Edwin
U. Federal de Paulo Ltda
Rua Soares Cabral 80, Ap 1301
Laranjeiras
CEP 22240-070
RJ RJ

# C R I S P  9 0

User's and Programmers's Guide

A. M. Britto

Volume 3, PC-386 Version

(1/6/91)

CONTENTS

# 1. INTRODUCTION

It is recommended that the users have a firm background in

* finite elements
* soil mechanics
* critical state soil mechanics theory

for using the CRISP program. They should have taken courses at post-graduate level in the above topics. The CRISP-90 program is specifically aimed at this particular group. It takes a few years of dedicated usage for users to acquire the ability to interpret the results obtained from a CRISP analysis.

This manual should be used in conjunction with the CRISP-90 brown manual - volume 1 (User's and Programmers's Guide). That manual deals with only the analysis parts of the program (GP and MP), and is the recommended starting point for reading. This manual deals with all other modules of CRISP-90 as implemented in PC-386 computers.

Chapter 2 deals with the analysis part of the program (GP and MP). This is to supplement, the information available in Volume 1 of the manual. It covers topics such as capabilities and limitations of CRISP-90, and also the assessment of results obtained. The later sections (assessment of results) should be read after the user has gained some experience in running some analyses (some of the examples given in appendix B of volume 1 and a few examples of your own choosing from published material). However these sections must be read before launching on a serious analysis which would be a design of a real structure. It is strongly recommended that the users evaluate the results from a CRISP analysis with alternate theoretical analysis,and where possible experimental results before it is put to such use. The authors of the program, Cambridge University, the Transport and Road Research Laboratory (Dept of Transport) disclaim any liability arising from errors in the program or erroneous interpretation of CRISP results.

Chapter 3 gives an over-view of all the modules in CRISP-90. It also explains which modules can be run in which environment. Some CPU time statistics, hardware requirement are also considered. Brief explanations of the new modules are also given.

Chapters 4 through 10 deals with each of the new modules as shown below.

Chapter 4 - super mesh program (SMP)

Chapter 5 - PRE-ADG program

Chapter 6 - Analysis Data Generation Program (ADG)

Chapter 7 - Geometry (GP) and Main Program (MP).

Chapter 8 - Post processing (PP) Program

Chapter 9 - CRISP-LOTUS 123 Interface program (CL)

Chapter 10 - Stress Path plotting program (SP)

# 2. ANALYSIS

## 2.1 Introduction

The information given in this chapter is intended to supplement the information given in CRISP-90 (brown) manual Volume 1, for the analysis part of the program (GP and MP). This section should be read by the user before embarking on any serious CRISP analyses. However some familiarity with CRISP analysis is expected in understanding the points raised in this section.

## 2.2 Capabilities and limitations of the CRISP-90 Main Program

CRISP-90 is a finite element program which is able to perform drained, undrained and time dependent analysis of static problems (not dynamic) under monotonic loading/unloading conditions. It is not suitable for stress cycling in its present form. The program uses an incremental approach without any stress correction (with the exception of the elastic perfectly plastic models - no. 5). The program therefore does not use any iterative procedure. The program uses the small strain small displacement approach and hence it is not suitable for large strain analysis.

Plane strain, axisymmetric and three dimensional analysis can be carried out; however the axisymmetric analyses are limited to loading that is also axisymmetric, hence torsion tests cannot be analysed using the axisymmetric option.

In its present form, CRISP cannot model the driving of a pile. However, it can model the behaviour of a pile which is already in place.

CRISP uses the incremental approach without any stress correction at the end of increments for Cam clay analysis and if the number of increments are not sufficient the response would drift away from the true solution. However for the elastic perfectly plastic model (model no. 5), when yielding occurs the stress state is corrected back to the yield surface at the end of every increment. The un-balanced load due to the stress correction is re-

applied in the subsequent increment. Additionally at the end of every increment the strains are sub-divided into smaller steps and the stress state is re-evaluated more accurately (see section 2.5.4.2). Ten steps are used for each increment. Therefore, any analysis which only uses model 5 (and the elastic models 1 and 2) would require fewer increments than would a Cam-clay analysis.

However, if both Cam clay models and elastic perfectly plastic models are employed in the same analysis, the number of increments required will be governed by the Cam-clay behaviour ie. one needs to use the same number of increments as in an analysis with only Cam clay models.

It should be pointed out that it is up to the user to decide whether the models that are available in CRISP-90 are adequate to represent the soil behaviour he/she comes across in a real field situation. Some of the deficiencies of the models are :

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

(2) Because the Mohr-Coulomb model uses the associated flow rule, the dilation predicted at yield may be unrealistically high.

(3) The current soil models in CRISP are not suitable for analysing partially saturated soils.

CRISP produces a large amount of output. Engineering judgement has to be exercised in interpretting the results produced by the program. Questions one should ask include :

(1) Does the deformation pattern look reasonable?

(2) Do the stress paths behave in an acceptable manner?

(3) Are the principal stress and strain directions consistent with the applied loading?

(4) Are the distribution of pore pressures acceptable?

(5) Are the distribution of the strains reasonable?

Plot the zones of yielding if you are using elastic-perfectly

plastic models or the Cam clay models. Zones of yielding should be concentrated around areas subjected to loading and/or unloading. However results may be subject to suspicion if you find that plastic zones are developing far away from loaded regions.

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

*2.2.1 Literature survey*

It is very difficult to assess the results of a CRISP analysis if you have not used CRISP for this particular type of analysis before, and even great if you have not used the CRISP program before. There are various means to assess how reliable a CRISP analysis is.

Some of these suggestions need to be considered even before the analysis is carried out. From published references seek out any results of analysis of similar problems. Are there sufficient number of publications in this area? (See the section on reference which by no means is complete). If there are, then it gives one more confidence because others have used the CRISP program to carry out similar analysis and compared it with field data or laboratory tests (centrifuge tests) or simply used it in a parametric study. This is the case for retaining wall, embankment construction analysis. In contrast if it is difficult to find any reference on a particular type of problem, it could either indicate that one is trying to use CRISP for a problem which it has not been used in the past or for a problem for which CRISP is not particularly suitable.

The second suggestion regards the suitability or unsuitability of the soil models that are available in CRISP to represent the soil (as encountered in the field, or remoulded to carry out laboratory tests). One needs to carry out a literature review to see what others have used to represent the same soil. Care is needed to ensure that the soil parameters reported applies to the situation you are examining. Two entirely different types of structures on the

same type of soil can lead to completely different types of behaviour. For example construction of an embankment is likely to cause the soil to behave in an entirely different manner to excavation behind a retaining wall on the same type of soil. The soil behaviour could be stress path dependent. The chosen soil parameters for a Plane Strain analysis could be different from that for an axisymmetric analysis of the same soil. Carry out a single element test by subjecting a sample of soil to stress paths it is likely to encounter in different parts of the structure. Compare it with laboratory test data.

If the soil representation is not adequate, however carefully one carries out the analysis, the results are bound to be unsatisfactory.

## 2.3 Mesh, No. of elements and increments

The other important aspects which depend on how a CRISP analysis is carried out are listed below:

a) Number of elements used in the finite element mesh.
b) Number of increments used in the analysis.

### *2.3.1 Number of elements*

Both these aspects involve 'experience' and one gains this as one carries out more and more CRISP analyses. For the benefit of new users Figures 2.1-6 illustrate typical finite element meshes. Even though the actual number of elements used in the mesh is important the critical point is how these are deployed. Regions which are subjected to loading or unloading are of main interest. One should concentrate a good percentage of the finer (smaller) elements around this region and around any stress concentrations that are anticipated. Then use slightly larger elements (making the transition gradual) as one moves away from the region subjected to loading/unloading.

Somewhat large elements can be used at bounderies distant from loaded regions. Rapid variation in stresses and strains and displacements takes place around loaded region. When preparing the

mesh one needs to ask the question whether the chosen element type and size of the element are adequate to represent this variation. This is illustrated if one considers a simple example which is analysed using constant strain triangle (this element is not used in CRISP). The example considered is the thick cylinder problem. Figure 2.7 shows the results when different number of elements are used. These results are not from CRISP which does not have constant strain triangles, and are used for illustrative purposes.

In general about 100 to 200 type 2,3,4 or 5 elements would be sufficient if these are deployed sensibly giving a graded mesh. More elements may be needed if the problem analysed is a complex one (geometrically or in terms of stress-strain behaviour). Very rarely does one come across a problem which requires 1000 or more elements (of type 2,3,4,5).

*2.3.2 Number of increments*

The following suggestions are for analyses which involve either elastic-perfectly plastic models or critical state models. Any analysis which uses purely linear elastic models does not require more than one increment and the following discussion does not apply to them. However if you mix linear elastic models with other soil models then the following suggestions still hold.

In contrast to iterative methods, the incremental method tends to give a response which continually drifts away from the true response where the stress strain behaviour is non-linear.

If one is analysing a problem where one has no prior experience with CRISP, then one is recommended to run at least two analyses with sufficiently different number of increments for the same finite element mesh. Then compare the results from the two analyses. For example if the analysis is that of the settlement under a flexible footing then plot the variation of central settlement (or average settlement) against loading for both analyses. Compare the final settlement. Are these significantly different. If they are then ignore the results from the analysis which uses fewer increments and repeat the analysis with twice the number of increments as used in the second analysis. Again repeat the procedure of comparing the

results. If a unique solution exists to the problem one is analysing, then using more increments should converge towards this solution.

Figure 2.8 shows the effect of using more increments with smaller load steps.

If the results from two different analyses with significantly different number of increments give results which are sufficiently close (within a few percent) then the number of increments you are using for the analysis is adequate.

*2.3.3 Where to place the outer boundary?*

The outer boundary should be placed as far away as possible from the region subjected to the largest change in loading so as not to influence the results. Consider the analysis of a retaining wall. If H is the height of the retaining wall, then, as a rule of thumb, place the vertical boundaries at a distance of 4 to 5H from the wall. The same applies to the base of the mesh. However if a hard stratum is encountered at a depth less than this, then the base of the mesh can be taken at the depth at which the hard layer is encountered. Then one tacitly assumes that the displacements that occur in the hard layer due to the structure above is negligible.

In setting up the mesh make use of any axes of symmetry, if these exist (see Figure 2.9).

*2.3.4 Whether the outer boundaries should be smooth or rough?*

If these outer boundaries are placed far enough so as not to influence the results then these can be treated as rough.

## 2.4 Types of analyses

The following type of analyses are possible with the CRISP program.

(a) Drained analysis (1,2,4,12,13). (6) (8)

The numbers defined above are the element type numbers that can be present in the same mesh. These are separated into 3 groups. The element types from one group cannot be mixed with element types from

another group. For example element types 12 and 6 cannot be present in the same mesh.

Specify effective stress parameters : $E'$ $\nu'$ $K_w = 0$

No change in the pore pressures takes place during the analysis. If you have specified in situ pore pressures with the in situ stresses, then these are treated as static head and hence remain unaltered during the analysis.

(b) Undrained analysis (1,2,4,12,13) (6) (8)

There are 2 choices. One could carry out an analysis in terms of effective stress parameters.

$$E' \quad \nu' \quad K_w \neq 0$$

This gives changes in effective stresses and the pore pressures separately.

The alternative is to specify total stress parameters :

$$E_u \quad \nu = 0.49 \qquad K_w = 0$$

A value of Poissons ratio close to 0.5 ensures that the analysis is undrained. The changes in the stresses during the analysis should then be treated as changes in total stresses. The program will not print any changes in pore pressures. So with this type of analysis it is not possible to separate the changes in effective stresses from changes in pore pressures.

The following combination of parameters is not permitted :

$$E_u \quad \nu = 0.49 \qquad K_w \neq 0$$

This is like specifying undrained behaviour twice over. It is likely to result in pore pressures which are unreliable.

(c) Time dependent or consolidation analysis
(1,2,3,4,5,12,13) (6,7) (8,9)

use effective stress parameters : $E'$ $\nu'$
Also specify appropriate permeabilities $k_x$ and $k_y$.

It is permitted to model zones or regions within the mesh which behave either in a drained or undrained manner as described in (a) and (b). Here element types 2,4,6,8 will be used for these inclusions or regions.

In a retaining wall analysis (2-dimensional) type 5 can be used for the soil and type 4 for the wall. Since one is only interested in the total stress changes in the wall specify $K_w = 0$ for the wall.

## 2.5 Equilibrium check and out-of-balance loads

### *2.5.1 What is an equilibrium error?*

In the following sections, the terms a) equilibrium errors, b) equilibrium check c) out-of-balance loads will appear frequently and is important that the user understands in what context these are being used and the exact meaning without any ambiguity.

### *2.5.2 Equilibrium check*

This is a check carried out 1) at the in situ stage and 2) at the end of every increment of the analysis. This is a check carried out by the program to ensure that externally applied loads are in equilibrium with the internal stresses. A more detailed mathematical explanation is given in appendix H of CRISP (brown) manual volume 1.

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

For comparison purposes the externally applied loads and the internal (element) stresses are converted (using numerical integration) into equivalent point loads for all the nodes in the mesh. These are the quantities which are compared. There is one set for the external loads $\{ P_t \}$ and one set for the internal stresses $\{ P_{sn} \}$. The largest value in the x and y directions are picked up from $\{ P_t \}$. These are the values printed under the heading 'MAXIMUM EXTERNAL LOAD' in the MP output. The difference $\{ P_{err} \} = \{ P_t \} - \{ P_{sn} \}$ is calculated for all the nodes. If equilibrium is satisfied (as it should be) then $\{ P_{err} \}$ should be nearly zero at all the nodes. However if one finds that these out-of-balance loads are not

zero then it would indicate data errors. The user must check the data input in records D, F, G1, H1, H2 or H3 to find the mistake.

$$\frac{\{ P_{err} \}_{max}}{\{ P_t \}_{max}} \times 100$$

is expressed as a % and printed under the heading 'PERCENTAGE ERROR IN EQUILIBRIUM'.

It should be less than 1% at the in situ stage (but usually much smaller). This level of equilibrium must be maintained throughout the analysis for the results to pass the first test of acceptability (only true for analysis using models 1, 2, 3, 4 and 5. NOT 5) → elastic perfectly plastic.

*2.5.3 Is the equilibrium check a sufficient condition ?*

The answer to the above question is an emphatic NO. The equilibrium check is just that and nothing more. It does not automatically confer on the results the label of 'acceptability'.

If one takes time at this point to quickly scan through the MP output listed in appendix B (CRISP brown manual, volume 1) and look at all the 'equilibrium checks' for the 4 example problems they will find one thing in common. The equilibrium is satisfied to less than 1% which appears to have met the condition mentioned above.

However if one looks at Fig. B.2 (Page B.6 of CRISP manual volume 1.) one will notice that the results are incorrect if only 6 increments are used in the analysis of a triaxial test.

The point illustrated here is that the obtained results can be erroneous even if the equilibrium check is satisfied. The example analysed is one of the simplest kind encountered in soil mechanics where uniform stress conditions prevail. Whereas most real field situations are more complex.

The error in the final q, p' and u values are given below.

| | p' | q | u |
|---|---|---|---|
| Theory | 106.8 | 106.8 | 78.7 |
| CRISP | 125.0 | 117.6 | 64.4 |
| error | 19 % | 17 % | 10 % |

The user should NOT conclude that:

(a) using about 50 increments is adequate for any type of analysis which involves Cam-clays (which may be a complex problem).

(b) using about 6 increments will result in only an error of 10-20% in any analysis which involves Cam-clays.

These conclusions are not valid and these points cannot be over-emphasized.

*2.5.4 Equilibrium check - During Analysis*

*2.5.4.1 Analysis using elastic and Cam-clay models*

CRISP program uses the tangential stiffness approach in an incremental method. It is assumed that the user sub-divides the whole analysis into sufficiently small load-increments. It is also assumed that the difference between the piecewise linear assumption and the true response is negligible. Since there is no correction made to the stresses at the end of the increment, there should not be any out-of-balance or residual stresses (or loads). Therefore there is no need for any iteration. The foregoing discussion is true when using Cam clay models in a CRISP analysis. The statements made above are specific to CRISP and are not necessarily true of all finite element programs.

Large equilibrium errors (a large % of out-of-balance loads) indicate that there is something wrong with the analysis. Under these circumstances it is wrong to re-apply these loads to the increments which follow, thereby propagating the effects of errors.

*2.5.4.2 Analysis using elastic perfectly plastic models (no. 5)*

If one is using the elastic perfectly plastic models in a CRISP analysis, when yielding occurs and the stress state moves outside of the yield locus/failure surface (point 1 in Figure 2.10), and

stresses are corrected back to the yield surface (point 2). In Figure 2.10 this correction is denoted by $\Delta\sigma_{c1}$ for a single integration point. When this is converted into a set of equivalent nodal loads denoted by $\Delta P_{c1}$, this represents the out-of-balance load. Out of the applied $\Delta P_1$ load only $(\Delta P_1 - \Delta P_{c1})$ is satisfied by the internal stresses. In a finite element program that uses an iterative technique, the load $\Delta P_{c1}$ is re-applied and the resulting incremental displacements are added to the current displacements. If during the application of $\Delta P_{c1}$ further yielding takes at some integration points then a second set of out-of-balance loads, $\Delta P_{c2}$ are calculated and the above procedure is repeated until the resulting incremental displacements or $\Delta P_{cn}$ is less than a preset tolerance. This is represented in Figure 2.11.

In CRISP, instead of calculating the incremental displacements due to the out-of-balance loads separately, the loads are carried over to the next increment and added to whatever loading is applied in that increment. Therefore the applied load increment in the next increment is $\Delta P_2 + \Delta P_{c1}$. This is represented in Figure 2.12.

In a CRISP analysis which uses elastic perfectly plastic models at the end of each increment, the increase in strain is divided into ten equal steps. Then the stress state is re-evaluated at the end of each step. This is illustrated in Fig. 2.13. Therefore the stress state calculated is closer to the true solution. This technique is only used for model no. 5 and hence any analysis which uses the elastic perfectly plastic models would require fewer increments than a Cam clay analysis.

If, at the end of an analysis, there are out-of-balance errors greater than about 1% you may have to add a few more increments than you had originally intended at the end of the analysis but without any further loading. This will ensure that the residual out-of-balance loads are re-distributed ensuring that equilibrium is satisfied. Then the displacements at the end of the analysis will be consistent with the applied loading. The stop-restart facility will be useful in this respect.

*2.5.5 In situ stage*

Equilibrium errors at in situ stage would indicate that the in situ stresses specified by the user are not consistent with the in situ boundary condition specified. It could mean either of the following.

(a) The user may have forgotten to restrain/fix the sides of some elements which lie along the mesh boundary. Check the displacement fixities of the left and right hand boundaries and the base of the finite element mesh.

(b) The in situ total vertical stresses specified by the user are not consistent with the bulk unit weight specified with the material properties or GRAVI (in record H1 in MP input data) has been specified incorrectly. If the current analysis is that of a field situation and if the in situ vertical stresses include the effects of self-weight then GRAVI must be set equal to 1. If the analysis is that of a laboratory sample and the vertical stresses are uniform at in situ stage then GRAVI = 0.

At each reference point (record G1) calculate the total vertical stress by summing the effective vertical stress and in situ pore pressure and check that this is equal to the bulk unit weight if the soil (PR(8) in record D) multiplied by the depth of the reference point. Repeat this for all the reference points.

If there is more than one type of soil present in the mesh with differing bulk unit weights then make allowance for that as shown below :

Consider reference point 3 in Fig. 2.14

$$\sigma_{v3} = \gamma_1 \times h_1 + \gamma_2 \times h_2$$

Where $h_1 + h_2$ is the depth of reference point 3. If we consider reference point 4 then

$$\sigma_{v4} = \gamma_1 \times h_1 + \gamma_2 \times h_2 + \gamma_3 \times y$$

The sum of the effective vertical stress and the in situ pore pressure for these reference points should equal the values given by the above equation.

The following assumptions are made in the above example.

(a) no surcharge is acting at in situ stage on the ground surface.

(b) GRAVI = 1. ie the example considered is that of a field situation.

(c) the soil is fully saturated.

The following have no bearing on the equilibrium check carried out at in situ stage and hence should not be used as an excuse to explain away any out-of-balance loads.

1) The problem analysed is that of a soil-structure interaction problem.
2) only single precision is used in the calculations.
3) soil model used is elastic perfectly plastic (model no. 5).

The equilibrium condition at in situ stage must be met unconditionally. If it is not the case then

DO NOT PROCEED WITH THE ANALYSIS

The results will be wrong anyway.

Irrespective of what type of soil model is used or whether the analysis is that of a soil-structure interaction problem (with or without a very stiff structure) equilibrium must be satisfied at in situ stage. The precision used in the calculations do not matter at in situ stage. Neither the choice of soil models nor the presence of a very stiff structure in any way affects the equilibrium condition from being satisfied at in situ stage.

In any analysis where in situ stresses are specified it is essential that one starts off with the level ground surface. In situ stress specification provided in CRISP (option 1 in record F) only caters for this. Therefore if the starting point of the analysis has a irregular ground surface or a structure already present it is not possible to use that as the in situ stage in a CRISP analysis.

The following examples are used to clarify this situation.

(i) embankment construction

In the geometry program the whole region (including the embankment) is meshed in the usual manner (Fig. 2.15(a). However (in the MP input data) at in situ stage the elements representing the embankment are removed to leave a horizontal surface for which in situ stresses are specified without any problem (Fig. 2.15(b)). Then build/add the embankment, layer by layer (if the descritisation used allows for that) until the whole embankment is built (Fig. 2.15(c)). Then use this as the starting point for the analysis.

There are a few snags in using this approach :

(1) One may not end up with the same stress distribution one would have wanted to specify in the first place after the embankment has been constructed.

(2) During the construction of the embankment the foundation is subjected to displacements and strains one was not interested in and these have to be subtracted from the end results, for interpretation.

(3) It also makes it impossible to model the embankment using Cam clay models. Because the embankment is not present in the mesh in increment 1, it cannot be specified a stress history (ie $p'_c$, initial stresses etc).

(ii) excavation behind retaining wall.

The mesh should cover the whole area including the region of soil in front of the wall which is going to be excavated (Fig. 2.16(a)). If the retaining wall is of the same bulk unit weight as the soil or if the difference can be ignored, the wall can be in place at in situ stage. For a realistic analysis the permanent slab and the soil it replaces should be modelled by 2 separate sets of elements occupying the same region and connected to the same set of nodes. If the bulk unit weight of the wall is different from that of the surrounding soil then again use 2 sets of elements to model the region occupied by the wall. One set to model the soil (before the wall is placed) and

the other for the wall. These 2 sets of elements should be connected to the same set of boundary nodes (ie the nodes along the boundary which define the region, in this example, the wall). Different material zone numbers should be assigned to the 2 sets of elements (see appendix I section 5 of CRISP-90 brown manual - volume 1).

If one is trying to analyse a retaining wall problem using the procedure outlined above one should note that it is not possible to use SMP, PRE-ADG and ADG programs to generate a mesh with super-imposed elements. The above programs can only handle one set of elements occupying any given region. The user will have to modify the input data to the GP and MP to incorporate super-imposed elements. This approach is littered with pit-falls and it is easy to introduce errors into the data files. Therefore not recommended to new users.

Procedure :

In situ stage

Remove the elements representing the wall and the permanent slab. Soil elements occupy the whole region at the beginning and the ground surface is horizontal (Fig. 2.16(b)).

Increment Block 1 - Remove soil and install wall

The soil elements occupying the wall region are removed and the wall is placed in position (Fig. 2.16(c)).

Increment Block 2...6 (say)

Excavate the soil in front of the wall in stages (layer by layer). In this example it is done over increment blocks 2 to 6. The last layer could be the excavation of the region where the permanent slab is placed (Fig. 2.16(d)).

Increment block 7

Permanent slab is cast. Elements representing the permanent slab are added (Fig. 2.16(e)).

Each of the increment blocks can have any number of increments.

The number of increments depend on the thickness of the layer being removed. The thicker the layer removed in an increment block the more increments that will be required. Typically 5 to 10 increments per layer should be adequate in general.

*2.5.6 Equilibrium check - during analysis (continued)*

Out-of-balance loads during the analysis can be present if model 5 is used and yielding is occurring. As explained above, these are legitimate out-of-balance loads. This should not give rise to any concern. Only point that should remembered is that the % out-of-balance should be kept to within reasonable limits (less than about 15-20 %), when elastic perfectly plastic models are used. This only applies to the actual analysis (increment 1 onwards) and not to the in situ stage.

The other form of equilibrium error arises from numerical problems (ill-conditioning). Let us consider the following matrix equation

$$P = K u$$

P - is a column vector (loads)
K - stiffness matrix
u - unknown displacements

$$P_{err} = P - K u$$

After solving for the unknown nodal displacements u is multiplied by K and compared with P. If the difference is expressed as a % of the largest applied load

$$\% \text{ error} = [ P - K u ] / P_{max}$$

When the above matrix K is well conditioned then P - K u is negligible, as is the % error. However if the matrix K is ill-conditioned then % error can be significant. This may happen in CRISP when very stiff structures are buried in soft soils. This is unacceptable and the whole point of the equilibrium check is to guard against such an event. The equilibrium check carried out at

the end of every increment is to trap this type of error from creeping into a CRISP analysis.

As explained above there are two sources of out-of-balance loads

(a) In an analysis which uses elastic perfectly plastic models (model no. 5) yielding and the resultant stress correction gives rise to out-of-balance loads. This is permitted. At the beginning of an analysis where the initial state is elastic and during the analysis until yielding takes place there should not be any out-of-balance loads. During elastic behaviour no stress correction is carried out, because there is no need.

(b) When very stiff structures are present in a soft soil, out-of-balance loads may result due to numerical problems. Out-of-balance errors can also occur in a consolidation analysis where an inappropriate time step has been used. Any out-of-balance loads from this source is unacceptable.

From the foregoing discussion it is obvious that numerical problems in the presence of soil model 5 would be hard to detect. The out-of-balance loads due to ill-conditioning can be masked by the out-of-balance loads due to yielding and stress corrections. If you suspect this then replace the model 5 by a linear elastic model (example model 1) and re-run the analysis. If significant out-of-balance loads are still present that would indeed indicate a numerical problem.

If there is a large difference in the stiffness of the soil and structure in a soil-structure interaction analysis this could lead to a precision problem. This can be investigated by re-running the analysis by reducing the stiffness of the structure. If the equilibrium errors disappear then inadequate precision is the problem. One will have to carry out the analysis, where actual stiffnesses are used for structure and soil, using double precision.

It should be pointed out that not all soil-structure inter-action

problems automatically run into numerical problems. However the user should be on the lookout against such an event.

Significant out-of-balance loads (resulting in equilibrium errors > 1%) during the course of an analysis (ie increment 1 onwards) when models 1, 2, 3, 4, 6 are used would indicate 'numerical problems' and no other. This is assuming that there were no equilibrium errors at in situ stage.

*2.5.7 For analysis using soil models 1, 2, 3, 4 and 6*

The equilibrium condition being met at in situ stage and throughout the analysis is a pre-requiste for a CRISP analysis to yield a acceptable solution. This simply means that numerical problems may be absent. However this condition alone does not guarantee that the results produced are acceptable.

It is possible to carry out an analysis with an ill-suited soil model (to the problem being analysed), with insufficient number of increments and an inadequate mesh (too few elements or elements deployed inefficiently) and still end up with an analysis which satisfies the equilibrium check throughout the analysis.

Therefore the compliance of the equilibrium check does NOT justify :

a) the appropriateness of the soil model.
b) the number of elements used and how these are deployed.
c) the number of increments used.

## 2.6 Assessment of Cam clay analysis (using models 3, 4 and 6)

CRISP uses the incremental method with the tangent stiffness approach. If a sufficiently large number of increments (small load steps) are not used, then the obtained results will continuously drift away from the the true response. It is not possible to estimate the amount of drift beforehand. Therefore the following suggestion is given.

Users embarking on any new analysis need to carry out at least two different runs with different number of increments for the same mesh and then compare the results. Here it is assumed that the finite element mesh one is using is adequate. See Fig.2.1-6 for some

typical meshes used. If these results are different by more than 5% then they need to repeat the procedure with increased number of increments until the results are within about 5%.

*2.6.1 Yield ratios (YR)*

In order to assess the size of the increment in any Cam clay analysis the parameter YR (yield ratio) was implemented in CRISP. It estimates by how much the yield locus has changed in size in a single increment. See section 5.2 (pages 5.12-13 of volume 1 of the manual) for an explanation of the yield ratio. YR is calculated for each integration point for all elements which were modelled as one of the critical state models (models 3, 4 and 6). This parameter was printed if the option for 'printing additional parameters for Cam-clay models' has been specified by the user (see record I, parameters IOUT). However since the printing of this information was dependent on whether the user requested it, this information was mainly overlooked or ignored. Because CRISP produced enormous amount of output and users were more interested in displacements and stresses, the yield ratios were rarely examined.

Looking at element centroids or just a range of elements never fully reveals what is happening. With this in mind a summary table was introduced with CRISP-90. This is automatically printed (the user cannot suppress this output) for each increment. It counts the number of integration points under different categories and also different ranges of YR values. Only integration points with Cam clay model behaviour are considered and no distinction was made between the 3 different models or different material zones. None of the other models was included in this summary. This gives an idea at a glance of the size of the loading in any given increment.

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

about the previous 99 increments.

Ideally most of the integration points which are yielding/hardening should have a yield ratio value close to 1.0 (less than 1.01).

The use of YR can be illustrated with example problem number 1 (see appendix B, volume 1 of manual). Presented is a strain controlled undrained triaxial test where 3% axial strain is applied to the sample in 6 equal increments. The obtained results are compared with an analysis of about 50 increments and with the theoretical solution. As can be seen, the analysis with 6 increments overestimates the deviatoric stress q at yield. This example is used to illustrate that errors can occur if sufficient number of increments are not used.

Let us examine the results obtained in terms of YR. Because of the uniform nature of the triaxial test, the stress state at any given integration point is representative of the behaviour of the entire mesh. However in an analysis of a field problem (example : retaining wall analysis) the behaviour at a given integration point will be different from the behaviour from any other integration point. See Figure 2.17 for example.

Looking at increment 1 the value of YR is less than 1.0 (0.844). A value of less than 1.0 means that either the stress state is elastic or it is undergoing softening with the size of the yield locus reducing. In the case of elastic behaviour the value of YR indicates how close it is to the yield locus. A stress state close to the yield locus will therefore give a YR value of nearly 1.

| Increment Number | Yield Ratio (YR) | |
|---|---|---|
| 1 | 0.844 | |
| 2 | 1.125 | ← |
| 3 | 1.023 | |
| 4 | 1.012 | |
| 5 | 1.007 | |
| 6 | 1.005 | |

In increment 1 the YR value is 0.844. In increment 2 the transition from elastic to plastic state occurs. The value of 1.125 indicates that the current yield locus is 12.5% larger than the previous. The value is much higher than the recommended upper limit of 1.05 and way out from the value of 1.01 which would be needed for accurate prediction.

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

## 2.7 Advice to new users

The users are expected to be familiar with the principles of soil mechanics and have followed lectures in soil mechanics at undergraduate level. Understanding of critical state soil mechanics is essential if using Cam clay models in the analysis. It should be remembered that these are hypothetical models.

Start with an elastic analysis for which there exists a well known solution ie settlement under a strip footing on a elastic medium. Compare the finite element results with the theoretical solution. If the results are significantly different then check whether you using sufficient number of increments.

Then try simulating the Terzaghi one-dimensional consolidation analysis (see section 9.1 of CSSMVFE). Again use a single element (8 noded, type 5). The first thing you will notice is that to analyse a time dependent problem you need to use a different element type to that for the drained/undrained type of analysis. The element used for a consolidation analysis has excess pore pressure as an additional variable, at some of the nodes (type 5).

Following the example given in appendix B of the CRISP manual volume 1, use a small time step of 1 sec for the first increment block in which the loading is applied. The increment block 2 consists of 7 increments and you will notice that the time steps are

increased gradually (see note in chapter 4 of the book CSSMVF2 on how to choose these time steps). Also note the pore pressure boundary condition applied to the top surface. The application of 100 units of pressure results in an excess pore pressure of 100 units everywhere within the mesh. In this analysis the sample is assumed to drain only from the top surface. Hence the excess pore pressures should be 0 at the top. This is achieved by specifying either -100 with fixity code 1 or 0 with fixity code 2. Both have the same effect for this particular problem. Use of the different pore pressure fixity codes are explained in Chapter 4 (Pages 4.68-4.70 of the CRISP brown manual - volume 1). Do not worry unduely about the pore pressure fixities at this stage.

Draw the theoretical settlement vs time plot and compare it with the CRISP results.

Try re-running the consolidation analysis with about 20 increments in the second increment block. Compare the results with the previous analysis.

Plot the pore pressure distribution within the sample for a selected time factor (say 0.5). The 8 noded elements use a 3 x 3 integration scheme. The pore pressures are printed under 2 headings: Under Nodal displacements (DX, DY) and excess pore pressures (DU). Both incremental and cumulative values are printed. Only the excess pore pressure at the top and bottom can be found for this example because there are only nodes at the top and bottom of the mesh. The other set of pore pressures which are printed under the heading 'General stresses' for each integration point or at element centroids. From the co-ordinates given establish the location of the integration points (see Fig. 4.5 of the manual volume 1). Here you have values of pore pressures at 3 different depths within the mesh. How does this compare with the theoretical distribution of the pore pressures. Is it sufficient to use a single element to model the consolidation analysis?

Here is an indication of modelling the spatial variation of pore pressure by a linear variation. We have run into a problem where the number of elements used is not sufficient in the y

direction (ie the direction of variation of the pore pressure). There is no variation of pore pressure along any horizontal line. Therefore it is sufficient to use only a single element in the X direction. Compare the results with Fig. 3.21 given in CSSMVFE.

Now try a mesh with more elements in the vertical direction (and graded in the same manner, smaller elements nearer the top drainage boundary) and see whether the results shown in Fig. 3.21 could be improved upon. What conclusions do you come to in terms of

a) Increasing the number of elements
b) Increasing the number of increments.

*2.7.1 Cam clay analysis*

Start with the simplest type of analysis, for example, a triaxial test (drained and undrained type). A single 8 noded quadrilateral element (type 4) is sufficient for this purpose. Try to reproduce the response of the undrained triaxial test example given in appendix B where 2 Cubic strain triangles were used. Try 2 different analyses with 6 and 50 increments but increasing the total strains from 3% to 5%. If you are familiar with the Critical state theory, calculate the theoretical end point (on the critical state line) of the analysis. Compare the values of q, p' and p and u and voids ratio at the critical state with the results obtained from the CRISP analysis.

If you are using the program for the first time, the best place to start is the examples given in the appendix B of the CRISP manual (volume 1). Familiarise yourself with the input data (refer to chapter 3 of the manual). Also look at the output. Try running the example of the undrained triaxial test with about 50 increments and compare the results with that shown in Fig.B.2 Now try running an analysis of a DRAINED triaxial test. You need to make only one change to the MP input data and no changes to the GP data.

Look up the the material property table (record D) and the 7th material property has the value of 5.E5 which represents the effective bulk modulus of water. By replacing this value by zero you make the response a drained one. No changes in pore pressures take

place. If you have specified a initial pore pressure then it remains the same throughout the analysis.

Now that you are familiar with the use of the CRISP program in terms of drained, undrained and time dependent (consolidation) analyses you can attempt a more difficult problem. At this stage try tackling problems for which you have a independent solution.

Remember never to over-reach yourself in terms of attempting a complex analysis at the initial stages. Avoid any analysis which includes slip elements. These are only for the more experienced users. Use this element type only if absolutely necessary. You can get by in most cases without applying slip elements.

It is important to examine the printed output from the Main Program. This is the most important step in assessing the reliability of the results obtained.

First of all check the output for any data errors that may have been picked up by the program. The program attempts to carry on with the analysis when non-fatal errors encountered. For example in specifying the nodal fixities, node numbers have been entered incorrectly. The program only crashes due to fatal errors.

For Cam clay analysis look at the following :

(1) error messages due to data errors. If you are using PC-386 version then these messages are displayed on the screen and a copy is written to the *.MPE file. Here * represents the name of the analysis you are currently carrying out.

(2) equilibrium check

(3) Yield ratio (YR) range

(4) Warning messages - negative p'

(5) Impermissible stress states

*2.7.2 For elastic perfectly plastic models*

1) Equilibrium check : Ensure that the out-of-balance loads in any one increment does not exceed about 15 - 20% of the maximum applied at that stage. By increasing the number of increments for the

increment block in which the largest out-of-balance loads occur one should be able to decrease the amount of out-of- balance loads. If you find out that this is not the case then it is likely that what you are attempting to do in the analysis is incorrect. Also ensure that ICOR is set equal to 1 in record C2. Even if the analysis contains a mixture of elastic (1, 2), Critical state model (3, 4, 6) and elastic perfectly plastic models always set ICOR = 1 (in the presence of even a single element which is modelled using elastic perfectly plastic model). For the regions modelled using elastic perfectly plastic model, before the very first integration point yields there should not be any out-of-balance loads associated with it. Out-of-balance loads for model 5 are always associated with yielding.

2) Warning messages :

Dlamda negative - this message at a few integration points should not raise any concern. However if such messages appear for whole regions or for many elements and with or without the following message :

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

This message indicates that the analysis may not be valid. The above message is printed if the stress state which has yielded and ventured outside the yield surface has been 'supposedly' corrected back to the yield surface, is still lies outside the yield surface. This can happen when you start an analysis with zero stresses using the Mohr-Coulomb model with C = 0. This means the whole region is on the tip of the yield surface and has presumably yielded. All these elements will be assigned plastic

stiffness instead of elastic. To avoid this specify an arbitrarily small value for C (say 1 in the material properties) to ensure that initial zero stress state lies within the yield locus rather than on it.

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

The occurrence of the message diamda Negative is rare in a well behaved analysis. Array VLDL is set to a size of 200. This is more than adequate for any situation and if this limit is reached then probably something is wrong with the analysis.

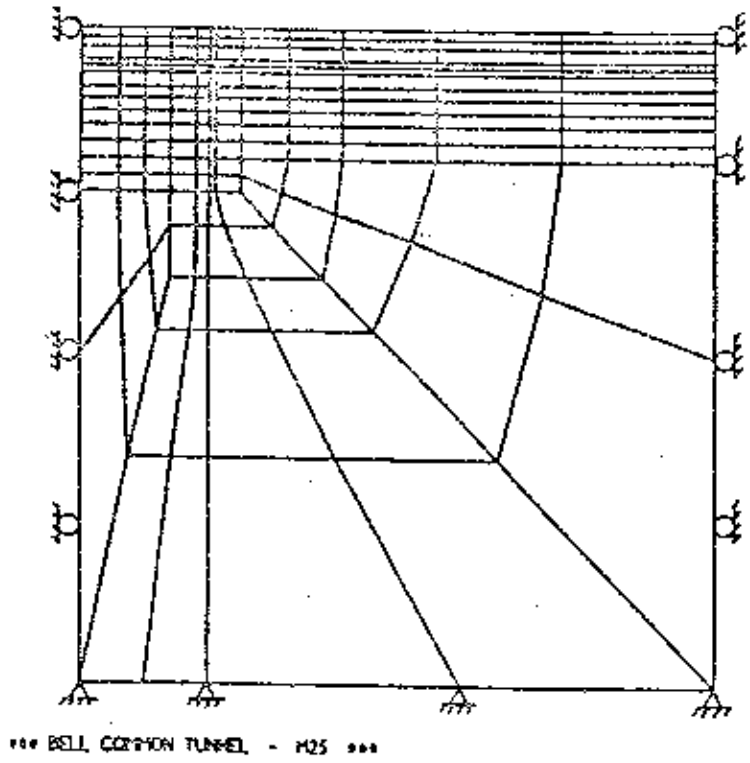



Figure 2.1 Retaining wall

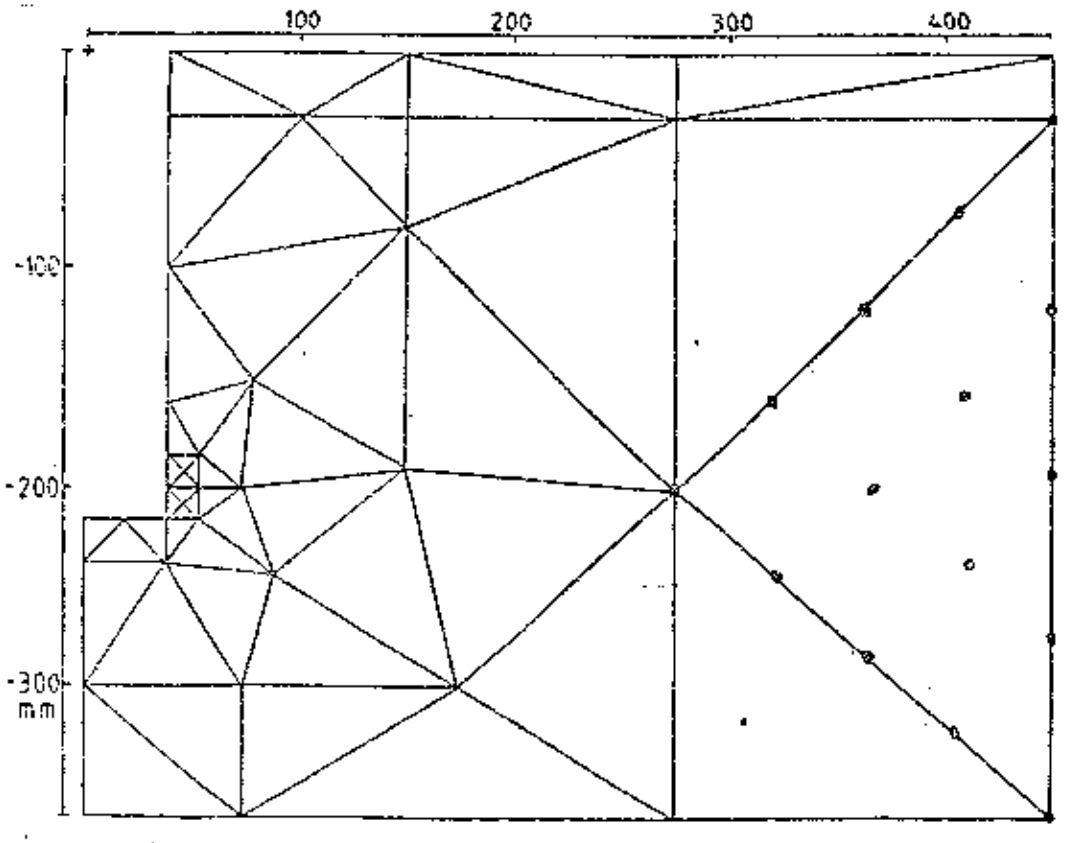



Figure 2.2 Shaft excavation (Phillips, 1984)

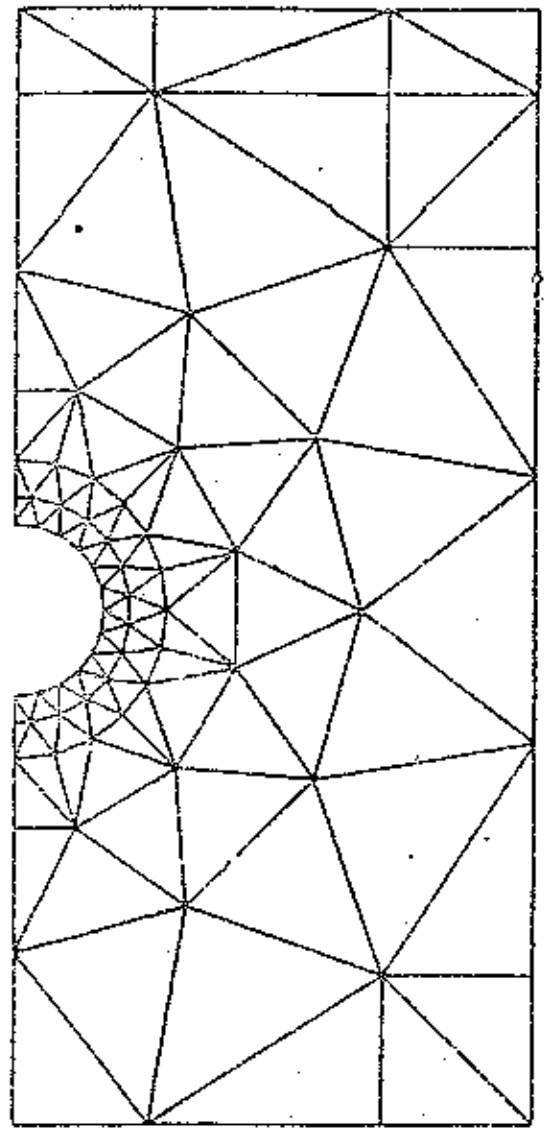


Figure 2.3 Tunnel (Taylor, 1984)

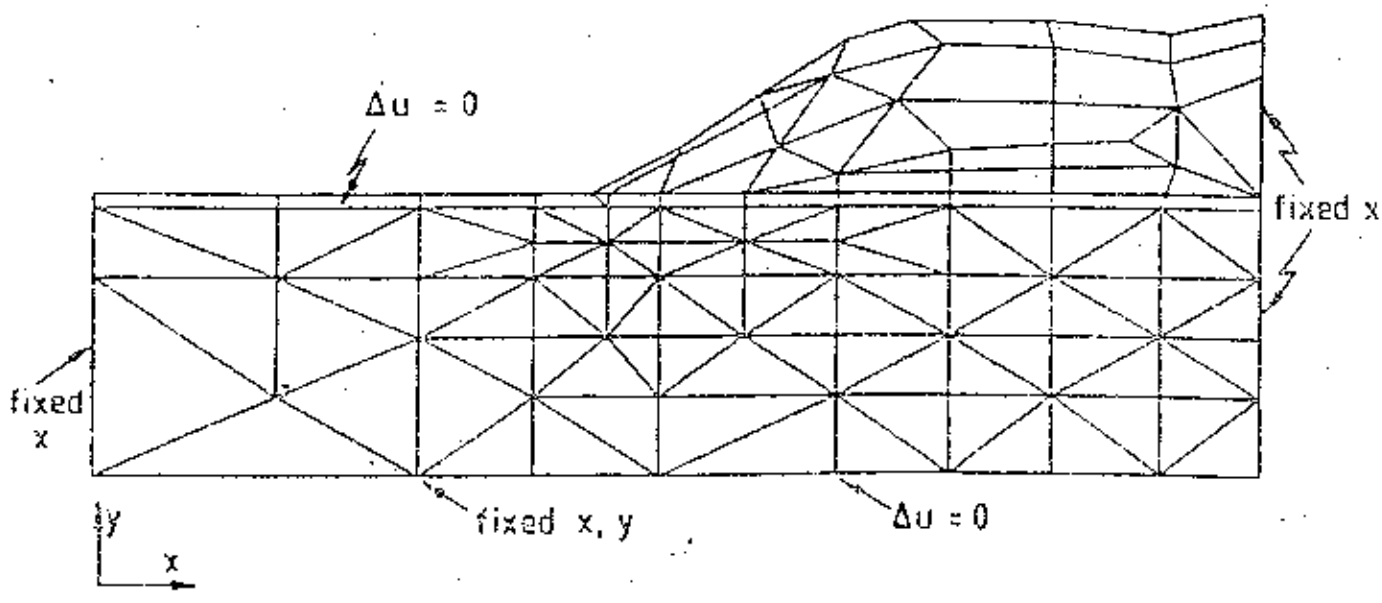



Figure 2.4 Embankment (Almeida, 1984)

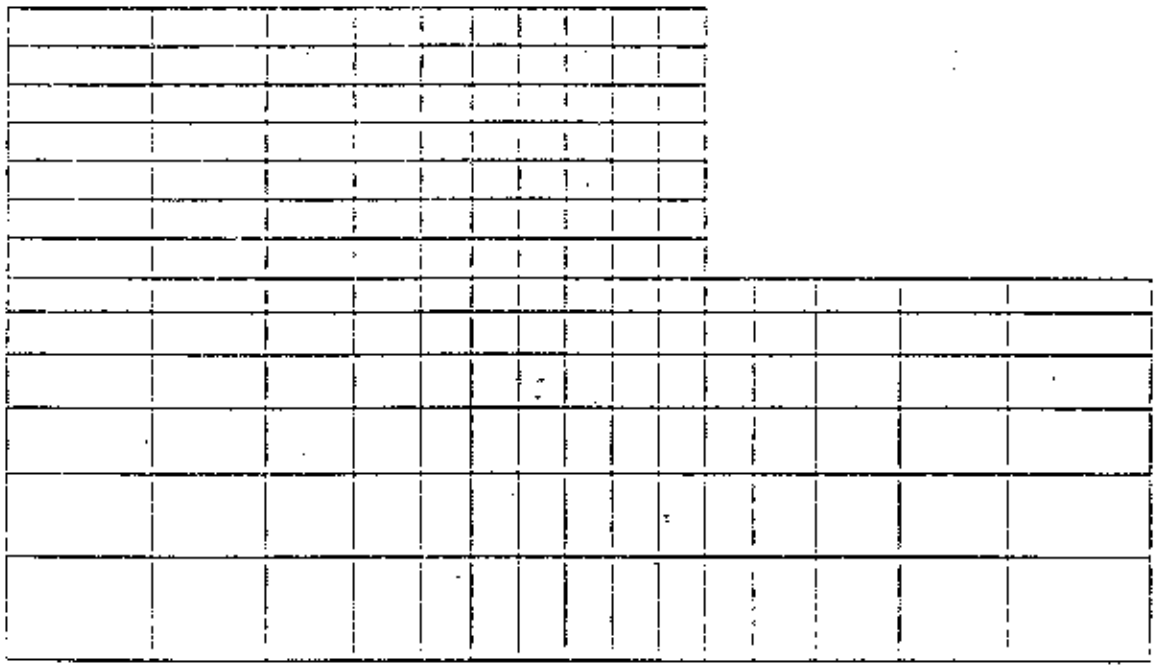


Figure 2.5 Molikpaq (Jeyatharan, 1991)

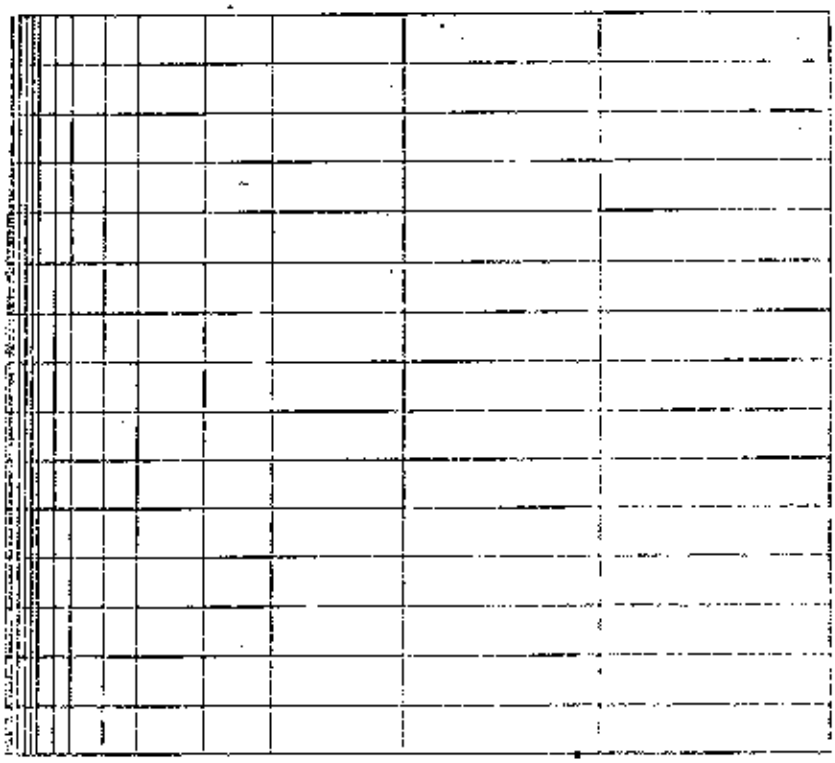


Figure 2.6 Pressure grouting (Mckinley, 1991)

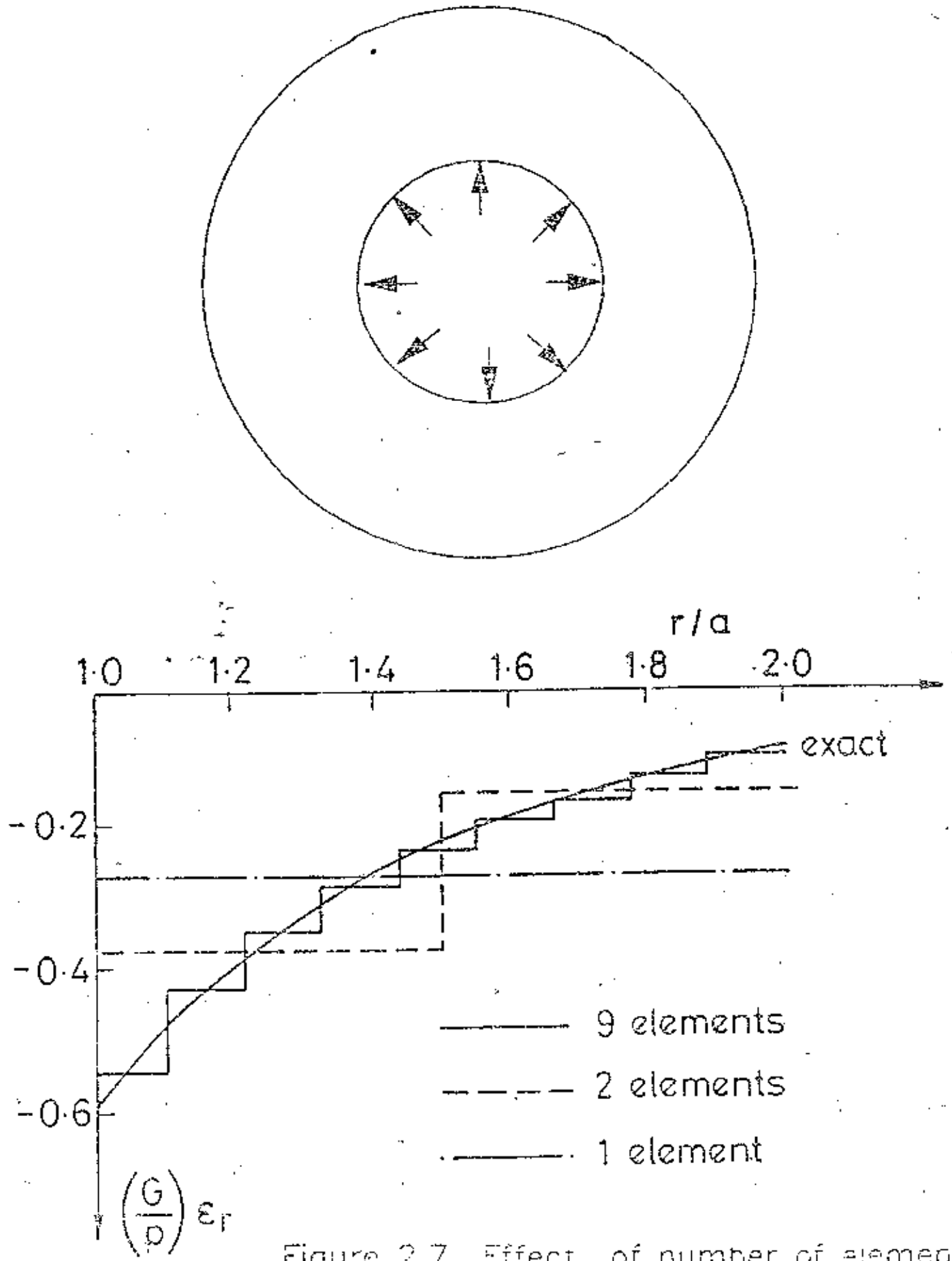
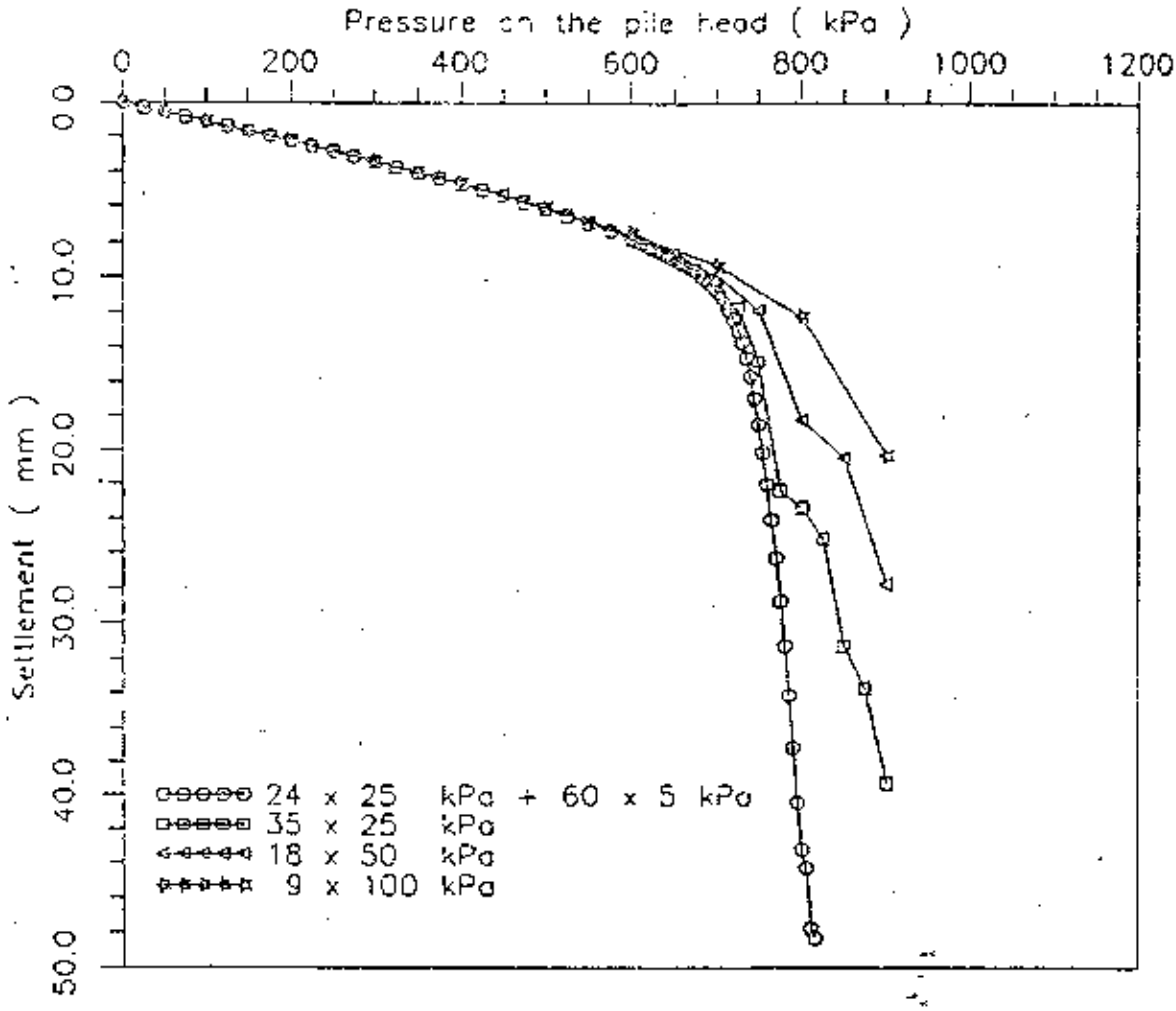



Figure 2.7 Effect of number of elements




Influence of the number of increments on the load-settlement curve.
CRISP – Undrained analyses, Ko'= 0.61

Figure 2.8 Effect of number of increments

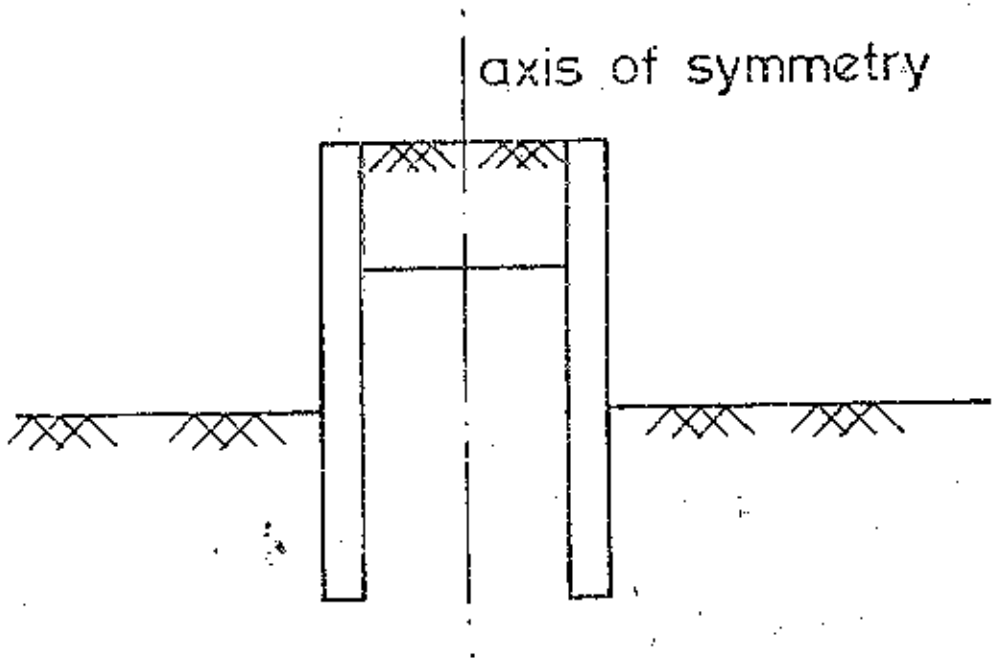
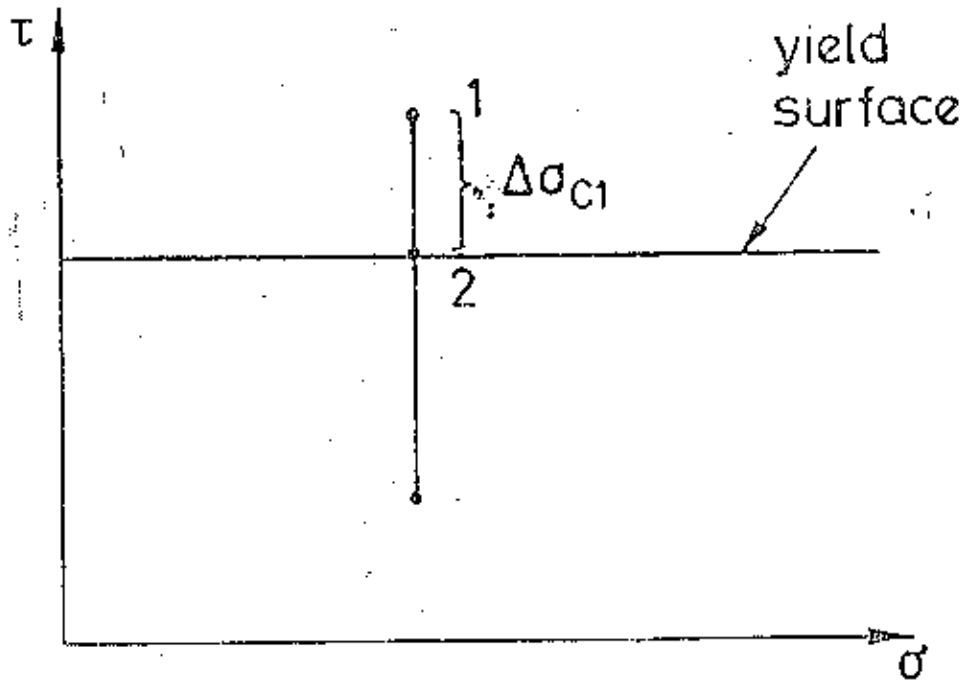



Figure 2.9 Use of axis of symmetry




$$\{ \Delta P_{C1} \} = \int B^T \Delta\sigma_{C1} \; d(vol)$$

Figure 2.10 Stress state correction

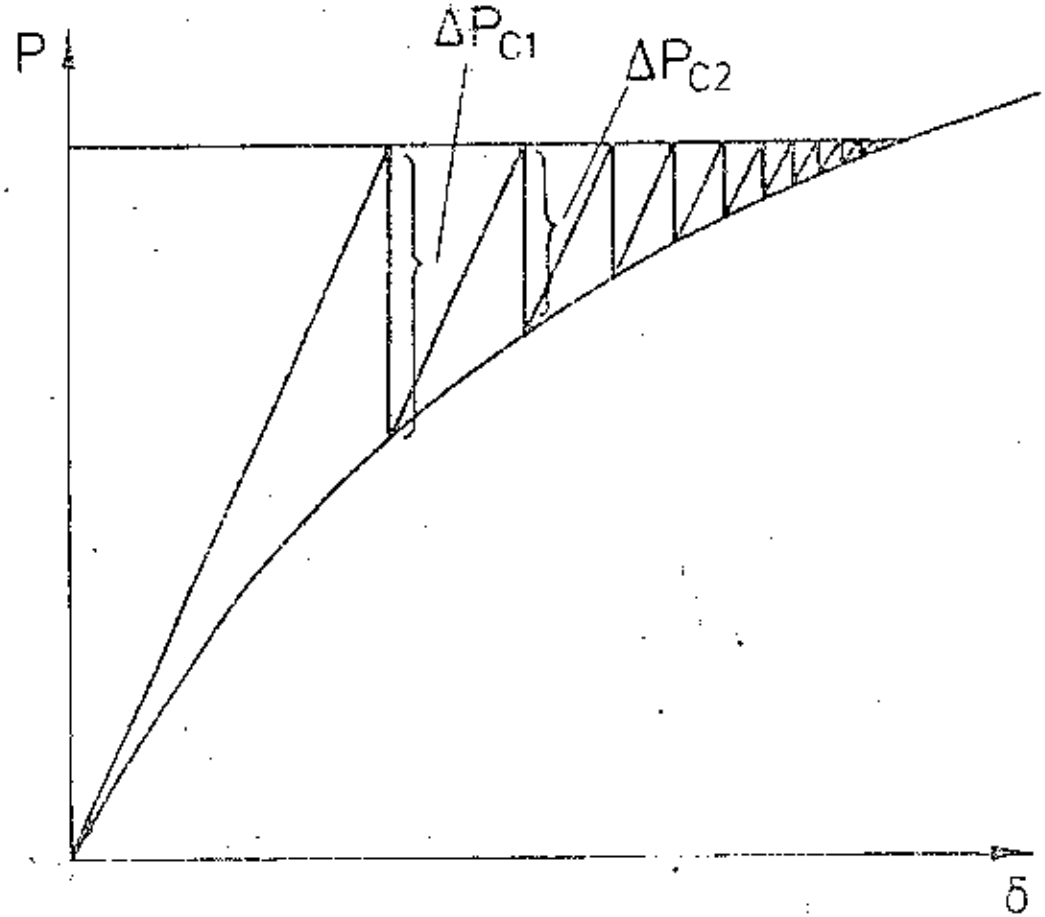



Figure 2.11 Iterative technique

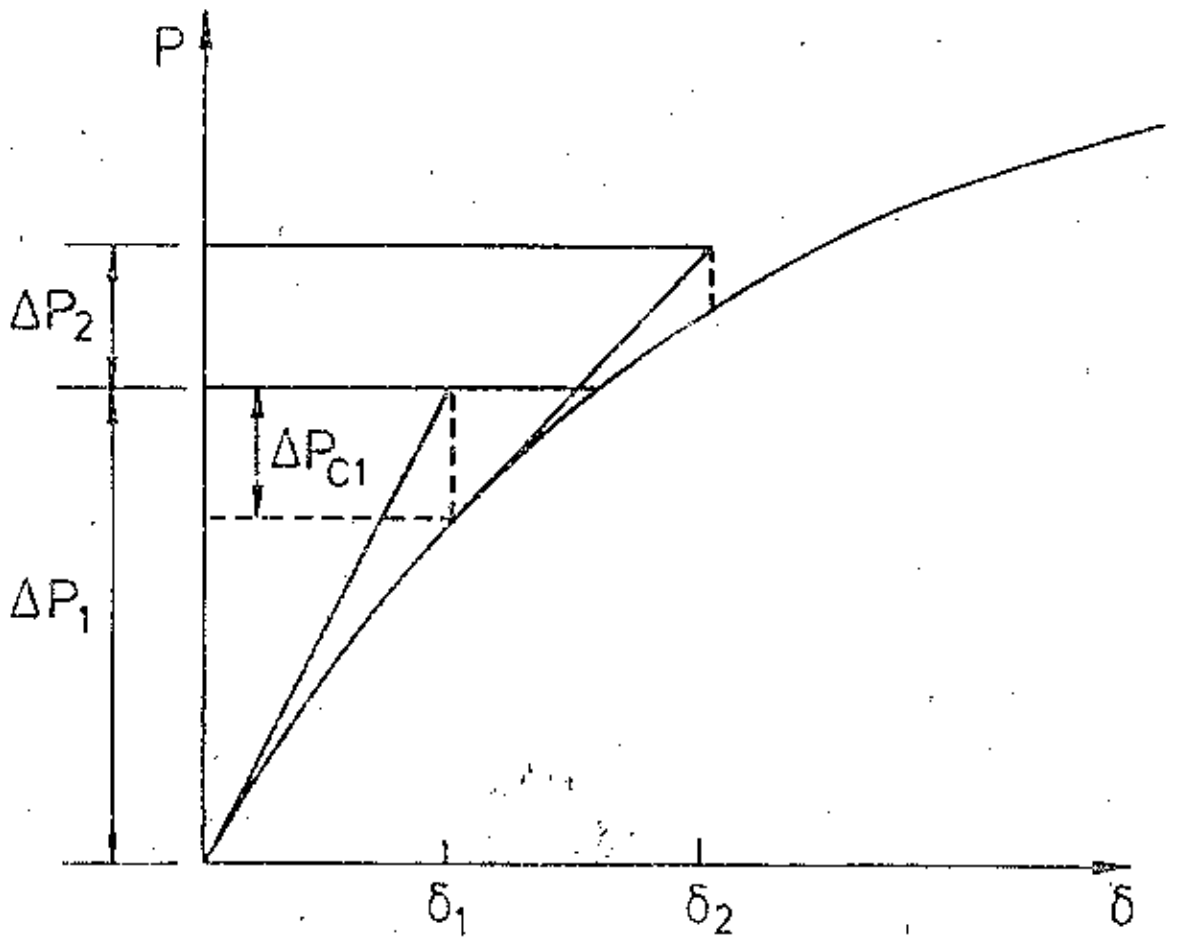



Figure 2.12 Carry over of un-balanced loads

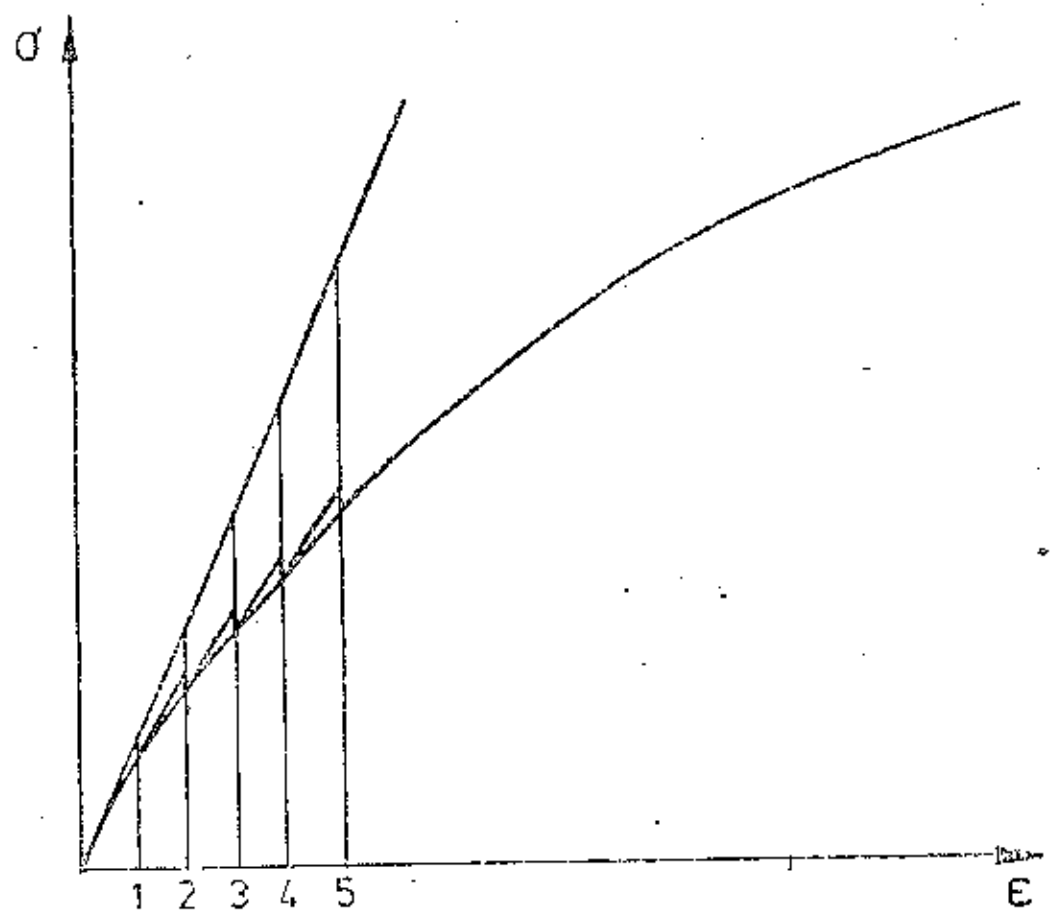



Figure 2.13 Use of sub-increments for model 5

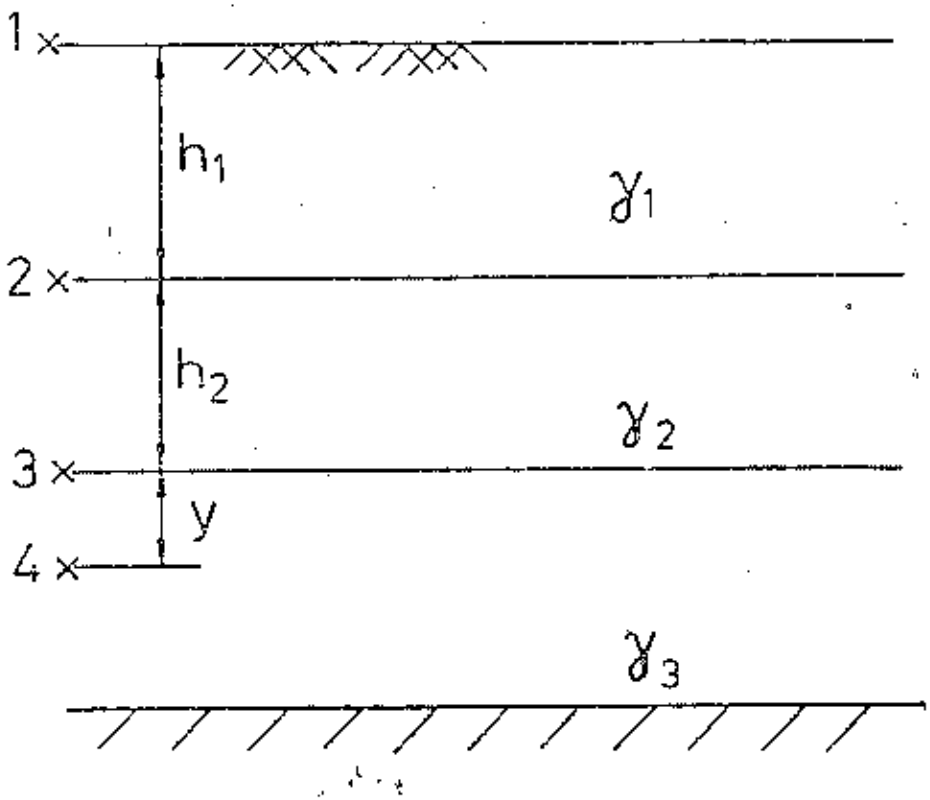



Figure 2.14 In situ stresses

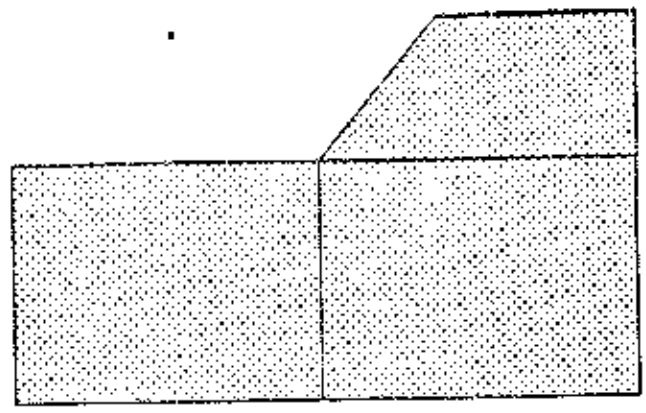


Figure 2.15(a) Complete mesh

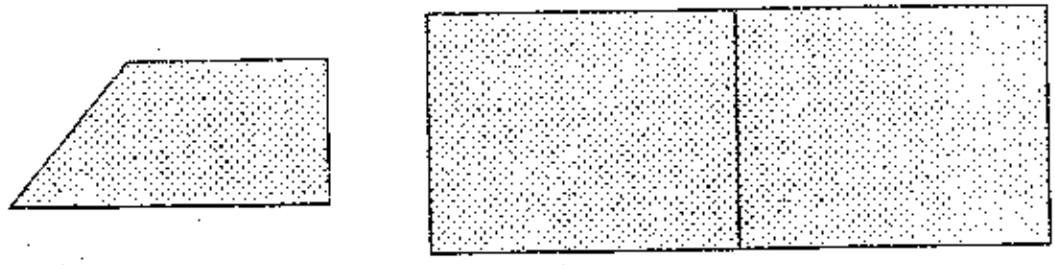


Figure 2.15(b) In situ stage

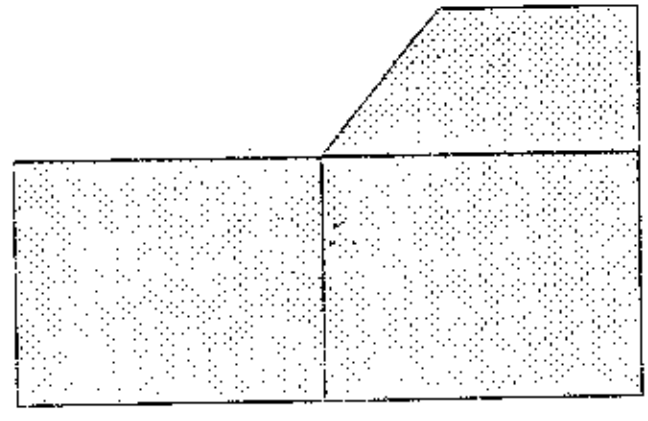


Figure 2.15(c) During analysis

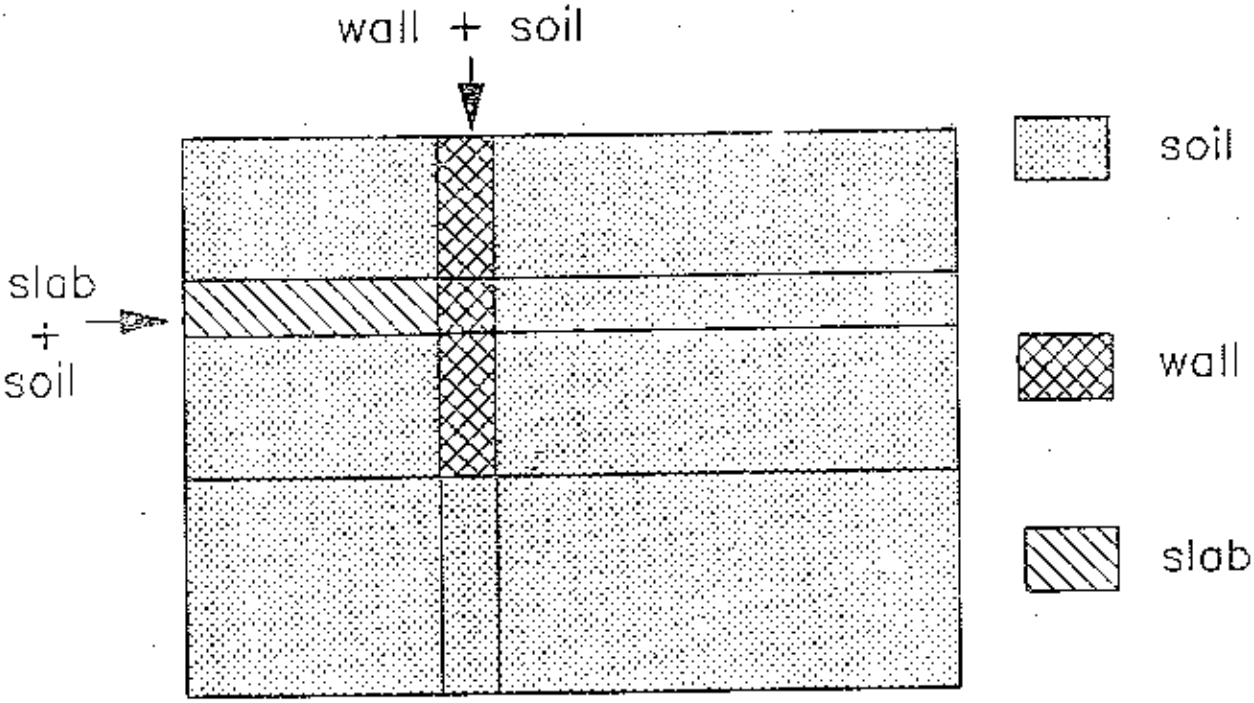



Figure 2.16(a) Complete mesh

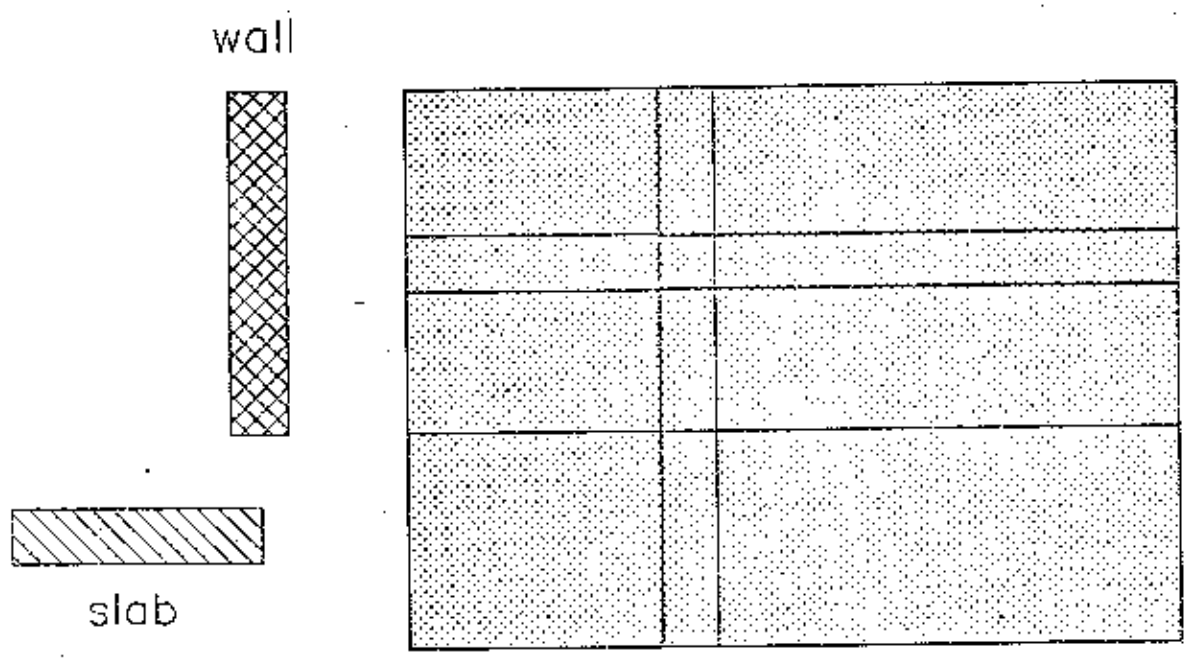



Figure 2.16(b) In situ stage

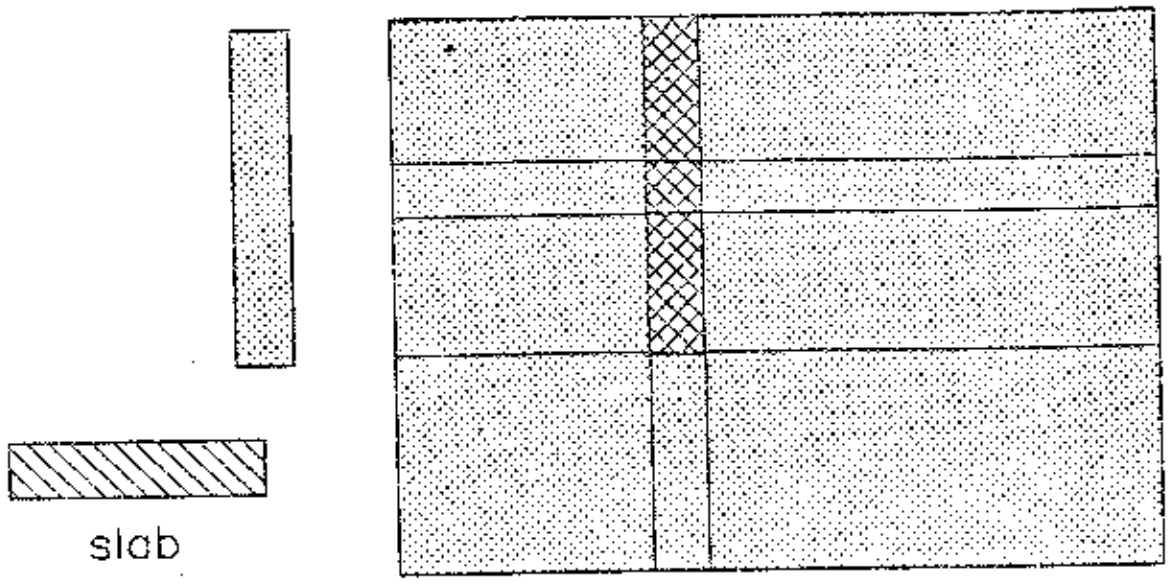



Figure 2.16(c) Increment block 1

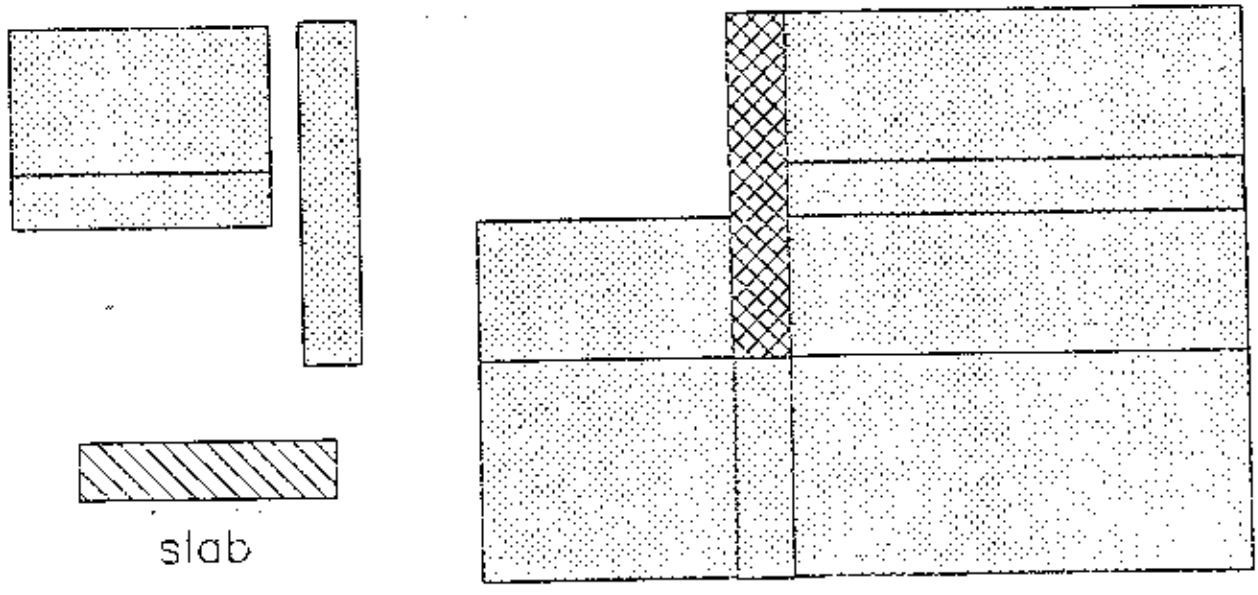



Figure 2.16(d). End of increment block 6

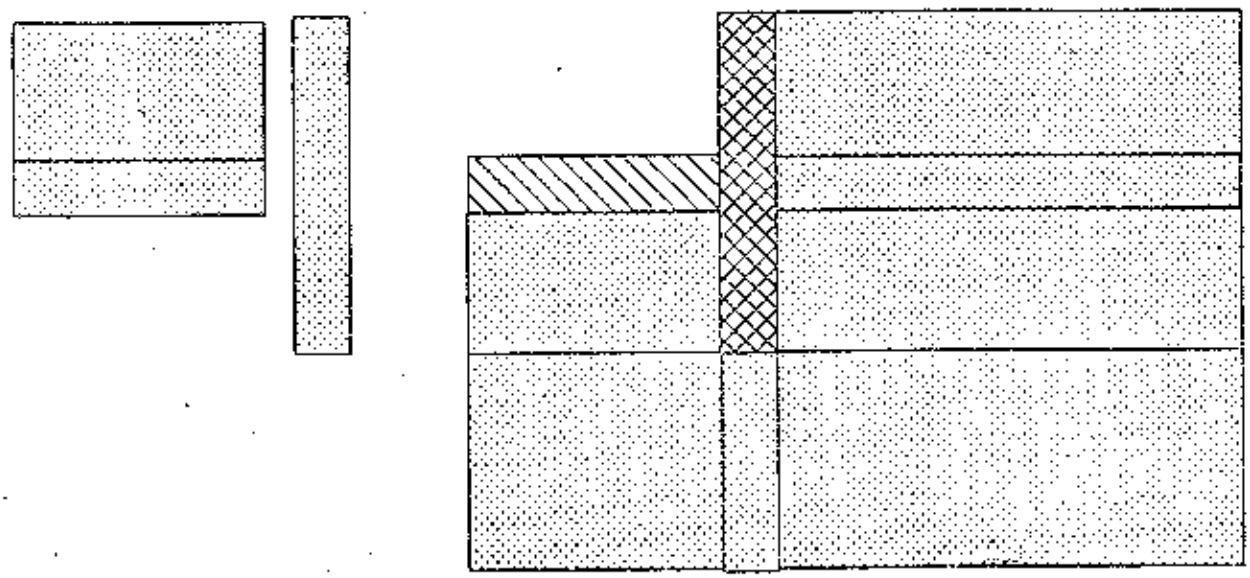


Figure 2.16(e) End of increment block 7

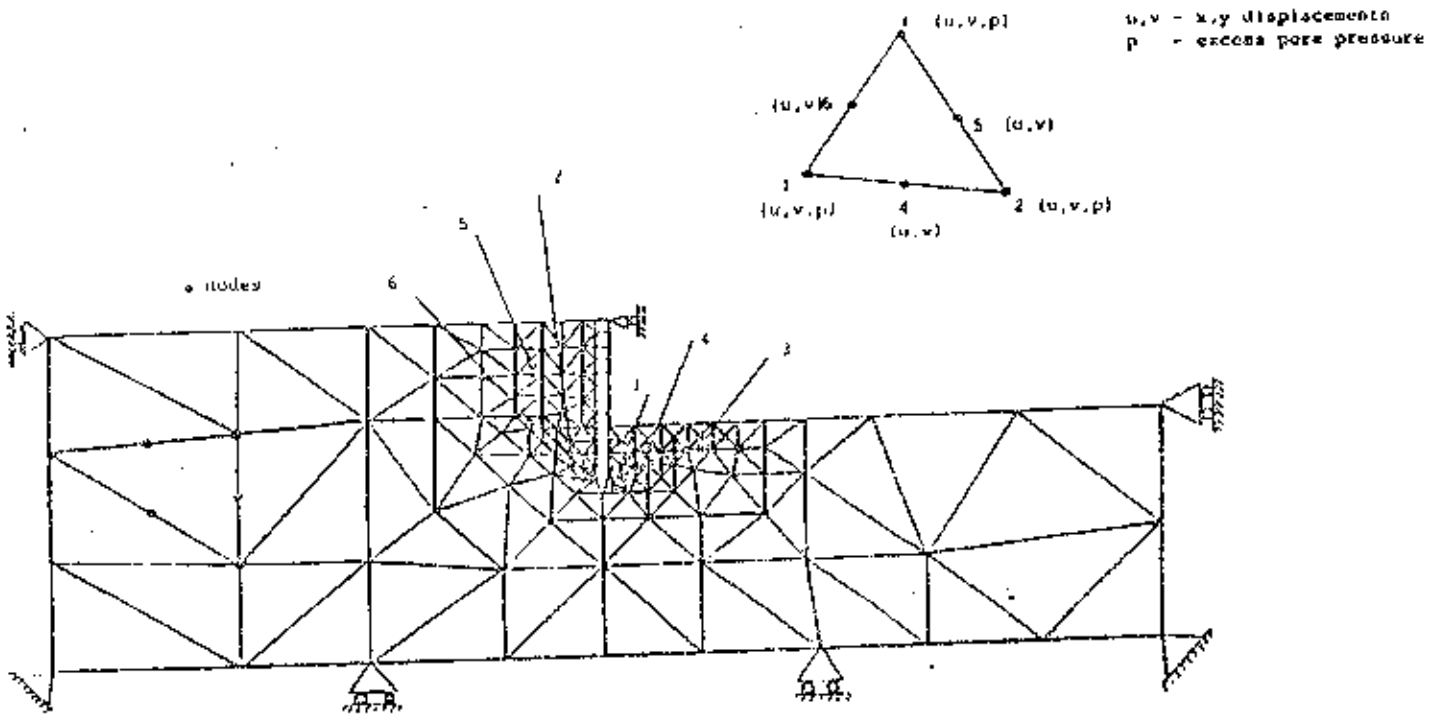



Figure 2.17 – Retaining wall (White, 1987)

# 3. CRISP-90 - AN OVERVIEW AND MODES OF USE

## 3.1 Introduction

Section 3.2 describes the new program modules and section 3.3 gives an overview of how all the program modules interact. It also covers topics such as hardware requirement and timings on some examples. Section 3.4 deals with the modes of use and describes which modules can be run on which computer environment.

## 3.2 New Program modules for running on 386 computers

Various new modules have been added to the CRISP-90 suite of programs. These are graphics based and include limited pre and post-processing capabilities. These programs do not make use of the 3-dimensional capabilities of the analysis part of the program. Therefore these programs are limited in scope. Here follows a brief introduction of all the modules in the CRISP-90 suite. The programs (on 386 computers) should be run in the sequence there appear below.

SMP (2D only)

Mesh generation program. Has only 2D-capability. This program creates an input data file (GPD) that can be directly used as input to the Geometry program (GP). The problem to be analysed is sub-divided into coarse elements (called super-elements). These are then sub-divided into triangular or quadrilateral finite elements. At present it is not possible to generate slip, bar or beam elements using this program. However users who need to use these type of elements can edit the GPD file created by the SMP program and add in these elements.

SQ (2D and 3D)

The front squasher program includes the element sequence of assembly which is efficient for the frontal method. This reduces the CPU time required by the analysis (MP). This reads in the data file created by the SMP program. GPD file is renamed as GPR file for

using it with the SQ program. The SQ program then creates an GPD file for use with the Geometry program. SQ is an optional program. This can be by-passed if running very simple examples (this is a reason why the SMP program creates a GPD file, for use directly with the GP program). But it is recommended that the users use the SQ program if running any large analysis. The improvement can be significant even for a mesh with about 40 to 50 elements.

PRE-ADG (2D only)

This program is run immediately after the SMP program. It generates information required by the ADG program for example works out the position of icons which are used in the ADG program. This program does not require any user input.

ADG (2D only)

This program generates the input data file for the Main program. It automatically generates the MPD file which can be directly input to the Main Program. The data for the ADG program are entered using menus and icons. Only capable of dealing with 2-dimensional analysis. See section 6.6 on the limitations of this program.

PP (2D only)

Post processor plotting program. Again limited to 2 dimensional analysis. It produces the following plots for a selected increment.

a) plots of undeformed and deformed mesh plot
b) zero extension line directions
c) principal strains
d) principal strain directions
e) displacement vectors
f) principal stress directions
g) location of data points
h) stress state codes (Cam-clays and elastic perfectly plastic models)

SP (2D and 3D; Cam clay analysis only)

Stress path plotting program. The stress path of any integration point in the mesh can be plotted.

CL (2D and 3D)

This is the CRISP - LOTUS 123 interface program. Creates an ascii file that can be imported into any LOTUS compatible spreadsheet program. This allows selected variables (from a group 40) to be plotted in

(a) graphs of variable vs Inrement number or time (consolidation analysis)

(b) distribution of a variable through a section of the mesh at selected increments. The user will have to define the nodes or integration points which lie along this section.

## 3.3 CRISP-90 - An Overview

The CRISP-90 suite of programs, which consists of 10 modules are written to run on INTEL 386 based computers and supersedes the CRISP-84 programs. The previous standard version of CRISP first distributed in 1984 is known as the CRISP-84 version. MICRO-CRISP which was first distributed in 1988 is a single program which can be run in any IBM compatible PC. This has limited features in comparison with other versions. MICRO-CRISP is independent of CRISP-84 and CRISP-90 and contains only the analysis part of it. No pre or post-processing facilities are provided with it.

CRISP-84 consisted of the following program modules (see Figure 3.1) :

SQ-84 - front squasher program
GP-84 - geometry program
MP-84 - main (analysis) program

These 3 program modules are superseded by GP-90, SQ-90 and MP-90 respectively. As in the case of the previous 84 version these 3 program modules can be run on any computer (mini, mainframe, CRAYS) but not in IBM PC XT, AT or compatibles. These 3 program modules are distributed in source code form for users who purchase the CRISP-90 program.

There is also an additional program module known as the CRISP-LOTUS 123 Interface program (CL90) which creates an ascii file (PRN

file) which can be imported into LOTUS 123 or any other compatible spreadsheet programs.

GP-90 and MP-90 forms the analysis part of the CRISP-90 suite of programs and the same version can be run any computer (386 computer which satisfies certain hardware requirement and minis, mainframes and CRAYs).

*3.3.1 Analysis type*

All the modules of CRISP-90 which can be used in a two-dimensional analysis are shown in Figure 3.2.

The modules which can be used in a 3-Dimensional analysis are shown in Figure 3.3.

There is only a single version and any program module denoted by 3D is capable of handling both 2 dimensional and 3 dimensional analyses.

*3.3.2 Distribution*

All the programs shown in Figure 3.2 are distributed in EXE file form in 1.44 MB (3.5 inch) disks or 1.2 MB (5.25 inch) disks. The programs shown in Figure 3.4 can be obtained in a 9-track magnetic tape. When ordering indicate which one is appropriate for your computer system.

*3.3.3 Hardware Requirement*

CRISP-90 can be run in INTEL 80386 processor based desktop computers which satisfy the following hardware requirement.

2 - 4 MB RAM (memory), Hard disk, Maths co-processor, VGA colour monitor (640 x 480 pixels), Mouse, Running MS DOS 3.30 or higher (not OS/2, Unix). DR DOS can be used instead. About 15 to 20 MBytes of hard-disk space is required. Depending on the size of the analysis more space may be required. 2 MBytes of RAM is the minimum. This enables one to carry out analysis with about 300 triangular elements. More memory will be needed to analyse very large problems. For example you will require 4 MBytes of RAM to carry out an analyses with about 1000 elements of type 2/3/4/5. However unless the problem you are analysing is geometrically very complex you

wouldn't usually require to use a mesh with 1000 elements.

About 3 MBytes of hard disk space is required to install CRISP-90 and the SALFORD FTN77 run time system. This is the minimum requirement and does not include the disk space required for running the analyses. Typical sizes of some of the files which are created during a CRISP run is shown in Table 3.1.

CRISP-90 runs on stand alone and and networked PC-386s. It is not possible to run CRISP-90 on a multitasking PC environment. It cannot be run with DESQVIEW or WINDOWS 3. It cannot be run in a MS-DOS window or environment under Unix.

*3.3.4 Running CRISP-90 on other computers*

Figures 3.4 and 3.5 shows the modules that can be run on any other computer. No graphical output is possible in this environment unless you make use of the commercial packages suppled by FEGS Ltd, FEMVIEW Ltd. FEMVIEW Ltd have versions of their software running in PC-386 computers. So it is possible to use these commercial packages for pre and post-processing in a PC-386.

As shown in figure 3.5, the CRISP to LOTUS 123 interface program can be run on these computers. The resultant ascii file (PRN) can be transferred to a PC and can be imported into any spreadsheet programs.

The above four programs (GP, MP, SQ and CL) are supplied in source code allowing users who have purchased the B4 version or the MICRO-CRISP version an update path. Source codes of the above four programs can also be obtained in a 9-track unlabelled magnetic tape. None of the other programs will be available in source code form.

This is because all other programs include graphic routines and the EXE files can only be run on 386 based computers.

It is not possible to run these programs on IBM PC compatibles or AT compatibles. The only program that can be run these two type of computers is the MICRO-CRISP Program (see Fig. 3.6). This is very limited in scope and only a small number of elements can be used (about 60 - 6 noded triangles) in any analysis if run with the MS-DOS restriction of 640 Kbytes.

It should be noted that even though there is only a single version which runs on mainframes to 386 computers, it is not realistic to run large three dimensional analysis on the 386 computer. For example see below the time taken for 3D analysis with a single increment. All the analyses were elastic, and contained a rectangular block with equal number of element in all 3 directions.

*3.3.5 Timings*

The time taken were on a IBM PS/2 model 70-121 computer running at 20 MHz. The time taken on A21 computers running at 25 MHz would be approximately 50% less.

| EXAMPLE | NEL | NODE | FW | DOF | TIME | ELEMENT CONFIGURATION |
|---|---|---|---|---|---|---|
| 1 | 27 | 208 | 174 | 624 | 3m | 3 X 3 X 3 |
| 2 | 125 | 756 | 360 | 2268 | 30m | 5 X 5 X 5 |
| 3 | 343 | 1856 | 618 | 5568 | 3h 40m | 7 X 7 X 7 |
| 4 | 512 | 2673 | 774 | 8019 | 8h 20m | 8 X 8 X 8 |

NEL - Number of elements in the mesh
NODE - Total number of nodes in the mesh
FW - maximum frontwidth
DOF - Number of degrees of freedom (variables) in the analysis
TIME - Time taken for 1 increment in hours and minutes.

For comparison a typical 2D analysis with 230 LST elements took approximately 2 minutes for one increment.

*3.3.6 Hard copy of Plots generated on the screen*

At present it is not possible to get good quality outputs of any of the plots generated on the screen. Screen dumps can be obtained on IBM Matrix printers or EPSON dot matrix printers. these can also can be obtained on Laser jet printers (example KYOCERA F1000) which can emulate a IBM matrix printer. However HP Laser jet printers cannot be used.

However it is hoped that future versions should be able to use the Laser jet II printers.

*3.3.7 Limitations in running CRISP-90 in PC-386 environment*

The limitation on producing good quality hard copy has already been mentioned.

## 3.4 Modes of Use (2-Dimensional analysis)

For the sake of simplicity all the modules in the CRISP-90 suite of programs are classified into the following four categories.

| | | |
|---|---|---|
| (a) Pre processors | : | SMP, PRE-ADG and ADG, (SQ) |
| (b) Analysis | : | GP, MP |
| (c) Assessment | : | ASP |
| (d) Post processors | : | PP, PQ and SP, CL |

PRE-ADG and ADG are paired together because ADG cannot be run without running PRE-ADG. Neither of these programs (PRE-ADG and ADG) can be run without running SMP. Similarly on the post processing side PQ and SP are paired together. SP cannot be run without running PQ.

The front squasher programs SQ is loosely grouped with the pre-processor even though it is not strictly a pre-processor. Similarly ASP is put into a separate category of its own called assessment. Because it is only used in the assessment of any analysis and does not produce any useful post-processing information.

For two dimensional analysis all the modules can be used. It should be noted that the SMP, PRE-ADG and ADG programs do not fully cater for all the facilities/options that are available in the analysis modules - GP & MP (see x.x). In addition the following modules can only be used for analysis using Cam-clay models for part of the mesh.

1) Assessment program - ASP.
2) Stress path plotting programs - PQ & SP.

Unless one is analysing a problem which requires the use of bar/beam/slip elements all the modules can be used.

*3.4.1 What modules one needs to use?* •

Because of the availability of so many modules it is understandable that this could lead to some confusion. The different options that are available are described below.

Pre-processing : The SMP, PRE-ADG and ADG program are bundled together. The user has the following options :

| | |
|---|---|
| (a) use SMP only | - GP data file is created (GPD). |
| (b) use all 3 programs | - GP and MP data files are created (GPD, MPD). |
| (c) use none | - no data files are created. |

For option (a) only the geometry program input data file (GPD) is created. The user will then have to create a main program input data file (MPD) by referring to the CRISP brown manual (Vol.1) and typing it himself/herself into an ascii file (this file should be given the same name as the geometry data file and should have the extension MPD). This allows the user to modify the finite element mesh to add in slip/bar/beam elements. Then the user has to edit the GPD file created by the SMP program using a word processor to incorporate the changes he/she has made to the f.e. mesh.

It should be remembered that once you make modifications to GPD it is not possible to run the PRE-ADG and ADG programs and create an MPD file and use it directly with the GPD file you have edited to incorporate slip/bar/beam elements. This is because the PRE-ADG and ADG program rely on information input to the SMP program and are not aware of the changes you have made to the GPD file. However if you are familiar with the CRISP brown manual you can edit the MPD file created by the ADG program so that the 'edited' files GPD and MPD are compatible. This method of using the CRISP-90 programs are full of pitfalls and is not recommended to new users. It is easy to introduce errors into the data.

In case of option (b) both Geometry data file (GPD) and Main (analysis) data file (MPD) are created. It is not possible to run the ADG program without running the SMP and PRE-ADG programs.

If you have other ways of preparing GPD and MPD then you choose option (c).. This is similar to using CRISP-84 where no pre-processors were provided with the programs. The user has to prepare both GPD and MPD by hand or use programs they have written themselves or use commercially available pre-processors.

It should be noted that the pre-processors (SMP, PRE-ADG & ADG) are independent of the post processing programs (PP, PQ & SP, CL).

Post-processing : The three programs available are (PP, SQ & SP, CL).

The four programs available are run independent of each other.

(a) PP - Post processor plotting program
(b) PQ ans SP - stress path plotting program.
(c) CL - CRISP - LOTUS 123 Interface program.

These make use of the NRS file created by the MP.

PP (Plotting Program)

This program can be used for any two dimensional analysis. It can be used for even an analysis where slip/bar/beam elements are included. Therefore you can use this program to look at the results for any 2D analysis carried out by the Main Program (MP).

Irrespective of whether you have used the SMP, PRE-ADG and ADG programs or not in generating the GPD, MPD files the PP program can be used for any 2D analysis. The same applies to the other two post processing programs.

PQ & SP (Critical state models only)

Again PQ and SP have to be run together to produce the stress path plots. PQ should be treated as PRE-SP program. Again stress path plotting program is only available for analysis using any one of the critical state models (Modified Cam clay, Cam clay, Hvorslev surface model). It also can be used in any analysis where zones of critical state models are present with other type of models (elastic - 1,2, elasto-plastic - 5). The user can then use these programs to plot

stress paths in the regions where the critical state models are used. However these programs cannot be used to plot stress paths for any part of the mesh modelled by any of the other models (models 1,2,5)

CL (CRISP - LOTUS 123 Interface program)

This program can be used for any analysis (2D or 3D) that can be carried out by the MP program. It is independent of all other pre and post processing modules.

Summary

| | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 |
|---|---|---|---|---|---|---|---|---|
| SMP..... | X | X | (.) | (.) | . | . | (.) | . |
| PRE-ADG & ADG..... | X | (.) | (.) | (.) | . | . | (.) | . |
| SQ..... | (X) | (X) | (X) | (.) | (X) | (.) | (.) | . |
| GP..... | X | X | X | X | X | X | X | X |
| MP..... | X | X | X | X | X | X | X | X |
| ASP..... | (C) | (C) | (C) | (C) | (C) | (C) | (.) | . |
| PP..... | (X) | (X) | (X) | (X) | . | . | (.) | . |
| PQ & SP..... | (C) | (C) | (C) | (C) | (C) | (C) | (.) | . |
| CL..... | (X) | (X) | (X) | (X) | (X) | (X) | (.) | . |

. - The program cannot be run.

(.) - The user has chosen not to run this program.

X - the program is run.

(X) - the running of the program is optional.

(C) - the running of the program is optional (can only be run for analysis that uses any one of the critical state models 3,4 6).

1 - 2D analysis where all the modules are used. Analysis which does not require slip/bar/beam elements. Alternatively GPD and MPD files created are modified by the user to incorporate these element types into the mesh.

2 - 2D analysis where the user may have modified the f.e. mesh (created by the SMP program) to incorporate slip/bar/beam elements and also creates his /her own MPD file (without using PRE-ADG and ADG file).

3 - 2D analysis where the user prepares the GPD and MPD files without using the SMP, PRE-ADG & ADG programs.

4 - Any 2D analysis of category 3 where the front squasher program SQ is not run.

5 - 3D analysis where front squasher program (SQ) is used.

6 - 3D analysis where front squasher program (SQ) is not used.

7 - Any 2D analyses where only the GP and MP programs are used.

8 - Any 3D analyses where only the GP and MP programs are used.

## 3.5 Running the programs in more than one system

Running the programs on PC-386 and any other Mini or mainframe

| Program | 1 | 2 | 3 | 4 |
|---|---|---|---|---|
| SMP | PC | PC | . | . |
| PRE-ADG & ADG | PC | PC | . | . |
| SQ | PC | MF | MF | MF |
| GP | MF | MF | MF | MF |
| MP | MF | MF | MF | MF |
| ASP | . | . | . | . |
| PP | . | . | . | . |
| PQ & SP | . | . | . | . |
| CL | MF | MF | MF | MF |
| Spreadsheet Program $ (eg. LOTUS 123) | PC | PC | PC | . |

$ Spreadsheet programs are not part of CRISP-90. No spreadsheet programs are distributed with CRISP. This information is included for the sake of completeness. Any spreadsheet programs which can import "LOTUS 123" compatible ascii files can be used.

In the above table

PC - this means PC-386 which satisfies hardware requirement listed elsewhere (not IBM PC XT or AT or compatibles).

MF - Any other computers (Mini, Mainframe computer).

For the purpose of the above table it is assumed that it is possible to transfer ascii files between the PC and the mainframe /mini computer in a straigtforward manner.

1 - This set is where the GPD and MPD files are created in the PC using SMP, PRE-ADG and ADG files. The GPD file is renamed as GPR and the SQ program is also run on the PC. The file GPD and MPD are transferred to the other computer (MF) and the analysis carried out. The CRISP-LOTUS 123 Interface program is run and the resulting ascii file (PRN) can then be transferred to the PC (Here any IBM PC compatible which has a spreadsheet program can be used - It need not be a PC-386).

2 - same as option 1 above except for the difference where the front squasher program is run on the MF environment instead of the PC.

3 - For a 3D analysis or in a 2D analysis where SMP, PRE-ADG & ADG programs are not used to generate the GPD and MPD files.

4 - In the absence of a PC-386 computer, analysis is carried out in MF environment for both 2D and 3D analysis.

Example 1 : Abutment wall - 105 elements of type 4 and 5.
Total nodes = 360
Total number of variables = 818
Maximum frontwidth = 63
Time taken by MP = 45 seconds per increment.
Size of NRS file for 10 increments

| | | | | |
|---|---|---|---|---|
| ABTWALL | SVO | 3220 | 21-08-90 | 3:44p |
| ABTWALL | GPD | 8053 | 11-09-90 | 12:56p |
| ABTWALL | SVN | 3220 | 11-09-90 | 1:46p |
| ABTWALL | SMD | 37368 | 11-09-90 | 12:54p |
| ABTWALL | ADG | 18122 | 11-09-90 | 1:52p |
| ABTWALL | MPD | 7965 | 11-09-90 | 2:11p |
| ABTWALL | GPR | 7621 | 11-09-90 | 12:53p |
| ABTWALL | PPD | 163 | 11-04-91 | 11:15a |
| ABTWALL | GPO | 24648 | 19-04-91 | 12:22p |
| ABTWALL | LIK | 22174 | 19-04-91 | 12:22p |
| ABTWALL | MSO | 10116 | 19-04-91 | 2:22p |
| ABTWALL | NRS | 827834 | 19-04-91 | 12:30p |
| ABTWALL | MPO | 178315 | 19-04-91 | 12:30p |
| ABTWALL | MAS | 152806 | 19-04-91 | 12:30p |

Example 2 : Retaining wall - 280 elements of type 3 and 5.
Total number of nodes = 579
Total number of variables = 1314
Maximum frontwidth = 88
Time taken by MP = 1.5 minutes per increment.
Size of NRS file for 25 increments

| | | | | |
|---|---|---|---|---|
| HALE1 | GPD | 15427 | 08-06-89 | 5:20p |
| HALE1 | MPD | 6953 | 14-06-89 | 5:21p |
| HALE1 | NRS | 4125128 | 13-04-91 | 5:12p |
| HALE1 | LIK | 39366 | 13-04-91 | 3:54p |
| HALE1 | PDF | 20352 | 13-04-91 | 3:54p |
| HALE1 | PPD | 163 | 13-04-91 | 5:52p |
| HALE1 | PPO | 218008 | 13-04-91 | 5:54p |
| HALE1 | PPE | 182 | 13-04-91 | 5:54p |
| HALE1 | MSO | 19633 | 13-04-91 | 6:02p |

TABLE 3.1 : SIZE OF FILES (INDICATION OF HARD DISK SPACE REQUIRED)

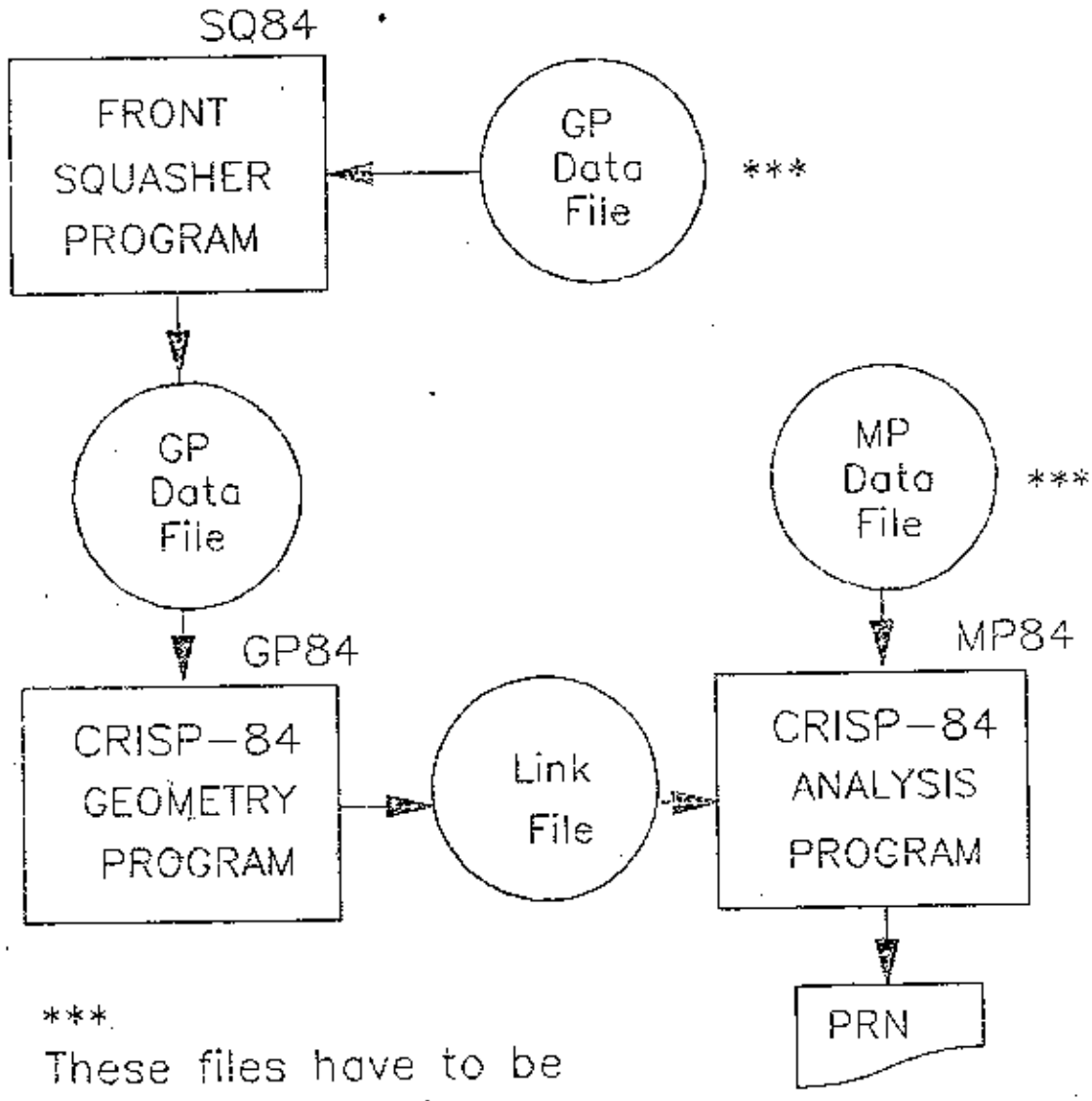



FIGURE 3.1 CRISP--84 ON MAINFRAME COMPUTERS

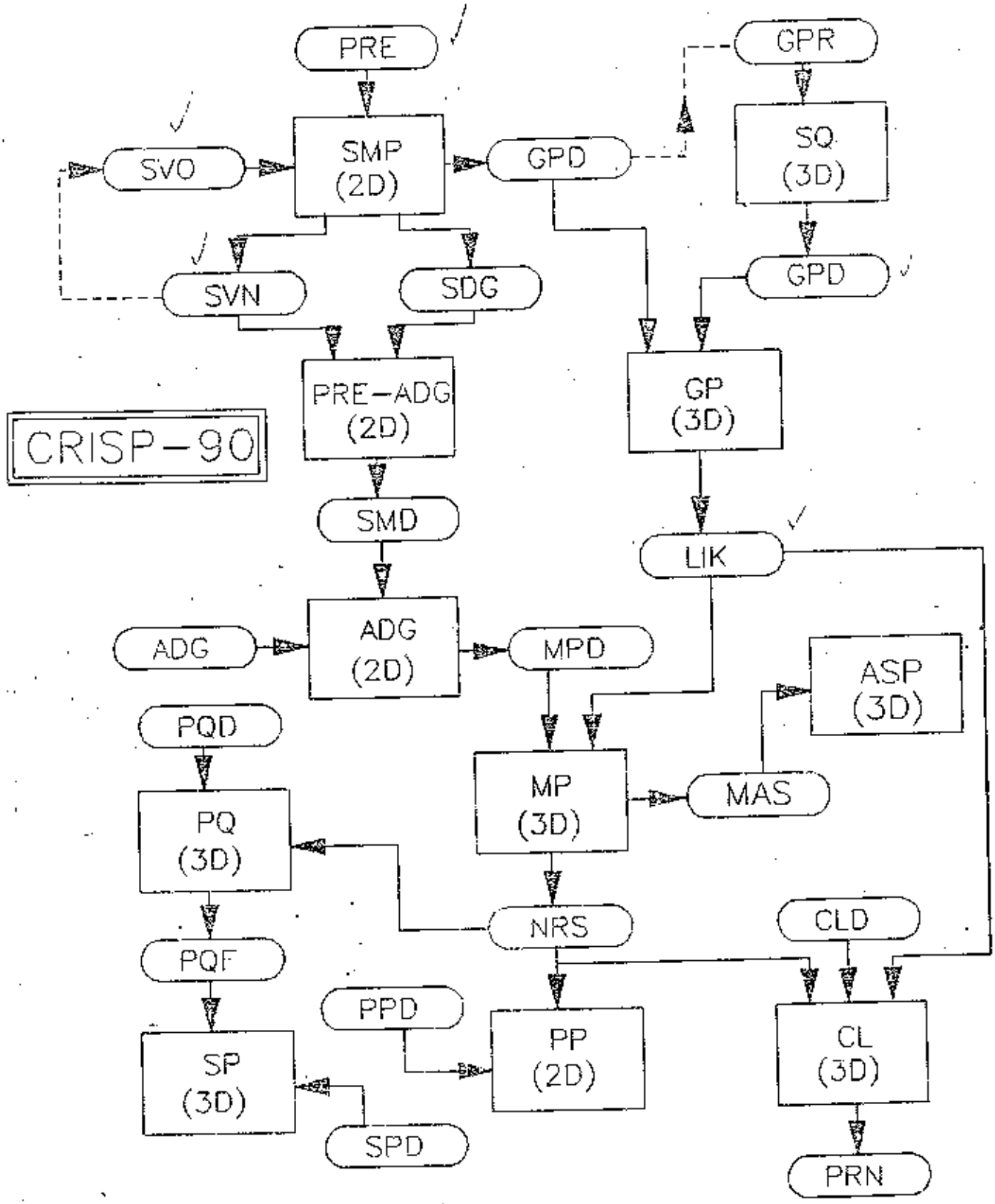



FIGURE 3.2 : CRISP-90 SUITE OF PROGRAMS
- FOR 2D ANALYSIS ON PC-386/486

( ) files are identified by extension names

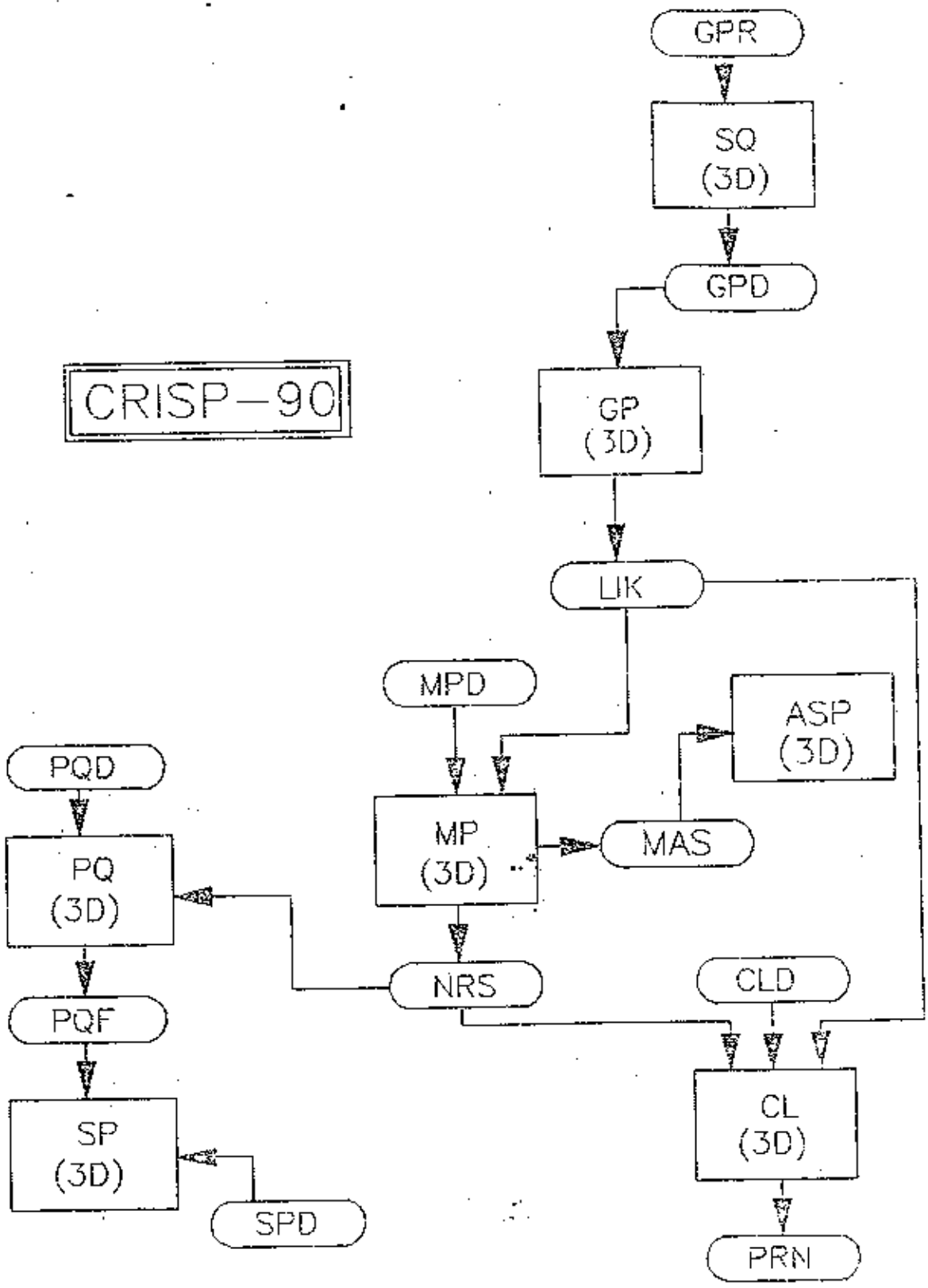



FIGURE 3.3 : CRISP-90 SUITE OF PROGRAMS FOR 3D ANALYSIS ON PC-386/486

( ) files are identified by extension names

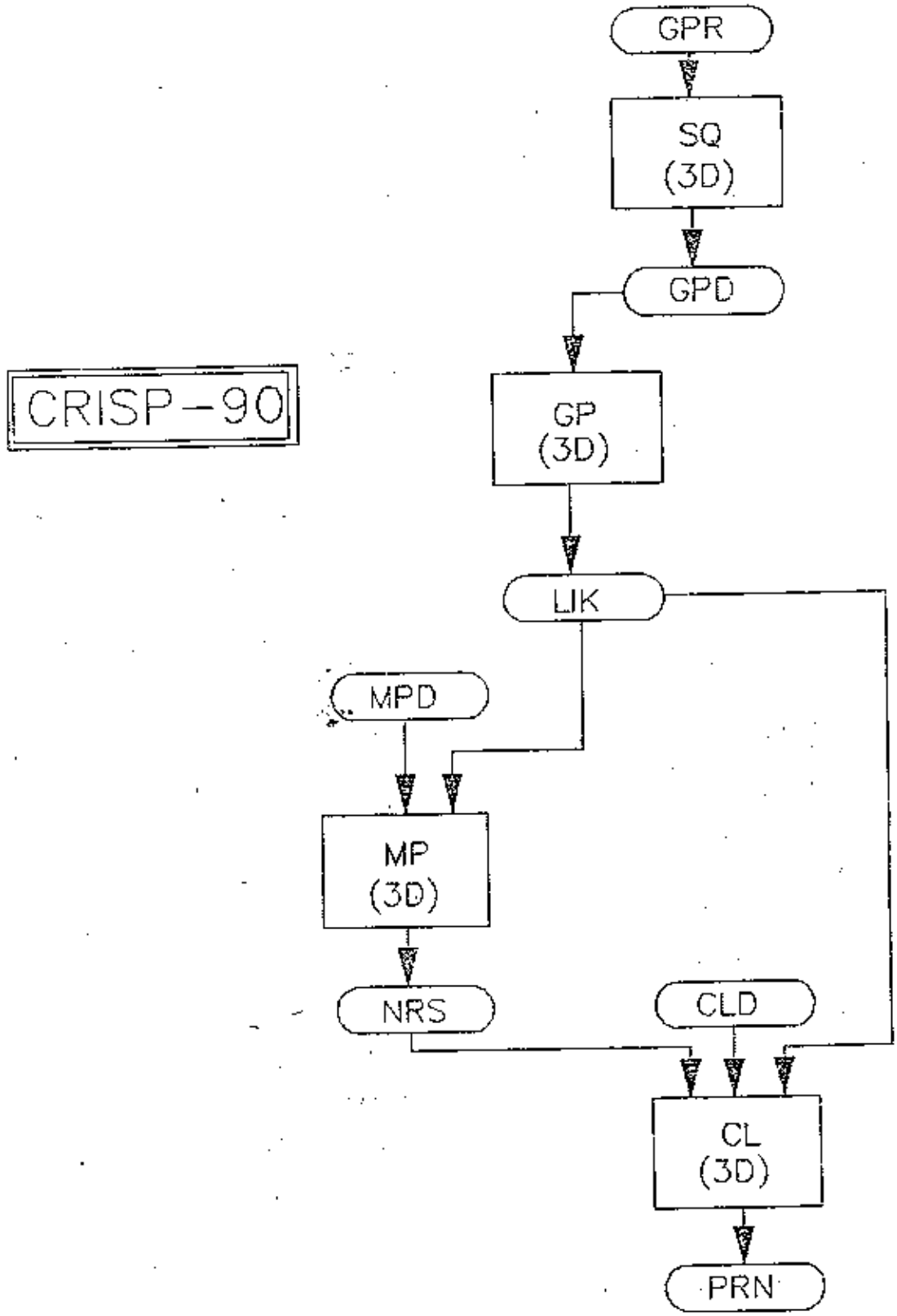
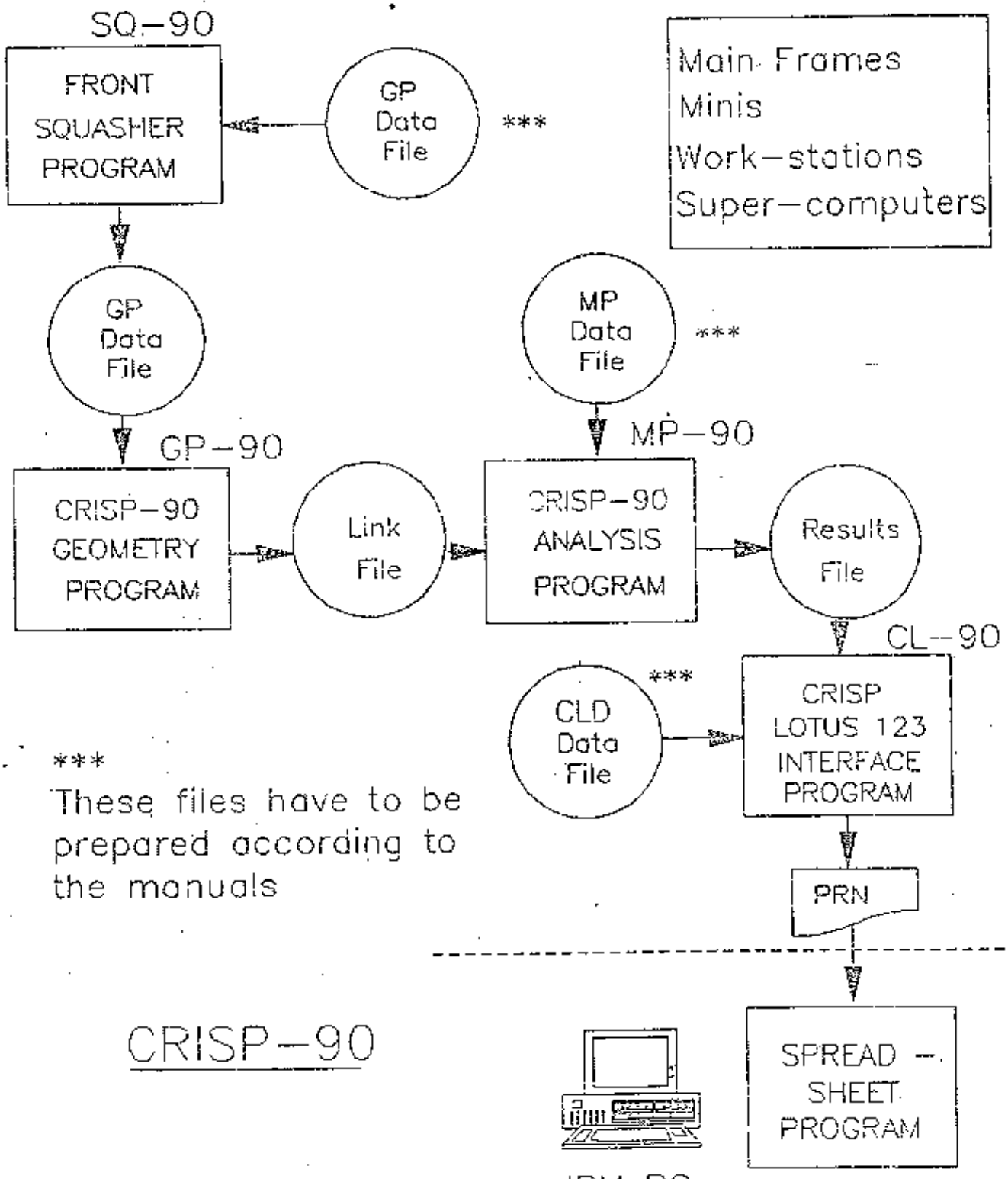



FIGURE 3.4 : CRISP-90 SUITE OF PROGRAMS ON ANY COMPUTER OTHER THAN A PC-386/486

( ) files are identified by extension names

# RUNNING CRISP-90 ON ANY COMPUTER OTHER THAN A PC-386




*** These files have to be prepared according to the manuals

CRISP-90

FIGURE 3.5

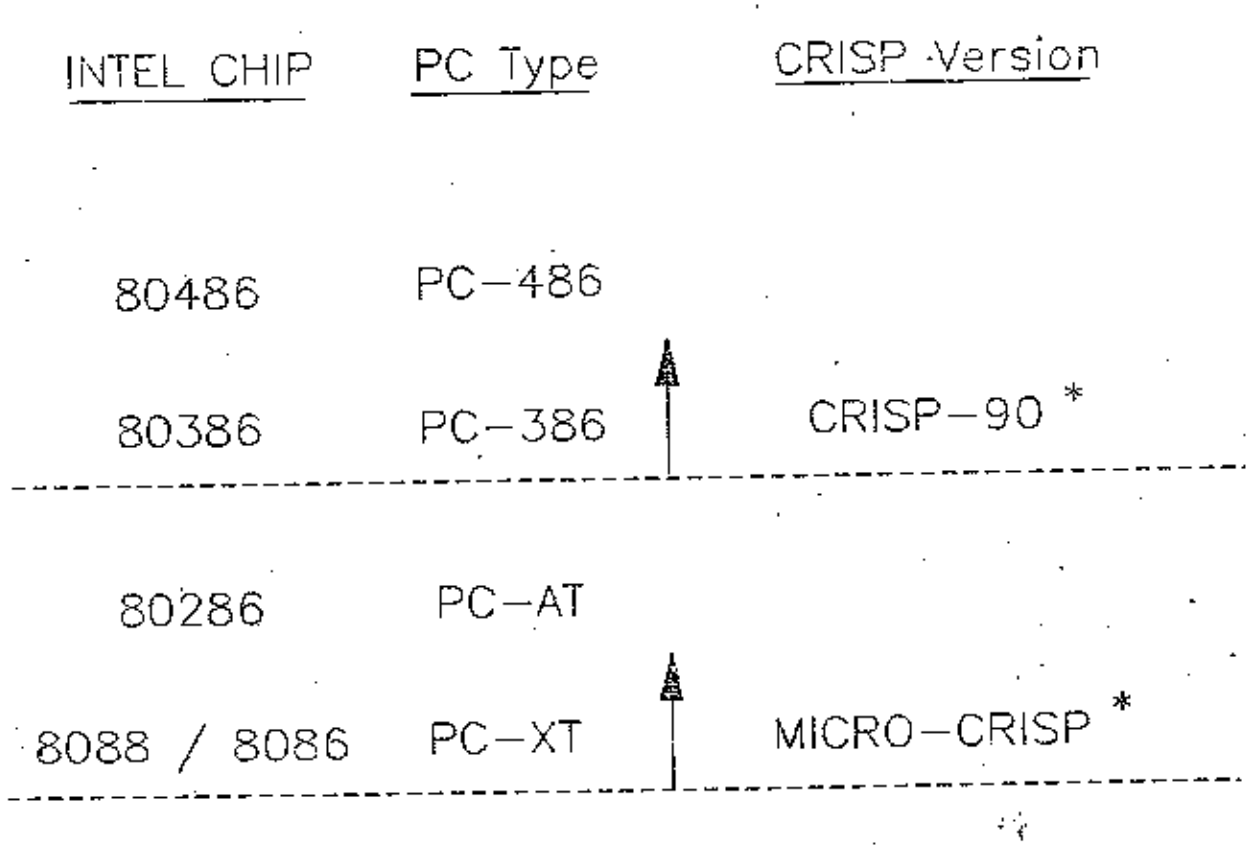



FIGURE 3.6 CRISP IN PC ENVIRONMENT

* These programs run on upwardly compatible hardware

# 4. MESH GENERATION PROGRAM - SMP

## 4.1 Introduction

The SMP program is not a general purpose mesh generation program nor is it versatile. It is not meant to be an alternative to commercial mesh generation programs. It does not have many checks and is not for users who want to generate the mesh on screen interactively without any preparation.

It is written with the assumption that the user has already drawn out the super-mesh in a sheet of paper and entered all the details required to generate the finite element mesh (Number of divisions, element types etc). If the user wants to subsequently change the super-mesh again it is advisable to draw out the changes in the sheet of paper first before starting to enter the data into the computer.

### *4.1.1 Interactive mode of data input*

The idea is to have all the information (correctly written down) before starting to use the SMP program. The SMP program is not meant for people who want to look at the super-mesh, half-way through the input of data and then want to carry on inputting data.

The SMP program expects all the information to be input in one go. The nodal co-ordinates and element nodal connectivity has to be entered completely if you have chosen the interactive input mode (see section 4.5 for alternatives). It is not possible to stop the program in the middle of the input and leave the program and come back later. All the information you have entered up to that point will be lost if you attempt to leave in the middle of data input in interactive mode. This only applies to entering the (i) super-nodes co-ordinates (ii) super-nodes connected to super-elements and the number of divisions.

However once the information about the super-mesh has been entered completely in the interactive mode the user can save that information by writing it to a "save" file (file with an extension

SVN). Once the main menu is reached then there is more flexibility and the user is free to leave the program and resume it at a later time. This is because all the information the user has entered has been safely saved.

*4.1.2 Input of data using a PRE file*

The alternative is to type in the information necessary to generate the super-mesh into a file with the extension PRE (ie prepared input data file, example : RETWALL1.PRE). The PRE file is simply an input file. The program reads the contents of this file if the option to read from a prepared input data file has been chosen. However the program does not modify or edit the contents of the PRE file. So it is not possible to input erroneous data through the PRE file and subsequently correct the errors in the data through the program and expect the corrections to be written to the PRE file. The SMP program only reads the contents of the PRE file and it does not write to it.

The logical step for the new users is to prepare the PRE file, Read it into the program and correct any mistakes and save the data. This way any changes made to data (ie corrections) can be written to the save file (SVN). Once you leave the SMP program you should rename the SVN file as SVO file before re-running the SMP program again. You will have to leave the CRISP-90 menu to do this. When running the SMP program again for the same analysis (either to correct mistakes in the input data or to make some changes to the super-mesh) you can read the SVO file to retrieve the previously input information about the super-mesh.

SVN - New SaVe file
SVO - Old SaVe file

Whenever the "save data" option is selected (option 6 in Main menu) the program rewinds the SVN file and writes the current data about the super-mesh. If you subsequently make any changes to the super-mesh you need to save the data again before quitting the program.

The SMP program cannot write to the SVO file neither can it read from a SVN file. This is the reason why a newly created SVN file has to be re-named as a SVO file before it can be read by the SMP program again.

Use the option 6 to save the data (main menu) once you have seen the plot of the super-mesh (option 4) and before using the checking options. This way even if the program happens to stop or crash the data you have input about the super-mesh will be written to the SVN file and hence saved. The option 6 to save the data about the super-mesh should be used before quitting the program. It does not matter at which stage of the program you decide to save the data.

Figure 4.1 shows the various files that are used with the SMP program. SMP8M.OUT is an output file which can be deleted when the SMP program has been run.

Figure 4.2 shows the different stages of the SMP program.

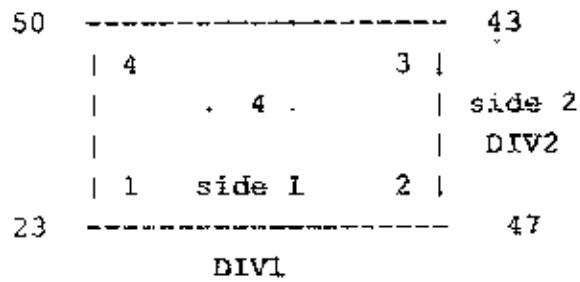
## 4.2 Manual for the input data

```
**************************************************
*                                                *
*          At present the program only           *
*    accepts replies in  upper case characters   *
*        (ensure that the CAPS LOCK is on)       *
**************************************************
```

This program is for interactive mesh generation and creation of the geometric input data file for the CRISP 90 program. The program is based on the concept of a supermesh which consists of a set of inter-connected super-elements. For consistency of naming conventions the nodes associated with the super-elements are referred to as super-nodes. Only quadrilaterals can be used in the super-mesh. It is not possible to use triangles in the super-mesh.

Each super-element is a 4 noded quadrilateral whose edges are assumed to be straight. Each super-element is assigned a CRISP element type number and a material zone number.

The super-nodes associated with a super-element is entered in anti-clockwise order starting with any super-node. The starting node has a special significance. The side between nodes 1 and 2 is referred to as side 1 and the one between nodes 2 and 3 is referred to as side 2.



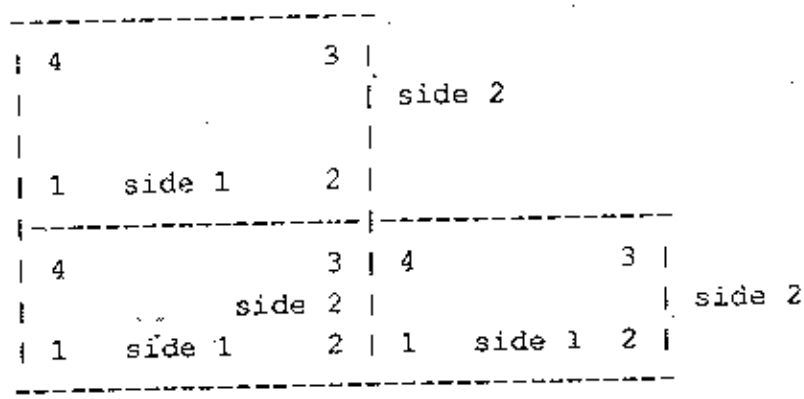

The numbers within the super-element near the 4 nodes represent the sequence in which these nodes were entered in the nodal-connectivity input. The numbers outside represent the actual super-node numbers.

The entries for the super-element 4 will be as follows:

| super element | nodes 1 | 2 | 3 | 4 |
|---|---|---|---|---|
| ... | .. | .. | .. | .. |
| 4 | 23 | 47 | ~~46~~ 48 | 50 |

Gaps are allowed in both the super-element number and the super-node numbers. However the user will find it simpler to adopt an ascending sequence of numbering for both super-element and super-node numbering without any gaps.

For consistency the same starting location for the node should be used for all super-elements. That way side 1 of all super-elements are lined up so are side 2 of all super-elements.




The additional information of how each of the super-element is to be sub-divided into FE elements is specified by tne division number for sides 1 and 2 (NDIV1, NDIV2). These division parameter at present represents a code which specifies the number of sub-division of the side into CRISP FE elements and also the relative size of the elements.

This division parameter is divided into 2 parts. If this consists of 1 or 2 digits then it is the number of divisions that side of the super-element is sub-divided into. Then all the elements have the same width. However if the division number is made up of 3 digits then the first digit defines the ratio of the first element side to the last element side created along that super-element side.

If the division number is made up of 4 digits then the first 2 digits define the factor with a decimal in between these 2 digits. The last 2 digits then define the number of divisions. By default when a factor is used the smaller sides are towards the first specified super-node. This can be changed by assigning a negative sign which means the larger sides will be towards first specified super-node.

The opposite sides of the super-element are meshed identically. The user has to ensure that along super-element sides that are shared the correct division parameter is specified. The program will check for any in-consistencies when the checking option is used.

Examples:

(a) x---------x---------x---------x---------x NDIV1 = 4
1 2

(b) x---x-----x------x-------x-----------x NDIV1 = 405
1 a b 2
b = 4 a

(c) x-----------x-------x-----x-----x---x NDIV1 = -405
1 b a 2
b = 4 a

(d) x---------x----------x-------------x NDIV1 = 1503
1 a b 2
b = 1.5 a

Input options:

1. Interactive input (input is from terminal)
2. Read from a previously saved file. The file name is assumed to be analysis.SVO ('analysis' is any name with up to 8 characters which the program will ask the user to specify at the beginning of each run).

   For example : WALL

   This implies a previously saved file of name WALL.SVO

3. Prepared input data file : (default name for the above example is WALL.PRE. This permits the user to type in the data into a file which is otherwise entered at the terminal interactively in response to questions by the program. (see below for the format of the PRE file).

These are only the default file names and the user is free to override it. Then the user should specify the file name including the extension.

Example : WALL.OLD

The save file created by the program from the current run will be

named WALL.SVN. This file contains the information about the super-mesh. Similarly the CRISP GP datafile created by the program will be given the name WALL.GPD. If files of these names already exist then these will be over-written.

If the user had created a SVN file from a previous run he/she should re-name it as a SVO file before re-running the program. Otherwise the program cannot read the information from that file.

### *4.2.1 Format for the PRE file*

Record

| | | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| A | NSE | NSN | NN | | | | | |
| B | KLT | KMAT | KSD1 | KSD2 | | | | |
| | ..... NSN records .... | | | | | | | |
| C | NODE | X | Y | | | | | |
| | ..... NSE records .... | | | | | | | |
| D | KSE | NS1 | NS2 | NS3 | NS4 | <LTYP> | <MAT> | <NDIV1> | <NDIV2> |

---

Explanations:

Record

A
- NSE - Number of super-elements in super-mesh
- NSN - Number of super-nodes in super-mesh
- NN - Ø

B
- KLT - default element type number for all super-elements
- KMAT - default material zone number for all super-elements
- KSD1 - default number of divisions along side 1 of all super-elements
- KSD2 - default number of divisions along side 2 of all super-elements

C
- NODE - super-node number
- X, Y - co-ordinates

D KSE - super-element number
NS1, NS2, NS3, NS4 - super-nodes associated with super-element KSE (should be in counter-clockwise direction)
LTYP - CRISP element type number for super-element KSE (omit if KLT is not zero)
MAT - material zone number for super-element KSE (omit if KMAT is not zero)
NDIV1 - division parameter for side 1 of super-element KSE (omit if KSD1 is not zero)
NDIV2 - division parameter for side 2 of super-element KSE (omit if KSD2 is not zero)

* these are default values for all the super-elements. If these parameters are specified a zero value that would indicate that the corresponding entry in record D (within angled bracket must be specified).

For example if KMAT = 0 then MAT for each super-element must be specified in record D. However if KMAT = 1 then MAT must be omitted from record D for all super-elements (all super elements will be assigned the default material zone number of 1). The other parameters operate in the same manner.

record B : If the parameters KLT KMAT KSD1 and KSD2 are different for super-element then set the corresponding value to 0 in record B. This simply means that these parameters will be specified in records D for all the super-elements. Record D is only used to reduce the data input via records D. For example if the material zone number is the same for all super-elements then set KMAT = 1 in record B. Then MAT can be omitted from record D for all the super-elements. Similarly for KLT, KSD1 and KSD2. If KMAT is different from one super-element to another then you have to set KMAT = 0 in record B. Then the program would expect you to specify MAT for each super-element in record D.

If all super-elements have identical values for any of the parameters KLT, KMAT, KSD1 and KSD2 then these can be specified in record B and the corresponding entry within angled brackets must be omitted from record D for all super-elements.

## 4.3 Procedure

The following sequence is recommended in running the SMP program.

1) Draw out the the super-mesh and number all the super-nodes and super elements. Read off the super-node co-ordinates and the super-nodes associated with each super-element type and write it down in a piece of paper. Also enter the material zone number and CRISP element type to be generated within each super-element. Enter the number of divisions for side 1 and 2 for each super-element.

2) You can either use option 1 (interactive input) or option 3 (prepared input data file). For new users option 3 is recommended. Because it is easy to make mistakes in interactive mode and the SMP program is not very error-tolerant during this phase of the program. As you become more used to the SMP program you can start using the interactive mode of data input. To use option 3 type in all the information into a file (PRE file; give it an extension name of PRE. Example : if you are going to call your analysis RETWALL1 then give it the name RETWALL1.PRE).

3) Follow the instructions given in section 4.4 in running the SMP program. Once the data has been input and the program displays the main menu plot the super-mesh (option 4) and visually check the super-mesh. If there appears to be mistakes, return to the main menu and use the option 3 to display the data input. Check these with the information you had written down in the first place. Return to main menu and use option 6 to save the data (The information input so far about the super-mesh is written to the SVN file. For this example the file RETWALL1.SVN is created). Correct any mistakes using the option 2 to make changes. Return to main menu, plot the super-mesh again and save the data again if you had made any changes. Run the checking options and then return to the main menu. As you may have noticed you don't have to strictly follow the options given in main menu in the sequence they appear in the menu.

4) create the finite element mesh (option 5 in main menu). The program would indicate how many elements and nodes have been

generated in the finite element mesh. This is followed by the the CRISP DATA/PLOT options menu. Every time you create a new finite element mesh this menu automatically appears on the screen. This menu is one level below the main menu and allows you to create the CRISP Geometry program data file and also view the finite element mesh. See section 4.7 for instructions on viewing the f.e. mesh.

At this stage you can choose to look at the finite element mesh (again there is a separate menu offering a number of options). Viewing the various plots and the options available needs getting used to. Facilities are available to zoom in and look at parts of the finite element mesh.

5) once you have looked at the finite element mesh choose option 9 to return to the previous menu (CRISP DATA/PLOT options menu). At this stage choose the option to create the Geometry program data file. The program will first prompt you for the title to be used for the analysis (up to 80 characters). Then once the program issues a message to say that the CRISP GP data file has been created then you can quit the program or return to the main menu, save the (super-mesh data) data (if you had not done it before) and quit the program. It is recommended that you quit the program from main menu in preference to quitting from anywhere else from the program.

6) When within the SMP program, once you have reached the main menu you can make changes to the super-mesh as many times as you want and also create the finite element mesh as many times as you want. Saving the information about the super-mesh on the SVN file and creating the Geometry program data file (GPD file) can be done any number of times. Each time you write to any of these files the previous contents are over-written by the latest set of data/information.

At present the following CRISP element types cannot be included in the finite element meshes generated by the SMP program.

| type | description |
|---|---|
| 1 | 3 noded bar element (used for 2D plane strain) |
| 12 | 3 noded beam element " |
| 13 | 6 noded slip element " |

The super-mesh can only be made up of 4 noded quadrilateral elements. It is not possible to have a triangular super-element. However each 4 noded quadrilateral super-elements can be sub-divided into either triangular or quadrilateral CRISP finite elements.

There are plans to introduce a 8 noded quadrilateral super-element with curved sides for modelling tunnels and buried pipes.

## 4.4 Running the Super Mesh Program - SMP

Make sure that the CAPS LOCK key is on. First of all before attempting to run the SMP program draw the super-mesh in a sheet of paper. Enter all the information in that figure : super-node numbers, super-element numbers element types, number of divisions. Enter the super-node co-ordinates and the list of super-elements associated with each super-element in another sheet of paper. Keep these 2 sheets of paper handy when running the various modules of the CRISP-90 suite of programs. See Fig. 4.3 for example. In this figure

Z - material zone number
E - CRISP element type number
D - divisions along element sides

(see also table 4.1)

There are 3 ways of entering information about the super-mesh into the SMP program.

a) Interactive input from terminal
(Typing in data as the program prompts for it).

b) Read in a previously saved file (only if you have used the program before for the current analysis).

c) Reading in a prepared data file which contains information about the super-mesh.

If you are starting a fresh analysis you can use either option 1 or 3.

If you have used the program before (say for example for a retaining wall analysis and the file is identified by WALL1. Then the program would have created a file called WALL1.SVN if you had saved the previous results). If you want to re-use this file and make some changes rename this file as the SVO file.

REN WALL1.SVN WALL1.SVO or COPY WALL1.SVN WALL1.SVO

If you are not already in the CRISP-90 menu type CRISP90.

If you are in the CRISP-90 menu then choose the option to run the SMP and PRE-ADG programs (option 1).

1) The screen with the header "Confirm Graphics Devices" appears on the screen.

Press the RETURN key once.

2) The super mesh program banner is printed by the program.

Press any key to continue.

This is followed by the input options listed below :

```
ENTER 1 - FOR INPUT FROM TERMINAL
ENTER 2 - FOR INPUT FROM A (SAVED) DATA FILE (SVO)
ENTER 3 - FOR INPUT FROM A (PREPARED) DATA FILE (PRE)
ENTER 4 - QUIT THE PROGRAM
```

These correspond to the 3 options explained above. Choose the

option which is appropriate to your case by typing a number in the range 1 to 4 and press the RETURN key.

Then the program will prompt for the file identifier with the following message :

ENTER NAME FOR THE FILE (UPTO 8 CHARACTERS). WITHOUT AN EXTENSION.

Type the filename
(for the above example ) WALL1

Then the program will ask you whether you want to create a SAVE file for subsequent runs of the program.

Type Y

The program will ask you to enter the name of the SAVE file. Within brackets the default file name will de displayed. If you accept that name then press the RETURN key. If you want you can type in a different file name at this stage.

Then the program will ask you whether you want to create CRISP GP data file.

Type Y

The program will ask you to enter the name of the GP data file. Within brackets the default file name will de displayed. If you accept that name then press the RETURN key. If you want you can type in a different file name at this stage.

If the files of the same name already exists then the program will insist that you type a different filename to prevent overwriting. If typing a different name you have to include the extension as well. Example : WALL1B.GPD

If you make any mistakes when inputting the data do not attempt to correct it unless the program spots it and allows you to re-type the data. The mistakes can be corrected when the main menu is offered later. Choose option 2 to change the super-element and super-nodes entries. Before doing this you can check the data you had typed by choosing option 1 or by plotting the super-mesh

(option 4).

When you have finished running the program do not forget to save the data before exiting. This is done in the main menu using option 6.

| | | |
|---|---|---|
| Input files | : | WALL1.SVO - previously created (saved) file<br>WALL1.PRE - data typed before-hand. |
| Output files | : | WALL1.SVN - currently saved data file.<br>WALL1.GPD - Input data file for the Geometry program. |

## 4.5 Inputting super-mesh data in stages

It is not necessary to input the complete data about the super-mesh in one go using the interactive mode. For example consider the super-mesh shown in Fig. 4.4.

You could enter the co-ordinates of the first 6 super-nodes (enter 6 when the program asks for the total number of super-nodes). Then input the super-element to super-node connectivity for the first 2 super-elements (enter 2 when the program prompts for the total number of super-elements). This corresponds to part of the super-mesh as shown in Fig. 4.4(a).

Once the main menu has been reached check the plot of the super-mesh (option 4). Then save the data input so far using option 6.

Then choose option 2 to make changes to the super-mesh. This will give a menu with the following options.

Change options

1) Change super-element entries
2) add a new super-element
3) delete a super-element
4) change super-node entries
5) add a new super-node
6) delete a super-node
7) return to main menu

Choose first option 5 to add in the super-nodes (7, 8, 9). Do it one at a time, Starting with super-node 7. After you have entered the co-ordinates of super-node 7, the program will ask

```
DO YOU WANT TO ADD ANYMORE (NEW)  SUPER-NODES ?
ENTER  Y  OR  N
```

Answer Y and then input co-ordinates of super-node 8. Do the same for node 9 and then answer 'N' to the above question.

This will return to the above 'Change options' menu. Now choose option 2 to add the super-elements 3 and 4. The program will prompt for the super-element number and then ask you to re-type the number again for confirmation (to avoid any errors). Enter 3 for the super-element number. It will also prompt for the CRISP element type, material zone number and the number of divisions for sides 1 and 2. Then it will ask the following question.

```
DO YOU WANT TO ADD ANYMORE (NEW)  SUPER-ELEMENTS  ?
ENTER  Y  OR  N
```

Answer Y and then input the details for super-element 4. Once this is done answer N to the above question which will return to the 'Change option' menu. Choose option 7 to return to the main menu. Look at the super-mesh plot and save the data. Then use the 'checking' and 'display' options to check that the information input are correct before generating the finite element mesh.

Note that in adding to the initially input super-mesh always enter the additional super-node details (in the above example 7, 8 and 9) before you input the information about the additional super-element to super-node connectivity. Otherwise you will introduce errors (you will lose the super-nodes numbers you have entered with the super-element numbers because these are unknown to the program).

## 4.6 Limitations in using the SMP Program

### *4.6.1 Introduction*

This note has been prepared for people who have been using the CRISP-84 program and who intend to update to the CRISP-90 program. The addition of new modules are intended to make the data preparation easier for both Geometry and Main (analysis) programs in a PC-386 compatible environment. However using the SMP program restricts the type of CRISP analysis that can be carried out.

The limitations of the SMP program are described below :

### *4.6.2 Limitations in using the SMP Program*

(a) It is not possible to carry out 3 dimensional analysis or 2 dimensional plane strain analysis with bar/beam/slip elements.

(b) Maximum of 10 material zones in any analysis. CRISP MP allows for 25 material zones.

(c) It is not possible to specify super-imposed elements and be able to distinguish between the 2 sets of finite elements that are generated ie 2 sets of elements occupying the same region - useful in analysis where soil is excavated to be replaced by a structure (example : retaining wall). Two sets of super-elements can occupy the same region. The SMP will indeed create 2 sets of finite elements occupying this region, however the ADG program (see chapter 6) will not be able to distinguish between the two sets, when attempting to remove or add these super-elements. This is likely to lead to hitherto unknown problems and should be avoided by the users.

The use of bar/beam/slip elements may be possible in future updates.

## 4.7 View commands

These commands can be used when the SMP program has drawn the finite element mesh on the screen. These commands are effective for all the f.e. mesh plot options that are available provided in the menu.

A f.e.mesh is drawn after any output command (?,E or G). Therefore window settings can be effected before the f.e.mesh is drawn.

Multiple commands can appear on each line, optionally separated by spaces, commas, or semicolons. If no separator is used.

? The current f.e.mesh is redrawn through the current window.

D[x] Move the new window Down. The default movement is half the window height, but a fraction of the height may be given as an additional real argument

E Draw the Entire f.e.mesh at the largest reasonable scale. The f.e.mesh will be oriented according to the last rotation command.

G Go. The f.e.mesh is redrawn as seen through the new window, which becomes current.

I[x] Move the new window In. The default movement is to half the previous window size, but a ratio in which the window is to be reduced may be given as an additional real argument.

L[x] Move the new window Left. The default movement is by half the window width, but a fraction of the width may be given as an additional argument.

O[x] Move the new window Out. The default movement is to twice the previous window size, but a number by which the window size is to be multiplied may be given as an additional real argument.

R[x] Move the new window Right. The default movement is by half the window width, but a fraction of the width may be given as an additional real argument.

W Return to the previous menu (wind up).

## 4.18 Errors in running the SMP program

1) Mistake in typing the analysis identifier.

If you notice that you have made a mistake in typing the analysis identifier, the program is going to use that name to search for PRE or SVO files and report that these files donot exist. Also the program will use the wrong name to create the SVN and GPD files. In order to avoid this, as soon as you notice the mistake press the CTRL/BREAK keys, and when the program prompts with the following message

Terminate batch job (Y/N)?

enter Y and then press the RETURN key to quit the program.

The DOS prompt (>) will be displayed at this point.

Type CLS (to clear the screen) followed by MODE 80 (to restore the text mode). Then type CRISP90 to get back the CRISP-90 menu and re-run the SMP program. However if you have already started to input super-mesh data pressing the CTRL/BREAK will cause you to lose all that data. Carry on with the running of the program until you reach 'main menu'. Save the data and then quit the program. Then use the REN or COPY commands to correct the mistakes in the file name.

If you have typed WALK1 instead of WALL1 then

```
COPY  WALK1.* WALL1.*
DEL WALK1.*
```

2) If you have forgotten to rename the SVN file from a previous session as a SVO file.

Press the CTRL/BREAK keys first and follow the instructions given above.

3) If you are stuck in the finite element mesh plot and unable to quit that screen.

Make sure that the 'CAPS LOCK' is on. Press the RETURN ket once. Then type N and press the RETURN key to get back to F.E. Plot

options menu.

4) Unexpected end-of-program.

It is possible that the program may stop or abort due to some errors in the super-mesh data. When this happens check the input data on the super-mesh (PRE file, super-mesh plot etc) for any possible errors. Also check that the division number specified are the same for super-element sides which are shared. If there are no apparent data errors in the input data then it could be due to some bug in the program. If the problem persists send a disk with the relevant files (PRE, SVO and SVN) to the authors of the program. This information would help in the investigation and fixing of the bugs.

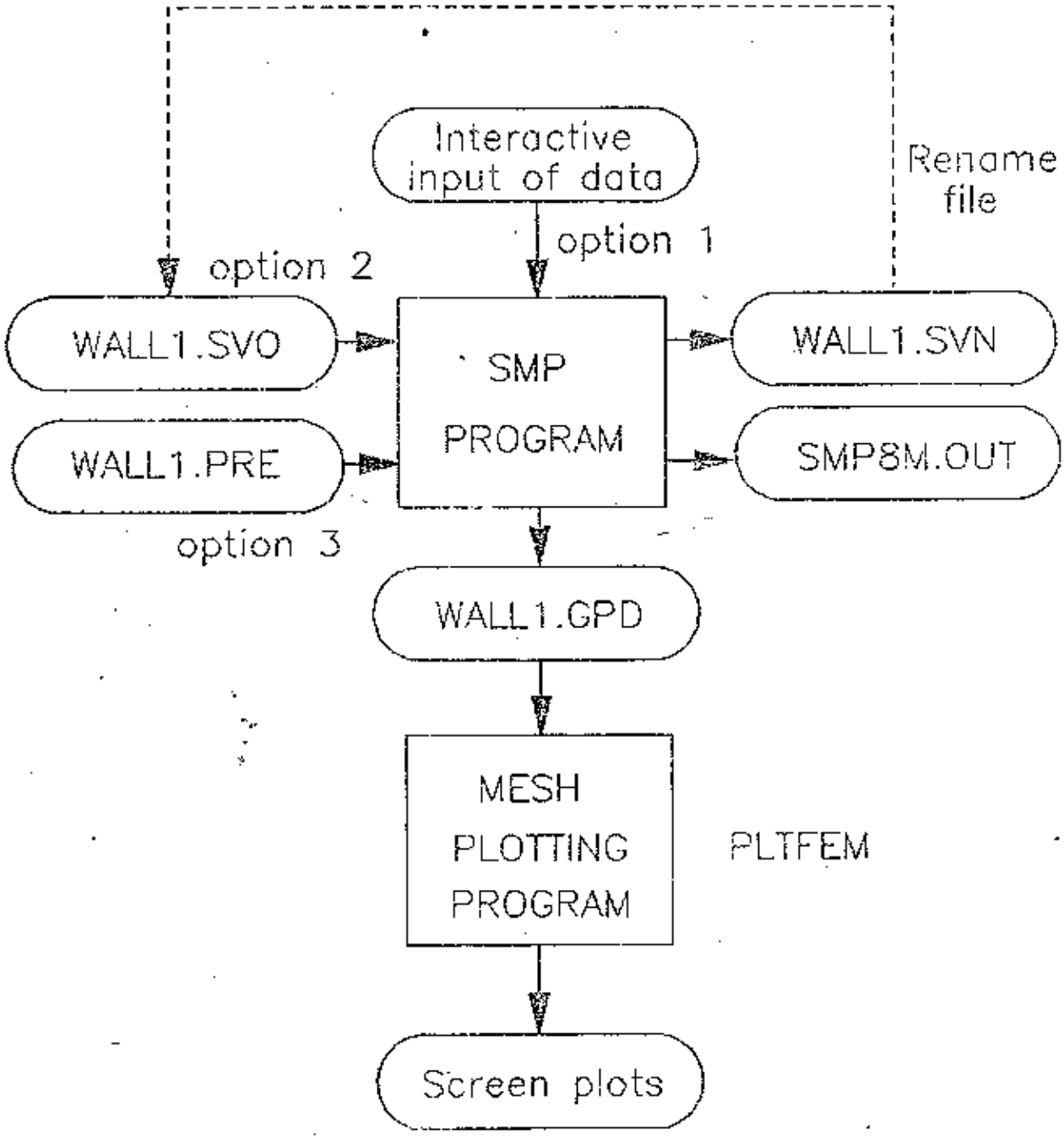



Figure 4.1 Files used by SMP

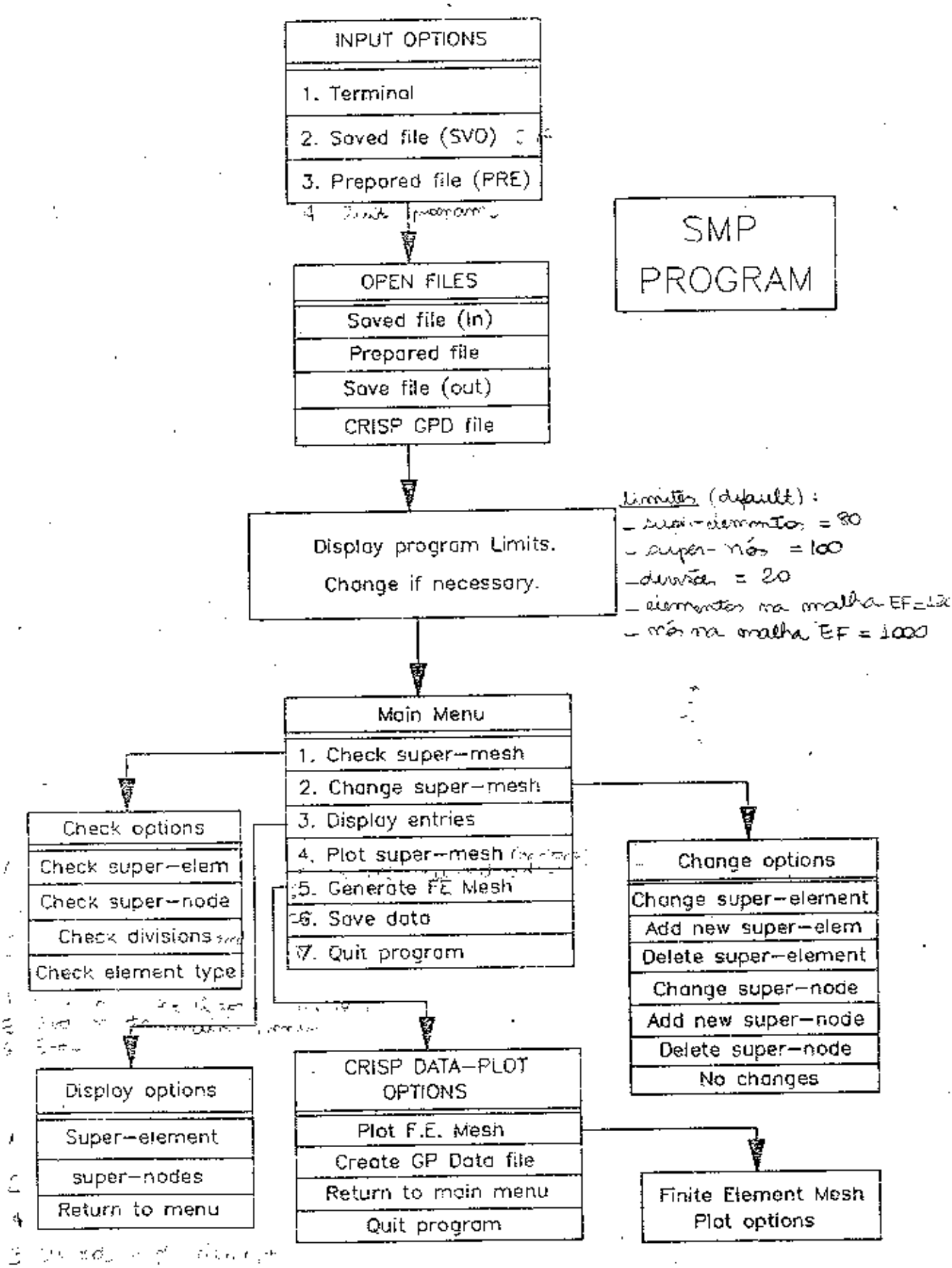
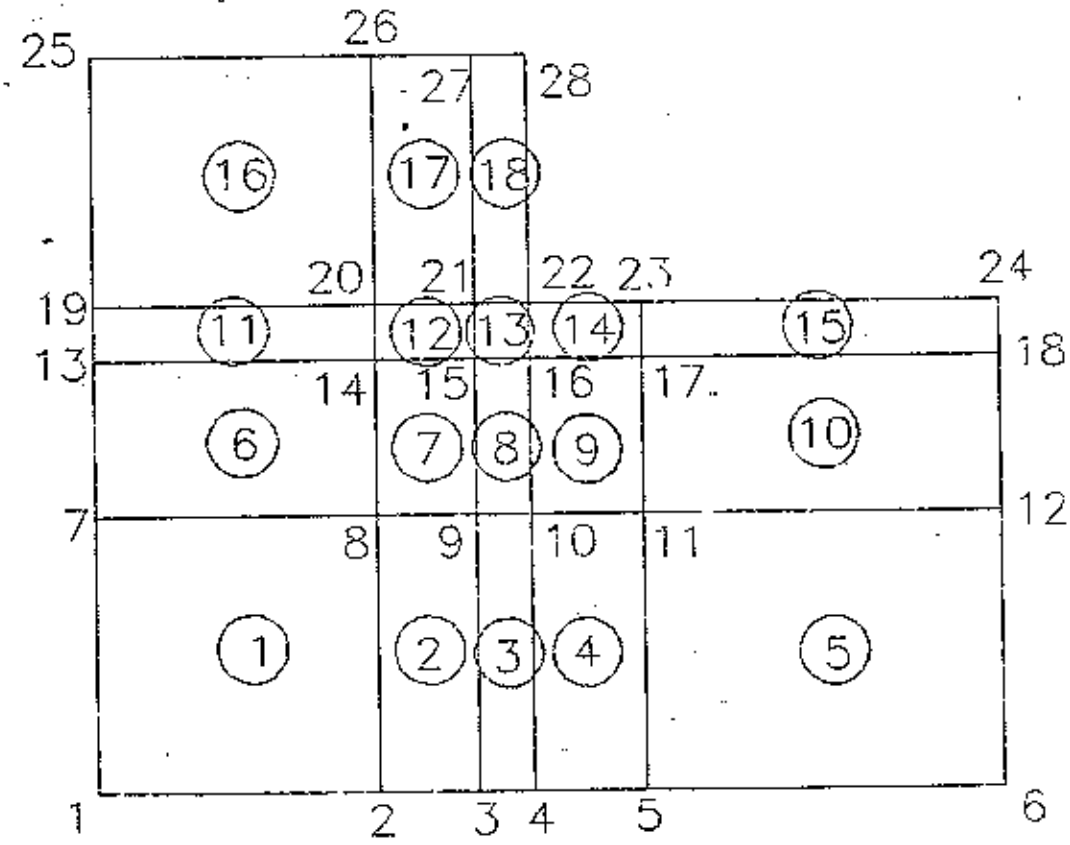



Figure 4.2 Different stages of the SMP program




(Example : ABTWALL)

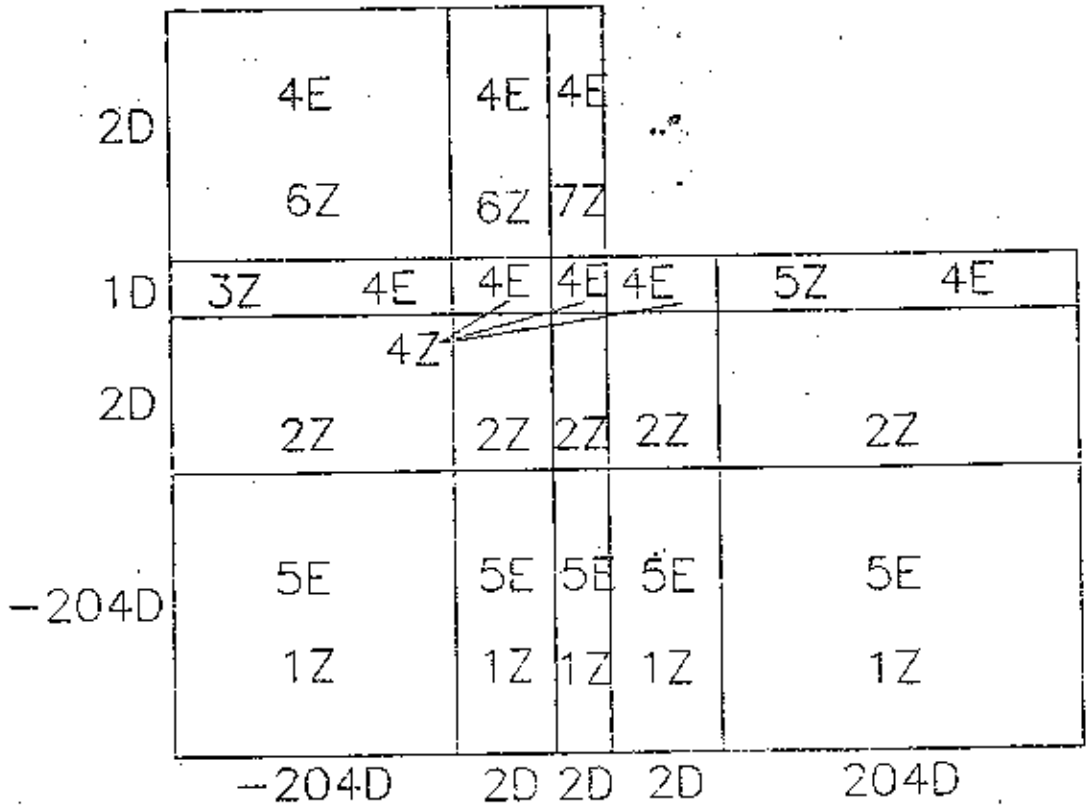
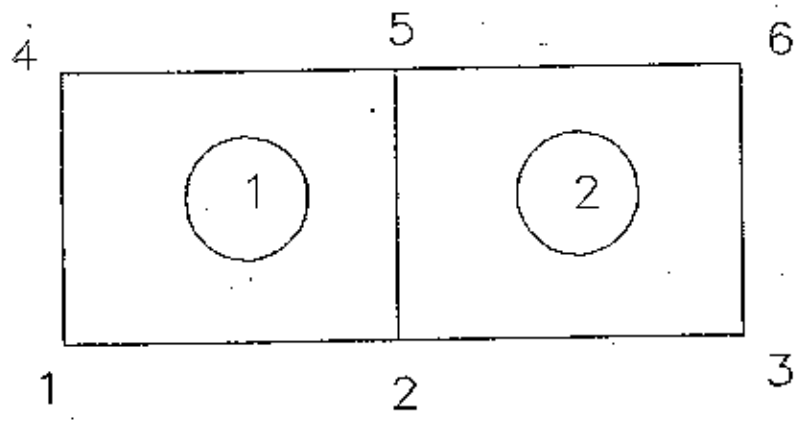
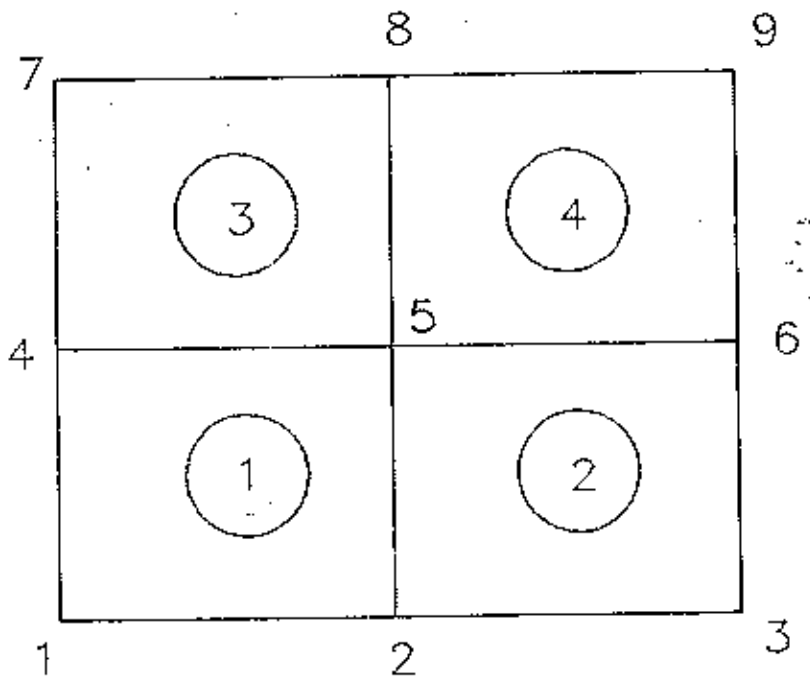



Figure 4.3 SMP input data




(a)




(b)

Fig. 4.4 Creating a super–mesh in stages

# 5. PRE-ADG PROGRAM

## 5.1 Introduction

This program is run back to back with the SMP program ie it is run immediately after the SMP program without any prompting from the user. It is run every time the SMP program is run irrespective of whether a new f.e. mesh is generated or an existing one is modified.

If the user senses that the f.e. mesh generated in the SMP program is not correct or that the input to the SMP program is only partially complete then he/she should press CTRL/C key after quitting the SMP program and just before starting the PRE-ADG program. This is the only instance the CTRL/C key should be pressed to prevent the PRE-ADG program from running. In the normal circumstances always run the PRE-ADG program, otherwise you may encounter problems with the ADG program.

As the name suggests the program sets-up various arrays for use in the ADG program (see chapter 6). The main task of this program is to prepare work-out the locations (for all boundary super-element sides) for displaying icons that may be assigned to the super-element sides which lie along the mesh boundary. All such locations are then displayed with the super-mesh. This is only done for visual check and the user need not be unduely concerned about the arrangements of these (icon) locations or sites at this point. The program displaying the above locations means that it has carried out its task successfully. If the above plot does not appear on the screen then there may be errors in the super-mesh data you have input to the SMP program. Re-run the SMP program and carry out all the checks and display the super-node numbers and co-ordinates and super-element numbers and the super-element to super-node connectivity.

It is always a good idea to sketch the super-mesh in a piece of paper which can be useful when running the PRE-ADG and ADG programs.

PRE-ADG program works with the super-mesh and not the f.e. mesh.

Figure 5.1 shows the display which comes up on the screen when you run the 'footing0' example. There are always 2 icon positions allocated to each super-element side (which are along the mesh boundary) which are sufficiently long enough to display the icons (ie for super- element sides which are longer than twice the width of the icons, see chapter 6 for information about the icons). Some of the arrangements used are shown in Figure 5.2. Some-times the line separating the locations may not be drawn. This also should not cause any concern to the user. In some occasions the icon locations may be displayed at a distance from the super-element-sides. This may happen from time to time. Any pair of icon locations (forming a rectangle) displayed is always associated with a super-element-side which forms the mesh boundary (even though it might be sometimes difficult to work out exactly to which super-element side it belongs to).

Wherever possible the locations are positioned symmetrically about the super-element side mid-point. However these icon locations may be shifted sideways to accommodate icons associated with adjacent super-element sides. The program has a scheme to shift and move icons around to make it possible to display as many icon locations as possible.

Super-element sides which are too short for 2 icons to be displayed adjacent to it are designated 'short' super-element sides and icons when assigned to these sides are not displayed along side it. These are then referred to as hidden icons.

It should be remembered that even though the PRE-ADG program displays icon locations it does not assign any icons to these lines. Icons are assigned to super-element sides by the user in the ADG program. PRE-ADG program just works out where to display these icons in anticipation.

The PRE-ADG program does not require any user input.

It reads the SVN file created by the SMP program and combines the information written to the CRISP90.SDG file and creates an SMD file. The locations of the icons are also written to this SMD file (see Figure 6.1 from the next chapter). The ADG program reads the

SMD file.

## 5.2 A further example

Figure 5.3 shows another example of the plot generated by the PRE-ADG program. The following super-element sides are too small to fit a pair of icon locations and hence considered as 'short' sides. Any icons assigned to these sides in the ADG program are designated 'hidden' and these are not displayed next to these sides.

2-3 3-4 4-5 18-24 22-23 26-27 13-19

In some of the rectangles the partition line is not shown (example : sides 19-25, 25-26) and this should not cause any concern to the user.

## 5.3 Running the PRE-ADG Program

This program is run immediately after the SMP program. It does not require any input data from the user.

1) The screen with the header "Confirm Graphics Devices" appears on the screen.

Press the RETURN key once.

2) The super mesh program banner is printed by the program.

Press any key to continue.

3) After a short delay the program will display the super-mesh and also the locations of icons. The more super-elements present in the mesh the longer will be the delay.

Press the RETURN key once, to exit the program and return to the CRISP-90 menu.

## 5.4 Errors in running the PRE-ADG program

(a) If the program crashes without producing a plot of the super-mesh with the icon locations then it is likely that there are some errors in the super-mesh data input to the SMP program. Re-run the SMP program and this time check all the data thoroughly using the checking option and display options. Make sure that this data is exactly what you had intended and that it is consistent with the manual (see section 4.2).

(b) If the plot of the super-mesh on display is different from the super-mesh you have been working with then it is likely that you did not save the previously created data or probably you did not generate the f.e. mesh ie the necessary files were not updated by the SMP program (CRISP90.SDG, *.SVN). In all probability the PRE-ADG program is reading the data from an old CRISP90.SDG file.

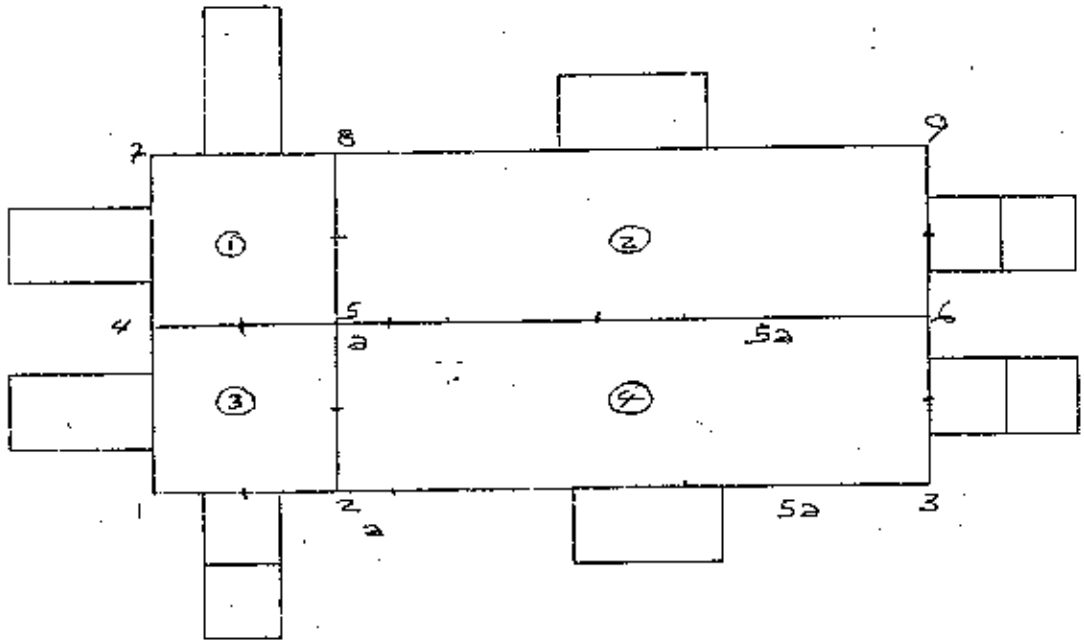



Figure 5.1 Locations of icons – FOOTING0

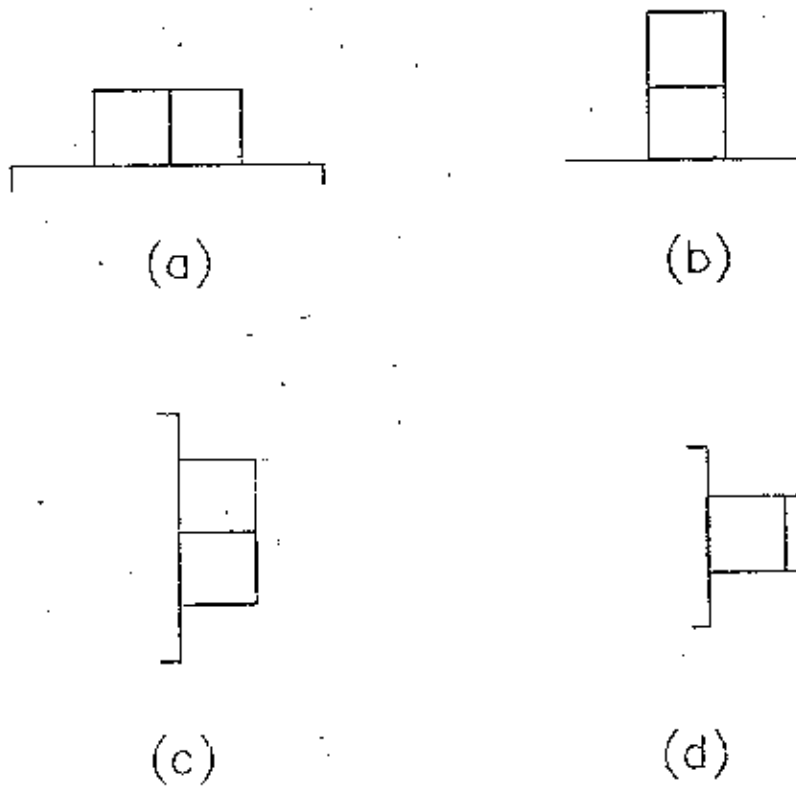



Figure 5.2 Arrangements of icons.

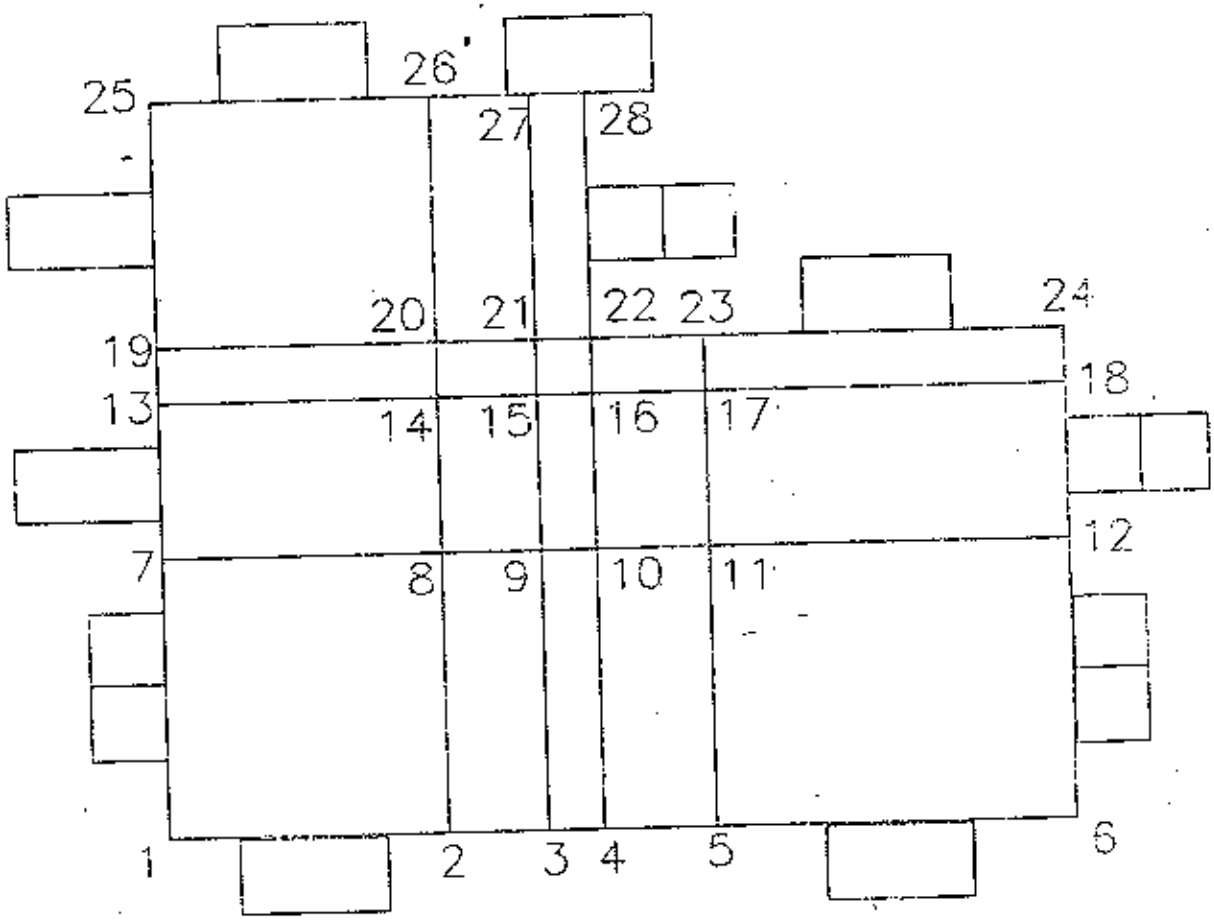



Figure 5.3 Locations of icons
(Example : ABTWALL)

# 6. ANALYSIS DATA GENERATION (ADG) PROGRAM

## 6.1 Introduction

Section 6.2 is an introduction to the ADG program. It is recommended reading for the first time user. Section 6.3 deals with the input of data using icons when using the ADG program. Section 6.4 deals with the specifying of a linear variation of in situ stresses. Section 6.5 covers the same ground as section 6.2 but is for use as a quick reference. Section 6.6 gives the limitations in using the ADG program to generate the input data file to the MP. Section 6.7 resolves some of the apparent conflicts in the information input to the ADG program and the input specification for the MP program (see chapters 3 and 4 of CRISP brown manual - volume 1). Section 6.8 deals with the generation of MPD files for use in a stop/restart run of MP.

## 6.2 Program description

The Analysis Data Generation (ADG) Program creates an input data file for the CRISP Main Program from data input by the user interactively. In order to use this program the user has to generate the finite element mesh using Mesh Generation Program (SMP). This is because the ADG program requires the details about the super mesh created by the SMP program. See Figure 6.1 which shows the relationship between the three program modules : SMP, PRE-ADG and ADG.

The ADG program consists of 4 types of pages

1) Information pages - These do not require any input from the user and is simply to convey some information or message to the user.

2) Menu pages - The user chooses an option by using the up or down arrow keys (in some instances Page up or Down key for scrolling 'pages').

3) Data input pages - Typing in data into certain fields.

Usually overtyping 'default' values.

4) Icon input page - This is used to specify displacement and pore pressure fixities and also removal and addition of super-elements.

Figure 6.2 shows the ADG program structure.

There are 2 options in running the ADG program.

(1) A fresh analysis where you have **only** used the SMP program to generate the finite element mesh ie you have not used the ADG program before for this particular finite element mesh. The information required by the ADG program is stored in a SMD file.

(2) You have used the ADG program before to generate the Main Program Data (MPD) file and you want to make some changes to the input data. If you had saved the data from the previous session, then you would have created an ADG file. Then you need to read in the ADG file in addition to the SMD file.

The ADG program is run by choosing the appropriate option from the CRISP-90 menu. This will produce the Metawindow graphics device page. Press the RETURN key and after a short pause the ADG program banner will be displayed. Press any key to continue.

The program will display the current directory name in the SMD directory page (page 1). In general all the relevant files will be stored in the current directory. In that case simply press the **RETURN** key for the next page (page 2). The program will then display **all** the file names with the extension SMD in that directory. If there are more than 10 such files then these are displayed in pages of 10. The current page number is displayed just above the file names. Use the 'Up and Down arrow' keys to make the choice and press the RETURN key.

If the file you want is not displayed in the current page then use the 'Page Up' or 'Page Down' key to scroll the page until you find the page which contains the file you want to use. If you find out that the file you want to use is not in the current directory then press the 'F2' key which should get you back to the SMD

directory page and then you can typeover the full path name to the directory you want to search for the files. Press the RETURN key for the program to display the SMD files in that directory.

If for some reason you want to quit the program at this stage press the 'F3' key which will cause the program to stop.

However if you decide to continue with the program .....

Once the SMD file has been selected the program would present a menu with 2 choices (page 3).

(a) Editing an existing ADG file
(b) Creating a new ADG file

As explained before if you have already used the ADG program and want to make some modification to previously input data then choose option (a). Otherwise choose option (b).

*6.2.1 Editing an existing ADG file*

The program will display the current directory name in the ADG directory page (page 4). In general all the relevant files will be stored in the current directory. In that case simply press the RETURN key for the next page (page 5). The program will then display all the file names with the extension ADG in that directory. If there are more than 10 such files then these are displayed in pages of 10. The current page number is displayed just above the file names. Use the 'Up and Down arrow' keys to make the choice and press the RETURN key.

If the file you want is not displayed in the current page then use the 'Page Up' or 'Page Down' key to scroll the page until you find the page which contains the file you want to use. If you find out that the file you want to use is not in the current directory then press the F2 key which should get you back to the ADG directory page and then you can overtype the full path name to the directory you want to search for the files. Press the RETURN key for the program to display the ADG files in that directory.

At this stage the program will read a code from both the SMD and ADG files to ensure they apply to the same mesh. If the codes are different then you are attempting to use the wrong ADG file with

a particular SMD file. Then the program will display a menu with the following choices:

a) Change SMD directory name
b) Change SMD file name
c) Change ADG directory name
d) Change ADG file name
e) Quit the program

This allows you to change the SMD and ADG files you have chosen.

*6.2.2 Creating a new ADG file*

At this stage the program will display the ADG LIMITS page (page 6). Press the RETURN key to continue.

*6.2.3 LIMITS Page*

The following limits apply to the ADG program for the present.

| | | | |
|---|---|---|---|
| Maximum number of Increment blocks | = | 10 | *** |
| Maximum number of material zones | = | 10 | *** |
| Maximum number of increments in a block | = | 50 | |

*** These restrictions only apply to the MPD files created by the ADG program. These are not the limits that apply to the CRISP MP.

When you press the RETURN key the following self-explanatory message will appear

*** reading data from SMD file ***

This is followed by the geometry details page.

*6.2.4 GEOMETRY DETAILS - DATA SUMMARY Page 1 (page ref no 7)*

The number of super-elements and super-nodes in the super-mesh and the number of elements and nodes in the finite element mesh are displayed by the program to refresh the user's memory. The different elements types are also listed under the different categories.

Press the RETURN key to continue.

There will be a short pause after the following message is displayed on the screen.

*** Carrying out calculations ***

*6.2.5 ANALYSIS DETAILS - DATA SUMMARY Page 2 (page ref no 8)*

This page will be displayed only if you are editing an existing ADG file. It gives a summary of the information previously input.

It gives the details of what the material types each zone is chosen to be and the number of increment blocks and analysis type (whether PLANE STRAIN or AXISYMMETRIC) and whether the analysis is CONSOLIDATION or DRAINED/UNDRAINED type. Press the RETURN key to continue the program.

*6.2.6 TITLE SECTION (consists of 3 Pages)*

This is the beginning of the ADG program proper.

Page 1 : Input the TITLE to be used in the MPD program (defaults to CRISP ANALYSIS) (page ref no 9)

Page 2 : Control parameters for the analysis. (page ref no 10)

Page 3 : Control parameters for the printed output. (page ref no 11)

You are expected to type the data in the appropriate fields. Use the 4 arrow keys to move around. Once you have entered all the relevant information for that page press the RETURN key to continue. Wherever possible the data you have typed will be checked for admissibility and you will be forced to correct any mistakes the program 'thinks' you have made.

Particular attention should be given to the Number of increment blocks. Check that you enter the correct number. This ensures that the correct number of options are displayed later in the increment block menu. Of course you can come back to this page and correct it, but it saves you that trouble in the first place.

Warning : The current version does not include an overall-check of all the parameters that you have input are admissible. It is hoped that these checks will be implemented in future updates. It is easy to get 'silly data' past the present simple checks implemented in the program.

The next page displayed is the Main Menu Page (page 12). This is the focal point of the program. All control emanates from this menu. This is also the point of NO RETURN as far as the file selection (choosing the ADG and SMD files) is concerned.

If you find that you have made a wrong choice of the SMD or ADG file choose the 'Quit without saving data' option from the main menu. Leave the program, check the files and re-run the ADG program. It is possible to create more than one ADG file if you have used the ADG program several times with a particular SMD file. Therefore the above mentioned scenario (predicament) is not an unlikely one. Avoid creating more than one ADG file, for a particular analysis.

*6.2.7 Main Menu*

The data input (that is needed to generate the input data to the MP) is divided into 4 sections. These appear as the first 4 options in the main menu and are listed in Figure 6.3.

1) Title section (the previous 3 pages you have already seen. These three pages can be re-accessed by choosing this option).

2) Material properties

3) In situ stress specification (includes in situ fixities, loading)

4) Analysis block (control parameters, displacement pore pressure fixities, loading, output options, time steps etc).

*6.2.8 Saving Data and Quitting the Program*

This is also the point of exit from the program. You need to generate the MDD file and save the data input during this session

before quitting. There is no auto-saving mechanism implemented in this program. If you do not save the data and quit the program you lose all the information you have input during that particular session of ADG.

When you choose the 'save data and quit' option you don't have to quit the program after saving. After the program has saved the data from the current session it will give you and opportunity to return to the main menu. Therefore it is quite acceptable to save the data at intervals to safeguard against program crashes.

When you save the data for the first time the program will ask you to specify a file name for saving the data. It is better to choose the same as specified for the SMP program. Since all the files are identified by having the same name, specifying a different file name would lead to problems.

Any subsequent saving will automatically write to the file you had specified and the program will not prompt you for the filename again. (This means once you have chosen a file name for saving data you cannot change it during that session of ADG program). This is not considered to be very critical since you can rename the ADG file using the REN option available in DOS after exiting the ADG program.

The only restriction is that all the files should have the extension ADG. Otherwise you cannot access that file from within the ADG program.

More on main menu .....

All these 4 sections require data input from the user to generate a complete MPD input data file. There is no particular restriction on the sequence in which the user decides to input the data for each of the above section. There is no need for in situ stresses to be specified before the information for the 'analysis block' is input.

However it is desirable to follow the 'top down' approach. For example if the material properties section is chosen first then the program will know whether any Cam-clay models have been used for modelling any of the zones in the mesh and will display a column for inputting the size of the Cam clay yield locus in the 'in situ

stress specification' page when specifying the in situ stresses at reference points later. The default option for all the zones is elastic model hence a 'pc' value column will not be automatically displayed if you have not chosen to specify the material properties. However once you have done that the in situ stress input page will be altered accordingly.

This is just to illustrate the advantages of inputting the data for the very first time in a top-down sequence. Of course if you are editing a previously input data it is not necessary to follow this approach.

Main menu again .....

The 4 sections mentioned above are further divided into various levels depending on the necessity, with each level having its own menus as listed below :

(1) Title Section

Consists of 3 pages of 'field' data input and no menus.

(2) Material properties

Menu to choose each of the material zone. The material properties input are then divided into 2 further pages. The first page offering the material 'type' selection which is then followed by the properties input page.

(3) In situ stress specification

Menu offers the choice of specifying the in situ stresses and the in situ displacement fixities/loading using the icons. The first option leads to a further menu which allows for the following 3 choices :

a) Zero in situ stresses (elastic analysis) ++++
b) Linear variation of in situ-stresses (elastic analysis)
c) Specifying the in situ stresses at a set of reference points. (This is the most general option and the only choice if you are using the Cam clay models).

*6.2.9 Zero in situ stress option*

++++ This option is to be phased out. Only useful for the most trivial of analyses. Since most soil mechanics problems does require the specification of in situ stresses there is no need for this option. Apart from causing confusion because the displacement fixities have to be specified in the first increment block rather than at in situ stage this option is not recommended. Of course the program can be modified to make it consistent (ie allowing the user to specify the displacement fixities at in situ stage even when the zero in situ stress option is chosen).

Choosing the first option (zero in situ stresses) puts up a page (page ref no. 18) informing the user that the in situ stresses are specified as zero allowing the user either to confirm it (thereby returning to the main menu) or changing his/her mind (by pressing the 'escape' key in which case returning to the above menu).

*6.2.10 In situ stresses at Reference Points*

Choosing the option c) to specify the in situ stresses at reference points gives a page where the user has to specify the number of reference points (page ref no. 20). This allows the program to display a field with the appropriate number of rows. One for each reference point. The ADG program only allows for a maximum of 10 reference points. This is again a ADG Program restriction and not a CRISP MP restriction.

Halfway through the input of the stresses at reference points if you decide to increase or decrease the number of reference points this can be done by pressing RETURN key and finding your way back when the next menu is offered.

When the RETURN key is pressed the program will display a menu with the following options (page 22).

1) Plot in situ stress profile.
2) edit in situ stresses specified at reference points.
3) Return to in situ stress menu (page 17).

Choose the last option which should take you to page 17. From the menu displayed in that page choose option 3 and this will present you with the option to change the number of reference points.

If the user is increasing the number of previously specified reference points no data typed in by the user during the current session will be lost. However if he/she decides to reduce the number of reference points the relevant entries for the deleted reference points are erased. It is not possible to change the number of reference points directly from the page on which you are inputting the in situ stresses.

*6.2.11 Linear variation of in situ stresses*

As mentioned before this option is only available for the elastic or elastic perfectly plastic models (models 1, 2 and 5). This is because it does not allow you to specify the size of the initial yield locus even if Cam clay models are used. This allows for the following distributions of in situ stresses (see Figure 6.4).

If you have chosen the 'linear variation' option to specify the in situ stresses in the presence of elastic perfectly plastic models ensure that the initial stresses do not violate the yield criterion chosen for that zone (this depends also on the material properties). See section 6.4 for more details.

*6.2.12 Critical state models*

If you are using the critical state models then you have no choice. You have to use the option which specifies the in situ stresses at reference points (option c). See also section 6.4. You need a minimum of 2 reference points. The reference points must be numbered from top to bottom. You have to input the y co-ordinates of the reference points as well.

The program remembers the option (out of the 3 available) you had input last, and accepts that as your choice. This is denoted by

the character 'I' which is displayed next to that menu option.

*6.2.13 In situ stresses at reference points*

Once you have entered the in situ stresses press the RETURN key and this gives you a menu with the following choices (page 22):

(i) a plot of the in situ stresses you have specified at reference points.

(ii) edit in situ stresses at reference points.

(iii) return to the in situ stress specification menu.

Option (i) is useful in checking the in situ stresses you have typed in.

*6.2.14 In situ displacement fixities and in situ loading*

Now you have to choose this option and specify the displacement fixities and also any in situ loading. At this stage you can remove any elements you are later going to use to simulate 'construction'. The 'icon' page and how to use the 'icons' is explained in the next section (the section under the heading 'specification of displacement fixities and loading using icons').

*6.2.15 Analysis Section (Increment block control parameters)*

Figure 6.5 shows the 3 levels of menu when the analysis section is chosen for input. Level 1 gives the user the choice of inputting data for each of the increment block in the analysis. The number of increment blocks displayed is the number requested by the user in the title section. A maximum of 10 increment blocks can be specified by the user. This is a ADG program limit and not a CRISP MP limit. CRISP MP does not have any limit on the number of increment blocks that can be used in any analysis.

The last option in this menu is a return to main menu.

Once an increment block is chosen from the level 1 menu, this leads to level 2 menu with the following choices :

a) Control parameters for increment block
b) Specifying fixities/loading using icons
c) Return to level 1 menu

Choosing the first option leads to a 'field' input page where the user has to specify the following control parameters (page 27):

1) Starting increment number of the increment block (INCA)
2) Finishing increment number of the increment block (INCB)
3) Increment in gravity level
4) Time step for the increment block
(consolidation analysis only)

After the numbers have been input pressing the RETURN key leads to the level 3 menu with the following options (page 28):

a) Output options (for the printed output)
b) Increment (load) factors
c) Time steps (only for consolidation analysis)
d) Return to level 2 menu

Choose each of this option in turn and from the menu provided which allows one to specify either separate values for each of the increment or a constant value for all the increments.

For example for the output options one could specify a standard output option for all the increment or specify a separate output option for each of the increment.

Similarly for the Increment (load) factors one could choose this to be the same for all the increments in the block or choose separate factors for each of the increment. At present there isn't a check to make sure that the individual load factors specified when summed is equal to unity.

Once all the parameters have been input the user can return to the main menu by choosing the last option in the level 3, 2 and 1 menus.

*6.2.16 Specifying fixities/loads using icons for an increment block*

Choosing the second option in level 2 menu will lead to the icon page. Wait till the cursor appears before pressing any of the

keys. See the notes given in the next section for how to input data.

## 6.3 Specification of displacement fixities and loadings using icons

When the choice is made (from the appropriate menu) to specify the displacement fixities and loadings the program draws the icon page (see Figure 6.6). If this is done for the first time you will notice the slow response as each icon and its variants are drawn. Wait till the whole page is drawn at which point the cursor will be displayed. It will erase some of the icons which are not required presently. At the top of the page you will find one of the following headers which indicates whether the data input is for the in situ stage or for one of the increment blocks.

a) In situ stage b) Increment block number i

The largest region below this header is known as the 'Mesh region'. This is where the super-mesh is drawn. On the right hand side you will notice that several icons are displayed which forms the 'icon menu'. There are three regions at the bottom of the screen. The left hand region with the label 'icon window' is used to display any icons attached to 'short' or 'inner' super-element sides. See below for an explanation of short and inner sides.

The middle bottom area is denoted as the 'message region'. The program whenever appropriate will display error messages, sign conventions etc in this region. The right hand region is denoted as the magnitude/value region, where data can be entered by the user when prompted by the program.

The mesh region, the menu region and all the icons have a blue background.

### *6.3.1 Icon menu*

Figure 6.7 lists all the icons. At the beginning none of the 'icons' have been chosen, hence all display the blue background. An icon is chosen by moving the mouse (hence the cursor) to the icon to be chosen and 'clicking' one of the mouse 'buttons'. It does not matter which mouse button is pressed. Irrespective of whether the mouse has 2 or 3 buttons all have the same effect.

The icon you have chosen will be displayed with a 'brown' background. This indicates that the particular icon is the 'current choice' or is 'active'. You will also notice that in the region immediately beneath the icon menu a label is displayed. It is a description of the icon you have chosen. As you select different icons this label will also keep changing.

Move the cursor to another icon and 'click' the mouse. You will notice that the newly chosen icon is displayed with a 'brown' background (this new icon now becomes 'active') and at the same time the previously selected icon returns to its original 'blue' background (it has become de-selected). Therefore only one icon can be 'active' at any given time.

*6.3.2 Long sides and short or inner sides*

The PRE-ADG program which is run immediately after the Mesh generation program (SMP) works out the icon locations for all the 'outer' sides of the super-mesh. If the outer super-element sides are long enough to display a maximum of 2 icons then they are classified as 'long' and any icons assigned to these sides will be displayed along side these lines. All other sides including 'inner' element sides are classified as 'short' sides. Any icons assigned to these sides are not displayed alongside the element-sides in question but are displayed in the icon window as soon as they are assigned.

These icons assigned to 'short' and 'inner' sides are also known as 'hidden' icons. Because these are not on display alongside the element-sides as done in the case of long sides. Nevertheless these remain assigned and this can be confirmed by the character 'H' displayed in brown near the centre-line of these sides.

*6.3.3 Excavation and construction icons*

The icons which appear in the first four rows (icons 1 to 11) can be assigned to any super-element side of the mesh. The next two icons (construction and excavation icons) can be assigned to any super-element. There are certain restrictions. Some super-elements may be displayed with a different background colour (brown). This

means that these super-elements are not present at the beginning of the current stage. These elements can be 'added' (using the construction icon) however for obvious reason cannot be 'deleted'. Similarly any super-elements with a blue background cannot be 'added'. However these elements can be removed using the 'excavation' icon. When you have assigned either the 'construction' or 'excavation' icon to an element you will notice that it is displayed on that element. However if the element is too small to fit the icon either the character 'E' or 'C' representing 'Excavation' and 'Construction' respectively will be displayed at the centre of the element in question.

These icons are assigned by clicking the mouse button when the cursor is on the super-element to which you want to assign that icon.

The next three icons in the icon menu are : 1) display 2) edit magnitude 3) erase icons respectively.

*6.3.4 Erasing icons*

The erase icon allows you to remove any previously assigned icons if you have changed your mind. The erase icon works directly on 'assigned' icons. Choose the 'erase' icon first by clicking on it and then move the cursor to the mesh and clicking the mouse on any icon you want to erase. You will notice the icon disappearing.

The last three are 1) open icon window 2) close icon window 3) wipe icons respectively.

*6.3.5 Wipe icon*

A word of warning about the 'Wipe' icon. Its task is to erase all previously input data on the icon page. It is not reversible and is used as follows. Choose the 'wipe' icon first by clicking the mouse on that icon so that it is displayed in 'brown'. Now move the cursor into the 'mesh' region and click the mouse (The cursor need not be on the super-mesh). The program will display a message asking for confirmation in the 'Message region'. If you decide to go ahead simply type 'Y' and press the RETURN key. Hey presto! all the icons assigned before are erased and you can make a 'fresh' start.

*6.3.6 Assigning icons to the mesh*

Once you have chosen any of the icons in the first four rows you can assign these to element-sides by moving the cursor to the relevant element-side and 'clicking' the mouse button. The cursor has to be on or very near the line. In that case the selected icon will be displayed by the side of the element side if it is a long line. Otherwise the icon will be displayed in one of the two locations within the icon window. If you had 'missed' the line, which is not unusual because of the leeway allowed on the line is very small, a message 'cursor not on any line' will be displayed in the message region. Try again.

> Note : Difficulties have been reported in trying to assign icons to small sized super-element sides. It is likely that the adjacent larger super-element sides are 'hit' by mistake.
>
> Perseverance is the name of the game. Keep moving the cursor around slowly while clicking the mouse to score a 'hit'. Even if the larger adjacent super-element sides are assigned icons unintentionally donot worry. It is easy to erase any icons accidentally assigned to nearby super-element sides.

If the icon you have chosen is associated with 'values' (icons 4 to 11) then a window will open up in the 'magnitude region'. The cursor will momentarily disappear from sight. Enter the relevant numbers in the appropriate field and press the RETURN key and the window will be removed from the 'magnitude region' and the cursor will re-appear. You can continue with further icon assignment. If you had made any mistakes in entering the values an 'E' will be displayed next to the field with 'error' and you have to correct any mistakes. You can move around using the 4 arrow keys between fields displayed.

*6.3.7 Assigning icons which have associated values*

Icons 4 to 11 have nodal values associated with the icons. These values are specified for the parameter in question

(incremental displacement, incremental or absolute value of excess pore pressure, normal and shear stresses) at the nodes at either end of the super-element side. The super-nodes numbers are not displayed with the super-mesh in the icon page. So use the super-mesh you had sketched for use with the SMP program.

When one of the above icon is assigned to any super-element side a window will be opened in the magnitude region and 2 or 4 fields of input will be displayed. Use the 4 arrow keys to move around the fields and input the required data. When you have done so press the RETURN key to carry on with further icon assignment.

*6.3.8 Assigning icons to more than one super-element side*

If you are going to assign the same icon to many different sides, you can do so by first selecting the icon by clicking the mouse button when the cursor is on that icon. Then move to the respective sides and click the mouse button and wait till the icon is displayed either adjacent to the line or in the icon window. Then move to the next side and repeat the procedure. However if the icon you are assigning requires input of values at the nodes at (either end) you need to input these values in the fields displayed in the magnitude region before you can assign the icon to a new side.

*6.3.9 Edit Magnitude icon*

This icon can be used for changing the values at nodes associated with a previously specified icon (only applicable to icons 4 to 11). This has no effect on icons 1, 2 and 3. Once the 'edit magnitude' icon has been selected in the usual manner (ie by clicking the mouse button when the cursor is on the icon) choose the icon which you want to change the specified value of the variable (displacement, pore pressure or shear/normal stress) and click the mouse button.

If the icon is one in the group of 4 to 11 then a window in the magnitude region will be opened and the values at the nodes will be displayed for you to change them. Depending on the particular icon 2 or 4 values will be displayed. This icon can be used not only to change the value but also to check the value previously specified.

Unlike some other icons, the 'edit magnitude' icon works both on previously assigned icons which are displayed alongside the super-mesh as well as any 'lines' which have been assigned 'hidden' icons. If the line chosen has been assigned any 'hidden' icons which will be apparent by the character 'H' displayed near the centre of the line, the icons assigned to the line will be displayed in the icon window.

The user then chooses the icon within the window for which he/she wants to change the associated values, if there are more than one icon assigned to this side. Even if the side contains only one hidden icon the values are not displayed automatically. You need to click on the icon in the window again for it to display the values associated with it.

To summarise first choose the 'edit magnitude' icon and then click on the line and then click on the icon which is displayed in the icon window.

Once the values have been changed or if the user had simply wanted to check the currently specified values, pressing the RETURN key will remove the values displayed in the magnitude region.

However the icon window remains open. This is in case where there are more than one icon associated with the line and the user may wish to inspect or change the values associated with the other icon. The 'icon window' can then be closed by choosing the 'close window' icon and then clicking the mouse button when the cursor is within the 'mesh' region. Note that the cursor need not be on the icon window or even on the super-mesh.

*6.3.10 Display icon*

This icon provides an alternate option to the 'edit magnitude' icon. This displays the values associated with any icon in the range 4 to 11. Unlike the 'edit magnitude' icon it does not operate on lines. It can only operate on 'visible' icons ie icons associated with 'long' lines or on any icons currently displayed in the icon window. Once the values are displayed in the magnitude region the values can be modified as with the 'edit magnitude' icon. Therefore the 'display' icon in a way serves the same purpose as the 'edit

magnitude' icon even though it operates in a slightly different manner.

The only difference is that the 'display' icon cannot operate directly on 'hidden' icons. Then one uses 'open window' icon to make the currently 'unseen' or 'hidden' icon visible.

*6.3.11 Open Window icon*

If the user wishes to find out what icons have been assigned to any 'inner' or 'short' side then he/she uses the 'open (icon) window' icon. The user first selects the 'open window' icon and then clicks the cursor on any side with a 'H' displayed adjacent to it. The icons assigned to this side are then displayed in the icon window. Then if the user wants to look at the values associated with any of the icons currently displayed within the window, choose either the 'edit magnitude' or 'display' icon and then click on the icon in question. This icon does not operate on long sides.

*6.3.12 Close Window icon*

This icon only serves to close the 'icon window' if that window is currently open. As mentioned earlier when the icon window is opened it is not closed automatically hence the need for a separate 'close window' icon.

The 'close window' icon is used to close the window by clicking the cursor in the mesh region and NOT in the icon window region.

*6.3.13 Quitting the icon page*

Use the function key 'F3' to leave the icon page once you have specified the displacement/pore pressure fixities, loading and element addition and removal.

It should be noted that as with the ADG program, the icon page deals with the super-mesh. The super-node numbers are used to specify values associated with displacements and pore pressures and any pressure/shear loading. Super-elements are removed and added.

You can only remove or add 'whole' super-elements. You cannot add or delete 'part' of a super-element. If you want to add or delete zones (or collection) of F.E. mesh elements then you need to

represent them as a separate super-element when using the SMP program.

No direct reference is made to the finite element mesh, or the elements and nodes in the f.e.mesh. The reference to the finite element is implied and inferred. The ADG program translates all the information you have specified for the super-mesh into finite element data for the CRISP MP program.

*6.3.14 Combining icons*

Because of the fact that only 2 icons at most can be assigned to any super-element side, the program wherever possible will combine certain icons. If you assign icon 2 to a side which has already been assigned icon 1, the program will combine icons 1 and 2 and will display it as icon 3. You could have used icon 3 in the first place. Similarly icons 4 and 5 are combined to give 6, 7 and 8 to give 9.

The program will carry out these combining operations automatically without any prompting from the user. However if the user has assigned 2 icons (which cannot be combined into a single icon).to a side and attempts to assign a 3rd icon the program will work out whether the 3rd icon can be combined with any one of the existing icons into a single icon. If that is the case it will seek confirmation before going ahead. It is vary rarely you need to assign 3 icons to a single side.

*6.3.15 Checks on errors*

The present version of the ADG program does not include any overall checks to see that the user specified data are consistent between the different increment blocks and the in situ stage. It is hoped that these checks will be implemented in future updates.

For example, at present you could restrain a super-element side in both x and y directions at in situ stage using the icons, in the icon page. Then in the first increment block you may apply a pressure loading to the very same side. This is not correct as applying a pressure load has no effect on a restrained side. This type of errors will not be spotted by the ADG program.

All the increment blocks and the in situ stage are treated as separate independent entities. The input data for the different increment blocks and in situ stage are are kept separate without any interaction. Therefore due care is needed when using the ADG program for the present, if one wishes to avoid any input data errors. It is a good idea to carry out a quick 'visual' check on all the increment blocks and in situ stage before generating the CRISP MPD file.

*6.3.16 Working with hidden icons*

Assigning :

This is straigtforward. Icons 1 to 11 can be assigned to "short" or "inner" sides in the same manner as "long" sides. You choose the icon first and then moving the cursor to the line in question and clicking the mouse button. If the side in question is "short" or is a "inner" side then the "icon window" (at the bottom left hand corner of the screen) will open and the icon displayed there. If the icon is associated with values (icon 4 to 11) the magnitude window will open up and 2 or 4 fields will be displayed (for the user to enter the appropriate nodal values).

It should be remembered that the "icon window" is only associated with any "short" super-element sides which lie along the boundary of the mesh and all the super-element sides ("inner" sides) which lie within the super-mesh. Super-elements sides which have icons displayed along side it are not associated with the "icon window".

Editing (changing magnitude and erasing) :

This applies to hidden icons already assigned. "short" or "inner" super-element sides with icons already assigned are identified by a 'H' displayed next to the line, in red.

To find out what icons have been assigned :

Select the "open window" icon by clicking on it. Move the cursor to the line with "hidden" icons and click the mouse button. This will display the 1 or 2 icons assigned to that line. If any of the icons assigned are associated with nodal values (icons 4 to 11) then if you want to find out what these values are choose either the

"edit magnitude" icon or "display" icon (icons 15 and 14 respectively) by clicking on it. Then move the cursor to the icon in question (which is displayed within the icon window) and click the mouse button. The nodal values will be displayed in the magnitude window for you to either (a) check the values or (b) change it.

Therefore this task involves four moves and are summarised below

(1) move the cursor to the icon menu region and choose the "open window" icon by clicking the mouse button.

(2) move the cursor to the super-mesh region and choose the super-element side by clicking on it. This displays the icons associated with that line in the icon window.

(3) move the cursor to the icon menu region and choose either the "display" icon (icon number 14) or the "edit magnitude" icon (icon number 15).

(4) move the cursor to the "icon window" and choose the icon by clicking on it.

The same task can be accomplished using three moves :

(1) move the cursor to the icon menu region and choose the "edit magnitude" icon by clicking the mouse button. Even if you only want to check on the current nodal values and not change it, you can still use it.

(2) move the cursor to the super-mesh region and choose the super-element side by clicking on it. This displays the icons associated with that line in the icon window.

(3) move the cursor to the "icon window" and choose the icon by clicking on it.

<u>erasing hidden icons</u> :

(1) move the cursor to the icon menu region and choose the "open window" icon by clicking the mouse button.

(2) move the cursor to the super-mesh region and choose the super-element side by clicking on it. This displays the icons associated with that line in the icon window.

(3) move the cursor to the icon menu region and choose the "erase icon (icon number 16).

(4) move the cursor to the "icon window" and choose the icon by clicking on it. This icon should be erased now.

To complete the above task and close the icon window choose the "close window" icon (icon number 18) from the menu and then move the cursor to the super-mesh region (the cursor need not be on the super-mesh) and then click the mouse button. This should clear the icon window region.

<u>There are two modes of inputting the nodal values</u> :
(in the fields in the magnitude region)

(1) Overstrike mode. (2) insert mode. Press the 'Ins' key once to toggle from one mode to the other. At present there is no way of telling apart the two modes. When you start entering the values from the response you get you should be able to determine the current mode.

### 6.4 In situ stress specification in ADG Program

The linear variation of in situ stress is a simple option which is not directly related to the CRISP input as given in the CRISP manual vol.1 (section 3). This is an alternative way of specifying the in situ stresses using option INSIT = 1 in record F (MP input data).

Instead of specifying the in situ stresses at user specified reference points the program calculates it from user specified data.

This option can only be used for analysis which uses the elastic or elastic perfectly plastic models. One cannot use this for Cam-clay analysis. This is because this option does not allow for the input of initial size of yield locus variation through the mesh. This information is necessary to carry out a Cam clay analysis.

The in situ stress variation options allowed for are shown in the Fig. 6.8. The following information is required from the user.

1. The in situ gravity level (GRAVI) **
2. dry unit weight of soil (DRYD)
3. bulk unit weight of soil (BULKD) **
4. unit weight of water (UNITW) **
5. depth of ground water table (DEPTH) **
6. coefficient of earth pressure at rest (KO) **
7. total vertical stress at the top (SURCH)

** essential. User should specify these values. All other parameters are optional.

1. a) If the vertical stress increases with depth and if you are analysing a field situation then this should be set to 1.
   b) If the analysis is that of a laboratory test set this to zero (default option).
   c) If the analysis is that of a centrifuge test and the starting point of the analysis is where sample has reached equilibrium at a given centrifugal acceleration then specify the g level.

2. Only needs to specified where the ground water level is below ground level and where you would like to specify a different unit weight for the part of the soil above the ground level. (default value is zero). This value is ignored if the ground water level is at ground level.

3. Bulk unit weight of the soil. This has to be specified.

4. Unit weight of water. Is used in calculating the initial pore pressure distribution. If set to 0 then the initial pore pressures will be set to zero.

5. Depth of ground water level (GWL). Default value is 0 ie GWL is assumed to be at the ground surface.

6. Coefficient of earth pressure. This has to be specified. The effective horizontal stress at any point is calculated by multiplying the effective vertical stress at that point by this parameter. If set to 0 then the effective horizontal stress is set to zero throughout the mesh.

7. Total vertical stress at the top. This allows one to take into account any surcharge acting at the beginning of the analysis at the top of the mesh. Default is 0 meaning no surcharge. Illustrated by Fig. 6.4(d).

The situations represented by each figure is listed below :

a) Ground water level is at ground surface.

GRAVI = 1. (field situation)
DRYD - not required
BULKD - specified
UNITW - specified
DEPTH = 0.
KO - specified
SURCH = 0 (not required)

b) Ground water level is below ground surface.

GRAVI = 1. (field situation)
DRYD - specified (for soil above ground water table)
BULKD - specified (for soil below ground water table)
UNITW - specified
DEPTH - specified (> 0)
KO - specified
SURCH = 0 (not required)

c) Submerged. Say the ground surface is h units under water.

GRAVI = 1. (field situation)
DRYD - not required
BULKD - specified
UNITW - specified
DEPTH = specified ( < 0 ) = (-)h
KO - specified
SURCH = 0 (not required)

d) Surcharge acting on the surface. The ground water level can be at ground level or below it. The figure 1(d) and the data below illustrates the case where the GWL is at ground surface.

GRAVI = 1. (field situation)
DRYD - not required
BULKD - specified
UNITW - specified
DEPTH = 0.
KO - specified
SURCH - specified ( > 0 )

For the case where the GWT is below ground surface, the following parameters are changed, the rest of the parameters are the same.

DRYD - specified
DEPTH - specified ( > 0 )

## 6.5 ADG Program - explanatory notes

Each analysis has its own SMD and ADG files. The files which have the same name are identified as belonging to the same analysis. In addition a code is used which is the same for all the files belonging to a particular analysis. The ADG program checks that the code is the same for both the SMD and ADG files selected by the user.

Pages 1 to 8 are gone through only once. Once Page 9 has been reached it is not possible access any of the previous 8 pages. It is also not possible change any of the files (SMD, ADG selected). If for some reason you want to use some other file (SMD or ADG) then quit the program from Main menu (Page 12) and re-run the ADG program again.

Future version may allow you to access some other file (if you find that you have selected a wrong file (SMD, ADG) without quitting the ADG program.

Here follows a page by page description of the interactive use of the ADG program. The page reference number for each page appears on the top hand corner of the screen. This section is meant to provide a 'help' facility and useful as a quick reference (note that the page reference numbers are not displayed in the icon page or in the plot of in situ stresses at reference points).

### *6.5.1 Page 1 : SMD directory name*

Displays the current directory name in a field. Press the RETURN key if the SMD file you are working with is in the current directory, which is the recommended way of using the ADG program. Accessing files in other directories may slow down the response of the program, hence not recommended. If for some reason you want to quit the program, simply press the F3 key. This should return you to the DOS prompt, without saving any of the information you have typed in.

If you are stuck on this page, try pressing the Esc (Escape) key once. If there is no response then try pressing the F3 key.

*6.5.2 Page 2 : SMD file selection*

Displays the files with the extension SMD which are in the current or user selected directory. If there are more than 10 SMD files then these are displayed 10 at a time. If the file name you are looking for is not found then use the 'Page up' key to scroll the file-names. Pressing the 'Page Up' key displays the next 10 SMD files, if these exist. 'Page down' key can be used to scroll backward.

Once the file you are looking for appears in the screen use the up or down arrow keys to highlight that file name and then then press the RETURN key. This means that you have selected the highlighted file. At this point if you find that you have made the wrong choice proceed to the main menu (page 12) and choose the 'quit without saving' option. Quit the program and re-run the ADG program again.

The SMD file (Super mesh data) contains information about the super- mesh and other geometric details. It does not contain any information pertinent to the main analysis. This information is stored in the ADG file (if you have used the ADG program before and saved the data).

*6.5.3 Page 3 : ADG file - menu*

A menu with the following choices :

(a) Read in a existing ADG file
(b) create a new ADG file.

By default the option (a) will be highlighted. If you have an existing ADG file for the following analysis you can choose it to be read in. If you have not used the ADG program before for the current analysis then choose option (b) using the down arrow key.

It should be noted you cannot read in a ADG file from an entirely different analysis and use it with a SMD file from a different analysis. For example you may have run an embankment analysis in the past. (Say) you have used the name EMBANK1 for this. This would have created EMBANK1.SMD and EMBANK1.ADG if you had used the SMP, PRE-ADG and ADG programs.

Now if you are embarking on a brand new analysis of (say) a retaining wall problem and let us assume that the name you have chosen is RETAIN1. If you have run the SMP and PRE-ADG programs then this would have created a file called RETAIN1.SMD. Now you cannot use the (new) RETAIN1.SMD file with the (old EMBANK1.ADG as a labour saving means. These files are incompatible and a code is used for each different analysis to ensure that this is not permitted.

If you have chosen option (b) you will next see page 6.

*6.5.4 Page 4 : ADG directory name*

This and page 5 are displayed in sequence if you have chosen option (a) in the previous page (page 3).

Displays the current directory name in a field. Press the RETURN key if the ADG file you are working with is in the current directory, which is the recommended way of using the ADG program. Accessing files in other directories may slow down the response of the program, hence not recommended. If for some reason you want to quit the program, simply press the F3 key. This should return you to the DOS prompt.

*6.5.5 Page 5 : ADG file selection*

Displays the files with the extension ADG which are in the current or user selected directory. If there are more than 10 ADG files then these are displayed 10 at a time. If the file name you are looking for is not found then use the 'Page up' key to scroll the file-names. 'Page down' key can be used to scroll backward.

Select the ADG file in the same manner as the SMD file above (page 2). For the sake of consistency the name of the ADG and SMD files should be the same. These need not be the same but helpful in avoiding confusion.

*6.5.6 Page 6 : ADG PROGRAM LIMITS*

The programs prints the limits of the ADG program. These are not necessarily the limits of the MP program.

```
Max number of increment blocks                    =   10   (no limit)
Max number of material zones                      =   10   (25)
Max number of increments in a increment block     =   50   (50)
```

The limits of the main program (MP) is displayed within brackets.

Press the RETURN key to continue.

This would display the following message :

*** Reading data from the SMD file ***

Once the information in the SMD file has been read the next page will be displayed.

### *6.5.7 Page 7 : GEOMETRY DETAILS*

At this stage the geometric details of the super-mesh and the f.e. mesh are listed. Also listed is the breakdown of the different types of elements present in the mesh. These are just to refresh your memory. Press the RETURN key to continue.

The following message will appear at the bottom of the page :

*** Carrying out calculations ***

If you are starting a new analysis (ie you have not read in a ADG file - option (b) in Page 3) then Page 9 will appear next (and page 8 will be skipped).

### *6.5.8 Page 8 : ANALYSIS DETAILS*

(only displayed if you have read in a ADG file)

This displays previously input MP data. For example the type of analysis the different number of material zones and the number of increment blocks and the starting and finishing increment numbers. This again is a reminder. None of the information on display can be changed in this page. Changes can be made later in the following pages.

Press the RETURN key when you have read the information in this page.

*6.5.9 Page 9 : TITLE SECTION - Records A and B*

This is first of 3 pages and the beginning of the ADG program proper. These are referred to as pages 9, 10 and 11 respectively. If you consult the CRISP manual (Vol.1) section 3 you will find that these 3 pages deal with records A, B and C1 and C2.

Record A : Title for analysis. Defaults to 'CRISP ANALYSIS'. Typeover, a meaningful title. [TITLE]

Record B : Link number (The program generates a link number automatically and you cannot change it). [LINK]

Type of analysis : (1) drained or undrained analysis
(2) consolidation analysis

The appropriate message which is printed is based on what type of elements are present in the mesh. This is not directly linked to any parameter in the MP input data.

Once you have typed over an appropriate TITLE press the RETURN key to continue.

*6.5.10 Page 10 : ANALYSIS CONTROL PARAMETERS - RECORD C1*

This page deals with the record C1.

All the parameters which form record C1 appears in the correct sequence except for IPRIM and ICOR. These parameters will be set by the program according to data input elsewhere in the ADG program, and hence not displayed here.

NPLAX - Analysis identifier (0 - Plane strain, 1 - Axisymmetric).
NMAT - Number of material zones in the mesh.
NOIB - Total number of increment blocks in analysis.
INC1 - Starting increment number for current analysis.
INC2 - Finishing increment number for current analysis.
IUPD - option to update nodal co-ordinates by the amount of displacements that have occurred in that increment.
ISR - stop/restart option and used for post-processing. (should always be set to 2).

### *6.5.11 Page 11 : PRINTED OUTPUT CONTROL PARAMETERS - Record C2*

This page deals with record C2.

INSOP - controls in situ stress output
(0 - no output, 1 - centroid only, 2 - all integration points)

IBC - controls the boundary conditions output from each increment (0 - not printed, 1 - printed for all increments in the analysis)

IRAC - controls the printing of reactions from each increment (0 - not printed, 1 - printed for all increments in the analysis)

NVOS - starting vertex node number for printing displacements and pore pressures (in consolidation analysis)

NVOF - Finishing vertex node number for printing displacements and pore pressures (in consolidation analysis)

NMOS - starting mid-side node number for printing displacements and pore pressures (in consolidation analysis)

NMOF - Finishing mid-side node number for printing displacements and pore pressures (in consolidation analysis)

NELOS - starting element number for printing general stresses, Cam clay parameters and strains .

NELOF - Finishing element number for printing general stresses, Cam clay parameters and strains

This record controls the information written to the Main program output file (MPO). This basically deals with what type of information is written to this file and also whether a detailed or slim line output is generated. These parameters do not impose any restriction on the type of information that can be post-processed (PP, CL and SP programs). That is independent of these control parameters.

These parameters only control the output generally destined for the printer (in mainframe computers, here written to the MPO file). The para -meters mentioned here control the printed output in conjunction with the IOUT parameters in Pages 30 and 31 (records I and K2 of MP input data).

*6.5.12 Page 12 : Main Menu*

The input is divided into 4 major sections :

1) Title section
2) material properties
3) in situ stress specification
4) analysis section

You have already input the data required for the title section by means of the 3 previous pages (pages 9 to 11). As a confirmation the second option 'material properties' is shown highlighted indicating that this is the next section for which data has to be input. However by choosing the option 1 from the menu the previous 3 pages can be re-accessed.

*6.5.13 Page 13 : MATERIAL ZONES menu*

This page is displayed when option 2 is chosen from the menu in the previous page.

The number of options on display is the same number as the material zones you have specified in the finite element mesh.

This section deals with record D of the input data. Record C3 is not relevant for the running of CRISP on the PC-386 environment and hence the program automatically generates a dummy record.

You need to choose each material zone in turn and identify the material type as one of the six given below (in the next page) :

1 - Linear elastic (isotropic/anisotropic)
2 - Non-homogenous elastic
3 - Modified Cam clay
4 - Cam clay
5 - Elastic perfectly plastic
6 - Hvorslev surface model

The order in which each material zone is chosen (from the menu) for inputting material properties is immaterial. This can be accessed any number of times.

Once the material properties for all the zones have been input, choose the option to return to the main menu.

*6.5.14 Page 14 : Material type number - Record D*

By default a type number of 1 is assigned by the program to all the materials at the beginning (ie when you use the ADG program for the first time). Overtype whatever is appropriate and then press the RETURN key. This is followed by page 15.

*6.5.15 Page 15 : Material properties - Record D*

Depending on your choice a default set of material properties or the material properties you have previously entered will be displayed.

Refer to Pages 3.12 and 3.13 of volume 1 of CRISP brown manual for a check and Pages 4.33 to 4.44 for more explanation. Also I.1 to I.3 of the brwon manual.

Once you have entered the material properties press the RETURN key which should take you back to the material zones menu (page 13) for you to input the data for the next material zone. Once the material properties for all the zones have been input, choose the option to return to the main menu.

This completes the data input for record D.

From the main menu, next choose the option to input data for the in situ stresses section. The following page is then displayed.

Note : In setting up the default material properties (see the file CRISP90.MAT) it is assumed that the units the user is working in is as follows :

Force - Kilo Newtons (kN)
Length - meters (m)
Time - sec

pressure, stress - kN/m**2
bulk unit weight - kN/m**3

permeability - m/sec

This by far the most commonly used units. However if the user is working with some other units he/she should check all the default values and type over whatever value is appropriate in the units of their choice.

Alternatively if these are the units one is commonly working on.then edit the CRISP90.MAT file and replace it with the appropriate values.

*6.5.16 Page 16 - IN SITU STRESS SPECIFICATION MENU*

This page is displayed when the in situ stress specification option is chosen (option 3) from the main menu in page 12.

The menu consists of the following categories :

1) In situ stress specification
2) In situ displacement fixities and loading.

Category 1 relates to the data for records F (NNI), G1, H1 (GRAVI).
Category 2 relates to the data for records E, H1, H2, H3 and C1 (IPRIM).

Data has to be entered for both these categories.

*6.5.17 Page 17 IN SITU STRESS SPECIFICATION MENU - RECORD F*

If you have chosen the option 1 from previous page you will be presented with a menu with the following options :

1) Zero in situ stresses
2) Linear in situ stresses
3) In situ stresses specified at reference points
4) Return to in situ stress menu (page 16)

If you have used the ADG program previously to input data then 'I' will be displayed to the right of the option previously chosen.

You need to input data for only one of these options (not all 3).

Your choice of the option is restricted by the type of analysis you are carrying out. If the analysis is a Cam clay analysis then you have to choose option 3. A Cam clay analysis is any analysis

where at least one zone of the mesh is represented by critical state model (model number 3 or 4 or 6) properties.

If you are carrying out an elastic analysis and you are not interested in specifying the in situ stresses then choose option 1. However even if you are carrying out an elastic analysis, but interested in specifying the in situ stresses (ie if you are not simply interested in the changes in the stresses generated during the analysis but also in the total stresses at the end of the analysis) then you have to choose option 2 or 3.

Option 1 represents INSIT = 0 in record F.

Option 3 gives the most flexibility and represents INSIT = 1 option in record F. Option 2 is explained in more detail elsewhere (Chapters 3, 4 in CRISP brown manual volume 1).

The option represented by INSIT = 2 and in situ stress specification using records G2 and G3 is not available with the ADG program (see ADG program limitations). The only way to specify this option (INSIT = 2) is to edit the MPD file created by the ADG program and adding that information using an editing or word-processor program. Use the 'non-document' mode if using the wordstar. This is an ascii file.

*6.5.18 Page 18 - ZERO IN SITU STRESS OPTION - RECORD F*

If you have chosen the first option (Zero in situ stresses) from the previous page then this page is displayed. This option can only be used for elastic analysis (models 1, 2 and also model 5 under certain circumstances).

By pressing the RETURN key you can confirm that the zero in stress option has been chosen. This will return you to the main menu. This is because you don't have to enter any data for the category 2 in page 16. Pages 19 to 24 are skipped for this option.

However if your choice was a mistake or if you have changed your mind then press the 'Esc' key and you will be returned to the menu in the previous page.

permeability - m/sec

This by far the most commonly used units. However if the user is working with some other units he/she should check all the default values and type over whatever value is appropriate in the units of their choice.

Alternatively if these are the units one is commonly working on.then edit the CRISP90.MAT file and replace it with the appropriate values.

*6.5.16 Page 16 - IN SITU STRESS SPECIFICATION MENU*

This page is displayed when the in situ stress specification option is chosen (option 3) from the main menu in page 12.

The menu consists of the following categories :

1) In situ stress specification
2) In situ displacement fixities and loading.

Category 1 relates to the data for records F (NNI), G1, H1 (GRAVI).
Category 2 relates to the data for records E, H1, H2, H3 and C1 (IPRIM).

Data has to be entered for both these categories.

*6.5.17 Page 17 IN SITU STRESS SPECIFICATION MENU - RECORD F*

If you have chosen the option 1 from previous page you will be presented with a menu with the following options :

1) Zero in situ stresses
2) Linear in situ stresses
3) In situ stresses specified at reference points
4) Return to in situ stress menu (page 16)

If you have used the ADG program previously to input data then 'I' will be displayed to the right of the option previously chosen.

You need to input data for only one of these options (not all 3).

Your choice of the option is restricted by the type of analysis you are carrying out. If the analysis is a Cam clay analysis then you have to choose option 3. A Cam clay analysis is any analysis

where at least one zone of the mesh is represented by critical state model (model number 3 or 4 or 6) properties.

If you are carrying out an elastic analysis and you are not interested in specifying the in situ stresses then choose option 1. However even if you are carrying out an elastic analysis, but interested in specifying the in situ stresses (ie if you are not simply interested in the changes in the stresses generated during the analysis but also in the total stresses at the end of the analysis) then you have to choose option 2 or 3.

Option 1 represents INSIT = 0 in record F.

Option 3 gives the most flexibility and represents INSIT = 1 option in record F. Option 2 is explained in more detail elsewhere (Chapters 3, 4 in CRISP brown manual volume 1).

The option represented by INSIT = 2 and in situ stress specification using records G2 and G3 is not available with the ADG program (see ADG program limitations). The only way to specify this option (INSIT = 2) is to edit the MPD file created by the ADG program and adding that information using an editing or word-processor program. Use the 'non-document' mode if using the wordstar. This is an ascii file.

*6.5.18 Page 18 - ZERO IN SITU STRESS OPTION - RECORD F*

If you have chosen the first option (Zero in situ stresses) from the previous page then this page is displayed. This option can only be used for elastic analysis (models 1, 2 and also model 5 under certain circumstances).

By pressing the RETURN key you can confirm that the zero in stress option has been chosen. This will return you to the main menu. This is because you don't have to enter any data for the category 2 in page 16. Pages 19 to 24 are skipped for this option.

However if your choice was a mistake or if you have changed your mind then press the 'Esc' key and you will be returned to the menu in the previous page.

*6.5.19 Page 19 - LINEAR VARIATION OF IN SITU STRESSES*

Option 2 from the menu in page 17 gives this page. The data input in this page deals with the records F (INSIT, NNI) and H1 (GRAVI).

This represents INSIT = 1 in record F. The program uses 2 or 3 reference points (depending on your case) and calculates the stresses at these reference points from information you enter in this page. This is an indirect way of inputting the same information that would have been input using Page 20 (option 3 in page 17).

See elsewhere for detailed explanation for the different parameters for which data has to be entered here. ??? where ???

Press the RETURN key once you have entered all the necessary data. This would return you to the menu in page 16. Choose option 2 from the menu which would give page 24. Then you can proceed with entering data using mouse/icons.

*6.5.20 Page 20 - IN SITU STRESSES SPECIFIED AT REFERENCE POINTS*

(RECORD F/H1)

Option 3 in page 17 leads to this page.

NNI - Number of reference points (minimum of 2) - record F.
GRAVI - In situ gravity level (record H1).
Set to 1 for analysis of field situation.
Set to 0 for analysis of laboratory tests (eg. Triaxial)
Set to n in an analysis of a centrifuge test where the starting point of the analysis is where initial equilibrium has been achieved at a centrifugal acceleration of 'n' g.

This represents INSIT = 1 in record F.

As you may have already gathered this is the same option as specified using page 19. Pages 19 and 20/21 represent alternate ways of specifying input data for the same CRISP MP input option (records F and G1).

This specifies the variation of in situ stress throughout the mesh (excluding the part of the mesh which is not present at the beginning of the analysis - example an embankment in an analysis where the embankment is not present at the beginning of the analysis

but is constructed during the course of the analysis).

For each reference point the stresses and size of the initial yield locus if using Cam clay models (as required for record G1 in the CRISP manual) has to be specified in the next page.

Pressing the RETURN key will lead to the next page (Page 21).

*6.5.21 Page 21 - In situ stresses at reference points*

This page corresponds exactly to the MP input record G1. The number of rows corresponds to NNI in the previous page ie the number of in situ reference points. The first column represents the reference point numbers. The second column the 'y' co-ordinate of the reference points. This is followed by the effective stresses in x, y and z directions (horizontal, vertical and the out-of plane-horizontal stresses respectively), the shear stress in the xy plane, the in situ pore pressure and where applicable the size of the initial yield locus.

The reference points should be defined from top to bottom ie the point 1 is at the top (ground level). Even though this is not a CRISP MP restriction the ADG program assumes this to be the case and carries out some checks on the y co-ordinates of the reference points.

The only difference from the manual (volume 1 - section 3) is that the column or voids ratio (e) is omitted. In the manual if the user is typing in the MPD data directly, he/she is expected to enter a 0 value for voids ratio. This is because e is calculated in the MP.

Once all the relevant information has been entered press the RETURN key. This will give you the option of producing a plot of the in situ stress distribution you have just entered.

*6.5.22 Page 22 - IN SITU STRESSES AT REFERENCE POINTS*

This page follows page 21 and is a menu with the following choices :

1) Plot in situ stress profile.
2) edit in situ stresses specified at reference points.
3) Return to in situ stress menu (page 17).

This allows you to look at the in situ stress profile you have specified in page 21 (option 1). Having looked at the stresses if you decide to make changes or correct any errors then choose option 2 which will return you to page 21. However you cannot go direct to page 20 from here (ie for example to change the number of reference points). Option 3 will return you to the in situ stress specification menu in page 17. Once you are in page 17 then choose option 3 to reach page 20.

*6.5.23 Page 23 - PLOT OF IN SITU STRESS PROFILE*

Plot of the in situ stress profile specified at reference points. The horizontal and vertical effective stresses, the pore pressure and the size of the initial yield locus (pc) are plotted against the y co-ordinate.

If two different parameters are identical (for example effective vertical stress and pore pressure) then only one will be displayed. One is over-drawn on top of the other.

Pressing the RETURN key displays the menu page (Page 17).

*6.5.24 Page 24 - IN SITU DISPLACEMENT FIXITIES AND LOADING*

This page is displayed when the 2nd option is chosen from page 16. See elsewhere on how to enter data in this page. (section B of the ADG program manual).

The data entered is this page deals with the following data records :

Records H1, H2, H3, E and record C1 (IPRIM)

The displacement fixities deals with record H3 and H1 (NFIXI). The in situ loading deals with record H2 and H1 (NLODI). Any element removed at this stage are not present at the beginning of the analysis (Parameter IPRIM in record C1 and record E). For example elements representing the embankment in the 'embankment construction' problem.

Quit this page by pressing the F3 key and then typing Y to confirm quitting. This will take you back to the menu in page 16.

To quit this page :

Press the F3 key once Type Y and then press the RETURN key.

*6.5.25 Page 25 - INCREMENT BLOCK MENU - LEVEL 1*

This is one level below the main menu and hence is referred to as Level 1 menu.

This menu is drawn when the option 3 in the main menu has been chosen. The number of options present in this menu corresponds to the number of increment blocks (NOIB) specified in page 10. This leads to level 2 menu.

Choose each increment block in turn and enter the necessary information.

There are 2 further levels of menu. Level 3 and an optional menu at the level below that.

Starting and finishing increment numbers for each increment block are displayed to the right of each option under the heading INCA INCB.

*6.5.26 Page 26 - ANALYSIS SECTION - LEVEL 2 MENU*

This menu has the following options :

1) Input control parameters
2) Displacement fixities and loading
3) Return to level 1 menu

It is necessary to enter information for option 1 (this leads to a data input page and the level 3 menu and further menus and data input pages). However the 2nd option is optional. It needs to be accessed only if you have got any displacement fixities to be specified or loading to be applied or want to remove or add elements.

*6.5.27 Page 27 - ANALYSIS SECTION - CONTROL PARAMETERS - RECORD 1*

Enter the necessary information in this page. The data input in this page relates to records 1 (INCA, INCB, DGRAV, DTIME)

The starting and finishing increment numbers (INCA, INCB) The change in gravity level (DGRAV). The time step for the increment block (DTIME) - in a consolidation analysis.

DTIME, DGRAV applies to the whole increment block. The various increments within the increment block are allocated a fraction of this according to parameters ILDF (for DGRAV) and ITMF (for DTIME) in record I.

Press the RETURN key when the necessary information has been entered.

This leads to the Level 3 menu if there are more than one increments in this increment block. Otherwise the page 30 is displayed on the screen.

*6.5.28 Page 28 - ANALYSIS SECTION - LEVEL 3 MENU*

This has the following choices :

1) printed output options - deals with IOUT (records I and K2)
2) Incr factor options - deals with ILDF (records I and K1)
3) Time step options - deals with ITMF (records I and K3)
4) Return to level 2 menu

The third of the above option is only present in a consolidation analysis.

Each of these options needs to be accessed in turn and data entered. Each option leads to menu with 2 choices. You need to only choose one of these and enter the necessary information.

*6.5.29 Page 29 - ANALYSIS SECTION - PRINTED OUTPUT OPTIONS*

The following choices are available

1) Standard printed output option
2) Individual printed output option
3) Return to level 3 menu.

Option 1 is chosen if you want the same sort of information to be printed for all the increments. However if you want outputs which are different for the different increments then you should choose option 2. For example if there are 10 increments in the increment block and if you only are interested in the detailed output for the

last increment in the increment block and no or slim line output from all the other increments in the current increment block then choose option 2.

If you have previously entered data for this increment block then an 'I' to the right of the menu against one of the 2 options, denotes which option has been chosen previously. You are free to change a previously chosen option ie for which data has already been entered.

You can also change the option half-way through the input or if you have decided otherwise. Simply press the RETURN key which will return you to this page.

Option 1 leads to page 30.
Option 2 leads to page 31.

*6.5.30 Page 30 STANDARD PRINTED OUTPUT - RECORD I*

The 5 numbers you enter (each number can take one of the 3 possible values 0, 1, 2) is combined into a 5 digit code (IOUT in record I) which controls the printed output of displacements and stresses produces the printed output of the same parameters (stresses, displacements etc) for all the increments in the current increment block. This means that this code is common to all the increments in the increment block.

Press the RETURN key which gives page 28.

*6.5.31 Page 31 SEPARATE OUTPUT OPTIONS - RECORD K2*

These parameters control the printed output for each of the increments in the current increment block. These represent IOUT(1) IOUT(2) .... in record K2. The explanation for this code is the same as for IOUT in record I (see the CRISP manual).

Press the RETURN key which gives page 28.

*6.5.32 Page 32 - INCREMENT FACTOR OPTIONS*

This menu has the following choices :

1) equal increment factor for all increments
2) separate increment factors for each increment
3) return to level 3 menu

Only one option has to be chosen from the above 2.

If you have previously entered data for this section then an 'I' will be displayed next to the category with data already entered.

Option 1 sets the parameter ILDF = 0 in record I. This means the load factor for each increment is set to 1/NOINC where NOINC is the total number of increments in the increment block.

Option 2 sets ILDF = 1. This means load factors specified via record K1 (see page 34 below).

The same set of increment factors applies to all the following.

(a) all types of loading. Pressure and nodal loading. Self-weight loading due to DGRAVI (only applicable to analysis of centrifugal tests and some type of collapse analysis using elastic perfectly plastic models).

(b) prescribed displacements and pore pressure changes which are specified using fixity code 1. any implied loading due to addition or

(c) removal of elements.

*6.5.33 Page 33 - EQUAL INCREMENT FACTOR OPTION*

This page is printed when you choose option 1 in page 32. This is just an informative page allowing you to either confirm setting ILDF = 0 or if you change you mind get back to the menu in the previous page. Use the RETURN key to confirm or press the 'Esc' key to get back to previous menu.

Pressing the RETURN key will take you to Page 23.

*6.5.34 Page 34 SEPRATE INCREMENT FACTORS - RECORD K1*

If you have chosen option 2 in page 32 then this page will be displayed.

This means ILDF = 1 and the increment factors you enter here corresponds to R(1) R(2) .... in record K1. The sum of all the

increment factors should be equal to 1.

Press the RETURN key once you have entered the information. This should return you to page 28.

*6.5.35 Page 35*

This menu has the following choices :

1) equal time steps for all increments
2) separate time steps for each increment

Only one option has to be chosen from the above 2.

If you have previously entered data for this section then an 'I' will be displayed next to the category with data already entered.

Option 1 sets the parameter ITMF = 0 in record I. This means that the time step for each increment is set equal to DTIME/NOINC where NOINC is the total number of increments in the increment block.

Option 2 sets ITMF = 1. This means time steps are specified via record K3 (see page 37 below).

*6.5.36 Page 36 - EQUAL TIME STEPS*

This page is printed when you choose option 1 in page 35. This is just an informative page allowing you to either confirm setting ITMF = 0 or if you change you mind get back to the menu in the previous page. Use the RETURN key to confirm or press the 'Esc' key to get back to previous menu.

Pressing the RETURN key will take you to Page 28.

*6.5.37 Page 37 SEPARATE TIME STEPS - RECORD K3*

If you have chosen option 2 in page 35 then this page will be displayed.

This means ITMF = 1 and the time steps you enter here corresponds to DTM(1) DTM(2) .... in record K3. The sum of all the increment factors should be equal to DTIME specified in page 27 (record I).

Press the RETURN key once you have entered the information. This should return you to page 28.

*6.5.38 Page 38 MENU TO CREATE AN ADG FILE*

The menu has the following 2 options :

(a) to create a new ADG file
(b) to overwrite the existing ADG file

Unless there is a special reason always choose option 2 ie to overwrite an existing ADG file. This way you are working with a single ADG file and this file is easily identified by the single analysis name. The files relevant to a single analysis can be easily identified using DOS command

DIR analysis-name.* example : DIR WALL1.*

*6.5.39 Page 39 : ADG directory name*

Displays the current directory name in a field. Press the RETURN key if the ADG file you are working with is in the current directory, which is the recommended way of using the ADG program. Otherwise enter the full path name for the directory. Then press the RETURN key.

If for some reason you want to quit the program, simply press the F3 key. This should return you to the DOS prompt.

*6.5.40 Page 40 : ADG file name*

If you have chosen option (a) in page 38 (ie asked to create a new ADG file), then this page is displayed. You are expected to type in a file name, possibly a variant of the analysis name used for the SMD file.

Once you have typed in the name pressing the RETURN key returns you to the main menu.

*6.5.41 Page 41 : Change SMD/ADG file/directory names*

Once you have selected the SMD and ADG files at the beginning of the program (pages 2 and 5 respectively) and if these files are incompatible ie pertaining to 2 different analyses then the program may display this page and allow you to return pages 1,2,4 or 5 to

select different files or directories.

*6.5.42 Page 42 : Quit program*

Whenever you choose the option to quit from the main menu this menu will be displayed to seek confirmation that you really want to leave. Choose option 2 and press the RETURN key to quit the program.

Choosing option 1 will return you to the main menu.

## Summary sheet for ADG Program

Page 1 : SMD file directory name is specified

Page 2 : SMD file name is selected

Page 3 : Menu for ADG file option (new or existing)

Page 4 : ADG file directory name is specified

Page 5 : ADG file name is selected

Page 6 : ADG Program limits (not necessarily for MP)

Page 7 : Geometry details

Page 8 : Analysis details (if editing existing file)

Page 9 : TITLE Block (Page 1 of 3) Records A & B

Page 10 : Analysis control parameters (Pge 2) Record C1

Page 11 : Printed output control parameters Record C2

Page 12 : Main menu

Page 13 : Choose Material Zones from menu

Page 14 : Choose Material Type for chosen zone

Page 15 : Enter Material properties

| | | |
|---|---|---|
| Page 16 | : | In situ stress stage |
| Page 17 | : | In situ stress specification menu |
| Page 18 | : | Zero in situ stress option |
| Page 19 | : | Specify linear variation of in situ stresses |
| Page 20 | : | Enter number of reference points |
| Page 21 | : | Specify in situ stresses at reference points |
| Page 22 | : | Menu with option to view in situ stresses |
| Page 23 | : | Plot of in situ stresses at reference points |
| Page 24 | : | In situ displacement fixities and loading |

| | | |
|---|---|---|
| Page 25 | : | Increment Block menu - level 1 |
| Page 26 | : | Analysis section - level 2 menu |
| Page 27 | : | Analysis section - ctrl Parameters - record I |
| Page 28 | : | Analysis section - level 3 menu |
| Page 29 | : | Analysis section - Printed output options |
| Page 30 | : | ~~Specify linear variation of in situ stresses~~ Όμοιο με κάτω σελίδας |
| Page 31 | : | Separate output options - Record K2 |
| Page 32 | : | Increment factor options |
| Page 33 | : | Equal increment factors |
| Page 34 | : | Separate increment factors - Record K1 |
| Page 35 | : | Time step options |
| Page 36 | : | Equal time steps |
| Page 37 | : | Separate time steps - Record K3 |

## 6.6 Limitations in using the SMP/ADG Programs

### *6.6.1 Introduction*

This note has been prepared for people who have been using the CRISP-84 program and who intend to update to the CRISP-90 programs. The addition of new modules are intended to make the data preparation easier for both Geometry and Main (analysis) programs in a 386 compatible environment. However using the SMP/ADG programs restricts the type of CRISP analysis that can be carried out.

The limitations of the SMP and ADG programs are described below :

### *6.6.2 Limitations in using the SMP/ADG Programs*

(a) It is not possible to carry out 3 dimensional analysis or 2 dimensional plane strain analysis with bar/beam/slip elements.

(b) Maximum of 10 material zones in any analysis. CRISP MP allows for 25 material zones.

(c) Maximum of 10 increment blocks. CRISP MP does not have any limit on the number of increment blocks.

(d) It is not possible to specify super-imposed elements. ie 2 sets of elements occupying the same region - useful in analysis where soil is excavated to be replaced by a structure (example : retaining wall).

(e) It is not possible to apply point (nodal) loads. In MP this is done using records B3 and L option (a).

(f) It is also not possible to release previously specified fixities. In the Main program it is possible to release previously fixed element sides by re-specifying the fixity records M, but by specifying a zero fixity code.

(g) It is not possible to fix individual nodes. Again this is possible using record N in the MP.

(h) Specifying in situ stresses using INSIT = 2 in record F and using records G2 and G3 is also not available when using the

SMP/ADG programs.

The use of bar/beam/slip elements may be possible in future updates.

## 6.7 Using the ADG Program to generate MP input data (MPD)

The MP input data generated by the ADG program (MPD) is consistent with the input specifications for the MP (which appears in section 3 of the CRISP brown manual Volume 1. However some minor conflicts may be apparent to the users of the ADG program.

Some parameters are automatically set by the ADG program and beyond the control of the user of the ADG program. For example the LINK number (record B of MP input data) is automatically set by the ADG program and cannot be altered by the user from within the ADG program.

The scope of the ADG program is restricted and as explained in section 6.6 some of the options provided in MP are not accessible from the ADG program. The ADG program is also somewhat restrictive. The maximum number of material zones allowed is 10 whereas MP can have upto 25 different material zones. Similarly the limit on the number of increment blocks is 10 in the ADG program whereas there is no limit in the MP program. These limits were imposed to simplify the ADG program.

The differences between the ADG program and section 3 of the brown manual, volume 1 are listed below. The ADG program is more restrictive and can be viewed as forming a sub-set of the full capabilities of the CRISP-90 MP. It should be remembered that the data file created by the ADG program (MPD) has to satisfy the 'input specification' to the MP program as given in section 3 of the brown manual (Vol. 1).

comparar c/ pág. 2.9 manual volumen 1

Record A

TITLE (restricted to 30 characters in ADG)

Record B

LINK (the ADG program gives the same number as in the GP input data. It consists of time and date. The user cannot change this and there is no reason to do so).

Record C1

IPRIM ICOR

These are omitted from page 2 of the TITLE BLOCK. IPRIM is set automatically to the total number of elements within the super-elements removed at in situ stage to form the primary mesh,

Similarly ICOR is set to 1 in the presence of soils model of type 5 (elastic perfectly plastic models).

Record C3 and C4

NINCP is set to 0 by default. This option of post processing is not available for the PC-386 version. This is obsolete because of the options provided by ISR in (record C2) and the results in file *.NPS.

Since NINCP = 0 record C4 is omitted.

Record D

Only inconsistency is in the Schofield model which

requires 16 input parameters. All other soils models require only 12. The permeability in regions which have developed tension cracks were specified as follows :

$k_{xt}$ - 15th parameter
$k_{yt}$ - 16th parameter

In ADG program, even for the Schofield model only 12 parameters are expected to be input. The 6th parameter location which is not used this model was used to specify a single permeability value which is assumed to be the same in both x and y directions. However if the user wants to specify different values of permeability in the tensile crack region, he can edit the MPD file after quitting the ADG program and returned to the DOS prompt with a text editing program. **

Record E

List of elements removed to form the primary mesh is automatically generated depending on whether the user has removed any super-elements at in situ stage via the icon page.

Record F

INSIT

INSIT = 0 or 1 are the only options available in the ADG program.

Record H1 and H2

NLODI > 0 option not available in ADG. Therefore it is not possible to specify nodal point loads from within the ADG program.

Record I

ICHEL is not specified directly. If super-elements have been removed/added in the icon page (for increment blocks) then the program automatically works out the number of finite elements.

NFXB - 0 Since it is not possible to specify individual nodal fixities in the ADG program.

Record L

NLOD > 0 option not available in ADG. Therefore.it is not possible to specify nodal point loads from within the ADG program.

** When using any editing programs make sure you use the 'non document' mode if it is available. Otherwise these programs may introduce control characters into these files causing the MP to abort.

Even though the above section deals with what cannot be input from the ADG program, the user is free to edit MP input data file (MPD) and make changes or add items. The only restriction is that the changes are consistent with the geometry.of the finite element mesh and the input specifications of the MP.

## 6.8 Using the ADG program with stop/restart facility in MP

You can also use the ADG program to generate a MPD data file for use with the stop/restart facility provided in MP.

If you refer to Chapter 3 of CRISP brown manual volume I it specifies that for a re-started analysis records E to H3 are omitted. If the value of INC1 is greater than 1 in record C1 in the MPD file the program assumes that the current run is a re-started analysis. The same procedure is used in the ADG program. When you enter a value of INC1 greater than 1 in page 10 of ADG the program

will skip writing the above mentioned records to the MPD file.

Use the ADG program in the normal manner but skip inputting the in situ details or changing previously input in situ details. You don't even have to delete previously input in situ details. The program simply ignores whatever information is there about the in situ stage. However these information remain accessible and can be viewed in the usual manner. Just choosing to create a MPD file for a stop/restart run does not cause the information about the in situ stage to be lost. This is in case you change your mind.

Future updates may allow the choice of generating the MPD file for a stop/restarted run to be made explicitly and thus suppressing the in situ details.

*6.8.1 Dealing with files in stop/restart run*

Care is needed in adopting a naming convention when using the stop/restart facility. The name WALL1 is used as an example.

(1) Starting a new analysis : WALL1.MPD will be used for the MP input data and it will create the following files : (a) WALL1.NRS (b) WALL1.MPO.

(2) Preparing data for the restarted run : When you use the ADG program again it is going to overwrite the contents of WALL1.MPD. Therefore first of all make a copy of the data file (use COPY or REN)

```
COPY   WALL1.MPD  WALL1A.MPD
```

Here 'A' is added to the end of the name to identify it as the first in a series of data files. Then ADG can now freely overwrite WALL1.MPD which can be directly used for the re-started run of MP.

(3) Running MP (restarted run) : The MP program expects to read the results from the previous run in WALL1.ORS. However this results is currently stored in WALL1.NRS. Therefore use the COPY command to rename the file.

COPY WALL1.NRS WALL1.ORS

When the MP runs it will read the previously stored results from WALL1.ORS and create a new WALL1.NRS file. It will also create a new WALL1.MPO (printed output file) which will overwrite the previous contents. In running MP no checks are made to see whether the above mentioned files exist or not. If any of the files of the same name exist these are overwritten.

If you want to keep backup copies for future reference then use the COPY command before carrying on with the next stage of the analysis.

COPY WALL1.MPO WALL1A.MPO

If you want to continue the re-started run again then use the character 'B' to identify the files used in the second stage. This way you end up with a series of files

WALL1A.MPD WALL1B.MPD WALL1C.MPD

in sequence for future reference.

All this is still done with a single ADG file. Of course you can adopt the same procedure to store copies of the ADG file corresponding to each stage of the MP run.

WALL1A.ADG WALL1B.ADG WALL1C.ADG

This way you can quickly recover any of the previously input information which corresponds to each stage.

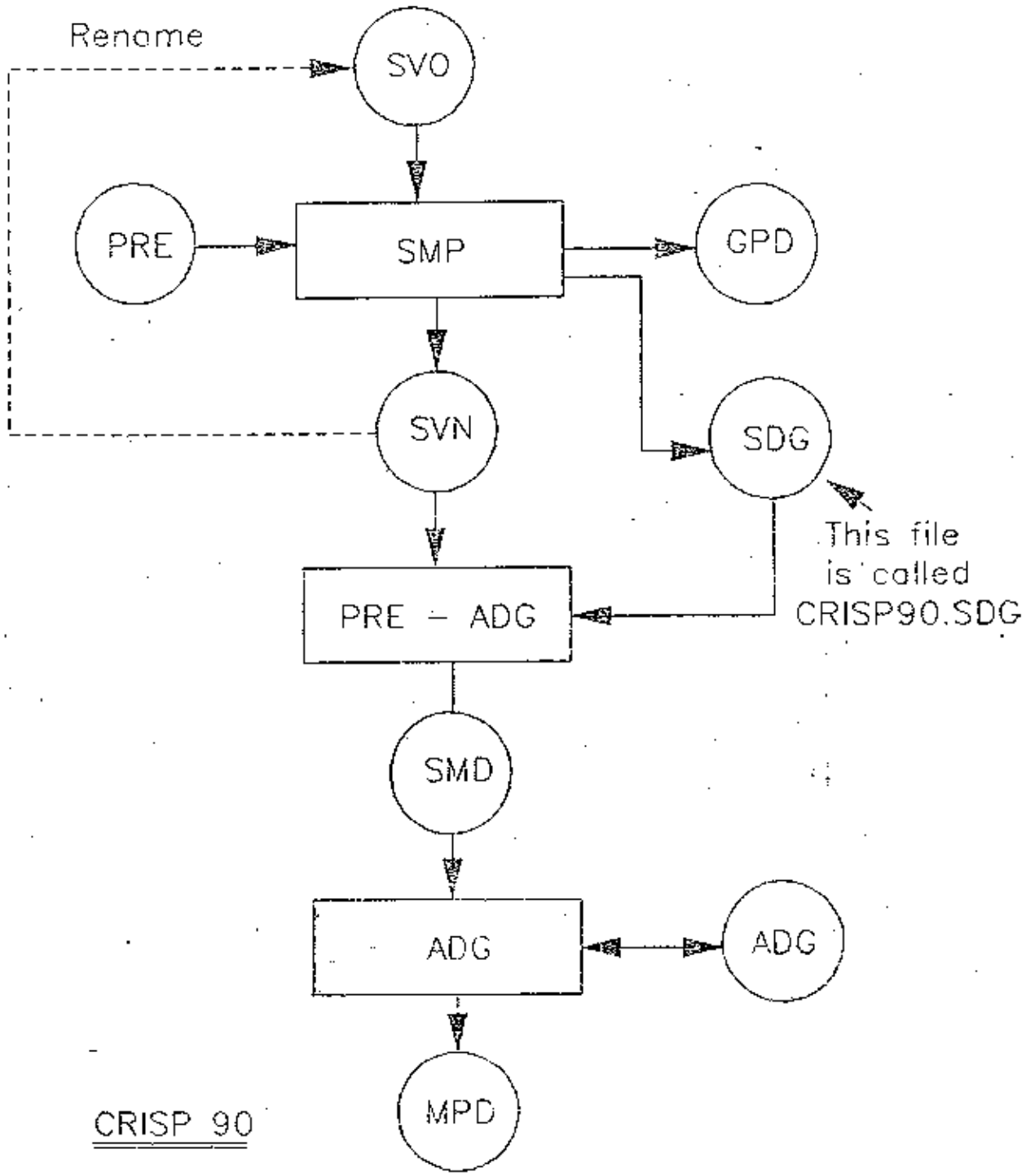



Figure 6.1

SMP, PRE–ADG and ADG Program modules

Files are identified by their extension names

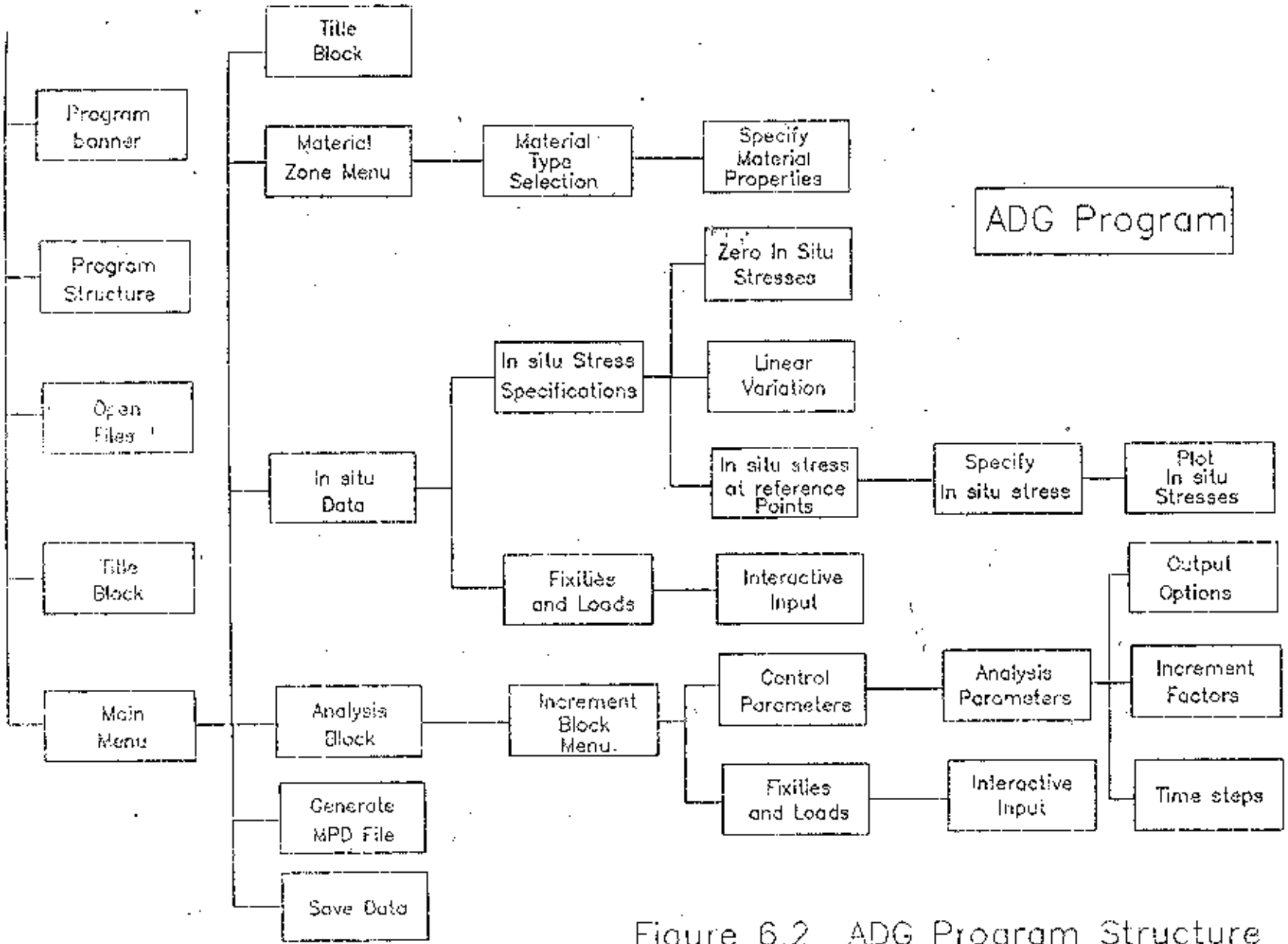



Figure 6.2 ADG Program Structure

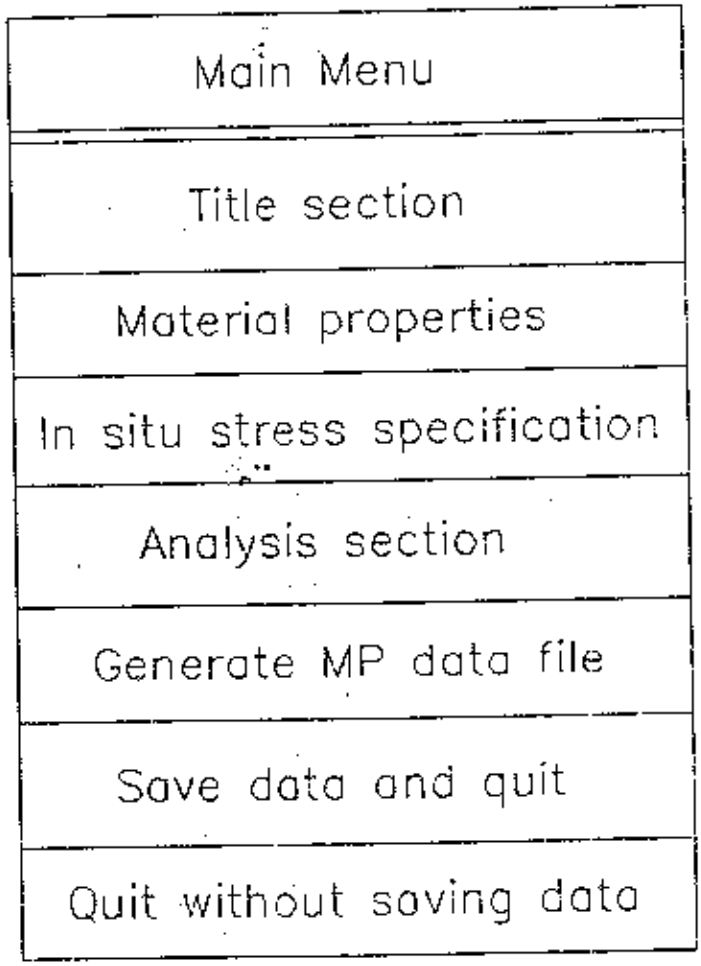



Figure 6.3 Main menu – ADG program

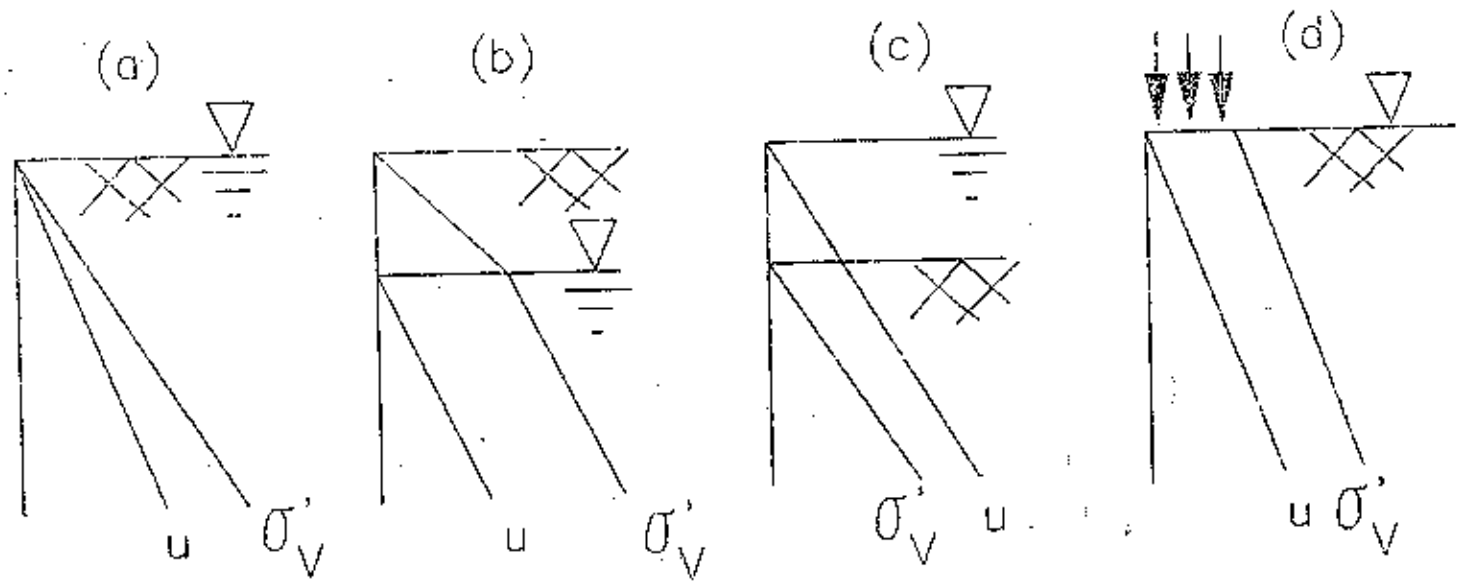



Figure 6.4 In-situ stress distributions - linear variation

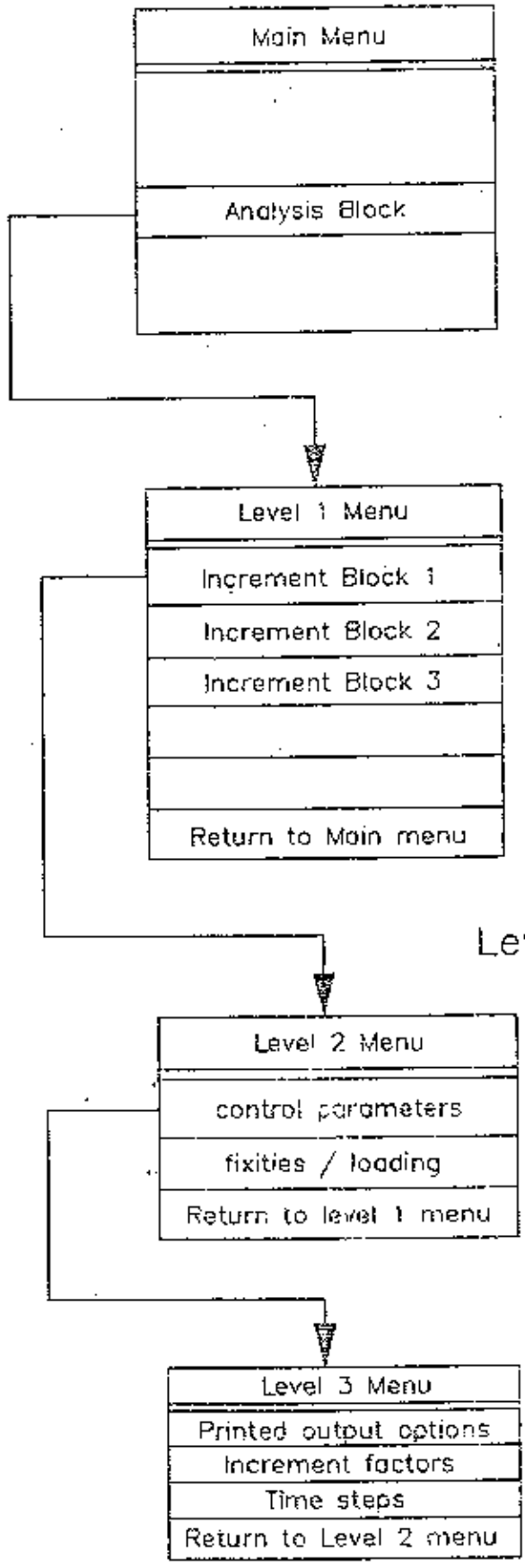



Figure 6.5

Level 1, 2, 3 menus

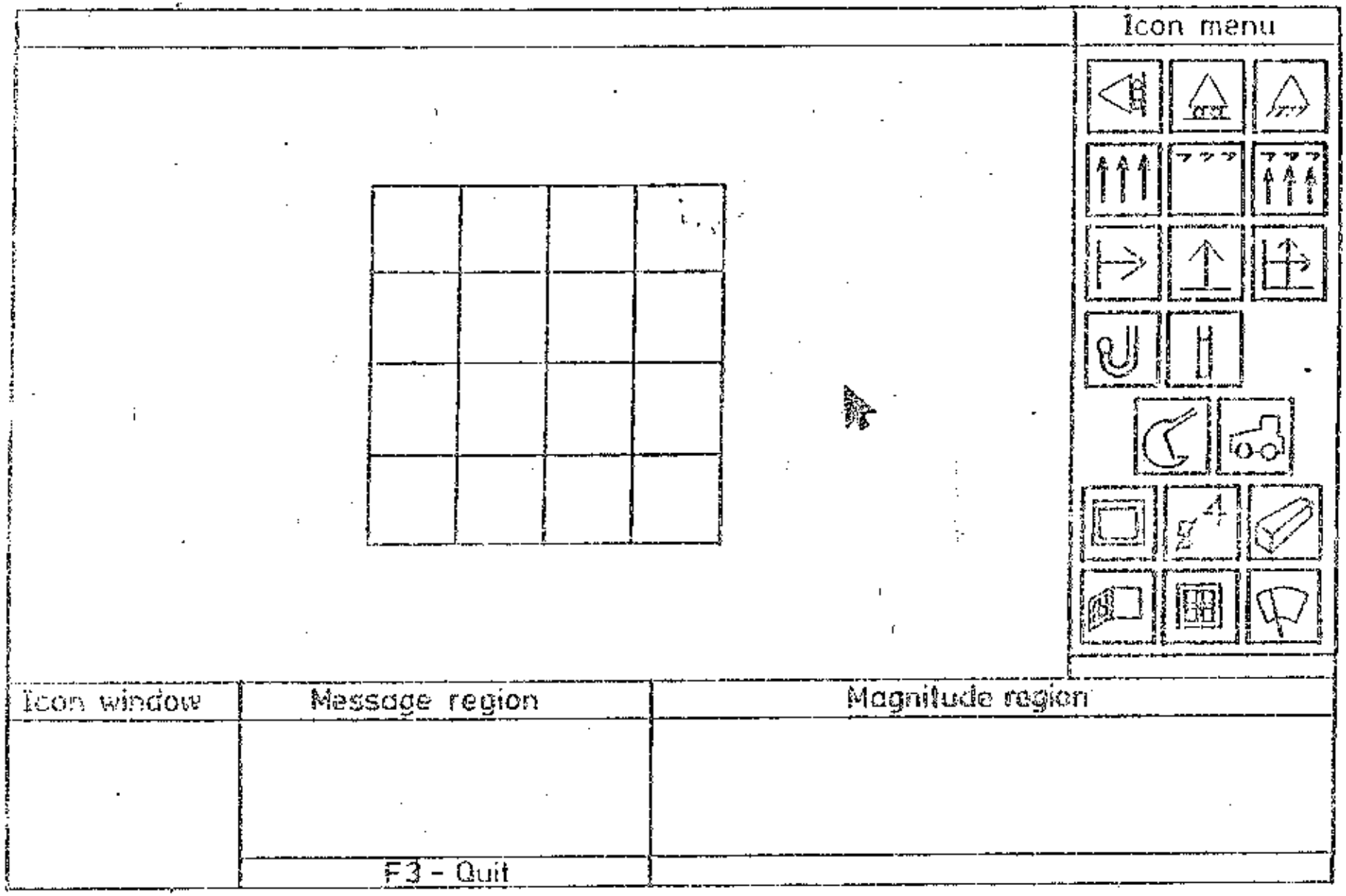



Figure 6.6 – Icon page

1.  x fixity (side restrianed in x direction)

2. y fixity (side restrained in y direction) 

3.  x and y fixities (side restrained in both x and y directions)

4. normal stress 

5.  shear stress

6. normal and shear stress 

7.  prescribed x displacement

8. prescribed y displacement 

9.  prescribed x and y displacements

10. incremental excess pore pressure fixity 

11.  absolute excess pore pressure fixity

12. excavation 

13.  construction

14. display values 

15.  edit magnitude

16. erase 

17.  open window

18. close window 

19.  erase all data

Figure 6.7 ICONS used in ADG Program

# 7. GEOMETRY AND MAIN PROGRAMS

(see the CRISP-90 brown manual volume 1, chapters 3 and 4).

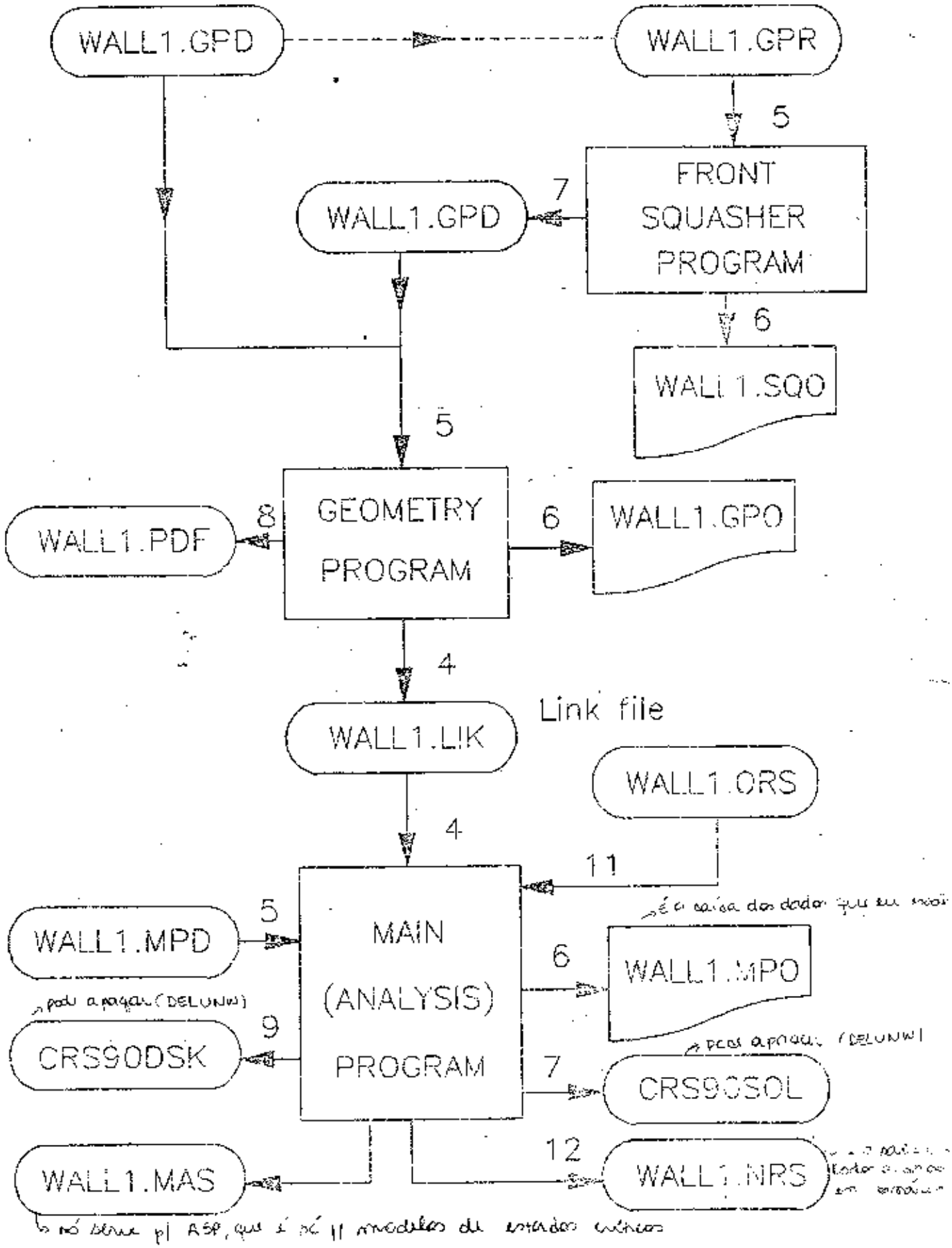



Figure 7.1 Files used by SQ, GP and MP

# 8. POST PROCESSING PROGRAMS - PP

## 8.1 Running the PP program

Choose the option to run the post processing program from the CRISP-90 menu. This will display the "Confirm Graphic Devices" page. Press the RETURN key. The PP program banner will be displayed. Press any key to continue. This will display page 1.

### *8.1.1 Page 1 : File section - NRS file directory*

Displays the current directory name in a field. Press the RETURN key if the NRS file you are working with is in the current directory, which is the recommended way of using the PP program. Accessing files in other directories may slow down the response of the program, hence not recommended. If for some reason you want to quit the program, simply press the F3 key. This should return you to the DOS prompt.

If you are stuck on this page, try pressing the Esc (Escape) key once. If there is no response then try pressing the F3 key. If there is no response then try re-booting the computer by pressing CTRL/ALT/DEL keys.

### *8.1.2 Page 2 : NRS file selection*

Displays the files with the extension NRS which are in the current or user selected directory. If there are more than 10 NRS files then these are displayed 10 at a time. If the file name you are looking for is not found then use the 'Page up' key to scroll the file-names. Pressing the 'Page Up' key displays the next 10 NRS files, if these exist. 'Page down' key can be used to scroll backward.

Once the file you are looking for appears in the screen use the up or down arrow keys to highlight that file name and then press the RETURN key. This means that you have selected the highlighted file. At this point if you find that you have made the wrong choice proceed to the next page (page 3) and choose the second option

'choose a different NRS file'.

*8.1.3 Page 3 : Change NRS directory/file or confirm*

A menu with the following options are displayed *

(1) confirm file selection. → goto page 4
(2) choose a different NRS file. → goto page 2
(3) choose a different directory. → goto page 1

If the NRS file you have selected in page 2 is the correct one, choose option 1 (it would be highlighted, by default) and press the RETURN key. If you had made a mistake in the choice of the NRS file then choose option 2. If you think the NRS file may be in a different directory then choose option 3. This should then return to page 1.

*8.1.4 Page 4 : TITLE for plots*

Type in appropriate title for the analysis you are post-processing. This will be displayed with the plots generated later. A default title 'CRISP ANALYSIS' will be displayed in the field. Typeover it and then press the RETURN key.

*8.1.5 Page 5 : Analysis type*

From the menu make the appropriate choice.

(a) Plane Strain
(b) Axisymmetric

*8.1.6 Page 6 : Select mode*

The menu with the following options will be displayed.

(1) Incremental mode
(2) cumulative mode
(3) Choose a different analysis
(4) Quit the program

Choose either option (1) or (2). Option (2) allows you to look at the results at the end of a given increment (INC2) for which the results must be present in the NRS file. In the incremental mode the

results at the end of increment INC1 are subtracted from the results at the end of increment INC2 and the differences can be viewed in the plots. The increment numbers INC1 and INC2 are specified via the next page. If using the incremental mode it is assumed that the results of both increments INC1 and INC2 are present in the NRS file. For cumulative mode INC1 = 0.

Option (3) is again if you want to look at a different analysis (a different NRS file). This option is provided because this is the page you return to once you have looked at the results you have selected and when you want to choose some other increment or analysis.

To complete the choice option (4) to quit the program is also included.

*8.1.7 Page 7 : Increment numbers*

For cumulative mode type in the increment number (INC2) in the given field. If you have chosen the incremental mode type in the increment numbers INC1 (in the first field) and INC2 (in the second field).

INC1 - starting increment number
INC2 - finishing increment number

INC2 must be greater than INC1. For incremental mode INC1 cannot be equal to zero. Once this is done press the RETURN key.

*8.1.8 Page 8 : Reading results from the NRS file*

The program will display the current choice of mode (incremental/cumulative) and display the increment number(s) and display the increment number currently being read from the NRS file.

Once the particular increment is read correctly, page 10 will be displayed. However if any errors are encountered or end-of-the NRS file has been reached without finding the results for the increments you have requested appropriate error messages will be displayed in the bottom half of this page. Press the RETURN key to continue.

Then if the program was able to read at least the results of one

increment from the NRS file (may not necessarily be the increment you had requested) then you have the choice to continue with the PP program. In that case the program will display page 9. Otherwise the program will be terminated and you have no choice.

### *8.1.9 Page 9 : Errors in NRS file. Change option*

If this page is displayed then something is wrong with the NRS file. However the results of at least one increment was readable (may be more) and hence you can view the results of any of the increments the program was able to salvage. Only increments upto and including the increment number displayed in the previous page can be viewed and no other.

The menu displays the following options.

| | |
|---|---|
| (1) change mode | → goto page 6 |
| (2) change increment number | → goto page 7 |
| (3) quit program | → goto page 18 |

Choose option (2) if you decide to choose any of the increments available for viewing. If you want to change the mode (incremental to cumulative or vice versa) then choose option (1). If you want to select a different analysis (NRS file) then choose option (1), which is catered for by the options available in page 6.

### *8.1.10 Page 10 : Menu 1 : Choice of plots*

The menu displayed has the following options:

| | |
|---|---|
| (1) View deformed mesh plots | → goto page 11 |
| (2) View all other plots | → goto page 13 |

All the plots available in this program are grouped under these 2 headings. The reason is that the plot of the deformed and undeformed mesh makes use of the nodal co-ordinates and the nodal displacements. All other plots make use of either the central integration point (centroid) or all the integration points within a element. Therefore it is logical to divide these into two separate sections.

Choose each option in turn and look at the available plots. To begin with choose option 1. This will lead to page 11. If you choose option 2 this will lead to page 13.

*8.1.11 Page 11 : Scale for displacement*

Type in an appropriate magnification factor for the displacement for plotting the deformed mesh and then press the RETURN key. This will lead to page 12.

*8.1.12 Page 12 : Selection of mesh plot*

This displays the following options.

| | | |
|---|---|---|
| (1) View undeformed mesh only | → | goto plot page |
| (2) View deformed mesh only | → | goto plot page |
| (3) View both | → | goto plot page |
| (4) Change scale for displacement | → | goto page 11 |
| (5) View other plots | → | goto page 13 |
| (6) Select another increment | → | goto page 6 |
| (7) Select another analysis | → | goto page 2 |

The first 3 choices lead to the mesh plot. The commands in section 4.7 gives the details of all the "windowing options". Press the W key to return to this menu. Option 4 allows to change the scale for the displacement if the one you have chosen before is unsuitable.

If you want to look at the other plots for the current choice of increments then choose option 5. This will take you to page 13. If you want to look at the results of a different increment then choose option 6. Option 7 allows you to select an entirely different analysis (NRS file).

*8.1.13 Page 13 : Menu - other plots and options*

The following options are displayed in this page.

| | | |
|---|---|---|
| (1) Menu of available plots | → | goto plot 17 |
| (2) Choose data point | → | goto plot 15 |
| (3) Change scale factors | → | goto plot 16 |
| (4) View deformed mesh plot | → | goto page 11 |
| (5) Choose another increment | → | goto page 6 |
| (6) Choose another analysis (NRS file) | → | goto page 2 |
| (7) Quit the program | → | goto page 18 |

*8.1.14 Page 14 : calculation in progress*

The message 'calculations are being carries out for the plots' will be displayed. Once the calculations are complete this page will be automatically cleared and page 17 will be displayed.

*8.1.15 page 15 : Data point selection*

The menu has the following choices.

(1) Centroids only
(2) all integration points

In option (1) only the central integration point will be used in producing the plots. In option (2) all the integration points in all the elements will be considered in producing the plots. Make the appropriate choice and then press the RETURN key to return to page 13.

*8.1.16 Page 16 : Scale factors used in the plots*

This page displays the following fields, for entering values. The default values are displayed.

| | | |
|---|---|---|
| (1) Scale factor for zero extension lines | (FSN) | 0.3 |
| (2) Scale factor for displacements | (DSPMG) | 1.0 |
| (3) Scale factor for prin strain direction lines | (SLEN) | 0.4 |
| (4) Scale factor for denoting data points | (GRID) | 1.0 |
| (5) Scale factor for principal strains | (RNOIP) | 4.0 |

FSN affects the size of the lines used in plotting the zero extension lines.

The parameter GRID governs the size of the lines representing the principal strain directions (plot number 2), stress directions (plot

number 5) and the size of '+' mark indicating the position of the data points.

The parameter SLEN also has an effect on the length of the lines representing the principal stress/strain directions.

RNOIP is the length of the line representing 1% strain.

*8.1.17 Page 17 : Menu for other plots*

The menu displays the available plots under the this category.

| | | | |
|---|---|---|---|
| (1) Zero extension lines | → | goto plot page | ++ |
| (2) Principal strain directions | → | goto plot page | ++ |
| (3) Displacement vectors | → | goto plot page | ++ |
| (4) Principal strains | → | goto plot page | ++ |
| (5) Data point locations | → | goto plot page | ++ |
| (6) Principal stress directions | → | goto plot page | ++ |
| (7) Stress state codes | → | goto plot page | ++ |
| (8) Return to previous menu | → | goto page 13 | |
| (9) Quit the program | → | goto page 18 | |

**++ this leads to the appropriate plots and then returns to this menu.**

Options (1) to (7) can be shown in turn and the plots viewed. At the completion of each plot the 'windowing option' can be invoked (see section 4.7 on the commands that can be used).

If you want to change any of the scales which affects any of the plots then choose option (8) and when page 13 is displayed choose option 3 (in that menu). This would then give page 16 for you to change the appropriate scale factors.

The same option (8) can be chosen to return to page 1 if you want to look at the results from a different increment or a different analysis.

Choose option (9) if you want to quit the program. This will display page 18 for confirmation.

### *8.1.18 Page 18 : Quit program*

The following options are displayed.

(1) Do not quit. Return to previous menu
(2) Quit the program

This page is for confirming that you wish to quit the program. The purpose of this page is to trap any unintentional (accidental) choice of the 'Quit program' option. If that is the case then press the RETURN key and the program will return to the page from where the choice was made. However choose option (2) and press the RETURN key to quit the program as intended.

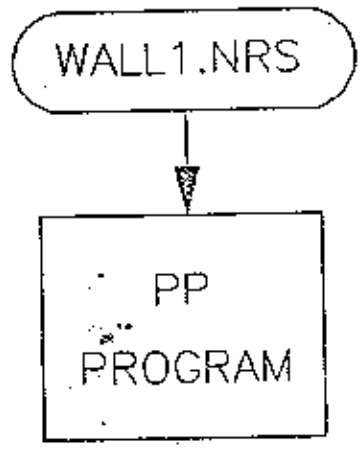



Figure 8.1 File used PP Program

# 9. CRISP - LOTUS 123 INTERFACE PROGRAM

## 9.1 Introduction

Section 9.2 is the manual for the input data to the CRISP - LOTUS 123 interface program (CL). Section 9.3 describes the format of the file created by the interface program.

This program creates an ascii file (a file with the same name as the analysis and with the extension PRN) which can be imported into any spreadsheet program (example LOTUS 123, SUPERCALC). For example if you have been analysing a problem identified by the name WALL1, then this file will be called WALL1.PRN.

See Figure 9.1 for the files used by the CL program.

## 9.2 Manual for the input data for the interface program

```
**********************************************************************

    M A N U A L   F O R   T H E   C R I S P - L O T U S  1 2 3

            I N T E R F A C E   P R O G R A M

        L A S T   M O D I F I E D   O N   2 4   S E P T   8 8

**********************************************************************
```

RECORD

| | | |
|---|---|---|
| A | | TITLE (up to 80 alphanumeric characters) |
| B | | LINK NUMBER |
| C | | IDB(1) ..... IDB(10) |
| D | | INC ⊕ |
| E | | NTAB ⊕ |
| F | | NMAT |
| G | | NTY(1) NTY(2) .... NTY(NMAT) |

<<<< for each table (repeat the following records H I J) >>>>

| | | |
|---|---|---|
| H | | NAME NOL |
| I | | LIST(1) LIST(2) ..... LIST(NOL) $ |

$ These are NODE numbers only if the variable NAME in record H are either XDISP or YDISP or ZDISP. For all other variables these must be element numbers.

<<< the following record only required if LIST( ) above >>>> are element numbers ie omit this if LIST( ) are nodes

J | IP(1) IP(2) ..... IP(NOL)

⊕ Even though there is no limit on the maximum number of increments or the number of tables there is a limit on the total number of lines that can be read by the LOTUS 123 program. The users should ensure that this limit is not exceeded in creating the PRN file.

---

Explanations :

Record A:

TITLE - Title for the LOTUS-123 Table output file

Record B:

LINK - Link number used in the CRISP MP analysis.

Record C:

IDB(1) ... IDB(10) - debugging options (used to test program. for normal run set all values to zero).

1 - print variable name and KUINV number.
2 - print co-ordinates of nodes or Element integration points.
3 - print KUINV number of variable and table no.
4 - print value of variable calculated, table no.
5 - print contents of array VARINT for each increment (warning : this creates an enormous amount printer output).

Record D:

INC - Number of increments to be processed from the NRS file

Record E:

NTAB - Number of tables to be created.

Record F:

NMAT - Number of material zones present in the CRISP MP analysis.

Record G:

NTY(1) NTY(NMAT) - material type numbers (MTP) for the material zones in sequence (same as in records D of CRISP MP input data).

| MTP | models |
|---|---|
| 1 | - Isotropic/Anisotropic elastic material |
| 2 | - Inhomogeneous elastic |
| 3 | - Modified Cam clay |
| 4 | - Cam clay |
| 5 | - Elastic perfectly plastic models |
| 6 | - Schofield soil model |

Record H:

NAME - name of the parameter for the table (as given below on the LHS). This must be enclosed in single quotes (Example: 'SIGXE')

| | |
|---|---|
| SIGXE | - EFFECTIVE HORIZONTAL STRESS (SIG-XX) |
| SIGYE | - EFFECTIVE VERTICAL STRESS (SIG-YY) |
| SIGZE | - EFFECTIVE OUT-OF-PLANE STRESS (SIG-ZZ) |
| TXY | - SHEAR STRESS (TXY) |
| TYZ | - SHEAR STRESS (TYZ) |
| TZX | - SHEAR STRESS (TZX) |
| SIGXT | - TOTAL HORIZONTAL STRESS (SIG-XX) |
| SIGYT | - TOTAL VERTICAL STRESS (SIG-YY) |

SIGZT - TOTAL OUT-OF-PLANE STRESS (SIG-ZZ)
UEXC ** - EXCESS PORE PRESSURE (UEXC)

** not implemented yet (gives UT at present)

UT - TOTAL PORE PRESSURE (UT)
SIG1E - MAJOR EFFECTIVE PRINCIPAL STRESS (SIG-1)
SIG3E - MINOR EFFECTIVE PRINCIPAL STRESS (SIG-3)
SIG1T - MAJOR TOTAL PRINCIPAL STRESS (SIG-1)
SIG3T - MINOR TOTAL PRINCIPAL STRESS (SIG-3)
S - PLANE STRAIN MEAN NORMAL STRESS (S)
T - PLANE STRAIN MAXIMUM SHEAR STRESS (T)
T/S - PLANE STRAIN STRESS RATIO (T/S)
XDISP $ - HORIZONTAL DISPLACEMENT (X-DIR) - Nodal variable
YDISP $ - VERTICAL DISPLACEMENT (Y-DIR) - Nodal variable

ZDISP $ - OUT-OF-PLANE DISPLACEMENT (Z-DIR) - Nodal variable
PE - MEAN NORMAL EFFECTIVE STRESS (PE)
PT - MEAN NORMAL TOTAL STRESS (PT)
Q - DEVIATORIC STRESS (Q)
PC - SIZE OF CRITICAL STATE YIELD LOCUS (PC)
(Cam clays only)
EPSX - HORIZONTAL STRAIN (EPS-XX)
EPSY - VERTICAL STRAIN (EPS-YY)
EPSZ - OUT-OF-PLANE STRAIN (EPS-ZZ)
GAMXY - SHEAR STRAIN (GAMMA-XY)
GAMYZ - SHEAR STRAIN (GAMMA-YZ)

GAMZX - SHEAR STRAIN (GAMMA-ZX)
VOID - VOID RATIO (E) (Cam clays only)
VOLSTR - VOLUMETRIC STRAIN (VOL)
DEVSTR - DEVIATORIC STRAIN (EPS)
MAXSTR - MAXIMUM SHEAR STRAIN
(diameter of Mohr's circle of strain)
MOBPHI - MOBILISED ANGLE OF FRICTION (PHI) DEG
DILANG - ANGLE OF DILATION (PSI) IN DEGREES
AXSTRS - AXIAL STRESS (SIG-AX) - 1D elements only
BM - BENDING MOMENT (BM) - Beam elements only
THXY - ANGLE THETA-XY (IN DEGREES)

SIGM - NORMAL STRESS (SIGM) = 3 x PE
(model number 5 only)
SIGBAR - DEVIATORIC STRESS (SIGBAR) = $Q/\sqrt{3}$

(model number 5 only)

LSFBM - LEAST SQUARE FIT BENDING MOMENT (LSFBM)
ETA - STRESS RATIO - ETA (=Q/PE) Cam clays
PSI - DIRECTION OF MAJOR PRINCIPAL STRESS (in degrees)
EPSILN - DIRECTION OF MAJOR PRINCIPAL STRAIN (in degrees)
ALPHA - DIRECTION OF ZERO EXTENSION LINE (in degrees)
BETA - DIRECTION OF ZERO EXTENSION LINE (in degrees)
B - RATIO (SIG2E-SIG3E)/(SIG1E-SIG3E)
K0 - EARTH PRESSURE COEFFICIENT

**** Note : At least a single space should separate the NAME and NOL in record H. Also note that the variable name must be enclosed in single quotes as shown below:

Examples : 'SIGXE' 10
'XDISP' 6

NOL - Number of elements or nodes which form a table (in sequence and should not exceed 15)

Record I:

LIST(1) ... LIST(NOL) - elements or nodes in sequence

Record J: (only present if record I contains element numbers)

IP(1)..... IP(NOL) - Integration point numbers in sequence

==========

*9.2.1 Example input data to the above program*

record

```
A     |   BACK ANALYSIS 7 OF TESTS HWS3 AND HWS4
B     |   1
C     |   0.0  0  0  0  0  0  0  0  0
D     |   107
E     |   3
F     |   9
G     |   3  1  1  5  5  5  5  5  1
H     |   'BM'     10
I     |   396 396 396 396 396 397 397 397 397 397
J     |   1 2 3 4 5 1 2 3 4 5
H     |   'MAXSTR'     1
I     |   120
J     |   7
H     |   'MOBPHI'     1
I     |   120
J     |   7
```

### 9.3 Format for the PRN file

This file is made up of a number of tables, one table for each variable. The first line in this consists of a TITLE in similar lines to the GP and MP programs. The input data is read from a CLD file. (for the above example called WALL1.CLD). This file has to be prepared according to the CRISP - LOTUS interface program manual (see the previous section).

Each table in the file PRN is given a table number and separated from each other by a line consisting soiely of the character '='. The next line contains the table number and the variable name for which the table contains the values.

example : TOTAL VERTICAL STRESS

or

HORIZONTAL DISPLACEMENT .

If the variable is a nodal parameter (example : displacement) then the next line contains a list of node numbers :

| | | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| TIME | ND> | 10 | 753 | 20 | 768 | 23 | 812 | 35 |

However if it is an integration point parameter (anything other than displacement, for example, stresses, strains) then the next 2 lines contains the list of element numbers (in the first line followed by a list of integration point numbers, in the second line).

| | | | | | | | |
|---|---|---|---|---|---|---|---|
| ELEM | | 24 | 35 | 20 | 23 | 56 | 39 |
| TIME | IP> | 7 | 7 | 7 | 7 | 7 | 7 |

The next line contains the x co-ordinates of these points (nodes or integration points).

X CO-ORD . . . . . .

This is followed by a line containing y co-ordinates.

Y CO-ORD . . . . . .

The next line contains a label DIST followed by a series of values. This is the distance calculated from the first point in the list to the rest of the points. You will notice that the first value is always zero, because it is used as a datum. This is useful if you are interested in plotting distributions.

DIST . . . . . .

This is followed by a number of rows which contain the values of the selected parameter for this table at the various points for every increment. The first value in each row is the increment number. The next is the total time at the end of that increment. This will be

zero if the analysis is not a 'consolidation' analysis. The next value in this row is the value of the selected variable (of this table) at the first point (node or integration point). This is followed by the values at the rest of the points in sequence, falling into respective columns.

The next line contains the values for the next increment. This is followed through for all the increments you had asked for (in the CLD file). This is followed by demarcation line consisting of the character '='.

If you had asked for further tables for the same variable or some other variable the procedure is repeated.

It should be remembered that each table is for a single variable. The list of node numbers or element and integration points could lie along either an approximate vertical section (see Fig. 9.2(a)) or horizontal section (see Fig. 9.2(a)). If this is the case then a row of values for a given increment number can be used to plot the spatial distribution of that variable along that section. This is an useful option. However these points need not represent an approximate section. These could have been selected at random from anywhere in the mesh (see Fig. 9.2(b)). In this later case one cannot use the values along a row to plot the distribution.

It should be stressed that in general a finite element mesh consists of different sized elements distributed unevenly. Very rarely does one find either a set of nodes or integration points lining up along a straight line. The exception is if you are using rectangular elements which are lined up in rows and columns.

For the more general case the user can identify a set of nodes or integration points which form approximately a straight line and this is illustrated in Fig. 9.2(c).

If you have used the SMP, PRE-ADG and ADG programs to generate the finite element mesh and the input data then you donot readily have access to the element and node numbers of the f.e.mesh (not the super-element and super-node numbers).

Use the program for plotting the mesh (type PLTFEM at the DOS prompt). This will prompt the name of the GPD file. Use the 'zoom

in' option (see section 4.7) to identify the nodes and elements of interest and either do a screen dump by pressing the 'Print Screen' on a IBM Matrix printer or in a laser printer which emulates a IBM Matrix printer. You need to select the appropriate emulation when the DOS prompt is displayed and before entering the CRISP-90 menu.

Table 9.1 shows a typical PRN file created by the CL program.

```
"******** ABTWALL  EXAMPLE ****"
"=============================================================================="
"TABLE NUMBER =     1"
"TOTAL HORIZONTAL STRESS (SIG-XX)"
" "          "ELEM"          49          50          33          34
"INC"      "TIME IP>"         9           9           9           9
"X    "        " "        29.500      29.500      29.500      29.500
"Y    "        " "        18.625      15.875      13.260      10.458
"DIS  "        " "         0.000       2.750       5.365       8.167
    1     0.1              113.6       166.4       209.5       248.3
    2     0.2              119.2       173.0       216.0       254.8
    3     0.3              124.7       179.4       222.4       261.3
    4     0.4              130.2       185.8       228.8       267.8
    5     0.5              135.7       192.3       235.3       274.3
    6     0.6              141.2       198.7       241.7       280.8
    7     0.7              146.6       205.1       248.2       287.3
    8     0.8              152.1       211.5       254.6       293.8
    9     0.9              157.5       218.0       261.1       300.3
   10      1.              162.8       224.4       267.6       306.8
"=============================================================================="
"TABLE NUMBER =     2"
"VERTICAL DISPLACEMENT (Y-DIR)"
"INC"      "TIME ND>"        106         107         108         109
"X    "        " "        30.000      31.500      33.000      37.618
"Y    "        " "        21.000      21.000      21.000      21.000
"DIS  "        " "         0.000       1.500       3.000       7.618
    1     0.1          -0.1714E-03  0.6210E-03  0.1421E-02  0.2302E-02
    2     0.2          -0.3364E-03  0.1250E-02  0.2850E-02  0.4863E-02
    3     0.3          -0.4889E-03  0.1873E-02  0.4254E-02  0.7339E-02
    4     0.4          -0.6222E-03  0.2511E-02  0.5669E-02  0.9804E-02
    5     0.5          -0.7369E-03  0.3170E-02  0.7108E-02  0.1228E-01
    6     0.6          -0.8377E-03  0.3839E-02  0.8553E-02  0.1475E-01
    7     0.7          -0.9230E-03  0.4520E-02  0.1001E-01  0.1727E-01
    8     0.8          -0.9929E-03  0.5214E-02  0.1147E-01  0.1983E-01
    9     0.9          -0.1051E-02  0.5916E-02  0.1294E-01  0.2242E-01
   10      1.          -0.1118E-02  0.6604E-02  0.1439E-01  0.2500E-01
```

Table 9.1 Contents of a typical PRN file

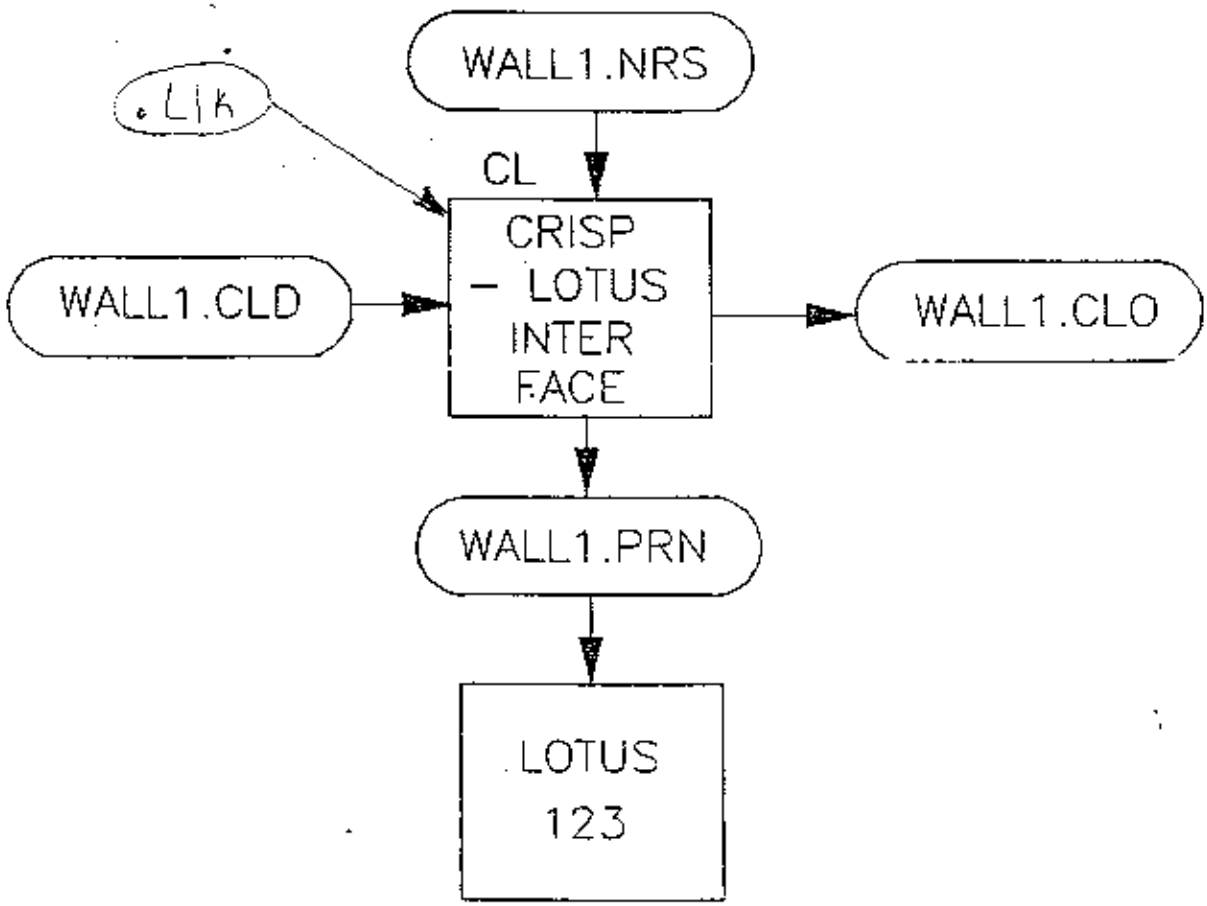



Figure 9.1 CRISP – LOTUS 123 INTERFACE

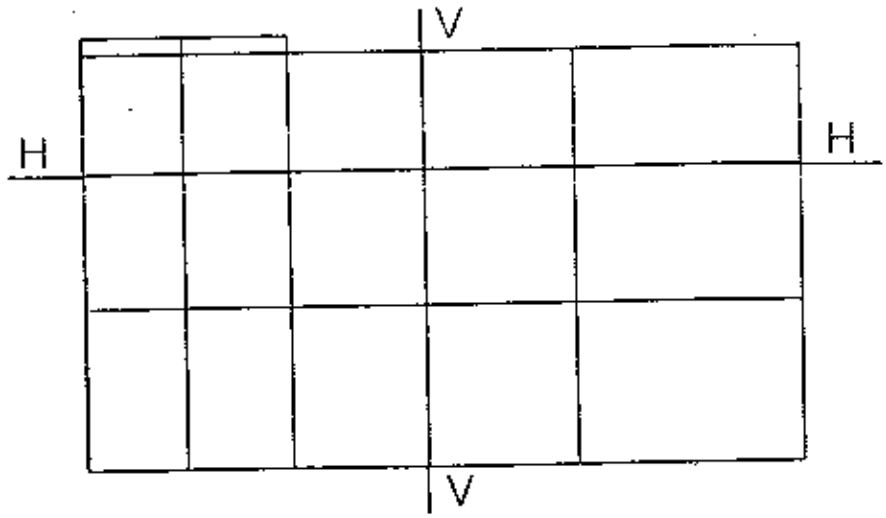



Figure 9.2(a)

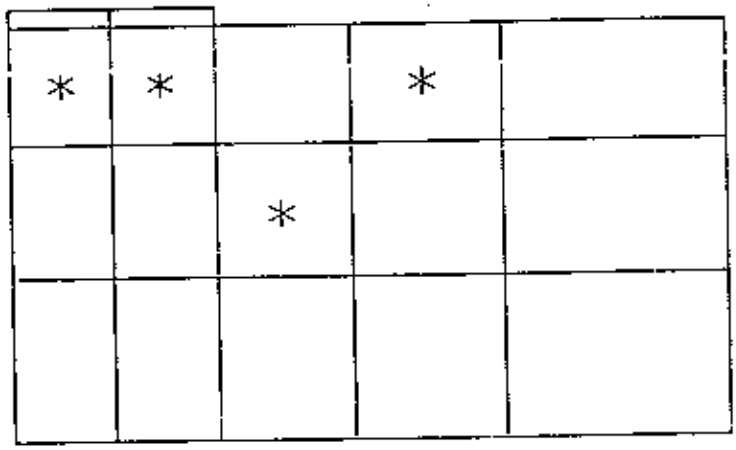


Figure 9.2(b)

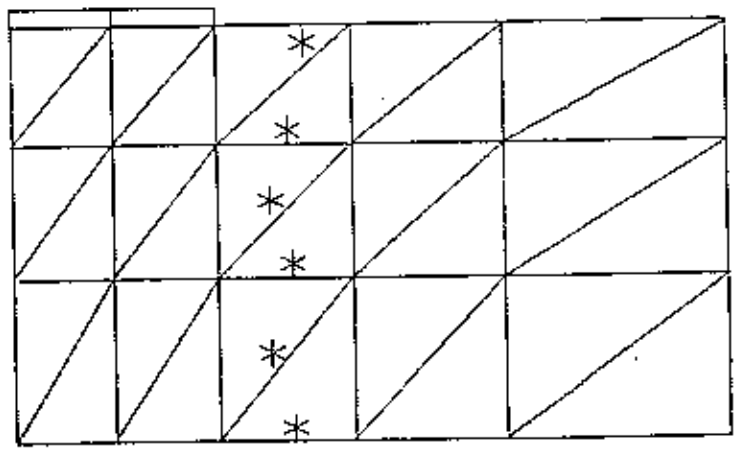


Figure 9.2(c)

# 10. STRESS PATH PLOTTING PROGRAMS (PQ AND SP)

## 10.1 Introduction

In order to run the Stress Path plotting program (SP) one has to run a PRE-SP program called PQ (see Figure 10.1). Section 10.2 gives the manual for the input data required by the PQ program (this file should be given the extension PQD). Section 10.3 gives the manual for the input data required by the SP program (this file should be given the extension SPD). Figure 10.1 shows the relationship between these 2 programs and the file NRS.

## 10.2 manual for the PQ program

**********

M A N U A L F O R P Q (P R E - S P) P R O G R A M.

(F O R C A M C L A Y S O N L Y)

**********

record A

NDP - number of data points (ie integration points)
(should not exceed 20)

record B <NDP records>

IDP MEL(IDP) MIP(IDP)

IDP ** - data point number
MEL(IDP) - element number
MIP(IDP) - integration point number

** The data point number IDP uniquely defines the element number MEL(IDP) and integration point MIP(IDP). The integration point and the associated element number is referred to by the unique "data point" number here onwards (ie in the Stress Path Plotting Program - SP data file).

## 10.3 Manual for the SP program

```
================================================================

M A N U A L   F O R   S T R E S S   P A T H   P L O T T I N G

                P R O G R A M      ( S P )

================================================================
```

record

```
A       TITLE  (12  characters)

B       ITEST

C       IDB(1)  IDB(2)  ..............  IDB(10)

        ....... only included if NDP > 1 ...........
D       NP

           < record E is omitted if ITEST = 0 in record B >

        <NP  records>   ....... only included if NDP > 1 .......
E       II   ND   NPDT(1)  NPDT(2) .......  NPDT(5)

           < record F is omitted if ITEST = 0 in record B >
F       NINCP

G       NYLD   KCSL   KNCL

H       NTH

        <NP  records>
I       I  IMODEL KAPPA  LAMBDA  ECS M GNU  PC  H  SLOPE PTRATIO

        <only present if NTH = 1>
J       IUD   IEXT

        <only present if NTH = 1>   NP records
K       I  INST(I)   SIGX  SIGY  SIGY   TXY   U   E   PCI

L       ISLCT
```

record B

ITEST = 0 testing the program. Does not read CRISP results from unit 3. (ie does not read a PQF file)

1 normal mode for running the program.

record C

IDB(1) .... IDB(10) - debugging options. Print out various arrays.

IDB(1) = N print CRISP file info for all data points for every Nth increment..
IDB(2) = 1 print limits of parameters on CRISP file
IDB(3) = 1 print yield locus values (p and q)
IDB(4) = N output frequency for yield locus values
IDB(5) = 1 print CSL and INCL values (p and q)
IDB(6) = N output frequency for CSL and INCL values
IDB(7) = 1 print limits of yield locus CSL and INCL values
IDB(8) = 1 print theoretical curve values
IDB(9) = N output frequency for theoretical curve values
IDB(10) = 1 print limits of theoretical curves

CSL : Critical state line
INCL : Isotropic normal consolidation line

N : output frequency. Every Nth point (or increment) in the calculation is printed.

NDP = No. of data (integration) points in CRISP results file (ie PQF file).

record D

+++ Records D and E are omitted if NDP = 1. It is assumed that NP = 1 ie. only one page of plots is produced.

NP = number of pages of plots (< 5)

note that records D and E are used to select which of the data points in the CRISP results file (PQF) is to be used in producing the plots. It is not necessary to use all the data points in producing the plots.

record E (record E is omitted if ITEST = 0 in record B )

II = page number
ND = number of data points to be plotted in page II
NPDT(1) ... NPDT(5) = the list of data points to be plotted in page II.

(< 5 - no more than 5 data points per page)

Note that the "data point" numbers uniquely identifies each integration point and the associated element.

record F (record F is omitted if ITEST = 0 in record B )

NINCP The number of increments to be used in plotting.

= 0 or -ve value would mean use all increments (ie NOINC) in PQF file in plotting.

+ve and < NOINC; Then use NINCP increments in plotting. This allows one to produce plots even when an analysis has one wrong towards the end of the analysis. This allows one to look at earlier part of the analysis without it being obscured by the end result.

record G

NYLD = 1 include the plot of the yield locus in all the pages.
0 no yield locus plotted in any of the pages.

KCSL = 0 no CSL plotted in any page.
1 CSL plotted in p' - q space in all the pages.
2 CSL plotted in p' - q and e - p' space in all

the pages.
3 CSL plotted in e - p' space in all pages.

KNCL = 0 no INCL plotted in any page.
1 INCL plotted in e - p' space in all the pages.

record H

NTH = 0 no theoretical stress/strain paths plotted,
1 theoretical stress/strain paths are plotted different curve for each page. see records I and J.

record I (this record is included if NTH = 1 or NYLD = 1 or KCSL = 1 or 2 or 3 or KNCL = 1)

I = page number

IMODEL = model material type number
3 modified Cam clay
4 Cam clay
6 Schofield soil model.

KAPPA = slope of swelling line in e - ln p' space
LAMBDA = slope of compression line in e - ln p' space
ECS = voids ratio at p' = 1 at critical state
M = slope of CSL in q - p' space
GNU = shear modulus or if GNU < 1 Poissons ratio
PC = size of yield locus drawn in q - p' space (see parameter KCSL, record G).
H = slope of Hvorslev surface (model 6 only)
SLOPE = slope of tensile crack region (model 6 only)
PTRATIO = demarcation point between tensile crack and Hvorslev region expressed in terms of p' (pd) as a ratio of critical state p' (px). Calculated within the program, user should specify a zero value. (model 6 only).

Note that for Cam-Clays and Modified Cam-Clays zero values should be specified for H, SLOPE and PTRATIO.

for example:

| page | IMODEL | KAPPA | LAMBDA | ECS | M | GNU | PC | H | SLOPE | PTRATIO |
|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 3 | 0.05 | 0.25 | 2.573 | 1.0 | 0.3 | 200. | 0. | 0. | 0. |
| 2 | 6 | 0.05 | 0.30 | 2.573 | 1.0 | 0.3 | 273. | 0.8 | 3. | 0. |

record J

IUD = 1 plot undrained theoretical paths (for Triaxial tests).
2 plot drained theoretical paths (for Triaxial tests).

IEXT = 0 compression test
1 extension test

record K

I = page number

INST(I) = 0 no theoretical curve to be plotted in page I.
1 theoretical curve to be plotted in page I.

SIGX SIGY SIGZ starting stress point for theoretical paths. (SIGX, SIGY etc are the EFFECTIVE stresses)

U = in situ pore pressure
E = initial voids ratio (not to be confused with ECS).
PCI = size of initial yield locus.

Record L

ISLCT = choice of the plots (total of 6) to be produced as listed below :

= 1 - all six plots are produced
= 2 - only 4 plots are produced (plots 1, 2, 3 and 6)
= 3 - only 2 plots are produced (plots 1 and 2)

---

Note that a "page" of plots in the above consists of a collection of 6 plots, as follows :

(1) q - p'
(2) q - ε where ε is the deviatoric shear strain.
(3) e - p'
(4) e - ε
(5) u - p' where u is the pore pressure.
(6) u - ε

The theoretical path is plotted in white in all these plots. In addition a yield locus is plotted is q - p' space. The CSL can be plotted in q - p' and e - p' spaces. The INCL can be included in e - p' space. The plots (3) and (4) are omitted if e values are all found to be zero (which is unlikely). Similarly plots (5) and (6) may be omitted if all the pore pressures are found to be zero (ie as in a drained analysis).

Only a simple option is included in plotting yield locus, CSL and INCL. They appear in all the pages or not at all. However it is possible to specify different curves depending on different model material properties.

In the above

CSL - Critical state line
INCL - Isotropical normal consolidation line

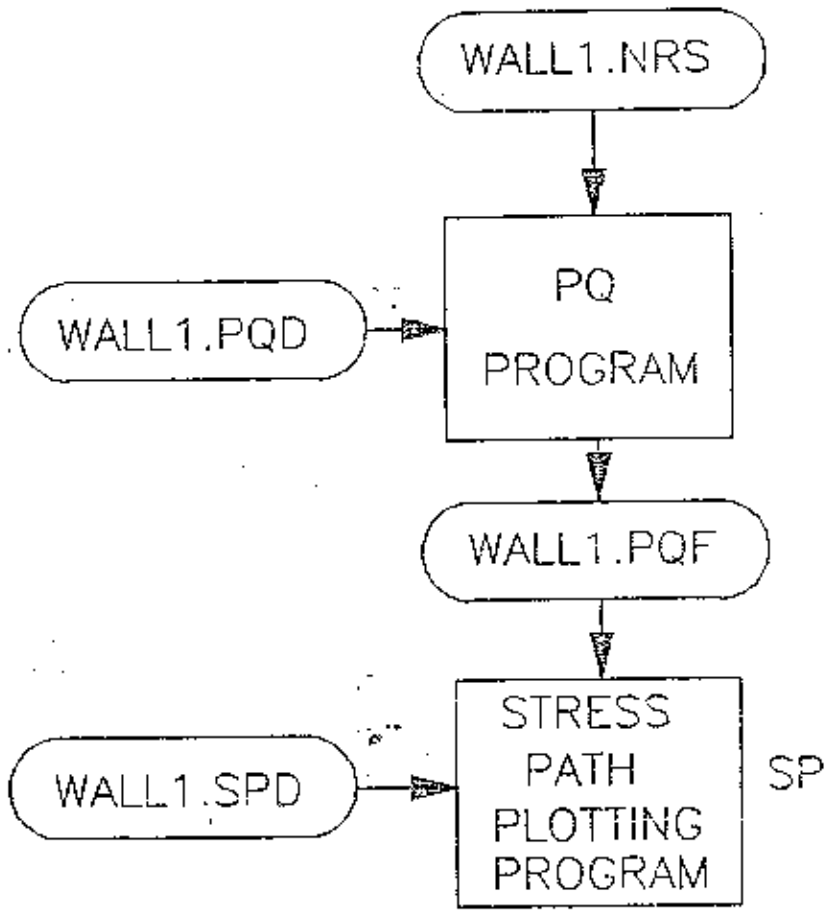



Figure 10.1 Files used by PQ and SP programs

# REFERENCES

## Embankments

(1) Almeida, M.S.S. (1981) Analysis of the behaviour of an embankment on Soft clay, M.Phil Thesis, University of Cambridge.

(2) Almeida, M.S.S. (1984) Stage constructed embankments on soft clays, PhD thesis, Engineering Department, Cambridge University.

(3) Almeida, M.S.S., Britto, A.M. & Parry R.H.G. (1986), Numerical modelling of a centrifuged embankment on soft clay, Canadian Geotechnical Journal, 23, 103-114.

(2), (3)
Analysis of centrifuge test on Kaolin clay foundation consolidated under a vertical pressure of 150 kPa. The Kaolin layer was overlain by a layer of Gault clay. The embankment was modelled using Leighton Buzzard sand in the centrifuge test and treated as a linear elastic material in the analysis. The clay was modelled using Modified Cam clay. Consolidation analysis. Centrifugal acceleration 100 g.

(4) Bassett, R.H., Davies, M.C.R., Gunn, M.J., & Parry, R.H.G., (1981), Centrifugal models to evaluate numerical methods, Proc. 10th ICSMFE, Stockholm.

(5) Davies M. C. R. (1982), Centrifugal Modelling of Embankment on Clay Foundations, Ph.D. Thesis, University of Cambridge.

(6) Vepsäläinen P., Arkima, O., Lojander M., Näätänen A. (1991), The trial embankment in Vaasa and Paimio, Finland, p 633-640, 10th ECSMFE, Florence.

(4), (5)
Analysis of centrifuge test on Kaolin clay foundation consolidated under a vertical pressure of 140 kPa. The embankment was modelled using Leighton Buzzard sand. Consolidation analysis. Modified Cam clay was used to model the clay.

## Oil Tanks on foundation

(7) Kusakabe O. (1980) Centrifuge Model tests of an Oil Tank, M.Phil Thesis, University of Cambridge.

## Shaft Excavations and Trenches

(8) Kusakabe O. (1982) Stability of Excavations in Soft Clay, Ph.D. Thesis, University of Cambridge.

(9) Phillips R. (1987) Ground deformations in the vicinity of trench heading, Ph D Thesis, University of Cambridge.

(10) White T. P. (1987), Finite element calculations involving the yielding of dilatant soils, M Phil Thesis, Cambridge University.

(8), (9), (10)
Analysis of centrifuge tests on Kaolin clay samples consolidated under a vertical pressure of 140 kPa. Axisymmetric shafts of diameters 30, 40, 60 mm and of depths 180 mm were tested at a centrifugal acceleration of 75 g.

(9)
Analysis of centrifuge tests on kaloin layer overlain by a layer of Gault clay. Both semi-circular shafts and axisymmetric shafts were tested. Clay samples were consolidated under a vertical pressure of 140 kPa. Test were conducted at a centrifugal acceleration of 75 g.

Tunnels

(11) Calabresi G., Tamagnini C., Rampello S. (1991), Influence of a soil nailing application on tunnel excavation in slightly overconsolidated clays. pp 677-681, 10th ECSMFE, Florence.

(12) Mair, R.J., (1979), Centrifugal modelling of tunnel construction in soft clay, PhD thesis, Cambridge University.

(13) Mair, R.J., Gunn, M.J., & O'Reilly, M.P., (1981), Ground movements around shallow tunnels in soft clay, Proc. 10th ICSMFE, Stockholm.

(14) Seneviratne H. N. (1979) Deformations and pore-pressures around model Tunnels in Soft Clay, Ph.D. Thesis, University of Cambridge.

(15) Seneviratne, H.N, & Gunn, M.J., (1985), Predicted and observed time-dependent deformations around shallow model tunnels in soft clay,
Proc. 5th Int. Conf. num. meth. in geomechanics, Nagoya, Japan.

(16) Taylor, R.N., (1984), Ground movements associated with tunnels and trenches, PhD thesis, Cambridge University.

Retaining Walls

(17) Sun H. W. (1987), Soil-structure interaction problem of retaining wall on compressible foundation, M Phil Thesis, Cambridge University.

(18) Sun H. W. (1990), Ground deformation mechanisms for soil-structure interaction, Ph D Thesis, Cambridge University.

(19) White T. P. (1987), Finite element calculations involving the yielding of dilatant soils, M Phil Thesis, Cambridge University.

(20) M. D. Bolton, A. M. Britto, W. Powrie and T. P. White (1989) Finite element analysis of a centrifuge model of a retaining wall embedded in a heavily overconsolidated clay. Computers and Geotechnics, Vol.7, No.4, p 289-318.

(21) M. D. Bolton and H. W. Sun (1991) Finite element analyses of bridge abutments on firm clay. (submitted for publication).

(22) Lung, R. K. W. (1982) Variation of the lateral pressure of clay against a model wall, M Phil Thesis, Cambridge University.

(23) Stewart, D.I. (1989) Groundwater effects on in situ walls in stiff clay, Ph.D.Thesis, Cambridge University.

Simple shear tests

(24) M. Budhu and A. M. Britto (1987) Numerical Analysis of Soils in Simple Shear Devices, Soils and Foundations, Vol. 27, No. 2, pp. 31-41.

Spill through abutments

(25) S. M. Springman (1984) Lateral loading on piles due to embankment construction, M.Phil Thesis, Cambridge University.

(25)
2D plane strain analysis where the row of piles are modelled as a plate of equivalent stiffness.

(26) S. M. Springman (1989) Lateral loading on piles due to simulated embankment construction, Ph.D. Thesis, Cambridge University.

(26)
3D linear elastic analysis where the stiffness of the embankment was ignored.

Consolidation tests

(27) Al-Taabaa, A. (1988) Permeability and stress-strain response of Speswhite Kaolin, Ph.D. Thesis, Cambridge University.

Footings

(28) Lau, C. K. (1988), Scale effects on tests on footings, Ph.D Thesis, Cambridge University.

# APPENDIX A

## INSTRUCTIONS FOR INSTALLING CRISP-90 AND RUNNING THE EXAMPLES

### A.1 Installing SALFORD FTN77/386 and CRISP-90

CRISP-90 is distributed in 3 disks. The other disk contains the SALFORD FTN77 Run time system files.

#### A.1.1 Installing SALFORD FTN77/386 run time system

Before installing the SALFORD FTN77 run time system read the relevant files from that disk. If you are not very sure about other softwares which use extended memory which may be used in the 386 computer in which you are planning to run CRISP-90 then seek expert advice.

**** WARNING ****** Otherwise your hard disk may become corrupted.

Following the instructions given in READ.ME file in the SALFORD FTN77/386 disk, install the SALFORD FTN77 run time system in a separate directory.

Example C:\FTN77 or D:\FTN77

Add the above directory name to the PATH statement in the AUTOEXEC.BAT

#### A.1.2 Installing CRISP-90 suite of programs

Create a directory called CRISP90. Copy all the files in the 2 CRISP-90 disks into this directory. Do not forget to copy the files in the MW directory in disk-1 to the CRISP90 directory (if this exists). Disk number 3 contains various input data files and also the output from the example problems given in appendix B of the CRISP-90 brown manual (volume 1). Only copy the files to hard disk when necessary. These files are not required for normal running of the CRISP-90 suite of programs. See chapter 3 for the amount of disk space necessary to install CRISP-90.

### A.1.3 Running a CRISP example to test the system

Install DBOS by typing IDBOS (this loads FTN77/386 run time system and Metawindows)

Then type CRISP90 which should give the menu for running the various modules.

Using the example data (FOOTING0) given in the FOOTING directory in disk 1 test the various programs.

Then try running the four examples which appear in the CRISP manual (volume 1). GP and MP data files are included in the EXAMPLE directory (disk 3). These four examples are for running the GP90 and MP90 programs only and are not for the other programs (example SMP, ADG).

When you have finished running CRISP type KDBOS to remove DBOS and Metawindow from memory. This is advisable before running any other software which make use of extended memory. You need to first type IDBOS if you want to run CRISP again.

## A.2 Notes on Running CRISP-90

If you have to re-boot the computer for some reason then wait till the following DOS prompt is displayed.

```
C:\>
```

It is assumed that you have installed CRISP-90 in the logical drive D. Change the logical disk drive to D by typing D:
Then change the directory to CRISP90 by typing CD CRISP90
This would display the following :

```
D:\CRISP90>
```

At this stage you need to re-load DBOS which is required by CRISP-90 and the Metawindows graphics library into memory before you can run the CRISP-90 suite of programs. This is done by typing IDBOS

```
D:\CRISP90>IDBOS
```

The computer will display the banner for the Salford FTN77/386 run time system. This will be followed by the Metawindow logo with the blue background.

When the DOS prompt is displayed you can run the CRISP-90 programs by first typing CRISP90.

If the following message is printed it means you have forgotten to type IDBOS after switching the computer on.

Salford University FTN77 programs require DBOS

If the following message is displayed and the computer 'locks-up' then it means that the file MWDRIVER.EXE is missing from the current directory.

Real mode library has terminated improperly - system must be re-booted.

You have to re-boot the system and load the softwares into memory again and then copy the MWDRIVER.EXE file into current directory. If the problem persists even after you have copied the MWDRIVER.EXE to the current directory then it is likely that the MWDRIVER.EXE has been corrupted. Contact the authors of the CRISP-90 program for a replacement file. Until you get a replacement you can run only the following programs.

SQ GP MP CL

All other program modules require the MWDRIVER.EXE file.

Then type CRISP90 for the CRISP-90 menu.

CRISP-90

The following programs can be run from the CRISP-90 menu program.

SMP (and PRE-ADG) - Super mesh program and the PRE-ADG program
ADG - Analysis data generation program

| | |
|---|---|
| SQ | - front squasher program |
| GP | - Geometry Program |
| MP | - Main Program |
| PP | - Post processor plotting program |

All the other programs (PQ, SP, CL) require input data to be set up before these programs could be run. Hence these program are always run at the DOS prompt. It is possible in the future that these program can also be run from the CRISP-90 menu.

## A.3 Troubleshooting

### General

If there are more than about 100 files in the current directory you may find that the ADG program becoming sluggish. Delete any unwanted files as soon as you have carried out any analysis. Since the files relevant to a particular analysis are readily identified store, the data files and delete the other output and results file (SQO, GPO, MPO, PPO, NRS etc).

Create a sub-directory with the name of the analysis you are currently carrying out. Then use the COPY command to copy the relevant files to this sub-directory. Check that these files have been correctly copied, and then delete it from the CRISP90 directory.

```
Example :   MD  WALL              | make a sub directory
            COPY WALL1.*  WALL    | copy all the files
            CD  WALL              | change to sub-directory
            DIR                   | check files have been copied
            CD ..                 | return to CRISP90 directory
            DEL WALL1.*           | delete the files
```

When you want to re-run the WALL1 analysis first copy all the files back into the CRISP90 directory.

Store the files with the following extensions (if these exist) :

SVO SVN PRE GPD MPD SMD ADG PPD SPD PQD CLD

Then all the other files can be re-created using these files and re-running the relevant programs.

SMP (make sure that the CAPS LOCK is on)

It is possible that the program may crash out due to some bugs. If this happens (or if any graphics based program crashes out - PP, ADG) type first CLS (to clear the screen) then type MODE CO80. This should restore the terminal to the normal text mode. Then try re-running the program.

If you happen to type the wrong file name or make a mistake the program will display

..... FILE DOES NOT EXIST. RE-ENTER

If you cannot remember the filename or typed the wrong name press the CTRL/BREAK key first and then press Y when the program seeks confirmation to terminate job, with the following question.

Terminate batch job (Y/N)?

When the DOS prompt > is displayed type CLS and press the return key, then MODE CO80 and press the return key. Then the program can be re-run after you have checked the file names. First type CRISP90 to get the menu.

PRE-ADG

This program is run automatically after the SMP program. First the 'Confirm graphic devices' page appears on the screen and then the banner. After this there will be a short pause, while the screen remains blank. The delay will depend on how complex the super-mesh is. Then the super-mesh is plotted with all icon positions displayed. Note that inclined sides and inner element sides are not assigned any icons. Press the RETURN key and there will be a further pause as the program is creating the SMD file (see chapter 5 for more details).

However if the program crashes with the following message

Attempt to read past end-of-file

Then the SMP program has not created the relevant files (SVN, SDG). Type Y when the program issues the following message

Terminate batch job (Y/N)?

If you are within the CRISP-90 menu, choose the option to return to DOS. Then use the CLS/MODE CO80 to restore the TEXT mode and try again.

Re-run the SMP program this time making sure you save the data and quit the program from main menu (not from any other menu).

ADG

When running the ADG program the MPD file created is given the same name as the SMD file if you are creating a new ADG file ie there is no existing ADG file. However if you are editing an existing ADG file and the ADG file name is different from the SMD file name then the program creates a .MPD file with the same name as the ADG file. For example if you are using FOOTING0.SMD and FOOTINGT.ADG files then the ADG program will create FOOTINGT.MPD.

This may cause some problem if you have been using the name FOOTING0 up to this point in running the various programs (SMP, GP). Because the previous files would have used the name FOOTING0 to create the various files. For example FOOTING0.LIK (the link file).

Therefore if you want to use the newly created FOOTINGT.MPD file in your analysis copy it to FOOTING0.MPD and then run the main program using the name FOOTING0 when the program prompts for the file name.

The next update of ADG program will fix this by always creating the MPD file in the same name as the SMD file (or let the user specify the MPD file name).

Using the mouse in the icon page of ADG

In the icon page if the mouse does not work (ie the cursor does not move as you move the mouse around) then quit the icon page by pressing the F3 and typing Y and pressing RETURN key. Return to main menu and choose the option to 'save and quit'. Then look for the file MOUSE.COM in root directory of the C drive or in the DOS directory (C:\DOS). Once this file is found simply type MOUSE. This would issue a 'MOUSE installed' message. It would be a good idea to include the following line in the AUTOEXEC.BAT file (to automatically install the mouse driver).

MOUSE

Then re-run the ADG program.

In the icon page some icons are erased from the icon menu, after all the icons have been drawn. The program only erases icons that are not required. This is nothing to be concerned about. The pore-pressure icons are not displayed at in situ stage (even for a consolidation analysis). This is because pore-pressure fixities are never specified at in situ stage (even for a consolidation analysis).

If running the ADG program for the first time and if you have chosen the option 'Editing an existing ADG file' by mistake when there is no existing ADG file press the function key number 3 (F3). Quit the program either at the directory page or file selection page. Then re-run the program.

If you want to get out of a page you are stuck on, (can happen if the error checking routine 'thinks' it has detected an error in the user input) then press the Esc key to get out of that page. However the Esc keep should not be used indiscriminately because the action of the Esc key has not been thoroughly checked. Some data may be lost. Therefore should not be used routinely. Use it only if absolutely necessary.

If for some reason you want to quit the ADG program use the F3 key. This allows the data to be saved before you leave the ADG program. Recommended method of quitting the ADG program is to use

the 'save data and quit' option in the main menu.

SQ

The SQ program very rarely crashes in the absence of any data errors in the GPD file. Therefore if you find that SQ has crashed do not attempt to run the program again without looking for the reason. Look at the *.SQO file for any indication (not easy).

It usually means that a data error or an incomplete GPR file. This can happen if you have been modifying the GPD file to incorporate some changes you have made to the mesh (for example to add bar, beam elements) by hand.

> Warning : If you run SQ program again when the first run had crashed you are in danger of losing (at least partly) the contents of the GPD file. This is because when you run the SQ program it copies *.GPD to *.GPR before program execution. In case of errors the newly created GPD file will be incomplete (or even empty). Running SQ program again without checking the GPD file will result in the incomplete GPD file overwriting the previous contents of GPR. This problem will be fixed in the next update.

## A.4 Files created when running CRISP-90

A number of files other than listed in page B.4 of 'Instructions For running CRISP-90 suite of programs' are created when running the various program modules. These files are listed below with the program names. These files do not contain any useful information as far as the user is concerned.

| Program | Files | | |
|---|---|---|---|
| SMP | SMP8M.OUT | | |
| PREADG | PREADGA.OUT | PREADGB.OUT | |
| ADG | TSTADG.OUT | TSTADG.X99 | GRAPH.OUT |

MP CRS90SOL CRS90DSK

PP P990PPE

It is believed that all other files are listed in page B.4 as mentioned before. The above files are created when each of the above program is run. These were partly used for checking the programs and for debugging purposes. The users can delete these files when the program has finished running. Alternatively these files can be left alone. Every time you run a new analysis these files will be re-created and the same names are used over and over again. However the contents of these files will vary from analysis to analysis. These files can be discarded and need not be stored like the other files (GPD, MPD, ADG etc.). Future updates may suppress these files from being created. For the present use the DELUNW (command to delete unwanted files). See the file DELUNW.BAT.

As mentioned elsewhere tidy up the files relevant to a particular analysis before embarking on a new analysis. Create sub-directories and move the files listed below to these sub-directories and delete them from the CRISP90 directory. This also helps to keep all the files relevant to a particular analysis separately.

You need to store the following files if you intend to re-run that analysis at a later date. Then copy these files back into the CRISP90 directory when you want to use these files again.

(SVO) SVN (PRE) SMD ADG GPD MPD (PQD)
(SPD) (CLD)

The files enclosed in brackets should also be stored if these exist.

The results file (binary) NRS could be quite large depending on the size of the finite element mesh (ie number of elements used in the analysis) and also on the number of increments. As an approximate guide it could be in the region of 200 to 400 kBytes per increment. Therefore it is up to the individual user to decide whether he/she wants to retain it for a short period of time or delete it. Remember that these files can be re-created from GPD and MPD files in the time taken to run the GP and MP program in the

first place.

A.4.1 Deleting unwanted files created during a CRISP-90 run

The command DELUNW can be used to delete any unwanted files listed in DELUNW.BAT.

APPENDIX B

# INSTRUCTIONS FOR RUNNING THE CRISP SUITE OF PROGRAMS PC-386 COMPUTERS AND COMPATIBLES

## The CRISP Suite of Programs

| Program name | Remarks |
|---|---|
| SMP | Super-mesh program. Generates the f.e. mesh. |
| PRE-ADG | Works out position of icons to be used in ADG program. |
| SQ | Front squasher program (efficient for the frontal method of solution). |
| ADG | Generates a MP input file. |
| GP | Geometry Program. |
| MP | Main (Analysis) program. |
| ASP | Analysis assessment program. |
| PP | Post processor plotting program. |
| CL | CRISP - LOTUS 123 Interface program. Creates an ascii file which can be imported into the LOTUS 123 - spreadsheet program. |
| PQ | Pre - Stress path plotting program. Creates a file that is used by the Stress path plotting program to produce plots. |
| SP | Stress path plotting program. |

==========

```
**********************************************************
*                                                        *
*     Note : In running the various CRISP programs       *
*          it is assumed that all the data and           *
*        input files exist in the current directory      *
*                                                        *
**********************************************************
```

All files relevant to a CRISP run are given the same file name (first 8 characters). The different files are identified by the different extension name. For example if you use EXAMPLE1 as the file identifier that name is used throughout all the CRISP programs. When you run the different programs that is the name you should type when the program prompts for the 'file identifier' at the beginning. For purposes of illustration the name 'WALL1' is used in the rest of this manual.

Before running the the various CRISP program certain softwares have to be loaded into memory as indicated below:

Type **IDBOS** to load the SALFORD FTN77/386 run time system and the Metawindow libraries. (This displays the Salford compiler banner, followed by the blue Metawindow logo).

If you look at the file IDBOS.BAT it contains the following statements :

```
DBOS
METASHEL/I
COMSPACE 50000
```

Then run the CRISP-90 program modules. When you have finished running these

Type **KDBOS** to remove the Salford DBOS and the Metawindow from memory.

If you look at the file IDBOS.BAT it contains the following statements :

METASHEL/K
KILL_DBOS

The computer should respond with the Metawindow blue logo followed by the message

MetaSHEL is removed.

This should be followed by the message

DBOS removed from memory.

However if you get the following message

MetaSHEL is not TSR loaded - cannot remove....

Then type METASHEL/K at the DOS prompt, and this should give the following message.

MetaSHEL is removed.

When the suite of CRISP programs are used the files with the following extensions will be created:

| Extension | Purpose | created by | Read by | File Type |
|---|---|---|---|---|
| PRS | Prepared input data for SMP90 | user | SMP | ascii |
| SVN | (new) saved info about super mesh | SMP | SMP | binary |
| SVO | (previously) saved info about super mesh | SMP | SMP | binary |
| SMD | super-mesh dat + info about f.e. mesh | PRS-ADG | ADG | ascii |
| ADG | Created by ADG program for storing user input analysis data | ADG | ADG | ascii |
| GPD | Geometry program input data | SMP | GP | ascii |
| GPR | GP input data used as input to SQ | SMP | SQ | ascii |
| GPO | Geometry program results | GP | - | ascii |
| LIK | Link file created by geometry program | GP | MP | binary |
| MPD | Main (analysis) program input data | user | MP | ascii |
| MPO | Main (analysis) results printed o/p | MP | - | ascii |
| NRS | new stop/restart file from current analysis (used for post processing) | MP | PP, CL, PQ | binary |
| ORS | old stop/restart file read when analysis is re-started. | MP | MP | binary |
| MPS | Error messages from Main program | MP | - | ascii |
| MAS | Results assessment file | MP | ASP | ascii |
| PQD | Input data for Pre - Stress path plotting program | user | PQ | ascii |
| PQR | File created by pre - stress path plotting program. | PQ | SP | binary |
| PQO | Output from Pre-stress path plotting program (used for debugging, checking) | PQ | - | ascii |
| SPD | input data for the stress-path plotting program | user | SP | ascii |
| SPO | Output from the stress-path plotting program | SP | - | ascii |
| CLD | Input data to the CRISP - LOTUS 123 interface program | user | CL | ascii |
| CLO | Output from the CRISP - LOTUS 123 interface program | CL | - | ascii |
| PRN | Output file created by CRISP - LOTUS 123 interface program used as input to the LOTUS 123 (or other spreadsheet programs) | CL | - | ascii |

## B.4 Commands to run the CRISP-90 modules

### B.4.1 Running the Super Mesh Program - SMP

Make sure that the CAPS LOCK key is on. First of all before attempting to run the SMP program draw the super-mesh in a sheet of paper. Enter all the information in that figure : super-node numbers, super-element numbers element types, number of divisions. Enter the super-node co-ordinates and the list of super-elements associated with each super-element in another sheet of paper. Keep these 2 sheets of paper handy when running the various modules of the CRISP-90 suite of programs. See Fig. 4.3 for example. In this figure

Z - material zone number
E - CRISP element type number
D - divisions along element sides

(see also table 4.1)

There are 3 ways of entering information about the super-mesh into the SMP program,

a) Interactive input from terminal
(Typing in data as the program prompts for it).

b) Read in a previously saved file (only if you have used the program before for the current analysis).

c) Reading in a prepared data file which contains information about the super-mesh.

If you are starting a fresh analysis you can use either option 1 or 3.

If you have used the program before (say for example for a retaining wall analysis and the file is identified by WALL1. Then the program would have created a file called WALL1.SVN if you had saved the previous results). If you want to re-use this file and make some changes rename this file as the

SVO file.

REN WALL1.SVN WALL1.SVO or COPY WALL1.SVN WALL1.SVO

If you are not already in the CRISP-90 menu type CRISP90.

If you are in the CRISP-90 menu then choose the option to run the SMP and PRE-ADG programs (option 1).

1) The screen with the header "Confirm Graphics Devices" appears on the screen.

Press the RETURN key once.

2) The super mesh program banner is printed by the program.

Press any key to continue.

This is followed by the input options listed below :

ENTER 1 - FOR INPUT FROM TERMINAL
ENTER 2 - FOR INPUT FROM A (SAVED) DATA FILE (SVO)
ENTER 3 - FOR INPUT FROM A (PREPARED) DATA FILE (PRE)
ENTER 4 - QUIT THE PROGRAM

These correspond to the 3 options explained above. Choose the option which is appropriate to your case by typing a number in the range 1 to 4 and press the RETURN key.

Then the program will prompt for the file identifier with the following message :

ENTER NAME FOR THE FILE (UPTO 8 CHARACTERS). WITHOUT AN EXTENSION.

Type the filename
(for the above example ) WALL1

Then the program will ask you whether you want to create a SAVE file for subsequent runs of the program.

Type Y

The program will ask you to enter the name of the SAVE file.

Within brackets the default file name will de displayed. If you accept that name then press the RETURN key. If you want you can type in a different file name at this stage.

Then the program will ask you whether you want to create CRISP GP data file.

Type Y

The program will ask you to enter the name of the GP data file. Within brackets the default file name will de displayed. If you accept that name then press the RETURN key. If you want you can type in a different file name at this stage.

If the files of the same name already exists then the program will insist that you type a different filename to prevent overwriting. If typing a different name you have to include the extension as well. Example : WALL1B.GPD

If you make any mistakes when inputting the data do not attempt to correct it unless the program spots it and allows you to re-type the data. The mistakes can be corrected when the main menu is offered later. Choose option 2 to change the super-element and super-nodes entries. Before doing this you can check the data you had typed by choosing option 1 or by plotting the super-mesh (option 4).

When you have finished running the program do not forget to save the data before exiting. This is done in the main menu using option 6.

| | | |
|---|---|---|
| Input files | : | WALL1.SVO - previously created (saved) file |
| | | WALL1.PRE - data typed before-hand. |
| Output files | : | WALL1.SVN - currently saved data file. |
| | | WALL1.GPD - Input data file for the Geometry program. |

### B.4.2 PRE-ADG PROGRAM

The PRE-ADG program generates information about the super-mesh that is needed by the ADG program. Also it calculates the locations of icons for the super-mesh. This program is immediately after the SMP program It does not require any input data from the user.

### B.4.3 Running the ADG program

Choose the option to run the ADG program from the CRISP-90 menu. Then follow the instructions given in chapter 6.

B.4.4 Running the front squasher Program - SQ

Provided that you have run the SMP program you would have created a WALL1.GPD file. If you want to run the front squasher program (which generates the element sequence which is efficient for the frontal solver used in the main analysis program reducing the CPU time used) then choose the option to run the SQ program from the CRISP-90 menu. file as WALL1.GPR. At present all the files with the extension name GPD are copied to the extension GPR for use by the SQ program. The SQ program then creates a WALL1.GPD data file which can be input to the geometry program.

The program will display the following banner and prompt for the file identifier.

```
*************************************************
*          CRISP SQ 90 PROGRAM                  *
*    (FRONTAL  ELEMENT NUMBERING PROGRAM)       *
*************************************************

Enter File/Analysis Identifier (up to 8 characters)
```

If you had typed the name correctly as WALL1, the following messages will appear informing you of the progress of the program.

```
****** STARTING SQ 90 RUN ******

****** SQ 90 RUN FINISHED ******

The following files have been created

Printed output (Results)                    - WALL1.SQO
Input data file for the geometry program    - WALL1.GPD
```

Input files : WALL1.GPR - Input data file (same as for GP)

Output files : WALL1.SQO - Output file.
WALL1.GPD - Input data file for the Geometry program.

B.4.5 Running the Geometry Program - GP

Choose the option to run the GP program from the CRISP-90 menu. The program will display the following banner and prompt for the file identifier.

```
*********************************
*      CRISP GP 90 PROGRAM      *
*       (GEOMETRY PROGRAM)      *
*********************************

 Enter File/Analysis Identifier (up to 8 characters)
```

If you type WALL by mistake then.....

```
*** Error : File - WALL.GPD does not exist.
**** Program Terminated ****
```

This allows the name of the data file to be checked. Re-run the program. If you had typed the name correctly as WALL1, the following messages will appear informing you of the progress.

```
****** STARTING GP 90 RUN ******

****** GP 90 RUN FINISHED ******

The following files have been created

Printed output (Results)          - WALL1.GPO
Link file (binary)                - WALL1.LIK
Plot data file (not used at present) - WALL1.PLT
```

Look at the output file WALL1.GPO and ensure it had run to its completion. This can be checked if the last statement in the output lists the total degrees of freedom. For example

```
TOTAL DEGREES OF FREEDOM IN SOLUTION  =  42
```

Also check the size of the link file ( in this case WALL1.LIK) using the directory command. The size indicates whether any information has been written to the link file.

WALL1.LIK

A file with zero size would probably mean some sort of an error. Check the GP input data for any errors. Also check the output file WALL1.GPO for any errors detected by the program.

B.4.6 Running the Main (Analysis) Program - MP

Choose the option to run the MP program from the CRISP-90 menu. The program will display the following banner and prompt for the file identifier.

```
**********************************
*       CRISP MP 90 PROGRAM      *
*    MAIN (ANALYSIS) PROGRAM     *
**********************************

 Enter File/Analysis Identifier (up to 8 characters)

Type   WALL1
```

If you had typed the name correctly as WALL1, the following messages will appear informing you of the progress.

```
****** STARTING MP 90 RUN ******
```

The equilibrium check for the in situ stage is printed first.

```
-------------------- IN SITU STAGE --------------------
EQUILIBRIUM CHECK : PERCENTAGE ERRORS
X DIR =    0.135          Y DIR =    0.187
```

This is followed by the equilibrium check for each of the increment as it is completed. This allows you to monitor the progress of the analysis. If the program detects any errors it will echo the error message on the screen. Some errors are non-fatal and the program will continue execution. It is up to the user to interrupt the program (by typing CTRL/BREAK and by typing Y to the question 'Do you really want to terminate the job?'. Then correct any mistakes in the input data and re-run the program. If all is well then the program will run the full course .....

****** MP 90 RUN FINISHED ******

The following files have been created

| | |
|---|---|
| Printed output (Results) | - WALL1.MPO |
| Stop/restart file (binary) | - WALL1.NRS |
| error messages (if any) | - WALL1.MPE |
| Analysis assessment info file | - WALL1.MAS |
| Results from selected increment (binary) | - CRS90DSK |

The file WALL1.NRS (NRS stands for New ReStart file) is then used in the following programs to produce plots.

a) Post processor plotting program - PP90
b) Stress path plotting program - SP90 (via PQ90)
c) CRISP - LOTUS 123 Interface program - CL90

The program may crash due to one of the the following reasons :

(i) Due to error in the input data.
(ii) Wrong data file has been specified.

In the case of (i) consult the list of error messages listed in the CRISP brown manual volume 1 - appendix E, under the routine name issuing the error message and correct the data error and re-run. Copies of the error messages are written to the results file (WALL1.MPO) as well as the the error-message file (WALL1.MPE).

B.4.7 Running the Analysis assessment Program - ASP

This produces 2 types of plots

(i) a plot of the equilibrium error against increment number. This is a quick way of checking whether there are any numerical problems in Cam-clay analysis and soil-structure interaction analysis.

(ii) A number of plots which allows the user to assess the reliability of Cam clay analysis in terms of yield ratio parameter and the distribution of stress state codes. This is only applicable to Cam-clay analysis.

Choose the option to run the ASP program from the CRISP-90 menu. The program will display the following banner and prompt for the file identifier.

```
**************************************
*      CRISP ASP 90 PROGRAM          *
*    (ANALYSIS ASSESSMENT PROGRAM)   *
**************************************
```

Enter File/Analysis Identifier (up to 8 characters)

Type WALL1

If you had typed the name correctly as WALL1, the following message will appear .....

****** STARTING ASP 90 RUN ******

The first of the plots (% equilibrium error against increment number) is drawn on the screen. If there are significant errors check for any input errors and re-run the MP 90 program.

Pressing the RETURN key clears the screen and offers a menu for the different plot options that are available. Each plot can be viewed in turn. When you have seen all the plots choose option 10 from menu to exit the program. If you decide that the results look

acceptable then you can proceed running the post-processing programs.

5.4.8 Running the Post Progessor Plotting Program - PP

Choose the option to run the PP program from the CRISP-90 menu. Then follow the instructions given in chapter 8.

The error messages are included in the output file (WALL1.PPO).

| | | | |
|---|---|---|---|
| Input file | : | WALL1.NRS | - Stop/restart file created by MP90. |
| Output files | : | WALL1.PPO | - output file contains tables of stresses/strains for data points and also printed displays. |

B.4.9 Running the PQ (Pre - SP) Program - Can clay analysis only

This is an interface program which has to be run before the stress path plotting program. You have to specify the element and integration point numbers for which you want to plot the stress paths. Then the program creates a file containing the necessary information to plot the stress path ( WALL1.PQF ).

This program requires the following 3 files :

WALL1.NRS - Stop/restart file.
WALL1.PQD - Input data file.
WALL1.LIK - Link file created by the geometry program.

Choose the option to run the PQ program from the CRISP-90 menu or type RPQ90 to start the program. The program will display the following banner and prompt for the file identifier.

```
**********************************
*      CRISP SQ 90 PROGRAM       *
*   (PRE-STRESS PATH PROGRAM)    *
**********************************
```

Enter File/Analysis Identifier (up to 8 characters)

Type WALL1

If you had typed the name correctly as WALL1, the following messages will appear informing you of the progress.

****** STARTING PQ 90 RUN ******

****** PQ 90 RUN FINISHED ******

The following file has been created

Information necessary to plot stress paths - WALL1.PQF

B.4.10 Running the SP Program - Cam clay analysis only

This program requires the following 2 files :

WALL1.SPD - Input data file.
WALL1.PQF - File created by the PQ90 program.

Choose the option to run the SP program from the CRISP-90 menu or type type RSP90 (if at the DOS prompt) to start the program. The program will display the following banner and prompt for the file identifier.

```
****************************************
*      CRISP SP 90 PROGRAM             *
*    (STRESS PATH PLOTTING PROGRAM)    *
****************************************
```

Enter File/Analysis Identifier (up to 9 characters)

Type WALL1

If you had typed the name correctly as WALL1, the following messages will appear informing you of the progress.

****** STARTING SP 90 RUN ******

This will then display the plots in sequence. Press the RETURN key after you have finished viewing each plot. Once all the plots have been drawn the following message is printed ....

****** SP 90 RUN FINISHED ******

The following file has been created

output from stress path plotting program - WALL1.SPO
(for checking purposes)

### B.4.11 Running the CRISP - LOTUS 123 Interface Program

This is an interface program which is run to create an input data file which can be used as input to LOTUS 123 or any compatible spreadsheet programs. You have to specify the node numbers and element numbers and integration point numbers for which you want to plot selected stress and strain parameters.

This program requires the following 2 files :

WALL1.NRS - Stop/restart file.
WALL1.CLD - Input data file.

Choose the option to run the CL program from the CRISP-90 menu or type RCL90 (if at the DOS prompt) to start the program. The program will display the following banner and prompt for the file identifier.

```
*********************************************
*              CRISP CL 90 PROGRAM          *
*    (CRISP - LOTUS 123 INTERFACE PROGRAM)  *
*********************************************
```

Enter File/Analysis Identifier (up to 8 characters)

Type WALL1 .

If you had typed the name correctly as WALL1, the following messages will appear informing you of the progress.

```
****** STARTING CL 90 RUN ******


****** CL 90 RUN FINISHED ******
```

The following files have been created

Input data file for the LOTUS 123 program - WALL1.PRN
Output file from the interface program - WALL1.CLO
(for checking purposes)

B.4.12 Running the LOTUS 123 Program

Consult The LOTUS 123 manual.

The file required is WALL1.PRN

Once the LOTUS 123 program has been started choose the following options:

/ FILE IMPORT NUMBERS

Select the file WALL1.PRN and the program will read that file into the working region.

This file consists of a series of tables. One table for each of the variable you had requested. The columns represent the values of the variable at selected locations (nodes or element and integration points). This enables the user to plot distributions in space through the mesh. The rows represent the increments. This allows the user to plot variation of the selected variable at a particular location through the increments (or time in a consolidation analysis).

APPENDIX C

INSTRUCTIONS FOR RUNNING THE EXAMPLE PROBLEM - FOOTING0

SMP

Type CRISP90 which would give the menu (for running the CRISP-90 suite of programs) and choose the option to run the SMP program.

| Successive page | Action |
|---|---|
| Confirm Graphic Devices page<br>Choice ? [ ] | Press RETURN |
| Program banner | Press RETURN |
| Input-options | Choose option 3 for prepared input file |
| Enter name for File<br>(up to 8 characters) | Type FOOTING0 |
| Enter name of Prepared input data file (FOOTING0.PRE) => | Press RETURN |
| Do you want to create a save file? | Y (for yes) |
| Enter name of Save file (FOOTING0.SVN) => | Press RETURN (to accept default) |
| Do you want to create a CRISP GP data file? | Y (for yes) |
| Enter name of the CRISP GP data file (FOOTING0.GPD) => | Press RETURN (to accept default) |
| Current program limits page | Choose option 7 (to continue without changing the limits) |

The following message is printed

*** Data from Prepared file has been read successfully ***

| | |
|---|---|
| Main menu | Choose option 4 to plot super-mesh |
| Super-mesh plot | Press RETURN |
| Main menu | Choose option 3 to display entries |
| Display options menu | Choose option 2 to display super-nodes and co-ordinates |

| | |
|---|---|
| Display options menu | Choose option 1 to display super-element entries |
| Display options menu | Choose option 3 to return to Main menu |
| Main menu | Choose option 5 to generate finite element mesh |

The following messages are printed

*** Mesh generation completed ***
No. of elements in FE Mesh =
No. of nodes in FE Mesh =

| | |
|---|---|
| CRISP Data/Plot options menu | Choose option 2 to create CRISP GP data file |
| Title ? | Enter title for analysis and press the RETURN key |

*** CRISP GP data file created ***

| | |
|---|---|
| CRISP Data/Plot options menu | Choose option 1 to look at the f.e. mesh plots. |
| F.E. Mesh Plot options | Choose options 1 to 7 in turn and look at the different plots. Explore the use of I, L, R, D, U commands. W to get back to the menu. Choose option 8 to return to previous menu. |
| CRISP Data/Plot options menu | Choose option 3 to return to main menu |
| Main menu | Choose option 6 to save data. |

The following message is printed

*** Super mesh data written to save file successfully ***

| | |
|---|---|
| Main menu | Choose option 7 to quit. |

PRE-ADG PROGRAM

This program is run immediately after the SMP program.

Press RETURN when the Metawindow confirm graphic devices page is drawn. This will display the program banner. Press any key to continue. There will be a short delay at this point. The program will next draw the super-mesh and also the locations of the icons for the super-

element sides.

Press RETURN to continue. There will be a short delay as the relevant file is created.

SQ PROGRAM

Type CRISP90 which would give the menu (for running the CRISP-90 suite of programs) and choose the option to run the SQ program. When the program has finished running look at the file FOOTING0.GPD.

ADG PROGRAM

This program creates the Main (Analysis) Program data input file.

Type CRISP90 which would give the menu (for running the CRISP-90 suite of programs) and choose the option to run the ADG program.

| Successive Pages | Page No. | Action |
|---|---|---|
| Confirm Graphic Devices page | - | Press RETURN |
| Program banner | - | " " |
| SMD directory name | 1 | " " |
| SMD file name (menu) | 2 | Choose FOOTING0.SMD and press the RETURN key. |
| Edit an existing ADG file/ To create a new ADG file? (menu) | 3 | Choose "edit an existing ADG file" option and press RETURN |
| ADG directory name | 4 | press RETURN key |
| ADG file name | 5 | Choose FOOTING0.ADG and press the RETURN key. |
| A.D.G. Program Limits page | 6 | Press RETURN |
| *** reading data from SMD file *** | | |
| Geometry Details page | 7 | Press RETURN |
| *** carrying out calculations *** | | |
| Analysis Details page | 8 | Press RETURN |
| Title Section (Page 1 of 3) | 9 | Press RETURN |
| Title Section (Page 2 of 3) | 10 | Press RETURN |
| Printed output control parameters (page 3) | 11 | Press RETURN |
| Main menu | 12 | Choose option 2 for material properties |
| Material zone menu | 13<br>14, 15 | Choose each mat zone in turn and press RETURN at each of the 2 pages displayed (Mat Type page and props page). Then return to main menu. |
| Main menu | 12 | Choose in situ stress specs option (3) |
| In situ stresses menu | 16 | Choose in situ stress specs option (1) |

| | | |
|---|---|---|
| In situ stress specs menu | 17 | Choose in situ stresses specs at reference points option (3) |
| In situ stress at Ref Points | 20 | Press RETURN |
| Input of in situ stesses at Reference points | 21 | Press RETURN |
| menu (In situ stresses at reference points - page 3) | 22 | Choose option to plot in situ stress profile (1) |
| Plot of in situ stresses | - | Press RETURN |
| In situ stress specs menu | 17 | Choose option to return to In situ stresses menu (4) |
| In situ stresses menu | 16 | Choose in situ displacement fixities and loading option (2) |
| Icon Page | - | Choose the erase icon and erase the previously specified fixities icons. Then choose the fixities icon and try to re-assign them. Press F3 to quit this page. Then type Y and press the RETURN key. |
| In situ stresses menu | 16 | Choose option to return to main menu (3) |
| Main menu | 12 | Choose analysis block option (4) |
| Analysis Block (level 1) menu | 25 | Choose increment block 1 |
| Analysis Block Level 2 menu | 26 | Choose input control parameters option |
| Control parameters | 27 | Press RETURN key |
| Analysis Block Level 3 menu | 28 | Choose output option (1). |
| Output options | 29 | Choose the standard output option. |
| Output options | 30 | Press RETURN. |
| Analysis Block Level 3 menu | 28 | Choose increment factor option. |
| Increment factor option | 32 | Choose equal incr factor option. |
| Increment factor option | 33 | Press RETURN. |
| Analysis Block Level 3 menu | 28 | Choose option to return to level 2 menu (3). |
| Analysis Block Level 2 menu | 26 | Choose displacement fixities/loading option (2) |
| Icon Page | - | Choose display icon and then click |

| | | |
|---|---|---|
| | | the cursor on the normal stress icon displayed on the super-mesh. Enter 100 at the field for first node and then use the down arrow key to move to the next field and enter 100. Press the RETURN key to close the magnitude window. Press F3 to quit the page. Type Y and press the RETURN key to confirm. |
| Analysis Block level 2 menu | 26 | Choose option to return to level 1 menu (5) |
| Analysis Block Level 1 menu | 25 | Choose option to return to main menu |
| main menu | 12 | Choose option to generate CRISP MP data file (5) |
| Main menu | 12 | Choose option to save data and quit |
| To create a new ADG file/ Overwrite existing ADG file | 38 | Choose option to create a new ADG file (1) |
| ADG file directory | 39 | Press the RETURN key |
| File section : ADG file name | 40 | Enter FOOTINGT and press the RETURN key |
| *** saving data in ADG file *** | | |
| Quit Program - menu | 42 | Choose quit the Program option |

GEOMETRY PROGRAM (GP)

Type CRISP90 which would give the menu (for running the CRISP-90 suite of programs) and choose the option to run the Geometry program.

Type FOOTING0 when the program prompts for the file name.

MAIN (ANALYSIS) PROGRAM (MP)

Type CRISP90 which would give the menu (for running the CRISP-90 suite of programs) and choose the option to run the Main (analysis) program.

Type FOOTING0 when the program prompts for the file name.

PLOTTING PROGRAM (PP)

Type CRISP90 which would give the menu (for running the CRISP-90 suite of programs) and choose the option to run the Plotting program.

Type FOOTING0 when the program prompts for the file name.

| Successive Pages | Page No. | Action |
|---|---|---|
| Confirm Graphic Devices page | - | Press RETURN |
| Program banner | - | " " |
| NRS directory name | 1 | " " |
| NRS file name (menu) | 2 | Choose FOOTING0.NRS and press the RETURN key. |
| Confirm file/directory selection or change | 3 | Confirm current file selection (option 1). |
| Title section | 4 | Type FOOTING EXAMPLE over 'CRISP ANALYSIS' |
| Analysis type | 5 | Choose axisymmetric option |
| Main menu | 6 | select cumulative mode |
| Choose increment No. for post processing | 7 | Type 22 (right justified) and press the RETURN key. |
| Reading NRS file | 8 | No action |
| Choice of plots | 10 | Choose to view deformed mesh |
| Displacement magnification for deformed mesh plot | 11 | Type 4.0 over 1.0 and press the RETURN key. |
| Selection of mesh plot | 12 | Choose option to view both (3) |
| Mesh plot page | - | See section 4.7 for commands for windowing options. Type W to return to the menu. |
| Selection of mesh plot | 12 | Choose option to change scale for displacement (option 4) |
| Displacement magnification for deformed mesh plot | 11 | Type a different value over 4.0 and press the RETURN key. |
| Selection of mesh plot | 12 | Choose option to view both plots (3) |
| Mesh plot page | - | See section 4.7 for commands for windowing options. |

| | | |
|---|---|---|
| | | Type W to return to the menu. |
| Selection of mesh plot | 12 | Choose option to view other plots (option 5) |
| Menu : Other plots and options | 13 | Choose 'Menu of available plots' |
| Calculation in progress | 15 | No action |
| Menu : All other plots | 17 | Choose 1 to 7 in turn and look at each of the plots. |
| Plot page | | See section 4.7 for commands for windowing options. Type W to return to the menu. |
| Menu : All other plots | 17 | Choose 'Return to previous menu' (option 8). |
| Menu : Other plots and options | 13 | Choose the option 2 to select data point |
| Data point selection | 15 | Choose 'All integration points' (option 2) |
| Menu : Other plots and options | 13 | Choose 'Change scale factor' option |
| Scale factors used in the plots | 16 | Type in a scale factor of 10.0 in the second field. |
| Menu : Other plots and options | 13 | Choose Option 1 - Menu of available plots |
| Calculation in progress | 15 | No action |
| Menu : All other plots | 17 | Choose option 2 to view displacement vectors. |
| Plot page | | See section 4.7 for commands for windowing options. Type W to return to the menu. |
| Menu : All other plots | 17 | Choose option to quit. (last option). |
| Quit Program - menu | 18 | Choose option to quit program (2). |

APPENDIX D

# USING BEAM, BAR AND SLIP ELEMENTS WITH SMP/PRE-ADG/ADG PROGRAMS

## D.1 Modifying GPD and MPD files

This note explains how beam, bar and slip elements can be included by modifying the GPD and MPD files created by the SMP and ADG program respectively.

If you want to use the bar, beam and slip elements in your analysis then you should run only the SMP, PRE-ADG and ADG programs and create the GPD and MPD files. But do not run the SQ, GP and MP programs. Then modify the GPD and MPD files as illustrated in the example below and then you can run the SQ, GP and MP programs.

The example chosen to illustrate this is an analysis of a retaining wall problem (see Fig D.1). Fig. D.1(b) shows the situation after excavation has been carried out and Fig. D.1(c) after the prop (modelled using a bar element) has been installed.

Figure D.2 shows the finite element mesh generated by the SMP program. For simplicity only a few elements are used. In general there will be lot more elements in a typical f.e. mesh.

The procedure to be followed is to draw the super-mesh and run the SMP program and PRE-ADG program without taking into account any bar, beam or slip elements. For the above example let us assume that we want to use slip elements between the wall and the soil. Figure D.2 shows the f.e. mesh generated without the slip and bar elements.

Run the ADG program to create the MPD file. Specify the appropriate information necessary to generate the MPD file. For example the material properties for the soil and the wall and the displacement boundary conditions. Also specify the in situ stresses and remove the super-element which is modelled by finite element 7 to simulate the excavation process. In using the ADG program only 1 increment block with a single increment is used. However when modifying the MPD file a second increment block is added.

For simplicity the analysis consists of 2 increment block. In increment block 1 the element 7 is removed. In increment block 2 the prop is installed. For reasons of simplicity both increment blocks contains a single increment.

At this stage both GPD and MPD files would have been created by running the SMP, PRE-ADG and ADG programs. However the GP or the MP programs are not run.

The contents of the GPD file is listed below :

```
SLIP EDIT Titulo
150609
  16    9    4    4    2    0
  0     0
  0     0    0    0    0    0    0    0    0    0
  0     0    0    0
    1        0.0        5.0
    2       20.0        5.0
    3        0.0        0.0
    4       20.0        0.0
    5       21.0        5.0
    6       21.0        0.0
    7        0.0       10.0
    8       20.0       10.0
    9       40.0        5.0
   10       40.0        0.0
   11       21.0       10.0
   12       40.0       10.0
   13        0.0       15.0
   14       20.0       15.0
   15       21.0       15.0
   16       40.0       15.0
  0
  1    4    1    3    4    2    1
  2    4    1    4    6    5    2
  3    4    1    1    2    8    7
  4    4    1    6   10    9    5
  5    4    2    2    5   11    8
  6    4    1    5    9   12   11
  7    4    1    7    8   14   13
  8    4    2    8   11   15   14
  9    4    1   11   12   16   15
```

If you are not familiar with the slip elements read appendix K of the CRISP Manual (volume 1) for how to use the slip elements.

Now the finite element mesh is modified to include the slip elements as shown in Figures 3 and 4. Figure 4 shows the detail around the wall. As can be seen all the changes are confined to the region defined by nodes 14 - 8 - 2 - 5 - 11 - 15 - 14. This way one knows that neither the node numbers nor the element-nodal connectivity has been changed outside this region. Care is needed in making such a local change. Ensure that the new elements and nodes are given numbers higher than the ones already used. There should not be any duplication of node or element numbers.

The changes are made by introducing 7 new nodes. Nodes 17 to 23. Node 23 is not shown in the above figure. The new nodes 17 to 22 can be given the same co-ordinates as the nodes 2, 5, 8, 11, 14, 15 respectively. The following entries should be added after the entry for the nodal co-ordinates for node 16 in the GPD file.

| Node No. | X Co-ord | Y Co-ord |
|---|---|---|
| 17 | 20. | 5. |
| 18 | 21. | 5. |
| 19 | 20. | 10. |
| 20 | 21. | 10. |
| 21 | 20. | 15. |
| 22 | 21. | 15. |
| 23 | 19. | 15. |

Similarly there are 6 new elements. 5 slip elements between the wall and the soil and a bar element modelling the prop. The interface is represented by slip elements 10 to 14.

The bar element simulating the prop is element 15 which connects node 23 to node 21. The prop is assumed to be of length 1 m.

The following entries should be added to the end of the GBD data file

| Element | Type | Mat | Zone | Node1 | Node2 | Node3 | Node4 |
|---|---|---|---|---|---|---|---|
| 10 | 13 | 3 | | 14 | 8 | 19 | 21 |
| 11 | 13 | 3 | | 8 | 2 | 17 | 19 |
| 12 | 13 | 3 | | 2 | 5 | 18 | 17 |
| 13 | 13 | 3 | | 5 | 11 | 20 | 18 |
| 14 | 13 | 3 | | 11 | 15 | 22 | 20 |
| 15 | 1 | 4 | | 23 | 21 | 0 | 0 |

Notice the 2 zeroes added to the node list for element 15. Because 4 node numbers are required for each entry.

In addition you will notice that because of the introduction of the interface layer the nodes associated with elements 5 and 8 have changed. However the nodes associated with elements 7, 3, 2, 6, 9 remain unchanged. This is because the new nodes 17 to 22 were introduced so that they were all on the side of the slip element attached to the wall. Care is needed how the new nodes and elements are introduced. It is a good idea to sketch this part of the mesh as shown in Figure D.4 before editing the GPD file. The new nodal connectivity for the wall elements 5 and 8 are as follows :

| Element | Type | Mat | Zone | Node1 | Node2 | Node3 | Node4 |
|---|---|---|---|---|---|---|---|
| 5 | 4 | 2 | | 17 | 18 | 20 | 19 |
| 8 | 4 | 2 | | 19 | 20 | 22 | 21 |

Now the total number of nodes are 23, and the elements 15. Make this change in Record C (line 3).

Incorporating all the above changes, the edited (new) GPD file is given below.

```
A   SLIP EDIT - AFTER ADDING SLIP AND BAR ELEMENTS
B   150409
C   23   15    4    4    2    0
D   0    0
E   0    0    0    0    0    0    0    0    0    0
F   0    0    0    0
    1        0.0        5.0
    2       20.0        5.0
```

```
    3       0.0       0.0
    4      20.0       0.0
    5      21.0       5.0
    6      21.0       0.0
    7       0.0      10.0
    8      20.0      10.0
    9      40.0       5.0
   10      40.0       0.0
   11      21.0      10.0
   12      40.0      10.0
   13       0.0      15.0
   14      20.0      15.0
   15      21.0      15.0
   16      40.0      15.0
              17              20.        5.
              18              21.        5.
              19              20.       10.
              20              21.       10.
              21              20.       15.
              22              21.       15.
              23              19.       15.
#   0
    1   4     1     3     4     2     1
    2   4     1     4     6     5     2
    3   4     1     1     2     8     7
    4   4     1     6    10     9     5
                5           4           2.          17    18    20    19
    6   4     1     5     9    12    11
    7   4     1     7     8    14    13
                8           4           2           19    20    22    21
    9   4     1    11    12    16    15
               10          13           3          14     8    19    21
               11          13           3           8     2    17    19
               12          13           3           2     5    18    17
               13          13           3           5    11    20    18
               14          13           3          11    15    22    20
               15           1           4          23    21     0     0
```

Interface elements are assigned material zone number 3.
Bar element is assigned material zone number 4.

It is not necessary to assign identical material properties to all

the slip elements. For example one could have used one set of properties for slip elements 10 and 14 (say material zone 3) and a different set for elements 11, 12 and 13 (say material zone 5).

After making these changes if you had used the same names for the files as before then you can run the SQ, GP and MP program from the CRISP90 menu. If necessary first run the SQ program.

Then run the GP program. From the output file GPO find the mid-side node number for the bar-element (no. 15). This is needed for modifying the MPD file as shown below.

Changes to MPD file:

You will notice that the geometry changes due to the introduction of the slip and bar elements does not change the elements or nodes numbers of the 2 vertical boundaries and the bottom of the mesh. Therefore no changes to displacement fixities as generated by the ADG program need to be made.

As mentioned before all the changes are confined to the region defined by nodes 14 - 8 - 2 - 5 - 11 - 15 - 14. The changes made to the mesh does not affect the part of the f.e. mesh outside this region. This is the reason why we can use the rest of the data (example : displacement fixities of the f.e. mesh boundary) generated by the ADG program. The changes made does not affect the outer boundary of the mesh. If the changes made did affect the outer boundary then the user should modify any displacement fixities (records H3 and M in the input data) generated by the ADG program accordingly.

The following changes need to be made to the MPD file. In reading the following sections it will be useful to refer to the CRISP manual (vol. 1) section 3.

Record C1

In the super-mesh used for running the SMP program there were only 2 material zones (soil and wall). Now the introduction of the slip elements and the bar elements means now that there are a total of 4 material zones. NMAT has to be increased from 2 to 4 in record C1. Since the prop (modelled by element 15) is not present at the

beginning of the analysis, remove it. IPRIM = 1.

Record D

Include the material properties for the slip and bar elements using records D (see the CRISP Manual - section 3) after the material properties for the soil and the wall.

Since IPRIM = 1 means record E which lists the elements to be removed at the beginning of the analysis has to be included in the data.

Record E

Include a line containing the element number 15.

Record I (for increment block 1)

Simulation of the excavation : Element 7 would have been removed to simulate the excavation. ICHEL = 1 (total number of elements to be removed). Remove the slip element 10 along with element 7. Therefore now

ICHEL = 2 in record I.

Add the element 10 to the list of elements to be removed in record J.

Record I (for increment block 2)

In the second increment block you need to add the bar element and at the same time restrain the free end of the bar element (node 23). Since the bar element has a mid-side node it is necessary to restrain this node as well. This is done using record N.

Record J (add the prop)

15

Record N

The mid-side node number from GPO (output) file is 786. It is necessary to restrain both nodes (node 23 and 786) in both X and Y directions. Otherwise it may result in a mechanism (CRISP MP will stop with a ZERO PIVOT error message). Also set NFXB = 4 in record I for increment block 2 for the 4 nodal fixities.

| node | IVAR | IFXC | VAL |
|---|---|---|---|
| 23 | 1 | 1 | 0 |
| 23 | 2 | 1 | 0 |
| 786 | 1 | 1 | 0 |
| 786 | 2 | 1 | 0 |

The MPD file created by the ADG program is listed in TABLE D.1. The edited MPD file is listed TABLE D.2.

D.2 Making local changes to the mesh

In Fig. D.4 the choice was made to retain the same node numbers as generated by the SMP program for the soil-side of the soil-wall interface. The reason for this choice is that the nodal-connectivity of the soil elements 7, 3, 1, 4, 6, 9 don't have not to be changed. Whereas only the nodal connectivity of the wall elements 8, 5 had to be changed ie the number of changes is less.

However there can be circumstances where you may want to retain the same node numbers on the wall side of the interface. Then introduce the new node numbers on the soil side of it. This is illustrated in Fig. D.5. This could be a different example where the wall may be replaced by a footing which is subjected to pressure. In this case the ADG program would have generated data records using the node numbers assigned by the SMP program. Changing the node numbers on footing side of the interface would require modifying the data records which represent the pressure on the footing.

Both the above mentioned choices are acceptable. The main point is to choose whatever is convenient, which avoids the introduction of data errors.

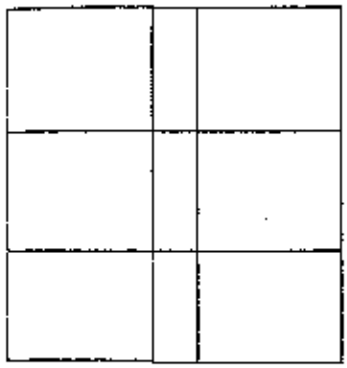


Fig. D.1(a)

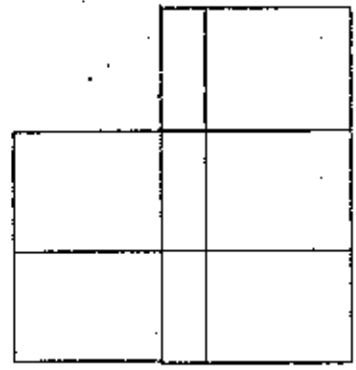


Fig. D.1(b)

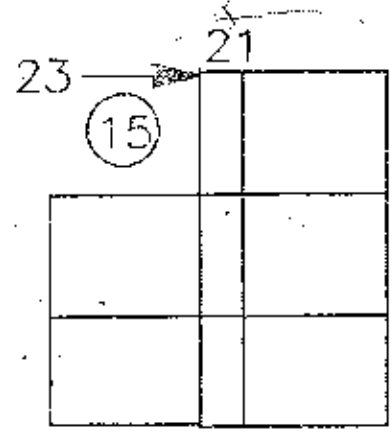



Fig. D.1(c)

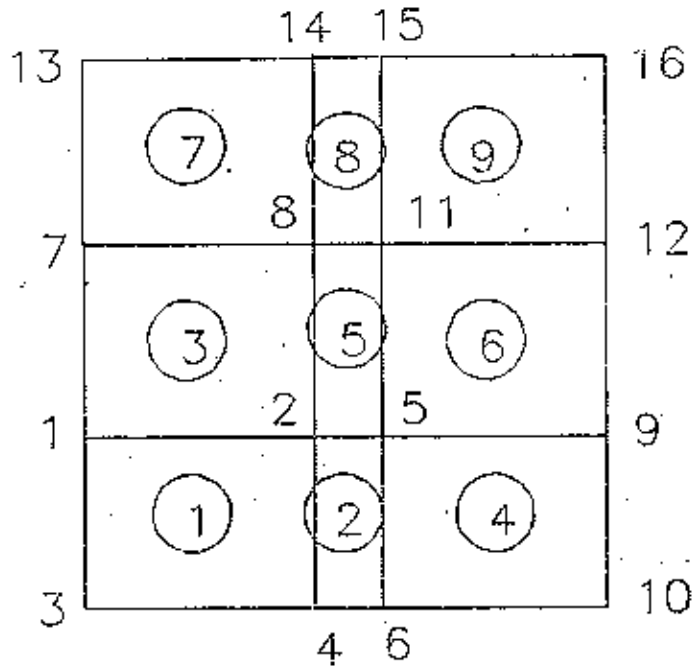



Fig. D.2

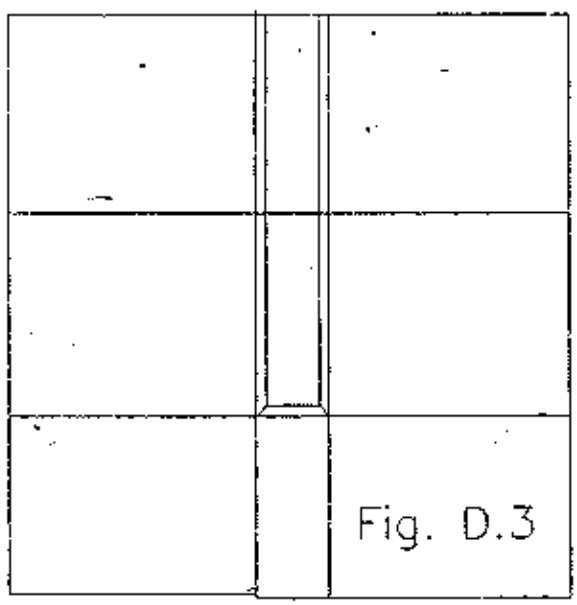


Fig. D.3

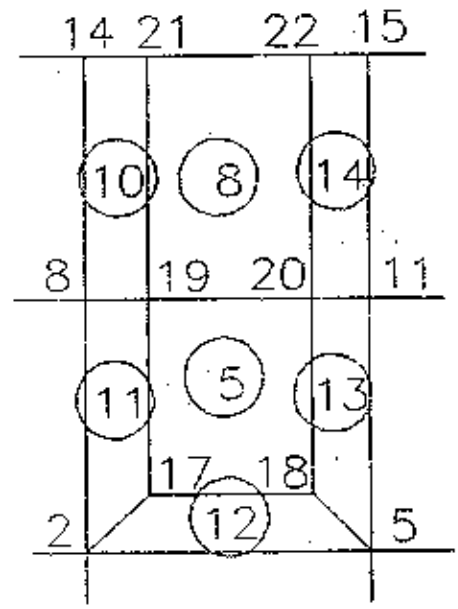



Fig. D.4

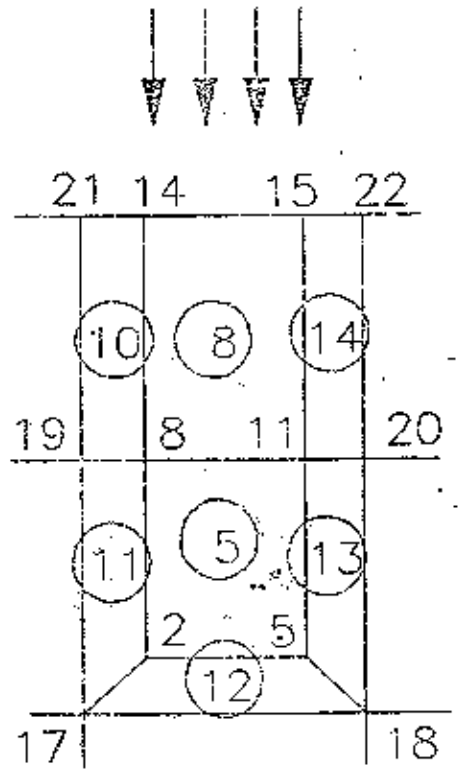



Figure D.5 Alternative scheme
(adding slip elements)

```
 CRISP ANALYSIS
C--------RECORD B : LINK NUMBER
     150409
C--------RECORD C1 : ANALYSIS CONTROL PARAMETERS
C---NPLAX     NMAT     NOIB     INC1     INC2     IPRIM    IUPD     ICOR     ISR
        0        2        1        1        1        0        0        0        2
C--------RECORD C2 - OUTPUT CONTROL PARAMETERS
C--    INSOP    IBC     IRAC     NVOS     NVOF     NMOS     NMOF     NELOS    NELOF
        1        0        1       -1       16        0        0        1        9
C--------- RECORD C3 - NO. OF INCS WRITTEN FOR PP
        0
C--------RECORD D :  MATERIAL PROPERTIES
C--      E1       E2       v1   v2     G        -    gw/Kw  Bulk     PERMX    PERMY
 1 1 0.10E+05 0.10E+05 0.25 0.25 0.40E+04 0. 0.00E+00 20.0    0.      0.     0 0
C--      E1       E2       v1   v2     G        -    gw/Kw  Bulk     PERMX    PERMY
 2 1 0.20E+06 0.20E+06 0.25 0.25 0.80E+05 0. 0.00E+00 20.0    0.      0.     0 0
C--------RECORD F : IN SITU STRESS OPTION
C     INSIT    NNI
        1        2
C--REP PT    Y-CO      SIG-X'     SIG-Y'     SIG-Z'      TXY    PORE PR  VOID    PC
      1       15.00      0.000      0.000      0.000       0.      0.00   0.    0.00
      2        0.00     75.000    150.000     75.000       0.    150.00   0.    0.00
C--------RECORD H1 : IN SITU DETAILS
C------- NDI     NFXI      GRAVI
        0       18       1.00
C---------IN SITU DISPLACEMENT FIXITIES - RECORD H3
C-- ELEM ND1   ND2 VAR    FIXC      VL1        VL2        VL3
      1    3     4    1      1      0.000      0.000      0.000
      1    3     4    2      1      0.000      0.000      0.000
      2    4     6    1      1      0.000      0.000      0.000
      2    4     6    2      1      0.000      0.000      0.000
      4    6    10    1      1      0.000      0.000      0.000
      4    6    10    2      1      0.000      0.000      0.000
      4   10     9    1      1      0.000      0.000      0.000
      4   10     9    2      1      0.000      0.000      0.000
      6    9    12    1      1      0.000      0.000      0.000
      6    9    12    2      1      0.000      0.000      0.000
      9   12    16    1      1      0.000      0.000      0.000
      9   12    16    2      1      0.000      0.000      0.000
      7   13     7    1      1      0.000      0.000      0.000
      7   13     7    2      1      0.000      0.000      0.000
      3    7     1    1      1      0.000      0.000      0.000
      3    7     1    2      1      0.000      0.000      0.000
      1    1     3    1             0.000      0.000      0.000
      1    1     3    2             0.000      0.000      0.000
C--------RECORD I : INCREMENT BLOCK NUMBER =     1
C-- IBNO INCA INCB ICHEL NLOD ILDF NFIX NFXB  IOUT IOCD    DTIME     ITMF DGRAV
      1    1    1     1    0    0    0    0 11111    0   0.000E+00    0    0.0
C--------RECORD J :
C--------LIST OF ELEMENTS ADDED/REMOVED
       7
```

Table D.1 MPD file (before adding slip elements)

```
 DATA EDITTED TO ADD SLIP AND BAR ELEMENTS
C---------RECORD B : LINK NUMBER
     150409
C---------RECORD C1 : ANALYSIS CONTROL PARAMETERS
C---NPLAX     NMAT     NOIB     INC1     INC2     IPRIM     IUPD     ICGR     ISR
        0        4        2        1        2         1        0        1       2
C---------RECORD C2 - OUTPUT CONTROL PARAMETERS
C--    INSOP    IBC     IRAC     NVOS     NVOF     NMOS     NMOF     NELOS   NELOF
         1        0        1        1       16        0        0        1       9
C---------- RECORD C3 - NO. OF INCS WRITTEN FOR PP
         0
C---------RECORD D :  MATERIAL PROPERTIES
C--      E1       E2       v1   v2      G        -  gw/Kw   Bulk     PERMX    PERMY
 1 1 0.10E+05 0.10E+05 0.25 0.25 0.40E+04 0.  0.00E+00 20.0 0.00E+00 0.00E+00 0
C--      E1       E2       v1   v2      G        -  gw/Kw   Bulk     PERMX    PERMY
 2 1 0.20E+06 0.20E+06 0.25 0.25 0.80E+05 0.  0.00E+00 20.0 0.00E+00 0.00E+00 0
C--      C       PHI      kn      ks     ksres     -        -        -        -    -
 3 8     1.      25.    1.2E4   0.4E4   0.4E2  0.001     0        0        0    0
C--      E       v        A       -       -        -        -        -        0    0
 4 8   1.E6     0.2      1.      0.      0.       0.       0.       0.       0    0
C---------RECORD E : LIST OF ELEMENTS REMOVED TO FORM PRIMARY MESH
C--       BAR ELEMENT MODELLING THE PROP
         15
C---------RECORD F : IN SITU STRESS OPTION
C     INSIT    NNI
          1       2
C--REF PT    Y-CO       SIG-X      SIG-Y      SIG-Z      TXY      PORE PR   VOID    PC
       1      15.00      0.000      0.000      0.000      0.       0.00      0.      0.
       2       0.00     75.000    150.000     75.000      0.     150.00      0.      0.
C---------RECORD H1 : IN SITU DETAILS
C-------- NDI      NFXI       GRAVI
          0        18         1.00
C----------IN SITU DISPLACEMENT FIXITIES - RECORD H3
C--- ELEM NDI   ND2  VAR   FIXC      VL1        VL2        VL3
        1    3    4    1     1     0.000      0.000      0.000
        1    3    4    2     1     0.000      0.000      0.000
        2    4    6    1     1     0.000      0.000      0.000
        2    4    6    2     1     0.000      0.000      0.000
        4    6   10    1     1     0.000      0.000      0.000
        4    6   10    2     1     0.000      0.000      0.000
        4   10    9    1     1     0.000      0.000      0.000
        4   10    9    2     1     0.000      0.000      0.000
        6    9   12    1     1     0.000      0.000      0.000
        6    9   12    2     1     0.000      0.000      0.000
        9   12   16    1     1     0.000      0.000      0.000
        9   12   16    2     1     0.000      0.000      0.000
        7   13    7    1     1     0.000      0.000      0.000
        7   13    7    2     1     0.000      0.000      0.000
        3    7    1    1     1     0.000      0.000      0.000
        3    7    1    2     1     0.000      0.000      0.000
        1    1    3    1     1     0.000      0.000      0.000
        1    1    3    2     1     0.000      0.000      0.000
```

(continued)

```
C--------RECORD I : INCREMENT BLOCK NUMBER =     1
C-- IBNO INCA INCB ICHEL NLOD ILDF NFIX NFXB  IOUT IOCD    DTIME    ITMF DGRAV
     1    1    2    0    0    0    0  11121     0    0.000E+00     0    0.0
C--------RECORD J :
C--------LIST OF ELEMENTS ADDED/REMOVED
       7    10
C--------RECORD I : INCREMENT BLOCK NUMBER =     2
C-- IBNO INCA INCB ICHEL NLOD ILDF NFIX NFXB  IOUT IOCD    DTIME    ITMF DGRAV
     2    2    2    0    0    0    4  11121     0    0.000E+00     0    0.0
C--------RECORD J :
C--------LIST OF ELEMENTS ADDED
       15
C--------RECORD N : NODAL FIXITIES
C--    TO FIX ONE END AND THE MIDPOINT OF THE BAR ELEMENT
C--NODE     IVAR   IFXC   VAL
      23      1      1     0
      23      2      1     0
     786      1      1     0
     786      2      1     0
```

Table D.2 MPD file (after adding slip elements)