CAMBRIDGE UNIVERSITY

CRISP 90

User's and Programmers's Guide

A. M. Britto and M. J. Gunn *

Engineering Department

Bruno Teixeira

The following sections of this manual are not applicable to the PC-386 version of CRISP-90.

1) section 2.3 (c) Plot results
2) section 4.5 Page 4.74
3) Appendix A

These are only applicable to running CRISP-90 in the Cambridge University IBM 3084 computer.

C R I S P 9 0

User's and Programmers's Guide

A. M. Britto and M. J. Gunn *

(1/6/90)

* Surrey University

PREFACE

The CRISP User's and Programmer's Guide describes the finite element program incorporating *Critical State* models of soil behaviour that has been developed by members of the Cambridge University Engineering Department Soil Mechanics Group.

The first part of the guide contains information which should enable a user of the program to apply it to a particular geotechnical boundary value problem. The second part of the guide should enable a programmer to modify the program for a particular purpose (e.g. addition of a new soil model) or to implement the program on a new computer. The second part of the guide describes versions of the programs which were current on 1 June 1990 (Geometry Program; version GP0. Main Program version MP0).

CONTENTS

VOLUME 1 (USER'S GUIDE)

1. INTRODUCTION

1.1 History

The computer program(s) known as CRISP were 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 the programs. Some enhancements to the programs have originated from the work of other members of the Soil Mechanics Group (John Carter, Nimal Seneviratne and Scott Sloan).

The program(s) were originally called "MZSOL" and in 1976 they were renamed "CRISTINA". The 1980 versions of the program(s) were called "CRISTINA 1980" and early in 1981 they were renamed "CRISP" (CRItical State Program(s)). Examples of the use of the program(s) can be found in the Ph.D. and M.Phil. theses of various members of the Soil Mechanics Group (Seneviratne 1979; Mair 1979; Davies 1982; Kusakabe 1980, 1982; Almeida 1981; Taylor 1984, Almeida 1984, Phillips 1987, White 1987, Sun 1987). The use of the program(s) was also reported in two papers published in the proceedings of the Tenth International Conference on Soil Mechanics and Foundation Engineering (Bassett et al 1981; Mair et al 1981).

1.2 What CRISP can do

This section contains a short summary of the analysis facilities available in CRISP.

a) 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.

b) Soil models:
Anisotropic elasticity, in-homogeneous elasticity (properties varying with depth), Critical State soil models (Cam Clay, Modified Cam Clay) and Elastic-perfectly plastic models (with Tresca, von Mises, Mohr-Coulomb or Drucker-Prager yield/failure surfaces).

c) Element types:
Linear strain triangle, cubic strain triangle linear strain quadrilateral and the 20-noded brick element (with extra pore pressure degrees of freedom for a consolidation analysis). Beam, bar and slip elements for two dimensional plane strain analysis.

d) Non linear techniques:
Incremental (tangent stiffness) approach. Options for updating nodal co-ordinates with progress of analysis. $\theta = 1$ for integration in time.

e) Boundary conditions:
Element sides can be given prescribed incremental values of displacements or excess pore pressures. Loading applied as nodal loads or pressure loading on element sides. Automatic calculation of loads simulating excavation or construction when elements are removed or added.

f) miscellaneous:
Free format input. Stop - restart facility allows analysis to be continued from a previous run using two magnetic tapes. Data checked for errors using a separate program. Post Processor (program) can be used by writing the results to a magnetic tape or writing results of selected increments to a disk file.

1.3 A Guide to the Guide

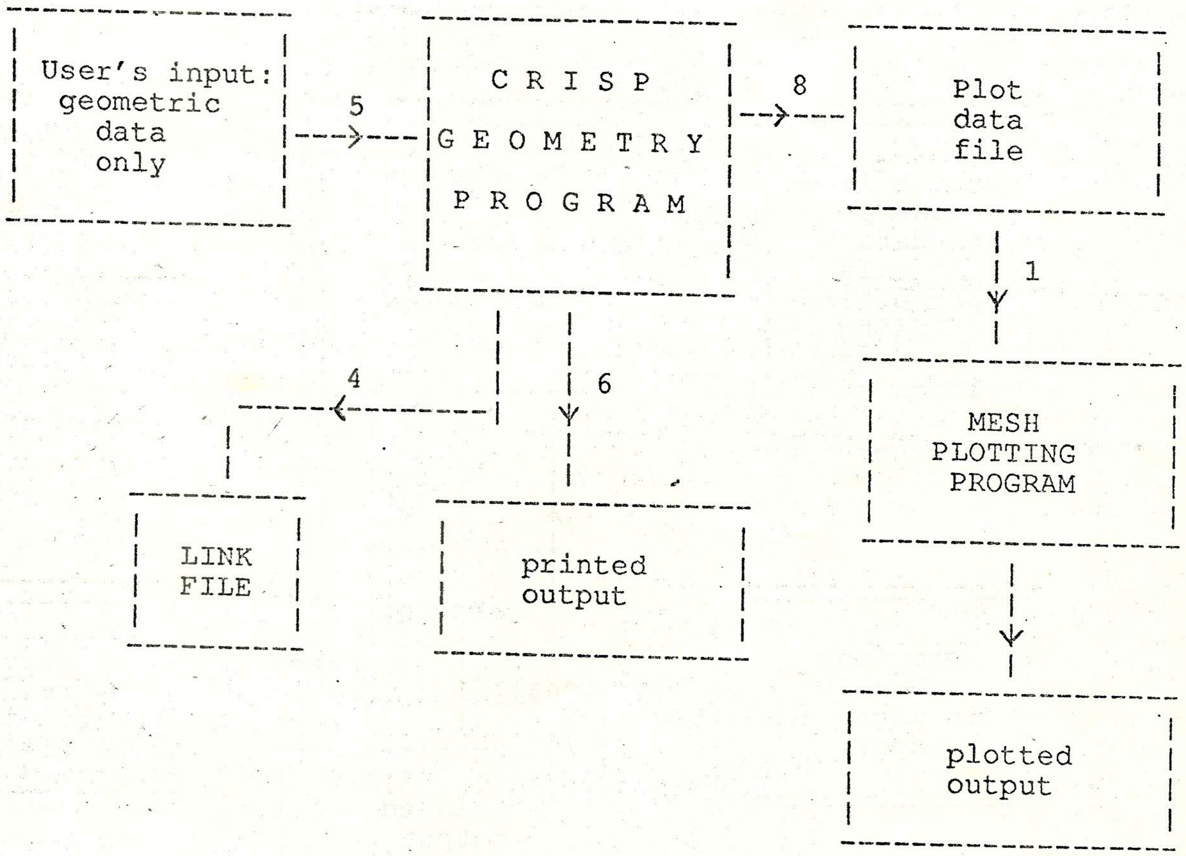
Chapter 3 (Input Specification) is intended as a concise yet complete description of how a user should prepare the input data to solve a particular problem. Chapter 4 contains extra explanatory information and hints designed for a new user of the program(s). It is expected that a user who becomes familiar with the use of the program(s) would mainly consult Chapter 3 and only make occasional reference to Chapter 4. The output produced by the program is described in Chapter 5. The authors believe that examples of the use of computer programs constitute an important and extremely useful part of documentation and for this reason four simple examples are presented in appendix B.

1.4 Further Source of Information

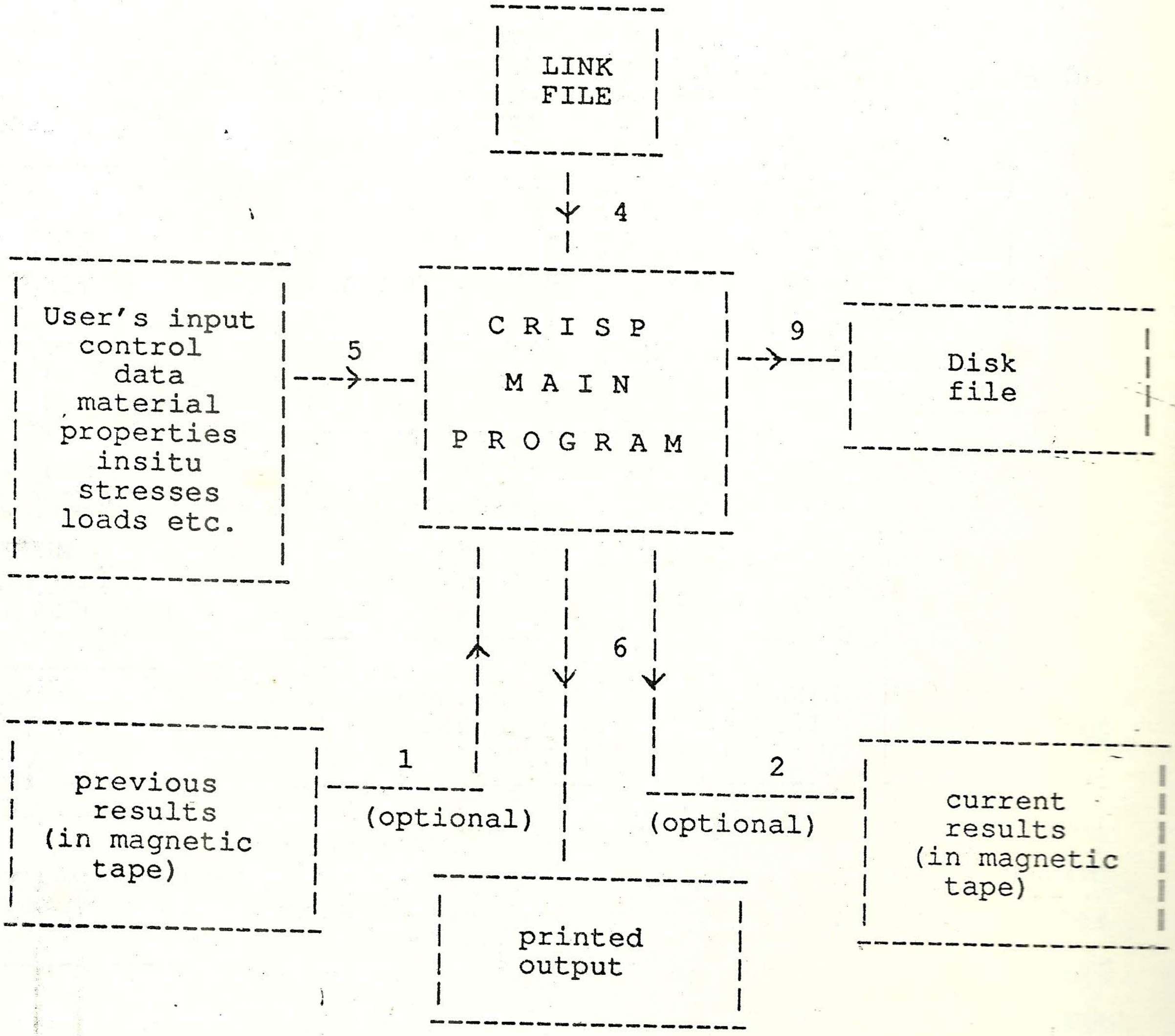
In writing this guide the authors have assumed that the reader will possess a certain amount of familiarity both with the ideas of Critical State Soil Mechanics and the Finite Element (Displacement) Method. Aspects of both these topics are discussed in the book **Critical State Soil Mechanics via Finite Elements** (Britto and Gunn 1987) and in a report (Chapters 2 and 3, Gunn 1981) 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). A large number of introductory texts are now available which describe the finite element method as applied to problems of linear elasticity. 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.

2. SYSTEM DESCRIPTION

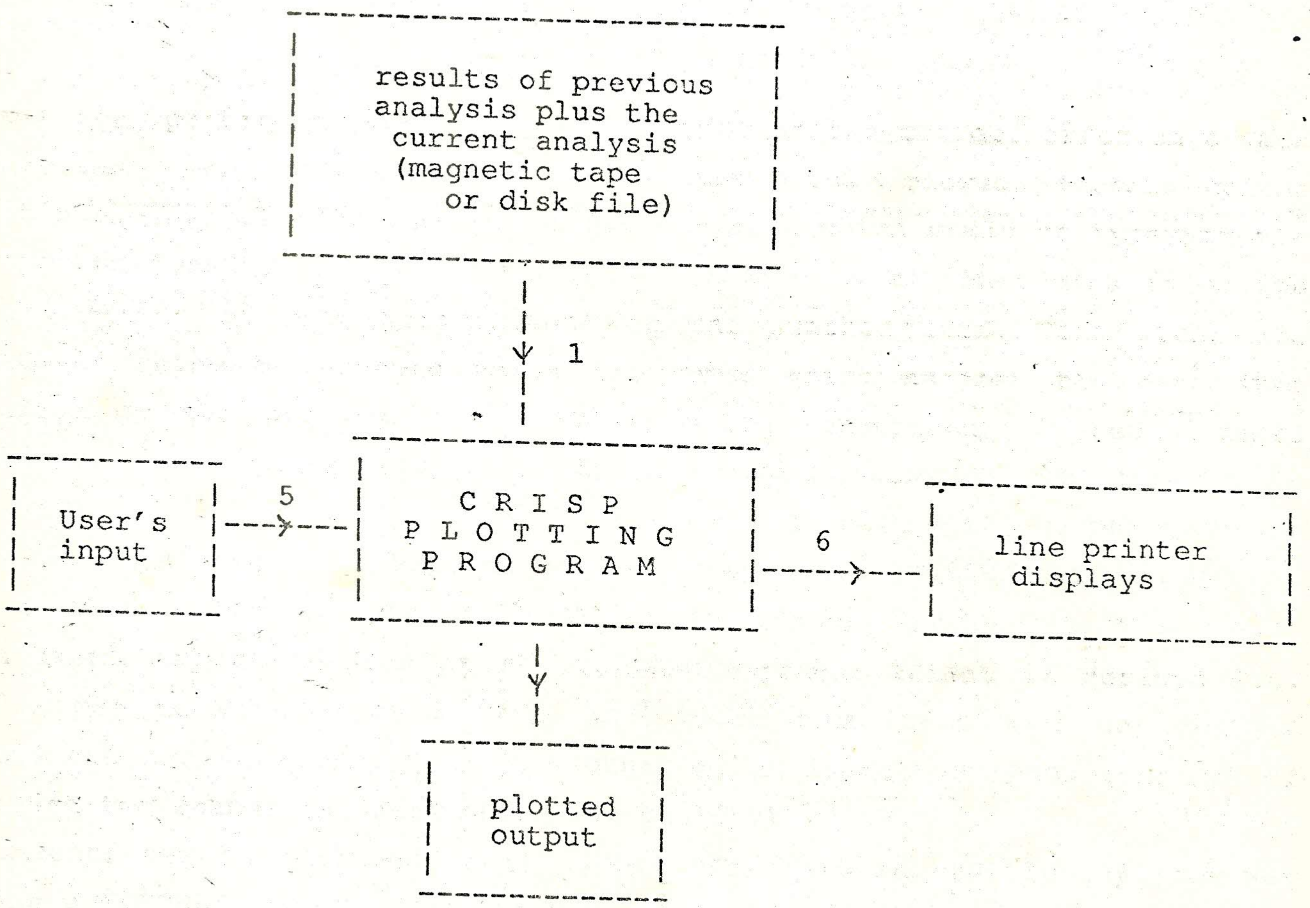
(a) <u>Prepare geometric data and check the mesh</u>



(b) Run analysis



(c) Plot results   (only available on the Cambridge University Computer)
(IBM 3084)



*** uses local CAMPLOT.HIGH graphics commands.

3. INPUT SPECIFICATION

3.1 Data format

The data for both the Geometry Program and the Main Program is Free Format i.e. particular data items must appear in the correct order on a data record but they are not restricted to appear only between certain column positions. The term "record" describes the data which would be typed in via a computer terminal and would occupy one line of a computer disk file (the normal method of data preparation) or one punched card. Data items are indicated below by memomic names ie. names which suggest the data item required by the program. The FORTRAN naming convention is used : names beginning with the letters I, J, K, L, M and N show that the program is expecting an INTEGER data item whereas names beginning with any other letter show that the program is expecting a REAL data item. INTEGER data items must not contain a decimal point but REAL data items may optionally do so. REAL data items may be entered in the FORTRAN exponent format if desired i.e. 0.0011 may be entered as 1.1E-3. Individual data items must not contain spaces and are separated from each other by at least one space (the use of at least two spaces is recommended for clarity).

Comments may be included in the input data file in exactly the same way as for a FORTRAN program. Any record (except for the first or TITLE record) that has the character C in column 1 is ignored by the programs. This facility enables the user to store information relating to parameter values, units assumed etc. permanently with the input data rather than separately.

The programs only read data from the first 80 columns of each record.

3.2 Geometry Program

* indicates extra explanation in chapter 4.

*Record A (one only)

```
------------------------------------------------
|    TITLE         (up to 80 characters)       |
------------------------------------------------
```

*Record B (one only)

```
------------------
|    LINK        |
------------------
```

LINK - a code number set by the user

*Record C (one 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
- 2 - Linear strain triangle with displacement unknown
- 3 - Linear strain triangle with displacements and excess pore pressures unknown (linear variation in porepressure)
- 4 - Linear strain quadrilateral with displacement unknown
- 5 - Linear strain quadrilateral with displacement and excess pore pressures unknown (linear variation in pore pressure)
- 6 - Cubic strain triangle with displacements unknown

7 - Cubic strain triangle with displacements and excess pore pressures unknowns (Cubic variation in pore pressure)
8 - Linear strain brick with displacements unknown
9 - Linear strain brick with displacements and pore pressure unknowns (linear variation in pore pressure) unknowns (plane strain only)
12 - 3-noded beam element with displacements and rotations unknowns
13 - 6-noded interface element with displacement 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 (one only)

NUMAX	MUMAX

+ NUMAX - Maximum value of user vertex node number
+ MUMAX - Maximum value of user element number

+ use of 0 is only valid if user node and element numbers begin with 1 and there are no gaps in the numbering

*<u>Record E</u> (one only)

```
-----------------------------------------------------
| ID1   ID2   ID3  ...................  ID10        |
-----------------------------------------------------
```

ID1...ID10 - debugging option. to printout various arrays in geometry part of program

0 - no printout
1 - list of arrays printed are given below; if set to 1 the following are printed

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 coordinates 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 coordinates and element-nodal connectivity 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 coordinates 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 (one only)

```
-------------------------------------------
| NSDZ    NSPZ    NDCUR    NPCUR          |
-------------------------------------------
```

the following 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.

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)

*Record G (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 (one only)

```
-------------------
| IRNFR           |
-------------------
```

IRNFR - option to specify separate list of optimum frontal numbering of elements

1 - read separate list (see record I)

0 - use the sequence in which elements are read (see record J)

*<u>Record I</u> [(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

*Record J (NEL records)

```
-------------------------------------------------------
| KEL   ITYP   IMAT   N1   N2.......NV               |
-------------------------------------------------------
```

```
KEL  - Element number

ITYP - element type number
        1 -  3 noded bar (Plane strain only)
        2 -  6 noded LST   (2-D)
        3 -  6 noded LST   (2-D consolidation)
        4 -  8 noded LSQ   (2-D)
        5 -  8 noded LSQ   (2-D consolidation)
        6 - 15 noded CuST  (2-D)
        7 - 22 noded CuST  (2-D consolidation)
        8 - 20 noded LSB   (3-D)
        9 - 20 noded LSB   (3-D consolidation)
       12 -  3 noded beam element (Plane strain only)
    ** 13 -  6 noded interface element (Plane strain only)

IMAT - material zone number in the range 1 to 10

N1,N2....NV - vertex node numbers listed in anticlockwise order

 where NV - the no. of vertex nodes  = MXNDV
            (based on element type MXTYP in record C)
```

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. Also see appendix K.

*Record K (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 element side
X1,Y1 )
X2,Y2 )      - coordinates of intermediate (displacement)
......)        nodes along curved element side.
..... )        for NSDZ displacement nodes.
..... )        (excluding end nodes)
XN,YN )

Z1,..ZN        for 3-D elements only
```

*Record L (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 element side
X1,Y1 )
X2,Y2 )      - coordinates of intermediate (pore-pressure)
......)        nodes along curved element side.
..... )        for NSPZ pore-pressure nodes.
..... )        (excluding end nodes)
XN,YN )

Z1, . ZN       for 3-D elements only.
```

3.3 Main Program

*Record A (one only)

```
-----------------------------------------------------------
|   TITLE   (up to 80 alphanumeric characters)     |
-----------------------------------------------------------
```

*Record B (one only)

```
-------------------
|   LINK          |
-------------------
```

LINK - the link code number

*Record C1 (one only)

```
----------------------------------------------------------------------
| NPLAX  NMAT  NOIB  INC1  INC2  IPRIM  IUPD  ICOR  ISR |
----------------------------------------------------------------------
```

NPLAX - plane strain/axisymmetric/3-D analysis option
  1 - axisymmetric
  0 - otherwise (Plane strain / 3-D 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 - coordinates are not updated after each increment
  1 - coordinates are updated after each increment

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
    (this is set to 1 only in the presence of elastic perfectly plastic models - model number 5)

ISR - option to stop and restart an analysis either using a disk file or a file in magnetic tape.
  0 - if the stop restart facility is not being used.

1 - stop restart facility using a disk file (only the results from the last increment in the current analysis is written to file). This is the partial stop/restart facility.
2 - stop restart facility using two magnetic tapes. This is the full stop/restart facility and the analysis can be restarted from the end of any previous increment.

*Record C2 (one only)

INSOP	IBC	IRAC	NVOS	NVOF	NMOS	NMOF	NELOS	NELOF

INSOP - in situ stresses output option
- 0 - in situ stresses are not printed
- 1 - in situ stresses printed only at element centroids
- 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 node number for output ***

NMOF - finishing midside node number for output ***

NELOS - starting element number for output ***

NELOF - finishing element number for output ***

*** This allows one 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.

*<u>Record C3</u> (one only)

```
---------------------
|   NINCP           |
---------------------
```

NINCP - total number of increments written to disk for post processing.

*<u>Record C4</u> (one only - present if NINCP > 0)

```
------------------------------------------------------------
| INCLST(1)  INCLST(2) ....... INCLST(NINCP)  |
------------------------------------------------------------
```

INCLST(1) .... INCLST(NINCP) - list of selected increments written to disk

The results from each of this increment are written to a disk file for post-processing.

*<u>Record D</u> (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 H will have the following properties

NTY - material property type as in the table below:
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 - The Schofield soil model (SCHO) (requires 16 properties)

NTY / property	1	2	3	4	5	6
P(1)	E_h	E_o	κ	κ	E_o	κ
P(2)	E_v	y_o	λ	λ	ν	λ
P(3)	ν_{hh}	m	e_{cs}	e_{cs}	C	e_{cs}
P(4)	ν_{vh}	ν	M	M	ϕ	M
P(5)	G_{hv}	0	G or ν'	G or ν'	y_o	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 ------→					

** these material properties are dependent on the element <u>types</u>.

P(11)	0	0	0	0	m_E	H
P(12)	0	0	0	0	m_C	S
P(13)						0
P(14)	not present for models 1 to 5					0
P(15)						k_{xt}
P(16)						k_{yt}

For the Schofield soil model material properties P(13) to P(16) are input in a second line.

6th property for model number 5:

J - yield criterion
1 - von Mises
2 - Tresca
3 - Drucker-Prager (Outscribing circle)
4 - Mohr-Coulomb

In the above

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 y_o

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

H - slope of Hvorslev surface along constant volume section in q - p' space.

S - slope of tensile crack region in q - p' space.

m_C - rate of increase in shear strength with depth

For the bar, beam and interface (slip) elements the material property type number 8 (NTY = 8) with the following material properties are specified.

property	bar	beam	interface
NTY	8	8	8
P(1)	E	E	C
P(2)	ν	ν	ϕ
P(3)	A	A	$k_n = \dfrac{E(1-\nu)}{(1+\nu)(1-2\nu)}$
P(4)	0	I	$k_s = G = \dfrac{E}{2(1+\nu)}$
P(5)	0	0	$k_{sres} = G_{res}$
P(6)	0	0	t

P(7) - P(12) are all zeroes.

where
- t - thickness or height of slip element
- A - cross sectional area
- I - Second moment of area of cross section
- k_n - modulus in the normal direction
- k_s - shear modulus
- k_{sres} - residual shear modulus

*Record E [(IPRIM-1)/10 + 1 records] - only present if IPRIM > 0
records E to H3 are omitted for a restarted analysis
using the stop/start facility

```
-----------------------------------------
| L(1)  L(2)  . . . . L(IPRIM)          |
-----------------------------------------
```

L(1) etc. - list of element numbers to be removed to form mesh at start of analysis

These elements are later "added" to simulate a construction event (eg. embankment). These elements do no have a stress history and therefore cannot be assigned critical state model properties ie the material type cannot be 3 (CC) or 4 (MCC) or 6 (SCHO).

there must be 10 element numbers per record (except for the last record in this group)

*Record F (one 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)

** note that 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 (NNI records) only present if INSIT = 1

| IN YN V(1) V(2) . . . V(NVRS) |

IN - in situ reference point
YN - y coordinate of reference point
V(1)...V(7) - σ'_x, σ'_y, σ'_z, τ_{xy}, u, 0 and p'_c

V(1)...V(9) - σ'_x, σ'_y, σ'_z, τ_{xy}, τ_{yz}, τ_{zx}, u, 0 and p'_c

for 2-D analysis NVRS = 7
for 3-D analysis NVRS = 9

All are effective stresses.
u - in situ pore pressure (the static head)
p'_c is specified as zero if not Cam-Clay

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 (only present if INSIT = 2)
There are NEL sets of records G2 and G3 - one set for each element

| IL |

IL - element number

Record G3 (NIP records - only present if INSIT = 2)

| VAR(1) VAR(2) .... VAR(NVRS) |

for 2-D analysis NVRS = 7

VAR(1) .... VAR(7) - stress parameters at each integration point (σ'_x, σ'_y, σ'_z, τ_{xy}, u, e, p'_c)

for 3-D analysis NVRS = 9

VAR(1) .... VAR(9) - stress parameters at each integration

point $(\sigma'_x, \sigma'_y, \sigma'_z, \tau_{xy},$
$\tau_{yz}, \tau_{zx}, u, e, p'_c)$

where
e - voids ratio
p'_c - size of the initial yield locus (see Fig. 4.3)

for all models other than the Cam-Clays e and p'_c must be set to zero

Record H1 (one only - omit if in situ stresses are set to zero in record F)

NLODI	NFIXI	GRAVI

NLODI - number of element sides with pressure loading (NLODI < 0) or number of nodes with point loads (NLODI > 0) (which are in equilibrium with the in situ stresses)

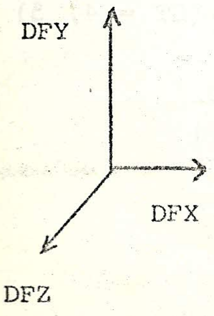
NFIXI - number of element sides which are restrained along boundary of the mesh. (ie displacement fixities)

GRAVI - In situ gravity acceleration field - 0, 1 or n
- 0 - for analysis which does not involve the effects of gravity (eg. laboratory testing : triaxial tests, consolidation tests)
- 1 - for analysis field problems or prototypes
- n - for analysis of centrifuge tests (for example in analysing a test at 100 g n is set to 100).

Record H2 (NLODI records) omit if NLODI = 0 in record H1

(a) NLODI > 0 (Nodal point loads)

N	DFX	DFY	DFZ



N - node number
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 3-D)

sign convention:
Point loads are positive in the direction of the axes.

(b) NLODI < 0 (this option only available for 2-D analysis)
Distributed pressure loading

L	N1	N2	T1	S1	T3	S3	T2	S2

see Figure in Page 3.23
- Linear strain triangles (LT = 2, 3)
- Linear strain quadrilateral (LT = 4, 5)

L	N1	N2	T1	S1	T3	S3	T4	S4	T5	S5	T2	S2

- Cubic strain triangles

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 shear 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.
Normal stresses - compressive stresses are positive.

<u>Record H3</u> (NFIXI records) - omit if NFIXI = 0 in record H1
(these are displacement fixities)

(a) for 2-D analysis See Figure in page 3.27

LNE	ND1	ND2	IVAR	IFX	0	0	0

- linear strain triangle (LT = 2, 3)
linear strain quadrilaterals (LT = 4, 5)

LNE	ND1	ND2	IVAR	IFX	0	0	0	0	0

- cubic strain triangle

LNE - element number

ND1)
ND2) - node numbers at the end of element side which is restrained

IVAR - the direction in which the side is restrained
1 - x direction
2 - y "

IFX - fixity code = 1

(b) for 3-D analysis see Figure in page 3.27

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 anticlockwise order

IVAR - the direction in which the face is restrained
1 - x direction
2 - y direction
3 - z direction

IFX - fixity code (= 1)

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/RE-START facility records E to H3 are omitted from the input data. Also IPRIM is set to 0 in record C1.

Record I (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

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
- 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
- a - out-of-balance loads
  - 0 - no out of balance loads
  - 1 - out of balance loads of 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

Note that IOUT is set to 0 if IOPT = 1

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 (record K2)

DTIME - time increment for consolidation analysis

ITMF - time increment option
  0 - time increment DTIME is equally divided 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.

Record J [(ICHEL-1)/10 + 1 records - only present if ICHEL > 0]

```
---------------------------------------
|  L(1)   L(2) .  . . . L(ICHEL)  |
---------------------------------------
```

L(1) etc. - list of element 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.

there must be 10 values per record (except for the last record in this group)

<u>Record K1</u> [(NOINC - 1)/10 + 1 records - only present if ILDF = 1]
where NOINC = INCB - INCA + 1

```
 ---------------------------------------
 | R(1)    R(2)   .    .    . R(NOINC)  |
 ---------------------------------------
```

R(1) etc. - the ratio of incremental loads or incremental prescribed displacements to be applied in each increment

there must be 10 values per record (except for the last record in this group)

<u>Record K2</u> [(NOINC - 1)/10 + 1 records - only present if IOPT = 1]

```
 ------------------------------------------------
 | IOUT(1)    IOUT(2)    .   .   . IOUT(NOINC)  |
 ------------------------------------------------
```

IOUT(1) etc. - the output options for each increment
Each value is a five digit code - abcde (see IOUT in record I; leading zeroes can be omitted.
Example : 00011 can be input as 11)

there must be 10 values per record (except for the last record in this group)

<u>Record K3</u> [(NOINC - 1)/10 + 1 records - only present if ITMF = 1]

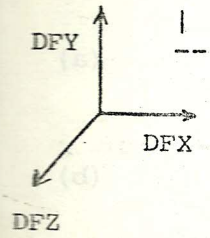
```
 ----------------------------------------------
 | DTM(1)    DTM(2)    .   .   .  DTM(NOINC)  |
 ----------------------------------------------
```

DTM(1) etc. - the time steps for each increment
(these are not ratios)

there must be 10 values per record (except for the last record in this group)

*Record L (NLOD records) - omit if NLOD = 0 in record I

(a) NLOD > 0 (Nodal point loads)



N	DFX	DFY	DFZ

Point loads are positive in the direction of the axes

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 3-D analysis)

(b) NLOD < 0 (this option is available for 2-D analysis only)

distributed (pressure) loading along element sides

L	N1	N2	T1	S1	T3	S3	T2	S2

- linear strain triangles, quadrilaterals

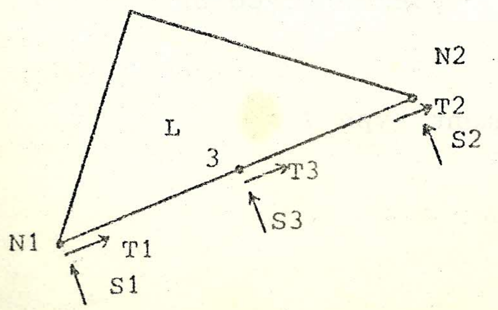
L	N1	N2	T1	S1	T3	S3	T4	S4	T5	S5	T2	S2

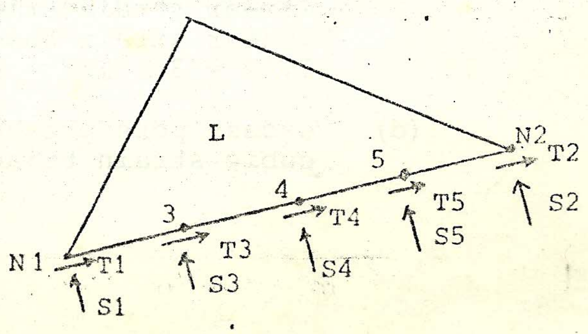
- cubic strain triangles

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 shear stress at N2
S2 - increment of normal stress at N2

sign convention for stresses:

Shear stresses which act in an anticlockwise direction about element centroid are positive.
Normal stresses - compressive stresses are positive (ie pressures which causes compression in element subjected to pressure, are positive)





Record M (NFIX records) - omit if NFIX = 0 in record I

(i) for 2-D analysis See page 3.27 (Figure)

```
-----------------------------------------------------------------
| LNE   ND1   ND2   IVAR   IFX   V1   V3  V2                     |   (a)
-----------------------------------------------------------------

-----------------------------------------------------------------
| LNE   ND1   ND2   IVAR   IFX   V1   V3   V4   V5   V2          |   (b)
-----------------------------------------------------------------

-----------------------------------------------------------------
| LNE   ND1   ND2   IVAR   IFX   V1   V2   0                     |   (c)
-----------------------------------------------------------------

-----------------------------------------------------------------
| LNE   ND1   ND2   IVAR   IFX   V1   V3   V4   V2   0          |   (d)
-----------------------------------------------------------------
```

LNE - element number

ND1,ND2 - node numbers at either end of element side which is fixed or has a prescribed variable.

IVAR - variable which is prescribed or fixed.
  1 - x displacement
  2 - y displacement
  3 - excess pore pressure

IFX * - fixity code
  1 - incremental value of variable
  2 - absolute value of excess pore-pressure

V1, V2 - prescribed values at end nodes
V3, V4, V5 - prescribed values at nodes along element 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 or 2

(d) excess pore pressure fixity
cubic strain triangle - element type 7

IVAR = 3 IFX = 1 or 2

(ii) for 3-D analysis see figure in page 3.27

LNE	ND1	ND2	ND3	ND4	IVAR	IFX	V1	V5	V2	V6	V3	V7	V4	V8	(e)
LNE	ND1	ND2	ND3	ND4	IVAR	IFX	V1	V2	V3	V4	0	0	0	0	(f)

- only for the 20-noded brick element

LNE - element number

ND1...ND2 - 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
 1 - incremental value of variable
 2 - absolute value of excess 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 or 2 or 3 IFX = 1

(f) excess pore pressure fixity
IVAR = 4 IFX = 1 or 2

* By re-specifying the record M, but with a fixity code of 0 (IFX = 0) previously specified displacement fixities can be released (ie the nodes are then free to move in the direction IVAR).

<u>Record N</u> (NFXB records) - omit if NFXB = 0 in record I

```
-----------------------------------
|    ND    IVAR   IFX   V         |
-----------------------------------
```

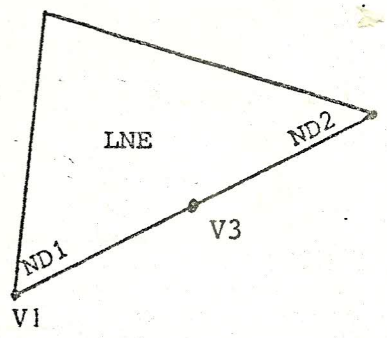
ND - node number

IVAR - variable that is prescribed or fixed
1 - x displacement
2 - y displacement
3 - excess pore pressure (for 2-D analysis)
z displacement (for 3-D analysis)
4 - excess pore pressure (for 3-D analysis)
5 - rotation (for 2-D plane strain analysis only)

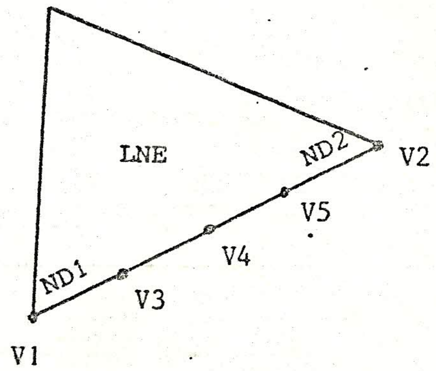
IFX - fixity code
1 - incremental value of variable
2 - absolute value of excess pore-pressure

V - prescribed value of variable

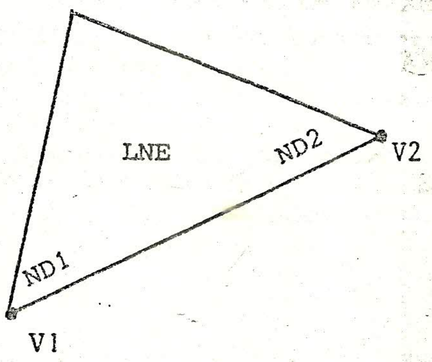
This option is provided to specify prescribed values to a variable or restrain it at a node on an individual basis. This is provided to supplement the side fixities (record M) and caters for fixities that cannot be handled by side fixities. ie fixing an individual node or giving it a prescribed value. This record should only be used to fix a beam element (type 12) from rotating or to apply a prescribed rotation to a beam at a node. Also can be used to restrain the nodes of bar elements (type 1).



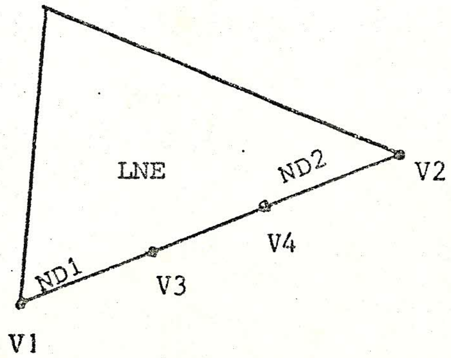
(a) LT = 2, 3, 4, 5



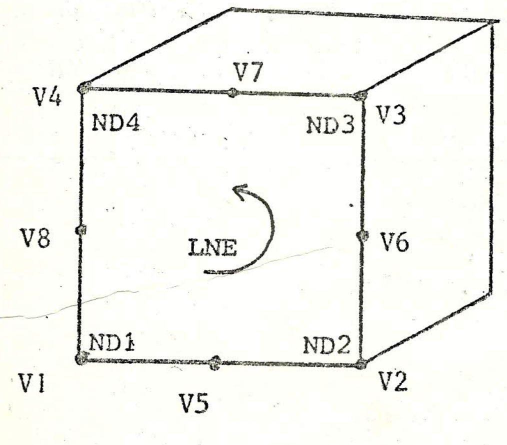
(b) LT = 6, 7



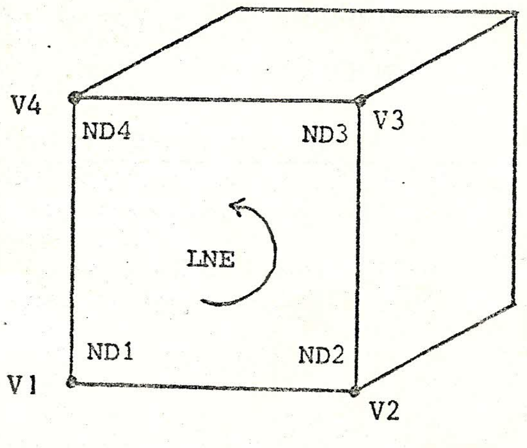
(c) LT = 3, 5



(d) LT = 7



(e) LT = 8, 9



(f) LT = 9

3.4 Data Summary

Geometry Program

Record Type	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	IRNFR
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 Type	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		NINCP
C4	1		INCLST(1) INCLST(2) ..... INCLST(NINCP)
D	NMAT		MAT NTY P(1) P(2) . . . P(12)
E	NM		L(1) L(2) . . . L(IPRIM)
F	1		INSIT NNI
G1	NIN		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 IOPT DTIME ITMF DGRAV
J	NCH		L(1) L(2) . . . L(ICHEL)

(contd)

Record Type	No. of Records		Data
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	NLOD		N DFX DFY DFZ or L N1 N2 T1 S1 T3 S3 T2 S2 or L N1 N2 T1 S1 T3 S3 T4 T5 S5 T2 S2
M	NFIX	2D	LNE ND1 ND2 IVAR IFX V1 V3 V2 or LNE ND1 ND2 IVAR IFX V1 V3 V4 V5 V2 or LNE ND1 ND2 IVAR IFX V1 V2 0 or LNE ND1 ND2 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

\+ NOINC = INCB - INCA + 1

NM = (IPRIM - 1)/10 + 1

NCH = (ICHEL - 1)/10 + 1

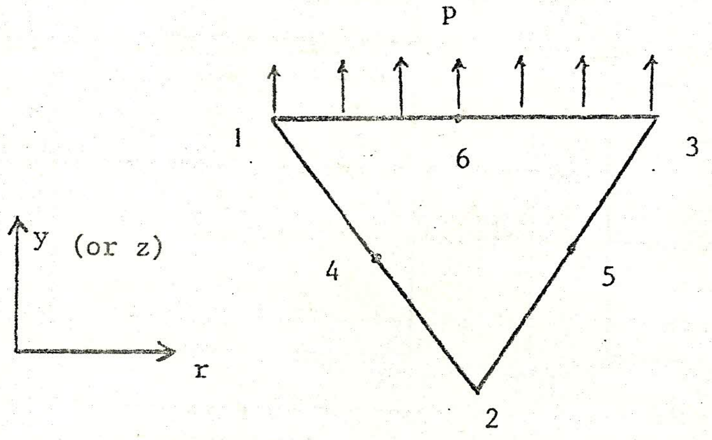
NS = (NOINC - 1)/10 + 1

2D : 2-Dimensional analysis

3D : 3-Dimensional analysis

Calculation of Nodal Loads Equivalent to Surface Loading

1) Axisymmetry



$$F_{y1} = \frac{\Pi p}{3} r_1 \cdot (r_3 - r_1)$$

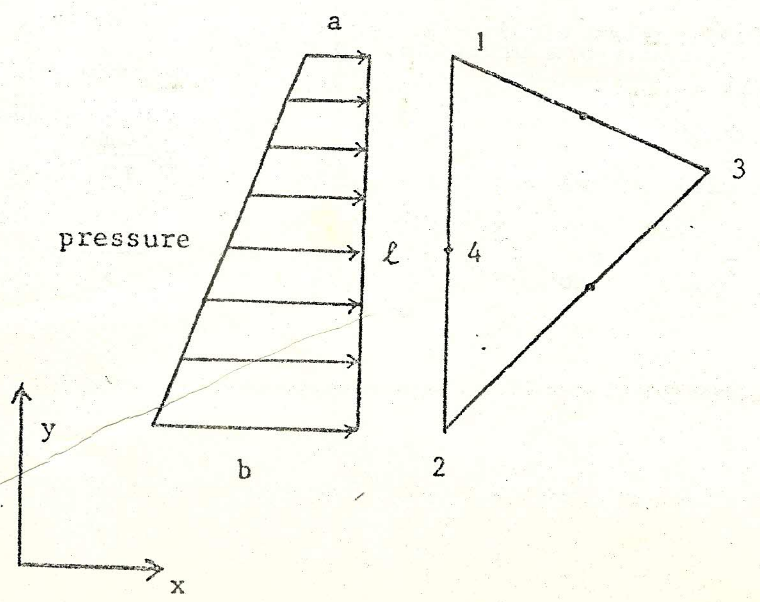
$$F_{y3} = \frac{\Pi p}{3} r_3 \cdot (r_3 - r_1)$$

$$F_{y6} = \frac{2\Pi p}{3} (r_3^2 - r_1^2).$$

by integrating

$$\int N^T\{p\}.dS$$

2) Plane Strain

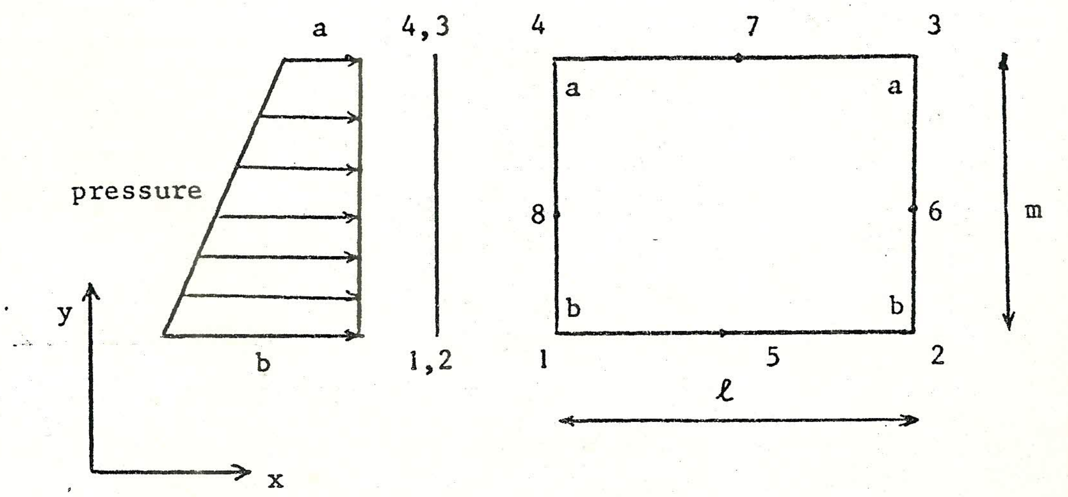


$$F_{x1} = \frac{a}{6} \cdot \ell$$

$$F_{x2} = \frac{b}{6} \cdot \ell$$

$$F_{x4} = \frac{1}{3} (a + b) \cdot \ell$$

3) Three Dimensional



$$F_{x1} = F_{x2} = (-)\frac{\ell m}{36}(2a + b)$$

$$F_{x3} = F_{x4} = (-)\frac{\ell m}{36}(a + 2b)$$

$$F_{x5} = \frac{\ell m}{9}(a + 2b)$$

$$F_{x7} = \frac{\ell m}{9}(2a + b)$$

$$F_{x6} = F_{x8} = \frac{\ell m}{6}(a + b)$$

if uniform pressure, $p = a = b$:

$$F_{x1} = F_{x2} = F_{x3} = F_{x4} = (-)\frac{\ell m}{12}p$$

$$F_{x5} = F_{x6} = F_{x7} = F_{x8} = \frac{\ell m}{3}p$$