8 nodes, 16 d.o.f.

(f) LSQ (element type 5)

(consolidation)

8 nodes, 20 d.o.f.

(for drained/undrained analysis
and zones)



(g) LSB (element type 8)

20 nodes, 60 d.o.f.

(for drained/undrained analysis
and zones)

(h) LSB (element type 9)

(consolidation)

20 nodes, 68 d.o.f.

Fig. 4.1(b) Different types of elements

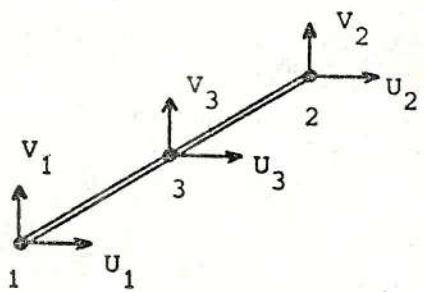
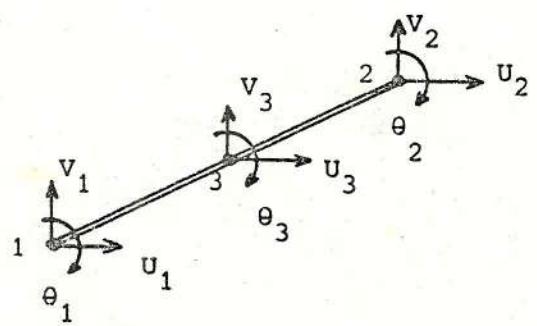
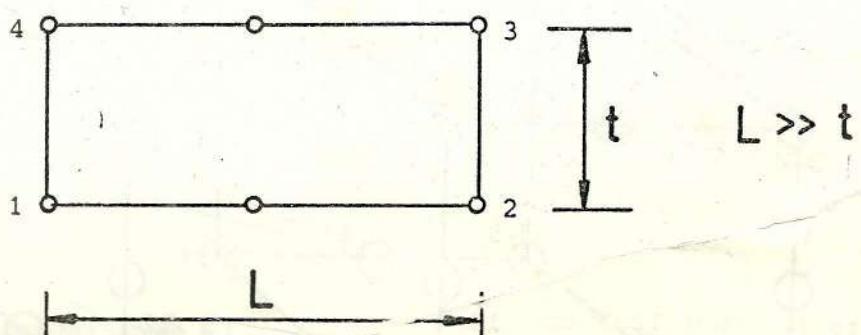
(a) Bar element ($LT = 1$)(b) Beam element ($LT = 12$)(c) Interface element ($LT = 13$)

Fig. 4.1(c) Element Types

The variation of displacements (and consequently strains) and where appropriate pore pressures are summarised in the table below.

<u>MXTYP</u>	<u>Element name</u>	<u>Displacement</u>	<u>Strain</u>	<u>Excess Pore Pressure</u>
1	Linear strain bar	Quadratic	Linear	N/A
2	Linear strain triangle (LST)	Quadratic	Linear	N/A
3	LST with linearly varying excess pore pressures	Quadratic	Linear	Linear
4	Linear Strain Quadri-lateral (LSQ)	Quadratic	Linear	N/A
5	LSQ with linearly varying excess pore pressures	Quadratic	Linear	Linear
6	Cubic strain triangle (CUST)	Quartic	Cubic	N/A
7	CUST with cubic variation of excess pore pressures	Quartic	Cubic	Cubic
8	Linear Strain Brick (LSB) element	Quadratic	Linear	N/A

9	LSB with linearly varying excess pore pressures	Quadratic	Linear	Linear
12	Linear Strain Beam element	Quadratic	Linear	N/A
13	Linear Strain slip element	Quadratic	Linear	N/A

All the elements are basically standard displacement finite elements which are described in most texts on the finite element method (e.g. Zienkiewicz, 1977).

Note that NVTX refers to the number of vertex (i.e. corner) nodes in the finite element mesh. The Program automatically generates node numbers and co-ordinates for any nodes lying on element sides or within elements.

Although CRISP allows the user complete freedom in the choice of element type, the following recommendations should lead to the selection of an appropriate element type:

(i) Plane strain analysis

For drained or undrained analysis use type 2 (Linear Strain Triangle) or 4 and for consolidation analysis use element type 3 or 5.

(ii) Axisymmetric analysis

For drained analysis or consolidation analysis where collapse is not expected then element types 2 to 5 will probably be adequate (i.e. the same as (i) above). For undrained analysis or situation where collapse is expected then element types 6 and 7 are recommended. Recent 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).

How many elements?

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

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

If possible an attempt should be made to analyse the problem with 2 different meshes one finer than the other (ie with more elements or using a higher order element). The comparison of these 2 analyses would indicate whether sufficient number of d.o.f.s are being used in the analysis. In theory the finer the mesh becomes the more closer the results become to the true solution (if such a solution exists).

Avoid triangular elements which have aspect ratios greater than about 3 (also avoid elements with $1/3 > \text{height/base} > 3$). For quadrilateral elements (types 4, 5) the aspect ratios can be much higher (say about 10).

Mixing different element types

As mentioned above the possibility exists of mixing different element types in a CRISP analysis. The only element types which it is recommended that could be mixed in the current program version are element type 2 (and/or 4) with element type 3 (and/or 5) and element type 6 with element type 7. This could be done in a consolidation analysis where part of the continuum is expected to behave in a completely drained or completely undrained mode in comparison to the rest. Also see section 3 of appendix I.

Record D

The parameters NUMAX and MUMAX need to be specified as non-zeroes only if there are gaps in the vertex node numbering and element numbering respectively. This information is necessary to allocate sizes to arrays which store the node numbers and element numbers. Rather than arbitrarily allocating sizes to these arrays which imposes a limit on the no. of vertex nodes and elements this procedure is preferred.

Record E

The normal option is to set all these values to zero. These flags need to be set only when the program is being tested (left in for the benefit of users/programmers who may want to change the program e.g. to incorporate a new element type). This feature helps to ensure that the changes made to the program are correct.

This debugging option may also be used to track down any errors in the specification of the finite element mesh (But this is best dealt by a data checking program).

Record F

The program calculates the co-ordinates of nodes along sides and element interiors by linear interpolation assuming that the elements are straight edged. However in some analyses (e.g. circular tunnel, buried pipe) it is more appropriate for the element sides to be curved to accurately model the physical problem. The program does not have the facility to calculate coordinates of nodes assuming that the element sides are curved. This feature is included so that (see records K and L) the user can directly specify the co-ordinates of the intermediate nodes along the (few) curved sides.

If all the element sides in the mesh are curved the user may consider writing a small program to generate the nodal co-ordinates automatically and input as described below (records K and L).

Records K and L are then used to specify the coordinates of intermediate nodes, for each element side. It should be noted that 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 (do not have pore pressure nodes along side) NSPZ and NPCUR must be set to zero. Records L are then omitted from input.

Records G and JElement and nodal numbering

The program user must assign each element and each vertex node in the finite element mesh unique (integer) numbers in the following ranges:

1 ≤ node number ≤ 750 (in general)

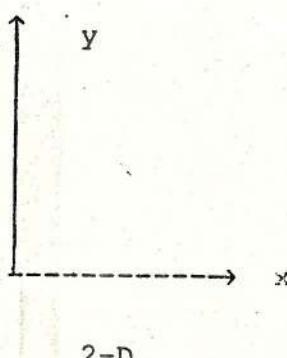
1 ≤ element number ≤ MUMAX (user specified; if equal to zero then NEL)

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.

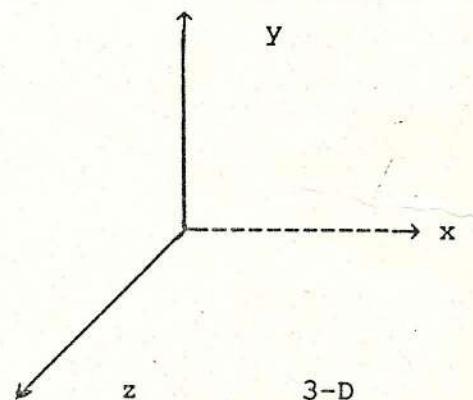
Co-ordinate System

It is recommended that the user adopts a co-ordinate system with the y axis pointing upwards:

i.e.



2-D



3-D

Note the the x axis points to the right - if the x axis points to the left then the program will calculate element areas and stiffness as negative quantities. (This recommendation is linked to the program's expectation that element node numbers are listed in record J 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 in records G1, G3 and L in the Main Program data).

The user 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:

- (1) specification of material self weight loads (excavation, construction and gravity increase - records D and I, in the input to Main Program)
- (2) elastic properties varying linearly with depth (record D)
- (3) axisymmetric analysis

When the axisymmetric analysis option is selected (record C1) it is assumed that the y axis is the axis of symmetry (i.e. the x axis is the radial direction).

In 3-D 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 the node is specified only for 3-D analysis. Interchanging or rotation of this set of axes is not allowed for the 3-D analysis.

Units

The user can choose any appropriate length for describing the co-ordinates of nodal points. It is important, however, that the units chosen to describe material properties, stresses and loads in the Program are consistent. In a drained or undrained analysis the user 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 the user has 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 meters (m) and kilo-Newtons (kN) respectively then stresses and elastic moduli must be in kN/m² and unit weights must be in kN/m³ (see Table 4.1).

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

	1	2	3	4
length	m	mm	mm	ft
force	kN	N	mN*	lbf
time	sec	sec	sec	hr
pressure, stress	kN/m ²	kN/m ²	mN/mm ² \$	lbf/ft ²
unit weight	kN/m ³	kN/m ³	mN/mm ³	lbf/ft ³
pmeability	m/s	mm/s	mm/s	ft/hr

Table 4.1 Consistent set of units

* milli Newtons

\$ mN/mm² and kPa are equal in magnitude

Material Zone Numbers (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). Thus 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 and zone 3 may be modelled by linear elastic properties. (note: "gaps" in the numbers of zones are not allowed.)

Number of vertex nodes (NV)

The number of vertex nodes (NV) after the element number, element type number and material zone number is governed by the element type number MXTYP in record C. For example if Linear Strain Triangles (type 2) and the 8-noded quadrilateral (type 4) are *mixed* then all records J must have 4 node numbers. For the triangular elements a 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 4.1).

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 any analysis of medium to large sized problems may prove to be prohibitively expensive. 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) becoming 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.

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 the user has 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. The frontal optimising programs 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 = $C \times f^2 \times NDF$

The amount of (backing) store required to store eliminated coefficients
 $\approx (f + 4) \times NDF$

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.

Record K and L

The coordinates 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 linear strain triangles of type 3 there are no excess pore pressure nodes along the element sides, therefore only record K needs to be specified. For element type 7 both types of records (K and L) have to be specified.

The element number is followed by nodes N1 and N2 (which are at either end of the side) to identify the element side. Then the nodal coordinates of intermediate nodes along the element side are given in sequence from node N1 to N2 (note that the coordinates of nodes N1 and N2 are not specified).

4.3 User's Guide to Input (Main Program)

Record A

The title specified by the user is printed out near the start of the program output. It is advisable to include in the title some identifying number of character which is changed for each new analysis (or re-analysis using different input parameters).

e.g.

FOOTING ANALYSIS - RUN 16 - ELASTIC PROPERTIES

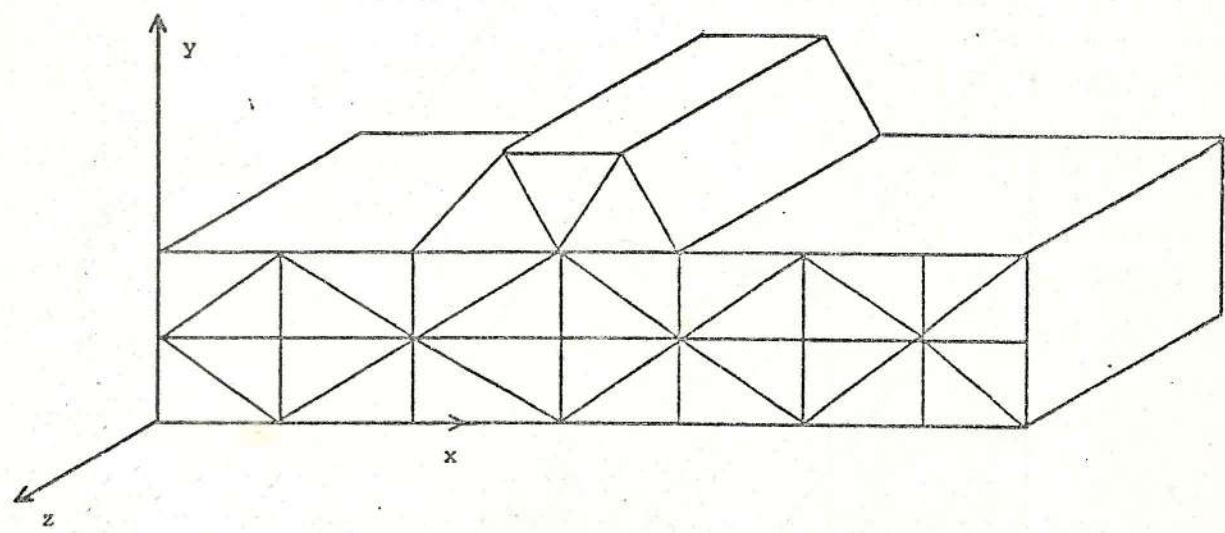
Note that comment lines cannot be included before the TITLE record. However these can be included anywhere in the data. Comment lines are identified by C in the first column.

Record B

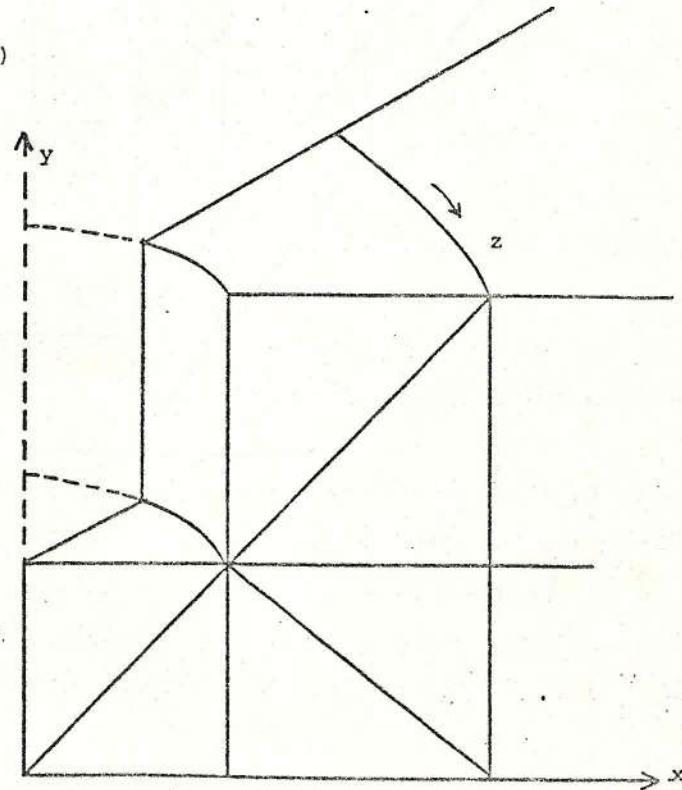
The link number must be the same as that specified in the geometry Program input data for the appropriate finite element mesh. (See Record B - input to Geometry Program).

Record C1

NPLAX = 0 (Plane strain)



NPLAX = 1 (Axisymmetry)

Figure 4.2

$\gamma_{xz} = \gamma_{yz} = 0$
 x is the radial direction
 z is the circumferential direction

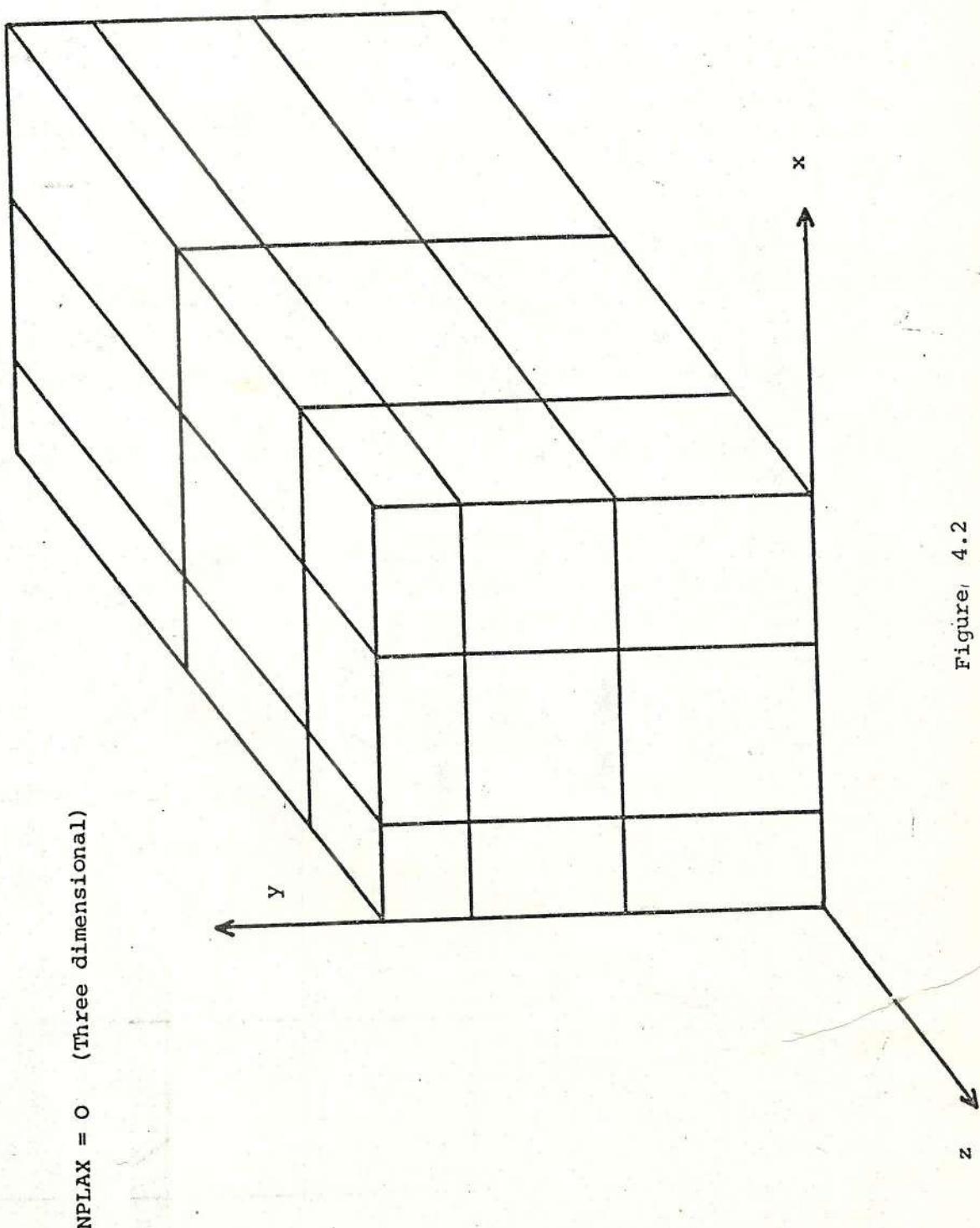


Figure 4.2

Record C1

NMAT must be equal to the number of different material zones specified in the geometry program.

INC2 \geq INC1

If INC1 > 1 then this analysis is a continuation of a previous analysis (see Section 4.4) and records E to H3 are omitted.

IPRIM

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. IPRIM is the number of finite elements that must be removed to form the primary finite element 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 represent a construction event cannot be assigned material type numbers 3 (MCC), 4 (CC), 6 (SCHO)). Therefore these elements will have to be modelled as elastic (type numbers 1 or 2) or elastic perfectly plastic (type number - 5). If using model 5 with the following yield criterions (a) Mohr-Coulomb (b) Drucker-Prager then specify a small value for C (if C is zero). 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 appendix E, routine YIELD).

Any element for which a stress history exists has to be assigned in situ stresses and these have to be present in the beginning of the first increment. Note that the elements listed in record E are not specified any in situ stresses. The in situ stress region need not cover (or span) this part of the mesh.

IUPD

IUPD = 0: this corresponds to the normal assumption that is 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: when this option is used the nodal co-ordinates are 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). 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 (i.e. IUPD = 0). Collapse loads can then be compared (and should correspond) with those obtained from a classical theory of plasticity approach.

ICOR

It is recommended that the user sets ICOR = 1 when elastic perfectly plastic models of behaviour 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. 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.

The parameter ICOR has no effect on Cam-clay 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 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.

Let us consider an analysis which is split into two separate runs and carried out using either magnetic tapes or private disks (ie using the stop-restart facility). 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:

		NPLAX NMAT ..	NOIB	INC1	INC2	IPRIM	IUPD	ICOR
run1	record C1 :	2	1	20	.	.	.
run2	record C1 :	2	21	40	.	.	.
			IBNO	INCA	INCB		
run 1	record I : increment block 1		1	1	10		
run 1	record I : increment block 2		2	11	20		
run 2	record I : increment block 1		1	21	30		
run 2	record I : increment block 2		2	31	40		

Record C2

This record allows the user to reduce the amount of lineprinter output produced by the Main Program. The parameters INSOP, IBC, IRAC operate independently. All other parameters operate in conjunction with parameter 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).

For example if the user requires the general stresses to be printed at all integration points for elements in a particular range then he/she should specify NELOS to the first element in that range and NELOF to the last element in that range. In addition he/she should specify $d = 2$ in the output code $abcde$ for IOUT (eg. 11121). Specifying zero values for both means no information will be printed for that category of output irrespective of the option set by IOUT. For example if both NMOS and NMOF are set to zero then even if option 2 is used for the nodal displacement ($e = 2$, eg. 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* (symbol is \wedge or \cap).

Records C3 and C4

In order to use the CRISP plotting program (a post processor)¹ to produce line-printer displays and plots of the results at selected stages of the analysis, the results at the end of selected increments are written to a disk file. The number of increments that can be written to this disk file is limited to 10. NINCP \leq 10.

Record C3 specifies the total number of increments (NINCP) to be written to the disk file and record C4 gives the list of increment numbers. These increment numbers must be specified in 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

Record C3 NINCP
 4

Record C4 INCLST(1) INCLST(4)
 10 20 80 100

¹ See the guide to the CRISP plotting program for more details

Record D

The parameters shown in the material properties table in the input specification have the meanings shown below. With a few possible exceptions (mentioned later) all the parameters should be regarded as being effective stress properties i.e. they either relate changes in strain to changes in effective stresses or they describe the soils strength in terms of the effective stresses that are acting in the soil skeleton.

Anisotropic elastic properties

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

$$\epsilon_x = \frac{1}{E_h} \sigma_x - \frac{v_{vh}}{E_v} \sigma_y - \frac{v_{hh}}{E_h} \sigma_z$$

$$\epsilon_y = -\frac{v_{hv}}{E_h} \sigma_x + \frac{1}{E_v} \sigma_y - \frac{v_{hv}}{E_h} \sigma_z$$

$$\epsilon_z = -\frac{v_{hh}}{E_h} \sigma_x - \frac{v_{vh}}{E_v} \sigma_y + \frac{1}{E_h} \sigma_z$$

$$\gamma_{xy} = \frac{1}{G_{hv}} \tau_{xy}$$

Note that suffixes h (for horizontal) and v (for vertical) have been adopted here to clarify the type of anisotropic properties which the program 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 v_{hh}) and another set relating to the vertical direction (v or y) and the coupling between horizontal and vertical directions (E_v , v_{hh} , v_{hv} , G_{hv}). The significance of these properties can be deduced from the above equations but the following may make the meanings clearer:

an increase in vertical stress leads to an increase in vertical strain $\Delta\sigma_y/E_v$ and a tensile strain $(v_{vh}/E_v)\Delta\sigma_y$ (in the absence of any changes in horizontal stresses). Hence v_{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 v_{hv} .

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

$$\frac{v_{hv}}{---} = \frac{v_{vh}}{---}$$

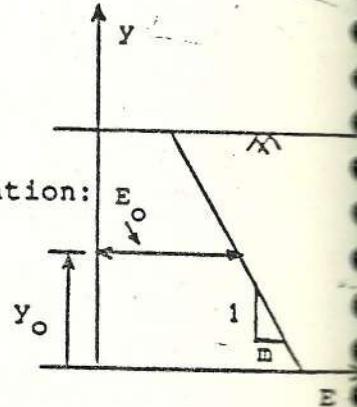
$$E_h \qquad E_v$$

Elastic, linear variation with depth

The elastic Youngs modulus at a depth y is given by the equation:

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

However v' is assumed to be a constant.



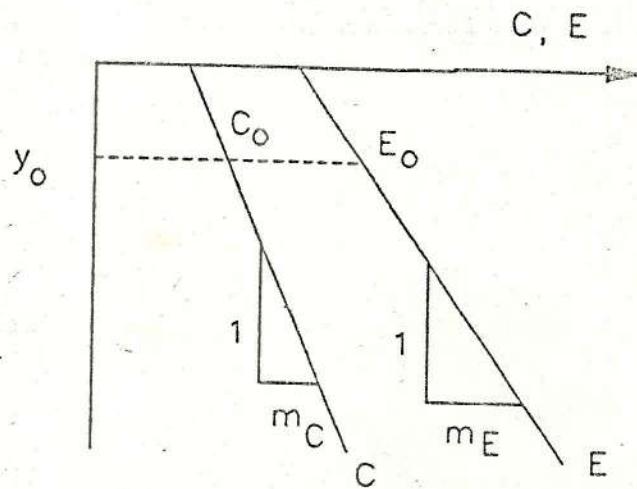
Critical State Parameters

λ , κ and M have the normal meanings (see Schofield & Wroth, 1968; Atkinson and Bransby, 1978). e_{cs} is the value of voids ratio on the critical state line for $p' = 1$ (using units of pressure consistent with other stresses input to the program). In the terminology of Critical State Soil Mechanics: $e_{cs} = \Gamma - 1$ The yield loci for Cam Clay and modified Cam Clay are shown on the following pages (Figure 4.3).

Elastic perfectly plastic models (model number 5)

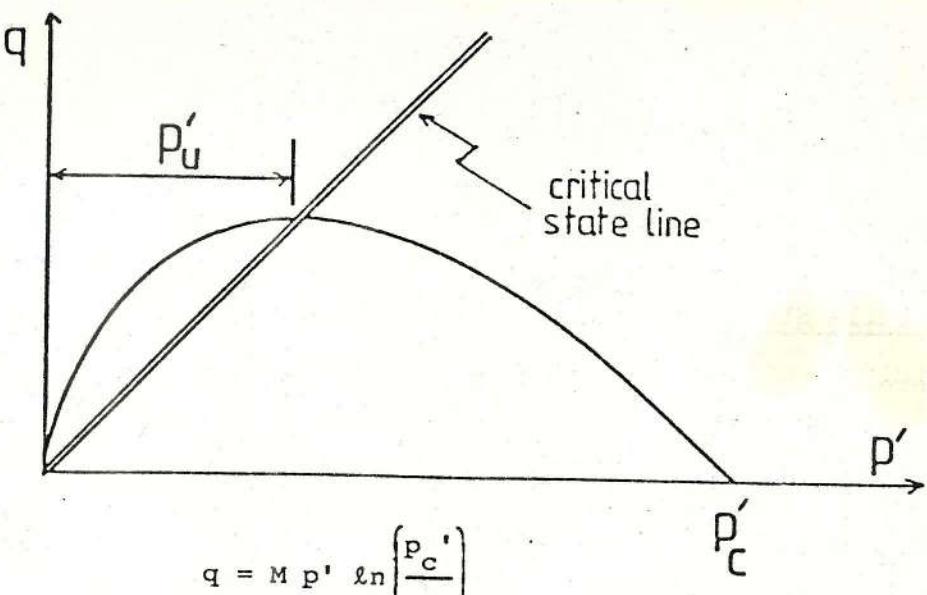
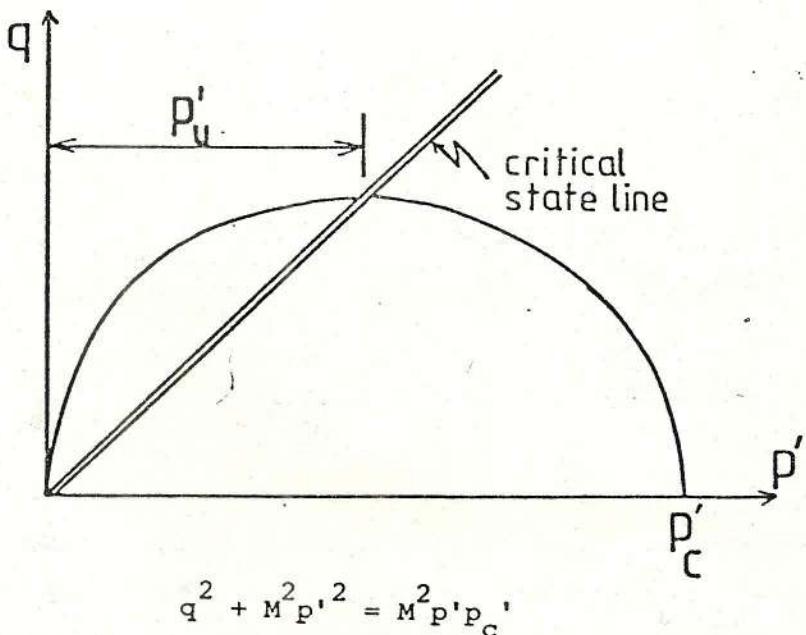
The parameters E_0 , C_0 , m_C , m_E allow for the linear variation of both the Young's modulus and the shear strength with depth (as shown in the figure below).

By specifying zero values for m_E and m_C one could specify constant values for E and C for the entire zone.



m_C - rate of increase of shear strength
with depth

m_E - rate of increase of Young's modulus
with depth

The Cam-Clay Yield LocusFigure 4.3 The Modified Cam-Clay Yield Locus

note: These yield loci are generalised in three dimensional principal stress space by rotating the loci about the hydrostatic (p') axis.

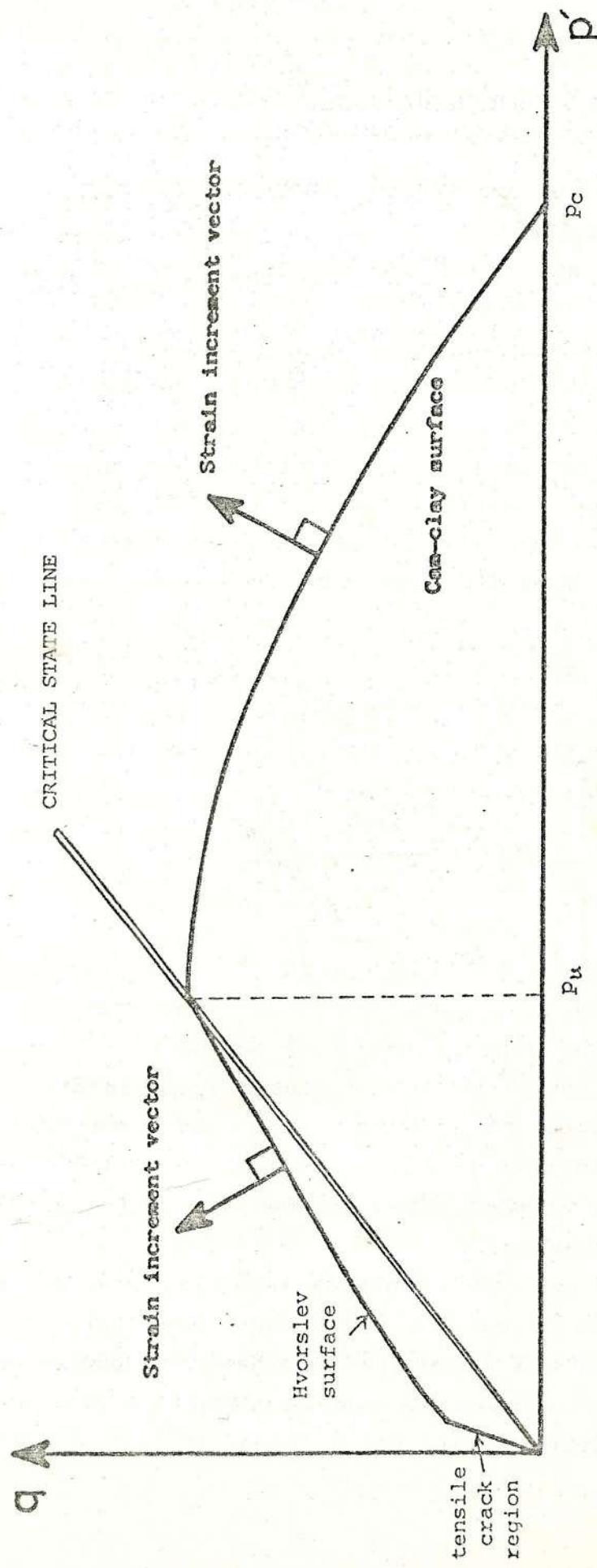


Fig. 4.3 (b) THE SCHOFIELD SOIL MODEL

$$q = (M - H)p_u (p'/p_u)^{\kappa/\lambda} + H p'$$

The other elastic property (G or v')

In the classical development of the Cam-Clay models (see Schofield & Wroth (1968) for Cam-Clay and Roscoe & Burland (1968) for modified Cam-Clay) 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 is equal to -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. The program 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:

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

It has recently 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 yield locus. If the effect of stress cycling is

important in a problem it is probably worthwhile considering the incorporation of a new soil 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 G is dependent on stress level (however it should be noted on the other hand 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 values of the critical state parameters which 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 the Cam 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_o and OCR (see records F to H3 for a discussion of how this is done).

K_w
 K_w is the bulk modulus of water. When an undrained analysis is performed K_w is normally set to a value between 50 and 500 times K' - the reason for this will be made clear following a description of how the program uses this value. The effective stress law can be written in matrix notation:

$$\underline{\sigma} = \underline{\sigma}' + \underline{m} u_w$$

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

$$\underline{\sigma} = [\sigma_x \sigma_y \sigma_z \tau_{xy} \tau_{yz} \tau_{zx}]^T$$

$$\underline{\sigma}' = [\sigma'_x \sigma'_y \sigma'_z \tau'_{xy} \tau'_{yz} \tau'_{zx}]^T$$

$$\text{and } \underline{m} = [1 \ 1 \ 1 \ 0 \ 0 \ 0]^T$$

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 $\Delta\underline{\epsilon}$. Suppose also that incremental effective stresses are related to incremental strains by the relationship

$$\Delta\underline{\sigma}' = D' \Delta\underline{\epsilon}$$

(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 $\underline{m}^T \Delta\underline{\epsilon}$ and the volumetric strain experienced by the pore water is equal to $[(1 + e)/e] \underline{m}^T \Delta\underline{\epsilon}$ where e is the current voids ratio. Then the change in pore water pressure is given by

$$\Delta u_w = K_w [(1 + e)/e] \underline{m}^T \Delta \underline{\epsilon}.$$

Combining this with the effective stress law and the incremental effective stress-strain relation the following equation is obtained

$$\Delta \underline{\sigma} = D' \Delta \underline{\epsilon} + \underline{m} K_w [(1 + e)/e] \underline{m}^T \Delta \underline{\epsilon}.$$

CRISP uses this equation in the following way:

- (i) The program expects in an undrained analysis that the material properties supplied relate to changes in effective stress.
- (ii) When calculating the element stiffness matrices the program adds in the terms corresponding to the volumetric stiffness of the pore water.
- (iii) Following the solution of the finite element equations the program calculates the changes in effective stresses and pore water pressure separately.

In a drained analysis the user sets $K_w = 0$ 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 elastoplastic material law. It is well known that in conventional linear elastic finite element analysis the use of a value 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.

Therefore use effective stress parameters (v' for Cam-clays, E' for elastic) and a K_w value (this has the advantage that pore pressures are calculated directly). Alternatively specify $v_u = 0.49$ and (E_u for elastic models) with $K_w = 0$. In the case of the latter no pore pressures are calculated.

γ_{bulk}

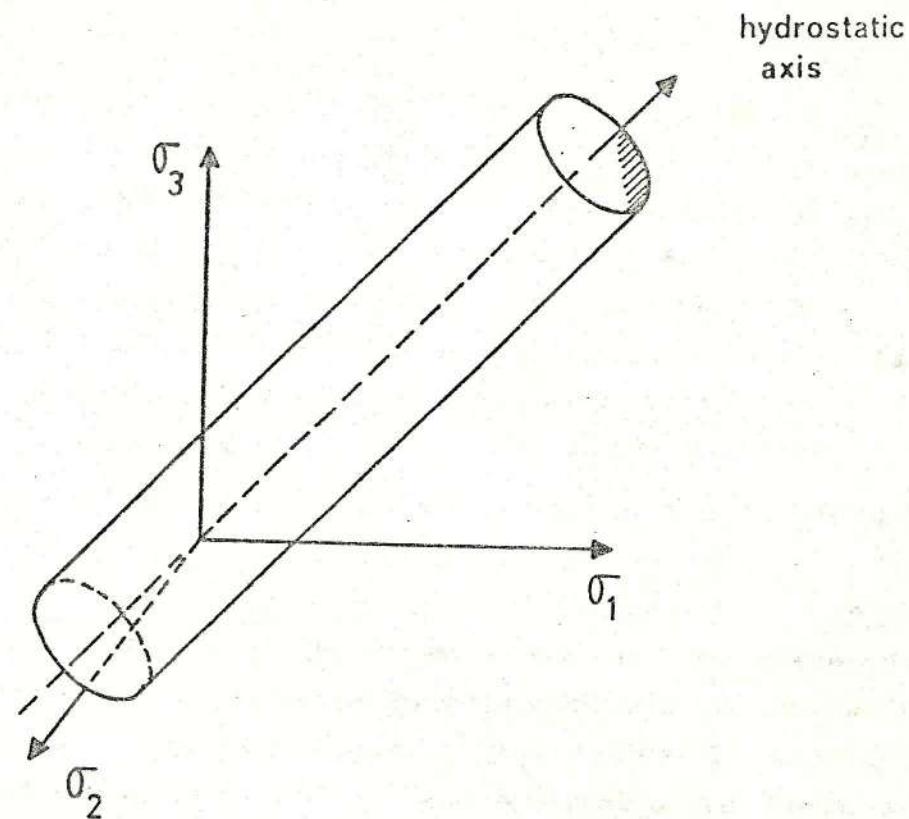
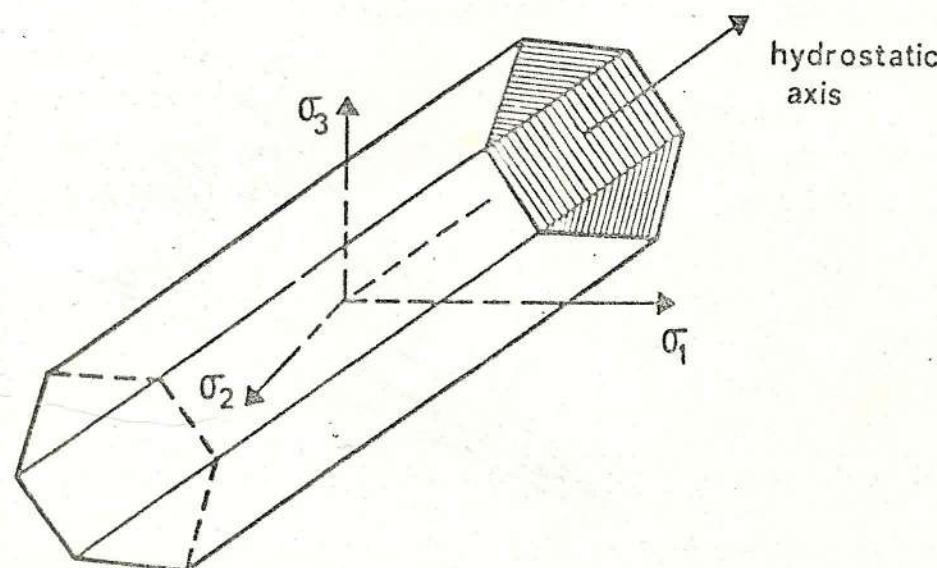
- γ_{bulk} is the bulk unit weight of the soil. This value is used by the program
- (a) when calculating implicit loads caused by excavation (removal of elements) or construction (addition of elements) sequences.
 - (b) when the gravity acceleration field is increased (or decreased) during an analysis (e.g. during geotechnical centrifuge test). (see notes on record I).

The unit for γ_{bulk} is weight/unit volume (see table 4.1).

C, ϕ , J

The shapes of the four possible yield surfaces in principal stress space are shown on the two following pages. 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.

Note that for the Drucker-Prager model (Yield criterion number 3) the yield surface represents a circle which outscribes the corresponding Mohr-Coulomb yield surface.

(1) The Von Mises Yield SurfaceFigure 4.4(2) The Tresca Yield Surface

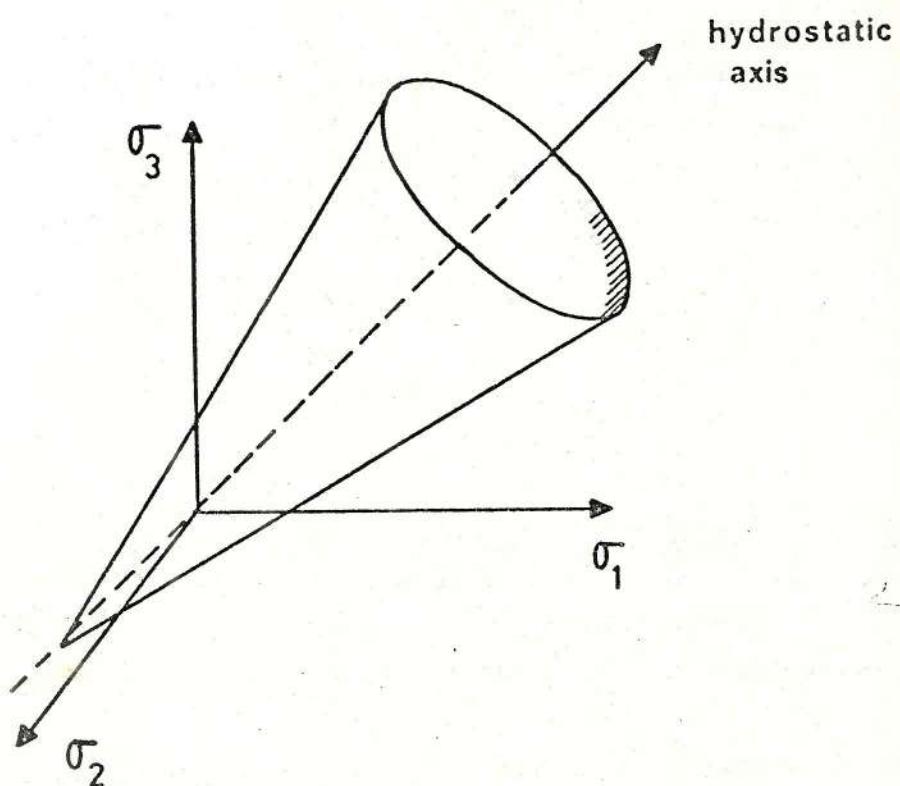
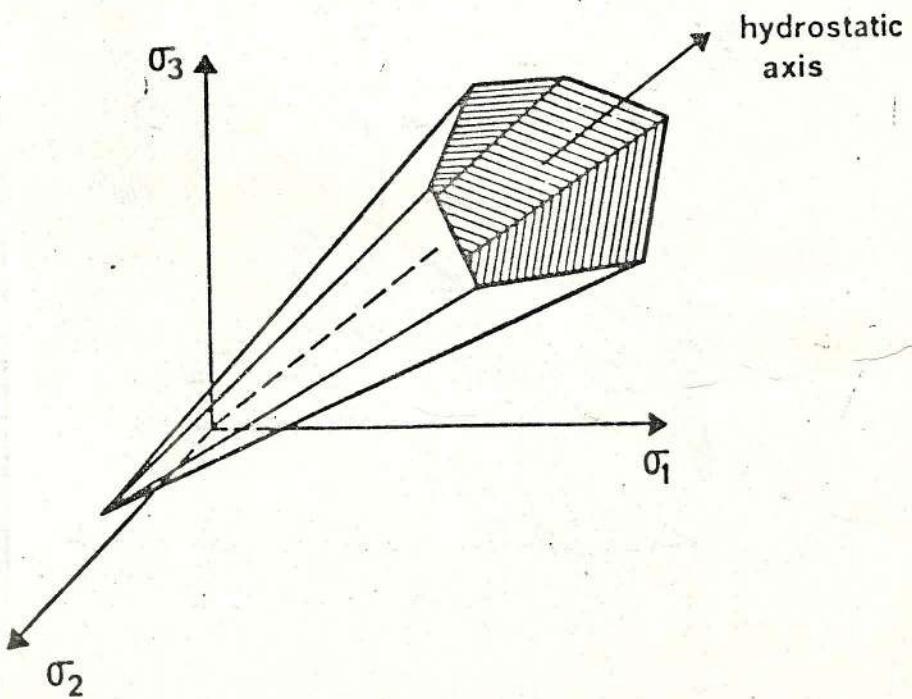
(3) The Drucker - Prager Yield Surface

Figure 4.4

(4) The Mohr - Coulomb Yield Surface

Records F, G1, G2 and G3

In an elasto-plastic analysis the stiffness matrix of a finite element will be dependent 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.

The purpose of record types F, G1, G2 and G3 is to enable the program to calculate the stresses (and for Cam-Clay the size of the yield locus specified by p'_c) before the analysis starts. For the purpose of specifying the in situ stresses the mesh is divided into a number of horizontal layers (option 1). 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 the user specifies a set of *reference points* along a vertical section and the stresses at these points. The in situ stresses at the integration points (see fig.4.5) are interpolated from the stresses specified at these points.

If all these parameters vary linearly throughout the mesh then it is sufficient to define these values at 2 points. The top and bottom of the mesh. These two points then form a single in situ layer. If there are N in situ reference points then there are N-1 in situ layers. It is then assumed that all these parameters have the same value at a given depth ie these only vary in the vertical (y) direction.

This even allows for a jump in a particular stress parameter (for example the effective horizontal stress can be different at an interface between two different layers). Then two reference points 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) have to be continuous and hence should have the same value at both these reference points.

Note that the in situ reference points should cover the whole of the mesh ie If there are N in situ reference points then no part of the mesh which requires any in situ stresses assigned to it should lie outside the range of y coordinates specified by the points 1 and N. However any part of the mesh

which is removed prior to the beginning of the analysis proper can lie outside the range of the in situ reference points. For example the elements (5 and 6) representing the embankment in the example problem number 2 in appendix B.

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, (θ - circumferential)
 τ_{xy} - shear stress in xy plane of plane strain (rz plane in axisymmetry)
 u - initial pore pressure (usually the static head)
 p'_c - size of the initial yield locus (the value at p' axis, $q = 0$).

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 given all the other information the voids ratio is calculated within the program.

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.

For Cam-Clays 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 (overconsolidated)/plastic straining that takes place. In specifying the above stress parameters, the most difficulty lies in the determination of σ'_x (and hence σ'_z). There are basically three approaches to determining the in situ stresses:

Determining the in situ stresses

- (i) 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.).

- (ii) 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:

$$K_{nc} = 1 - \sin\phi' \quad (1)$$

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 then proposes two relationships between K_o , K_{nc} and OCR ($OCR = \sigma'_{vmax}/\sigma'_v$):

$$K_o = OCR K_{nc} - \frac{v'}{1 - v'} (OCR - 1) \quad (2)$$

and

$$m \left| \frac{3(1 - K_{nc})}{1 + 2K_{nc}} - \frac{3(1 - K_o)}{1 + 2K_o} \right| = \ln \left| \frac{OCR(1 + 2K_{nc})}{1 + 2K_o} \right| \quad (3)$$

The first equation 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 v' necessary to fit the observed data were determined by Wroth to be in the range 0.254 to 0.371 for eight different soils. The second equation was proposed as 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. m is an empirical constant which Wroth shows is linearly related to Plasticity Index. (PI) for a number of soils.

CRISP allows only one method for specifying the in situ stresses for Cam-Clays. This is the direct specification of in situ stresses (σ'_x , σ'_y , σ'_z , τ_{xy} , u and p'_c) either at in situ reference points or at all integration points (INSIT = 1 or 2). For a 3-D analysis τ_{yz} and τ_{zx} are also specified after τ_{xy} and before u .

The user can calculate the in situ stresses using the approach (ii) outlined above. The value of PI can be used to obtain the empirical constant m

$$m = 0.022875 \text{ PI} + 1.22 \quad (4)$$

and K_{nc} can be calculated from equation (2) and the equation

$$\sin\phi' = \frac{3M}{6 + M} \quad (5)$$

(iii) An empirical relationship

$$K_o = K_{nc} \cdot OCR^\phi$$

where ϕ is in radians.

Calculation of the size of the initial yield locus (p'_c)

Knowing the past stress history of the ground or the laboratory sample one can determine the variation of OCR with depth. Using an empirical approach or other data one has to establish the relationship between OCR and K_o as mentioned above. Knowing K_o one can calculate the σ'_x variation. Using the OCR value one can calculate the $\sigma'_y \text{ max}$ ($= \sigma'_y \text{ max}$).

$$\sigma'_y \text{ max} = OCR \times \sigma'_y \quad (6)$$

OCR	- overconsolidation ratio
K_o	- coefficient of in situ earth pressure
K_{nc}	- as above, but under normally consolidated state
σ'_y	- effective vertical stress
$\sigma'_y \text{ max}$	- (past) maximum effective vertical stress

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

$$\sigma'_x \max = K_{nc} \times \sigma'_y \max$$

$$p'_\text{max} = (2 \times \sigma'_x \max + \sigma'_y \max) / 3$$

$$= (2 \times K_{nc} + 1) \times (\sigma'_y \max / 3) \quad (7)$$

$$q_\text{max} = \sigma'_y \max - \sigma'_x \max = (1 - K_{nc}) \times \sigma'_y \max \quad (8)$$

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

for Cam clay:

$$p'_c = p'_\text{max} \times e^{(q_\text{max}/(Mp'_\text{max}))}$$

for Modified Cam clay:

$$p'_c = (q_\text{max}/M)^2 / p'_\text{max} + p'_\text{max}$$

where M - is the Critical state frictional constant.

Using equations (6) to (8) one can derive a relationship between p'_c and σ'_y which can be used to calculate p'_c directly from σ'_y . This is repeated for any other depth (and in situ reference point). If the variation of any of these parameters is non-linear (usually σ'_x) then one has to approximate the variation to be of piecewise linear over a number of in situ layers.

Within the program 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 2 in situ reference points which lie on either side of it (see Figure 4.6). 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. This only leaves the calculation of the voids ratio at each integration point which is described in the next section.

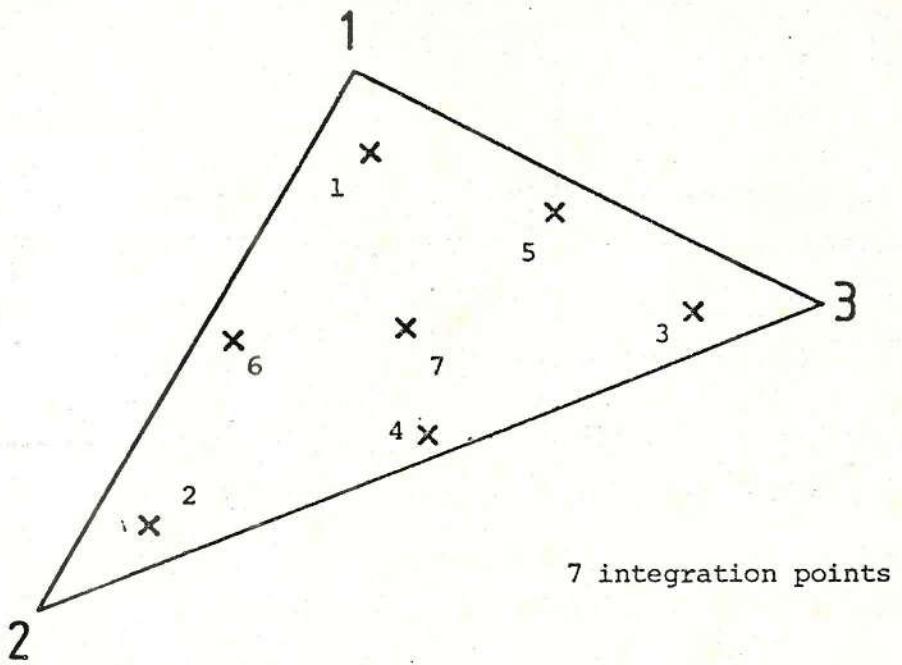
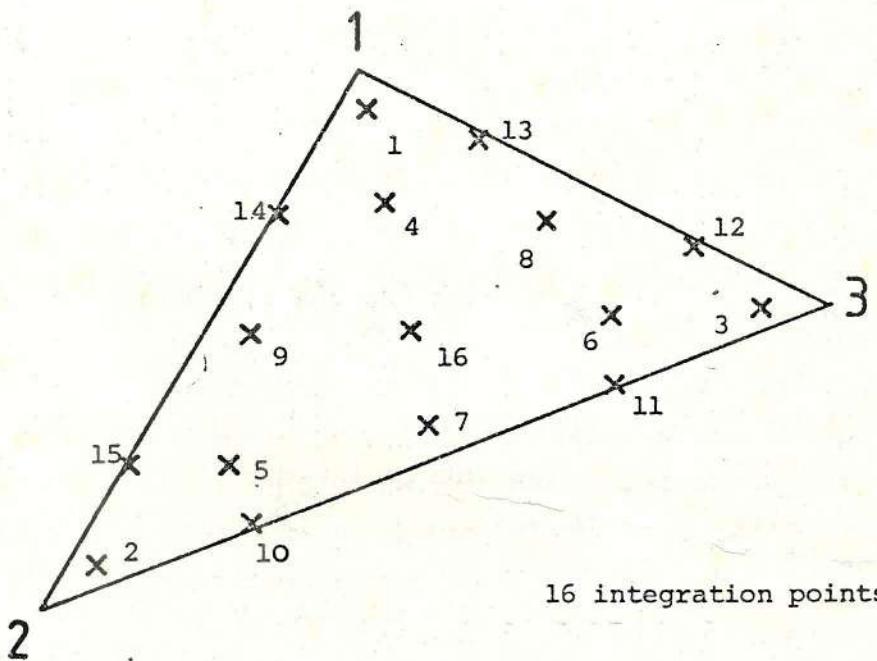
Linear Strain TriangleCubic Strain Triangle

Fig. 4.5 Integration Points

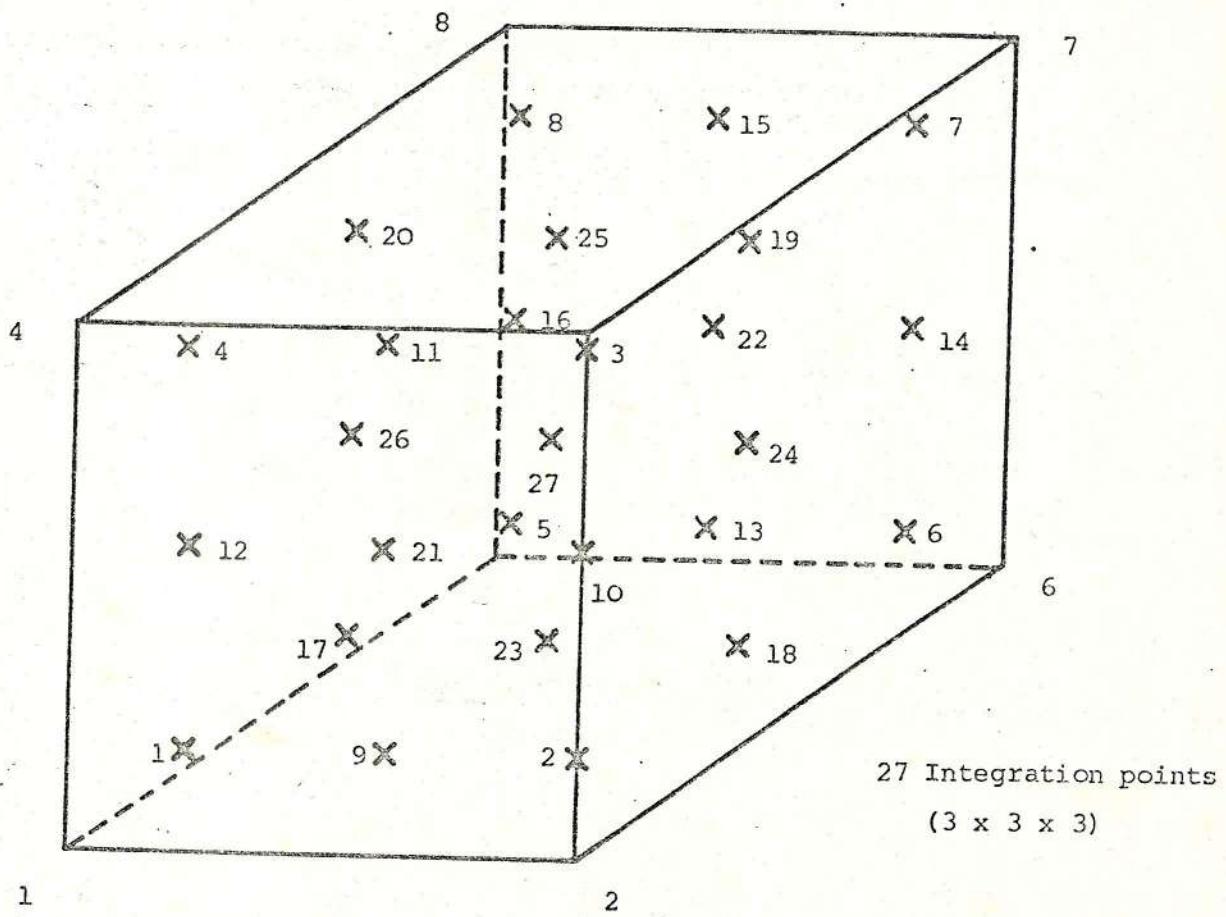
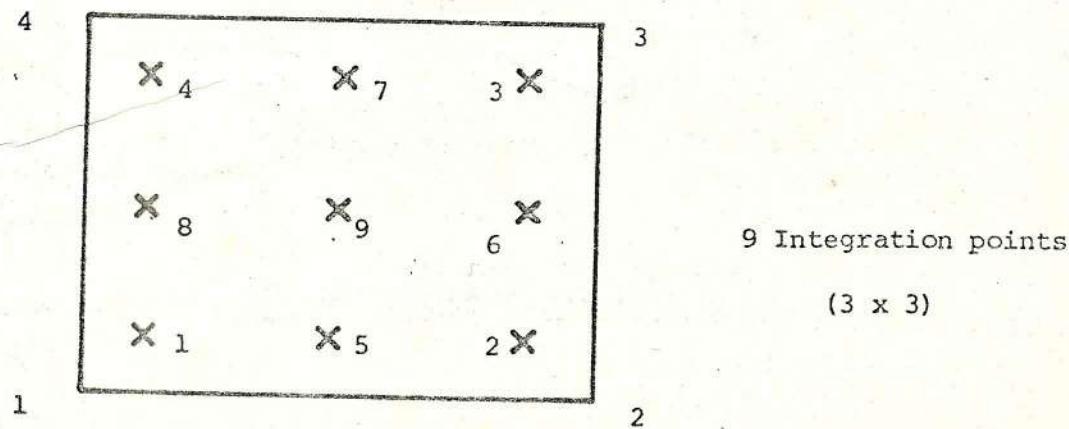
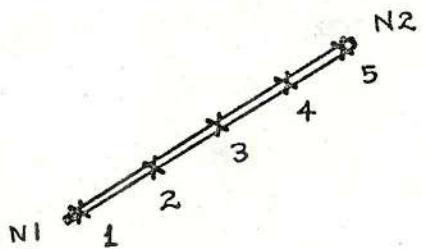
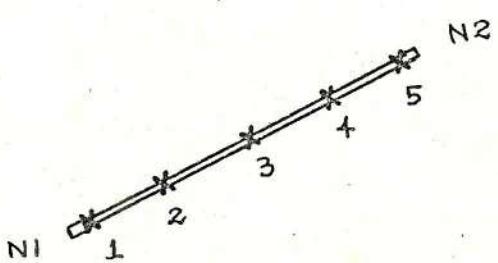
Linear Strain BrickLinear Strain Quadrilateral

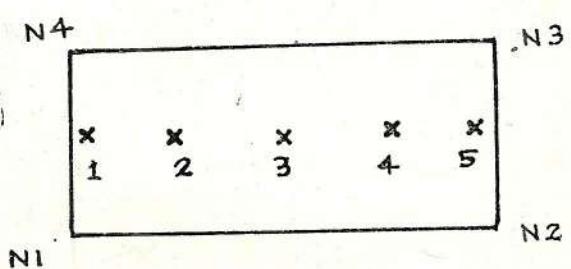
Fig. 4.5 (b) Integration Points



(i) bar element



(ii) beam element



(iii) slip element

Fig. 4.5(c) Integration Points

in situ reference points

M

y_M

⋮

layer (M-1)

⋮ ⋮

N

y_N

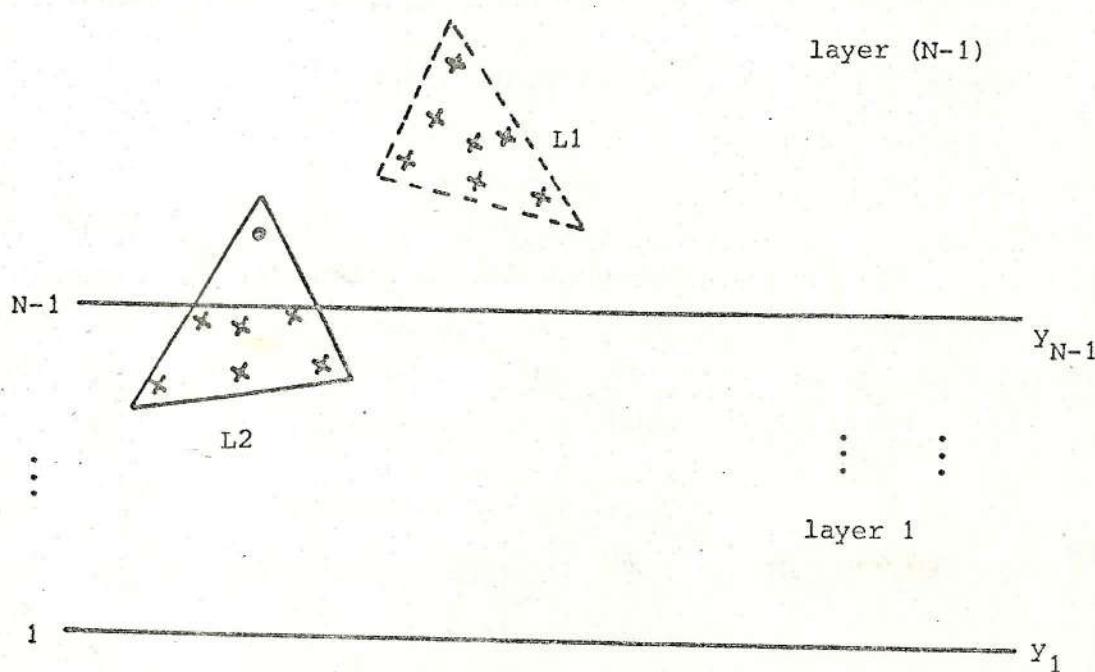


Figure 4.6 : Interpolation of in situ stresses at Gauss points from in situ reference points

Illustrated with a single element (L1) and its Gauss points, which lies within in situ layer (N-1) ie between reference points N-1 and N. Element L2 which is across 2 layers is also permitted. Each Gauss point stress state is established from the in situ layer it falls on.

Calculation of 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 parameters,

- e_{cs} (= $\Gamma - 1$) the voids ratio at $p' = 1$ on CSL.
- κ - slope of swelling line in $e - \ln p'$ space
- λ - slope of compression line in $e - \ln p'$ space

where CSL is Critical State Line

the voids ratio is calculated as follows:

Let us 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 4.7). Assuming that e and p' represent the initial voids ratio and effective mean normal stress respectively (point P in figure). Then for CSL,

where

$$e_a = e_{cs} - \lambda \times \ln(p'_a)$$

$$p'_a = p'_c / 2. \quad (\text{for Modified Cam clay})$$

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

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

$$e - e_a = \kappa (\ln p' - \ln p'_a)$$

$$\text{giving } e = e_{cs} - (\lambda - \kappa) \times \ln p'_a + \lambda \times \ln p'$$

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

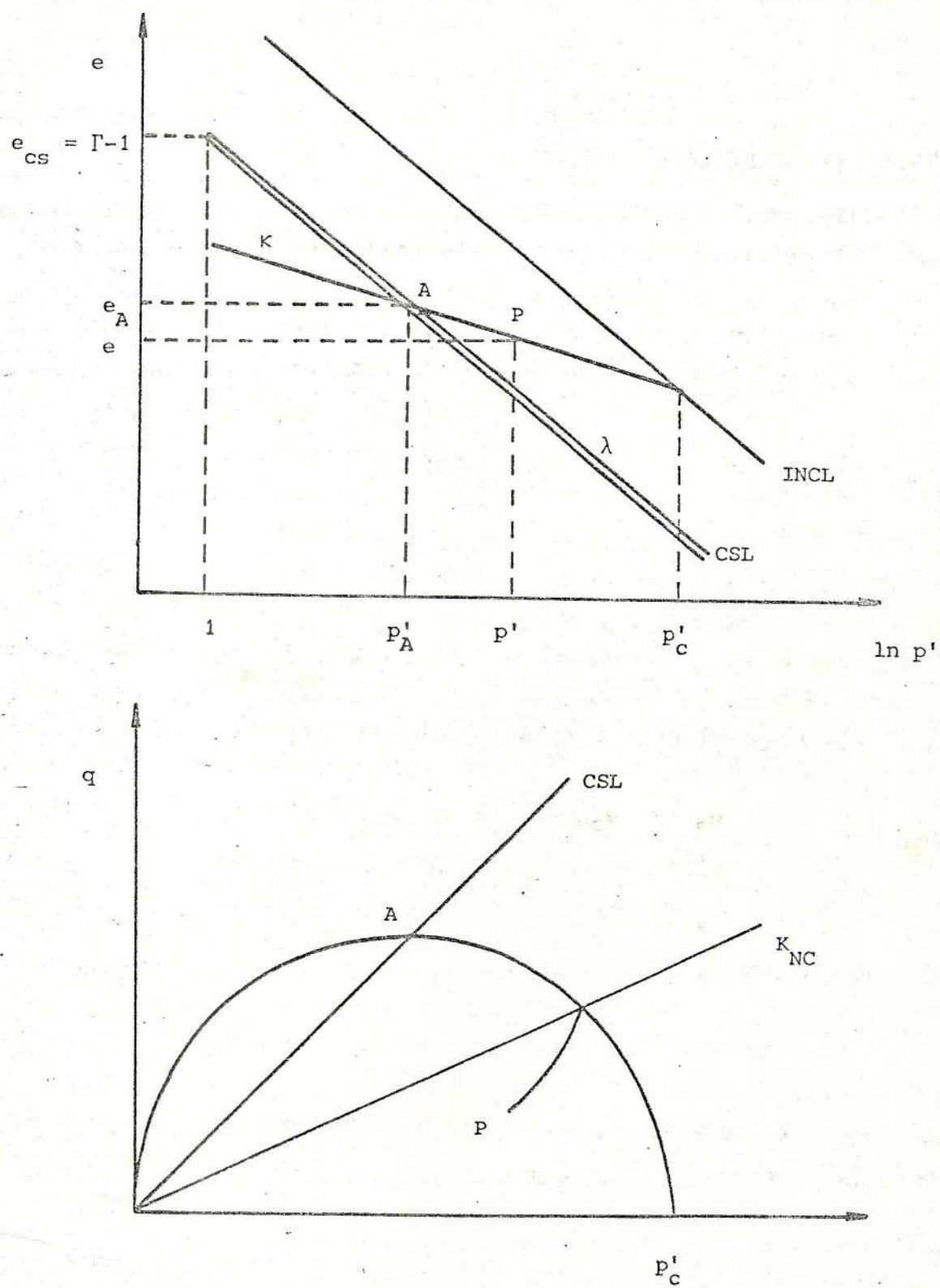


Figure 4.7 : Calculation of initial voids ratio

In situ stress history

Unlike the elastic models the critical state models require the specification of an initial non-zero stress state as a starting point (ie 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 also cannot be zero (size of the initial yield locus) at any integration point. This would indicate a yield locus of zero size. Note that these apply to values at integration points. Therefore one 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. A zero value of p'_c at the ground surface is permitted. Since none of the integration points of an element in any element type lie along the element sides. All of them lie within the element, some however lie close to the side, example (integration points 10 to 15 lie very close to the element sides for the Cubic strain triangular element Fig 4.5). Therefore the integration points close to the ground surface will have a very small p' and a very small value of p'_c if the soil is in a normally consolidated state. This results in a very small stiffness near the ground surface which would result in large deformations.

Depending on whether a load is put on the ground surface which causes it to shear some problems may be encountered with the analysis (watch out for p' becoming negative, which always spells trouble). In an analysis of an embankment on a soft foundation during loading the integration points nearest the surface were stressed such that the yield ratio increased at an unacceptably large rate. This problem was circumvented by giving the initial p'_c , near the surface, a large enough value (eg. stiff crust) or giving the stresses a value corresponding to an arbitrary small surcharge. Then you should specify this small surcharge in records H2 as well to ensure that equilibrium is satisfied at the in situ stage.

This also rectified the problem of integration point having negative p' due to unloading. During the analysis any integration point attaining zero p' or negative p' would make it impossible to carry out some of the calculations or result in zero stiffness. These circumstances should be avoided. If analysis is not possible at all it may be appropriate to model the particular region or zone by a thin layer of elastic or elastic-

perfectly plastic model.

Record H1/G1

At all in situ reference points (if the in situ stresses have been specified) the following condition should be satisfied.

$$\sigma_y = \sigma'_y + u_o = \text{GRAVI} \times \gamma_{\text{bulk}} \times (Y_N - Y_{GS}) \\ + \text{in situ surface pressure}$$

σ'_y	- V(2)
u_o	- V(5) for 2D, V(7) for 3D
γ_{bulk}	- P(8) in record D

where Y_N - is the y coordinate of the reference point
 Y_{GS} - is the y coordinate at the ground surface

$\text{GRAVI} = 0$ in any analysis of laboratory tests
 (triaxial tests, 1-D consolidation tests)

= 1 in for field situations or analysis of prototype
 or fullscale problems.

= n (where $n > 1$) in an analysis of a centrifuge test
 the g (gravity) level.

If the above condition is not satisfied it would lead to an error in equilibrium (see the values printed under the heading PERCENTAGE ERROR IN EQUILIBRIUM) before the Cam clay stress codes are printed. These in general should be negligible (see output from examples 1 and 2).

If the above equation is satisfied and if equilibrium errors are still present then check all the fixities along the entire mesh boundary. Also check the output for any errors detected by the program in the input data for record H3. Because nodes associated with element side which is restrained may have been incorrectly specified. Check also the pressures applied at in situ stage for compatibility with the in situ stresses. Ensure that the surcharge pressure in the above equation is specified in record H2.

Records H1, H2 and H3

The user has 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 the user 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 2-D case where the element side is restrained for the 3-D case an element face can be fixed in a particular direction. However at present there is no option in the program to calculate the nodal loads equivalent to the pressure applied on an element face, for the 3-D case. Therefore the user has to calculate the equivalent nodal loads and input it directly as nodal loads (using option (a) in record H2). When calculating the nodal loads it should be remembered that the intermediate nodes along the element side also carry part of this applied load.

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 this stage and should not be included in the input data.

Displacement fixities

Any displacement fixities (ie 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. See examples 2 and 3 for these two options respectively. Once specified these restraints (displacement fixities, zero prescribed displacements) remain effective during the rest of the analysis. Therefore these need not re-specified for

each and every increment block.

Record I

When a non linear or consolidation analysis is performed using CRISP it is necessary to divide either the loading or the time span of the analysis (or both if there is consolidation with non-linear material properties) into a number of increments. Thus if a total stress of 20 kPa is applied to part of the boundary of the finite element mesh 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 either the Cam Clay or the modified Cam Clay model. 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 inevitably mean that some compromise is made between accuracy and cost. The recommended way of reviewing the results to determine whether enough increments have been used in an analysis is to examine the values of yield ratio (YR) at each integration point (see chapter 5). 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 Cam Clay models is softening (i.e. yielding dry of critical) smaller increments (than the size suggested by the above discussion) may be necessary.

The time intervals for consolidation analysis (DTIME) should be chosen after giving consideration to 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 a good idea 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). Thus a suitable scheme might be:

<u>Increment No.</u>	<u>DTIME</u>	<u>Total Time</u>
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 (see 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.
- (v) when a change in pore pressure boundary condition is applied 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 variables that are close to the boundary. If this is not done then the solution will predict excess pore pressures that show oscillations (both in time and space)

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

- (a) apply loads in the first increment (or first few increments for a nonlinear analysis) but do not introduce any pore pressure boundary

conditions.

- (b) introduce the excess pore pressure boundary conditions in the increment following the application of the loads.

Boundary Conditions (NLOD, NFIX, ILDF, ITMF)

CRISP allows the user to describe a sequence of increments as an "increment block". This facility is provided for two reasons:

- (1) 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).
- (ii) When performing an excavation (or construction) analysis the program calculates the implied loadings to the removal (or addition) of the elements specified by the user. These implied loadings will often be too large to be applied in a single increment when the material behaviour is non linear. 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).

The program user should note the significance of specifying incremental loads in the input data. The total loads acting at any particular time are given by adding together all the previous incremental loads. Thus if part of the mesh is loaded and then subsequently these loads are removed it will be necessary to specify negative incremental loads. Total loads and total fixities remain in force from incremental block to incremental block if there is no action to remove them. A previously fixed node may be released by using a fixity code of 0 (zero).

The following example is intended to clarify these points for a consolidation analysis:

- (a) part of the boundary of a soil mass is loaded with a load of ten units (this is applied in ten equal increments)
- (b) consolidation takes place for some 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 load acting.

Loads		
<u>Increment No.</u>	<u>Incremental load applied</u>	<u>Total load acting</u>
1	1	1
2	1	2
3	1	3
4	1	4
5	1	5
6	1	6
7	1	7
8	1	8
9	1	9
10	1	10
11	0	10
12	0	10
.	.	.
.	.	.
.	.	.
21	-2	8
22	-2	6
32	-2	4
24	-2	2
25	-2	0
26	0	0
27	0	0
28	0	0
29	0	0
30	0	0

etc.

One possible way of translating 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 increment block (increments 11 to 20) would have no incremental loads and

the third (increments 21 to 25) would have an incremental load of -10 with 5 load factors equal to 0.2.

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.

Records I, K1, L and M

The loading (NLOD), self weight loads (DGRAV) and prescribed displacements (and pore pressures) (NFIIX) 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* 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).

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) (NFIIX - 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 under record K3.

Records I and K3

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. in record K1) these are actual time steps for the increments and not rations. 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 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

in record I

IBNO	INCA	INCB	TCHEL	NLOD	ILDF	.	.	.	DTIME	ITMF	DGRAV
1	1	9	0	-	1	.	.	.	100	1	0

in 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

in record K3

DTM(1)	DTM(2)	DTM(3)	DTM(4)	DTM(5)	DTM(6)	DTM(7)	DTM(8)	DTM(9)
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.

record I

IBNO	INCA	INCB	ICHEL	NLOD	ILDF	DTIME	ITMF	DGRAV
1	1	3	0	-	0	3.	0	0
2	4	9	0	-	0	97..	1	0

record K1

not present for both increment blocks (ILDF = 0 in record I)

record K3

	DTM(1)	DTM(2)	DTM(3)	DTM(4)	DTM(5)	DTM(6)
incr block 1			not present	(ITMF = 0 in record I)		
incr block 2	2	5	10	10	20	50

More on option (a):

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.

Record M (Pore pressure fixity)Pore pressure fixity

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 the user need not specify any pore pressure boundary conditions. All boundaries are automatically assumed to be impermeable, unless specified otherwise. In the case of the latter 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 fixity code 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 the user subtracts the in situ (normally hydrostatic) pore pressure from this value and then specifies this with a fixity code of 2 (see the next section for details). For example this will be the case in an oedometer test, along the top draining surface after the application of the load. When it is necessary to specify pore pressure boundary condition along a boundary along which the current pore pressure distribution is not known then one uses a fixity code of 2.

Finally if these pore pressure fixities are used in an increment block consisting of more than one increment, the changes specified using fixity code 1 is spread over all the increments exactly in the same manner as the displacement fixity (see records K1). However pore pressure fixity code 2 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.

Pore pressure fixity 2

The incremental changes of excess pore-pressure 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). But in the CRISP program it is not possible to fix the total value of the pore pressure directly. This has to be done indirectly by fixing the value of excess pore-pressure.

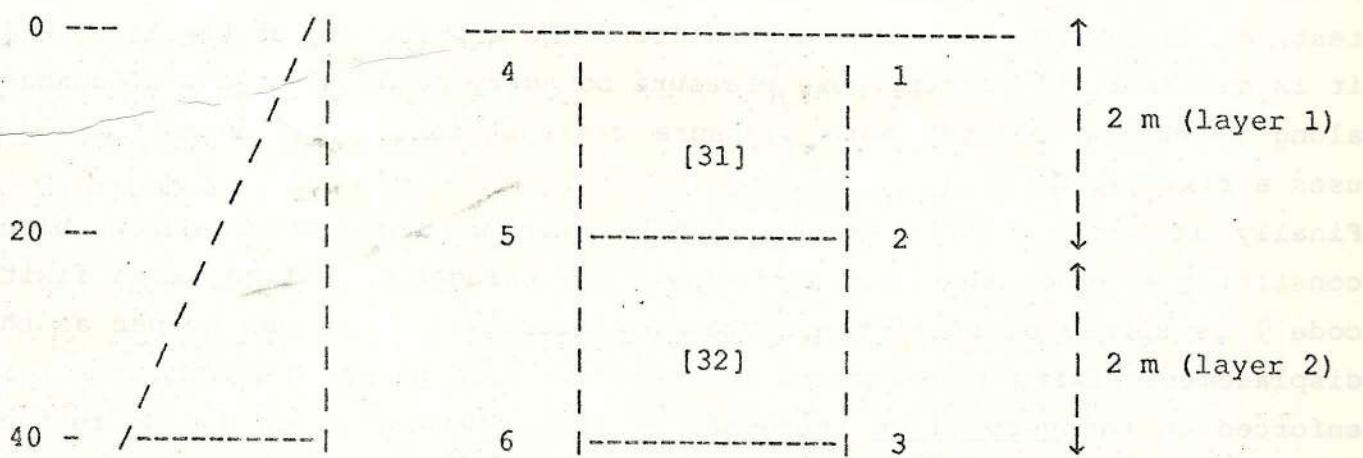
If u represents the total pore pressure at which a particular node is to be maintained, and if u_o is the in situ (hydrostatic) value of the pore pressure at that node, then:

$$\text{total p.p.} = \text{in situ p.p.} + \text{excess p.p.}$$

$$u = u_o + u_e$$

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

The user then specifies u_e in the input data with a fixity code 2, and not u . This is 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.



in situ pore pressure

[] - element numbers

For node 2 after excavating the first layer total pore pressure = 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 the users. The most common mistake is to incorrectly fix the excess pore pressure to 0 using a fixity code of 2. The fixity input is as follows 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 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-pressure 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.

(*). In the above example 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 pressure (Δu_e).

$$\text{incremental excess pore pressure} = \Delta u_e$$

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:

$$\text{absolute excess pore pressure } (u_e) = \sum_i^n \Delta u_e$$

Summarising, it could be said that when using fixity code 1 it is the Δu_e which is specified and when using fixity code 2 it is u_e 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 the latter the total pore pressure is held at u (corresponding to a excess pore pressure of u_e) in 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 2 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.

4.4 STOP/RESTART FACILITY

Input Data for an Analysis using Stop/Restart Facility

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 IRL and the results from the current run are stored on unit IW2. The restart files for ISR = 2 will be large and probably require use of magnetic tapes.

Option 1 : ISR = 2

The STOP/RESTART facility which uses two magnetic tapes 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 (i.e. without using the magnetic tapes) 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 upto 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 makes use of disk files (in the absence of magnetic tapes) and permits a large analysis to be broken into a number of smaller runs. Because of the limitations on the size of disk files only the results from the last increment is 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 the analysis can only be continued from the point left off.

4.5 CRISP Plotting Program (a post processor)

The CRISP plotting program² makes use of the cumulative displacements and stresses at the end of a given increment. This data for plotting can come from two sources.

- (i) <FILE> stored in a magnetic tape (requires use of STOP/RESTART facility when running CRISP)
- (ii) <DF> stored on disk.

When using STOP/RESTART facility cumulative results (displacements, stresses, strains etc.) of all increments are written to tape. This produces a rather a large file, but allows post processing at the end of any increment (cumulative mode) or between any two increments (incremental mode).

If it is known beforehand at which increments post processing is required then results from selected increments may be stored on the disk file.

For example, if in an analysis of 100 increments, storing results after increments 10 20 80 and 100 allows cumulative post processing for any of these increments and incremental post processing between any two of the stored increments.

The creation of a disk file and writing results to magnetic tape, are independent functions. The disk file is created solely to make use of the plotting program. The results written to a magnetic tape is essential to stopping and restarting an analysis. As a spin-off the results written to the tape can be used by the plotting program as well.

The data written to the disk file is controlled by data input (records C2 and C3) to the Main program.

² This plotting program is different from the MESH plotting program described elsewhere which only produces the plot of the initial geometry mesh