CRISP2D

GEOTECHNICAL FINITE ELEMENT ANALYSIS SOFTWARE

EXAMPLES

Prepared by

The CRISP Consortium Limited

www.mycrisp.com

Contents

INTRODUCTION 　1

1.1　How To Use This Guide

This volume is designed to introduce you to progressively more advanced features of CRISP2D by way of a series of worked examples. Each example comprises a brief description of each of these stages followed by step-by-step instructions as to how to complete every stage.

Each worked example introduces either new features or different ways of using existing features. By the end of the last worked example you will have a good idea as to the use of every feature in CRISP2D.

It is assumed that you have a good knowledge of soil mechanics, that you have used Windows before and that you know the standard terminology used to describe Windows artefacts.

Please Note: only examples 1 - 3 can be carried out using the evaluation version of CRISP2D and that some of the features shown in the manual have been disabled.

1.1.1　Examples

The examples are intended to provide a source of guidance for the use of CRISP2D interface. It is important to note that the examples are limited in size to allow the completion of the example in an acceptable time. For more complex problems you should refer to the publications directory supplied with CRISP2D. Five examples are included that cover simple lab tests to major engineering activities.

- Triaxial Test

- Oedometer Test

- Analysis of a strip footing

- Construction of an embankment, and

- Construction of a retaining wall

1.2　Using The Manuals

Volume I contains detailed descriptions on how to use every feature in the interface to help you create meshes, apply analysis conditions, run the analysis and post process the output. It also contains a chapter that introduces you to Windows and the terminology you need to know to make the most efficient use of these manuals.

Volume II contains a wealth of information detailing how you should go about setting up a finite element analysis (FEA) including selecting the correct type of finite element, choosing a soil model, deriving material parameters and setting in situ conditions to name but a few sections.

The Publications Directory is provided as a source of reference information so that you can see the different ways in which CRISP has been used to model particular problems in the past.

ONE DIMENSIONAL CONSOLIDATION TEST

2

2.1   Introduction

The subject of this example is a one-dimensional Terzaghi consolidation.

The estimated time is 30 - 45 minutes.

The following steps will be covered:

- Starting The Project

- Defining Units

- Creating The Finite Element Mesh

- Defining A Material Zone

- Assigning Element Properties

- Defining In Situ Stress Conditions

- Defining Increment Blocks

- Defining Boundary Conditions

- Defining Loads

- Scale Factors

- Running The Analysis

- Displacement Plots

- Duration Graphs

2.2   The Problem

The consolidation problem under consideration is illustrated in Figure 2-1. The finite element mesh that will be used consists of 2 linear strain triangle (LST) elements.
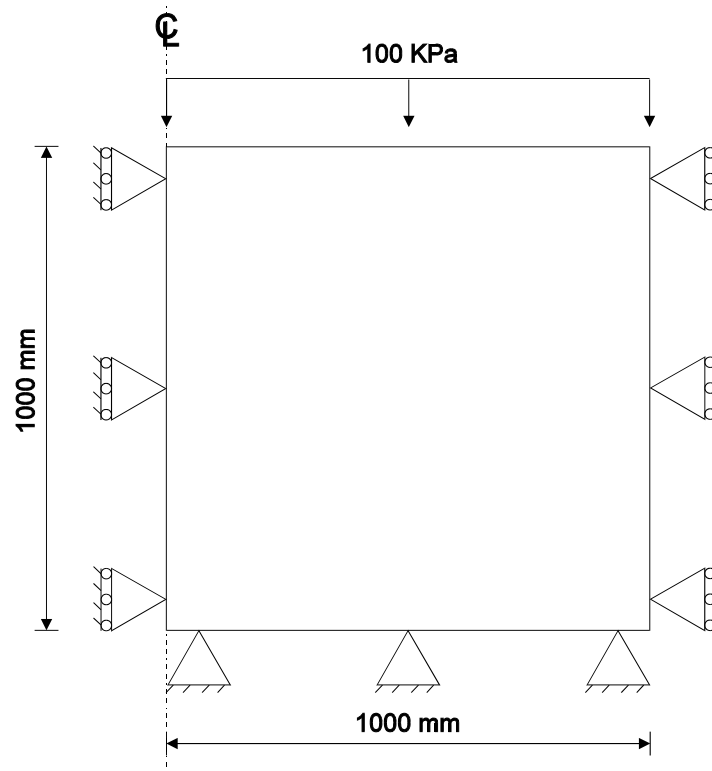
Figure 2-1:One Dimensional Terzaghi Consolidation

The one-dimensional consolidation of an axisymmetric sample is chosen to illustrate the use of consolidation elements. Two linear strain triangles are used in the mesh.

Figure 2-1 shows the mesh and the boundary conditions. The vertical sides and bottom of the sample are restrained and they are assumed to be smooth. The sample is initially unstressed, i.e. in situ stresses are set to zero. The sample has a radius of 1000 mm and is 1000 mm in height. The top surface is loaded with 100 kPa instantaneously. For computational purposes this load is applied over a small time period of 0.1 sec. The transient response during 20 secs after the load application is studied in the analysis. In the first increment, the drainage conditions are applied. The load is applied in the second increment. The third increment block consists of increments 3 to 9, whose time steps are 1, 1, 1, 1, 1, 5 and 10 secs respectively.

It   is recommended in consolidation analyses involving applied loads that the pore water pressure boundary conditions be specified in a separate increment block to the loads. In this example, the total pore water pressure is set to zero along the top of the sample in the first increment block and the load is applied in the second increment block.

2.3   Pre-Processing

This section describes all of the stages that must be completed using the Pre-Processor in order to run the finite element analysis. It is recommended that you save the Project regularly (by selecting Save Project from the File menu).

2.3.1   Starting The Project

This section describes the basic processes that should be completed in advance of creating your mesh. This involves entering data to set-up various run-time options for the finite element analysis and to descriptively identify the Project. The base units are also verified.

The following procedure describes the initial setup of the Project.

On opening the CRISP2D Pre-Processor, select New Project... from the File menu. This displays the Mesh Mode dialogue box.

Click on the Finite Mesh button. This exits the Mesh Mode dialogue box, sets you up in Finite Element mode and opens the Project Setup dialogue box.
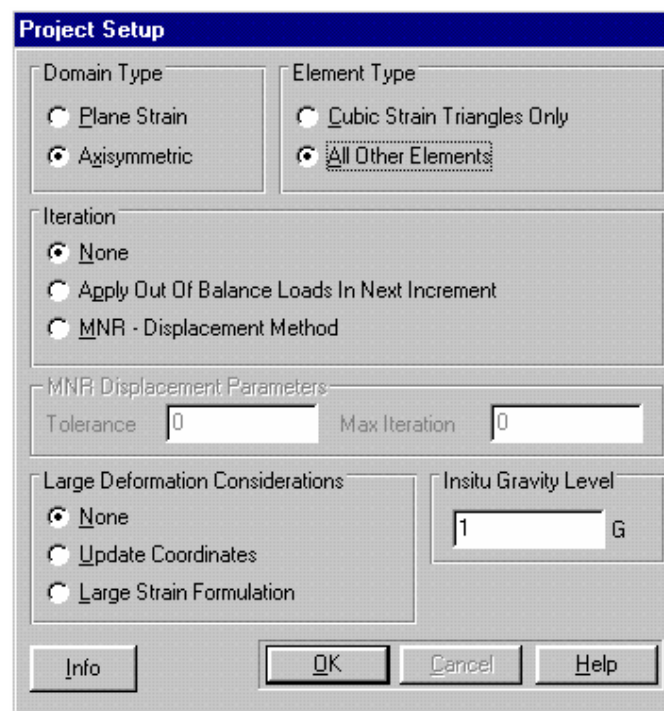
Figure 2-2:Project Setup dialogue box

Click on the Axisymmetric option button and on All Other Elements to select the analysis type and permissible element types, respectively.

The Project Setup should now look like Figure 2-2.

Click on the Info... button. This displays the Project Information dialogue box.
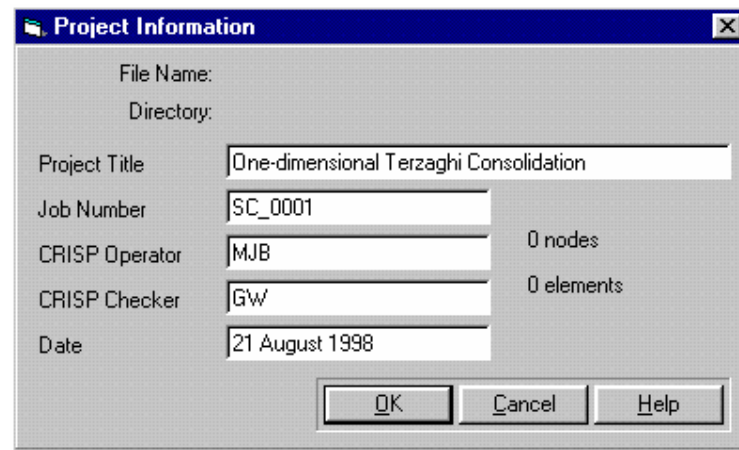
Figure 2-3:Project Information dialogue box

Enter data as appropriate. The date is automatically entered but may be manually altered.

The Project Information dialogue box should now look similar to Figure 2-3. Click on OK to return to the Project Setup dialogue box once you have finished.

Click on OK to exit.

2.3.2   Defining Units

At this point it is useful to ensure that the correct system of base units is being used. For this example, kilonewtons, millimetres, seconds and degrees form the base units.

Step 1   In the Options menu, click on Units... to display the Units dialogue box.

Figure 2-4:Defining Base Units

Step 2   Use the list boxes to select the appropriate units for Distance, Force, Time and Angle as shown in Figure 2-4. Click on OK to exit once you have finished.

2.3.3   Creating The Finite Element Mesh

The Project is now ready for you to define your finite element mesh. The quickest, most accurate way to create a small, regular mesh is to use the Node List to define the vertices and then join them together to form the finite elements.

Step 1	In the Mesh menu, click on Node List... to display the Node List dialogue box.
Step 2	Click on the empty x co-ordinate cell and type '0'. Press the TAB key to move to the y co-ordinate cell and enter '0' here.
Step 3	Press the TAB key again. This moves you into the x co-ordinate cell for the next node. Enter an x value of '1000' and a y value of '0'.
Step 4	Using the TAB key to move between cells, enter a total of four nodes with co-ordinates (0 , 0), (1000 , 0), (1000 , 1000) and (0 , 1000), as shown in Figure 2-5.
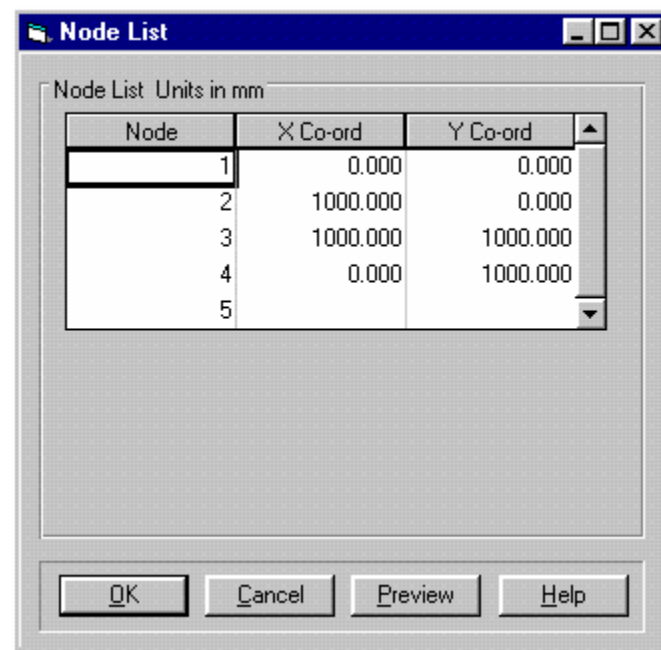
Figure 2-5:Node List dialogue box

Step 5	Click on OK to exit once you have finished.
Step 6	The Graphics Window will now display the nodes you have created. To view them at optimum scale select Full Page from the Zoom list box on the main toolbar.
Step 7	In the View menu, click on Node Numbers to switch on the display of node numbers.
Step 8	In the Mesh menu, click on Create Element. Using the left-hand mouse button click on (or close to) node 1, node 2, node 3 and then node 1 again. Lines will appear joining these nodes and creating an element. Repeat the process, this time clicking in turn on nodes 3, 4, 1 and then 3 again.
Step 9	When you click again on the first node, the element's edges change colour to indicate that the element is now closed (i.e. complete). When you start creating the second element, the first element's edges will again change colour- this time to indicate that you have begun working on a new, separate element.

Tip	Note that if you try to start creating the second element by clicking again on node 1, you will actually un-create the first element.
	This is because each time you click on a node you toggle its selected status. This means that by trying to start the second element from the lead node of the first element you are deselecting that node and making it no longer form a vertex for the first element.

> *Step 10*   Once the second element is created the Graphics Window will look something like Figure 2-6. In the Edit menu, click on Commit Element Creation.
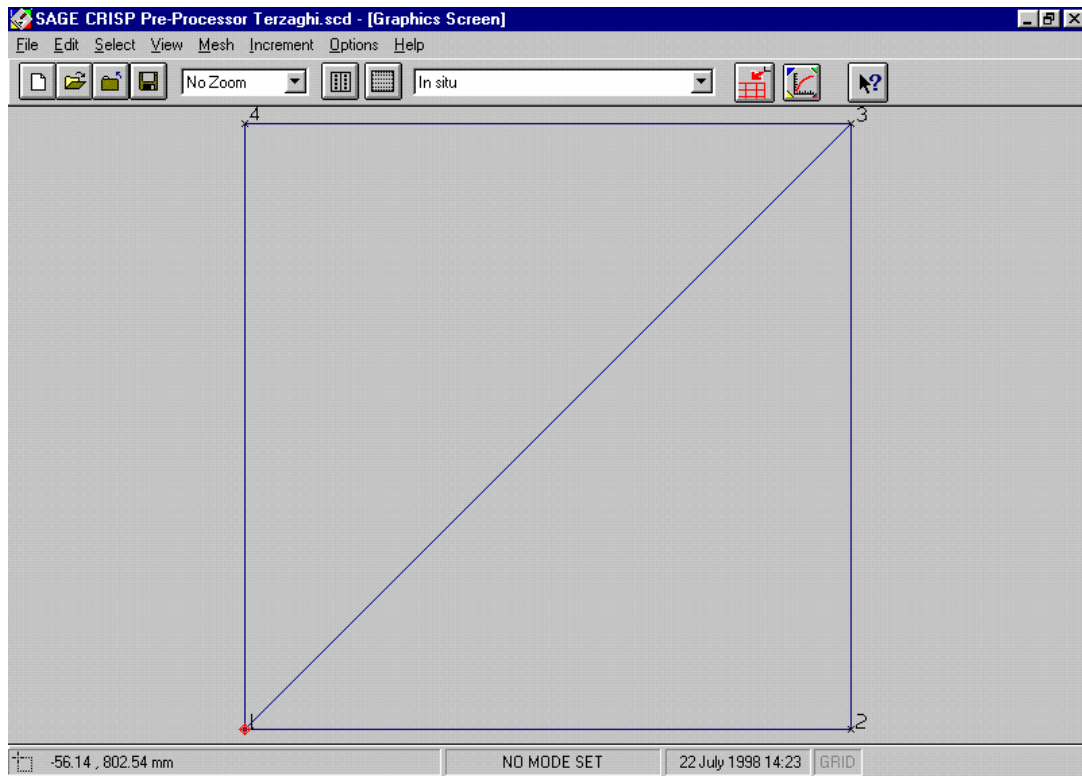
Figure 2-6:The Finite Element Mesh

2.3.4   Defining A Material Zone

A Material Zone contains information about the soil model and associated soil parameters that are being used to model particular regions of the mesh.

It is necessary to define the number of material zone(s) at this point in order to be able to allocate them using the Element Properties dialogue box. As they are referenced by name, a meaningful title is also given.

Having set up a zone, the soil model and parameters may be entered or changed at any time prior to the analysis being run.

> *Step 1*   In the Mesh menu, click on Material Properties... to display the Material Properties dialogue box.
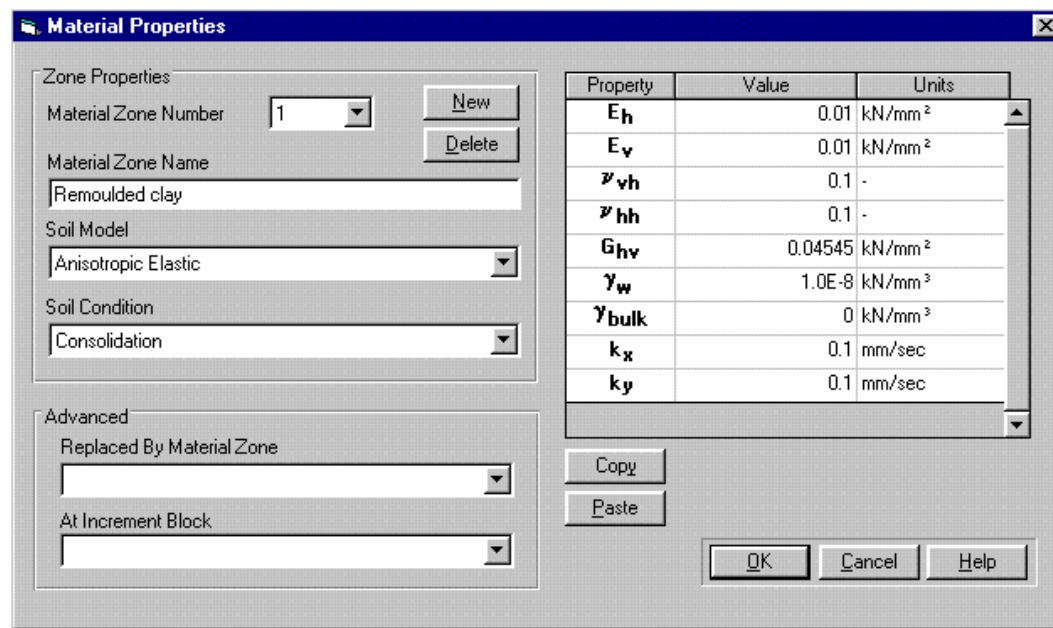
Figure 2-7:Material Properties for Oedometer Test

Step 2   Enter the name "Remoulded clay" from the Soil Models list box. Elastic is already shown, being the default.

Step 3   Enter the following soil parameters:

Property	Value	Units
E_h	0.01	kN/mm^2
E_v	0.01	kN/mm^2
v_{hh}	0.1	-
v_{vh}	0.1	-
G_{hv}	4.545 E-2	kN/mm^2
K_w / γ_w	1.0 E-8	kN/mm^3
γ_{bulk}	0	kN/mm^3
k_x	0.1	mm/s
k_y	0.1	mm/s

Step 4   The Material Properties dialogue box should now look similar to Figure 2-3. Click on OK to exit once you have finished.

Tip   In this example the soil properties are entered at this stage. If you prefer, you can enter and change the values at any time prior to running the analysis.

2.3.5   Assigning Element Properties

Every element in the mesh has two properties associated with it:

■   Finite element type, and

■ Material zone

These properties are assigned to finite elements in the following manner:

Step 1 In the Select menu, click on Domain Elements and select both elements by clicking on them. Selected elements are indicated by a user defined colour fill.

Step 2 You can deselect elements by clicking on them again.

Step 3 In the Mesh menu, click on Element Properties... to display the Element Properties dialogue box. Because the elements both have three sides, the quadrilateral element options are unavailable.
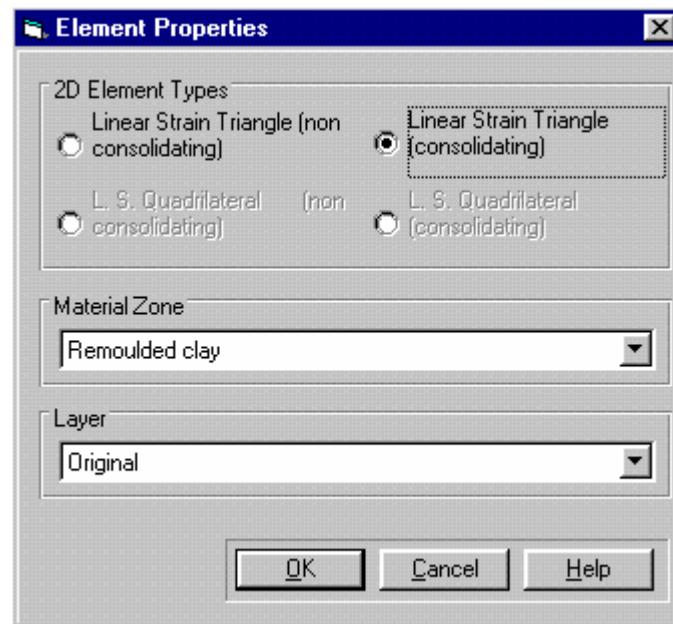
Figure 2-8:Defining Element Type and Zone

Step 4 Click on Linear Strain Triangle (consolidating).

Step 5 Select 'Remoulded clay' from the Material Zone list box.

Step 6 The Element Properties dialogue box should now look like Figure 2-8. Click on OK to exit once you have finished.

2.3.6   Defining In Situ Stress Conditions

This allows you to define the initial state of stress experienced by the mesh. However, in this example all in situ stresses are zero.

Step 1 In the Increment menu, click on Define In Situ Stress Conditions... to display the In Situ Stress Setup dialogue box.
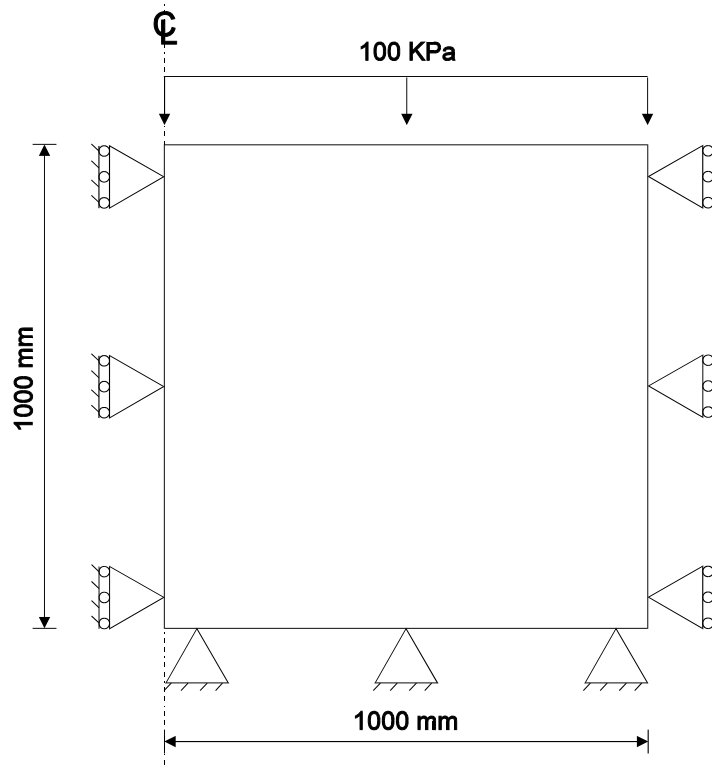
Figure 2-9:Zero In Situ Stresses

> *Step 2*   Click on OK to exit. This records a zero in situ stress state.

2.3.7   Defining Increment Blocks

Increments are the units by which the finite element program advances the analysis. In each increment, the effect of applied loads and boundary conditions are calculated and the accumulated strains and pore water pressures are updated. Increments are grouped together in blocks which then model particular events in the analysis.

This analysis requires three increment blocks. The first models the pore water pressure boundary conditions. The second models the instantaneously applied load. The third simulates the dissipation of excess pore pressures and the resulting consolidation using seven increments.

> *Step 1*   In the Increment menu, click on Define Increment Block Parameters... to display the Increment Block dialogue box.
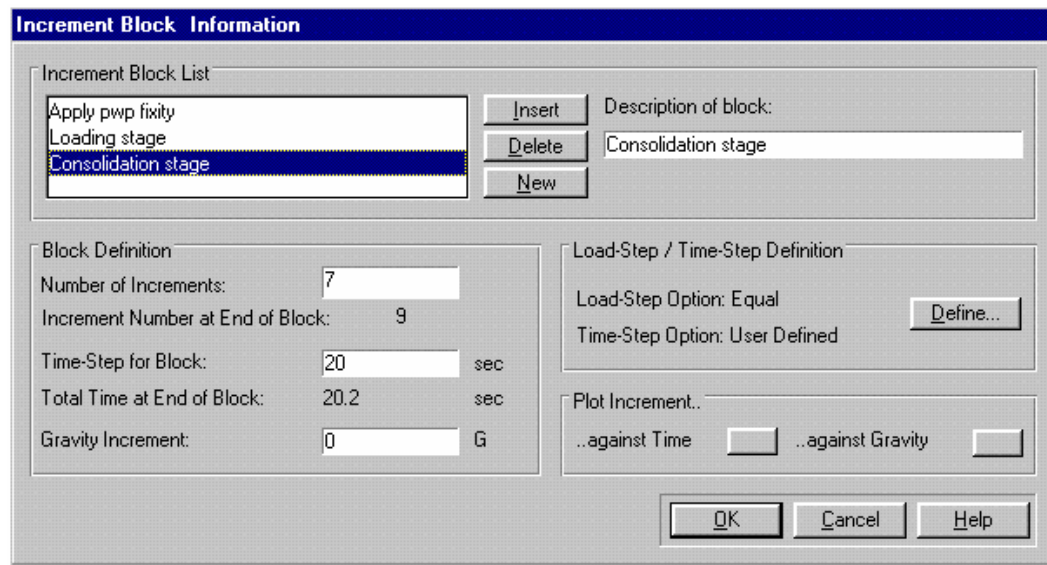
Figure 2-10:Increment Block dialogue box

Step 2 A default block called 'Increment Block (untitled)' is automatically created because no other blocks exist. Change the name to "Apply pwp fixity" in the Description of Block text box.

Step 3 Allocate a single increment and a time-step for the block of 0.1 seconds.

Step 4 Click on New to create another increment block and call it "Loading stage".

Step 5 Allocate a single increment and a time-step for the block of 0.1 seconds.

Step 6 Click on New to create another increment block and call it "Consolidation stage".

Step 7 Assign seven increments and a time-step for the block of 20 seconds to this block.

Step 8 Click on the Define... button. This displays the Define Load/Time Steps dialogue box.
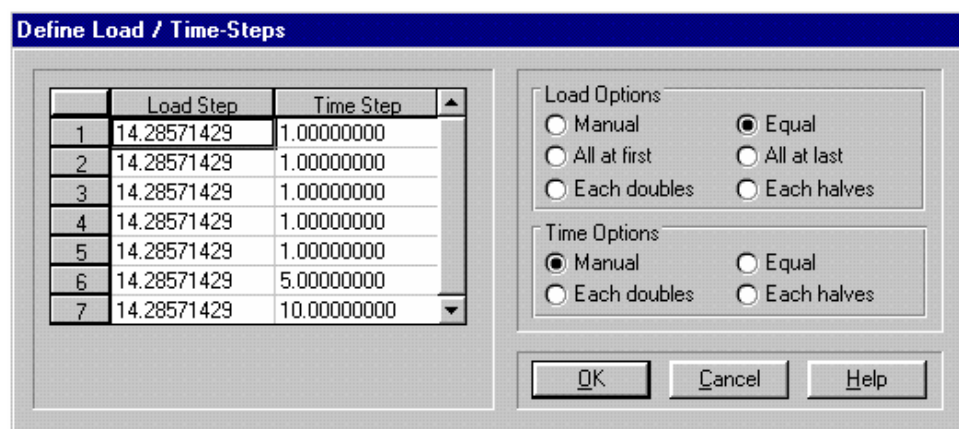
Figure 2-11:Defining Load and Time Steps

Step 9	In the Time Step column enter 1 second into the first five increments, 5 into the sixth and 10 into the seventh. This is the exact distribution of the 20 second total time-step.
Step 10	The Define Load/Time Steps dialogue box should now look like Figure 2-11. Click on OK to return to the Increment Block dialogue box, once you have finished.
Step 11	Upon clicking OK, a prompt will appear because the time-steps do not equal 100%, asking if you require automatic smoothing. Select Yes and the values will be multiplied by a common factor so that they total 100.
Step 12	Click on OK to exit the Increment Block dialogue box

2.3.8   Defining Boundary Conditions

Boundary conditions describe to the finite element analysis the ways in which the mesh is physically restrained from moving and the drainage features whose effects must be accounted for. This is achieved by applying fixities to edges and nodes in the mesh.

Once defined, fixities apply in all successive increment blocks. The Fixities dialogue box and the Graphics Window (if View Fixities is toggled on) will only display any fixity that is applied during the <u>current</u> increment block.

This example incorporates a pore water pressure fixity along the top edge. It is recommended that pore pressure fixities not be applied in the same increment block as any loads are applied. According to this, displacement fixities will be applied at the in situ stage, pore pressure fixities in the first increment block and the load in the second block.

See Section 6.5 in the Technical Reference Guide for further details about applying pore pressure fixities

The following procedure defines the fixities:

Step 1	Ensure that 'In situ' is selected from the Increment Block list box on the main toolbar.
Step 2	In the Select menu, click on Edges.
Step 3	Select the left-hand and right-hand (vertical) edges of the mesh by clicking on them. Both of these edges will have zero X direction fixity, so they can be defined simultaneously.
Step 4	In the Increment menu, click on Fixities... to display the Edge Fixity dialogue box.
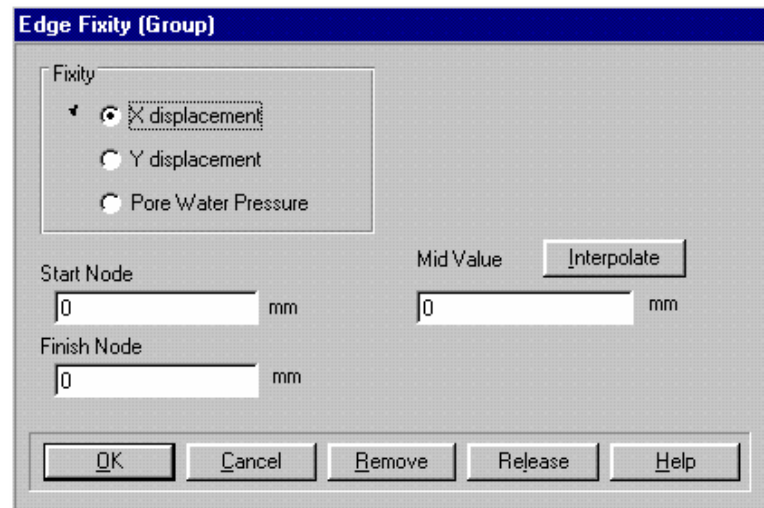
Figure 2-12:Zero X Direction Fixities

Step 5   Click on the X displacement option and type '0' into the Start Node and Finish Node text boxes. Now set the Mid Value fixity to zero by either pressing the Interpolate button or by typing in '0' directly.

Step 6   The Edge Fixity dialogue box should now look like Figure 2-12. Click on OK to exit once you have finished.

Step 7   Click on Clear Selection in the Select menu to ensure that no edges are selected (Select Edge remains active). Then select the bottom (horizontal) edge of the mesh by clicking on it.

Step 8   In the Increment menu, click on Fixities....

Step 9   Click on the Y displacement option and define zero fixities, as above.

Step 10   Click on OK to exit once you have finished and click on the bottom edge of the mesh to de-select it.

Step 11   The Graphics Window should now look similar to Figure 2-13. Note that the display of fixity icons can be turned on and off by clicking on Fixities in the View menu and that their size can be controlled in the Default Settings dialogue box (accessed via the Options menu).
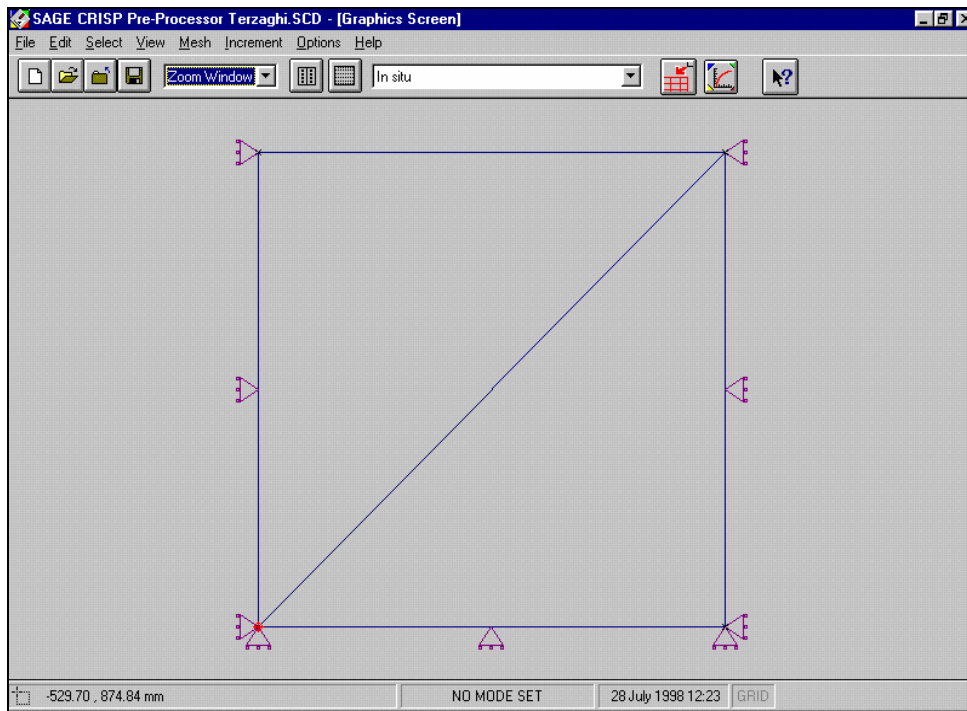
Figure 2-13:Displacement Fixities (at the In Situ Stage)

Step 12   Select 'Apply pwp fixity' from the Increment Block list box on the main tool bar.

Step 13   Click on Clear Selection in the Select menu to ensure that no edges are selected and select the top (horizontal) edge of the mesh by clicking on it.

Step 14   In the Increment menu, click on Fixities....

Step 15   Click on the Pore Water Pressure option. This will activate the pore pressure fixity group of options.
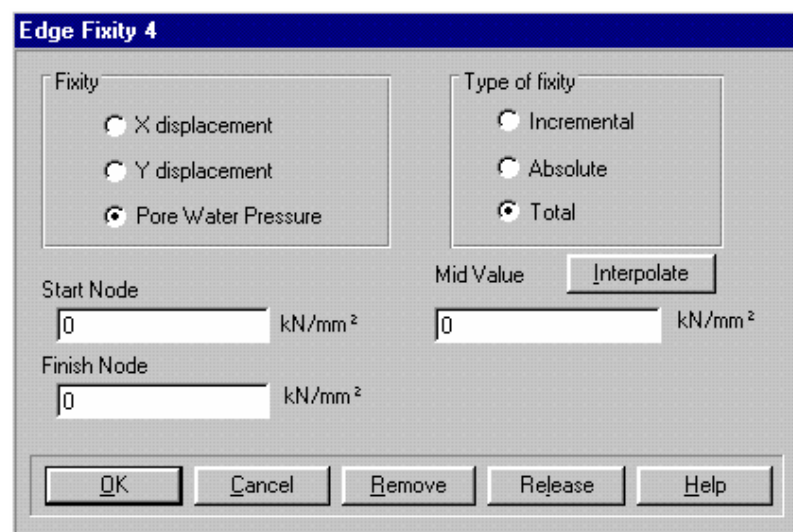
Figure 2-14:Applying A Pore Pressure Fixity

Step 16   Click on the Total option and enter zeroes into the Start Node, Finish Node and Mid Value text boxes.

Step 17   Click on OK to exit. This concludes the application of boundary conditions.

2.3.9   Defining Loads

A load is applied to the top edge of the mesh during the second increment block, but with a very small time increment so that the effect is one of instantaneous application.

As with fixities, once defined loads apply in all successive increment blocks. The Loads dialogue box and the Graphics Window (if View Loads is toggled on) will only display any load that is applied during the <u>current</u> increment block.

Step 1   Select 'Loading stage' from the Increment Block list box on the main toolbar.

Step 2   Click on Clear Selection in the Select menu to ensure no edges are selected.

Step 3   Select the top (horizontal) edge of the mesh by clicking on it.

Step 4   In the Increment menu, click on Loads... to display the Edge Load dialogue box.

Step 5   The applied loading is to be 100 kN/m^2, so enter a value of 0.0001 kN/mm^2 into the Normal Start node and Normal Finish node text boxes. Click on Interpolate and the Normal Mid Value will automatically be entered as 0.0001 kN/mm^2. Alternatively the value can be entered manually.

Step 6   The Edge Load dialogue box should look like Figure 2-15. Click on OK to exit once you have finished.
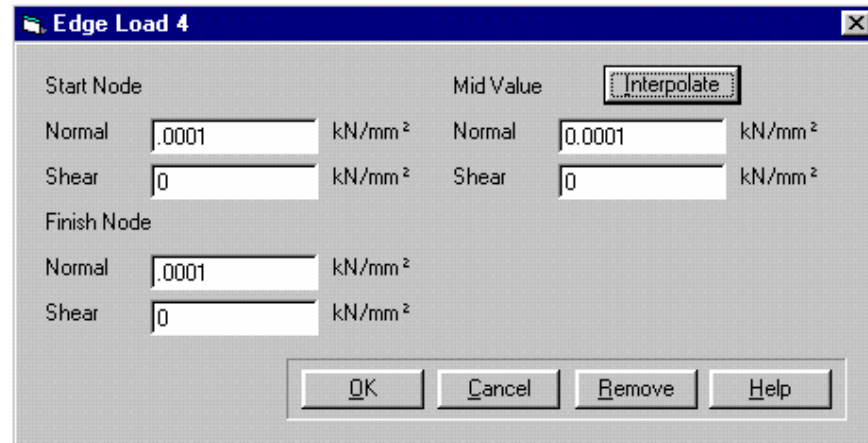
Figure 2-15:Loading the Top Edge

2.3.10 Scale Factors

Loads are displayed on the Graphics Window as arrows whose length is proportional to the size of the load. In order to optimise the display of these arrows, a scaling factor can be specified that will increase or decrease the length of every load arrow in the mesh by the same proportion.

Customising the display of loads in this way takes three forms- Shear load scale, Normal load scale and Offset (the distance between the arrow head and the edge or node to which it applies).

This dialogue box also controls the display of text (node numbers, element numbers, etc.) and icons (such as those used to represent fixities). The displayed size of these can be controlled by changing the scale factor values in the Text and Icons section.

Step 1   In the Options menu, click on Default Settings... to display the Defaults dialogue box.
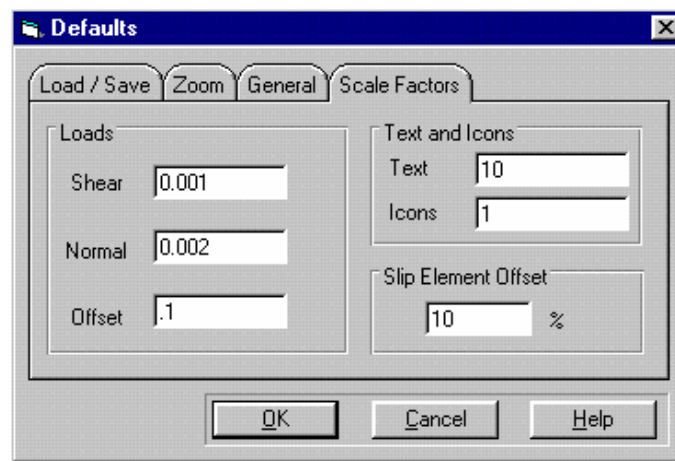
Figure 2-16:Setting Scale Factors

Step 2   Click on the Scale Factors tab.

Step 3   Enter values of 0.002 in the Normal text box and 0.1 in the Offset text box.

Step 4   If your fixity icons do not look anything like those shown in Figure 2-13, try changing the size by altering the basic display size. Values of 10 and 1 (as shown in Figure 2-17) were used to produce Figure 2-13.

Step 5   The Defaults dialogue box should look like Figure 2-17. Click on OK to exit once you have finished. The Graphics Window should look similar to Figure 2-17.
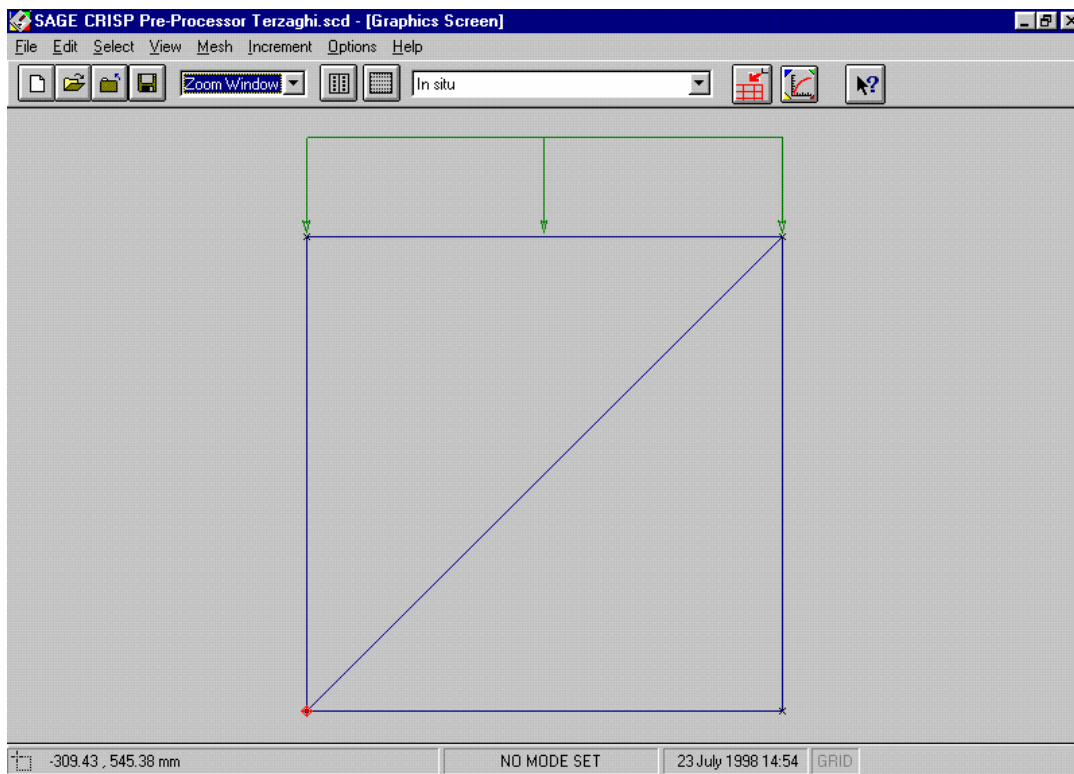
Figure 2-17:Loading Applied to the Top of the Sample

2.3.11 Running The Analysis

The Project should now be saved. This concludes the pre processing for this particular problem so all that remains is for you to run the finite element analysis program.

Step 1 In the File menu, click on Run Analysis... to run the finite element analysis. This displays the Run Analysis dialogue box.
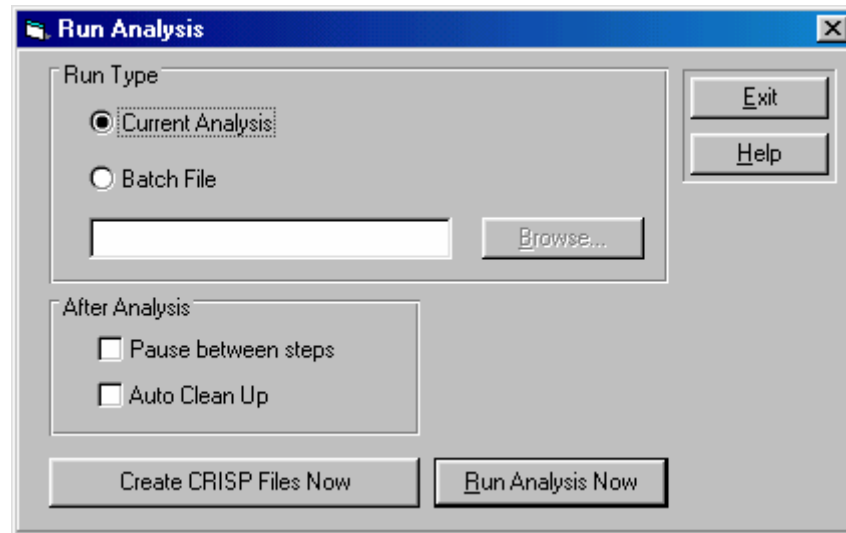
Figure 2-18:The Run Analysis dialogue box

Step 2 Click on Create CRISP Files Now. This will write two files with the extension .GPR and .MPD to disk. These files contain the input data required by the finite element analysis program. If any errors in the analysis data are encountered, you will be informed of the problem by a message.

Step 3 Provided no errors were encountered, click on Run Analysis Now. This will envoke the CRISP analysis engine modules (geometry program and main program).

Step 4 When the analysis is finished, you will be returned to Windows and the output from the analysis is converted into a Microsoft Access 2.0 database. A percentage bar informs you as to how long you have left to wait.

Step 5 You can load your results directly into the Post-Processor by clicking on the Post-Processor button in the Analysis Docker dialogue box.

2.4 Post-Processing

The Post-Processor contains a set of features that allow you to look at the output from your analysis.

In this section you will be introduced to the following features:

- Displacement Plots

- Duration Graphs (using nodes and integration points)

2.4.1 Introduction

If you entered the Post-Processor directly from running the analysis, your Project will already be loaded. However you may also enter the Post-Processor by double-clicking on its icon in the CRISP2D program group in Windows' Program Manager. In this case, you should select Open Project... from the File menu and, provided the analysis has been successfully run, you will be able to post process the output from your analysis.

The example assumes that you have successfully loaded the one dimensional consolidation Project into the Post-Processor.

2.4.2   Displacement Plots

Displacement plots are drawn directly onto the Graphics Windows. They provide a representation of how the mesh has deformed at any stage in the analysis. You can view either vector displacements or the deformed mesh at any specified magnification.

Step 1   In the Plots menu, click on Displacement Plots... to display the Displacement Plot dialogue box.
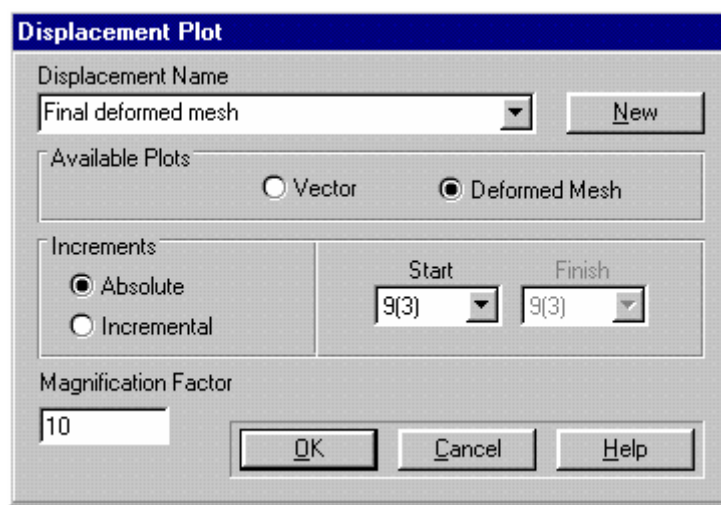
Figure 2-19:Displacement Plot dialogue box

Step 2   Click on the New button to create your plot and change the title to 'Final deformed mesh'.

Step 3   Select the Deformed Mesh option.

Step 4   Ensure that Absolute is selected and that the Start increment is 9.

Step 5   Enter a Magnification Factor of 10. The Displacement Plot dialogue box should look similar to Figure 2-19.

Step 6   Click on OK to exit once you have finished. The plot will be displayed on the Graphics Window similar to Figure 2-20.
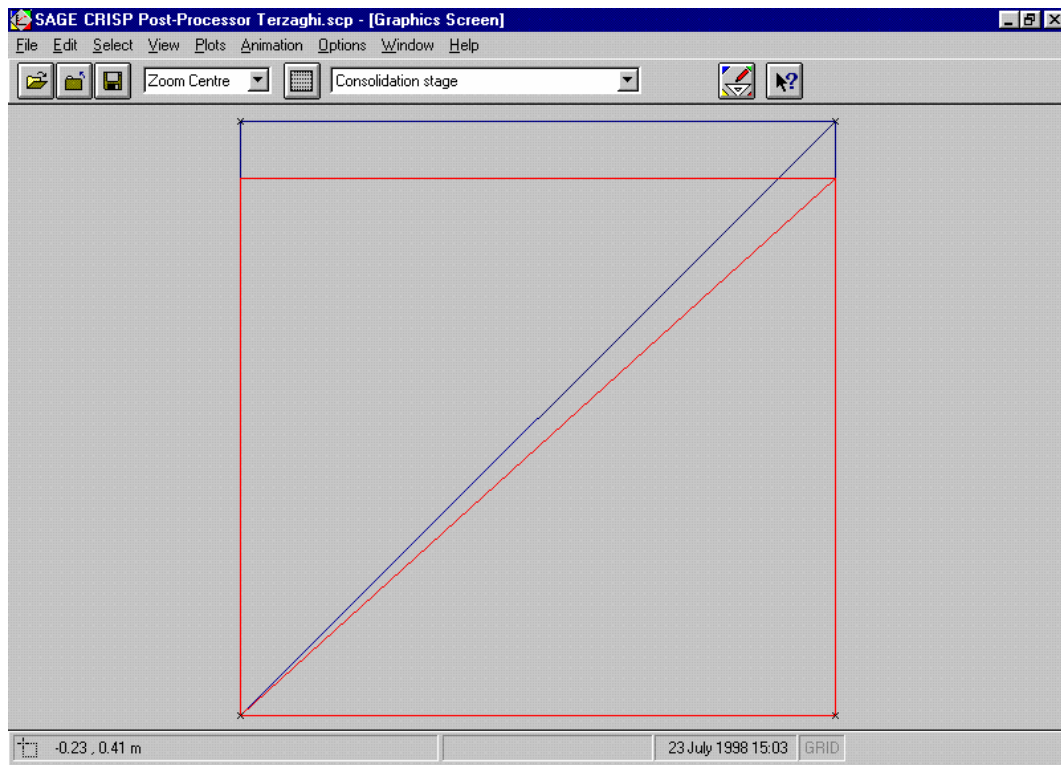
Figure 2-20:Deformed Mesh After Full Consolidation

Tip	Displacement plots look clearer if you select good line styles and colours. To change the line styles, select Line Styles... from the Options menu. To change the line colours, select Line Colours... from the options menu.

2.4.3   Duration Graphs

Duration graphs allow you to investigate the variation of output from individual integration points or nodes over the course of a specified range of increments.

Up to 10 data sets (each one representing a different integration point or node) can be viewed on the same graph, but for this example a single node will be used.

Two Duration graphs will be produced here:

- Vertical Displacement plotted against Time, for the top left node

- Total Pore Pressure plotted against Time, for two integration points

Vertical Displacement vs Time

Displacement in this example is purely in the vertical direction (as confirmed by the Displacement Plot produced earlier). Therefore, the amount of vertical displacement experienced by the top edge can be plotted on a graph drawn from either one of the nodes along the top edge.

The following procedure describes how to create a graph plotting the movements of the top, left-hand node:

Step 1   In the Select menu, click on Nodes and then on Clear Selection. Select the node at the top left corner of the mesh by clicking on or close to it.

Step 2   In the Plots menu, click on Duration Graphs... to display the Duration Graphs dialogue box.
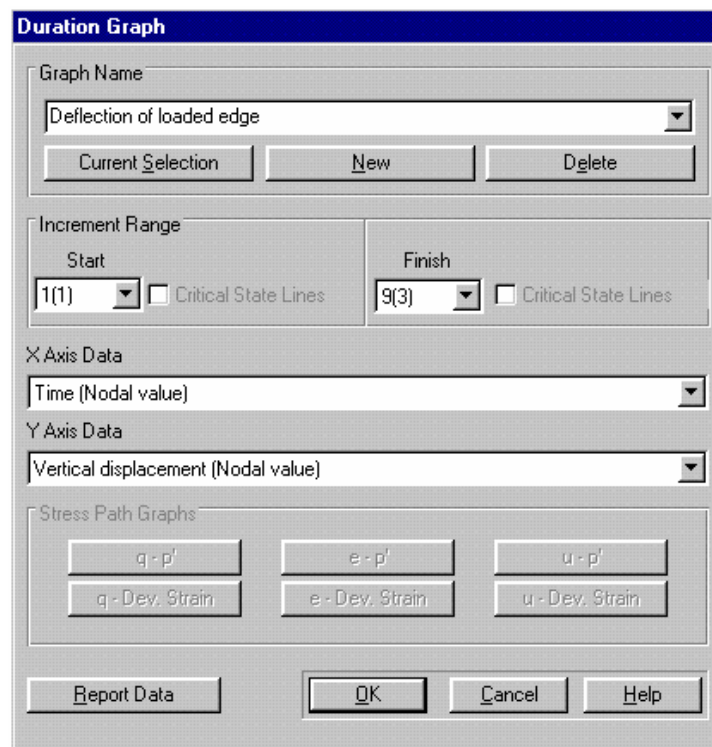
Figure 2-21:Duration Graph dialogue box

Step 3   Click on the New button to create your graph and change the title to "Deflection of loaded edge".

Step 4   Ensure that the Increment Range starts at 1 and finishes at 9.

Step 5   Select 'Time' as X Axis data and 'Vertical displacement' as Y Axis data. The Duration Graph dialogue box should now look similar to Figure 2-21.

Step 6   Click on OK to exit once you have finished. The graph will be displayed with a configuration that depends upon your default settings. It should look similar to Figure 2-22.

Step 7   If the graph does not look very similar, you can change the settings (axis labels, titles and legend) in the Graph Configuration dialogue box. This can either be accessed from the Options menu, or by double clicking on the graph itself. This graph has no legend, no grid lines, 11 X axis divisions, 10 Y axis divisions, X axis minima and maxima of 0 and 22, Y axis minima and maxima of -0.01 and 0.
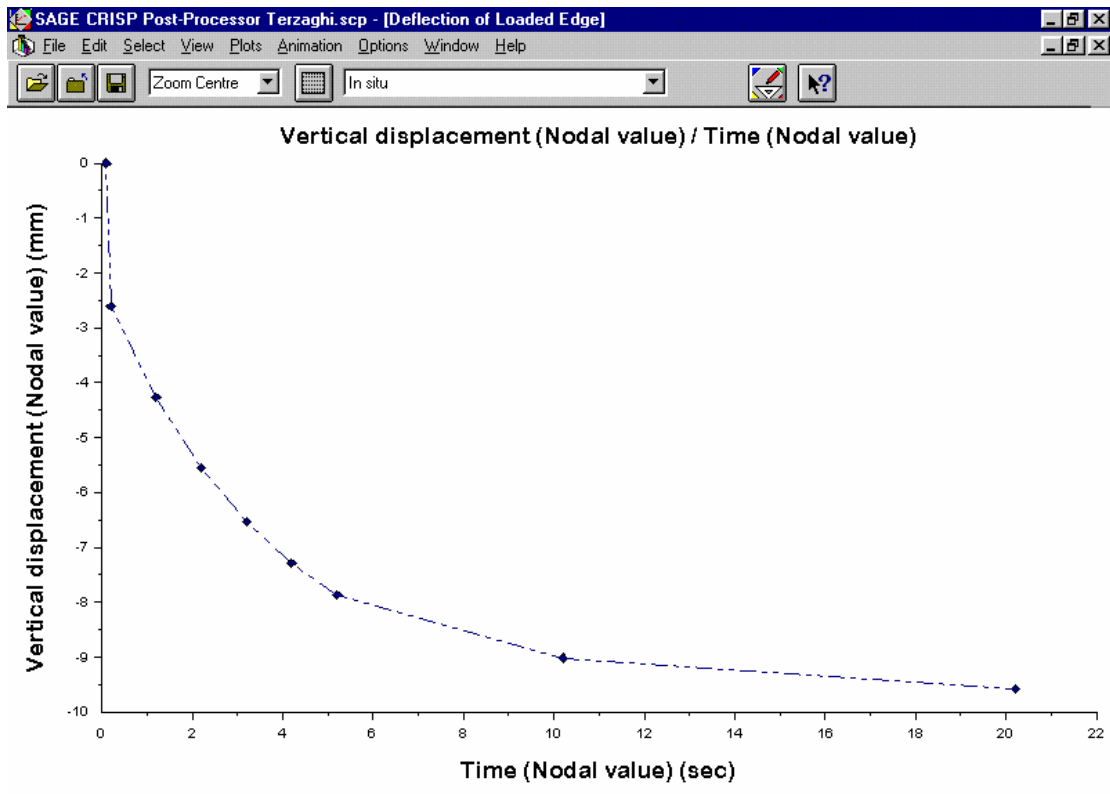
Figure 2-22:Graph Showing Vertical Deflection of the Sample

Total Pore Pressure vs Time

The following procedure describes how to create a plot showing the dissipation of pore pressure with time for two points in the mesh. These points are the centroids of the two elements and are respectively 1/3 in and 2/3 up, and 2/3 in and 1/3 up from the bottom left-hand corner.

The graph is plotted according to the following procedure:

Step 1   To reactivate the Graphics Window, click on Cascade or on Tile Horizontally in the Window menu. These options allow you to switch between the Graphics Window and any graphs that are on display.

Step 2   In the Select menu, click on Integration Points. This will automatically action Integration Points in the View menu so that they appear on screen.

Step 3   Select the integration point at the centre of each element.

Step 4   In the Plots menu, click on Duration Graphs... to display the Duration Graphs dialogue box.

Step 5   Click on the New button to create your graph and change the title to "Dissipation of Pore Pressure".

Step 6   Ensure that the Increment Range starts at 1 and finishes at 9.

Step 7   Select 'Time' as X Axis data and 'Total Pore Pressure' as Y Axis data.

Step 8   Click on OK to exit once you have finished. The graph displayed should look similar to Figure 2-23.

Step 9   This graph is configured with a legend, no grid lines, 11 X axis divisions, 6 Y axis divisions, X axis minima and maxima of 0 and 22, Y axis minima and maxima of 0 and 120.
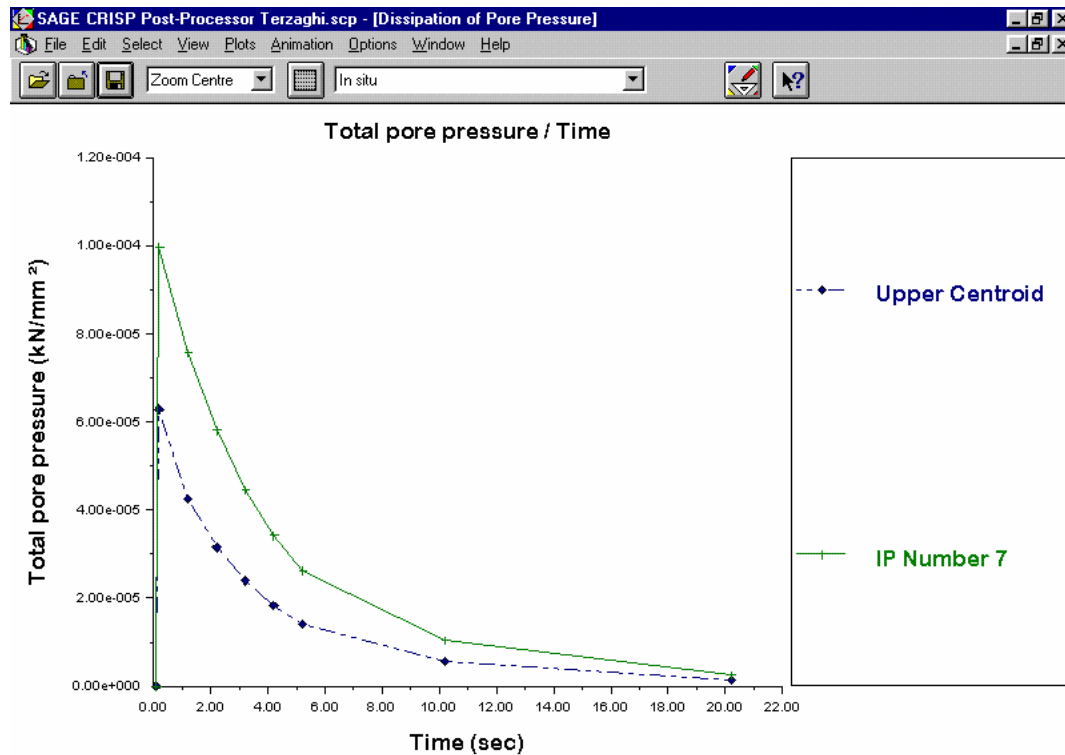
Figure 2-23:Graph Showing the Dissipation of Total Pore Pressure

Note that the γ_{bulk} value of the soil was set to zero in this analysis so that the in situ stresses could be set to zero. This allows the dissipation of pore pressure to be illustrated much clearer. It is only very rarely that the in situ stresses are set to zero.

2.5   Summary

By the end of this example you will have made use of the following features:

■   Units

■   The Node List

■   Creating finite elements

■   Material zones

■   In situ stresses

■   Increment blocks

■   Displacement and pore water pressure fixities

■   Loads

■   Load scale factors

■   Displacement plots

■   Duration graphs

2.6   Things To Try Next

■   Try using more increments (with smaller time-steps) to more accurately capture the sample's behaviour at the beginning of consolidation.

■   Try creating a mesh with 8 elements to see how mesh geometry affects the results.

■   Try using cubic strain triangles to see how the use of a higher order element influences the results.

■   Try using non consolidation elements and drained material values to see if the drained condition is the same as the long term consolidation condition.

UNDRAINED TRIAXIAL TEST

3

3.1 Introduction

This example is designed to introduce you to one of the family of critical state soil models, namely the Cam clay model. The example does not aspire to impart you with a knowledge of critical state soil mechanics, so it is recommended that it only be attempted if you are already familiar with critical state theory. The subject of the example is a strain controlled, undrained triaxial test.

The following steps will be covered:

- Starting The Project

- Defining Units

- Creating The Finite Element Mesh

- Defining A Material Zone

- Assigning Element Properties

- Defining In Situ Stress Conditions

- Defining Increment Blocks

- Defining Boundary Conditions

- Defining Loads

- Scale Factors

- Running The Analysis

3.2 The Problem

The triaxial problem under consideration is illustrated in Figure 2-1. Because of consideration of symmetry only a quarter section needs to be considered for this analysis. The finite element mesh that will be used consists of 2 linear strain triangle elements. Cubic strain triangles are recommended for axisymmetric problems, but for this simple example their use is not required.

See the Technical Reference Guide for further information about choosing the right type of finite element for your problem.
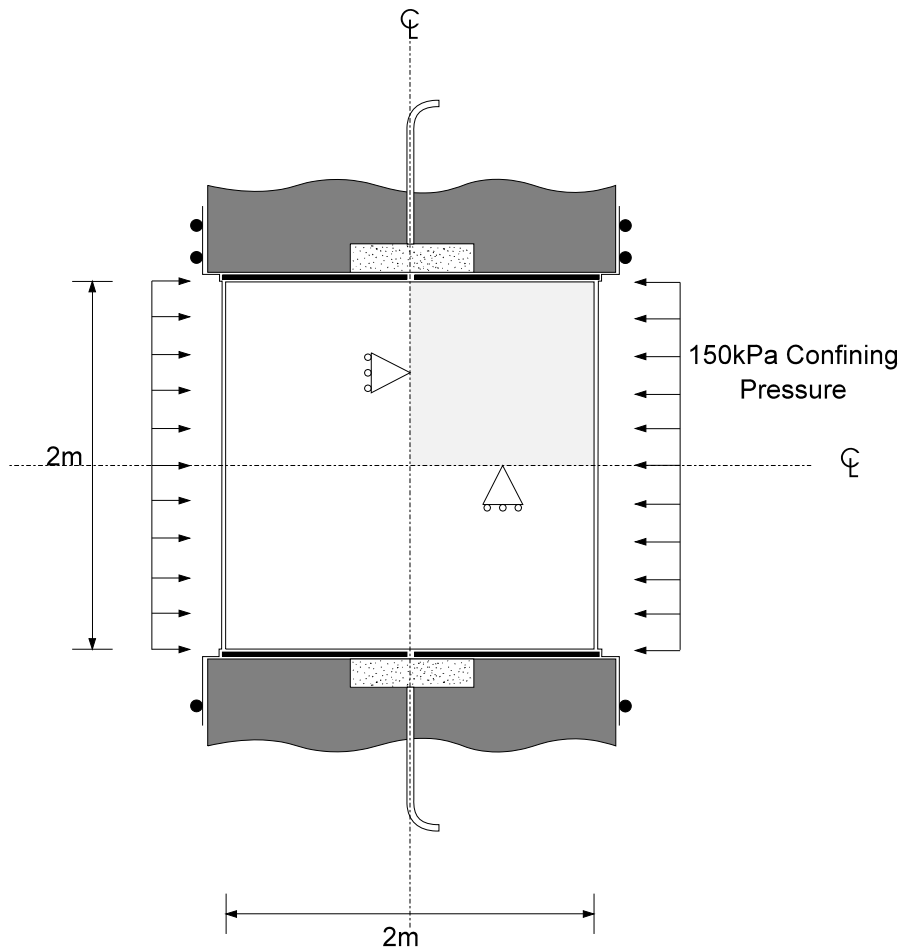
Figure 3-1:Triaxial Test Cell

The soil sample is isotropically consolidated to 200 kPa and then isotropically unloaded to a mean normal stress of 150 kPa. A standard undrained compression test is then carried out. The Cam clay parameters selected for the soil are as follows:

$\lambda = 0.30$ $\qquad$ $\kappa = 0.05$ $\qquad$ $M = 1.0$

$e_{cs} = 2.953$ $\qquad$ $\nu' = 0.3$

As this is an undrained analysis, the bulk modulus of water is required to complete the data on material properties.

$$\text{Effective bulk modulus of soil} \left(K'\right) = \frac{(1+e) \cdot p'}{\kappa}$$

$$K' = \frac{(1+1.5517) \cdot 150}{0.05}$$

$$K' = 7655 \text{ kPa}$$

For undrained analyses the bulk modulus of water approximately 100 K' (7.655×10^5 kPa). In the analysis presented here a value of 5.0×10^5 kPa was used.

More than 50 load increments are recommended for an finite element analysis which simulates a triaxial test. For the purposes of this example however, only six load increments will be used.

A strain controlled test is considered here with 0.5% axial strain in each increment, leading to a total axial strain of 3%. The left-hand edge is restricted to move vertically and the top and bottom edges are restricted to move horizontally. The top edge is displaced vertically downwards to simulate a strain controlled test.

The results of the analysis are plotted in q - ε_a, u - ε_a, and q - p′ space

. These are compared with the theoretical and also with an analysis using a large number of increments. These differences between theory and predictions are due to the large increment size which was used solely for illustrative purposes. During the second increment the yield ratio (YR) is on average about 1.12, i.e. the yield locus has grown in size by 12%. Stricter control on YR is recommended so that the change in size of the yield locus is not more than 3% (i.e. YR 1.03 when hardening or 0.97 when softening). The results emphasise the importance in selecting the size of load increment: as shown here large load increments may lead to erroneous results.

3.3 Pre Processing

This section describes all of the stages that must be completed using the Pre-Processor in order that the analysis can be run. It is recommended that you save the Project regularly.

3.3.1 Starting The Project

This section describes the basic processes that should be completed in advance of creating your mesh. This involves entering data to set-up various run-time options for the finite element analysis and to descriptively identify the Project. The base units are also verified.

The following procedure describes the initial set-up of the Project.

Step 1	On opening the CRISP2D Pre-Processor, click on New Project... in the File menu. This displays the Mesh Mode dialogue box.
Step 2	Click on the Finite Mesh button. This exits the Mesh dialogue box, sets you up in Finite Element mode and opens the Project Setup dialogue box.
Step 3	In the Domain Type section, click on the Axisymmetry option button and click on All Other Elements in the Element Type section.
Step 4	Enter an in situ gravity level of 1 G.
Step 5	The dialogue box should now look like Figure 3-2. Click on OK to exit once you have finished.
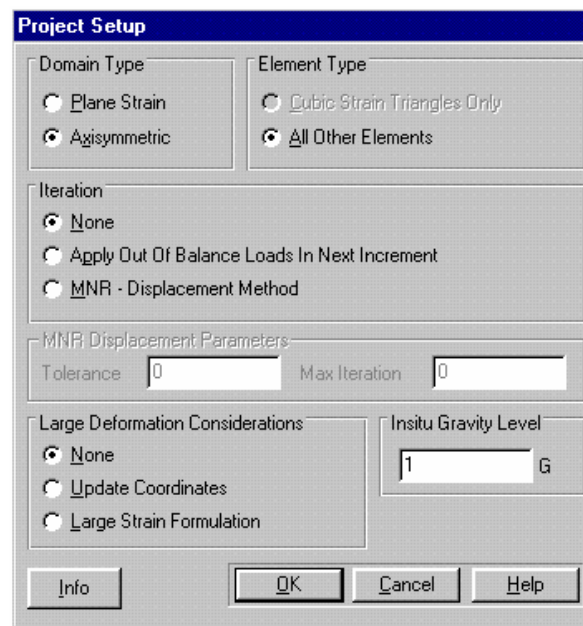
Figure 3-2:Project Setup dialogue box

Step 6   Click on the Info... button. This displays the Project Information dialogue box.
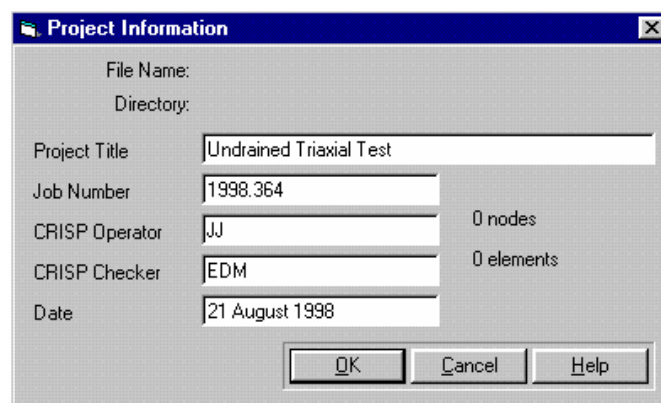
Figure 3-3:Project Information dialogue box

Step 7   Enter data as appropriate. The date is automatically entered but may be manually altered.

Step 8   The Project Information dialogue box should look similar to Figure 3-3. Click on OK to return to the Project Setup dialogue box once you have finished.

Step 9   Click on OK to exit.

3.3.2   Defining Units

At this point it is useful to ensure that the correct system of base units is being used.

Step 1   In the Options menu, click on Units... to display the Units dialogue box.

Figure 3-4:Defining Base Units

Step 2   Use the list boxes to select the appropriate units for Distance, Force, Time and Angle as shown in Figure 3-4.

Step 3   Click on OK to exit once you have finished.

3.3.3   Creating The Finite Element Mesh

The Project is now ready for you to define your finite element mesh. The quickest, most accurate way to create a regular mesh is to first use the Node List to define the vertices, which can then be easily linked to form the mesh elements.

Step 1   In the Mesh menu, click on Node List... and enter four nodes with co-ordinates (0 , 0), (1 , 0), (1 , 1) and (0 , 1) into the Node List dialogue box, as shown in Figure 3-5.

Figure 3-5:Node List dialogue box

Step 2   Click on OK to exit once you have finished.

Step 3   The Graphics Window now displays the nodes you have created. To view them at optimum scale, select 'Full Page' from the Zoom list box on the main toolbar.

Step 4   In the View menu, click on Node Numbers to switch on the display of node numbers.

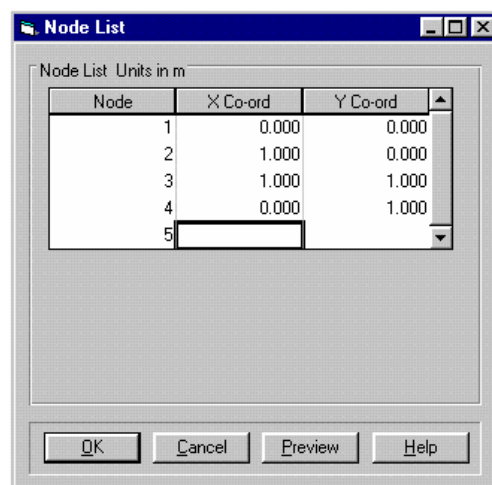*Step 5*   In the Mesh menu, click on Create Domain Element. Using the left-hand mouse button click on (or close to) node 1, node 2, node 3 and then node 1 again. Lines will appear joining these nodes and creating an element. Repeat the process clicking in turn on nodes 3, 4, 1 and then 3 again. As you start creating the second element, the first will change colour (the colours are definable in the Colours dialogue box, which is accessed via the options menu).

Tip   Note that if you try to start creating the second element by clicking again on node 1, you will actually un-create the first element.

This is because each time you click on a node you toggle its selected status. This means that by trying to start the second element from the lead node of the first element you are deselecting that node and making it no longer form a vertex for the first element.

Step 6   Once the second element is created, in the Edit menu, click on Commit Elements. This fixes the newly created elements, confirming that they are correctly created, the Graphics Window will look something like Figure 3-6.

Figure 3-6:The Finite Element Mesh

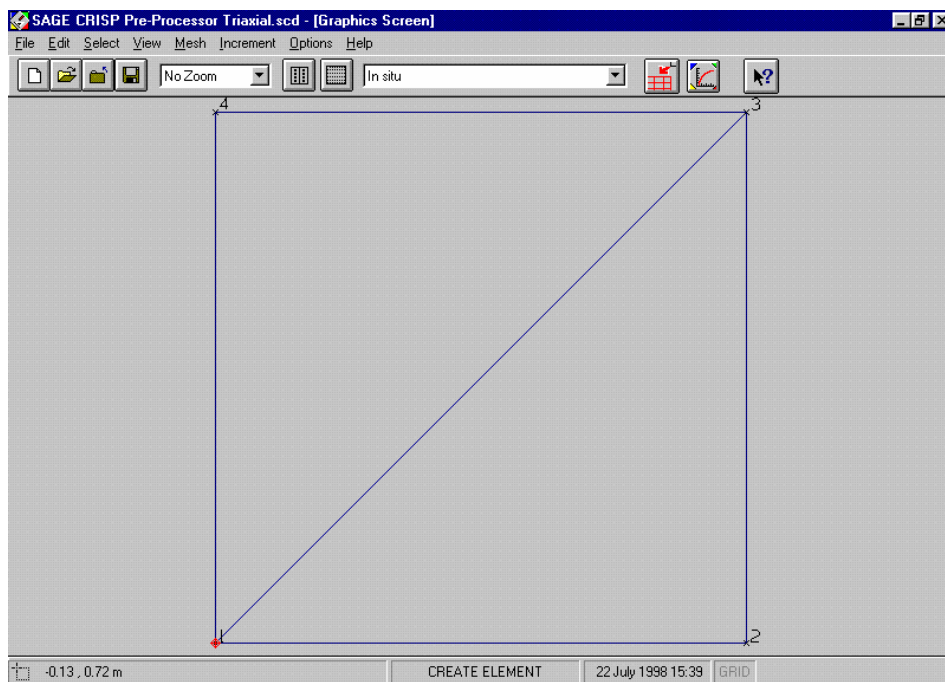# 3.3.4 Defining A Material Zone

A Material Zone contains information about the soil model and associated soil parameters that are being used to model particular regions of the mesh.

It is necessary to define the number of material zone(s) at this point in order to be able to allocate them using the Element Properties dialogue box. As they are referenced by name, a meaningful title is also given.

Having set up a zone, the soil model and parameters may be entered or changed at any time prior to the analysis being run.

Step 1   In the Mesh menu, click on Material Properties... to display the Material Properties dialogue box.

Figure 3-7:Material Properties for Undrained Triaxial Test

Step 2   Enter the name "Overconsolidated Kaolin Clay" and select 'Cam clay' from the Soil Models list box.

Step 3   Enter the following soil parameter values:

Property	Value	Units
κ	0.05	None
λ	0.3	None
e_{cs}	2.953	None
M	1.0	None
G or ν'	0.3	None
K_w / γ_w	5.0E05	kN/m^3
γ_{bulk}	20	kN/m^3
k_x	0	m/s
k_y	0	m/s

Step 4   The Material Properties dialogue box should now look like Figure 3-7. Click on OK to exit once you have finished. Note that k_x and k_y are not required for an undrained analysis.

Tip   It is not necessary to define permeability values for analyses which do not have consolidation elements.

3.3.5   Assigning Element Properties

Every element in the mesh has two properties associated with it:

- Finite element type, and

- Material zone

These properties are assigned to finite elements in the following manner:

Step 1   In the Select menu, click on Domain Element and select both elements by clicking on them.

Step 2   In the mesh menu, click on Element Properties... to display the Element Properties dialogue box. Because the elements have the same number of sides, their properties can be defined simultaneously.
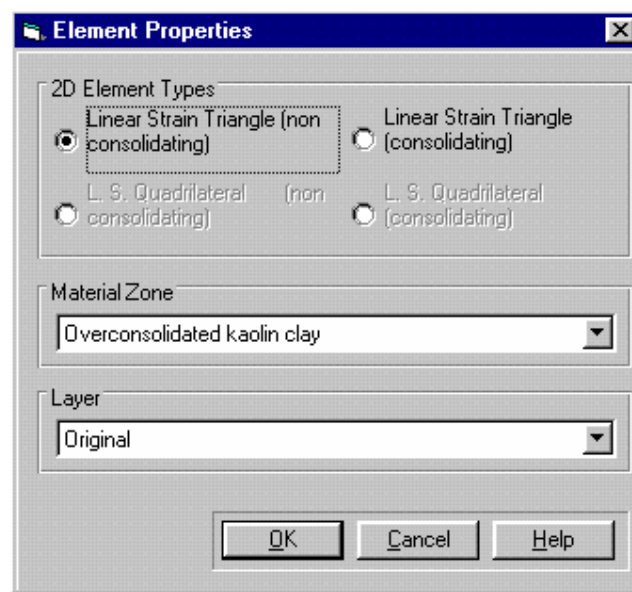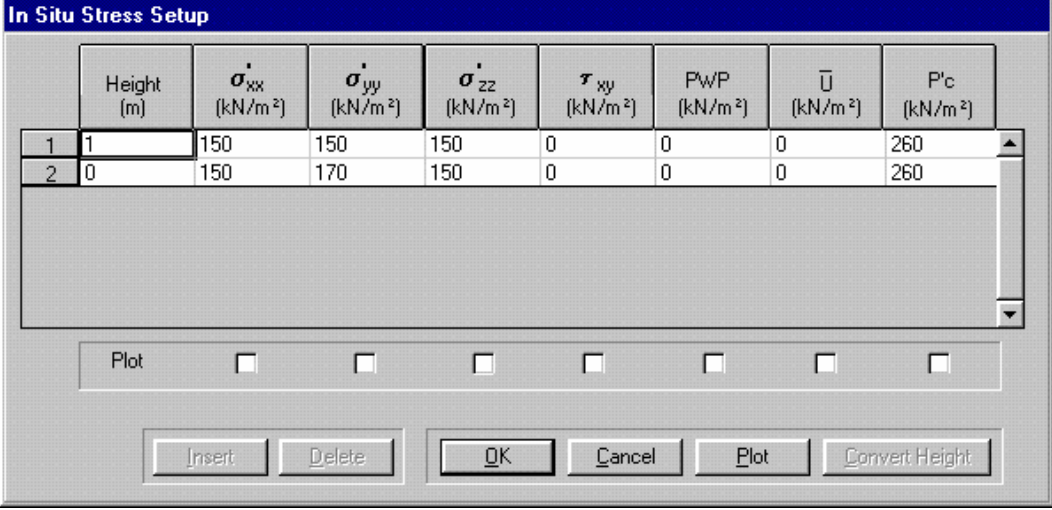
Figure 3-8:Defining Element Type and Zone

Step 3   As the selected elements have 3 sides, their Element Type is automatically fixed as Linear Strain Triangle. Select the (non consolidating) option.

Step 4   Select 'Overconsolidated Kaolin Clay' from the Material Types list box. If you have been following the example this will be the only option in the list.

Step 5   The Element Properties dialogue box should now look like Figure 3-8. Click on OK to exit once you have finished.

3.3.6   Defining In Situ Stress Conditions

This allows you to define the initial state of stress experienced by the mesh. Although the Stress Conversion Parameters dialogue box provides a means of automatically calculating in situ stresses, we already know the values in this case.

Step 1   In the Increment menu, click on Define In Situ Stress Conditions... to display the In Situ Stress Setup dialogue box.

Figure 3-9:In Situ Stress dialogue box

Step 2   Enter the values as shown below in Figure 3-9.
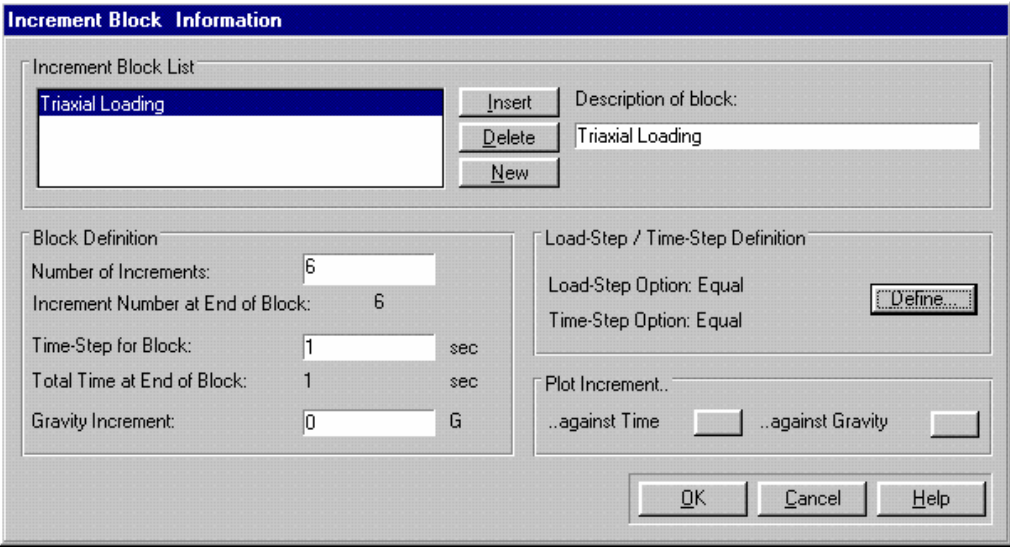
Step 3   Click on OK to exit once you have finished.

3.3.7   Defining Increment Blocks

Increments are the units by which the finite element program advances the analysis. In each increment, the effect of applied load and boundary conditions is calculated and the accumulated strains and pore water pressures are updated. Increments are grouped together in blocks which then model particular events in the analysis.

This analysis requires only one increment block. This models the application of a prescribed displacement. The block comprises 6 increments.

Step 1   In the Increment menu, click on Define Increment Block Parameters... to display the Increment Block Information dialogue box.

Figure 3-10:Increment Block dialogue box

> ***Step 2*** A default block called 'Increment Block (untitled)' is automatically created because no other blocks exist. Change the name to "Triaxial Loading" in the Description of Block text box.

Tip As mentioned in Section 2-2, 6 increments are insufficient for the problem being analysed. If you prefer, you may enter 50 increments here, but the penalty is that the finite element analysis will take longer to run.

> ***Step 3*** Click on the Define... button. This displays the Define Load/Time Steps dialogue box.
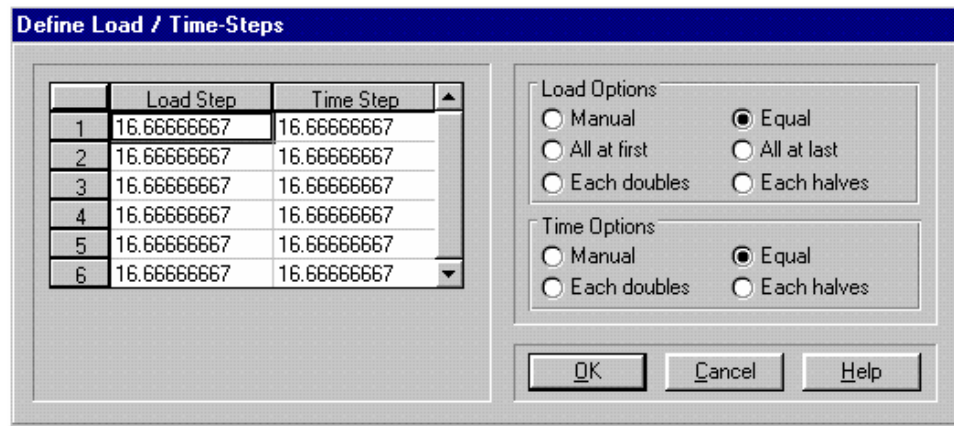
Figure 3-11:Defining Load and Time Steps

> ***Step 4*** Select Equal in the Load Options group box. The Define Load/Time Steps dialogue box should now look like Figure 3-11. Click on OK to return to the Increment Block Information dialogue box, once you have finished.

> ***Step 5*** Click on OK to exit.

3.3.8 Defining Boundary Conditions

Boundary conditions describe to the finite element analysis, the ways in which the mesh is physically restrained from moving. This is achieved by applying fixities to edges and nodes in the mesh.

Once defined, fixities apply in all successive increment blocks. The Fixities dialogue box and the Graphics Window (if View Fixities is toggled on) will only display any fixity that is applied during the <u>current</u> increment block.

For this example, boundary conditions need to be applied to both the in situ stage and to the triaxial loading stage.

The following procedure defines the fixities for the in situ stage.

> ***Step 1*** Ensure that 'In situ' is selected from the increment block list box on the main toolbar.

> ***Step 2*** In the Select menu, click on Edges.

> ***Step 3*** Select the bottom (horizontal) edge of the mesh by clicking on it.

> ***Step 4*** In the Increment menu, click on Fixities... to display the Edge Fixity (Group) dialogue box.

Step 5   Click on the Y displacement fixity option and type "0" into the Start Node and Finish Node text boxes. Now set the mid-side fixity to zero by either pressing the Interpolate button or by typing in "0" directly.

Step 6   The Edge Fixity dialogue box should look like Figure 3-12. Click on OK to exit once you have finished.
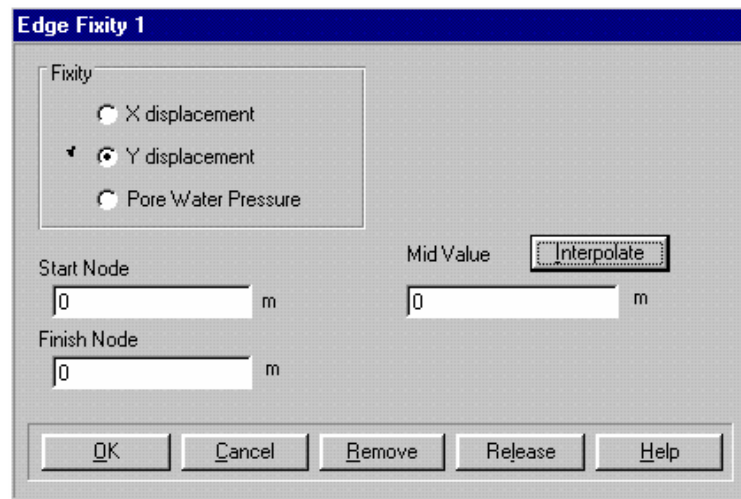
Figure 3-12:Zero Y displacement Fixities

Step 7   In the Select menu, click on Clear Selection to ensure that no edges remain selected (Select Edge is still active). Then select the left-hand (vertical) edge of the mesh by clicking on it.

Step 8   In the Increment menu, click on Fixities... and define the fixity as before except as zero X displacement fixities rather than zero Y displacement fixities.

Step 9   Click on OK to exit once you have finished and click on the left-hand edge of the mesh to de-select it.

Step 10   The Graphics Window should now look similar to Figure 3-13.
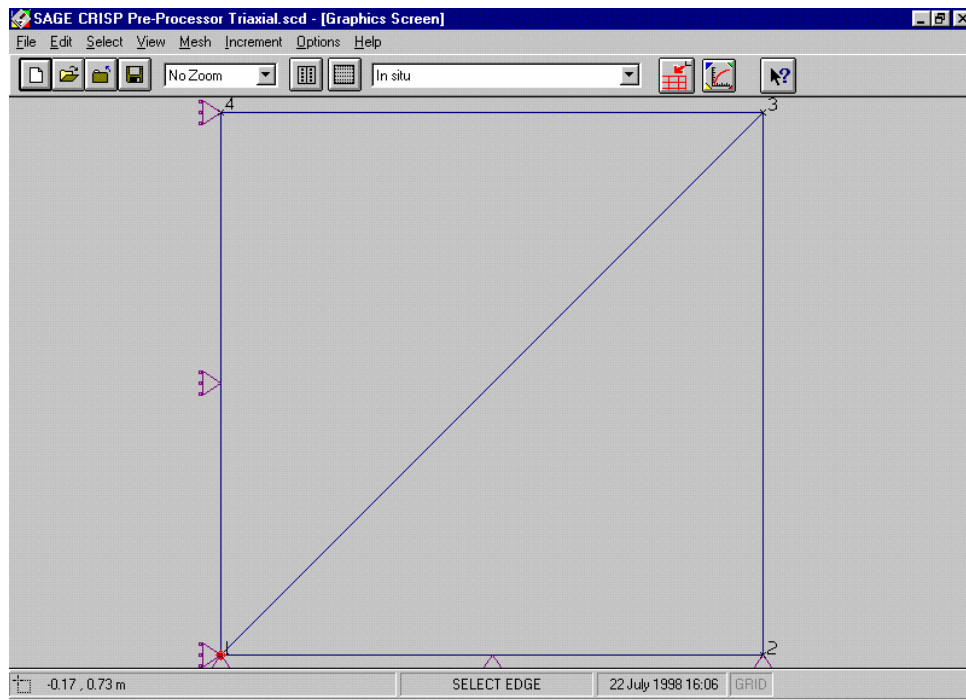
Figure 3-13:Displacement Fixities at the In Situ Stage

In this example the only fixity that alters during the test is the top edge, which undergoes a prescribed displacement of -0.03m. The following procedure defines this fixity in the triaxial loading stage.

> *Step 11*   Select 'Triaxial Loading' from the increment block list box on the main toolbar.

> *Step 12*   Select the top (horizontal) edge of the mesh by clicking on it.

> *Step 13*   In the Increment menu, click on Fixities... to display the Edge Fixity dialogue box.
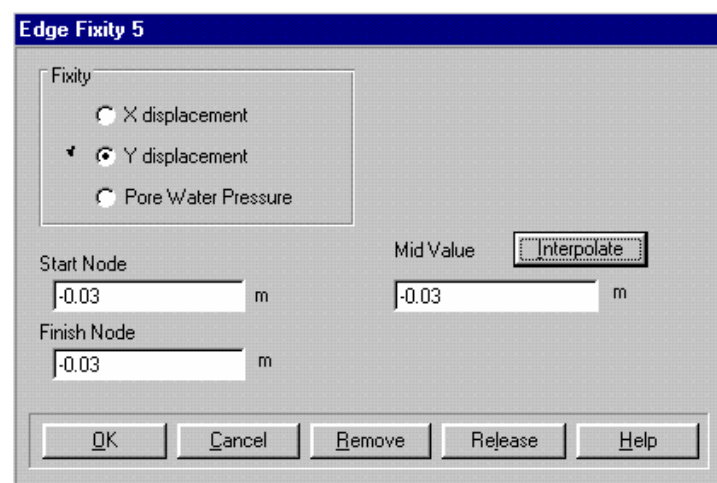
Figure 3-14:Defining a Displacement

> *Step 14*   Click on the Y displacement fixity option and enter the value "-0.03" into the Start Node and Finish Node text boxes. Pressing the Interpolate button automatically enters the Mid Value as -0.03. Alternatively you can enter it manually.

Step 15	The Edge Fixity dialogue box should now look like Figure 3-14. Click on OK to exit once you have finished.

3.3.9   Defining Loads

A load is applied to the right-hand edge of the mesh to simulate the cell pressure. As with fixities, once defined loads apply in all successive increment blocks. The Loads dialogue box and the Graphics Window (if View Loads is toggled on) will only display any load that is applied during the current increment block.

Step 1	Select 'In situ' from the increment block list box on the main toolbar.
Step 2	In the Select menu, click on Clear Selection to ensure no edges are selected.
Step 3	Select the right-hand (vertical) edge of the mesh and the top (horizontal) edge by clicking on them.
Step 4	In the Increment menu, click on Loads... to display the Edge Load dialogue box.
Step 5	Enter a value of 150 kN/m^2 in the Normal Start node and Normal Finish node text boxes. Click on the Interpolate button and the Mid Value will automatically be entered as 150 kN/m^2. Alternatively the value can be entered manually.
Step 6	The Edge Load dialogue box should now look like Figure 3-15. Click on OK to exit once you have finished.
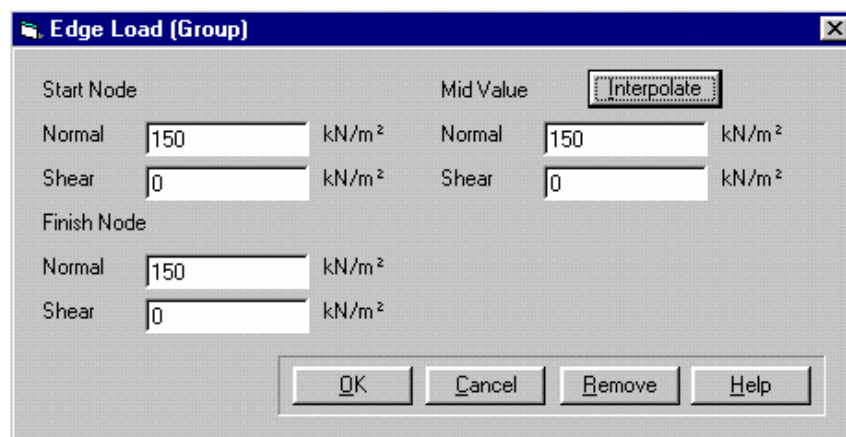
Figure 3-15:Defining a Triaxial Cell Pressure

3.3.10   Scale Factors

Loads are displayed on the Graphics Window as arrows whose length is proportional to the size of the load. In order to optimise the display of these arrows, a scaling factor can be specified that will increase or decrease the length of every load arrow in the mesh by the same proportion.

Customising the display of loads in this way takes three forms - Shear load scale, Normal load scale and Offset from the line to which the loads apply.

Step 1	In the Options menu, click on Default Settings... to display the Defaults dialogue box.
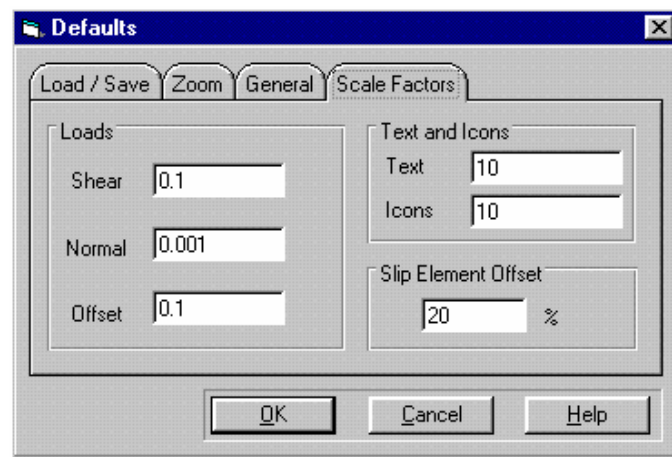
Figure 3-16:The Defaults dialogue box

Step 2   Click on the Scale Factors tab.

Step 3   Enter a value of 0.001 in the Normal text box.

Step 4   The Defaults dialogue box should look like Figure 3-16. Click on OK to exit once you have finished.

Step 5   In the View menu, click on Loads (a tick will appear to the left of the item when it is selected).

**Step 6**   The Graphics Window should now look similar to Figure 3-17.
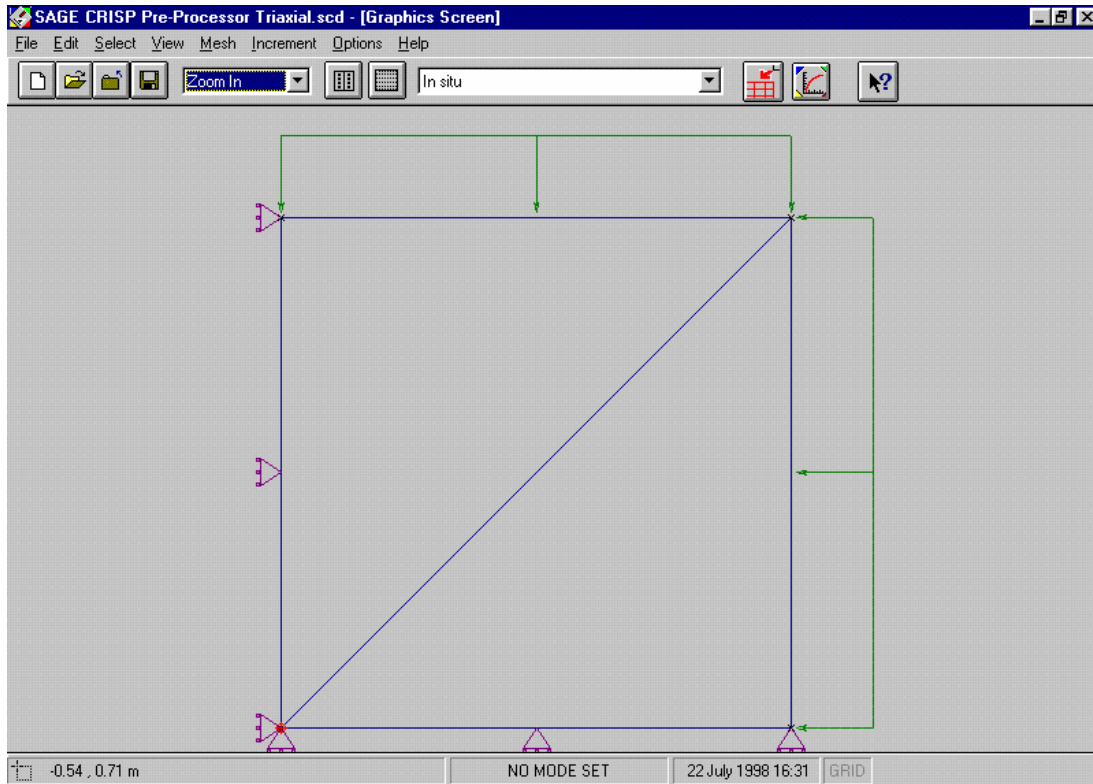
Figure 3-17:Loads and Fixities at In Situ Stage

3.3.11 Running The Analysis

You should save the Project now. This concludes the pre processing for this particular problem so all that remains is for you to run the finite element analysis program.

Step 1   In the File menu, click on Run Analysis... to run the finite element analysis. This displays the Run Analysis dialogue box.
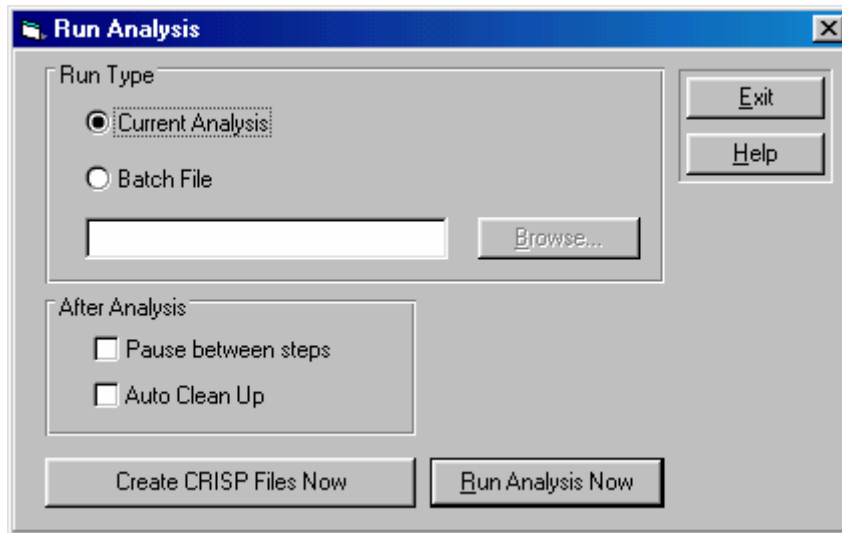
Figure 3-18:The Run Analysis dialogue box

Step 2   Click on Create CRISP Files Now. This will write two files with the extensions .GPS and .MPS to disk. These files contain the input data required by the finite element analysis program.

Step 3   Click on Run Analysis Now. This will task switch you to a DOS box and will run the finite element program using the data files just created.

*Step 4*   When the analysis is finished, you will be returned to Windows and the output from the analysis is converted into a Microsoft Access 2.0 database. A percentage bar informs you as to how long you have left to wait.

Step 5   The Post-Processor is loaded as soon as the conversion is complete and you can begin to look at your results.

3.4 Post Processing

In this section you will be introduced to the following features:

- ■   Duration Graphs

- ■   Displacement Plots

3.4.1 Introduction

If you entered the Post-Processor directly from running the analysis, your Project will already be loaded. However you may also enter the Post-Processor by double-clicking on its icon in the CRISP2D program group in Windows' Program Manager. In this case, you should select Open Project... from the File

menu and, provided the analysis has been successfully run, you will be able to post process the output from your analysis.

For this example, ensure that you have the Parent Project loaded in the Post-Processor.

3.4.2   Duration Graphs

Duration graphs allow you to investigate the variation of output from individual integration points or nodes over the course of a specified range of increments.

Up to 10 data sets (each one representing a different integration point or node) can be viewed on the same graph, but for these examples a single integration point will be used.

Three graphs will be plotted:

- Deviatoric Stress against Mean Normal Effective Stress (q-p')

- Deviatoric Stress against Deviatoric Strain (q-Dev. Strain)

- Total Pore Pressure against Deviatoric Strain (u-Dev.Strain)

The graph of Deviatoric Stress against Mean Normal Effective Stress is created as described by the following procedure:

Step 1   Click on Integration Points and then on Clear Selection in the Select menu. Select an integration point in the top right corner of the mesh by clicking on it.

Step 2   In the Plots menu, click on Duration Graphs... to display the Duration Graph dialogue box.
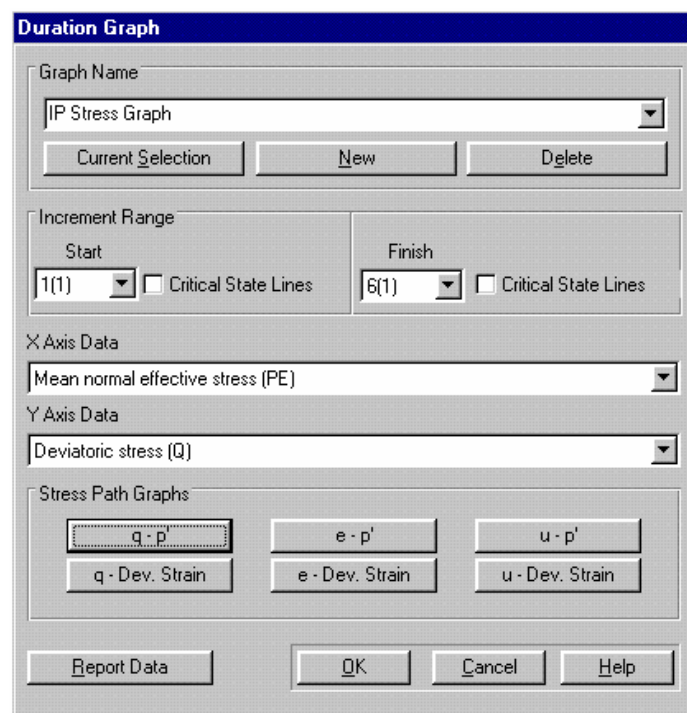

Figure 3-19 :Duration Graph dialogue box

Step 3   Click on the New button to create your graph and change the title to 'IP Stress Graph'.

Step 4   Ensure that the Increment Range starts at 1 and finishes at 6.

Step 5	Click on the q - p' button. This automatically enters Mean Normal Effective Stress (PE) as X Axis data and Deviatoric Stress (Q) as Y Axis data. The Duration Graph dialogue box should now look like Figure 3-19.
Step 6	Click on OK to exit once you have finished. The graph displayed should look similar to Figure 3-20. You can configure the way the graph looks (axis labels, titles and legend) by double-clicking on the graph window and changing the set-up in the Graph Configuration dialogue box.

Figure 3-20::Integration Point Stress Graph

The Integration Point that was used for the first Duration Graph remains selected after the first plot is created and can be used to create further graphs.

The graph of Deviatoric Stress against Deviatoric Strain is created as described by the following procedure:

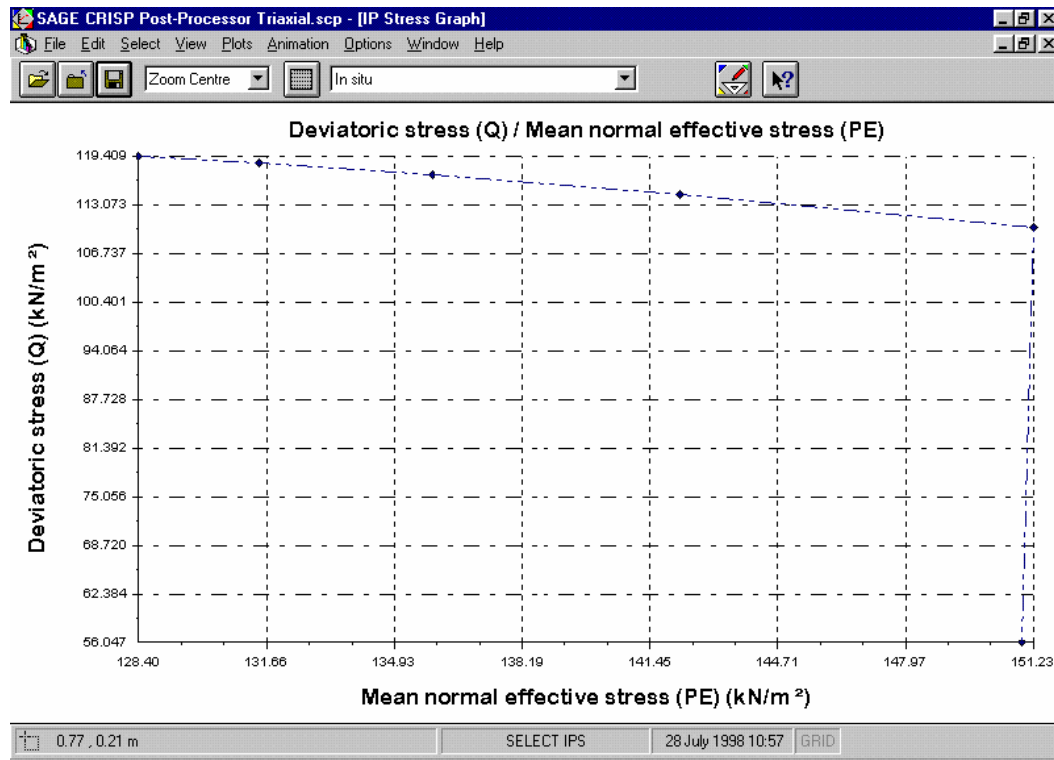|	
Step 7	In the Plots menu, click on Duration Graphs... to display the Duration Graphs dialogue box.
Step 8	Click on the New button to create your graph and change the title to 'IP Stress/Strain Graph'.
Step 9	Ensure that the Increment Range starts at 1 and finishes at 6.
Step 10	Click on the " q - Dev. Strain " button. This automatically enters Deviatoric Strain (EPS) as X Axis data and Deviatoric Stress (Q) as Y Axis data.
Step 11	Click on OK to exit once you have finished. The graph displayed should look similar to Figure 3-21. You can configure the way the graph looks (axis labels, titles and legend) by double-clicking on the graph window and changing the set-up in the Graph Configuration dialogue box.

Figure 3-21:Integration Point Stress/Strain Graph

The third graph of Total Pore Pressure against Deviatoric Strain is created as described by the following procedure:

Step 12   In the Plots menu, click on Duration Graphs... to display the Duration Graph dialogue box.

Step 13   Click on the New button to create your graph and change the title to 'IP Pore Pressure/Strain Graph'.

Step 14   Ensure that the Increment Range starts at 1 and finishes at 6.

Step 15   Click on the " u - Dev. Strain " button. This automatically enters Deviatoric Strain (EPS) as X Axis data and Total Pore Pressure as Y Axis data.

Step 16   Click on OK to exit once you have finished. The graph displayed should look similar to Figure 3-22. You can configure the way the graph looks (axis labels, titles and legend) by double-clicking on the graph window and changing the set-up in the Graph Configuration dialogue box.
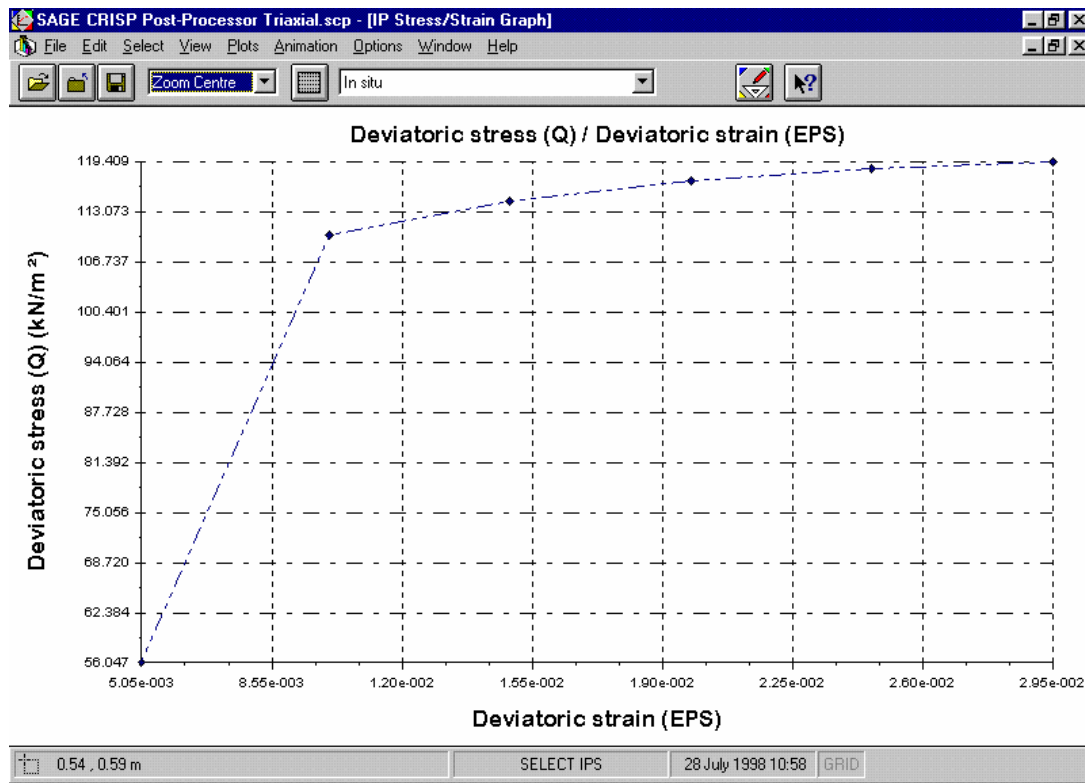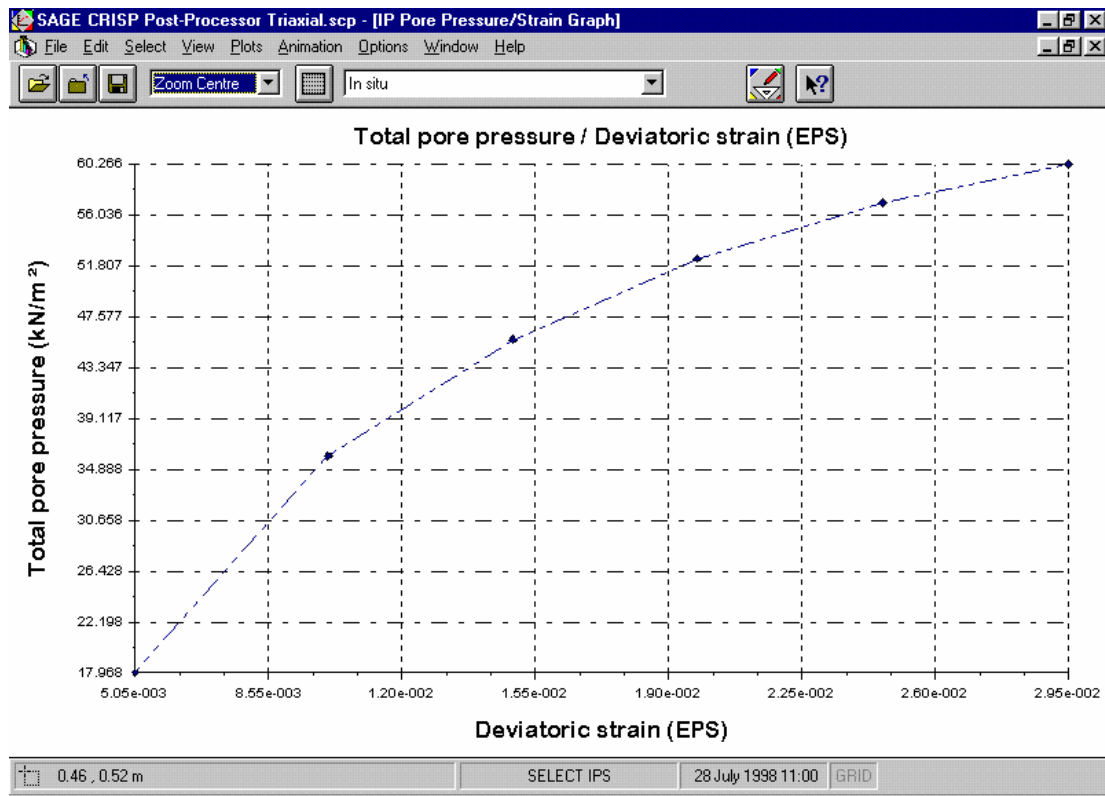
Figure 3-22 : Integration Point Pore Pressure/Strain Graph

3.4.3   Displacement Plots

Displacement plots are drawn directly onto the Graphics Windows. They provide a representation of how the mesh has deformed at any stage in the analysis. You can view either vector displacements or the deformed mesh at any specified magnification.

Step 1   Select Displacement Plots... from the Plots menu. This displays the Displacement Plot dialogue box.

Figure 3-23 : Displacement Plot dialogue box

Step 2   Click on the New button to create your plot and change the title to 'Fully deformed mesh'.

Step 3   Select the Deformed Mesh option.

Step 4   Ensure that Incremental is selected, that the Start increment is 1 and the Finish increment is 6.

Step 5   Enter a Magnification Factor of 5. The Displacement Plot dialogue box will then appear as shown in Figure 3-23.

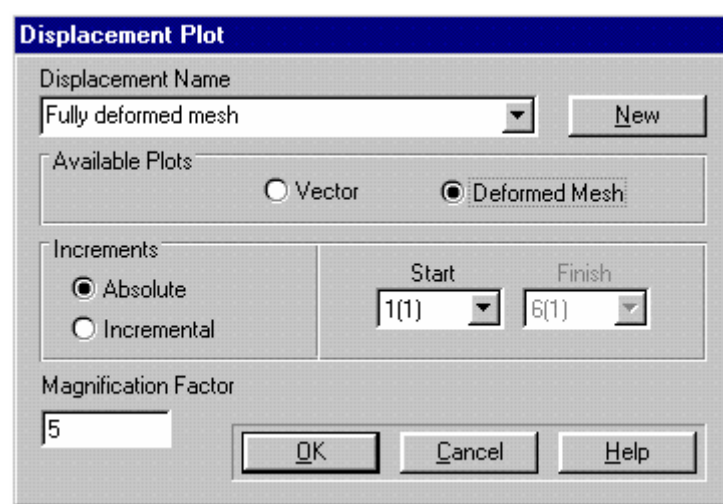*Step 6*   Click on OK to exit once you have finished. The plot will be displayed on the Graphics Window. Once a zoom window has been executed to view the area of interest, the plot will appear as shown in Figure 3-24.
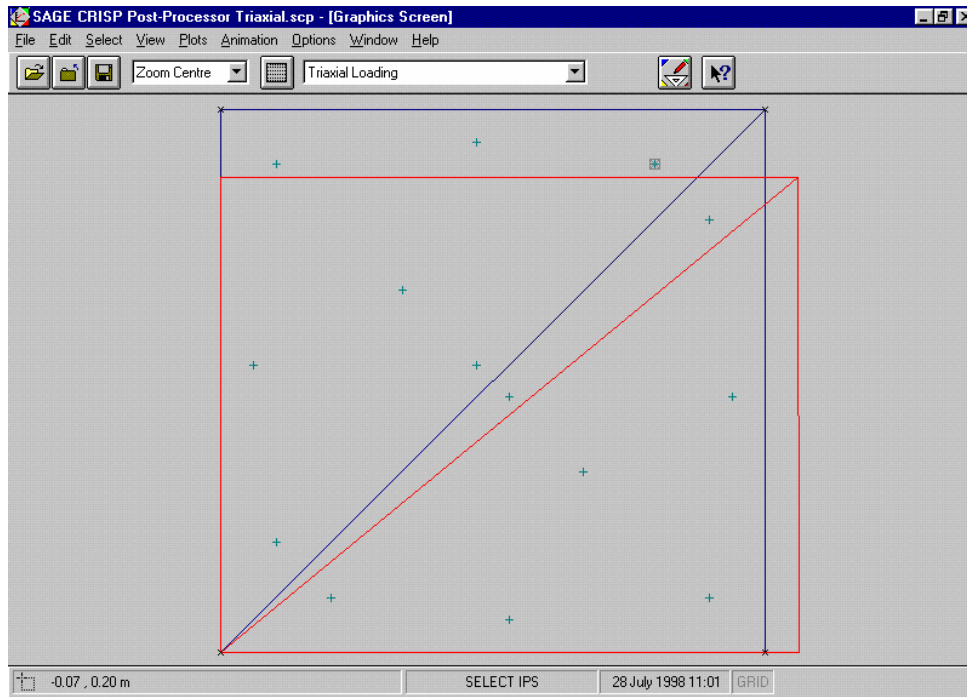
Figure 3-24:Deformed Mesh Plot

3.5 Summary

By the end of this example you will have made use of the following features:

- Cam clay

- Duration graphs

- Displacement plots

3.6 Things To Try Next

- If you look at the analysis summary it can be seen that the yield ratios for increment 2 are unacceptably high. This is a good indicator that the are not enough increments in the analysis. Increase the number of increments to 50 and compare the results. Note - 50 increments will give the theoretical solution.

STRIP FOOTING ON GIBSON SOIL **4**

4.1   Introduction

This example is designed to introduce you to the automatic mesh generation feature (using the structured mesh generator) and to the use of the Parent/Child feature to carry out a parametric study. This feature is used to compare results from the finite element analysis with results predicted by theory. The subject of the example is a strip footing on an elastic soil, with linear variation of shear modulus with depth.

The following steps will be covered:

- ■   Starting The Project

- ■   Defining Units

- ■   Creating The Finite Element Mesh

- ■   Defining A Material Zone

- ■   Assigning Element Properties

- ■   Defining In Situ Stress Conditions

- ■   Defining Increment Blocks

- ■   Defining Boundary Conditions

- ■   Defining Loads

- ■   Scale Factors

- ■   Running The Analysis

- ■   Displacement Plots

- ■   Duration Graphs
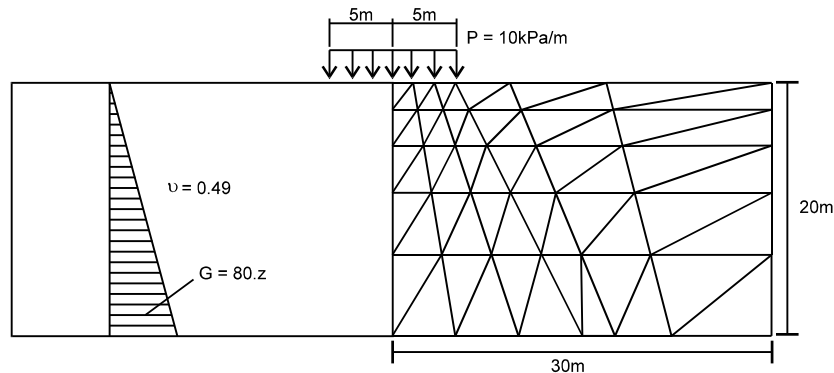
4.2 The Problem

Figure 4-1:Strip Footing Geometry

Figure 4-1 shows the mesh and the soil data for a plane strain calculation of the settlement of a strip load on Gibson soil.

The Gibson soil is an elastic material in which the shear modulus increases linearly with depth. Using z to denote depth, the shear modulus (G) is given by:

$$G = \alpha \cdot z$$

This can be related to Young's modulus using the equation:

$$E = 2G(1 + \nu)$$

The material is then defined by inputting a value of E at the top of the mesh (E_0) of zero and calculating the gradient of variation of Young's modulus with depth.

An exact solution to this problem is only available for the case of a Poisson's ratio of 0.5. In this example a value of 0.49 is used in order to approximate this incompressibility condition.

Taking a value of 80 kN/m²/m for α yields the gradient, m:

$$E = 2 \cdot 80 \cdot z \cdot (1 + 0.49)$$
$$E = 238.4 \cdot z$$
$$m = \frac{E}{z} = 238.4$$

The results from this example show an almost uniform settlement of the soil surface beneath the strip load, as can be seen if you produce a deformed mesh plot or contour vertical displacements using the Post-Processor. The observed settlement is 0.0598 m at the centre of the strip load.

The analytic solution is exact only for an infinite half-space, whereas the solution obtained here is for a layer of finite depth. However, the effect of a shear modulus that increases linearly with depth is to localise the deformations near the surface; it would therefore be expected that the finite thickness of the layer will only have a small effect on the results. The exact solution for this particular problem, as given by Gibson[1], gives a uniform settlement beneath the load of magnitude:

---

[1] Gibson, R.E. (1967), Some results concerning displacements and stresses in a non-homogeneous elastic half-space, Geotechnique, Vol 17, 58-64.

$$\text{Settlement} = \frac{P}{2\alpha}$$

where

$$P = \text{pressure load per unit length}$$

In this case, the exact solution gives a settlement of 0.0625 m and the finite element analysis gives a result of 0.0598 m.

4.3 Pre-Processing

This section describes all of the stages that must be completed using the Pre-Processor in order to run the finite element analysis. It is recommended that you save the Project regularly (by clicking on Save in the File menu).

4.3.1 Starting The Project

This section describes the basic processes that should be completed in advance of creating your mesh. The steps you will follow include:

- Selecting a mesh mode

- Entering Project set-up information

- Entering QA information

- Verifying the base units system

- Activating autosave

- Defining a background grid

- Choosing a background colour

Selecting A Mesh Mode

The first thing that you are asked to do when you start a new Project is to select a starting mesh mode. This means that you must decide whether you want to create the finite element mesh using either the structured or the unstructured automatic mesh generators, or if you want to create it directly.

This example involves generating a finite element mesh from a structured super mesh.

Step 1   On opening the CRISP2D Pre-Processor, select New Project... from the File menu. This displays the Mesh Mode dialogue box:
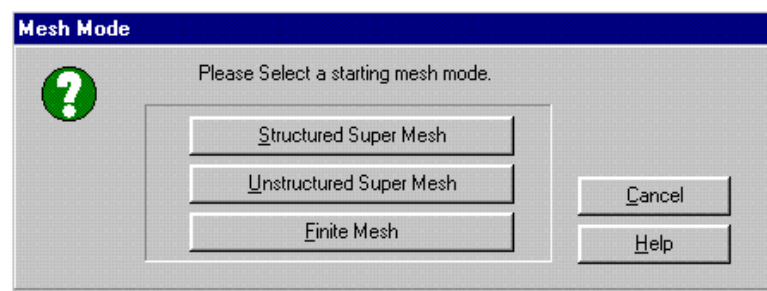
Figure 4-2 :Mesh Mode dialogue box

> *Step 2*   Click on the Structured Super Mesh button. This exits the Mesh Mode dialogue box, sets you up in Structured Super Mesh mode and opens the Project Setup dialogue box.

Setting Up The Project

Setting up a Project involves defining the analysis type, specifying whether or not the mesh is to be composed of cubic strain triangles and setting up various run-time options.

> *Step 1*   In the Domain Type section, click on the Plane Strain option button and click on All Other Elements in the Element Type section.

> *Step 2*   Enter an in situ gravity level of 1 G.

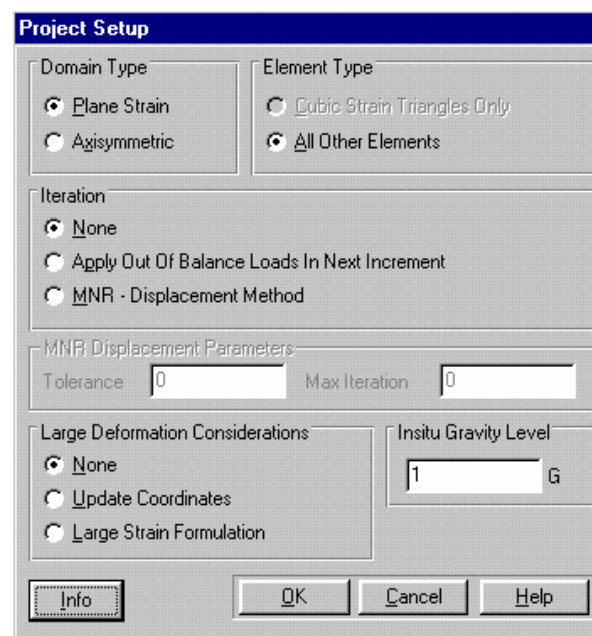> *Step 3*   The Project Setup dialogue box should now look similar to Figure 4-3.

Figure 4-3:Project Setup dialogue box

Entering Quality Assurance Information

> *Step 1*   Click on the Info... button in the Project Setup dialogue box. This displays the Project Information dialogue box. This contains fields for you to enter QA information such as job number, operator name, checker name, etc.
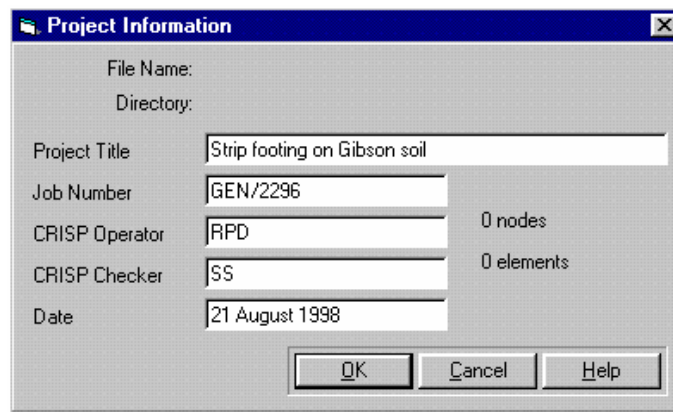
Figure 4-4:Project Information dialogue box

> *Step 2*   Enter data as appropriate. The date is automatically entered but may be manually altered.

> *Step 3*   The Project Information dialogue box should now look similar to Figure 4-4. Click on **OK** to return to the Project Setup dialogue box once you have finished.

> *Step 4*   Click on **OK** to exit.

Defining Units

At this early stage it is useful to ensure that the correct system of base units is being used.

> *Step 1*   In the Options menu, click on **Units...** to display the Units dialogue box.

Figure 4-5:Units dialogue box

> *Step 2*   Use the list boxes to select the appropriate units for Distance, Force, Time and Angle as shown in Figure 4-5. Click on **OK** to exit, once you have finished.

Autosaving

This is an opportune moment to introduce the Autosave feature. If for any reason you lose your Project (it happens!) you need not worry about having to start all over from scratch, or even from when you last

saved. Autosave creates a backup copy of your Project at definable time intervals that will be automatically restored in the event of an unexpected exit.

To activate Autosave, follow this procedure:

Step 1   Click on Default Settings... in the Options menu to display the Defaults dialogue box, and click on the Load/Save tab.
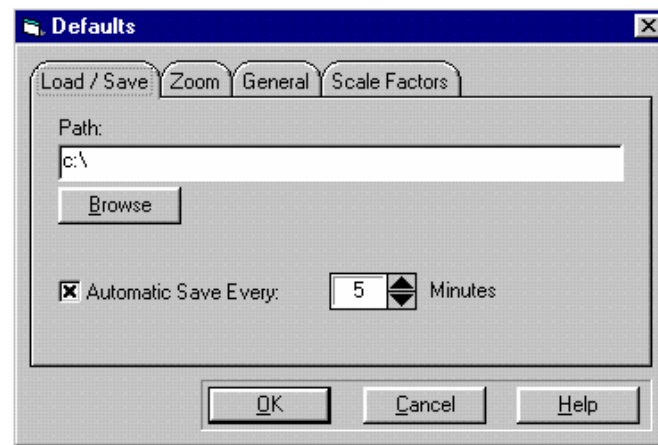

Figure 4-6:Load/Save Defaults dialogue box

Step 2   Click on the Browse button, which displays the Load/Save Path dialogue box.

Step 3   Select the drive and directory where your backup copies will be stored and click OK to exit, once you have finished.

Step 4   Click on the Autosave checkbox. This enables the spin control so that you can define how often your Project is saved.

Step 5   Adjust the spin control to read 5 minutes. The Defaults dialogue box should now look similar to Figure 4-6. Click on OK to exit, once you have finished.

Step 6   In the File menu, click on Save. This brings up a standard Windows File Save dialogue box. CRISP2D Projects are stored with a .SCD extension.

Defining A Background Grid

A Background Grid feature is provided as an aid to drawing regular meshes. Two types of grid are available: an ordinary fixed grid, such as is used in this example and an auto grid. The fixed grid allows you to specify the physical distance between one grid point and the next according to the base units of length that you have chosen (in this case, metres).

Warning   Using the fixed grid, if you zoom out too far the screen will become completely overcrowded with grid points. Any actions you then attempt to make will be very slow because of the shear volume of data being displayed on the Graphics Window.

The Autogrid feature is designed to avoid this problem. Instead of showing grid points a set distance apart it allows you to specify the maximum number of points you want to see on the screen in the X and Y directions. A scale is then displayed at the borders of the Graphics

Window to help you keep track of the physical distance that separates these points.

Step 1   Click on Grid... in the Options menu or double click on the word 'GRID' at the bottom of the Graphics Window. This displays the Grid Options dialogue box.
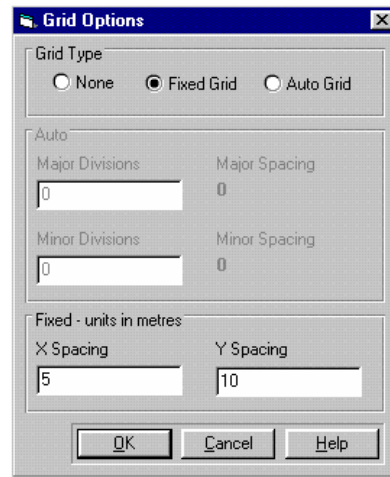
Figure 4-7 : Grid Options dialogue box

Step 2   Select Fixed Grid and enter the X spacing as 5 and the Y spacing as 10.

Step 3   The dialogue box should now look like Figure 4-7. Click on OK to exit, once you have finished. The Graphics Window will then display a grid of dark grey dots.

Changing The Background Colour

The fixed grid can be difficult to see if the background of the Graphics Window is grey, so it is useful to be able to change this colour. To change the background colour, follow this procedure:

Step 1   Click on Colour in the Options menu.

Step 2   Click on white in the background selection frame

Step 3   Click on OK to exit.

4.3.2   Creating A Mesh Using The Structured Mesh Generator

The Super Mesh provides a simple method of generating large numbers of finite elements. In this example 60 finite elements are automatically created from a super mesh comprising just 2 super elements. This saves an enormous amount of time and effort.

This section covers all of the stages you must have completed in order to automatically generate the finite element mesh. The steps you will follow include:

■   Drawing A Super Mesh

■   Defining Edge Gradings

■ Defining A Material Zone

■ Assigning Element Properties

■ Generating The Finite Element Mesh

Drawing The Super Mesh

The mesh of Super Elements is drawn directly onto the Graphics Window using a combination of mouse point-and-click actions and keystrokes. These combinations are summarised below:

■ To position the vertex of a Super Element you must hold the CTRL key down and left click on the Graphics Window at the point where you wish to locate the vertex. These vertexes are termed *Super Nodes*.

■ By holding down both SHIFT and CTRL when you left click, the cursor snaps to the nearest grid point and creates a new Super Node. This facilitates the design of rectilinear meshes.

The following procedure should be strictly followed in order to create the Super Mesh:

Step 1   In the Mesh menu, click on Create Super Elements. The fact that you can now create elements is indicated by the mode box in the status bar at the bottom of the screen.

Step 2   Select 'Window' from the Zoom list box on the main toolbar.

Step 3   With the cursor located over a point about 3/4 of the way down the screen and 1/6 of the way in from the left (over the top left corner of the black box in Figure 4-8), press the <u>right</u> mouse button. Holding the right button down, move the mouse to size the zoom window. Release the button when you have highlighted the area you wish to magnify. The Graphics Window should briefly look similar to Figure 4-8, before you release the button. Note that the zoom window may look different depending on your default configuration.
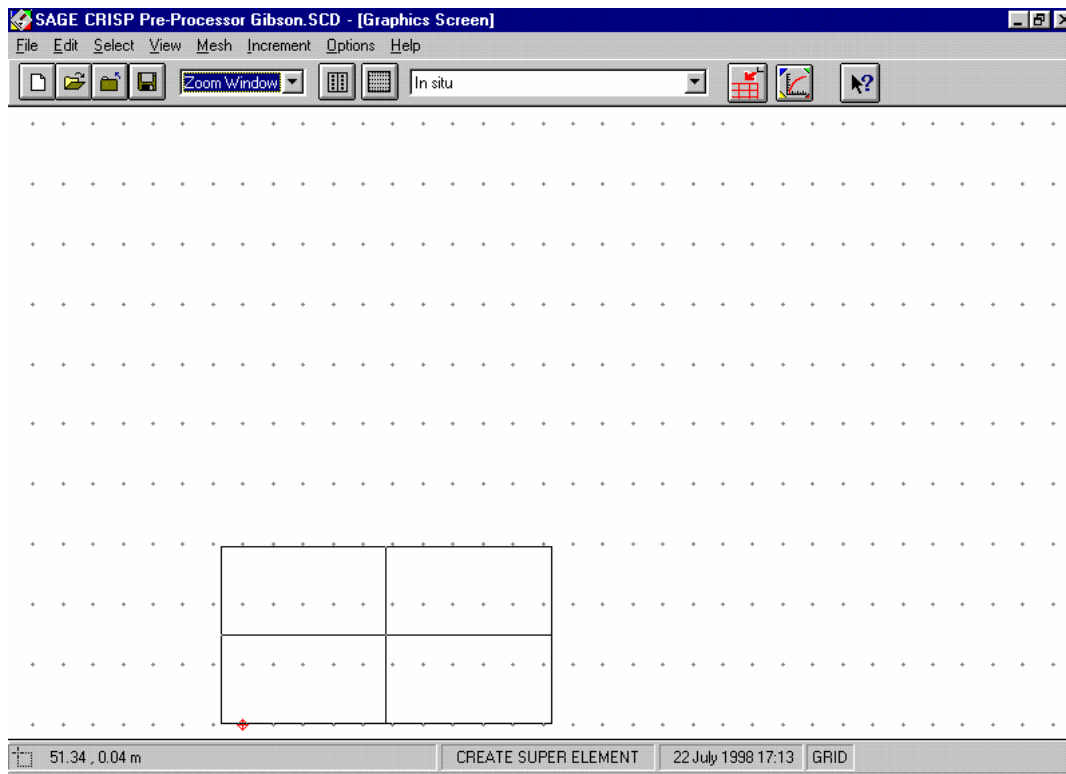
*Figure 4-8:Using Zoom Window to Move Around the Graphics Window*

Step 4   Whilst holding down both the CTRL and the SHIFT keys, click close to the origin (indicated by the co-ordinates on the status bar) to create the first node at (0 , 0).

Step 5   Continue clicking on the Graphics Window to place the vertex nodes of the Super Elements. Figures a to j depict the entire process of creating the Super Mesh. Remember to <u>hold down</u> CTRL and SHIFT when you are creating new nodes (Figures a, b, c, d, g and h), but to **<u>release</u>** them when you wish to use an existing node (Figures e, f, i and j).

e.

f.

g.

h.

i.

j.
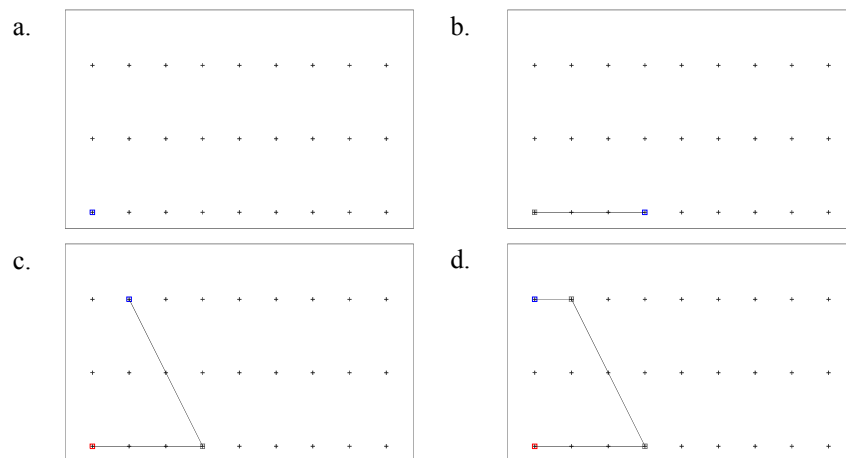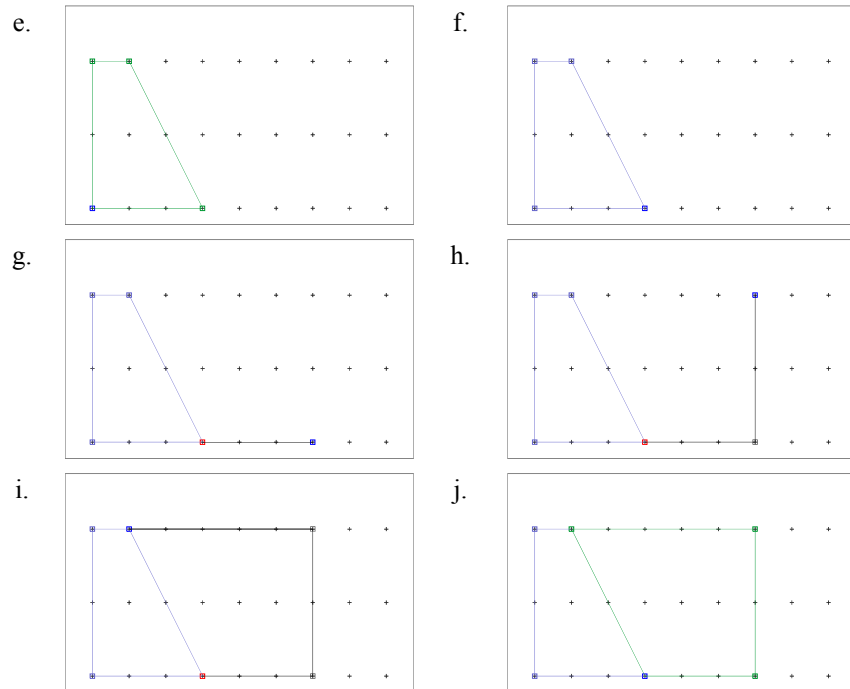
Step 6   In the Edit menu, click on Commit Super Element Creation. This fixes the elements, confirming that they are correctly created. The elements will change colour when they committed.

Step 7   To verify that you have correctly positioned all of the nodes, click on Super Node List... in the Mesh menu. This displays the Super Node List dialogue box.
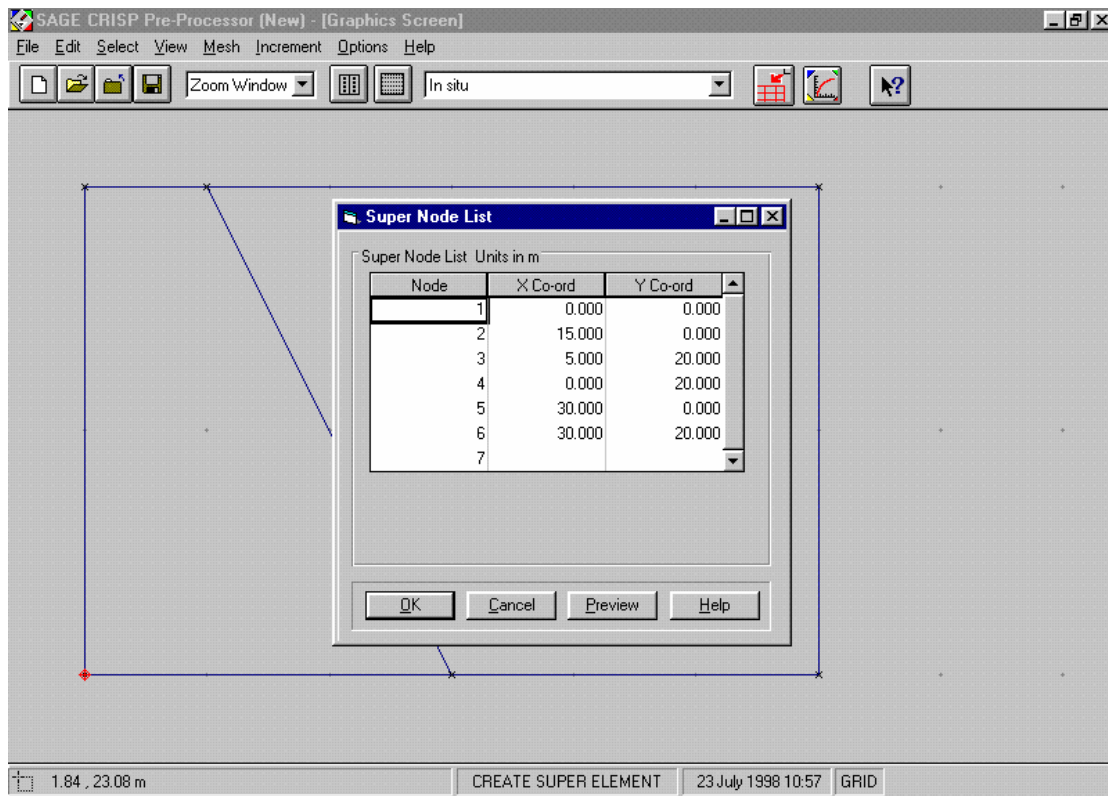
Figure 4-9:Co-ordinates of Super Nodes

Step 8   If any of your entries are different to those shown in Figure 4-9, click on the number(s) you need to change and type in your amendment(s). You can check your progress by clicking on the Preview button.

Step 9   If you have made any changes you should exit by clicking on OK. Otherwise you can click on Cancel or press ESC to exit.

Defining Edge Gradings

Edge Gradings define how the Super Elements will be divided up into finite elements by the mesh generator. In general, better results will be obtained by using smaller finite elements in the parts of the mesh that experience the most rapid and/or the largest changes in stress, during the course of the analysis.

Step 1   In the Select menu, click on Super Edges.   Select the top left-hand (horizontal) edge of the mesh by clicking on or close to it.

Step 2   In the Mesh menu, click on Super Edge Grading... to display the Super Edge Gradings dialogue box.

Figure 4-10:Defining Edge Gradings for the Top Left-Hand Edge

Step 3 Enter '3' into the Number of Divisions text box and click on the Equal Spacing option. Weighting values will be automatically entered and the dialogue box should now look like Figure 4-10.

Step 4 Click on OK to exit, once you have finished.

Step 5 In the Select menu, click on Clear Selection, or click again on the selected edge to de-select that edge. Click on the top right-hand (horizontal) edge. When an edge is selected, one or other of that edge's nodes is also highlighted. Ensure that the highlighted node is the one on the left-hand end of the edge by clicking closer to this end than the other when you select the edge. If the wrong node is highlighted, your gradings will appear the wrong way around.

Step 6 In the mesh menu, click on Super Edge Grading... to again display the Super Edge Gradings dialogue box.

Figure 4-11:Defining Gradings for the Top Right-Hand Edge

Step 7   Enter a value of '3' into the Number of Divisions and Factor (k) text boxes.

Step 8   Click on the End 1 is k times End 2 option. The dialogue box should now look like Figure 4-11.

Step 9   Click on OK to exit, once you have finished.

Step 10   In the Select menu, click on Clear Selection.

Step 11   Select the left-hand vertical edge, ensuring that the node at the top is highlighted.

Step 12   Apply edge gradings to this edge, such that there are 5 divisions, with the division at the bottom of the edge being 3 times larger than the division at the top. The dialogue box should look like Figure 4-12.

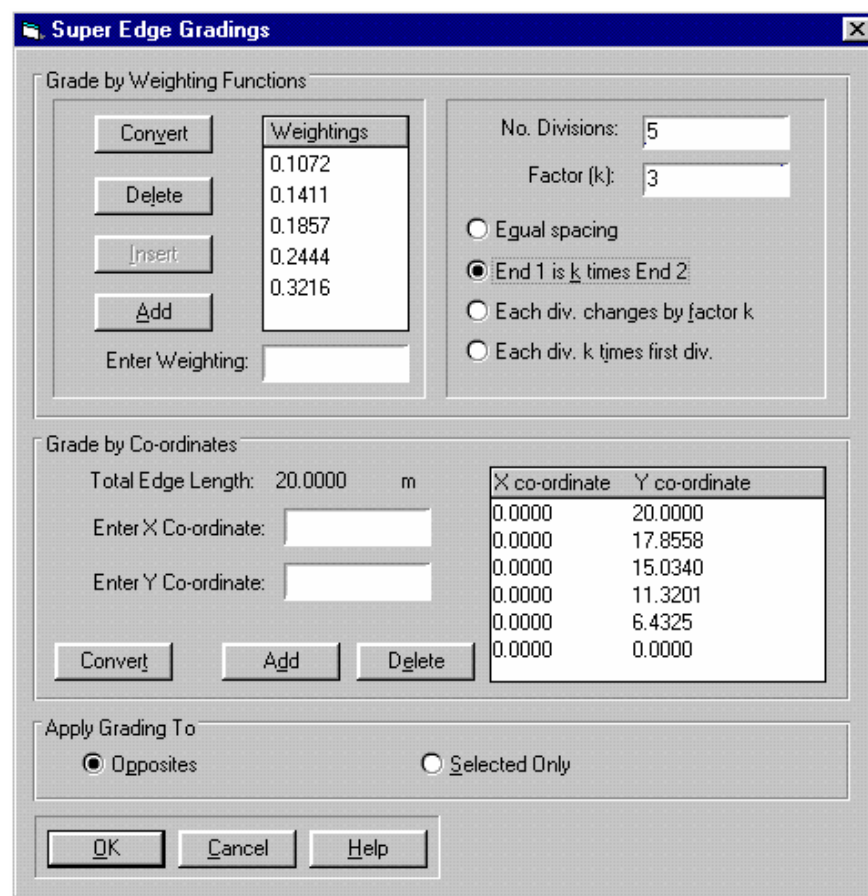*Figure 4-12:Defining Gradings for the Left-Hand Vertical Edge*

Step 13   Click on OK to exit, once you have finished.

Step 14   Opposite edges of each element are always graded in identical proportions, so by defining gradings for just these three edges, all edges in the mesh have been graded. This can be seen from the Graphics Window, which should now look similar to Figure 4-13.
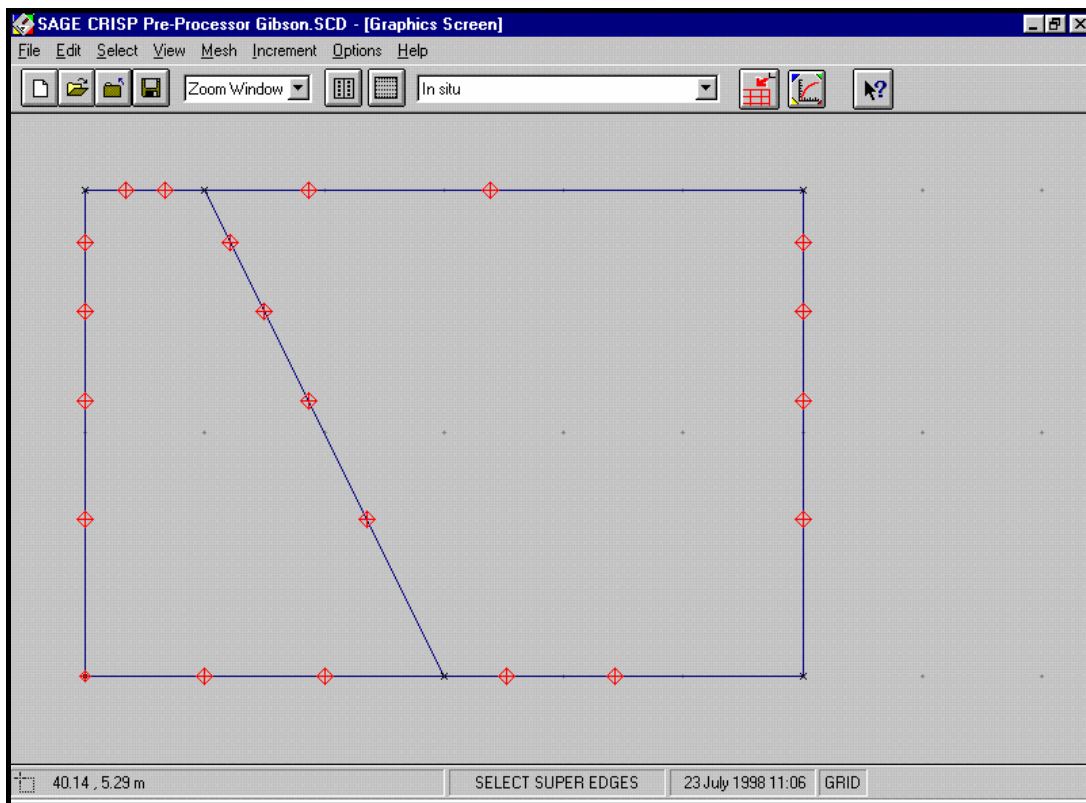
Figure 4-13:Fully Assigned Edge Gradings

Defining A Material Zone

A Material Zone contains information about the soil model and associated soil parameters that are being used to model particular regions of the mesh.

It is necessary to define the number of material zone(s) at this point in order to be able to allocate them using the Element Properties dialogue box (the next stage).

Once you have set up a zone, the soil model and parameters may be entered or changed at any time prior to the analysis being run.

> *Step 1* Select Material Properties... from the Mesh menu. This displays the Material Properties dialogue box.

Figure 4-14:Material Properties dialogue box

> ***Step 2*** Enter the name 'Gibson soil' and select 'Elastic (Linear variation with depth)' from the Soil Model list box. Material zones are referenced by name elsewhere in the Project, so it is good practice to give each zone a meaningful title here.

> ***Step 3*** Enter the following material property values:

Property	Value	Units
E_0	0	kN/m^2
Y_0	20	m
m	238.4	$kN/m^2/m$
ν	0.49	None
K_w / γ_w	0	kN/m^3
γ_{bulk}	20	kN/m^3
k_x	0	m/sec
k_y	0	m/sec

> ***Step 4*** The Material Properties dialogue box should now look similar to Figure 4-14. Click on OK to exit, once you have finished.

> ***Tip*** Although values for the soil properties have been defined here, it is possible to delay defining them right up until just before you run the analysis.

Assigning Element Properties

Every element in the mesh has two properties associated with it:

■   Finite element type, and

■   Material zone

These properties are assigned to finite elements in the following manner:

Step 1   In the Select menu, click on Domain Super Elements and select both elements by clicking on them.

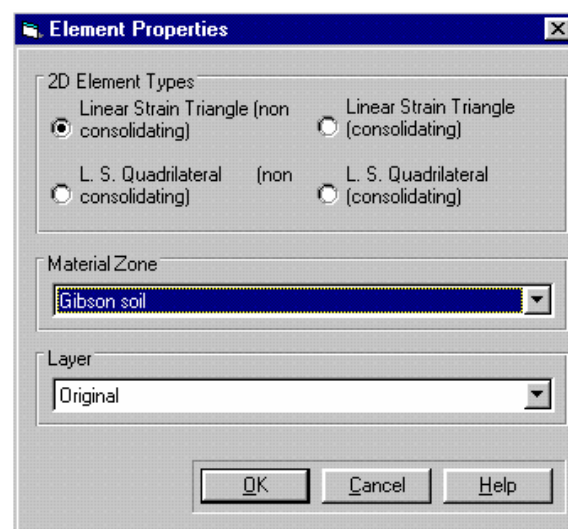*Step 2*   In the Mesh menu, click on Element Properties... to display the Element Properties dialogue box.

Figure 4-15:Defining Element Type and Zone

Step 3   Select the Linear Strain Triangle (non consolidating) option.

Step 4   Select 'Gibson soil' from the Material Zone list box.

Step 5   The Element Properties dialogue box should now look like Figure 4-15. Click on OK to exit, once you have finished.

Generating The Finite Element Mesh

You are now at a stage where you can generate the finite element mesh from your Super Mesh.

Step 1   In the Mesh menu, click on Generate Finite Element Mesh. This will first check that you have correctly graded every edge and that you have assigned a generated finite element type to each super element. It will then spend a short time performing the automatic generation of the finite element mesh.

Step 2   Provided you have completed the above stages correctly, the finite element mesh will be displayed as shown in Figure 4-16. The menu bar will now allow you access to the additional options available in Finite Element Mesh mode.
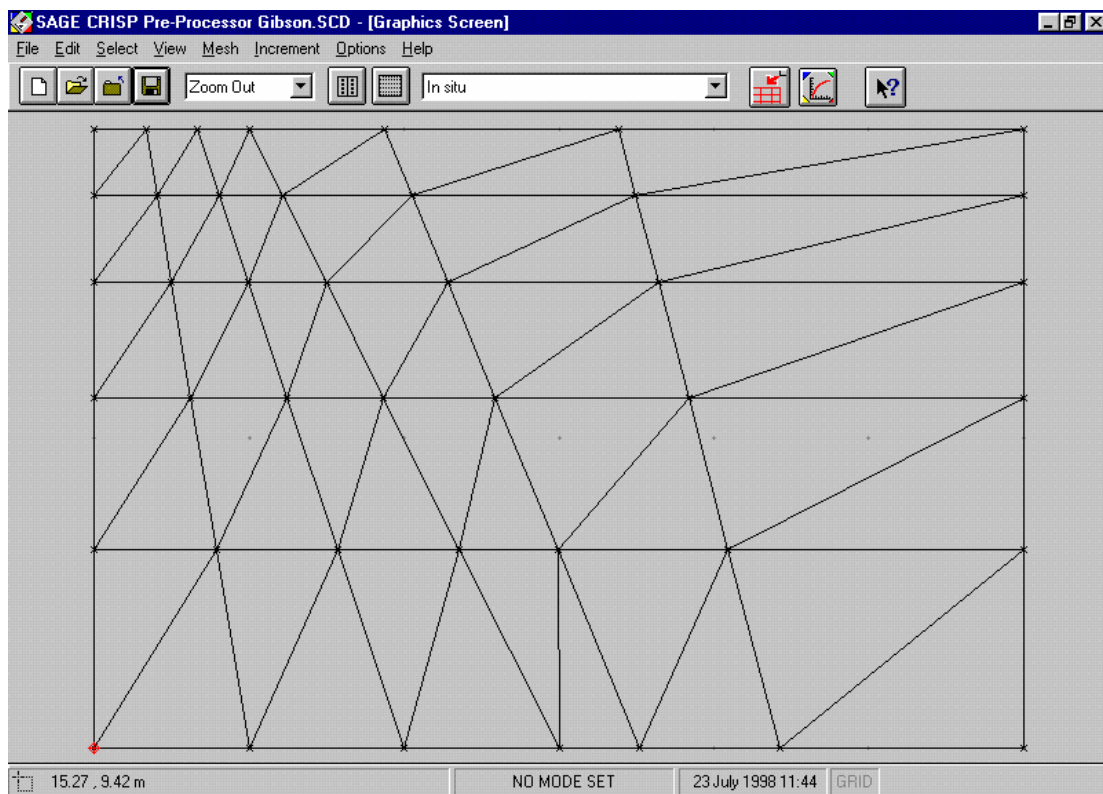
Figure 4-16:Automatically Generated Finite Element Mesh

4.3.3 Defining In Situ Stresses Using The In Situ Stress Convertor
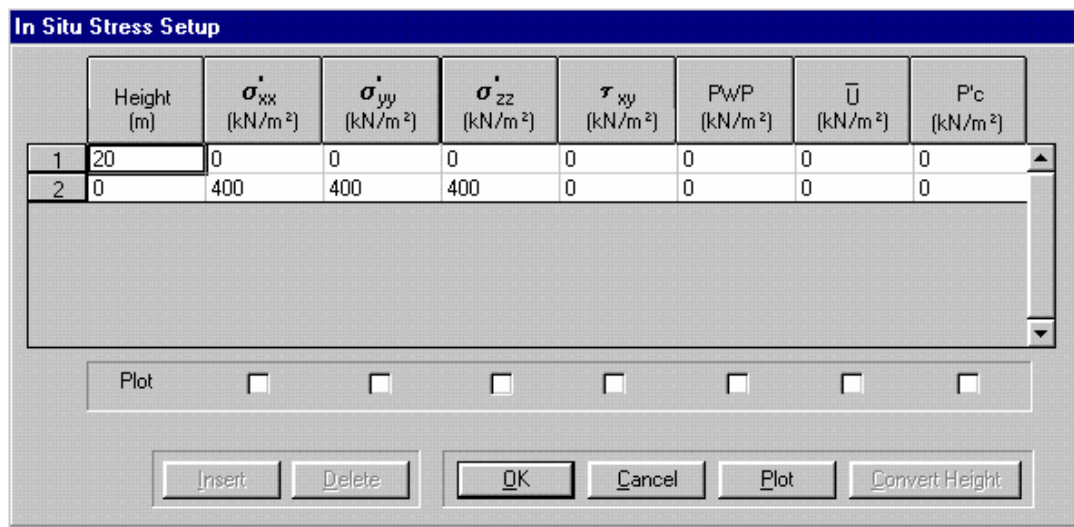
In situ conditions are defined in terms of stresses at different elevations within the mesh. These stresses are derived from soil parameters that are obtained (almost) directly from lab testing of your particular soil.

The derivations can either be performed by hand, in which case you enter the values directly into the In Situ Stress Setup dialogue box, or you can use the Stress Conversion Parameters dialogue box to automatically perform the derivation for you.

The latter method requires you first to select the reference elevation for which you wish to calculate the stresses. You can then input the soil parameters and perform the automatic derivation.

>   *Step 1*   In the Increment menu, click on Define In Situ Stresses... to display the In Situ Stress Setup dialogue box.

Figure 4-17:In Situ Stress Setup dialogue box

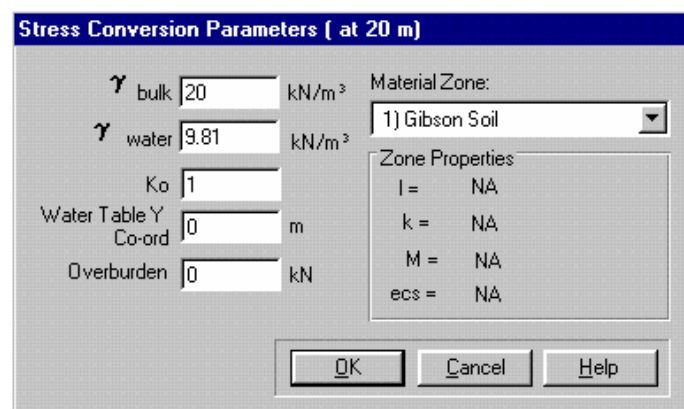*Step 2*   Select the second row of the in situ stresses grid by clicking on the row number (to the left of the Height column). This enables the Convert Height... button, which you should click on to display the Stress Conversion Parameters dialogue box.

Figure 4-18:Using the In Situ Stress Convertor

Step 3   Enter values as shown in Figure 4-18. Click on OK to exit, once you have finished. The stress convertor takes these parameters , converts them into stresses and enters them into the highlighted row in the in situ stresses grid.

Step 4   The In Situ Stress Setup dialogue box should now look like Figure 4-17. Click on OK to exit.

4.3.4   Defining Increment Blocks

Increments are the units by which the finite element program advances the analysis. In each increment, the effect of applied loads and boundary conditions is calculated and the accumulated strains and pore water pressures are updated. Increments are grouped together in blocks which then model particular events in the analysis.

This analysis requires only one increment block during which the strip load is applied to the mesh. The block comprises 10 increments.

Step 1   In the Increment menu, click on Define Increment Block Parameters... to display the Increment Block dialogue box.
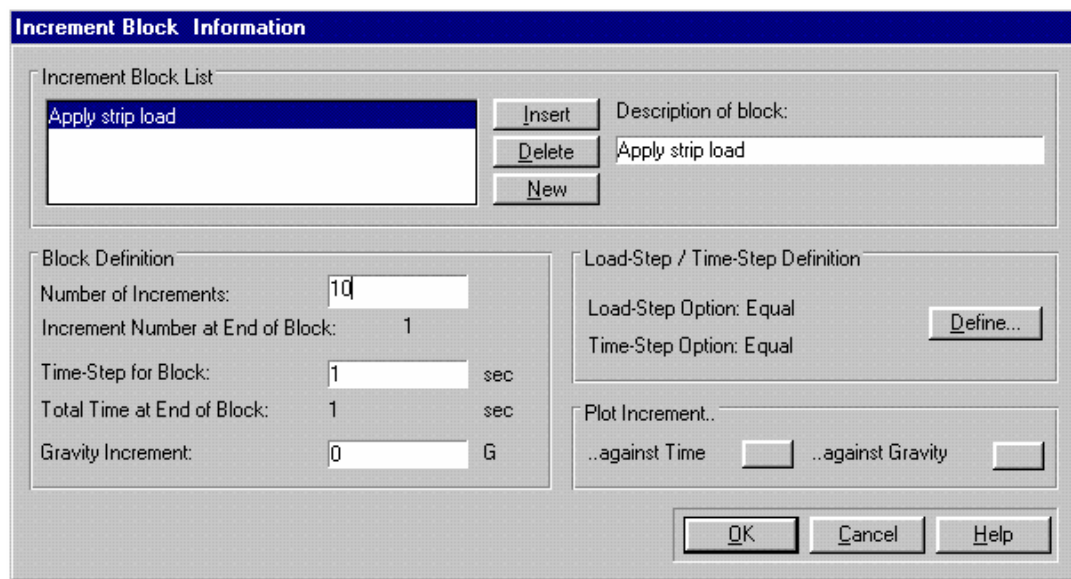
Figure 4-19 :Defining Increment Blocks

Step 2   A default increment block is created when you enter this box for the first time. In the Description of block text box, change the name of the block from 'Increment Block (untitled)' to 'Apply strip load'.

Step 3   Allocate ten increments to the block.

Step 4   Since this is a non-consolidation analysis, any time-step specified for the block will be ignored.

Step 5   Click on OK to exit.

4.3.5   Defining Boundary Conditions

Boundary conditions describe to the finite element analysis the ways in which the mesh is physically restrained from moving. This is achieved by applying fixities to edges and nodes in the mesh.

Once defined, fixities apply in all successive increment blocks. The Fixities dialogue box and the Graphics Window (if View Fixities is toggled on) will only display any fixity that is applied during the current increment block.

In this example, the left and right edges are restrained horizontally and the base is restrained both horizontally and vertically.

The following procedure defines fixities for the in situ stage:

Step 1   Select 'In situ' from the Increment Block list box on the main toolbar.

Step 2   In the Select menu, click on Edges.

Step 3   Select all of the vertical edges on the left and right-hand sides of the mesh by clicking on them. All of these edges have zero X direction fixities, so they can be defined simultaneously.

Step 4   In the Increment menu, click on Fixities... to display the Edge Fixity dialogue box.
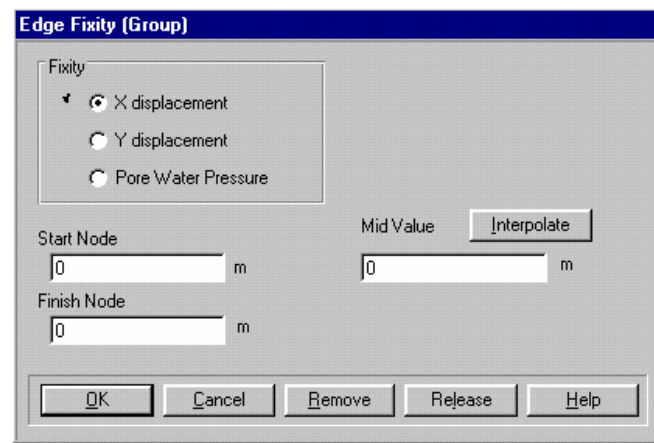
Figure 4-20:Defining Edge Fixities

Step 5   Click on the X displacement option button.

Step 6   Specify X direction fixities by typing '0' into the Start Node and Finish Node text boxes. Now set the Mid-Value fixity to zero by either pressing the Interpolate button or by typing in '0' directly.

Step 7   The Edge Fixity dialogue box should now look like Figure 4-20. Click on OK to exit, once you have finished.

Step 8   The Graphics Window should look similar to Figure 4-21. If no fixities are visible, click on Fixities in the View menu (a tick will appear to the left of the item when it is active).
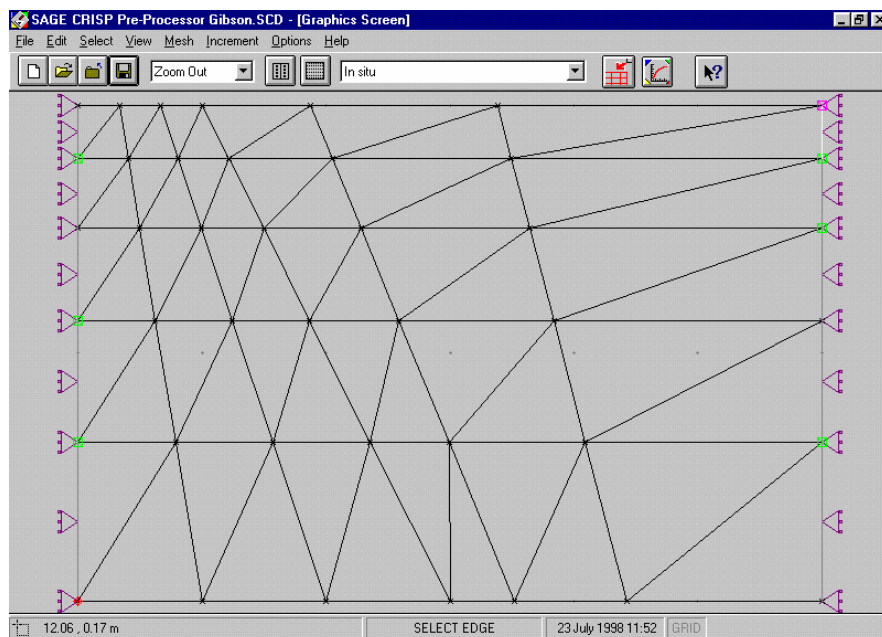
Figure 4-21:Fixities Defined for the Left and Right Vertical Sides

Step 9   Click on Clear Selection in the Select menu to ensure that no edges are selected (note that Select Edge mode remains active). Then select the bottom (horizontal) edges of the mesh by clicking on them.

Step 10   In the Increment menu, click on Fixities... and define zero X displacement fixities as before.

Step 11   Click on the Y displacement option button.

Step 12   Specify Y direction fixities of zero in the Start Node, Finish Node and Mid-Value text boxes.

Step 13   Click on OK to exit, once you have finished. The Graphics Window should look similar to Figure 4-22.
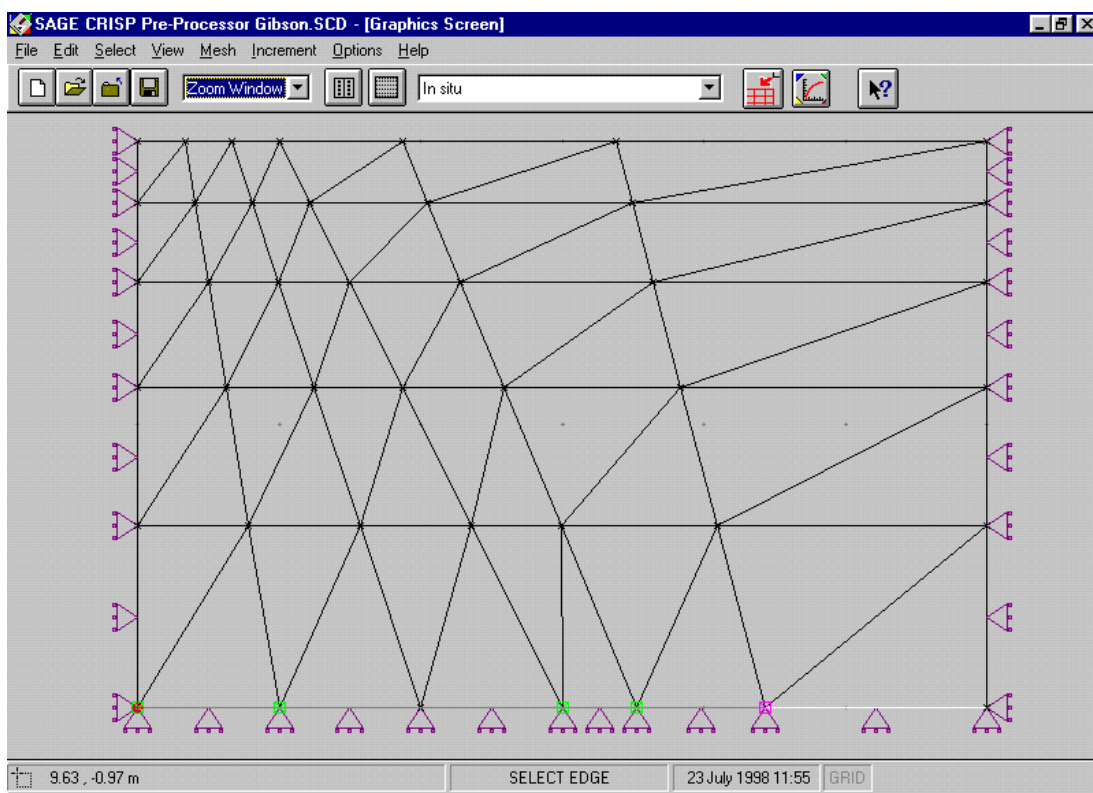
Figure 4-22:In Situ Fixities Fully Defined

4.3.6   Defining Loads

A load is applied to edges along the left-most 5m of the top of the mesh to simulate the strip footing. As with fixities, once defined, loads apply in all successive increment blocks. The Loads dialogue box and the Graphics Window (if View Loads is toggled on) will only display any load that is applied during the current increment block.

The sign conventions that apply to pressure loads are:

Positive loads are displayed pointing towards the bottom left corner of the Graphics Window.

This convention is enforced as follows:

■   Downwards is positive;

■   Right-to-left is positive;

■   When applied to inclined edges, the downwards positive criterion takes precedence over right-to-left;

■   When applied to vertical edges, right-to-left is positive.

For further clarification, see Section 5.9.5 in the User Guide. The following procedure defines loads for this problem.

Step 1   Select 'Apply strip load' from the Increment Block list box on the main toolbar.

Step 2   Click on Clear Selection in the Select menu to ensure no edges are selected.

Step 3   Select the edges of the mesh that correspond to the top edge of the first super element you created. On the finite element mesh these are the first three edges along horizontally from the top left-hand corner.

Step 4   In the Increment menu, click on Loads... to display the Edge Load dialogue box.

Step 5   Enter a value of 10 kN/m^2 in the Normal Start node and Normal Finish node text boxes. Click on the Interpolate button and the Normal Mid Value will automatically be entered as 10 kN/m^2. Alternatively the value can be entered manually.

Step 6   The Edge Load dialogue box should now look like Figure 4-23. Click on OK to exit, once you have finished.
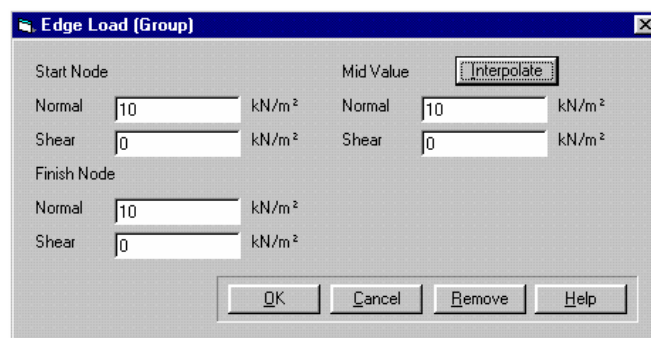
Figure 4-23:Defining the Strip Load

Step 7   Once you have finished, the Graphics Window should look something like Figure 4-24. Here, the loaded region has been magnified using the Zoom Window feature. The appearance of the load arrows can be modified by changing the default scale settings (see section 4.3.7).
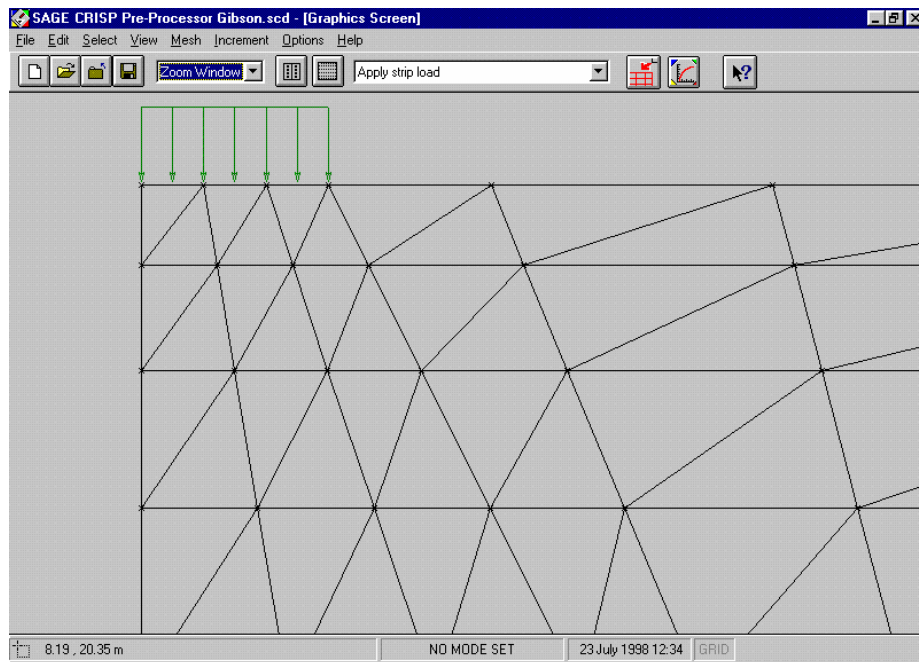
Figure 4-24:Strip Load Applied at the Top of the Mesh

4.3.7 Scaling The Displayed Load Arrows

Loads are displayed on the Graphics Window as arrows whose length is proportional to the size of the load. If the load arrows on the screen look too long or too short, you can adjust this using the Scale Factors function.

Step 1   In the Options menu, click on Default Settings... to display the Defaults dialogue box.
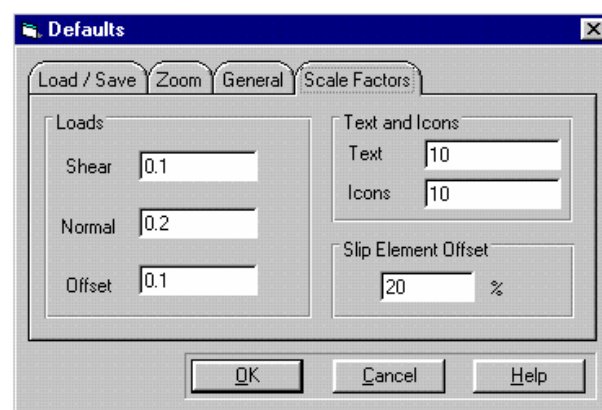
Figure 4-25:Scale Factors for Strip Footing Loads

Step 2   Click on the Scale Factors tab.

Step 3   Enter values of 0.2 in the Normal text box and 0.1 in the Offset text box.

Step 4   The Defaults dialogue box should now look similar to Figure 4-25. Click on OK to exit, once you have finished.

4.3.8   Setting Up A Parent/Child Analysis

In order to be able to compare results from two separate runs of the finite element analysis, the Project must be turned into a Parent Project and a number of Child Projects.

A Child Project is initially a copy of the original (Parent) Project. The mesh cannot be altered but features such as material properties and number of increments may be varied.

Due to the choice of the elastic soil model, the number of increments used should not increase the accuracy of the analysis. To demonstrate this a Child Project is made with just 5 increments (remember that the Parent has 10 increments).

Step 1   The Project must be saved before any Child Projects can be created. Click on Save Project in the File menu to save your Project to disk.

Step 2   In the File menu, click on Parent/Child... to display the Parent/Child dialogue box.

Step 3   Enter the name 'Stripft2' into the New Child Name text box. This will be the filename of the Child Project and is therefore restricted to 8 characters.

Step 4   Click on the ⟩⟩ button to assign it to the family tree. This creates an identical Project to the one you currently have open with the filename you just assigned to it. The Parent/Child dialogue box should look similar to Figure 4-26.

Figure 4-26:Setting Up A Parent-Child Relationship

Step 5   Make sure that the new child Project is highlighted in the Family Tree list and click on Load. This will exit the Parent And Children dialogue box and load the new child into the Pre-Processor.

Step 6   Select Define Increment Block Parameters... from the Increment menu and change the number of increments in the 'Strip Loading' block to 5.

Step 7   This concludes the set-up of the child analysis.

4.3.9   Running The Analysis

The Project should now be saved. Provided you have followed all of the steps described above, preparation of the Project using the Pre-Processor is complete.

Since there are two Projects to run, it is convenient to use the Batch Run utility. This allows you to line up a series of Projects that will be run consecutively. The sequence of events is outlined below.

- Creating The Output Files

- Setting Up A Batch Run

- Performing The Analysis

Creating Output Files

The first stage is to create the output files for the two Projects. The following procedure describes how this is done (it assumes that you still have the Child Project loaded).

Step 1   In the File menu, click on Run Analysis... to display the Run Analysis dialogue box.
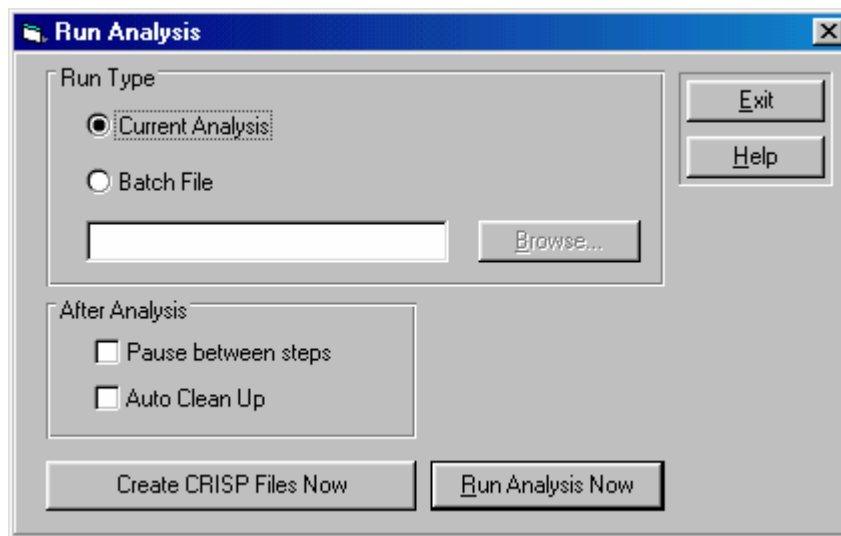
Figure 4-27:Run Analysis dialogue box

Step 2   Click on the Create CRISP Files Now button. Provided you have followed all of the above steps correctly, this will create .GPR and .MPD files for your Project. A message box will appear after each stage has been completed.

Step 3   Click on Exit once you have finished.

Step 4   In the File menu, click on Parent/Child... to display the Parent/Child dialogue box.

Step 5   Highlight the Parent Project in the Family Tree list and load it.

Step 6   Create the CRISP output files as described above and press Exit once you have finished.

Step 7   By now you should have created output files for both the Parent and Child Projects.

Setting Up A Batch Run

A Batch run is simply a list of Projects that are to be run consecutively. The following steps describe how to set up a Batch run to carry out the Parent and Child analyses in this way:

Step 1	In the File menu, click on Setup Batch Run... to display the Setup Batch Run dialogue box.
Step 2	Change to the drive and directory in which the Projects are saved. When you are in the correct directory, the names of both the Parent and Child Projects should appear in the Analysis Names list.
Step 3	Highlight these two files and click on Add to add them to the Files to run list.
Step 4	Click on Save Batch File As... to display the standard Windows File Save dialogue box.
Step 5	Give the batch file a name (with a .SBF extension) and click on OK to return to the Setup Batch run dialogue box.
Step 6	Click on Exit once you have finished

Performing The Finite Element Analysis

Now the analysis is ready to run:

| *Step 1*	In the file menu, click on Run Analysis... to display the Run Analysis dialogue box.
Step 2	Click on the Batch File option.
Step 3	Either enter the path to the batch file you just created manually, or locate it using the Browse utility.
Step 4	Click on Run Analysis Now. This will prompt you to save the current Project and then close down the Pre-Processor and run the Analysis program.
Step 5	You will be task switched from the Analysis Program into a DOS box where the analysis will commence. Equilibrium error percentages are issued for each increment as the analysis progresses. If all of the above stages have been carried out successfully, no error should be greater than about 1%.
Step 6	Once each analysis is complete, the output is converted into a Microsoft Access 2.0 database and, once this is finished, you will return to the Analysis dialogue box.
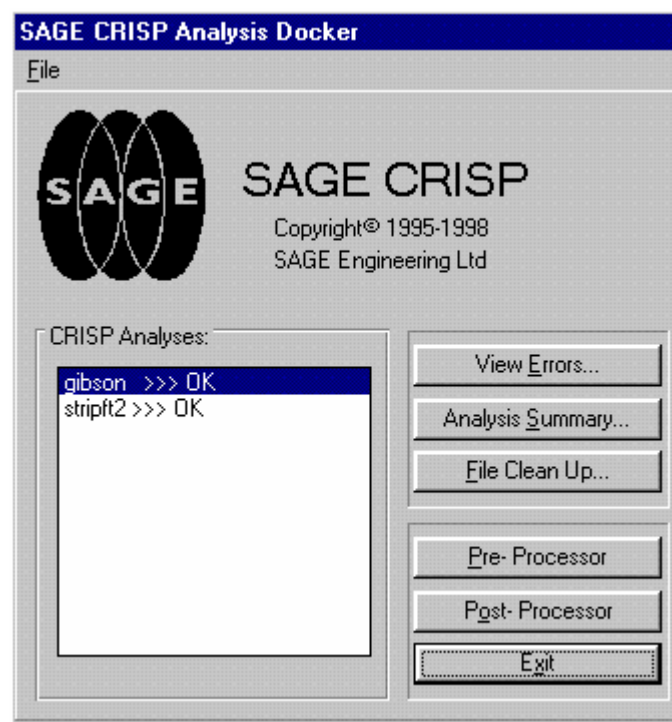
Figure 4-28:Analysis Docking Module

Step 7   Provided both Projects have been run successfully (i.e. without errors- the message to the right of the Project name should be OK), highlight the Parent Project in the list and click on the Post-Processor button. This loads the Project into the CRISP2D Post-Processor.

4.4   Post Processing

In this section you will be introduced to the following features:

- ■   Displacement Plots

- ■   Instance Graphs

- ■   Duration Graphs

4.4.1   Introduction

If you entered the Post-Processor directly from running the analysis, the strip footing Project will already be loaded. However you may also enter the Post-Processor by double-clicking on its icon in the CRISP2D program group in Windows' Program Manager. In this case, you should select Open Project... from the File menu and, provided the analysis has been successfully run, you will be able to post process the results from your analysis.

For this example, ensure that you have the Parent Project loaded in the Post-Processor.

4.4.2   Displacement Plots

Displacement plots are drawn directly onto the Graphics Windows. They provide a representation of how the mesh has deformed at any stage in the analysis. You can view either vector displacements or the deformed mesh at any specified magnification.

Step 1   Select Displacement Plots... from the Plots menu. This displays the Displacement Plot dialogue box.
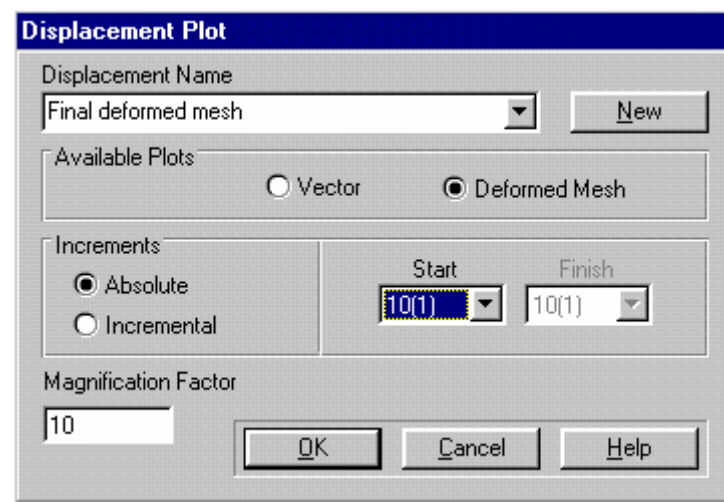
Figure 4-29:Setting Up A Displaced Plot

Step 2   Click on the New button to create your plot and change the title to 'Final deformed mesh'.

Step 3   Select the Deformed Mesh option.

Step 4   Ensure that Absolute is selected and that the Start increment is 10.

Step 5   Enter a Magnification Factor of 10. The Displacement Plot dialogue box should now look similar to Figure 4-29.

Step 6   Click on OK to exit, once you have finished. If you execute a Zoom Window to magnify the top quadrant of the mesh, the plot should look similar to Figure 3-24.

Figure 4-30:Deformed Mesh, Magnified 10 Times

4.4.3   Instance Graphs

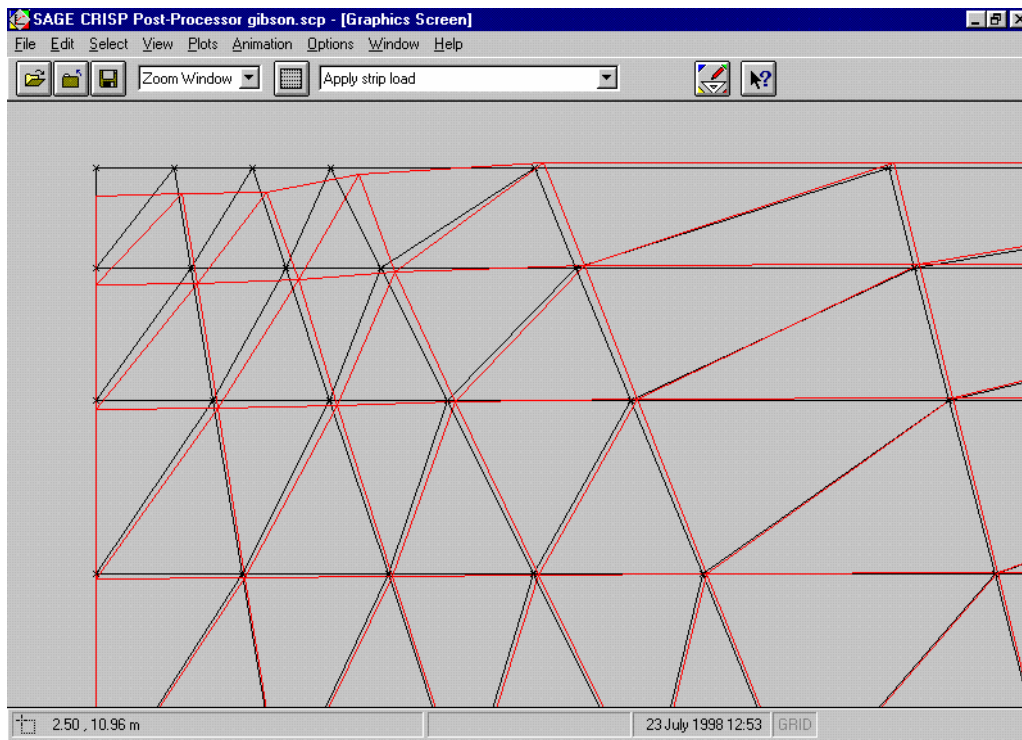Instance Graphs allow you to investigate output for selected integration points or nodes at particular increments in the analysis.

In this example the results from the Parent analysis, with 10 increments, and the Child analysis, with 5 increments, are plotted on the same graph to show that the values are identical. In both cases data is taken from the last increment when all loading has been applied.

> *Step 1*   In the Select menu, click on Nodes.
>
> *Step 2*   Select all of the nodes along the top edge of the mesh. These are the points from which the graph will draw its data.
>
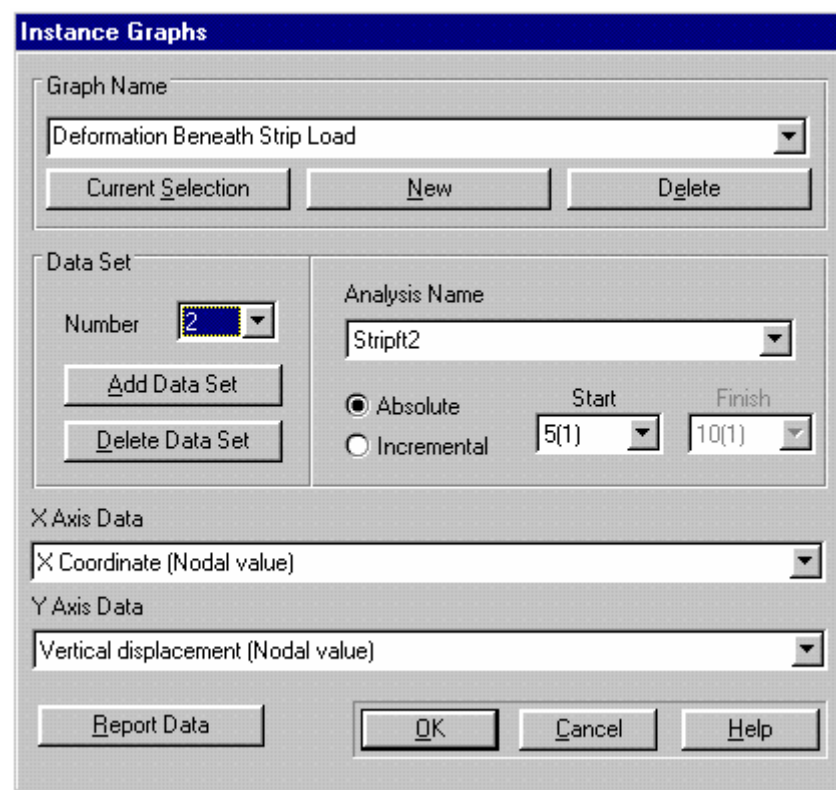> *Step 3*   In the Plots menu, click on Instance Graphs... to display the Instance Graphs dialogue box.

Figure 4-31:Setting Up An Instance Graph

Step 4   Click on the New button to create a new graph whose data is taken from the selected nodes. Change the title to 'Deformation Beneath Strip Load'.

Step 5   Ensure that the Data Set Number is 1 and that Analysis Name shows the name of the Parent Project you previously created.

Step 6   Click on Absolute and enter a Start value of 10.

Step 7   Choose 'X Co-ordinate' as X Axis Data and 'Vertical displacement' as Y Axis Data.

Step 8   Click on the Add Data Set button and select Data Set 2.

Step 9   Select the Child Project from the Analysis Name list (the Parent and Child should be the only two Projects in the list).

Step 10   Select Absolute and enter the Start number as 5. The Instance Graph Dialogue box should look like Figure 4-31.

Step 11   Click on OK to exit, once you have finished. You should see a graph similar to that displayed in Figure 4-32.

Step 12   The appearance of the graph can be altered in the in the Graph Configuration dialogue box. This is accessed via the Options menu, or by double-clicking on the graph itself. The options used in Figure 4-32 were 6 X axis divisions with minima and maxima of 0 and 30 respectively, 10 Y axis divisions with minima

and maxima of -.1 and 0.1 The legend, axis labels and graph title can also be changed in this dialogue box.

Step 13   The resulting graph confirms that in this case varying the number of increments does not alter the result.
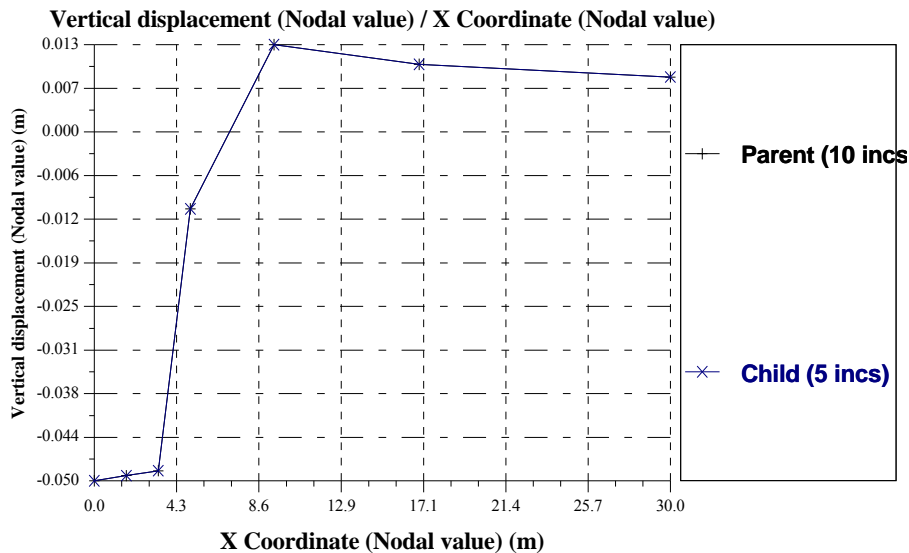

Vertical displacement (Nodal value) / X Coordinate (Nodal value)

Figure 4-32:Comparison Of Vertical Deformations

4.4.4   Duration Graphs

Duration graphs allow you to investigate the variation of output from individual integration points or nodes over the course of a specified range of increments.

Up to 10 data sets (each one representing a different integration point or node) can be viewed on the same graph. For this example, two integration points beneath the footing will be used.

Step 1   To reactivate the Graphics Window, click on Cascade, Tile Horizontally or on Tile Vertically in the Window menu. These options allow you to switch between the Graphics Window and any graphs that are on display.

Step 2   In the Select menu, click on Integration Points and then on Clear Selection. Select the integration points at the centroids of the left-most two finite elements directly beneath the strip footing.

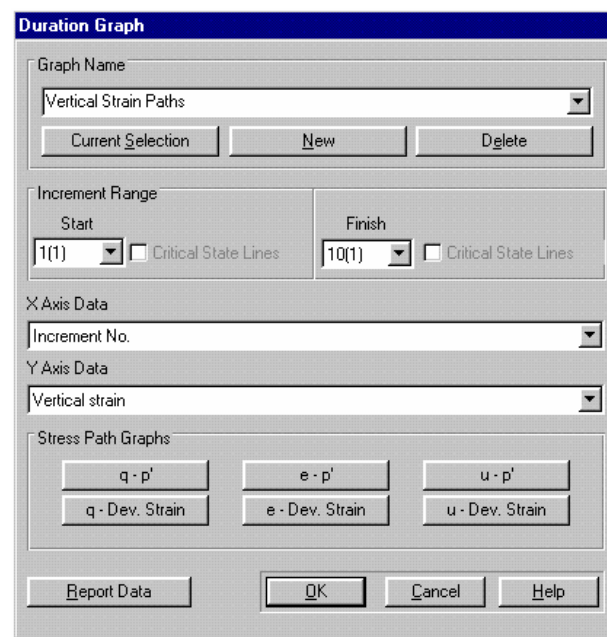*Step 3*   In the Plots menu, click on Duration Graphs... to display the Duration Graphs dialogue box.

Figure 4-33:Setting Up A Duration Graph

Step 4   Click on the New button to create your graph and change the title to 'Vertical Strain Paths'.

Step 5   Ensure that the Increment Range starts at 1 and finishes at 10.

Step 6   Select 'Increment No.' as X Axis data and 'Vertical strain' as Y Axis data. The Duration Graph dialogue box should now look like Figure 4-33.

Step 7   Click on OK to exit, once you have finished. The graph displayed should look similar to Figure 4-34. You can configure the way the graph looks (axis labels, titles and legend) by double-clicking on the graph window and changing the set-up in the Graph Configuration dialogue box.
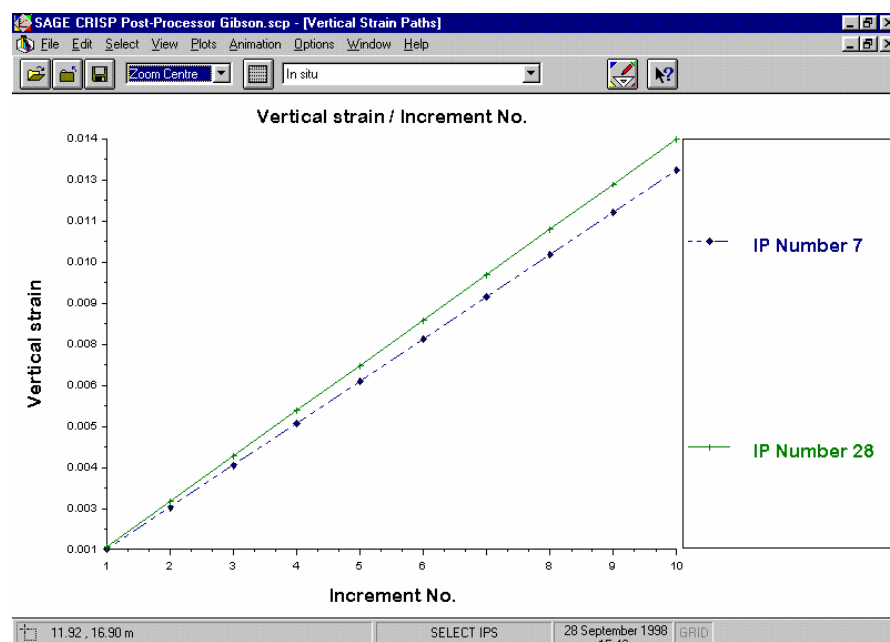
Figure 4-34:Graph Showing Strain Paths for Two Integration Points

4.5 Summary

By the end of this example you will have made use of the following features:

■ Autosave

■ Units

■ The background grid

■ The Structured Mesh Generator

■ The In Situ Stress converter

■ Parent/Child analyses

■ Instance graphs

■ Duration graphs

■ Displacement plots

4.6 Things To Try Next

■ The mesh used for this example will not necessarily give you the best results.

■ Try using different edge grading values to create a mesh that has smaller elements around the base of the footing and larger elements further away. The mesh shown in Figure 4-35 will give better results than the mesh used in the example- see if you can duplicate this mesh:
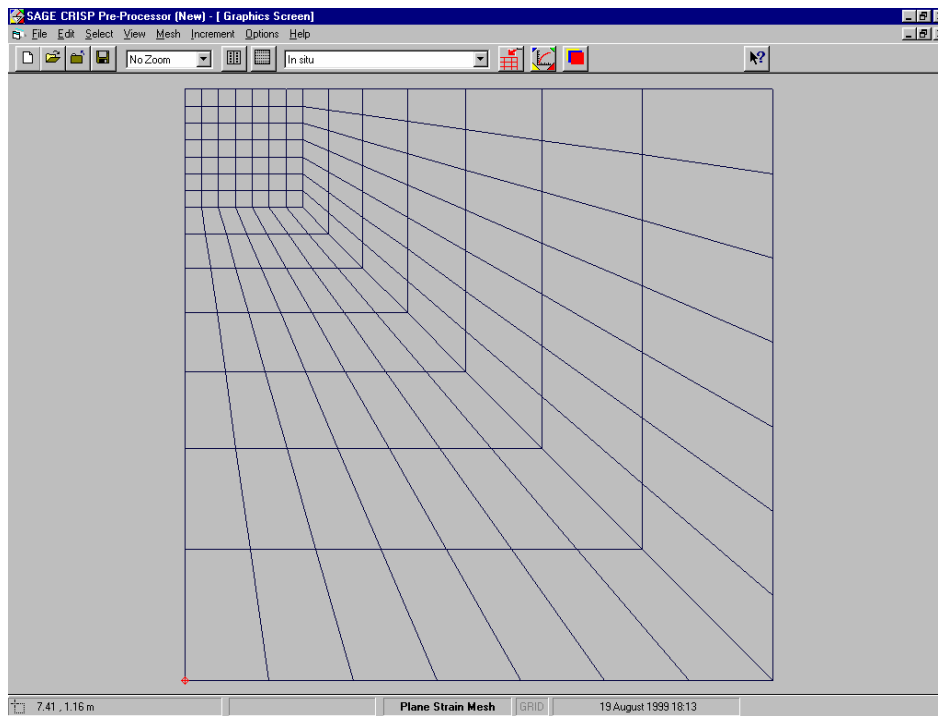
Figure 4-35:Refined Finite Element Mesh Created Using The Structured Mesh Generator

- ■   Try using the unstructured mesh generator to obtain a smoother transition from small to large elements.

- ■   The mesh in Figure 4-36 was created using a single super element (a mega element) with super nodes at the four corners and 4 additional super nodes at (0, 15) (0, 16.667) (5,20) and (7.5, 20)
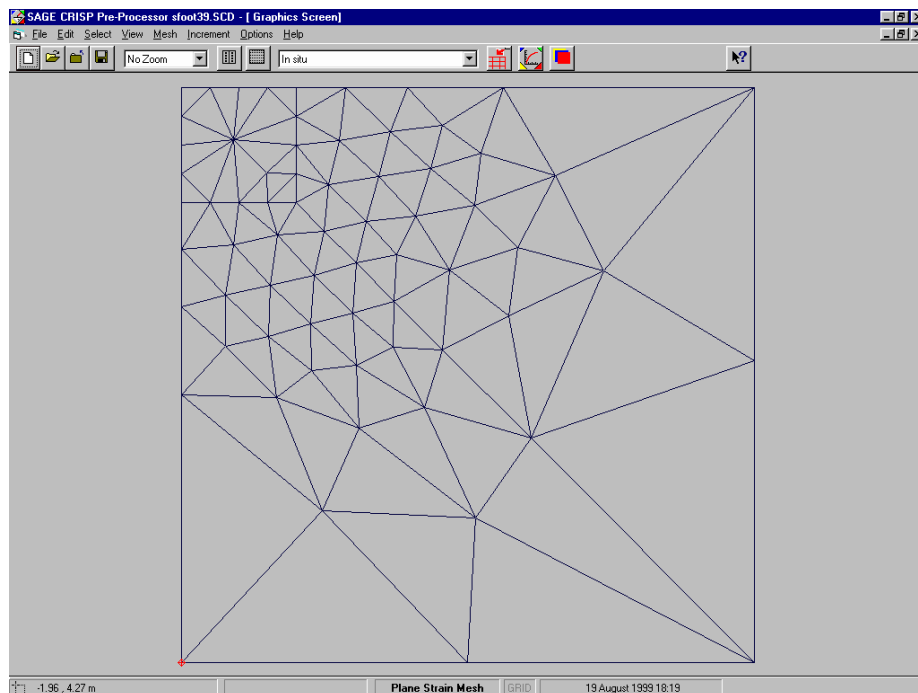

Figure 4-36 :Refined Finite Element Mesh Created Using The Unstructured Mesh Generator

OPEN AND BRACED EXCAVATION

5

5.1   Introduction

The following steps will be covered:

- ■   Starting The Project

- ■   Defining Units

- ■   Creating The Finite Element Mesh

- ■   Defining A Material Zone

- ■   Assigning Element Properties

- ■   Defining In Situ Stress Conditions

- ■   Defining Increment Blocks

- ■   Defining Boundary Conditions

- ■   Defining Loads

- ■   Scale Factors

- ■   Running The Analysis

- ■   Displacement Plots

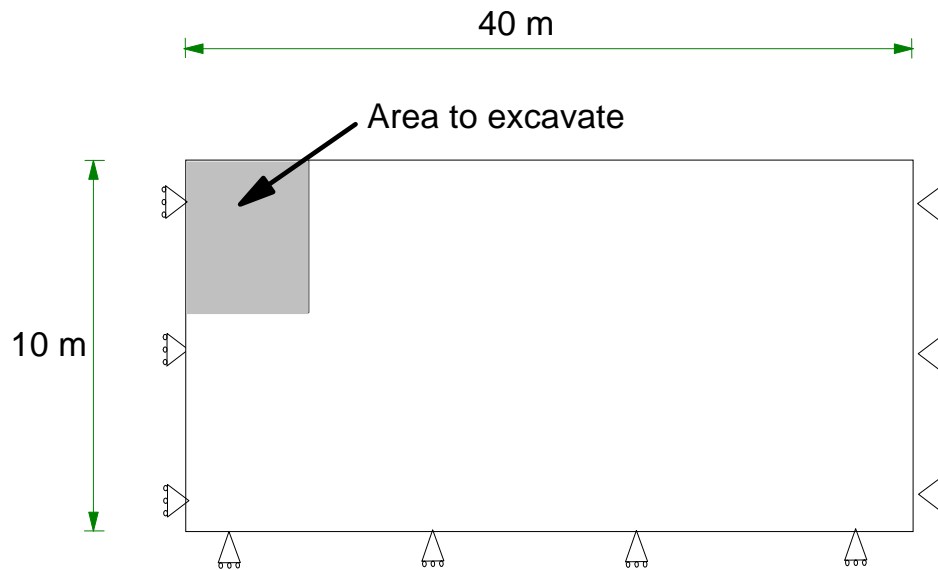- ■   Duration Graphs

5.2   The Problem

Figure 5-1 :Excavation Geometry

The excavation of a basement has been chosen to illustrate the construction techniques which can be set up within CRISP2D. It uses very simple elastic models to enable the construction methods to be modelled without too much complexity.

One of CRISP's main uses over the years has been to analyse earth excavation and earth retaining structures. There are numerous papers included in the Publications Directory which detail analyses which model these types of problems.

This example is a very simple representation of how the program can be used to set up braced and unbraced retaining wall problems. It first analyses an unbraced excavation similar to the example problem described in Britto and Gunn then returns back to the Pre-Processor to add a propped retaining wall to the excavation and compares the results of the two analyses.

5.3 Pre-Processing

This section describes all of the stages that must be completed using the Pre-Processor in order to run the finite element analysis. It is recommended that you save the Project regularly (by clicking on Save Project in the File menu).

5.3.1 Starting The Project

This section describes the basic processes that should be completed in advance of creating your mesh. The steps you will follow include:

- ■   Selecting A Mesh Mode

- ■   Entering Project Set-Up Information

- ■   Entering QA Information

- ■   Verifying The Base Units System

The following procedure describes the actions required to start and set-up a new Project.

Step 1   On opening the CRISP2D Pre-Processor, select New Project... from the File menu. This displays the Mesh Mode dialogue box:
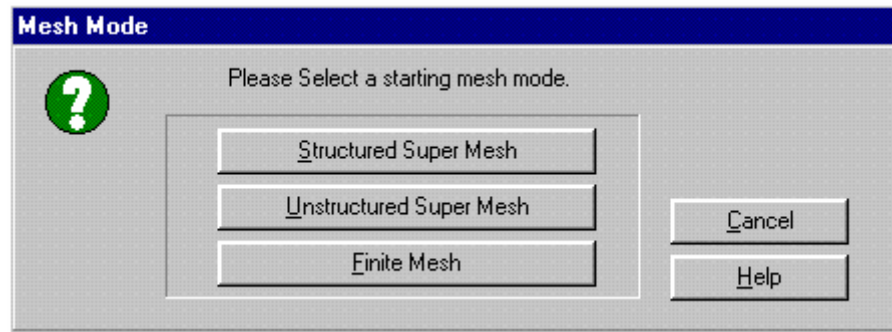

Figure 5-2 :Mesh Mode dialogue box

Step 2   Click on the Structured Super Mesh button. This exits the Mesh Mode dialogue box, sets you up in Structured Super Mesh mode and opens the Project Setup dialogue box.
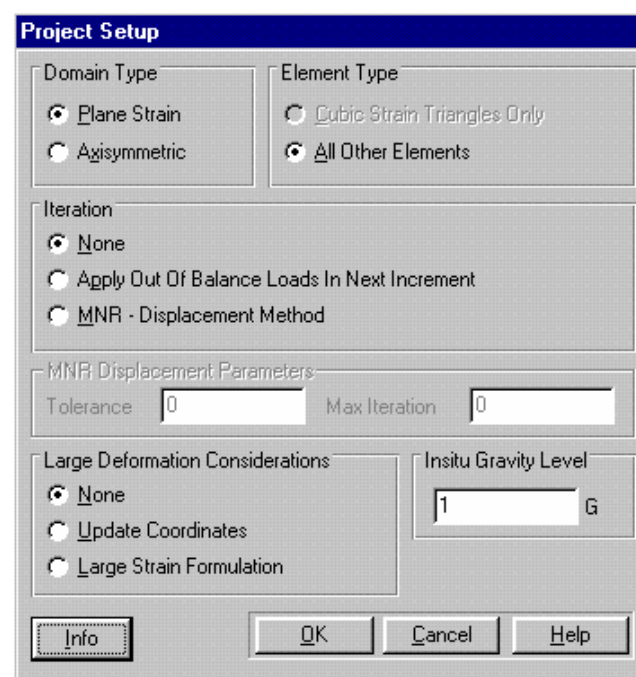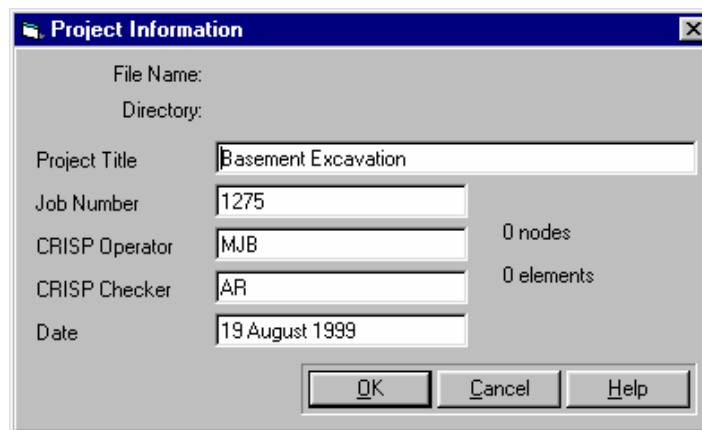

Figure 5-3:Project Setup Box

Step 3   In the Domain Type section, click on the Plane Strain (default) option button and click on All Other Elements in the Element Type section.

Step 4   The Project Setup should now look like Figure 5-3, with an in situ gravity level of 1G, the Apply in Next Increment and Update After Each Increment check boxes unchecked and Use The Standard Program selected.

Click on the Info... button. This displays the Project Information dialogue box which allows you to enter QA information such as job number, operator name, checker name, etc.

Figure 5-4:Project Information Box

Step 5   Enter data as appropriate. The date is automatically entered but may be manually altered.

Step 6   The Project Information dialogue box should now look similar to Figure 5-4.

Step 7   Click on OK to return to the Project Setup dialogue box once you have finished.

Step 8   Click on OK in the Project Setup dialogue box.

Defining Units

At this early stage it is useful to ensure that the correct system of base units is being used.

Step 1   In the Options menu, click on Units... to display the Units dialogue box.

Figure 5-5:Units dialogue box

Step 2   Use the list boxes to select the appropriate units for Distance, Force, Time and Angle as shown in Figure 5-5. Click on OK to exit, once you have finished.

5.3.2   Saving a Project

It is best to save the project at regular intervals throughout the preparation period.

Step 1   In the file menu click on Save Project...

Step 2   Select the required directory and enter a filename (with the .scd extension).

Step 3   Click on OK to save the project.

The project name will now appear on the banner at the top of the Pre-Processor.

Tip   Once a project has a filename, you can save the project quickly by using the Save Button on the toolbar (the button has a picture of a disk on it).

If you set the Autosave feature (select Default Settings in the Options menu), you will be able to restore the current file after any unofficial shut down of your system.

5.3.3   Creating A Mesh Using The Structured Mesh Generator

The Structured Super Mesh provides a method of generating large numbers of finite elements from a few Super Elements. For this example, the Super Mesh is used to automatically generate all the soil elements, with the wall elements created manually later.

This section covers all of the stages you must have completed in order to automatically generate the finite element mesh. The steps you will follow include:

- ■   Creating A Super Mesh

- ■   Defining Edge Gradings

- ■   Defining Material Zones

- ■   Assigning Element Properties

- ■   Generating The Finite Element Mesh

5.3.4   Creating The Super Mesh

The easiest way of creating nodes at precise co-ordinates is to create them manually in the Super Node List dialogue box.

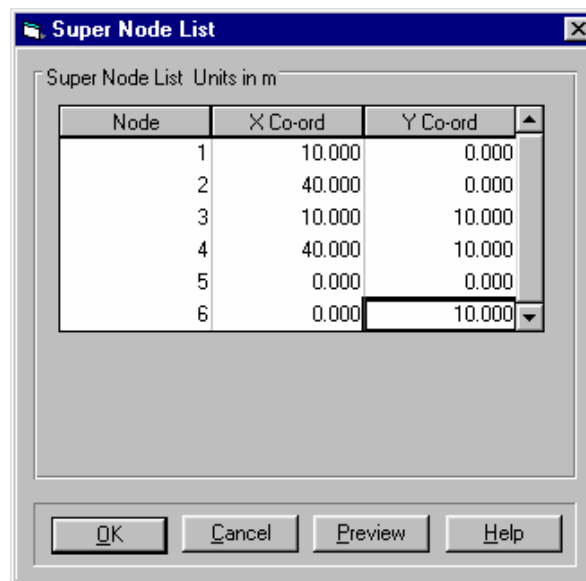*Step 1*   In the Mesh menu, click on Super Node List... to display the Super Node List dialogue box.

Figure 5-6:Super Node List dialogue box

Step 2   Enter the following nodes into the Super Node List box, using the TAB key to move between the X and Y co-ordinate columns. Each time you enter a Y co-ordinate for the last node in the list, press TAB to create a new node and move into it's X co-ordinate column.

Step 3   Once all the node co-ordinates have been entered click OK.

Node	X Co-ordinate	Y Co-ordinate
1	10	0
2	40	0
3	10	10
4	40	10
5	0	0
6	0	10

Step 4   In the View menu, click on Zoom and then on Full Page. The Graphics Window will resize to display the nodes at optimum scale.

Step 5   To aid you with the creation of the Super Elements it will be best to turn the node numbers on. In the View menu, click on Super Node Numbers

Step 6   In the Mesh menu, click on Create Super Elements. The fact that you can now create elements is confirmed by the mode indicator in the status bar, at the bottom of the screen.

Step 7   Join the nodes together to form the Super Mesh depicted in Figure 5-7.

Step 8   To create each Super Elements, you must click on each of its vertex nodes in turn, and then click again on the first node to complete the element. The complete sequence of nodes to join together is ; 1 - 2 - 4 - 3 - 1 (first super element created) 5 - 1 - 3 - 6 - 5 (second Super Element created)

Tip	If you make a mistake, you can backtrack one node at a time through the elements you have created simply by clicking again on each node.

> ***Step 9*** When you have finished, you should have two four sided Super Elements in a mesh identical to that shown in Figure 5-7.

> ***Step 10*** In the Edit menu, click on Commit Super Element Creation. This fixes the elements, confirming that they are correctly created. The elements will change colour when they committed. If you are not up to speed with creating elements, it is a good idea to Commit elements that have been correctly created as soon as each is finished. This way, if you make a mistake you can use the Abort Super Element Creation feature to undo all changes made since Commit Super Element Creation was last clicked on.
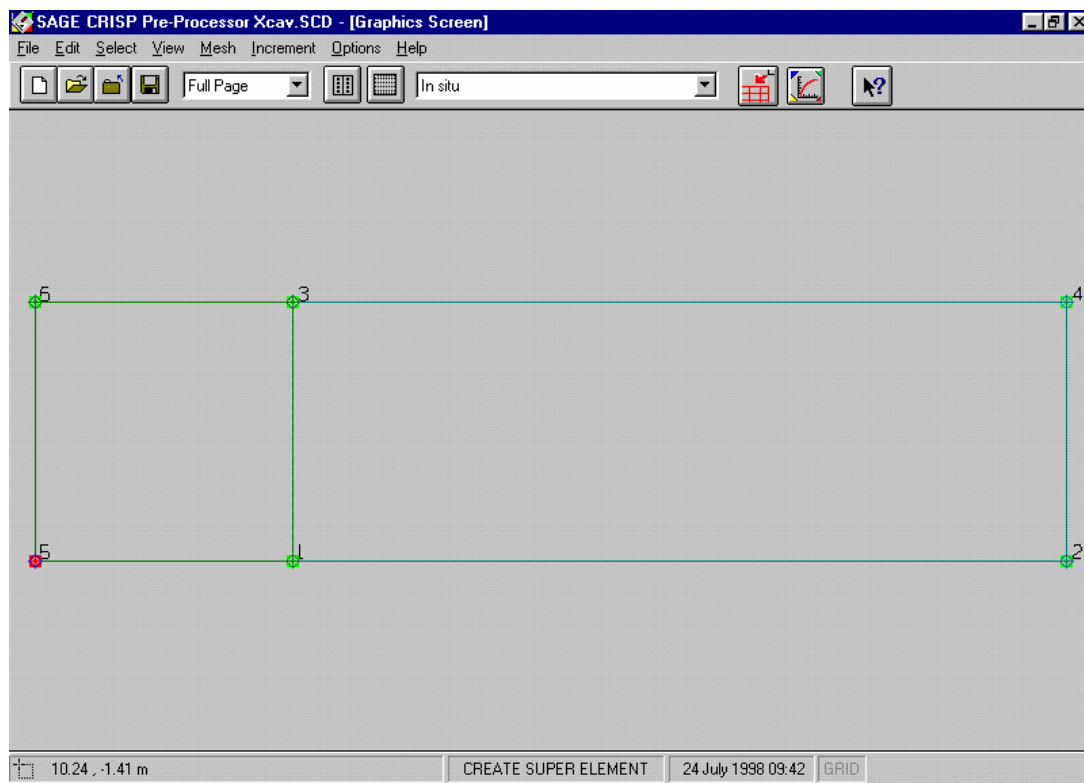

Figure 5-7:Structured Super Mesh

Defining Edge Gradings

Edge Gradings define how the Super Elements will be divided up into finite elements by the mesh generator. In general, better results will be obtained by using smaller finite elements in the parts of the mesh that experience the most rapid and/or the largest changes in stress, during the course of the analysis.

The following procedure defines gradings for the entire Super Mesh.

> ***Step 1*** In the Select menu, click on Super Edges. Select the top left-hand horizontal edge.

> ***Step 2*** In the Mesh menu, click on Super Edge Grading... to display the Super Edge Gradings dialogue box.
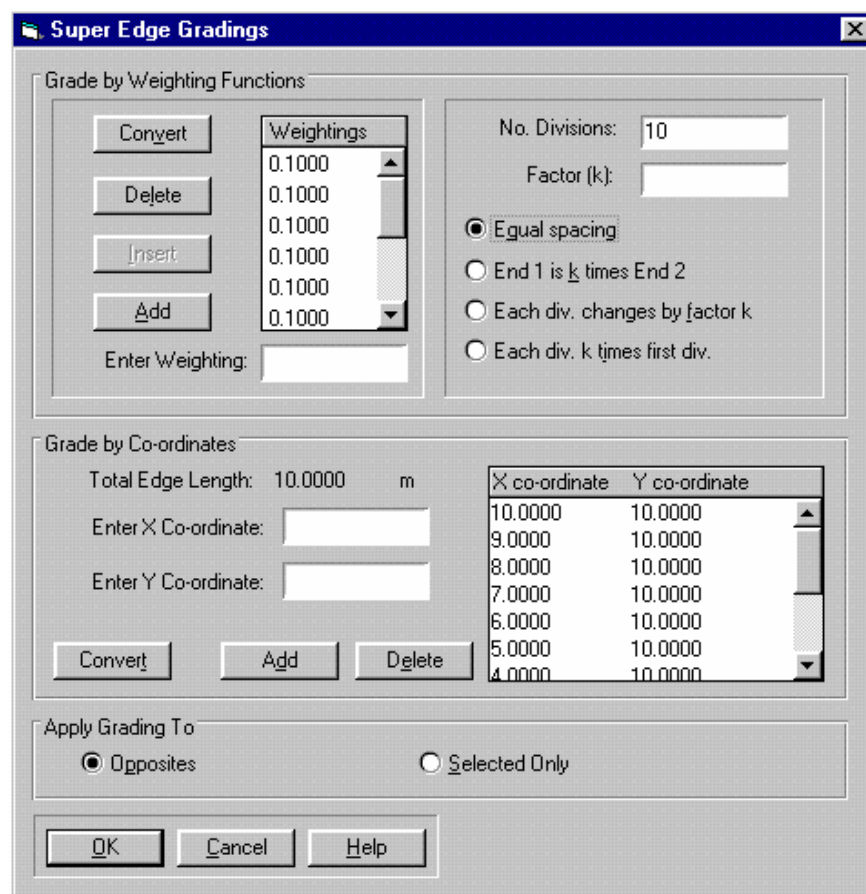
Figure 5-8:Defining Super Edge Gradings

Step 3   Enter a value of '10' into the Number of Divisions text box and click on the Equal Spacing option. Weighting values will be automatically entered into the list and the dialogue box should now look like Figure 5-8.

Step 4   Click on OK to exit, once you have finished.

Step 5   Opposite edges of each element are always graded in identical proportions, so by defining edge gradings for this edge, gradings have also been automatically defined for an additional edge. Grading markers are shown on the Graphics Window as diamond shape boxes positioned on graded super edges. The mesh should now look similar to Figure 5-9.
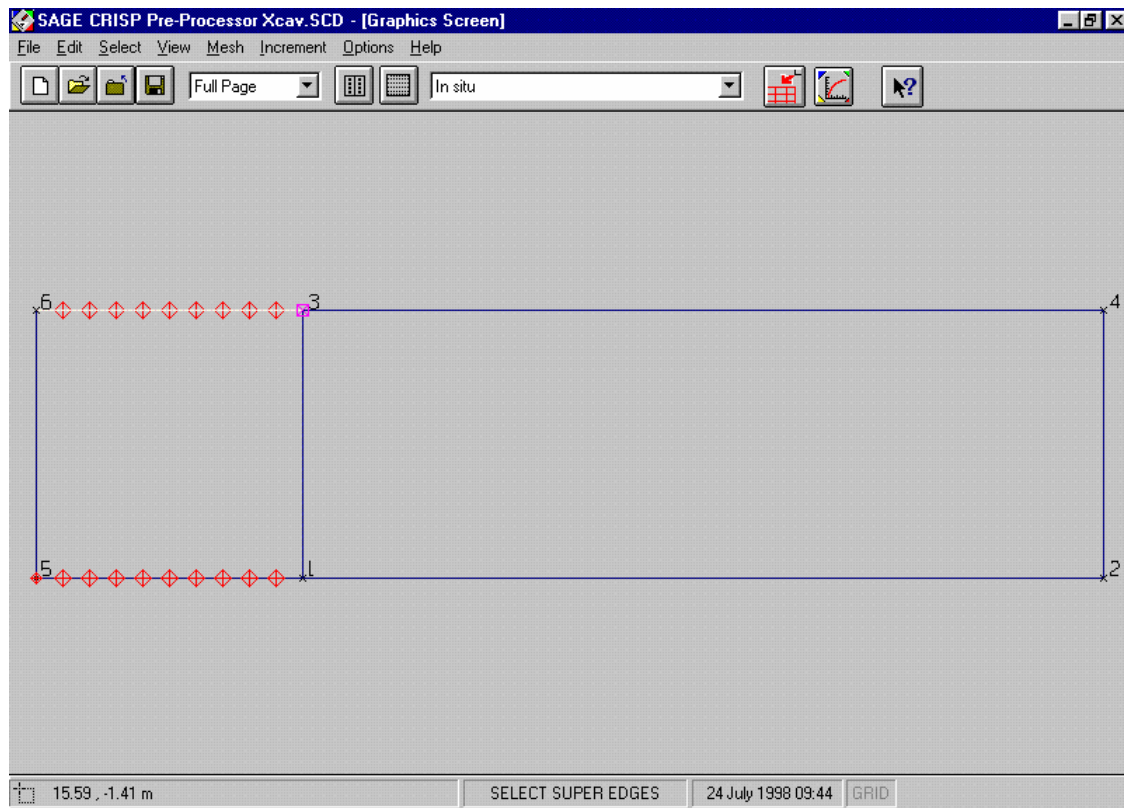
Figure 5-9:Super Edge Gradings

Step 6   In the Select menu, click on Clear Selection, or click again on the selected edges to de-select them. You can also use the short-cut key, CTRL+L, to clear the selection.

Step 7   Select the left-hand vertical edge.

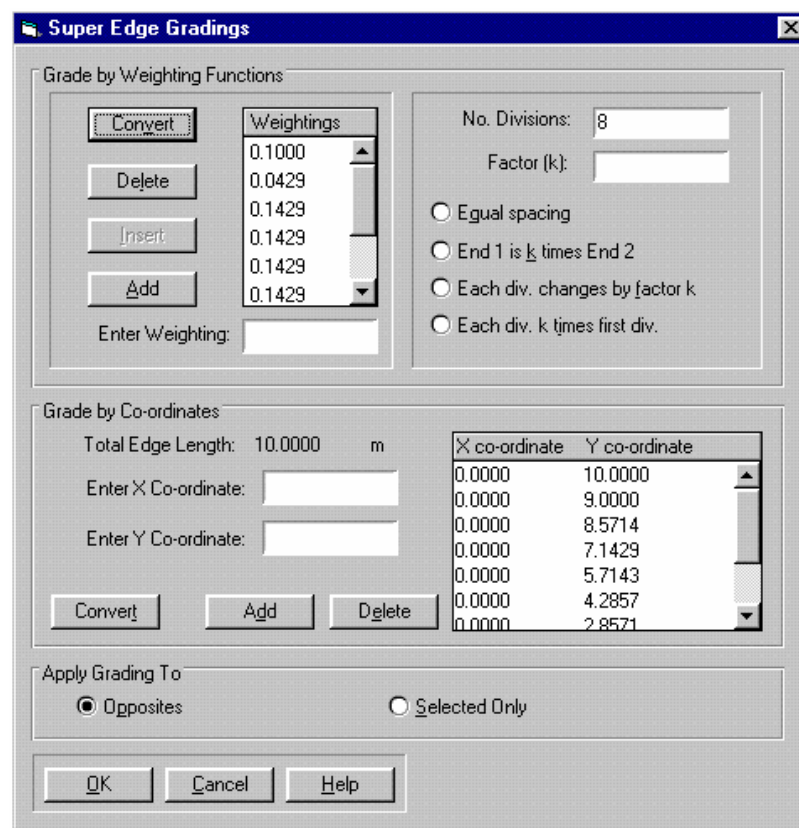*Step 8*   In the Mesh menu, click on Super Edge Grading... to display the Super Edge Grading dialogue box.

Figure 5-10:Defining Super Edge Gradings

Step 9   Type '7' in the Number of Divisions text box and click on Equal Spacing. To add an extra grading at 9m enter the X and Y co-ordinate as in Figure 5-10 and click Add. This will show the co-ordinate in the list box. Click Convert and then OK to assign these grading to the selected edge.

Step 10   Select Clear from the Select menu.

Step 11   Select the top right edge and assign 6 divisions with a factor of 3 and End 1 is k times End 2.

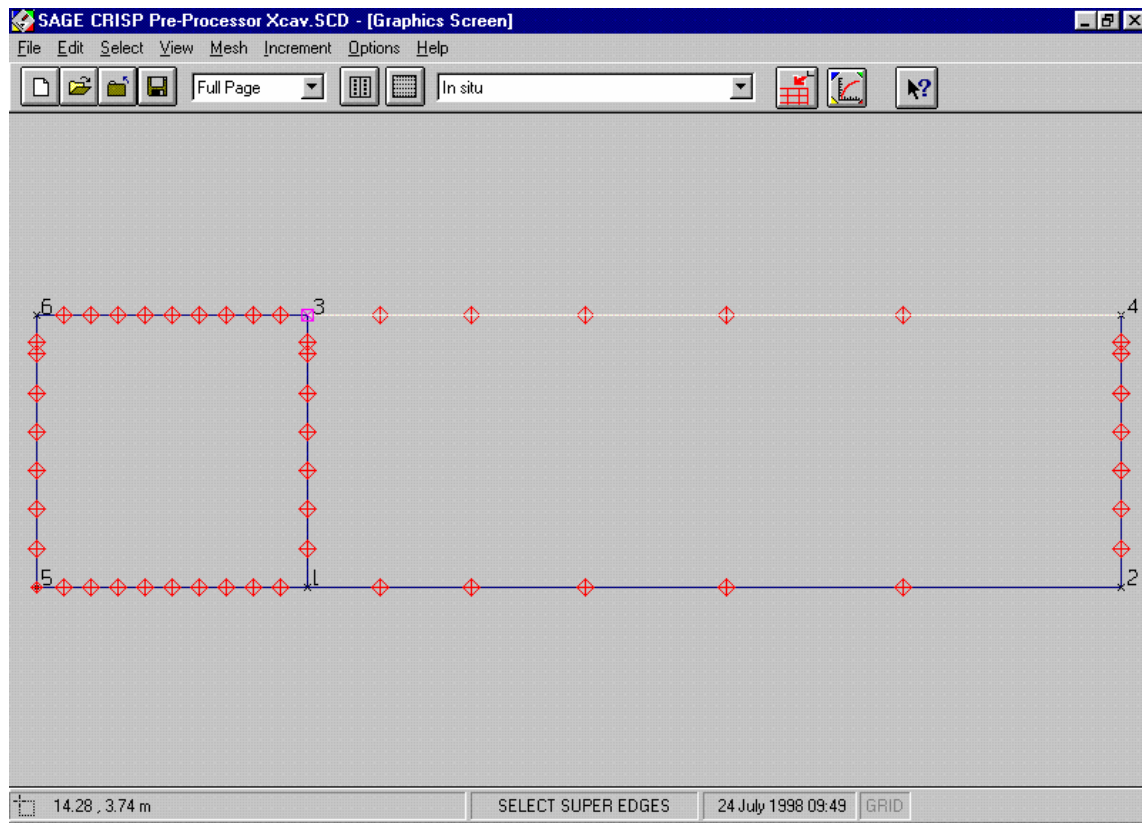*Step 12*   Your supermesh should now appear the same as Figure 5-11.

Figure 5-11:Super Edge Grading dialogue box

5.3.5   Defining Material Zones

A Material Zone contains information about the soil model and associated soil parameters that are being used to model particular regions of the mesh.

It is necessary to define the number of material zones at this point in order to be able to assign elements to zones (the next stage).

Once you have set up a zone, the soil model and parameters may be entered or changed at any time prior to the analysis being run.

> *Step 1*   Select Material Properties... from the Mesh   menu.   This displays the Material Properties dialogue box.

Figure 5-12:Material Properties dialogue box

Step 2   Enter the name "Elastic Soil" and select 'Elastic'(default) from the Soil Models list box. Material zones are referenced by name elsewhere in the Project so it is good practice to give each zone a meaningful title here.

Step 3   Enter the following material property values:

Property	Value	Units
E_h	3.0E3	kN/m^2
E_v	3.0E3	kN/m^2
ν_{hh}	0.25	None
ν_{vh}	0.25	None
G_{hv}	1.2E3	kN/m^2
K_w/γ_w	0	kN/m^3
γ bulk	20	kN/m^3

Step 4   The Material Properties dialogue box should now look similar to Figure 5-12.

Step 5   Click on OK to exit, once you have finished.

Tip   Although values for the soil properties have been defined here, it is possible delay defining them right up until just before you run the analysis.

5.3.6   Assigning Element Properties

Every element in the mesh has two properties associated with it:

- ■   Finite element type, and

- ■   Material zone

These properties are assigned to Super Elements in the following manner:

Step 1   In the Select menu, click on Domain Super Elements and select both the Super Elements.

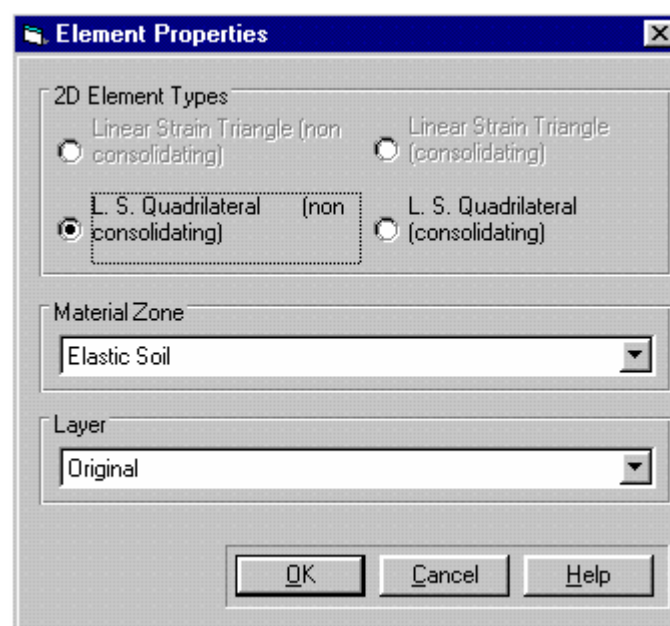*Step 2*   In the Mesh menu, click on Element Properties... to display the Element Properties dialogue box.

Figure 5-13:Assigning Element types and Material Zones

Step 3   Select the Linear Strain Quadrilateral (non consolidating) option.

Step 4   Select 'Elastic Soil' from the Material Types list box.

Step 5   The Element Properties dialogue box should now look like Figure 5-13.

Step 6   Click on OK to exit, once you have finished.

Step 7   This concludes the definition of element properties.

5.3.7   Generating The Finite Element Mesh

You are now at a stage where you can generate the finite element mesh from your super mesh.

Step 1   In the Mesh menu, click on Generate Finite Element Mesh. This will first check that you have correctly graded every edge and that you have assigned a

generated finite element type to each Super Element. It will then spend a short time performing the automatic generation of the finite element mesh.

Step 2   Provided you have completed the above stages correctly, the finite element mesh will be displayed as shown in Figure 5-14 and the main menu will change to allow you access to the additional options available in Finite Element mesh mode.
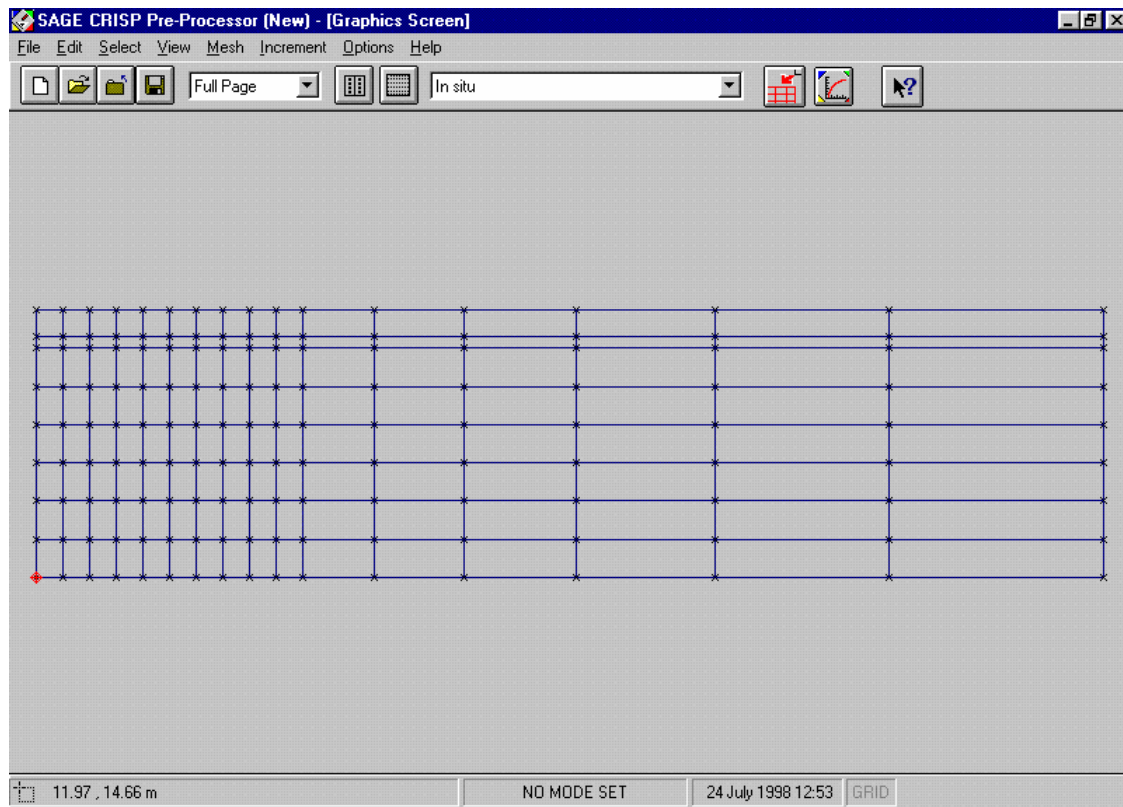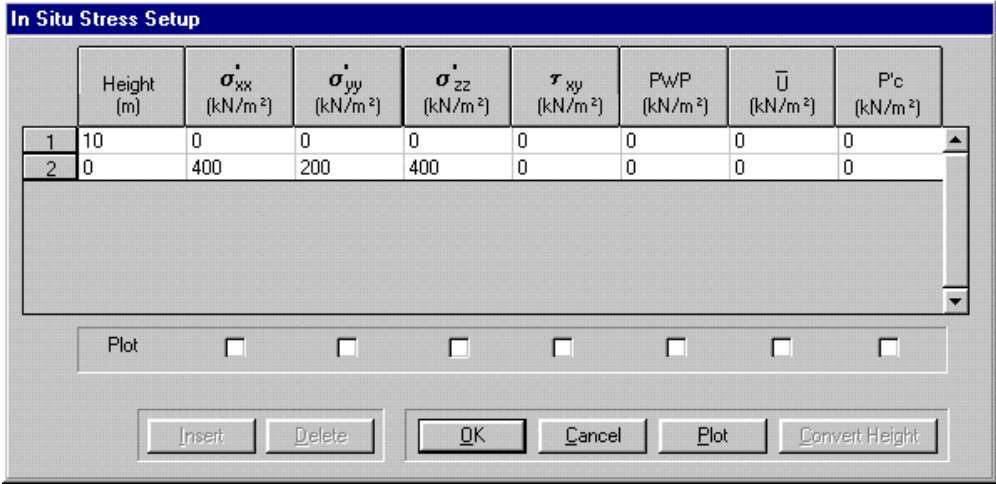

Figure 5-14:Automatically Generated Finite Element Mesh

5.3.8   Defining In Situ Stresses

In situ conditions are defined in terms of stresses at different elevations within the mesh.

Step 1   In the Increment menu, click on Define In Situ Stress Conditions... to display the In Situ Stress Setup dialogue box.

Figure 5-15:In Situ Stress Setup dialogue box

> *Step 2*   The top and bottom co-ordinates of the mesh will automatically be entered for you.
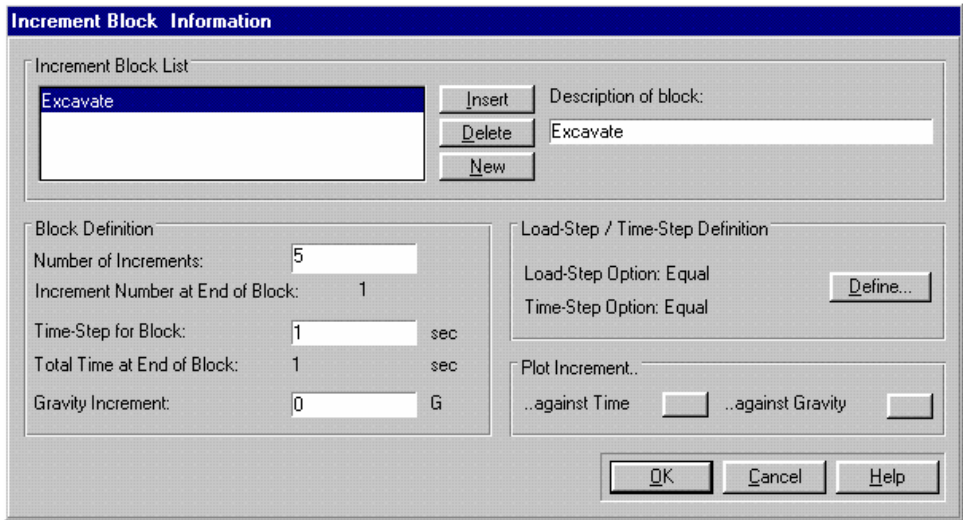
> *Step 3*   Enter the values as shown in Figure 5-15. Click on OK to exit, once you have entered and double-checked the values.

5.3.9   Defining Increment Blocks

Increments are the units by which the finite element program advances the analysis. In each increment, the effect of applied load and boundary conditions is calculated and the accumulated strains and pore water pressures are updated. Increments are grouped together in blocks which then model particular events in the analysis.

The first analysis in this example requires only 1 increment block, consists of five increments and allows the excavation to be carried out without any structural support.

> *Step 1*   In the Increment menu, click on Define Increment Block Parameters... to display the Increment Block Information dialogue box.

Figure 5-16 :Defining Increment Blocks

Step 2   Change the name of the first increment block from "Increment Block (untitled)" to "Excavate" by entering this into the Description of block text box. Change the number of increments to 5.

Step 3   The Increment Block Information dialogue box should now look like Figure 5-16.

Step 4   Click on OK to exit.

5.3.10 Setting Up The Construction Sequence

The construction of the excavation is modelled in a single stage.

The following procedure describes how to remove elements from the mesh in order to simulate this construction sequence.

Step 1   Select 'Excavate' from the increment block list box on the main toolbar.

Step 2   In the Select menu, click on Domain Elements and select all of the elements highlighted in Figure 5-17.

Step 3   In the Increment menu, click on Remove Elements. This removes the elements from the current, and all subsequent increment blocks.
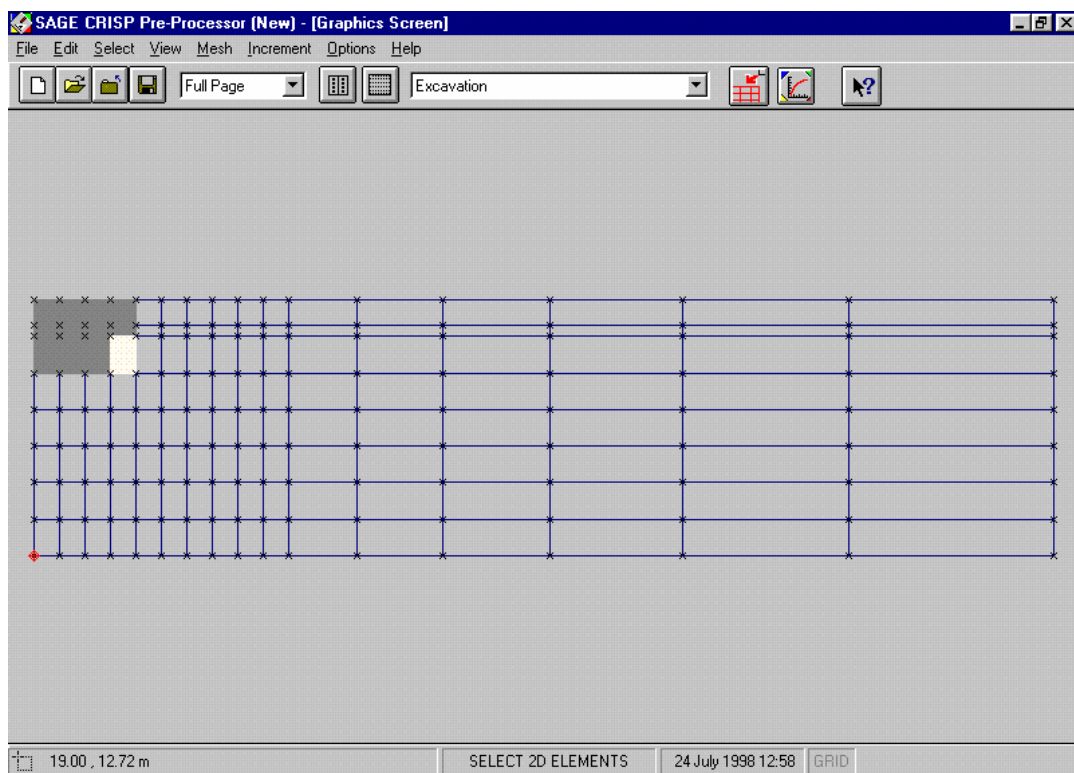

Figure 5-17:Removing Elements in "Excavate" Increment

5.3.11 Defining Boundary Conditions

Boundary conditions describe to the finite element analysis the ways in which the mesh is physically restrained from moving. This is achieved by applying fixities to edges and nodes in the mesh.

Once defined, fixities apply in all successive increment blocks. The Fixities dialogue box and the Graphics Window (if View Fixities is toggled on) will only display any fixity that is applied during the current increment block.

Only in situ fixities are applied in this example.

Defining In Situ Fixities

In this example the left and right-hand edges are restrained horizontally, and the base is restrained both horizontally and vertically during in situ stage. The following procedure defines fixities for the in situ stage:

Step 1   Select 'In situ' from the Increment Block list box on the main toolbar.

Step 2   In the Select menu, click on Edges.

Step 3   Select all the far-left-hand and far-right-hand (vertical) edges of the mesh by clicking on them.

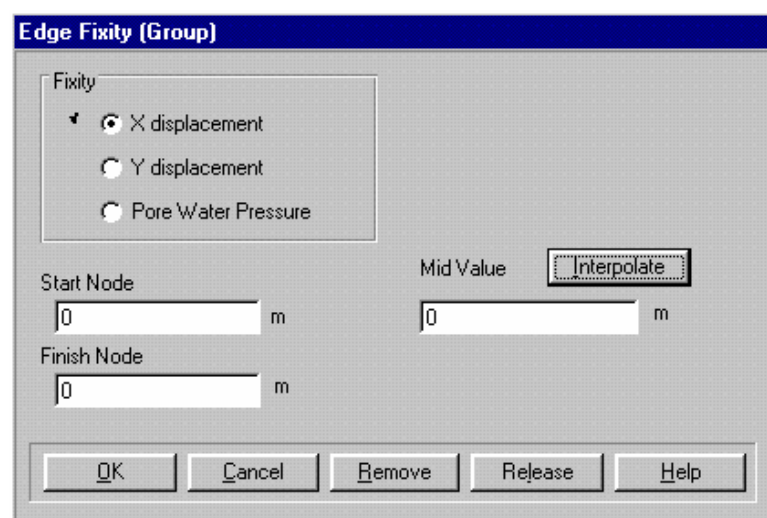*Step 4*   In the Increment menu, click on Fixities... to display the Edge Fixity (Group) dialogue box.

Figure 5-18:Defining Edge Fixities

Step 5   Click on the X Displacement option.

Step 6   Specify '0' X direction fixities in the Start Node and Finish Node text boxes. Set the Mid Value to zero either by entering it directly, or by clicking on the Interpolate button.

Step 7   The Edge Fixity dialogue box should now look like Figure 5-18. Click on OK to exit, once you have finished.

Step 8   In the Select menu, click on Clear Selection to ensure that no edges remain selected. Select the bottom (horizontal) edges of the mesh by clicking on them.

Step 9	In the Increment menu, click on Fixities... and define zero X displacement fixities as before.
Step 10	Click on the Y Displacement option.
Step 11	Specify Y direction fixities of zero in the Start Node and Finish Node and Mid Value text boxes.
Step 12	Click on OK to exit, once you have finished.
Step 13	The Graphics Window should now look similar to Figure. The size of the fixity icons can be changed in the Default Settings dialogue box (accessed via the Options menu).

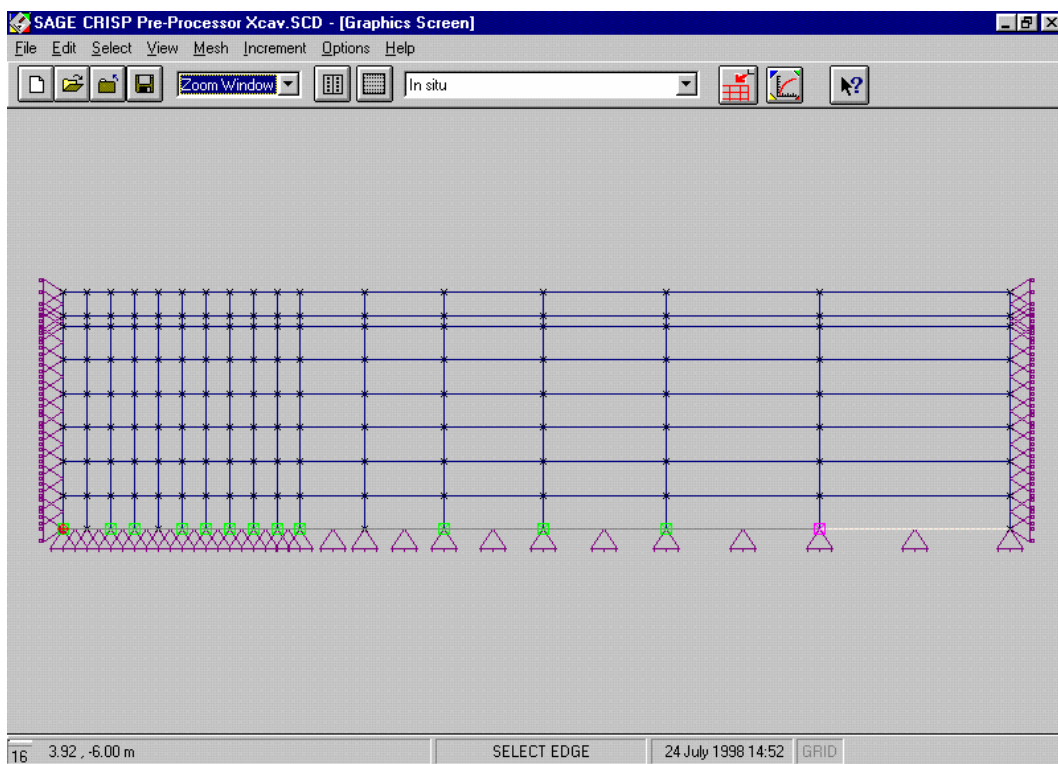*Figure 5-19:Fully Defined In Situ Fixities*

5.3.12 Running The Analysis

The Project should now be saved. Provided you have followed all of the steps described above, preparation of the Project using the Pre-Processor is complete and the analysis is ready to be run, as described below.

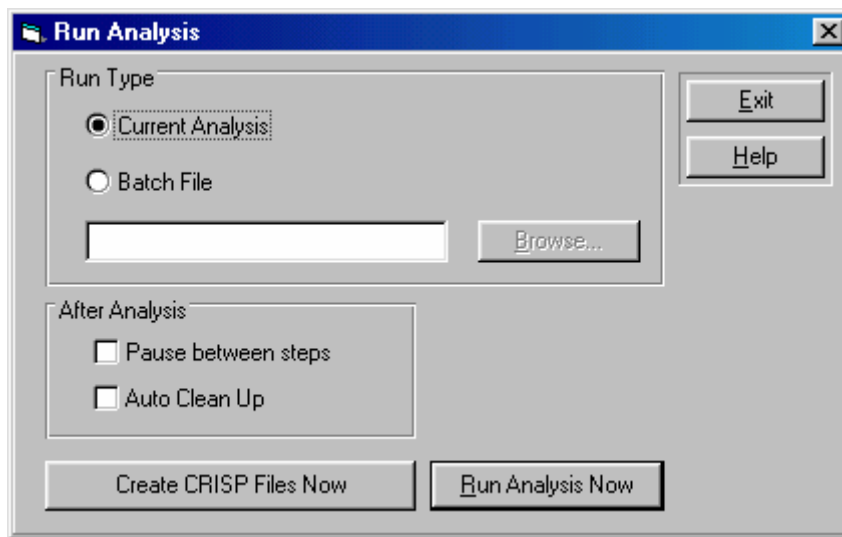|	
Step 1	In the File menu, click on Run Analysis... to display the Run Analysis dialogue box.

Figure 5-20:Run Analysis dialogue box

Step 2 Click on the Create CRISP Files Now button. Provided you have followed all of the above steps correctly, this will create .GPR and .MPD files for your Project. A message box will appear after each stage has been completed.

Step 3 Click on the Run Analysis button. This will prompt you to save your Project and will then run the finite element analysis.

Step 4 You will be task switched into a DOS box where the analysis will commence. Equilibrium error messages are issued for each increment of the analysis.

Warning If the error ever exceeds about 1% then this is an indication that something is wrong with the input data. You should quit the analysis immediately (by pressing CTRL-BREAK) and double check all data in the Project in the Pre-Processor.

 If there is a problem, first check the in situ stress conditions, then check the material properties. Check also that fixities are correctly defined and that the Project Setup is correct.

Step 5 Once the analysis is complete, the output is converted into a Microsoft Access 2.0 database. Once this is complete, you will see the Analysis Docker dialogue box:
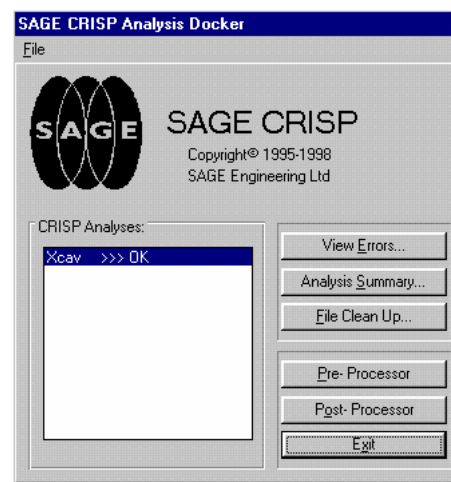
Figure 5-21:The Analysis Docking Module

> ***Step 6*** If the Project ran successfully (without errors- the message to the right of the Project name should say 'OK'), click on the Post-Processor button to load the Project into the CRISP2D Post-Processor.

5.4 Post-Processing

In this section you will be introduced to the following features:

- Displacement Plots

- Instance Graphs

5.4.1 Introduction

If you entered the Post-Processor directly from running the analysis, the excavation Project will already be loaded. However you may also enter the Post-Processor by double-clicking on its icon in the CRISP2D program group in Windows' Program Manager. In this case, you should select Open Project... from the File menu and, provided the analysis has been successfully run, you will be able to post process the results from your analysis.

Initially this example will look at the deflected shape of the excavation surface and the excavation side before returning to the pre processor to make modifications to the analysis. The deformed shape of the excavation will be shown using an displacement plot and two instance graphs.

5.4.2 Displacement Plots

Displacement plots are drawn directly onto the Graphics Windows. They provide a representation of how the mesh has deformed at any stage in the analysis. You can view either vector displacements or the deformed mesh at any specified magnification.

The displacement plot described below illustrates the final shape of the mesh at the end of the last analysis increment.

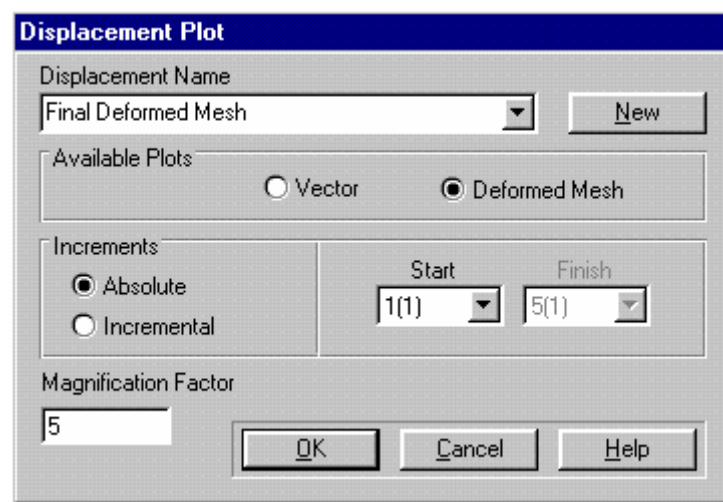> ***Step 1*** Select Displacement Plots... from the Plots menu. This displays the Displacement Plot dialogue box.

Figure 5-22:Setting Up A Displaced Mesh Plot

Step 2   Click on the New button to create your plot and change the title to "Final deformed mesh".

Step 3   Select the Deformed Mesh option.

Step 4   Ensure that Absolute is selected and enter a Start increment of 5.

Step 5   Set a Magnification Factor of 5. The Displacement Plot dialogue box should now look similar to Figure 5-22.

Step 6   Click on OK to exit, once you have finished. Figure 5-23 shows the displaced mesh plot in Graphics Window (a zoom window has been executed to show more clearly the excavated region).
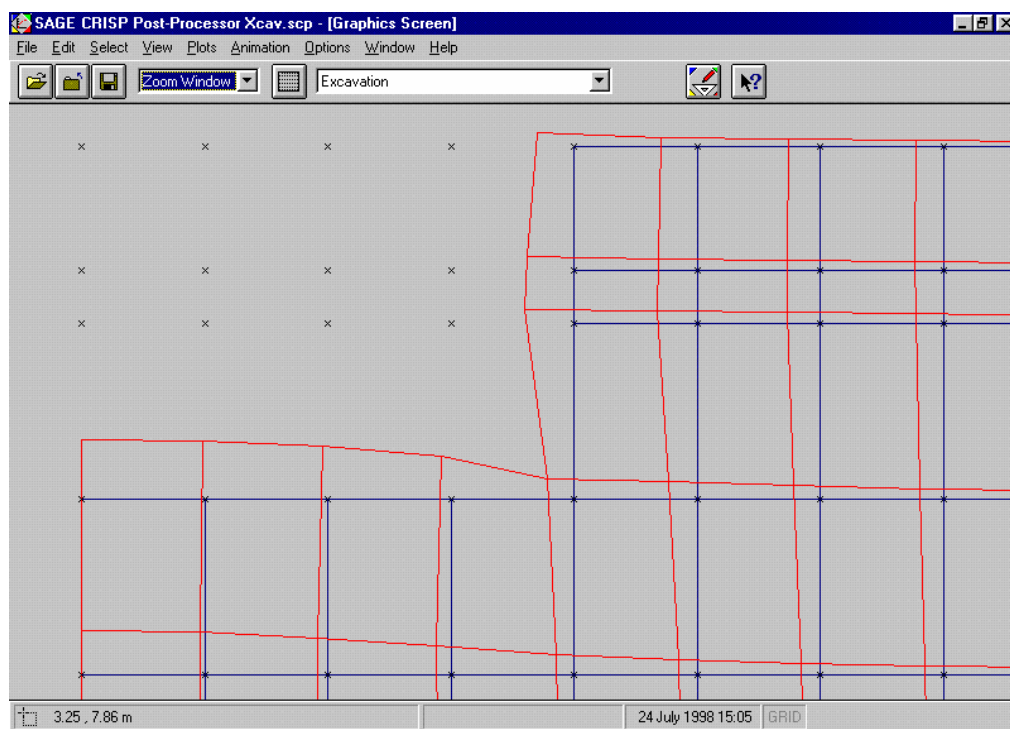
Figure 5-23:Deformed Mesh at Collapse

Tip	Displacement plot look clearer if you select good line styles and colours. To change the line styles, Select Line Styles... from the Options menu. To change the line colours, select Colour... from the Options menu.

5.4.3   Instance Graphs

Instance Graphs allow you to investigate output for selected integration points or nodes at particular increments in the analysis.

The graph described below shows the deformation of the base of the excavation at the end of the analysis

Step 1   In the Plots menu, click on Clear Current Plot. This makes it easier to see which nodes you are selecting.

Step 2   Select 'Excavate' from the Increment Block list box on the main toolbar - this will make the selection of nodes clearer.

Step 3   In the Select menu, click on Nodes.

Step 4   Select all of the nodes along the excavated surface from left to right.

Step 5   In the Plots menu, click on Instance Graphs... to display the Instance Graphs dialogue box.

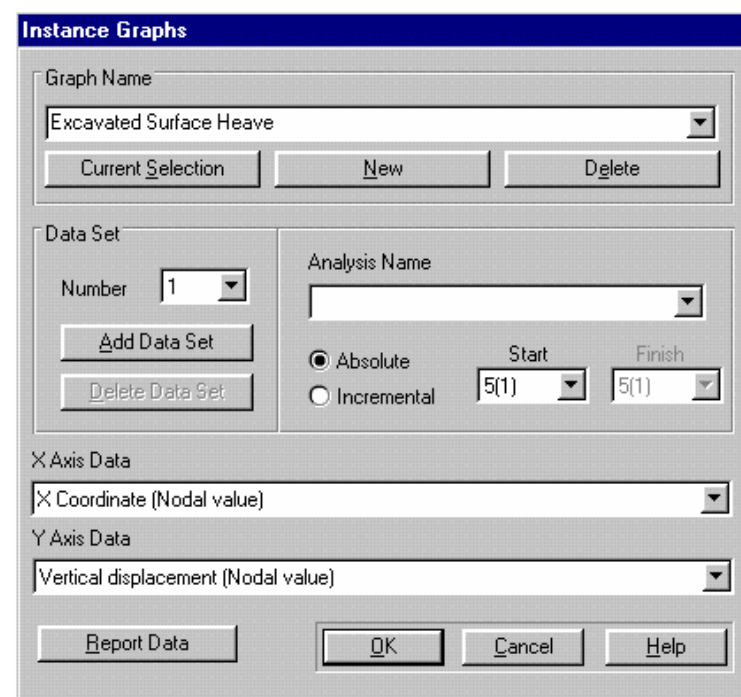*Figure 5-24:Setting Up An Instance Graph*

Step 6   Click on the New button to create a new graph whose data is taken from the selected nodes. Change the title to "Excavated Surface Heave".

> ***Step 7*** Click on Absolute and enter a Start value of 5.
>
> ***Step 8*** Choose 'X Co-ordinate' as X Axis Data and 'Vertical displacement' as Y Axis Data.
>
> ***Step 9*** Click on OK to exit, once you have finished. You should see a graph similar to that displayed in Figure 5-25.

The appearance of the graph can be altered in the in the Graph Configuration dialogue box. This is accessed by double-clicking on the graph itself.
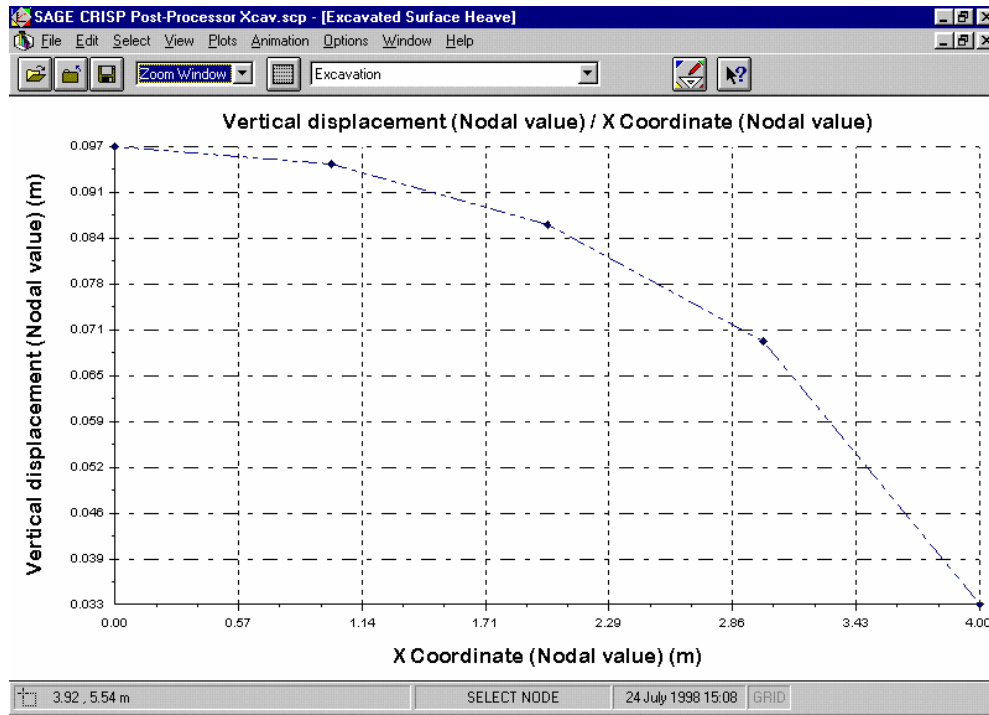
Figure 5-25:Heave of basement surface after excavation

The graph described below shows the deformation of the side of the excavation at the end of the analysis.

> ***Step 10*** In the Select menu, click on Clear Select.
>
> ***Step 11*** Select all of the nodes down the excavated surface from top to bottom.
>
> ***Step 12*** In the Plots menu, click on Instance Graphs... to display the Instance Graphs dialogue box.
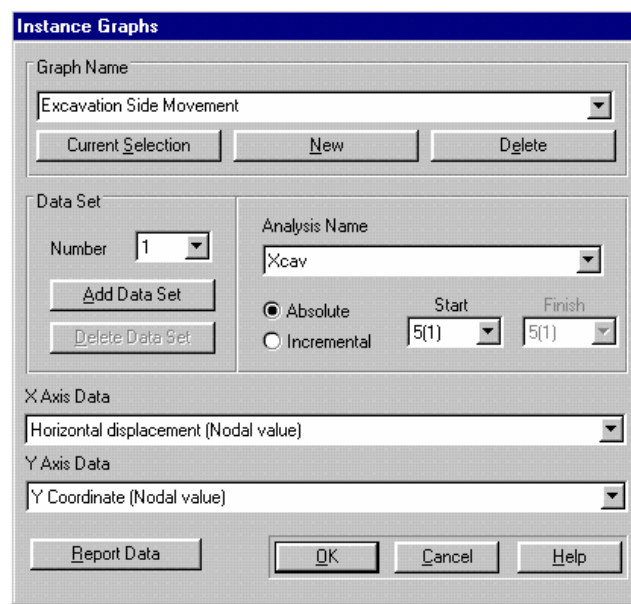
Figure 5-26:Setting Up An Instance Graph

Step 13   Click on the New button to create a new graph whose data is taken from the selected nodes. Change the title to "Excavated Side Movement".

Step 14   Click on Absolute and enter a Start value of 5.

Step 15   Choose 'Horizontal displacement' as X Axis Data and 'Y Co-ordinate' as Y Axis Data.

Step 16   Click on OK to exit, once you have finished. You should see a graph similar to that displayed in Figure 5-27.

The appearance of the graph can be altered in the in the Graph Configuration dialogue box. This is accessed by double-clicking on the graph itself.
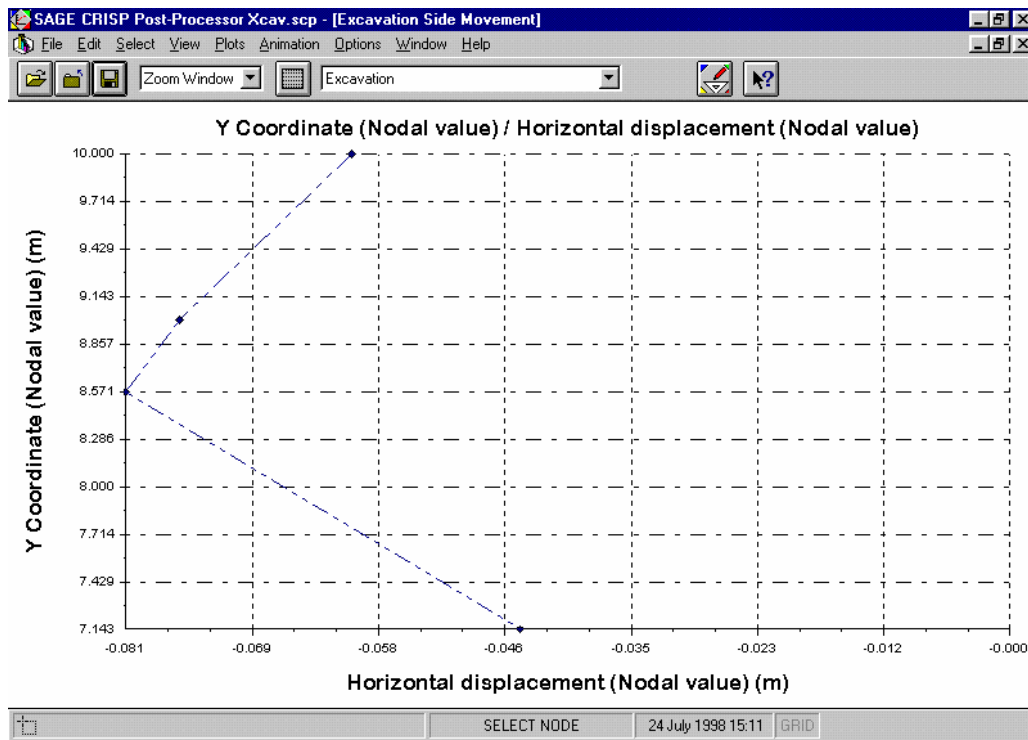
Figure 5-27:Horizontal Displacements of excavation side.

On first inspection this graph may appear to have an unexpected profile but on closer inspection of the horizontal stresses (Select the integration points down the back of the wall and plot an instance graph as in Figure 5-28) the reasons for the apparent pinning of the top of the excavation can be seen. As the sides of the excavation move the horizontal stress down the side of the wall decreases proportionally with the height from the top of the excavation, however the material model is Elastic and so does not experience any yielding. A much better result would be attained from either an elastic, stiffness varying with depth, model or an elastic perfectly plastic model.

As discussed at the beginning, this example has been made a simple as possible, but still shows the important of understanding the results. Under normal circumstances this analysis should be re-done with a yielding soil model, however, this example will continue with the basic model.
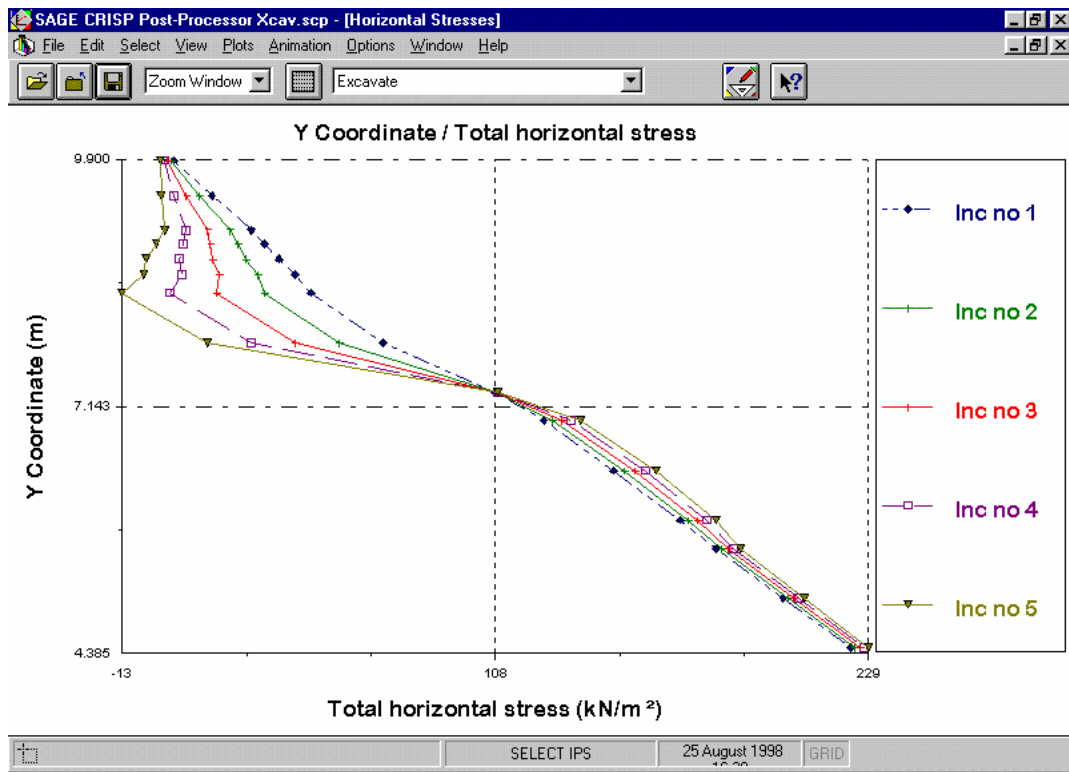
Figure 5-28:Comparison Of Horizontal Stresses during analysis

5.4.4　Saving the Project

The example will now lead you through some alterations to the pre processing of the file. Before we do this you should save the project. Select Save Project from the File menu. Select Exit from the File menu to leave the Post-Processor.

5.5　Pre-Processor - (Adding a retaining wall)

5.5.1　Opening a project

In the File menu, click on Open File and select the scd file from the appropriate directory. The example will then be displayed on the graphics screen.

5.5.2　Defining a new Material Zone

Step 1　　　　Select Material Properties... from the Mesh menu. This displays the Material Properties dialogue box.
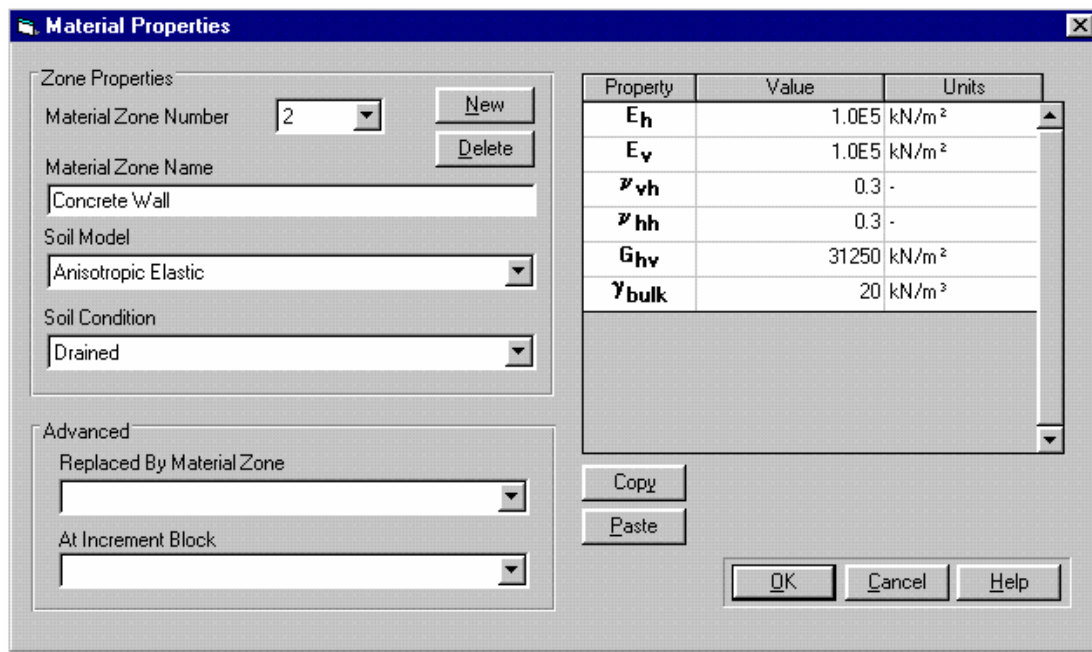
Figure 5-29:Material Properties dialogue box

Step 2   Click on the New button and enter the name "Concrete Wall" and select 'Elastic'(default) from the Soil Models list box. Material zones are referenced by name elsewhere in the Project so it is good practice to give each zone a meaningful title here.

Step 3   Enter the following material property values:

Property	Value	Units
E_h	1.0E5	None
E_v	1.0E5	None
ν_{hh}	0.30	None
ν_{vh}	0.30	None
G_{hv}	31250	kN/m^2
K_w/γ_w	0	kN/m^3
γ bulk	20	kN/m^3

Step 4   The Material Properties dialogue box should now look similar to Figure 5-29.

Step 5   Click on OK to exit, once you have finished.

Tip   Although values for the soil properties have been defined here, it is possible delay defining them right up until just before you run the analysis.

5.5.3 Swapping elements.

To simulate the construction of a retaining wall we need to swap a column of soil elements in the mesh for a column of wall elements. CRISP2D does not allow you to change the material properties of elements without using a Stop - Re start analysis. It is possible however to define Super Imposed Elements.

Super-imposed elements are two "Layers" of elements that overlap. As overlapping elements are not permitted in CRISP2D, it is necessary to remove the old elements, in this example, the soil, and replace them with the new elements, in this example, the Wall.

The first process in this sequence is to define a super-imposed set of elements

Defining Super Imposed elements

Step 1 In the Select menu, click on Elements. Select the elements down the side of the excavation, to include all but the bottom two elements in the strip (see Figure 5-30).
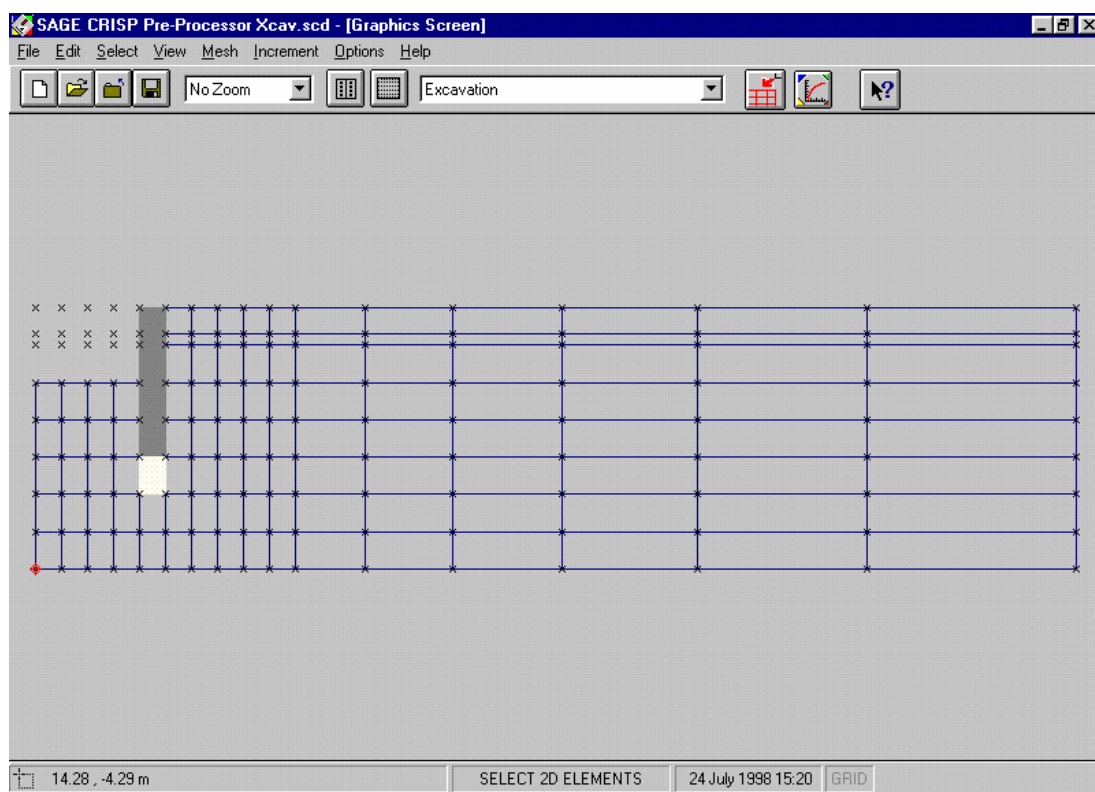

Figure 5-30:Defining Super Imposed elements

Step 2 In the Mesh menu, click on Super-Impose Elements... to display the Super-Impose Elements dialogue box.

Step 3 Give the layer the name of "Wall", select "Concrete Wall" from the Material Name list and Non Consolidation for the Finite Element Type.
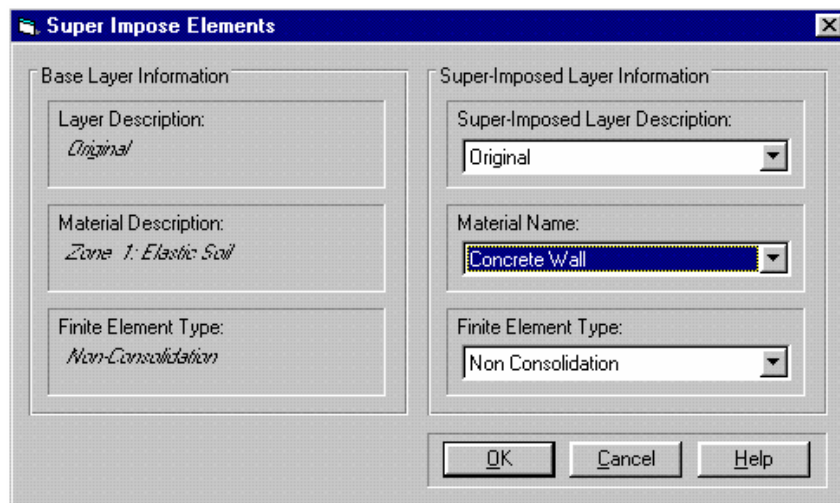
Step 4 Click OK

Figure 5-31:Super Impose Elements dialogue box:

This process will create a set of elements with the same geometry as the selected set and will have placed them on top of the existing element set.

As the wall is not in the analysis at the in situ stage, it must be removed.

Creating a "Swap" Increment

An increment in which elements are swapped is no different to any other increment, it is simply an increment where elements are added at the same location as elements which have just been removed.

The first step is to create the Increment Block.

Step 1 In the Increment menu, click on Define Increment Block Parameters... to display the Increment Block Information dialogue box.

Step 2 Highlight Excavate increment that was created earlier and click on Insert. This will add an increment before the Excavate increment.

Step 3 Change the name of the inserted increment block from "Increment Block (untitled)" to "Install Wall" by entering this into the Description of block text box. Change the number of increments to 2.

Step 4 The Increment Block Information dialogue box should now look like Figure 5-32.
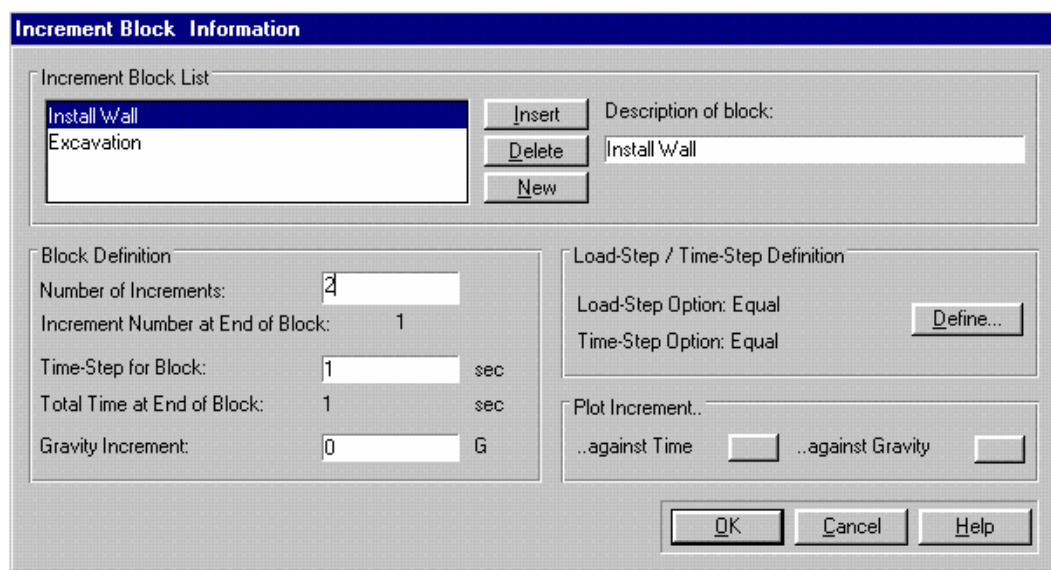
Step 5 Click on OK to exit.

Figure 5-32 :Defining Increment Blocks

Adding and Removing Elements

The final stage or this operation is to remove the wall in the in situ stage, remove the soil and add the wall at the "Install Wall" stage.

When you click on a point which two elements occupy the multi-selection box will appear as shown in Figure 5-33. Use the material property of the elements, shown in the multi select box, to distinguish between the elements.
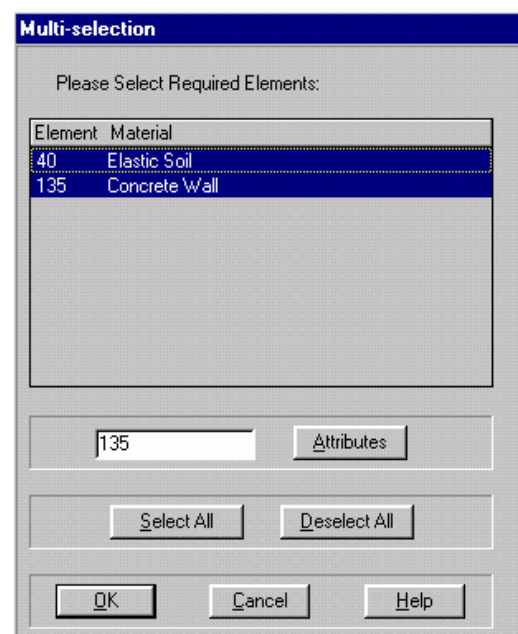

Figure 5-33:Multi-selection dialogue box

Step 1   Make sure that the in situ increment is chosen from the Increment box.

Step 2   In the Select menu, click on Clear Selection to clear the current element selection.

Step 3 Select the Wall Elements.

Step 4 In the Increment menu, click on Remove Elements to remove the wall elements from the in situ stage.

Step 5 Select "Install Wall" from the Increment Box.

Step 6 In the Select menu, click on Clear Selection to clear the current element selection.

Step 7 Select the soil elements which are to be replaced by the wall elements.

Step 8 In the Increment menu, click on Remove Elements to remove the soil elements from the "Install Wall" stage.

Step 9 Click on the Removed Elements icon on the toolbar to view the elements currently removed at the "Install Wall" stage. Both the wall and soil elements should be present in the screen view.

Step 10 In the Select menu, click on Clear Selection to clear the current element selection.

Step 11 Select the Wall Elements

Step 12 In the Increment menu, click on Add Elements to add the wall elements at the "Install Wall" stage.

This example has been made particularly simple to show the principles of the program. It does not reflect a standard that professional analyses should be conducted to. For more information on installation effects and retaining walls refer to the Publications Directory.

Before running the analysis chose Save As.. from the File menu and save the project under a different name but with the same .scd extension. This will stop the data from the original .scd file being overwritten.

5.5.4 Running The Analysis

The Project should now be saved. Provided you have followed all of the steps described above, preparation of the Project using the Pre-Processor is complete and the analysis is ready to be run, as described below.

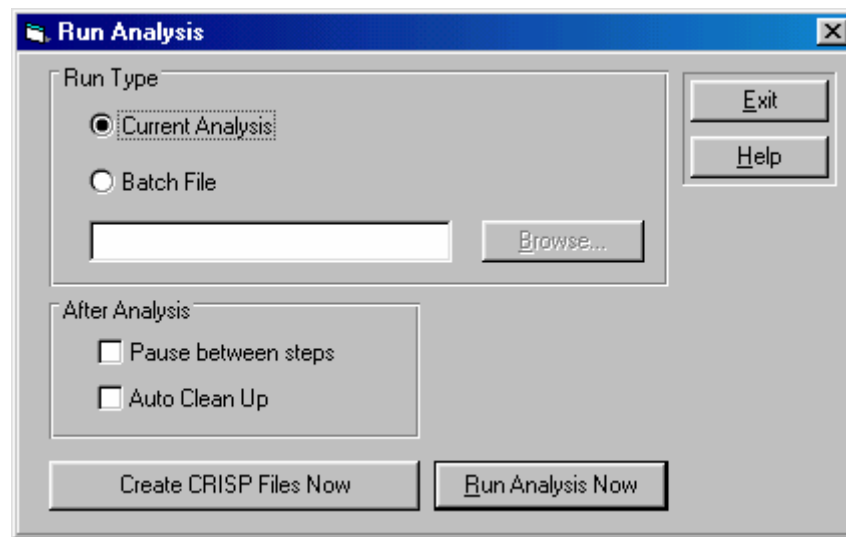*Step 1* In the File menu, click on Run Analysis... to display the Run Analysis dialogue box.

Figure 5-34:Run Analysis dialogue box

Step 2 Click on the Create CRISP Files Now button. Provided you have followed all of the above steps correctly, this will create .GPR and .MPD files for your Project. A message box will appear after each stage has been completed.

Step 3 Click on the Run Analysis button. This will prompt you to save your Project and will then run the finite element analysis.

Step 4 You will be task switched into a DOS box where the analysis will commence. Equilibrium error messages are issued for each increment of the analysis.

Warning If the error ever exceeds about 1% then this is an indication that something is wrong with the input data. You should quit the analysis immediately (by pressing CTRL-BREAK) and double check all data in the Project in the Pre-Processor.

If there is a problem, first check the in situ stress conditions, then check the material properties. Check also that fixities are correctly defined and that the Project Setup is correct.

Step 5 Once the analysis is complete, the output is converted into a Microsoft Access 2.0 database. Once this is complete, you will see the Analysis Docker dialogue box:
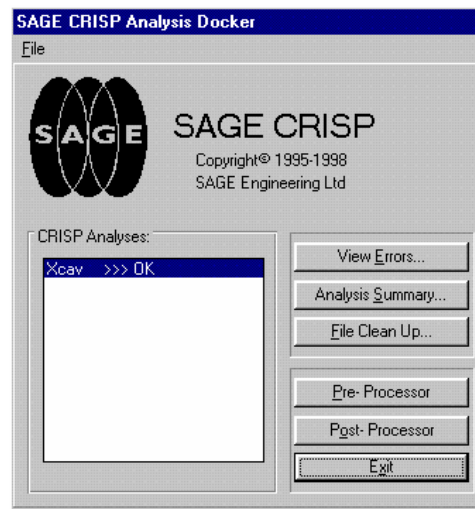
Figure 5-35:The Analysis Docking Module

If the Project ran successfully (without errors- the message to the right of the Project name should say 'OK'), click on the Post-Processor button to load the post processor.

5.6 Post-Processor

The post processor options for the two .scd files (one with and one without the wall) are the same. To produce the deformation plot and the two instance graphs - refer back to the previous post processor description earlier in the example.

Remember that the last increment is now 7 and not 5 when plotting the results.
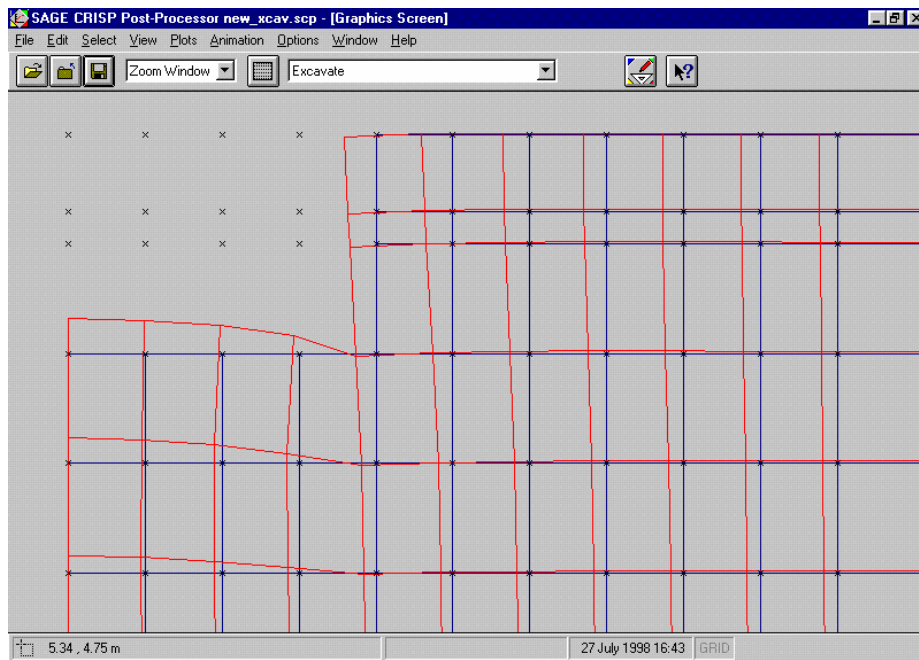
Compare your results to the ones below.

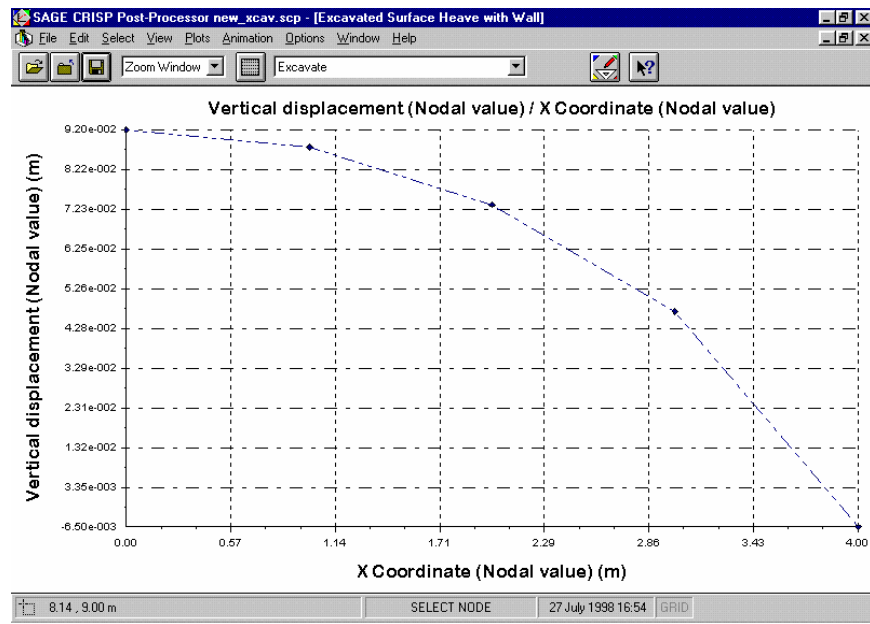*Figure 5-36:Displacement Plot with Wall*
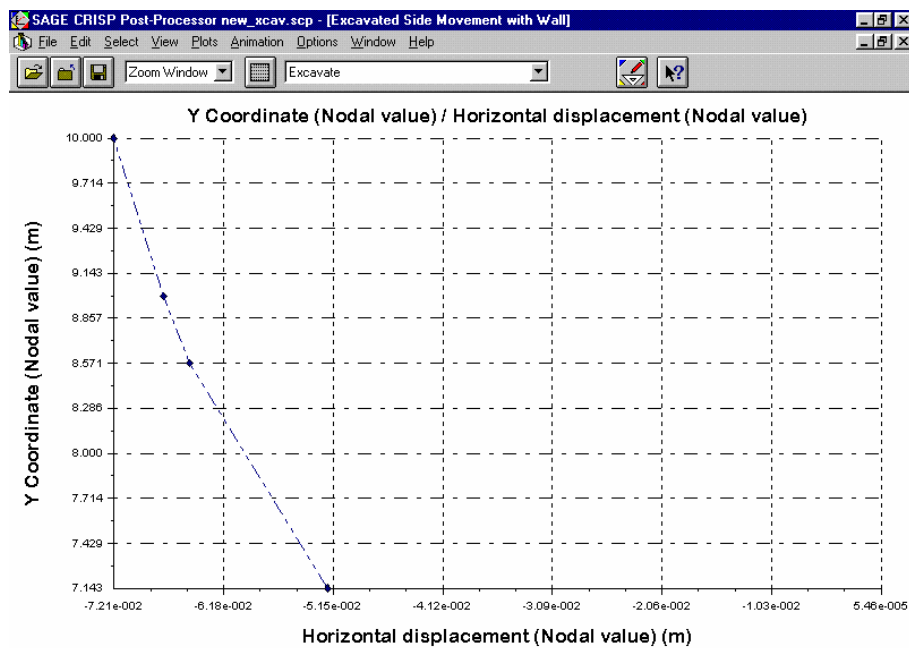
Figure 5-37:Basement Heave

Figure 5-38:Wall deflection

5.7 Summary

By the end of this example you will have made use of the following features:

■ Autosave

■ Units

■ The Structured Mesh Generator

■ Creating finite elements manually

■ Construction sequences

■ Super imposing elements

■ Removal of in situ elements

■ Displacement plots

■ Instance graphs

5.8 Things To Try Next

■ Try applying a horizontal fixity to the second node down the wall, during the excavation stage. This will model a perfectly stiff prop.

■ Applying a Beam or Bar element to the wall to model the prop forces in the wall.

CENTRIFUGE TEST OF AN EMBANKMENT

6.1   Introduction

This example is taken from the paper "Numerical modelling of a centrifuged embankment on soft clay", by Almeida, Britto and Parry (see the Publications Directory). It is designed to introduce you to a centrifuge type analysis, with coupled-consolidation (Biot) drainage conditions, using the Modified Cam clay soil model and combining automatic mesh generation with manual, finite element creation. The mesh used is almost identical to the mesh described in the paper, with some slight modifications to simplify the mesh creation process. Results from the analysis are then compared with observed results, obtained from an experimental centrifuge analysis on a prototype model of the embankment.

The subject of the example is the modelling of a sand embankment constructed on soft clay.

The following steps will be covered:

- Starting The Project

- Defining Units

- Creating The Finite Element Mesh

- Defining A Material Zone

- Assigning Element Properties

- Defining In Situ Stress Conditions

- Defining Increment Blocks

- Defining Boundary Conditions

- Defining Loads

- Scale Factors

- Running The Analysis

- Displacement Plots

- Duration Graphs

6.2 The Problem

The analysis simulates the construction of a sand embankment on an overconsolidated clay crust overlying a normally consolidated clay layer. The embankment is constructed in five stages. After each of the first three stages, time is allowed for the equalisation of pore pressures to occur. The final two layers are then constructed relatively quickly in order to induce failure.

6.3 Pre-Processing

This section describes all of the stages that must be completed using the Pre-Processor in order to run the finite element analysis. It is recommended that you save the Project regularly (by clicking on Save Project in the File menu).

6.3.1 Starting The Project

This section describes the basic processes that should be completed in advance of creating your mesh. The steps you will follow include:

- ■ Selecting a mesh mode

- ■ Entering Project set-up information

- ■ Entering QA information

- ■ Verifying the base units system

The following procedure describes the actions required to start and set-up a new Project.

Step 1   On opening the CRISP2D Pre-Processor, select New Project... from the File menu. This displays the Mesh Mode dialogue box:
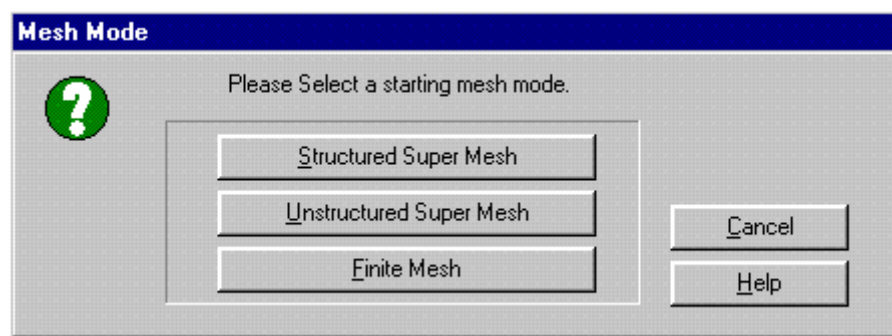

Figure 6-1:Mesh Mode dialogue box

Step 2   Click on the Structured Super Mesh button. This exits the Mesh Mode dialogue box, sets you up in Structured Super Mesh mode and opens the Project Setup dialogue box.
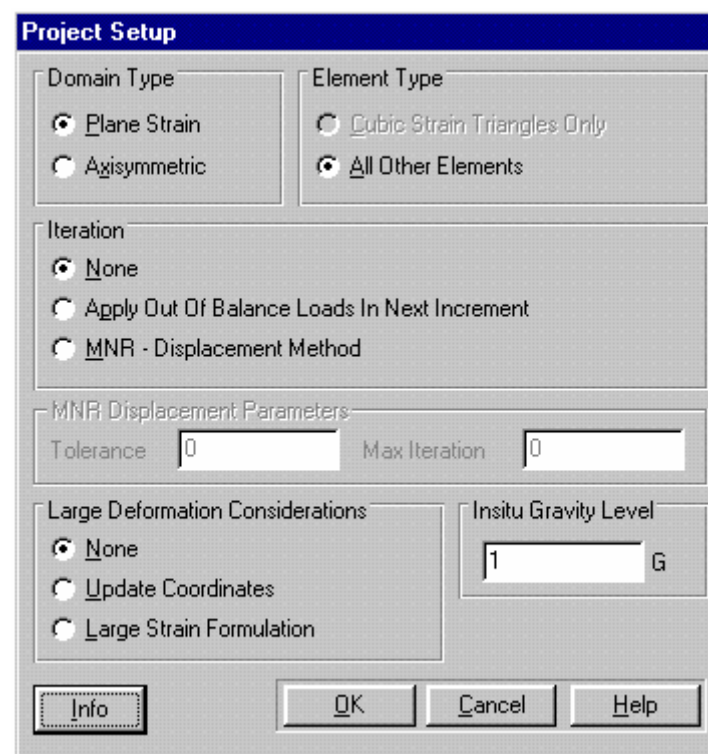
Figure 6-2:Project Setup dialogue box

Step 3   In the Domain Type section, click on the Plane Strain option button and click on All Other Elements in the Element Type section.

Step 4   The Project Setup should now look like Figure 4-3, with an in situ gravity level of 1G, the Apply in Next Increment and Update After Each Increment check boxes unchecked and Use The Standard Program selected.

Step 5   Click on the Info... button. This displays the Project Information dialogue box which allows you to enter QA information such as job number, operator name, checker name, etc.
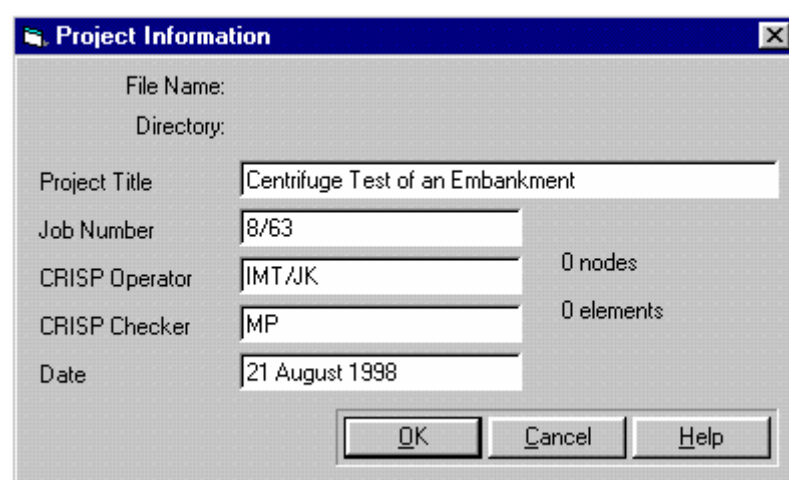

Figure 6-3:Project Information dialogue box

Step 6	Enter data as appropriate. The date is automatically entered but may be manually altered.
Step 7	The Project Information dialogue box should now look similar to Figure 6-3. Click on OK to return to the Project Setup dialogue box, once you have finished.
Step 8	Click on OK to exit.

Defining Units

At this early stage it is useful to ensure that the correct system of base units is being used.

Step 1	In the Options menu, click on Units... to display the Units dialogue box.

Figure 6-4:Units dialogue box

Step 2	Use the list boxes to select the appropriate units for Distance, Force, Time and Angle as shown in Click on OK to exit, once you have finished.

6.3.2 Creating A Mesh Using The Structured Mesh Generator

The Structured Super Mesh mode provides a method of generating large numbers of finite elements from a few super elements. For this example, the super mesh is used to automatically generate the lower portion of the mesh (comprising the two clay layers), with the remainder (the sand embankment) created manually.

This section covers all of the stages you must have completed in order to automatically generate the finite element mesh. The steps you will follow include:

- Creating a super mesh

- Defining edge gradings

- Defining material zones

- Assigning element properties

■   Generating the finite element mesh

6.3.3   Creating The Super Mesh

Nodal co-ordinates for the super mesh are taken straight out of the paper to which this example is attributed. The easiest way of creating nodes at these precise co-ordinates is to create them manually in the Super Node List dialogue box.

Step 1   In the Mesh menu, click on Super Node List... to display the Super Node List dialogue box.
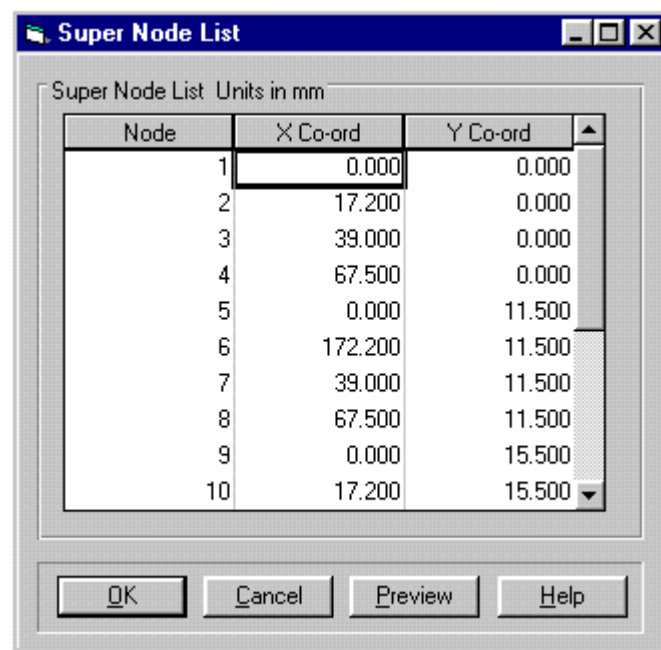

Figure 6-5:Super Node List dialogue box

Step 2   Enter the following nodes into the Node List box, using the TAB key to move between the X and Y co-ordinate columns. Each time you enter a Y co-ordinate for the last node in the list, press TAB to create a new node and move into its X co-ordinate column. Once you have entered all the nodes, click on OK to display them.

Node	X Co-ordinate	Y Co-ordinate
1	0	0
2	17.2	0
3	39	0
4	67.5	0
5	0	11.5
6	17.2	11.5
7	39	11.5
8	67.5	11.5
9	0	15.5
10	17.2	15.5
11	39	15.5

Node	X Co-ordinate	Y Co-ordinate
12	67.5	15.5
13	0	16.4
14	17.2	16.4
15	39	16.4
16	67.5	16.4
17	20.8	14.2
18	35.4	14.2
19	20.8	15.5
20	35.4	15.5

Step 3　　In the View menu, click on Zoom and then on Full Page. The Graphics Window will resize to display the nodes at optimum scale.

Step 4　　In the Mesh menu, click on Create Super Elements. The fact that you can now create elements, is confirmed by the mode indicator in the status bar that runs along the bottom of the screen.

Step 5　　Join the nodes together to form the super mesh depicted in   To create each super element, you must click on each of its vertex nodes in turn, and then click again on the first node to complete the element. Remember that you do not ever need to press CTRL because you are not creating any new nodes.

Tip　　If you make a mistake, you can backtrack one node at a time through the elements you have created simply by clicking again on each node.

Step 6　　When you have finished, you should have eleven four sided super elements in a mesh identical to that shown in Figure 6-6.
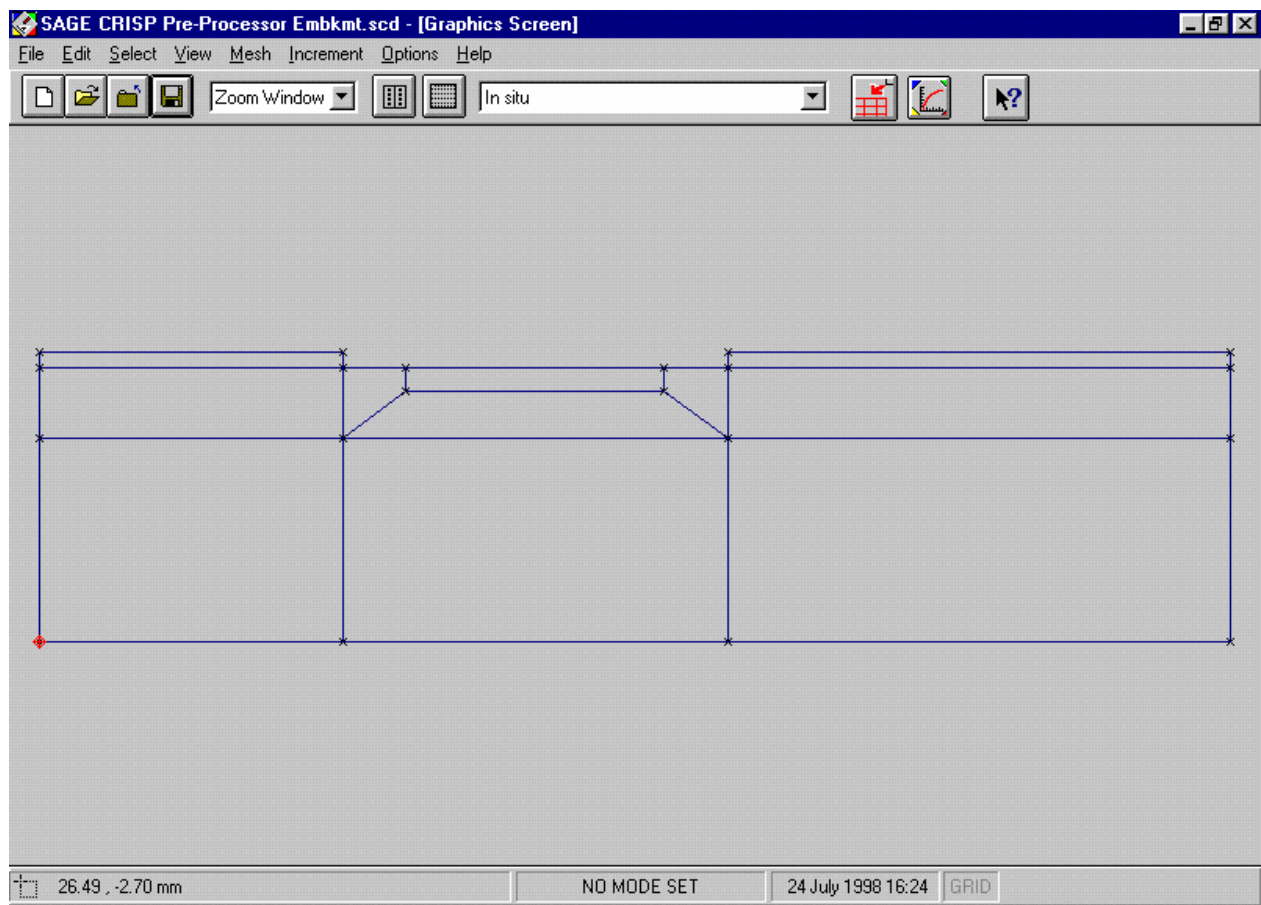
Figure 6-6:Structured Super Mesh

> ***Step 7***   In the Edit menu, click on Commit Super Element Creation. This fixes the
> elements, confirming that they are correctly created. The elements will change
> colour when they committed. If you are not up to speed with creating elements,
> it is a good idea to Commit elements that have been correctly created as soon as
> each is finished. This way, if you make a mistake you can use the Abort
> Super Element Creation feature to undo all changes made since Commit
> Super Element Creation was last clicked on.

Defining Edge Gradings

Edge Gradings define how the super elements will be divided up into finite elements by the mesh
generator. In general, better results will be obtained by using smaller finite elements in the parts of the
mesh that experience the most rapid and/or the largest changes in stress, during the course of the analysis.

The following procedure defines gradings for the entire super mesh.

> ***Step 1***   In the Select menu, click on Super Edges. Select the two leftmost, horizontal,
> edges with a y co-ordinate of 15.5.

> ***Step 2***   In the Mesh menu, click on Super Edge Grading... to display the Multiple
> Super Edge Gradings dialogue box.

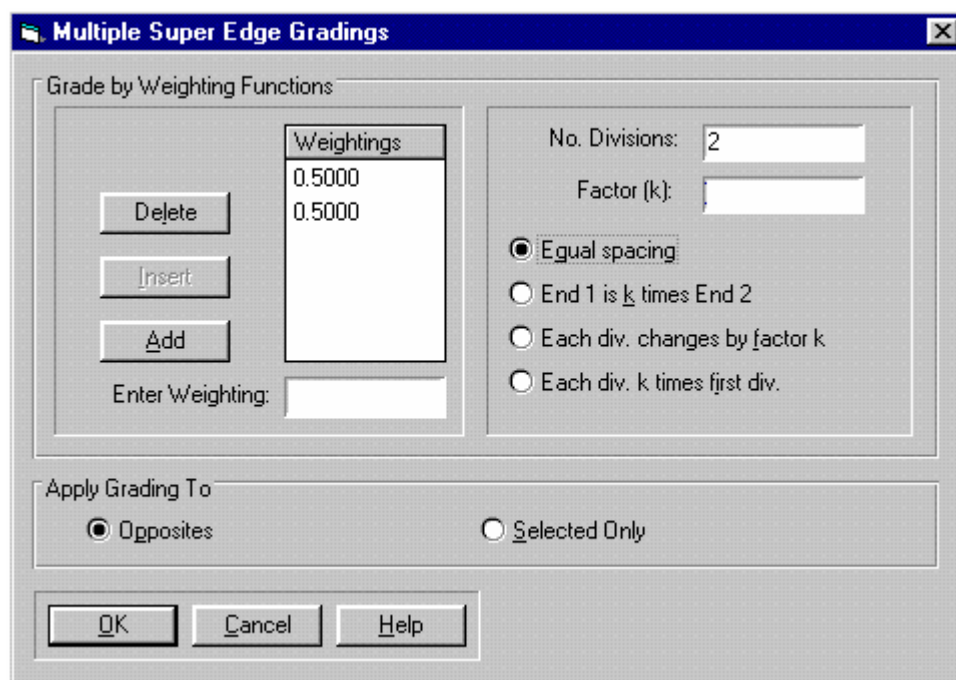*Figure 6-7:Defining Super Edge Gradings*

Step 3   Enter a value of '2' into the No. Divisions text box and click on the Equal Spacing option. Weighting values will be automatically entered into the list and the dialogue box should now look like

Step 4   Click on OK to exit, once you have finished.

Step 5   Opposite edges of each element are always graded in identical proportions, so by defining edge gradings for these two edges, gradings have also been automatically defined for six additional edges. Grading markers are shown on the Graphics Window as diamond shape boxes positioned on graded super edges. The mesh should now look similar to Figure 6-8.
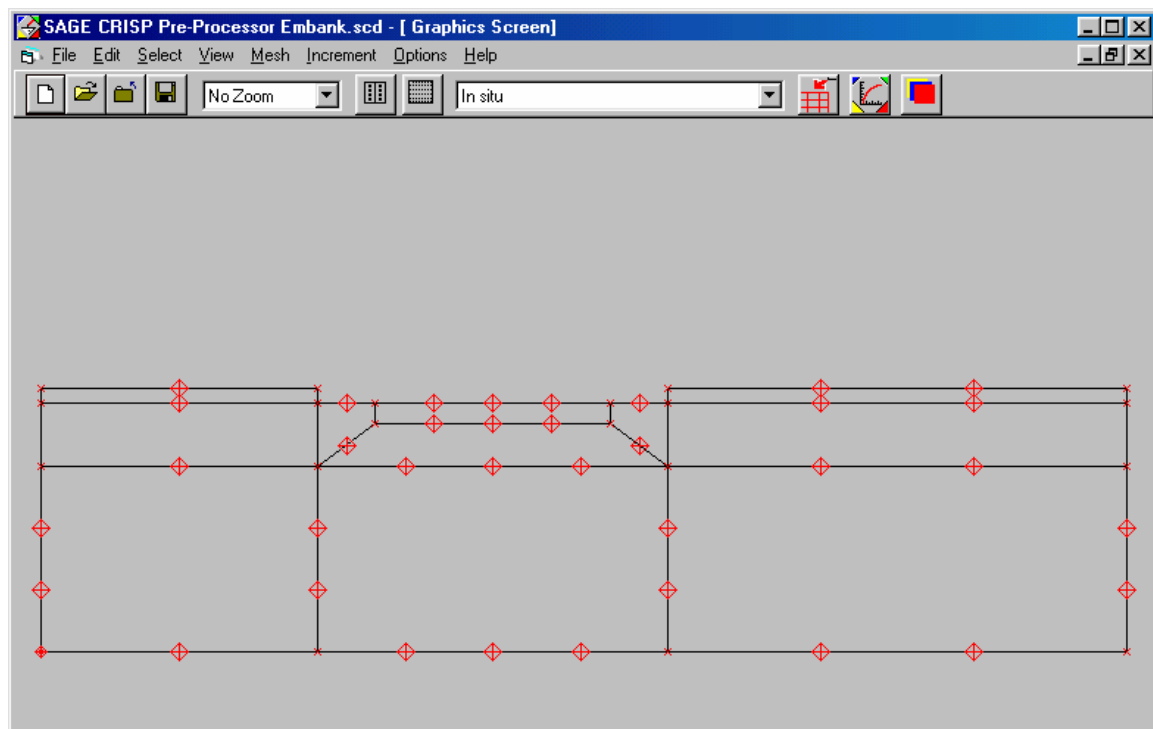
Figure 6-8:Super Edge Gradings

Step 6   In the Select menu, click on Clear Selection, or click again on the selected edges to de-select them. You can also use the short-cut key, CTRL+L, to clear the selection.

Step 7   Select the top two left-hand vertical edges and the uppermost right-hand vertical edge.

Step 8   In the Mesh menu, click on Super Edge Grading... to display the Multiple Super Edge Grading dialogue box.
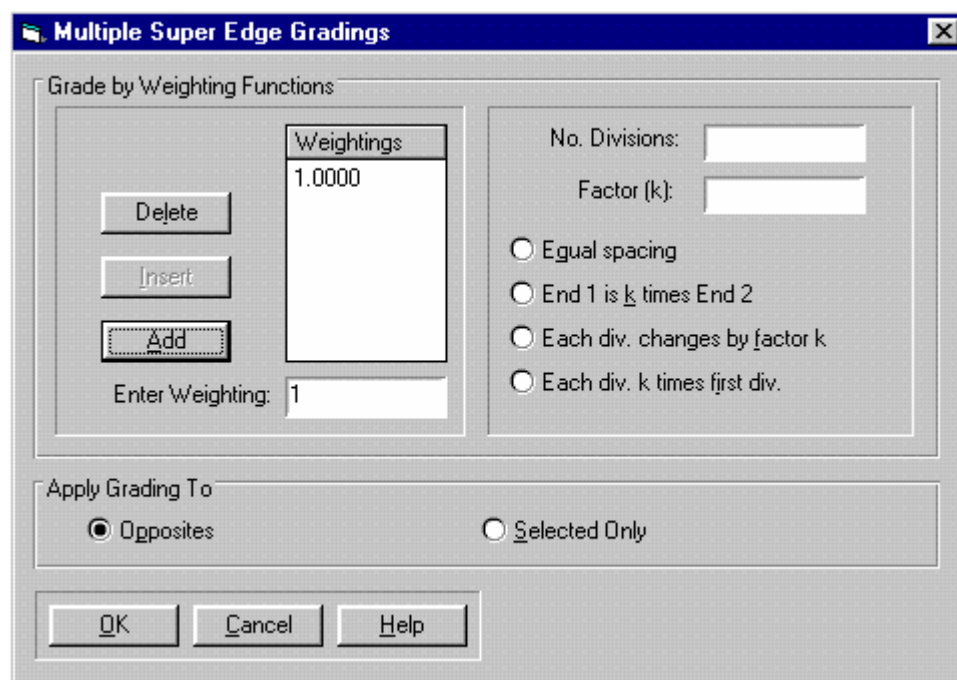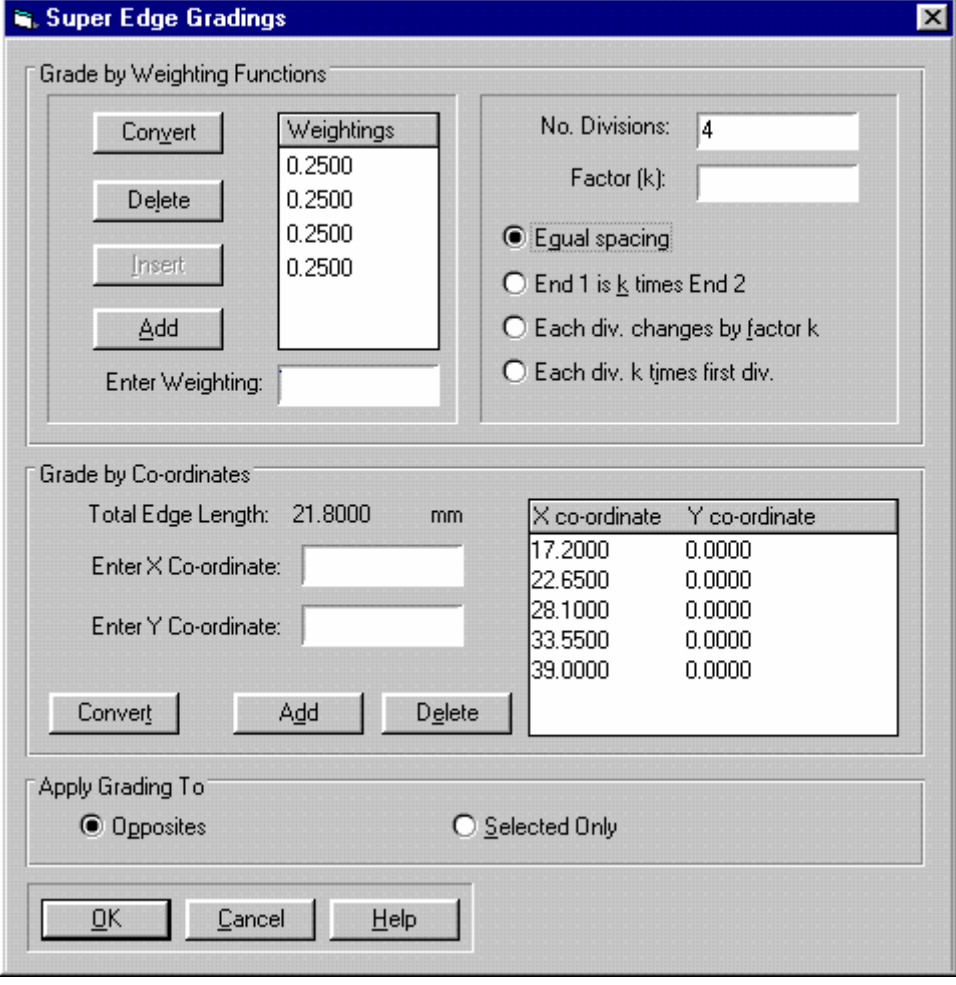
Figure 6-9:Defining Super Edge Gradings

Step 9   Type '1' in the Enter Weighting text box and click on Add. This will add a single grading of 1.0 to the list. The dialogue box should now look like

Step 10   Click on **OK** to exit.

Step 11   Clear the current selection and then select the bottom middle horizontal edge.

Step 12   Assign it four equally spaced divisions, as described above (in steps 2 to 4). The dialogue box should look like when you have finished.

Figure 6-10:Super Edge Grading dialogue box

Step 13   Clear the selection and then select the bottom left-hand vertical edge and the bottom right-hand horizontal edge.

Step 14   Assign them three equally spaced divisions, as described above.

Step 15   This concludes the definition of edge gradings. Every edge in the mesh should now have gradings assigned to it. To check this, make sure that each edge in the mesh is displayed with at least one grading marker on it, except for those edges with just one division. The final mesh should look similar to
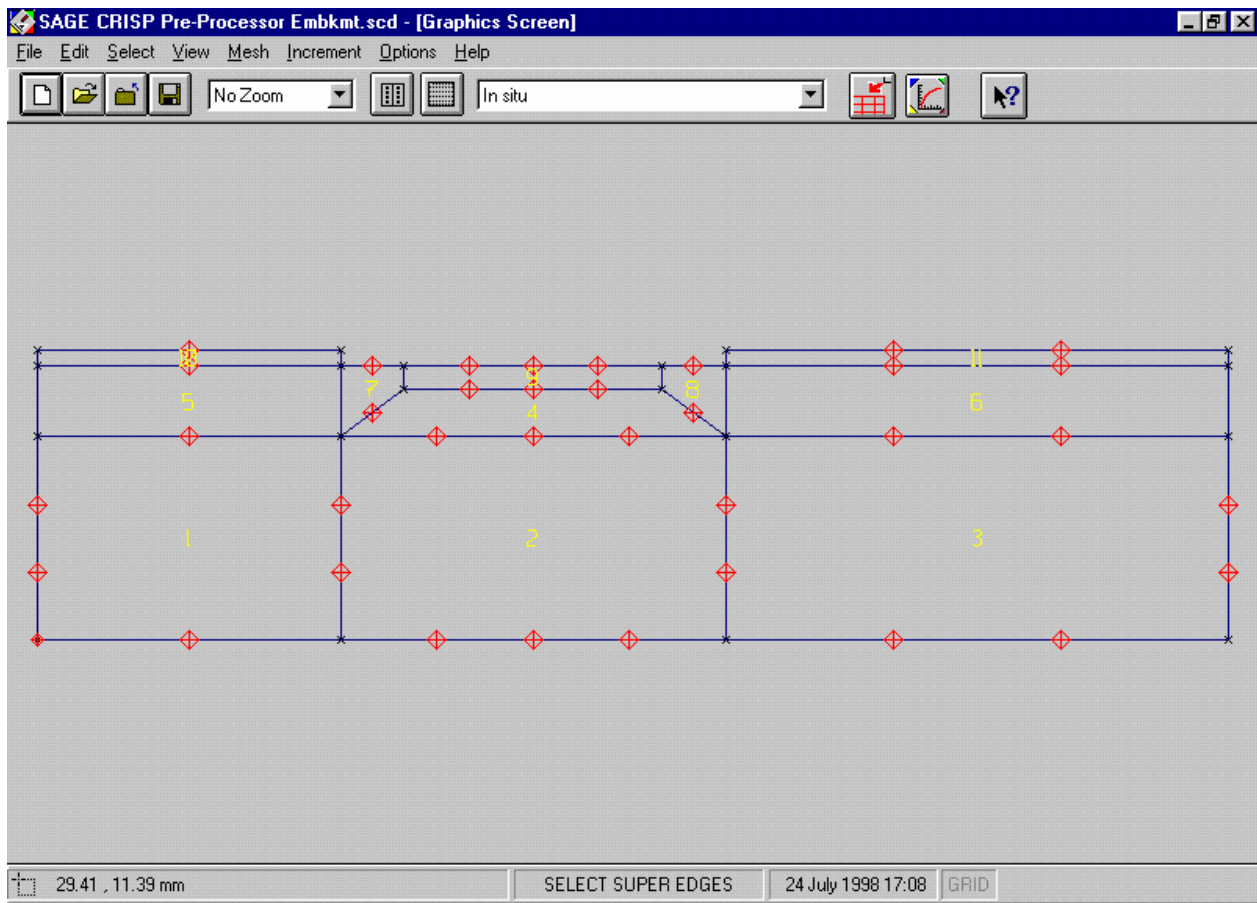
Figure 6-11:Fully Assigned Super Edge Gradings

6.3.4   Defining Material Zones

A Material Zone contains information about the soil model and associated soil parameters that are being used to model particular regions of the mesh.

It is necessary to define the number of material zones at this point in order to be able to assign elements to zones (the next stage).

Once you have set up a zone, the soil model and parameters may be entered or changed at any time prior to the analysis being run.

> ***Step 1***   Select Material Properties... from the Mesh   menu.   This displays the Material Properties dialogue box.

Figure 6-12:Material Properties dialogue box

Step 2 Enter the name "Kaolin". Select "Consolidating" from the Soil Condition list box and "Modified Cam clay" from the Soil Models list box. Material zones are referenced by name elsewhere in the Project, so it is good practice to give each zone a meaningful title here.

Step 3 Enter the following material property values:

Property	Value	Units
κ	0.05	None
λ	0.25	None
e_{cs}	2.44	None
M	0.9	None
G	3.2E3	kN/m^2
ν	0	None
γ_w	9.81	kN/m^3
γ_{bulk}	15.4	kN/m^3
k_x	2.0E-9	m/sec
k_y	2.0E-9	m/sec

Step 4 The Material Properties dialogue box should now look similar to

Step 5   Click on the New button to create a second soil model and enter the name "Gault clay". Select "Consolidating" soil condition and the "Modified Cam Clay" model, then enter the following material property values:

Property	Value	Units
κ	0.035	None
λ	0.219	None
e_{cs}	1.96	None
M	1	None
G	3.5E3	kN/m^2
ν	0	None
γ_w	9.81	kN/m^3
γ_{bulk}	15.4	kN/m^3
k_x	8.64E-5	m/sec
k_y	8.64E-5	m/sec

Step 6   Click on the New button to create a third soil model and enter the name "Sand". Select the "Elastic" model and enter the following material property values:

Property	Value	Units
E_h	3.0E3	kN/m^2
E_v	3.0E3	kN/m^2
v_{vh}	0.3	None
v_{hh}	0.3	None
G_{hv}	937.5	kN/m^2
γ_w	9.81	kN/m^3
γ_{bulk}	16.9	kN/m^3
k_x	0	m/sec
k_y	0	m/sec

Step 7   Click on OK to exit, once you have finished.

Tip   Although values for the soil properties have been defined here, it is possible to delay defining them right up until just before you run the analysis.

6.3.5 Assigning Element Properties

Every element in the mesh has two properties associated with it:

■ Finite element type, and

■ Material zone

These properties are assigned to finite elements in the following manner:

Step 1 In the Select menu, click on Domain Super Elements and select the bottom three super elements by clicking on them.

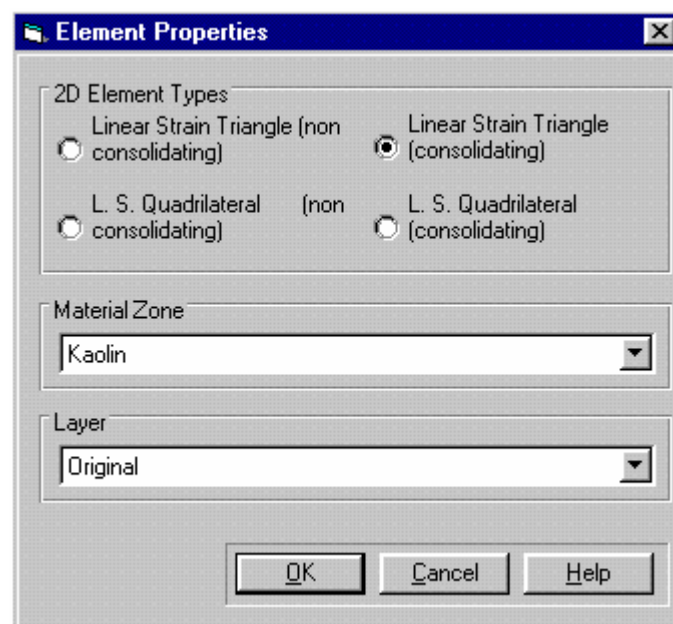*Step 2* In the Mesh menu, click on Element Properties... to display the Element Properties dialogue box.

Figure 6-13:Assigning Element types and Material Zones

Step 3 Select the Linear Strain Triangle (consolidating) option.

Step 4 Select 'Kaolin' from the Material Zone list box.

Step 5 The Element Properties dialogue box should now look like Figure 6-13. Click on OK to exit, once you have finished.

Step 6 In the Select menu, click on Clear Selection. Select the two narrow elements in the top row (one on the left, and one on the right).

Step 7 Define the element type as L. S. Quadrilateral (consolidating) and the material zone as Gault clay.

Step 8 In the Select menu, click on Clear Selection. Select the remaining elements (in a horizontal swathe across the middle of the mesh).

Step 9   Define the element type as Linear Strain Triangle (consolidating) and the material zone as Gault clay.

Step 10   This concludes the definition of element properties.

6.3.6   Generating The Finite Element Mesh

You are now at a stage where you can generate the finite element mesh from your super mesh.

Step 1   In the Mesh menu, click on Generate Finite Element Mesh. This will first check that you have correctly graded every edge and that you have assigned a generated finite element type to each super element. It will then spend a short time performing the automatic generation of the finite element mesh.

Step 2   Provided you have completed the above stages correctly, the finite element mesh will be displayed as shown in Figure 6-14 and the main menus will change, to allow you access to the additional options available in Finite Element Mesh mode.
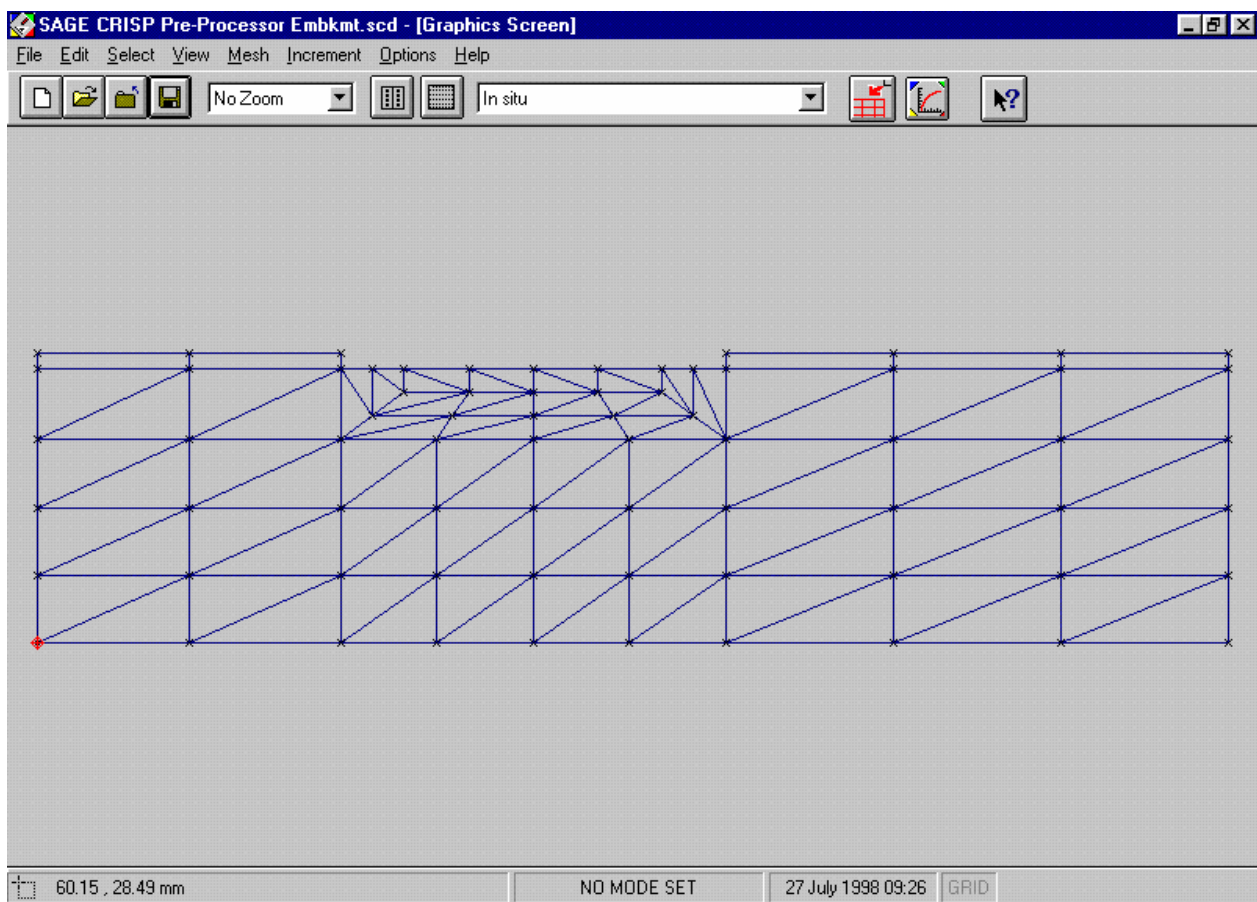
Figure 6-14:Automatically Generated Finite Element Mesh

6.3.7 Defining In Situ Stresses

In situ conditions are defined in terms of stresses at different elevations within the mesh. In this example, all of the stresses are defined in terms of *effective stress*. The effective vertical stresses have been calculated directly from the bulk modulus of soil (specified earlier in the defining material zones stage) and the bulk modulus of water (taken as 9.81 kN/m^3). The other parameters have been derived from the paper.

Step 1 In the Increment menu, click on Define In Situ Stress Conditions... to display the In Situ Stress Setup dialogue box.

In Situ Stress Setup

	Height [mm]	σ'_{xx} [kN/mm²]	σ'_{yy} [kN/mm²]	σ'_{zz} [kN/mm²]	τ_{xy} [kN/mm²]	PWP [kN/mm²]	$\bar{U}$ [kN/mm²]	P'c [kN/mm²]
1	16.4	0	0	0	0	0	0	0
2	15.5	12.78	5.03	12.78	0	8.83	0	120
3	13.5	17.33	16.21	17.33	0	28.45	0	62
4	11.5	21.88	27.39	21.88	0	48.07	0	49.1
5	8.8	30.3	42.54	30.3	0	74.5	0	41
6	0	64.3	91.68	64.3	0	160.88	0	87.9

Plot ☐ ☐ ☐ ☐ ☐ ☐ ☐

Insert Delete OK Cancel Plot Convert Height

Figure 6-15:In Situ Stress Setup dialogue box

Step 2 Select the second row of the in situ stresses grid by clicking on the row number (to the left of the Height column). This enables the Insert button, which you should click on four times to create the required number of reference elevations.

Step 3 Enter the values as shown in Figure 6-15. Click on OK to exit, once you have entered and double-checked the values.

6.3.8 Defining Increment Blocks

Increments are the units by which the finite element program advances the analysis. In each increment, the effect of applied load and boundary conditions is calculated and the accumulated strains and pore water pressures are updated. Increments are grouped together in blocks which then model particular events in the analysis.

This analysis requires eleven increment blocks. The first block consists of only one increment and allows the water table to be defined. The subsequent ten blocks, of ten increments each, allow the embankment to be built in five stages, with a consolidation period after each stage.

Step 1 In the Increment menu, click on Define Increment Block Parameters... to display the Increment Block Information dialogue box.

Figure 6-16 :Defining Increment Blocks

Step 2   Change the name of the first increment block from "Increment Block (untitled)" to "Define Water Table" by entering this into the Description of Block text box.

Step 3   Assign a time-step of 100 seconds by entering '100' into the Time-Step for Block text box.

Step 4   Click on the New button to create a new increment block.

Step 5   Change the name to "Add Layer 1".

Step 6   Click on OK to exit.

Step 7   In the Options menu, click on Units... and change the units for time from seconds to days.

Step 8   In the Increment menu, click on Define Increment Block Parameters....

Step 9   Click on 'Add Layer 1' in the Increment Block list.

Step 10   Enter a time-step of 2 days, and allocate ten increments to the block.

Step 11   Define nine more increment blocks with the following properties:

Block No.	Description	No. of Incs	Total Time-Step
3	Layer 1 consolidation	10	250 days
4	Add Layer 2	10	2 days
5	Layer 2 consolidation	10	648 days
6	Add Layer 3	10	2 days
7	Layer 3 consolidation	10	848 days
8	Add Layer 4	10	2 days
9	Layer 4 consolidation	10	28 days
10	Add Layer 5	10	2 days
11	Layer 5 consolidation	10	28 days

Step 12   The Increment Block Information dialogue box should now look like Figure 6-16. Note that the entry for the first time-step has been automatically converted into Days. Click on OK to exit, once you have finished.

6.3.9   Restricting Output

This analysis comprises 101 increments in eleven increment blocks. By default, output is written to disk for every integration point of every element, eighty four times (once at in situ stage plus once for each increment). This volume of output will take up approximately 40 Mb hard disk space. If you wish to limit this demand for space, you can restrict the volume of output written by only requesting it at specified increments. The procedure below describes how to request that output only be written for the last increment of each increment block.

Step 1   In the Increment menu, click on Specify Output... to display the Output Options dialogue box.
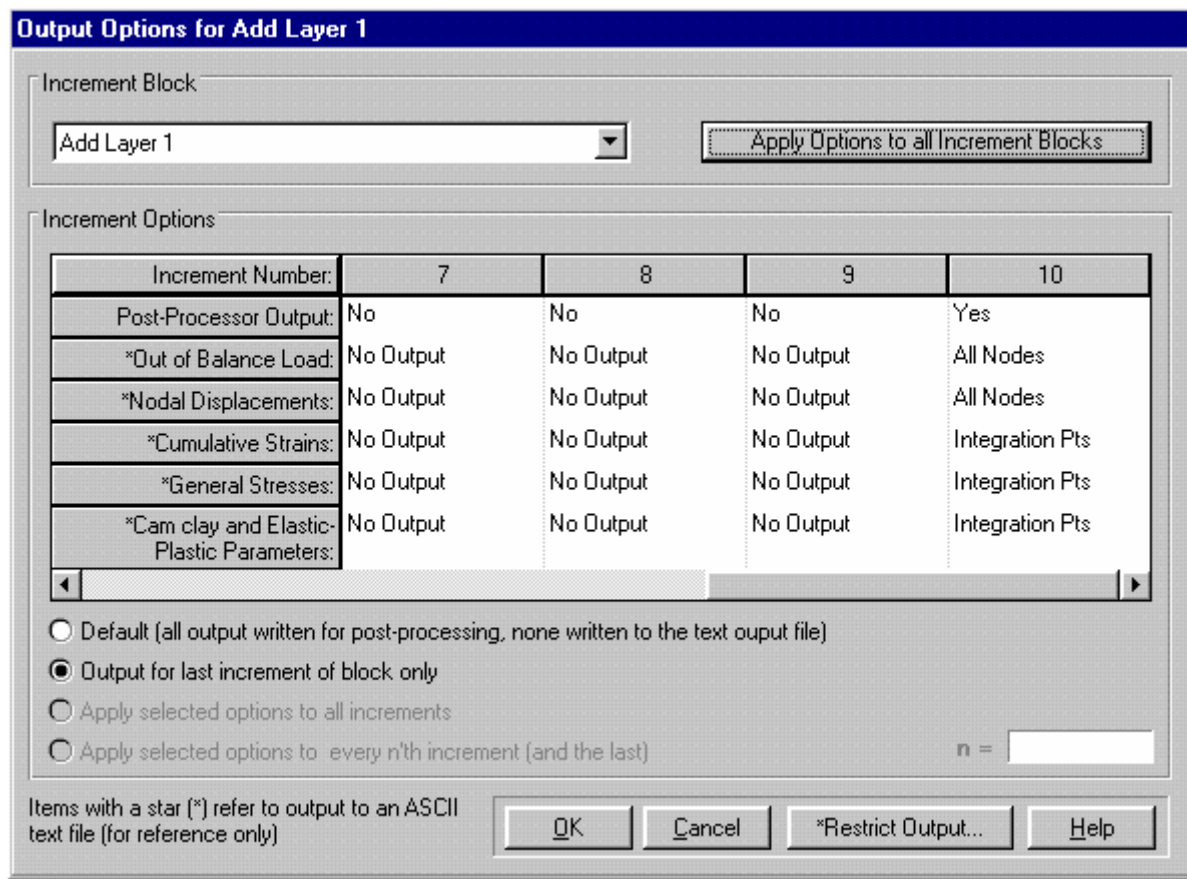
Figure 6-17:Specifying Restricted Output

Step 2   In the Increment Block list, click on 'Add layer 1'.

Step 3   Click on the 'Output for last increment of block only' option.

Step 4   Click on the Apply Options to all Increment Blocks button.

Step 5   The dialogue box should look similar to when you have finished. Click on OK to exit.

6.3.10 Creating The Embankment

The positions of some of the nodes in the embankment are defined in the paper by the positions of the sand layers in the centrifuge prototype. The easiest way to create a mesh for the embankment that matches the prototype is to specify these nodes directly (using the Node List) and then join the nodes to form elements. This is the same method as was used to build the super mesh.

The stages you will follow here include:

- Adding the nodes

- Saving the Project as a Bookmark

- Drawing the elements

Adding The Nodes

Step 1   In the Mesh menu, click on Node List... to display the finite element Node List dialogue box. Add the following nodes to the bottom of the list:

Node No.	X co-ordinate	Y co-ordinate
72	46.9	18.2
73	54.3	19
74	59.3	18.6
75	33.7	16.4
76	46	21.3
77	51.6	22.1
78	57.2	21.7
79	62.8	21.7
80	67.5	23.2
81	27.2	16.4
82	34.9	19.6
83	45.4	26.4
84	49.6	26.7
85	60.7	26.4
86	67.5	28.2
87	29.6	16.4
88	31.6	16.4
89	37.2	19.9
90	37.2	18.8
91	44	23.2
92	44	22.3
93	48.4	24.6
94	48.4	23.8
95	55.7	23.8
96	55.7	24.9
97	61.3	23.8
98	61.3	24.9
99	67.5	26.1
100	67.5	24.9
101	53.3	26.2
102	34	20
103	40	23
104	22.5	16.4

Step 2   Click OK to exit, once you have finished.

Making A Bookmark

It is a good idea to save the Project as a Bookmark before you start drawing the finite elements. This way, if anything goes seriously wrong with the elements that you draw you can easily return to this point simply by loading the saved Bookmark.

Step 1   Save the Project by clicking on Save Project in the file menu.

Step 2   In the Edit menu, click on Save As Bookmark... to display the Save Bookmark dialogue box.
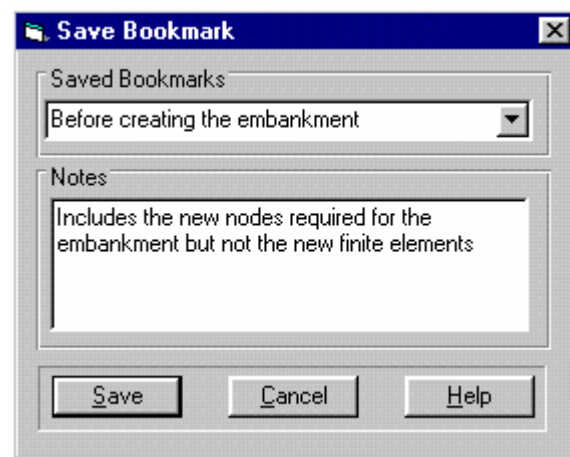
Figure 6-18:Saving a Bookmark

Step 3   Enter a descriptive save name and any notes you wish to add.

Step 4   Click on Save to store the bookmark and exit.

Drawing The Elements

Several features may be of use when you come to draw the elements:

You are informed when each element is complete by its changing colour. The colour will change again when you start creating a new element.

If you make a mistake, you can undo your changes by backtracking through the newly created elements by clicking again on the vertex nodes.

Remember not to try to start creating a new element from the last node of the element you have just completed as this will instead take you back a step.

If things go severely wrong, you can go back to the mesh as it existed before you began creating elements by loading the saved bookmark (click on Load Bookmark... in the Edit menu).

Step 5   In the Mesh menu, click on Create Elements.

Step 6   Join the nodes together to form triangular and quadrilateral elements as depicted in below.
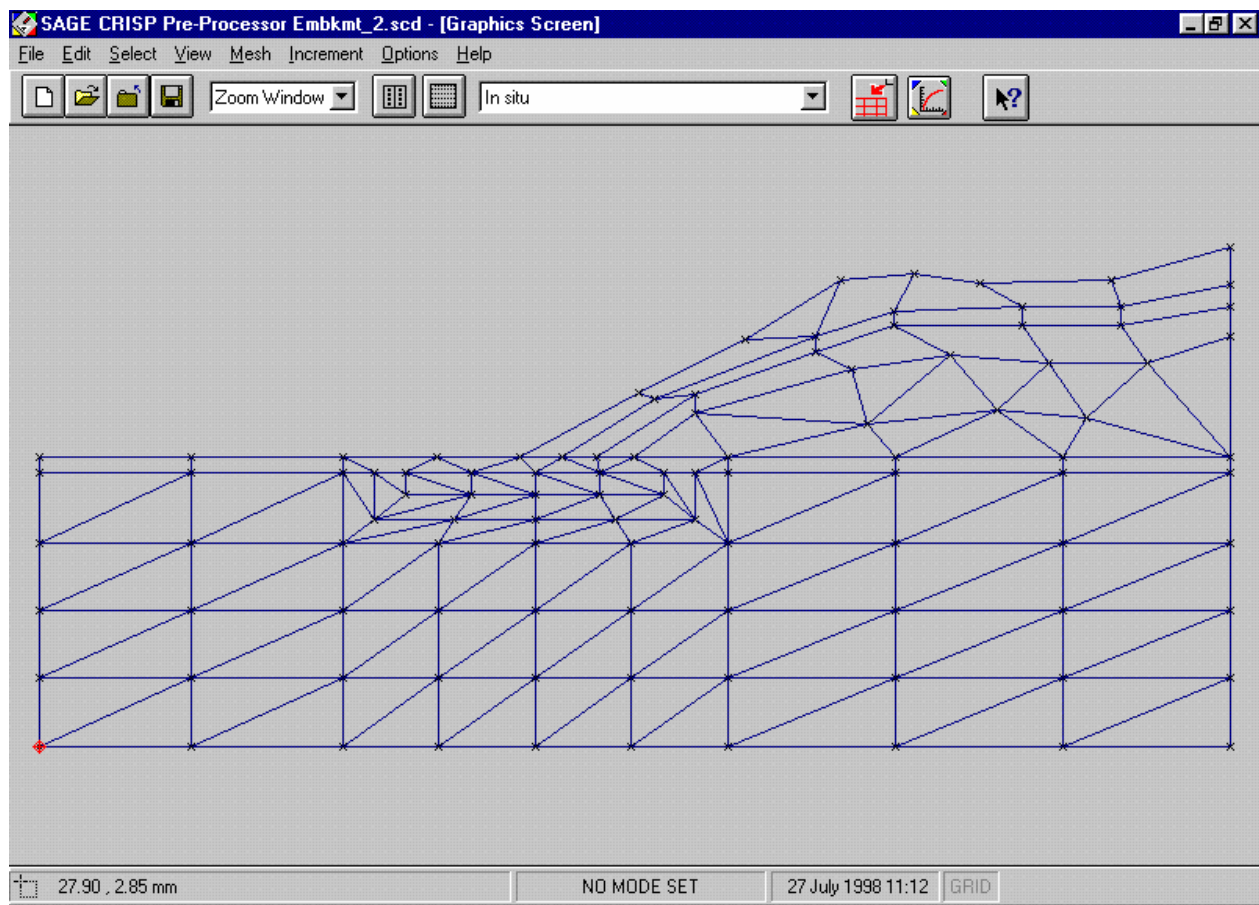
Figure 6-19:The Completed Embankment Mesh

Tip	The best way of exactly copying this mesh is to draw the elements layer by layer (see Figure 6-21 to Figure 6-27)
	As soon as you complete each layer, click on Commit in the Edit menu to make the changes permanent. If you make a mistake, you can then undo your changes by clicking on Abort Element Creation in the Edit menu. This will take you back to the stage you were at when you last clicked on Commit.

6.3.11 Setting Up The Construction Sequence

The construction of the embankment is modelled in five two-part stages. During each stage, a layer of sand is added over a short period of time. A longer period of time is then allowed for the excess pore pressures, induced by the weight of the additional layer, to dissipate, after which the next layer is added.

The following procedure describes how to add and remove elements to and from the mesh in order to simulate this construction sequence.

Step 1	Select 'In situ' from the Increment Block list box on the main toolbar.
Step 2	In the Select menu, click on Domain Elements and select all of the elements above or equal to a Y co-ordinate of 16.4 by clicking on them, as shown in Figure 6-20.
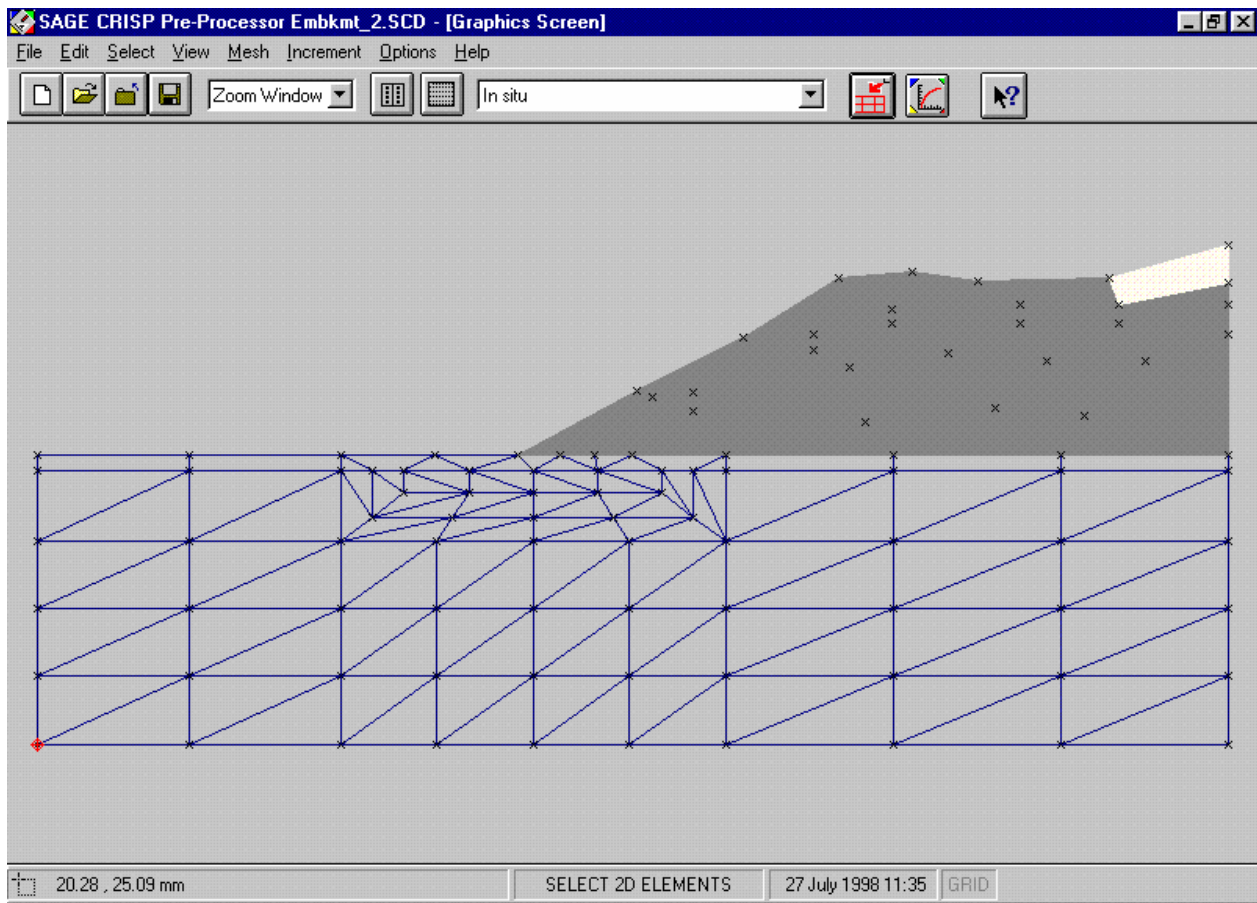
Figure 6-20:The Elements That Will Form the Embankment

Step 3   In the Increment menu, click on Remove Elements. This removes the elements from the current, and all subsequent increment blocks.

Step 4   Select 'Add Layer 1' from the Increment Block list box on the main toolbar.

Step 5   In the View menu, click on Removed Elements . This displays all elements that are not present in the primary mesh.

Step 6   In the Select menu, click on Clear Selection and select the elements shown in Figure 6-21 by clicking on them.
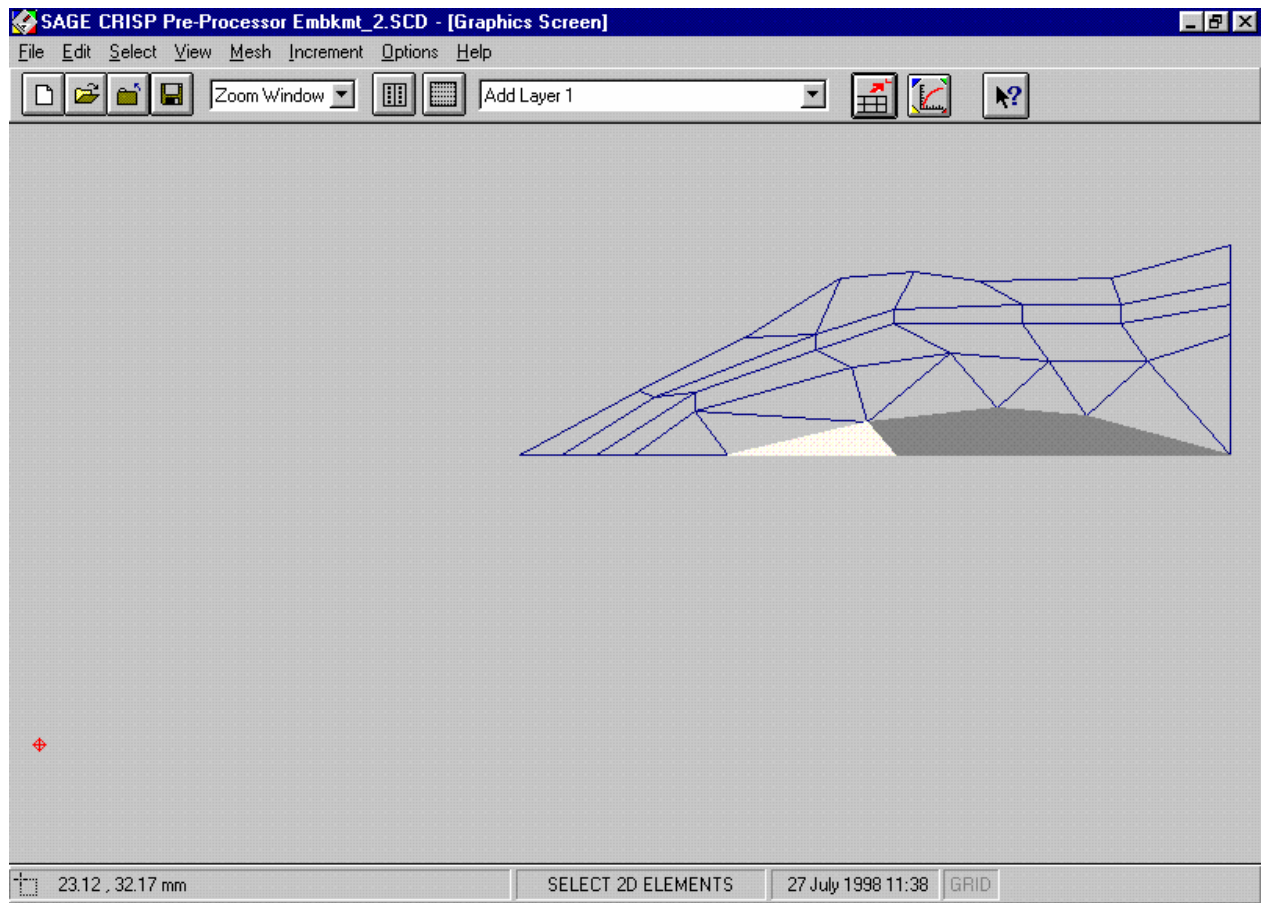
Figure 6-21:Add Layer 1

Step 7   In the Increment menu, click on Add Elements. This adds the elements to the current and all subsequent increment blocks, simulating the construction of the first layer of the embankment.

Step 8   Repeat steps 4 to 7 for each of the 'Add layer' increment blocks, selecting the elements shown and adding them to the correct increment blocks.

Remember to select the relevant increment block for each constructed layer from the Increment Block list box <u>before</u> you add the elements back into the primary mesh.
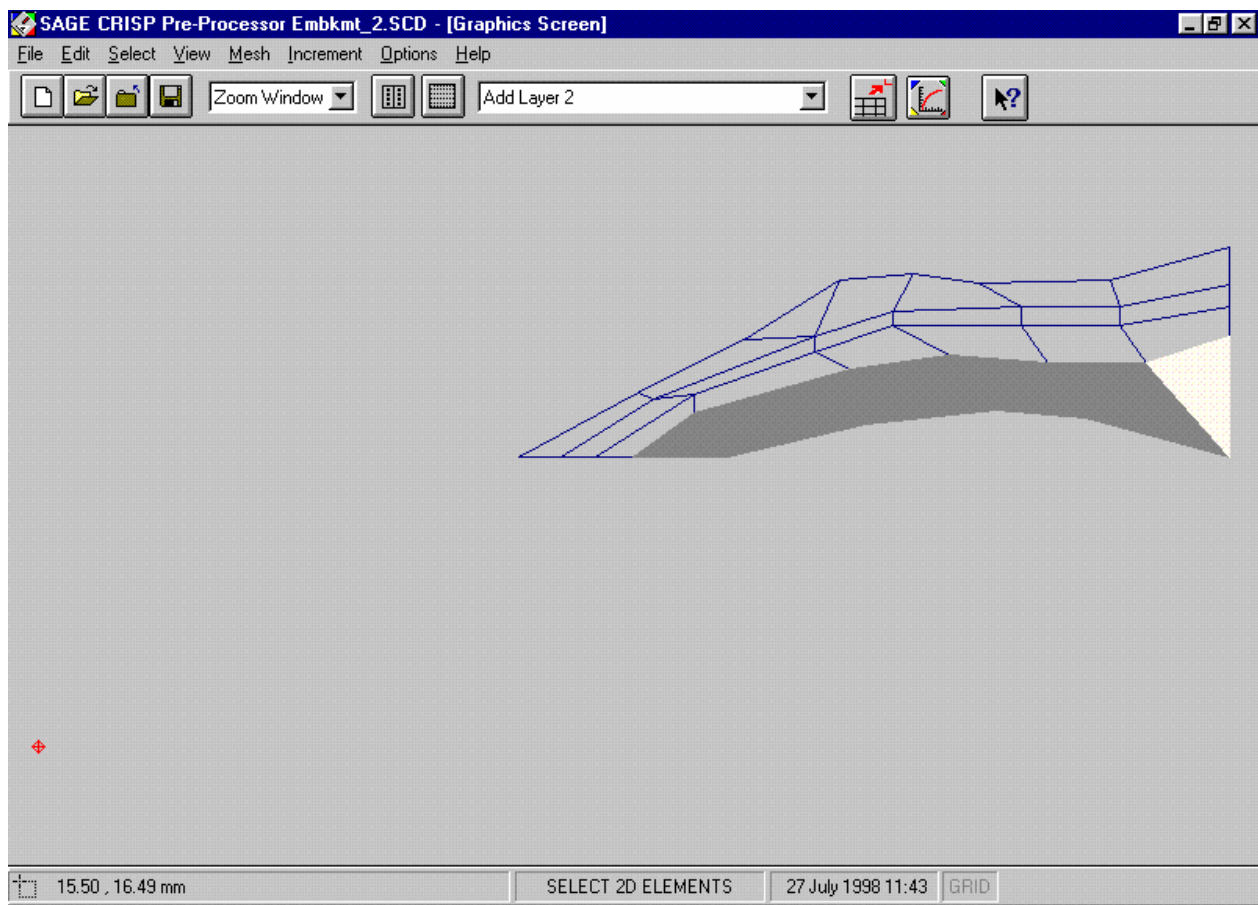
Figure 6-22:Add Layer 2
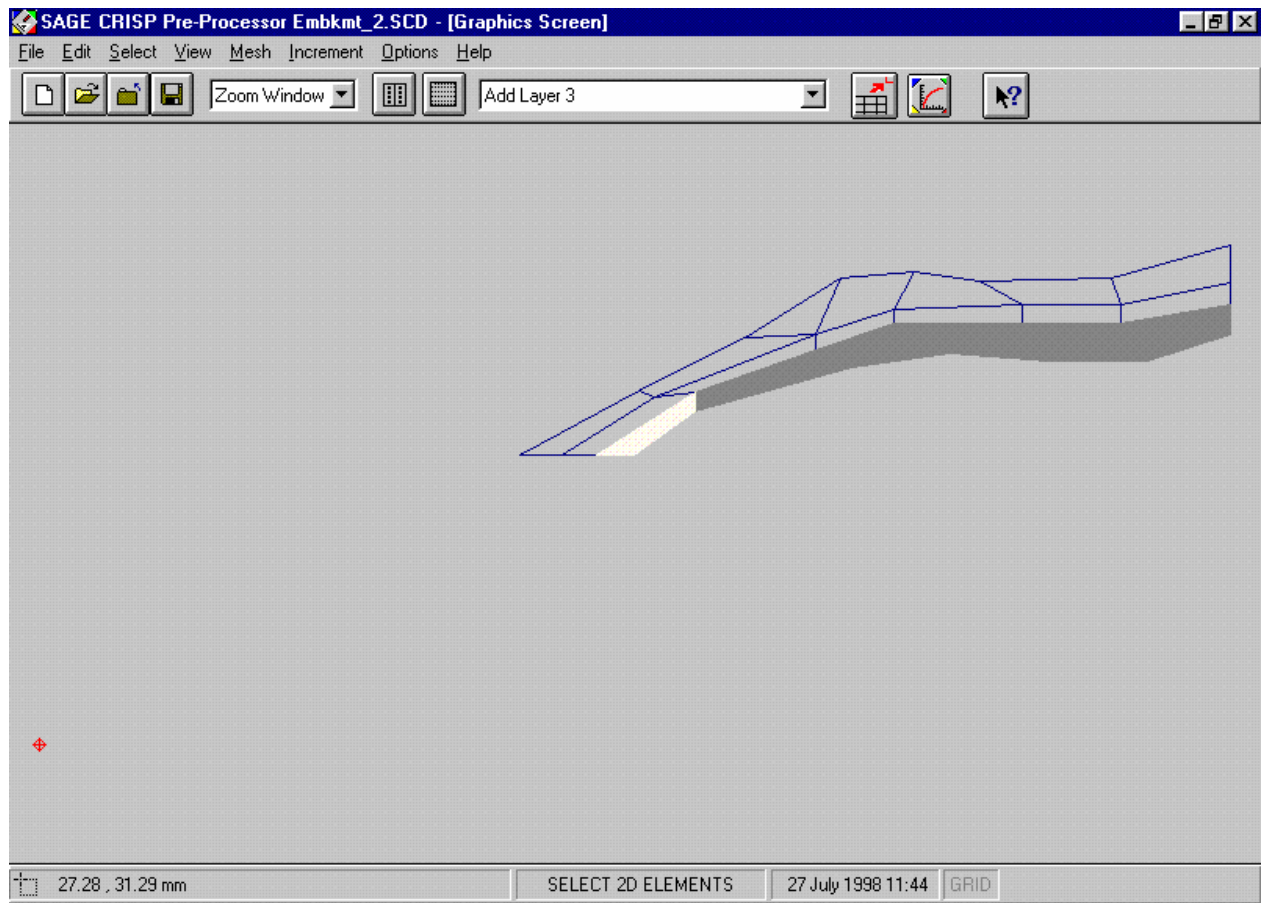
Figure 6-23:Add Layer 3
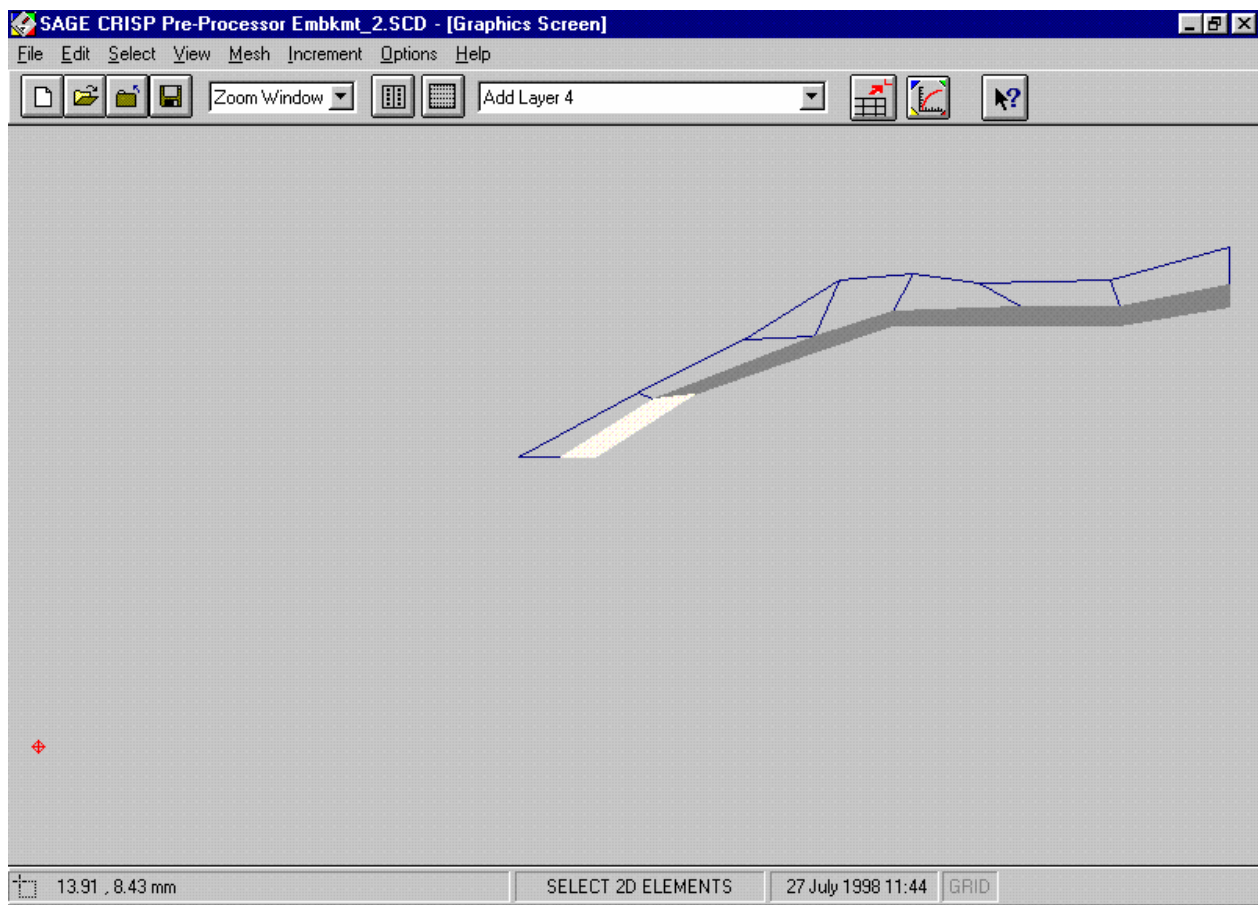
Figure 6-24:Add Layer 4
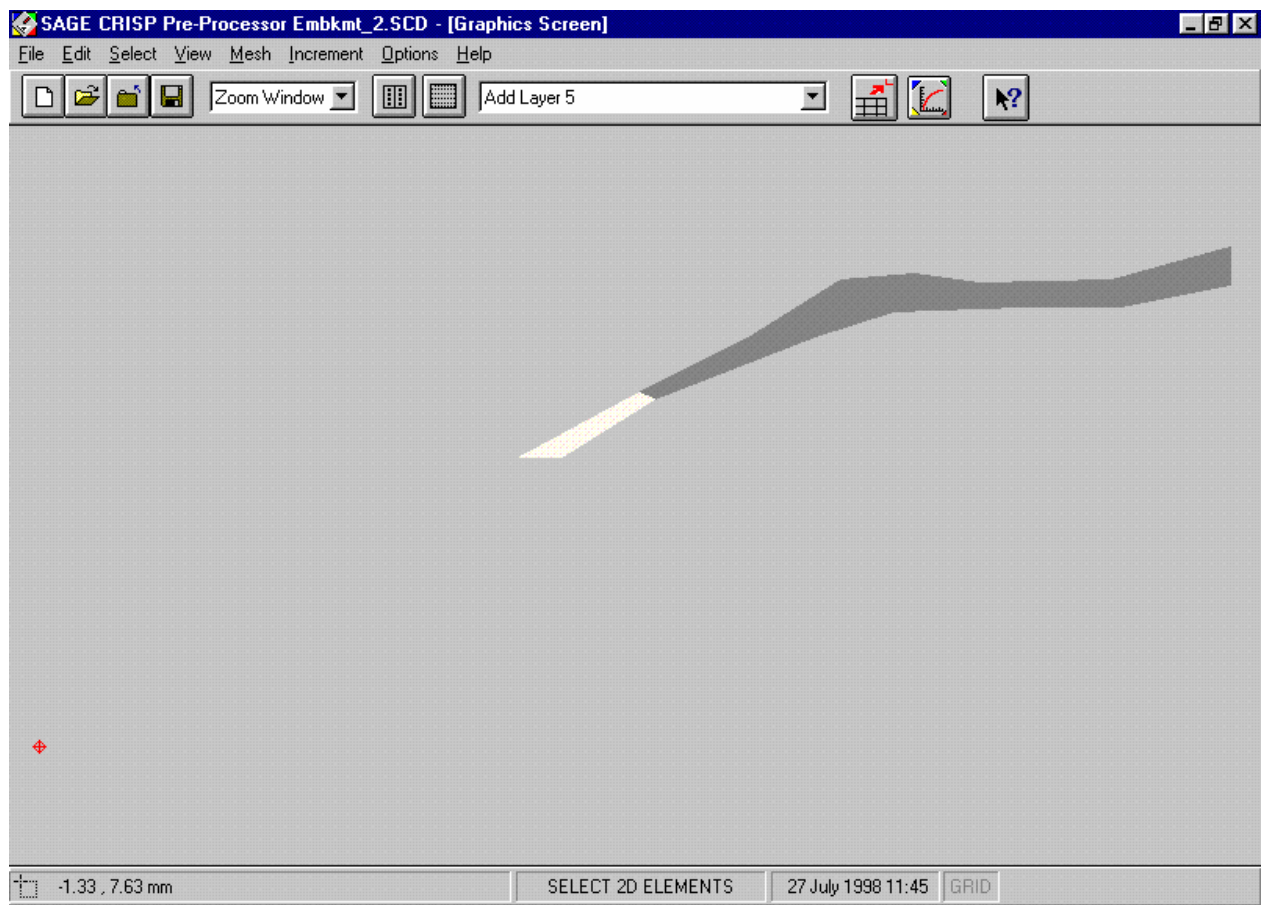
Figure 6-25:Add Layer 5

Step 9 In the View menu, click on Present Elements.

Step 10 You should now check that the sequence is correct. Click on the Increment Block list box so that the scrollable list of available increment blocks appears.

Step 11 Click once on 'In situ' so that the list disappears, leaving the In situ stage highlighted.

Step 12 Press the Down and Up cursor keys (↓ and ↑) to view the construction sequence as it progresses through the increment blocks. Ensure that the layers are added in the correct blocks.

Step 13 This concludes the construction sequence set-up.

6.3.12 Assigning Element Properties

All of the newly created finite elements need to be assigned a finite element type and a material zone. Elements in the embankment will all be non consolidating triangles or quadrilaterals in the 'Sand' material zone. The remaining new elements are consolidating triangles and quadrilaterals in the 'Gault clay' material zone.

Step 1   Select 'Add layer 5' from the Increment Block list box on the main toolbar. This increment block is chosen so that all of the elements in the mesh are present at once.

Step 2   To make it easier to see how far you have got with defining element properties, click on Legend in the View menu then click on Material Zones (also in the View menu).

Step 3   This displays the three material zones as coloured regions. The Legend contains the key to these colours.

Tip   To change the colours, click on Colour... in the Options menu to display the Colour Configuration dialogue box.

                To change a colour, double click on the coloured square adjacent to the material zone or finite element type that you wish to change and select a colour from the dialogue box that appears.

Step 4   In the Select menu, click on Domain Elements.

Step 5   Select all of the triangular elements in the embankment part of the mesh.

Step 6   In the Mesh menu, click on Element Properties... to display the element properties dialogue box.
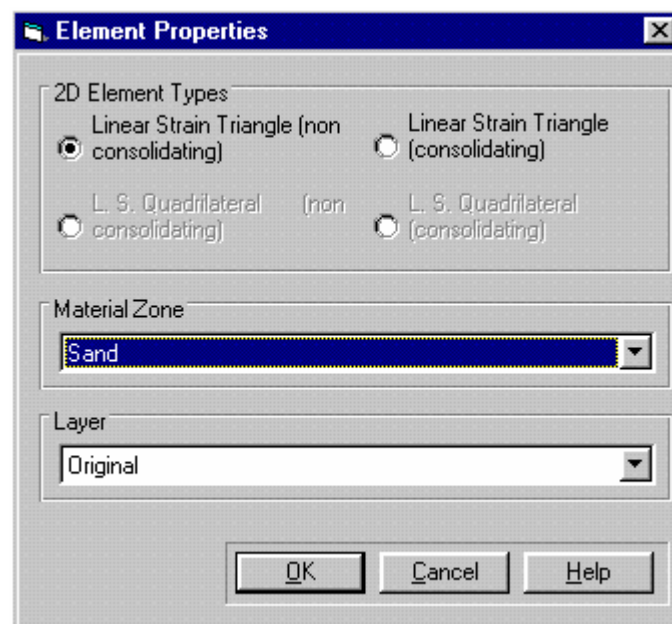
Figure 6-26: Element Properties for the Embankment

Step 7   Select the Linear Strain Triangle (non consolidating) option and the 'Sand' material zone. The Quadrilateral options are greyed out because you have selected three sided finite elements.

Step 8   Click on OK to exit, once you have finished.

Step 9	Clear the selection, then select all of the quadrilateral elements in the embankment part of the mesh.
Step 10	Assign these elements to the 'Sand' material zone and give them a L. S. Quadrilateral (non consolidating) finite element type.
Step 11	Clear the selection, then select all of the new triangular elements that you created beneath the toe of the embankment.
Step 12	Assign these elements to the 'Gault clay' material zone and give them a Linear Strain Triangle (consolidating) finite element type.
Step 13	Clear the selection, then select the two quadrilateral elements beneath the toe of the embankment.
Step 14	Assign these elements to the 'Gault clay' material zone and give them a L. S. Quadrilateral (consolidating) finite element type.
Step 15	You should check that all element types and material zones have been correctly assigned. When you are viewing material zones, the Graphics Window should look as shown below
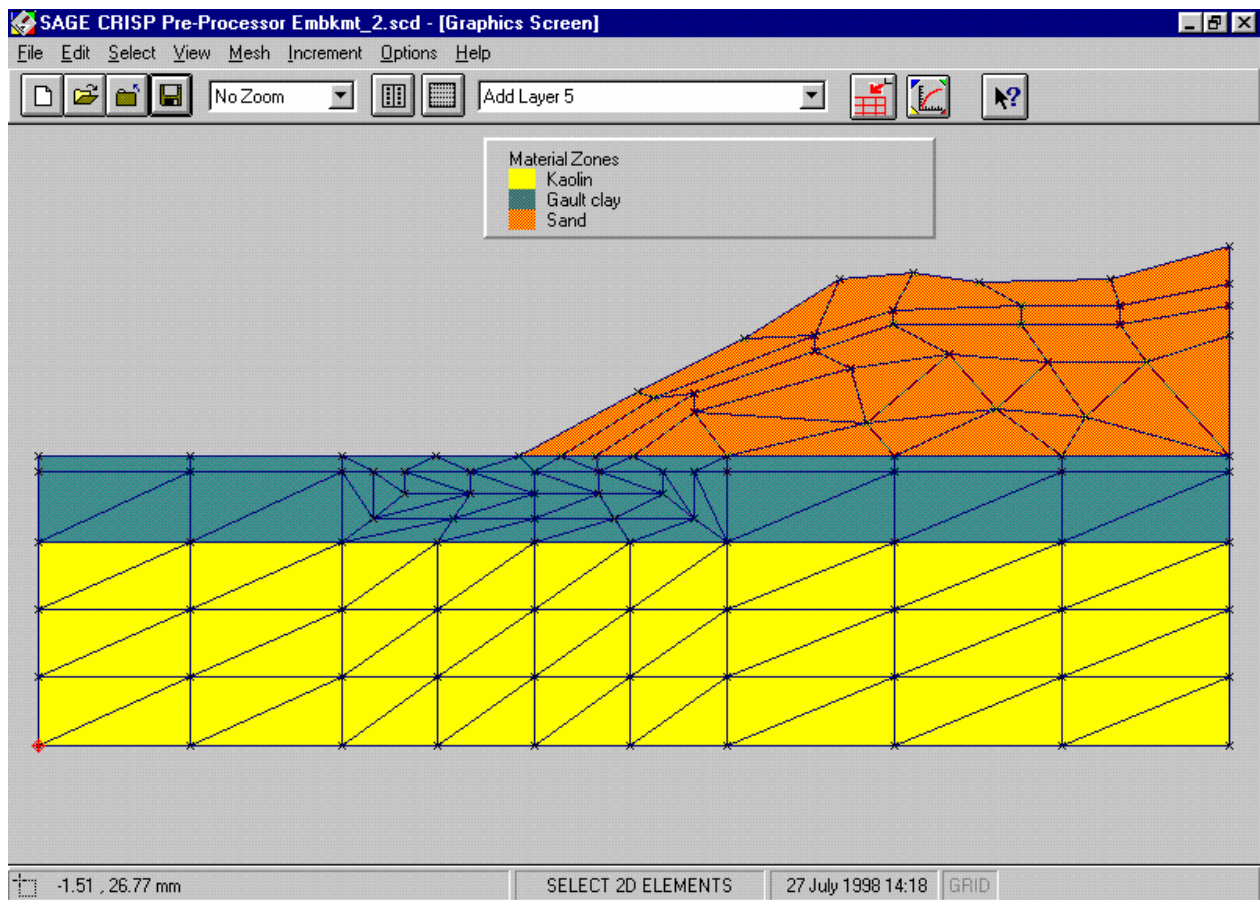
Figure 6-27:Fully Assigned Material Zones

Step 16　　　In the View menu, click on Element Types. The Graphics Window should look as shown below, provided that all of the elements have been created in identical fashion to those in this example.
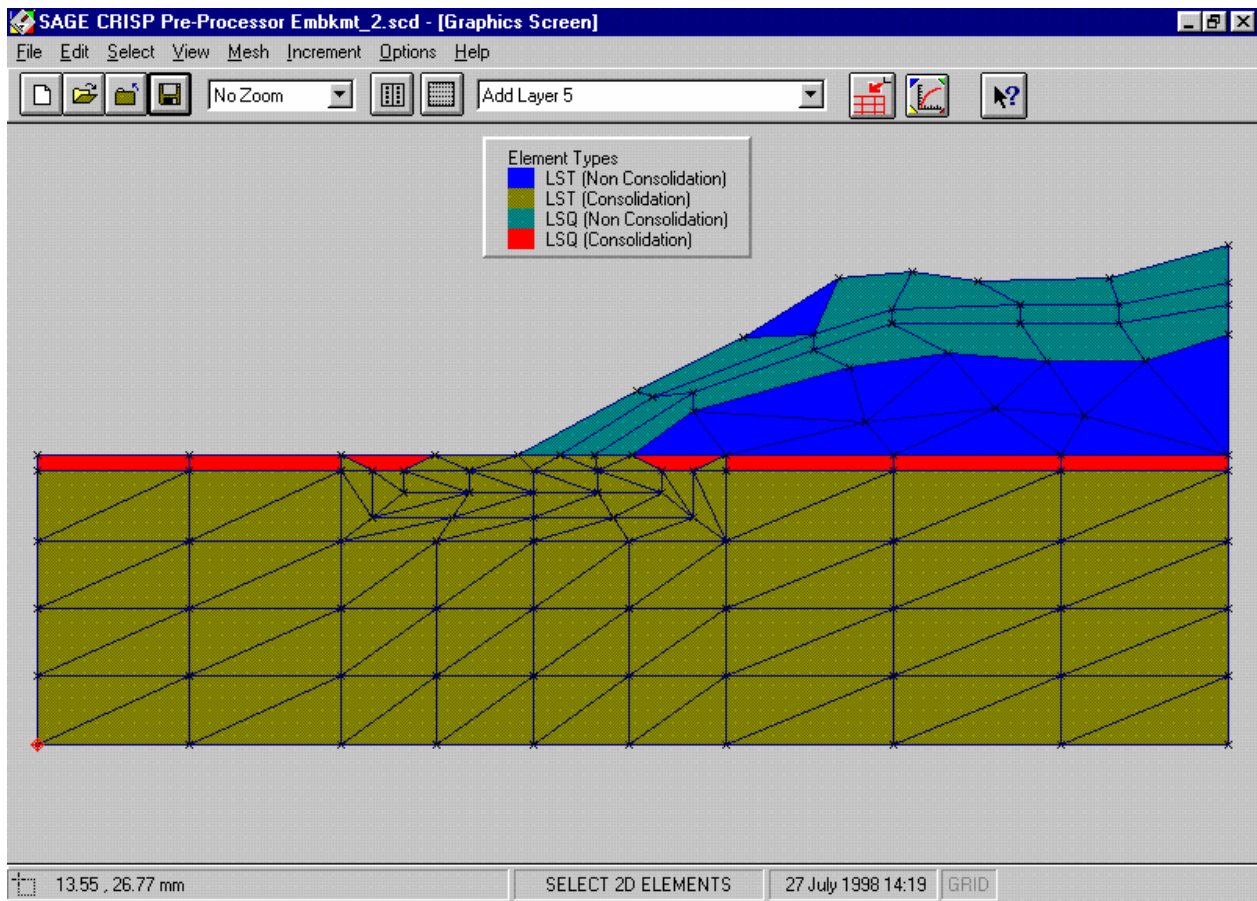
Figure 6-28:Fully Assigned Finite Element Types

6.3.13 Defining Boundary Conditions

Boundary conditions describe to the finite element analysis, the ways in which the mesh is physically restrained from moving. This is achieved by applying fixities to edges and nodes in the mesh.

Once defined, fixities apply in all successive increment blocks. The Fixities dialogue box and the Graphics Window (if View Fixities is toggled on) will only display any fixity that is applied during the current increment block.

Fixities are applied in the following stages:

■　　　In situ fixities

■　　　Pore pressure fixities (the water table)

■　　　Construction sequence fixities

Defining In Situ Fixities

In this example the left and right-hand edges are restrained horizontally, and the base is restrained both horizontally and vertically during in situ stage. The following procedure defines fixities for the in situ stage:

Step 1   Select 'In situ' from the Increment Block list box on the main toolbar.

Step 2   In the Select menu, click on Edges.

Step 3   Select all the far-left-hand and far-right-hand (vertical) edges of the mesh by clicking on them.

Step 4   In the Increment menu, click on Fixities... to display the Edge Fixity dialogue box.
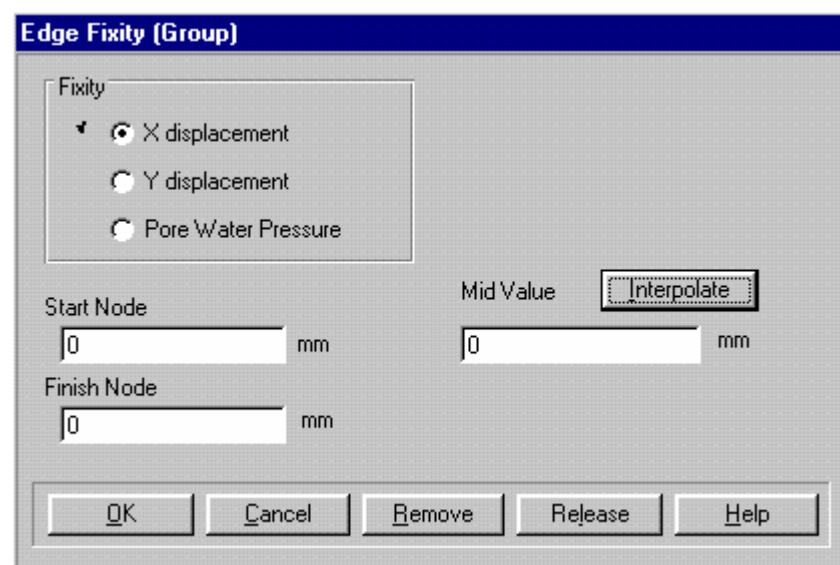
Figure 6-29:Defining Edge Fixities

Step 5   Click on the X Displacement option.

Step 6   Specify '0' X direction fixities in the Start Node and Finish Node text boxes. Set the Mid Value to zero either by entering it directly, or by clicking on the Interpolate button.

Step 7   The Edge Fixity dialogue box should now look like Figure 6-29. Click on OK to exit, once you have finished.

Step 8   In the Select menu, click on Clear Selection to ensure that no edges remain selected. Select the bottom (horizontal) edges of the mesh by clicking on them.

Step 9   In the Increment menu, click on Fixities... and define zero X displacement fixities as before.

Step 10   Click on the Y Displacement option.

Step 11   Specify Y direction fixities of zero in the Start Node and Finish Node and Mid Value text boxes.

Step 12   Click on OK to exit, once you have finished.

*Step 13*   The Graphics Window should now look as shown below. The size of the fixity icons can be changed in the Defaults dialogue box (accessed via the Options menu).
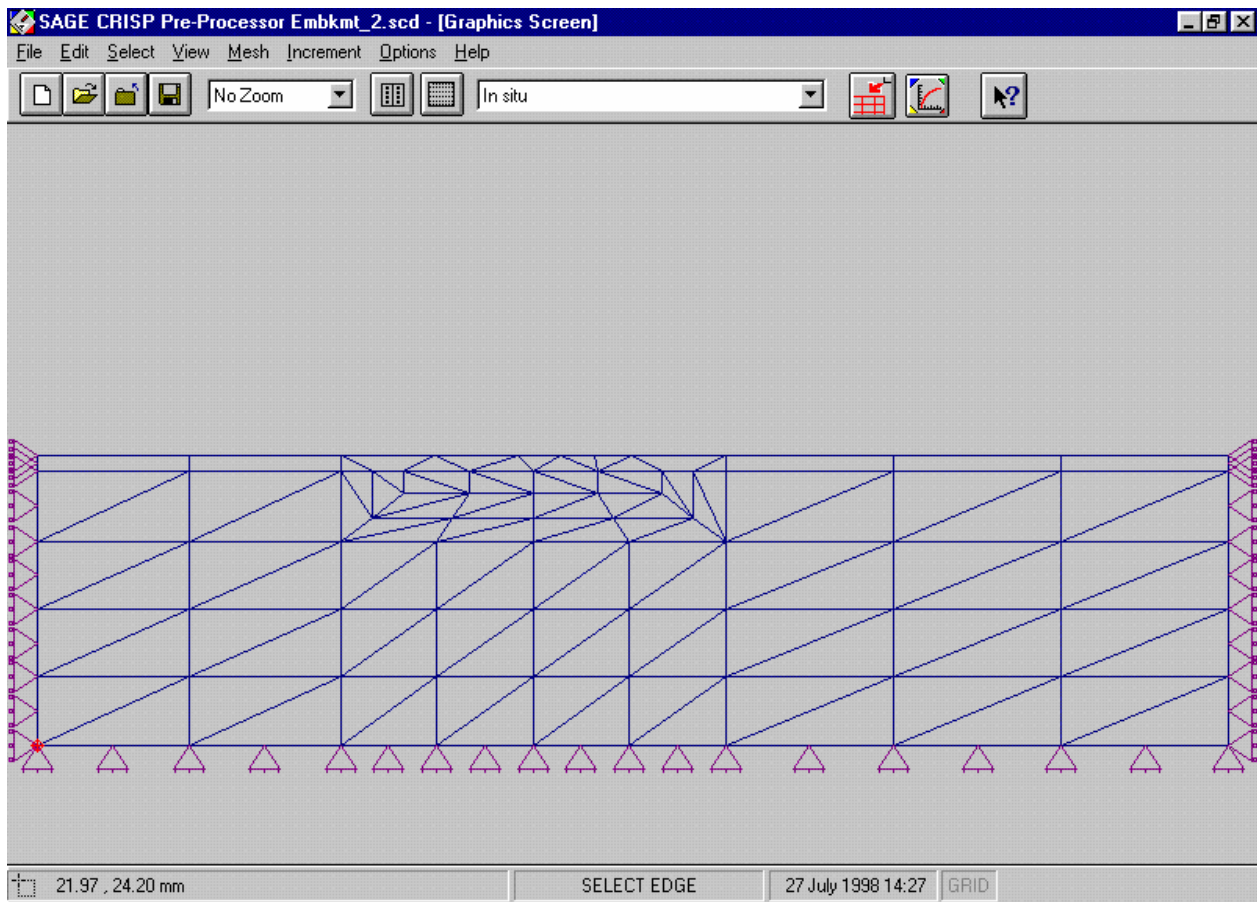
Figure 6-30:Fully Defined In Situ Fixities

Defining The Water Table

A water table is added in the first increment block by setting the total pore water pressure along material zone boundaries. The following procedure defines pore water pressure fixities:

Step 1   Select "Define water table" from the Increment Block list box on the main toolbar.

Step 2   In the Select menu, click on Edges.

Step 3   Select all horizontal edges along the bottom of the mesh by clicking on them (you will not have to do this unless you have cleared the selection).

Step 4   In the Increment menu, click on Fixities... to display the Edge Fixity dialogue box.
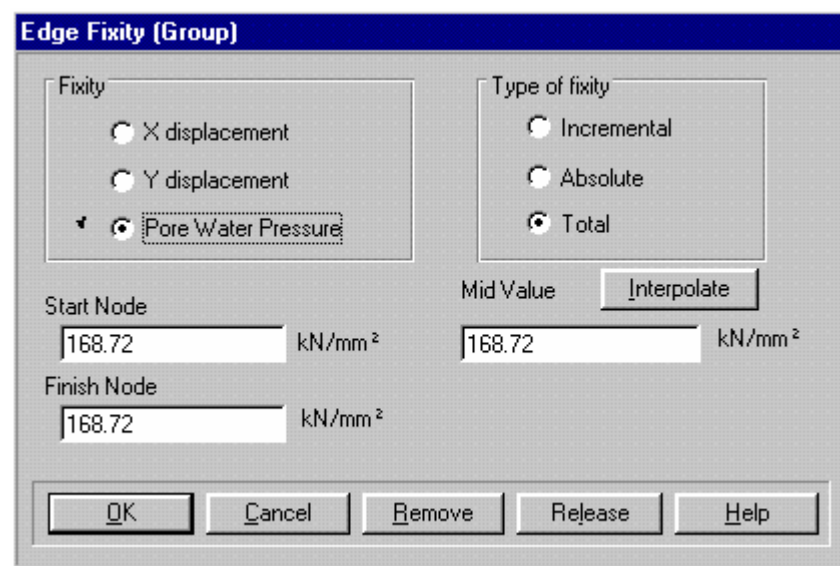
Figure 6-31:Setting Pore Water Pressures

Step 5   Click on the Pore Water Pressure option.

Step 6   Click on the Total option.

Step 7   Specify total pore water pressures of '168.72' in the Start Node and Finish Node text boxes. Set the Mid Value by clicking on the Interpolate button.

Step 8   The Edge Fixity dialogue box should now look like Figure 6-31. Click on OK to exit, once you have finished.

Step 9   In the Select menu click on Clear Selection, then select all horizontal edges with a Y co-ordinate of 15.5 metres.

Step 10   In the Increment menu, click on Fixities... and define total pore water pressure fixities as before, but with a value of '8.82'.

Step 11   Click on OK to exit, once you have finished.

Step 12   In the Select menu click on Clear Selection, then select all of the top horizontal edges (i.e. with a Y co-ordinate of 16.4 metres).

Step 13   In the Increment menu, click on Fixities... and define zero total pore water pressure fixities.

Step 14   Click on OK to exit, once you have finished.

Step 15   The Graphics Window should now look as shown below
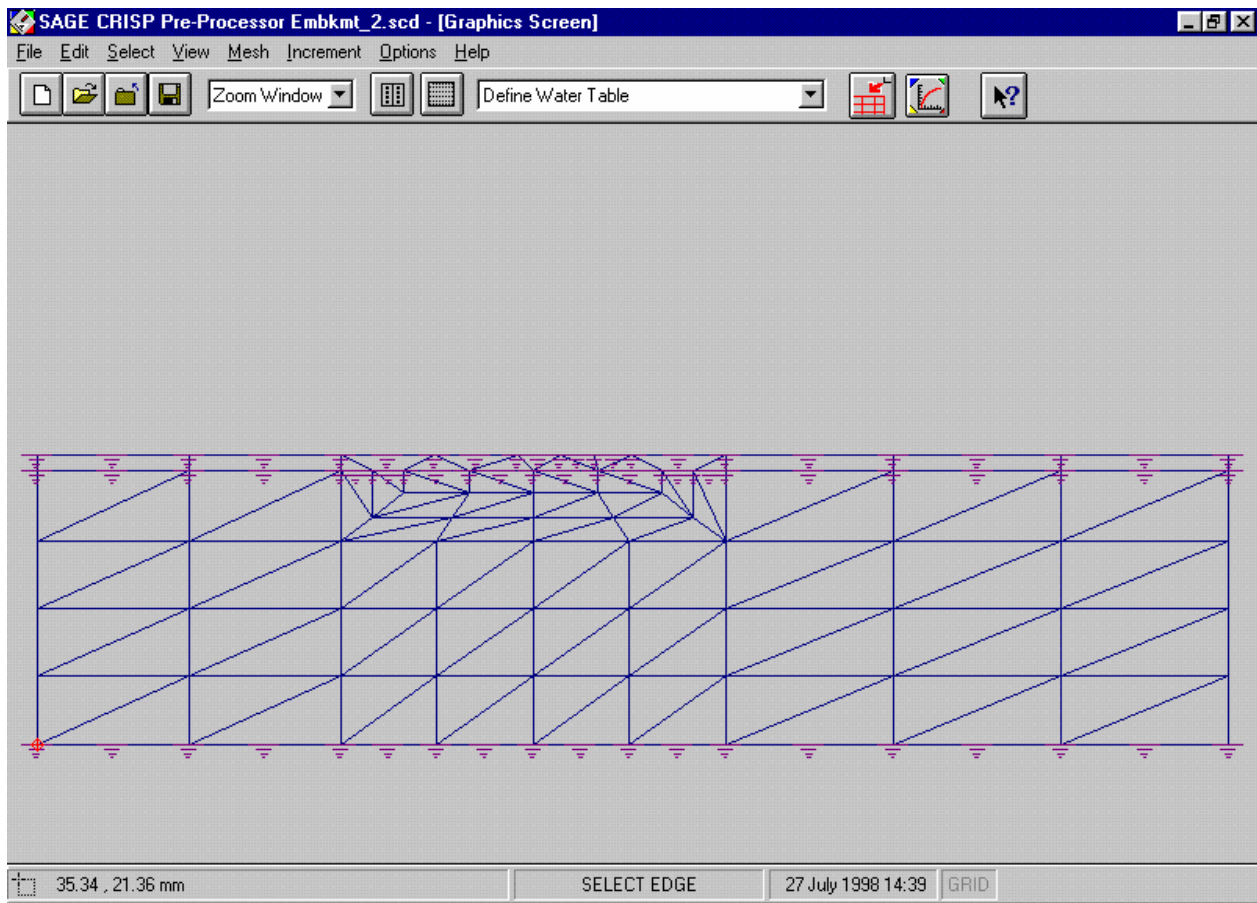
Figure 6-32:Fully Defined Pore Pressure Fixities

Defining Construction Sequence Fixities

The right-hand vertical edges of the embankment are fixed horizontally as the layers are constructed. The following procedure defines fixities during construction:

Step 1   Select 'Add layer 2' from the Increment Block list box on the toolbar.

Step 2   In the Select menu, click on Clear Selection, then select the right-hand vertical edge of the layer just added to the embankment.

Step 3   In the Increment menu, click on Fixities... and define zero X displacement fixities. This procedure is the same as already described in Defining In Situ Fixities.

Step 4   The Graphics Window should now look as shown below.
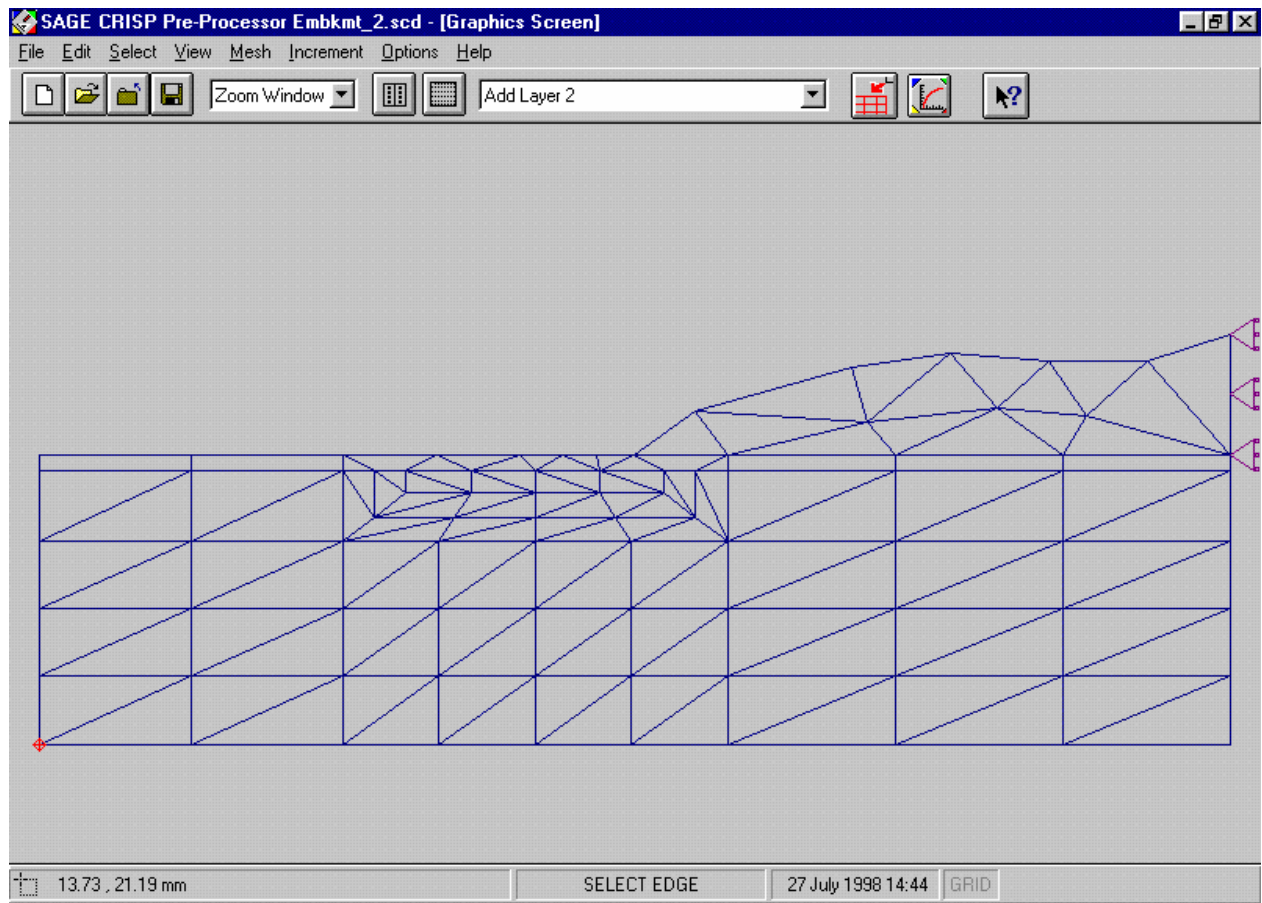
Figure 6-33:Defining Horizontal Fixities for the Second Layer

> *Step 5*   In this way, define horizontal fixities for the right-hand vertical edge of each of
> layers 3, 4 and 5 as they are added.

6.3.14 Running The Analysis

The Project should now be saved.   Provided you have followed all of the steps described above,
preparation of the Project using the Pre-Processor is complete and the analysis is ready to be run, as
described below.

> *Step 1*   In the File menu, click on Run Analysis... to display the Run Analysis
> dialogue box.
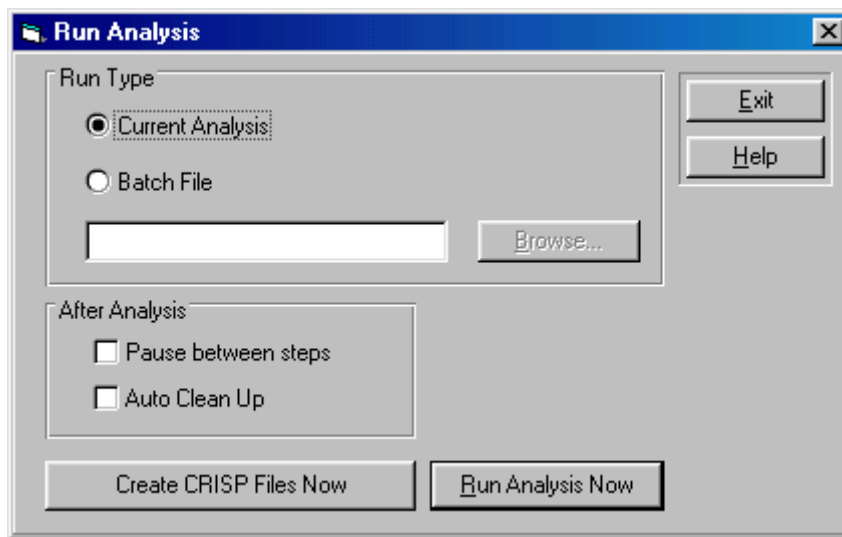
Figure 6-34:Run Analysis dialogue box

Step 2   Click on the Create CRISP Files Now button. Provided you have followed all of the above steps correctly, this will create .GPR and .MPD files for your Project. A message box will appear after each stage has been completed.

Step 3   Click on the Run Analysis button. This will prompt you to save your Project and will then run the finite element analysis.

Step 4   You will be task switched into a DOS box where the analysis will commence. Equilibrium error messages are issued for each increment of the analysis.

Warning   If the error ever exceeds about 0.02% then this is an indication that something is wrong with the input data. You should quit the analysis immediately (by pressing CTRL-BREAK) and double check all data in the Project in the Pre-Processor.

         If there is a problem, first check the in situ stress conditions, then check the material properties. Check also that fixities are correctly defined and that the Project Setup is correct.

Step 5   Once the analysis is complete, the output is converted into a Microsoft Access 2.0 database. Once this is complete, you will see the Analysis Docker dialogue box:
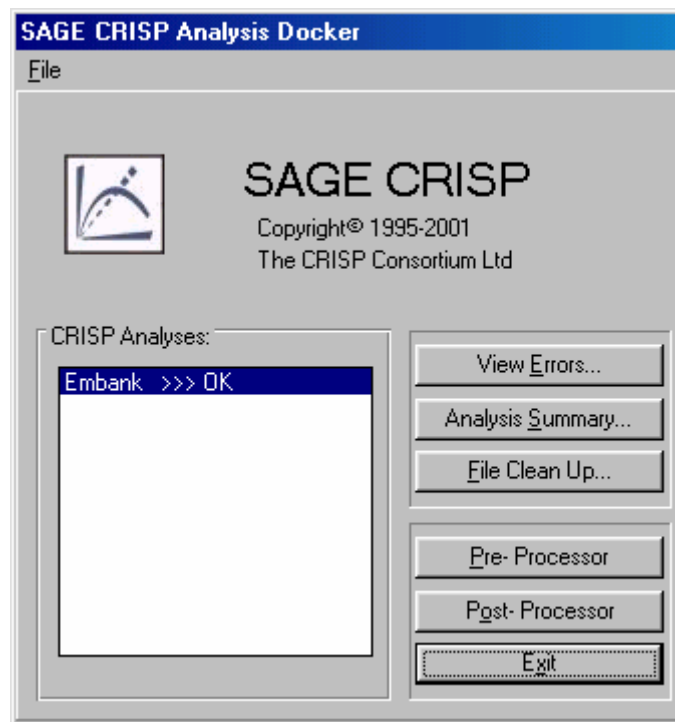
Figure 6-35:The Analysis Docking Module

> ***Step 6***   If the Project ran successfully (without errors- the message to the right of the
> Project name should say 'OK'), click on the Post-Processor button to load
> the Project into the CRISP2D Post-Processor.

6.4   Post-Processing

In this section you will be introduced to the following features:

- ■   Displacement Plots

- ■   Instance Graphs

- ■   Duration Graphs

- ■   Stress State Code Plots

6.4.1   Introduction

If you entered the Post-Processor directly from running the analysis, the embankment Project will already
be loaded. However you may also enter the Post-Processor by double-clicking on its icon in the
CRISP2D program group, in Windows' Program Manager. In this case, you should select Open
Project... from the File menu and, provided the analysis has been successfully run, you will be able to
post process the results from your analysis.

6.4.2   Displacement Plots

Displacement plots are drawn directly onto the Graphics Windows. They provide a representation of how the mesh has deformed at any stage in the analysis. You can view either vector displacements or the deformed mesh at any specified magnification.

The displacement plot described below, illustrates the final shape of the mesh at the end of the last increment of the analysis By comparison with the experimental work, this point represents collapse of the centrifuge prototype embankment.

Step 1　　　　Select Displacement Plots... from the Plots menu. This displays the Displacement Plot dialogue box.
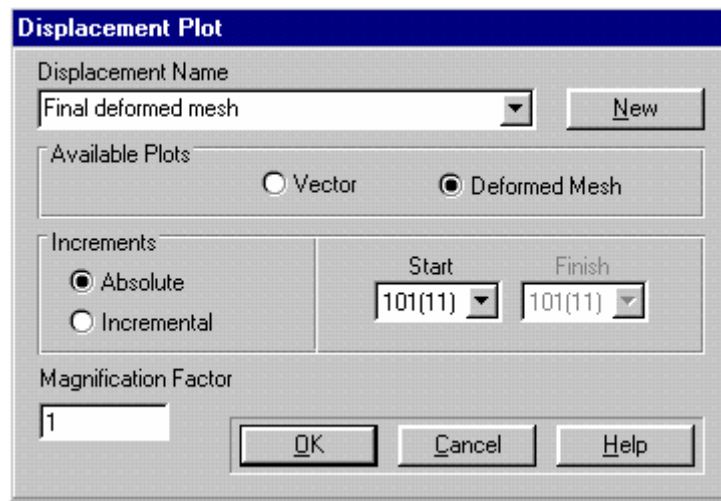
Figure 6-36:Setting Up A Displaced Mesh Plot

Step 2　　　　Click on the New button to create your plot and change the title to "Final deformed mesh".

Step 3　　　　Select the Deformed Mesh option.

Step 4　　　　Ensure that Absolute is selected and enter a Start increment of 101.

Step 5　　　　Set a Magnification Factor of 1 (the default). The Displacement Plot dialogue box should now look similar to Figure 6-36.

Step 6　　　　Click on OK to exit, once you have finished. Figure 6-37 shows the displaced mesh plot in Graphics Window (a zoom window has been executed to show more clearly the region beneath the embankment).
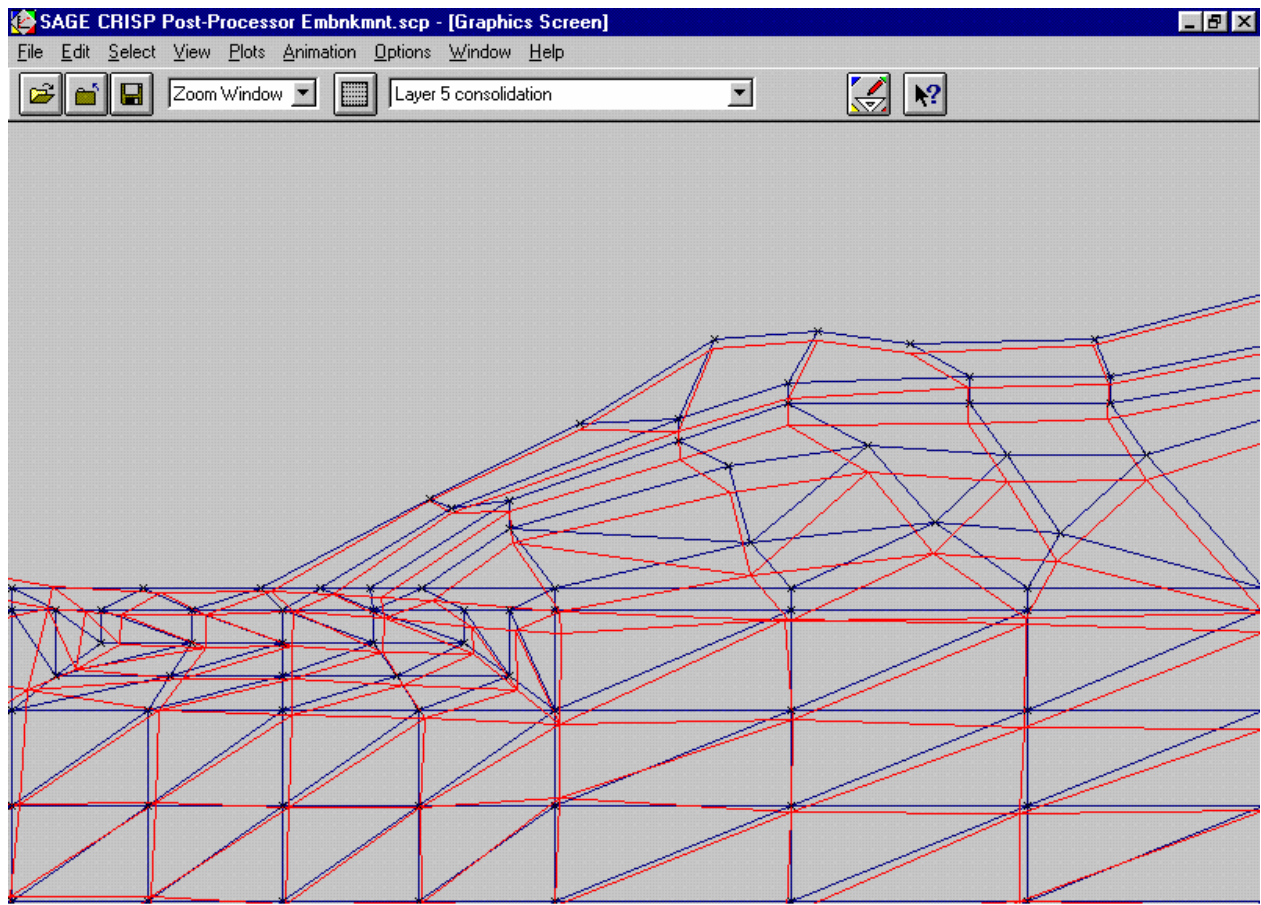
Figure 6-37:Deformed Mesh at Collapse

6.4.3   Instance Graphs

Instance Graphs allow you to investigate output for selected integration points or nodes at particular increments in the analysis.

The graph described below compares the deformation occurring between the start of the analysis and the end of the third layer's consolidation with deformations occurring between this point and the end of the fifth layer's consolidation.

> *Step 1*   In the Plots menu, click on Clear Current Plot. This makes it easier to see which nodes you are selecting.

> *Step 2*   In the Select menu, click on Nodes.

> *Step 3*   Select all of the nodes whose Y co-ordinate is 15.5. These points, just beneath the top of the clay layer, are the source from which the graph will be drawn.

> *Step 4*   In the Plots menu, click on Instance Graphs... to display the Instance Graphs dialogue box.
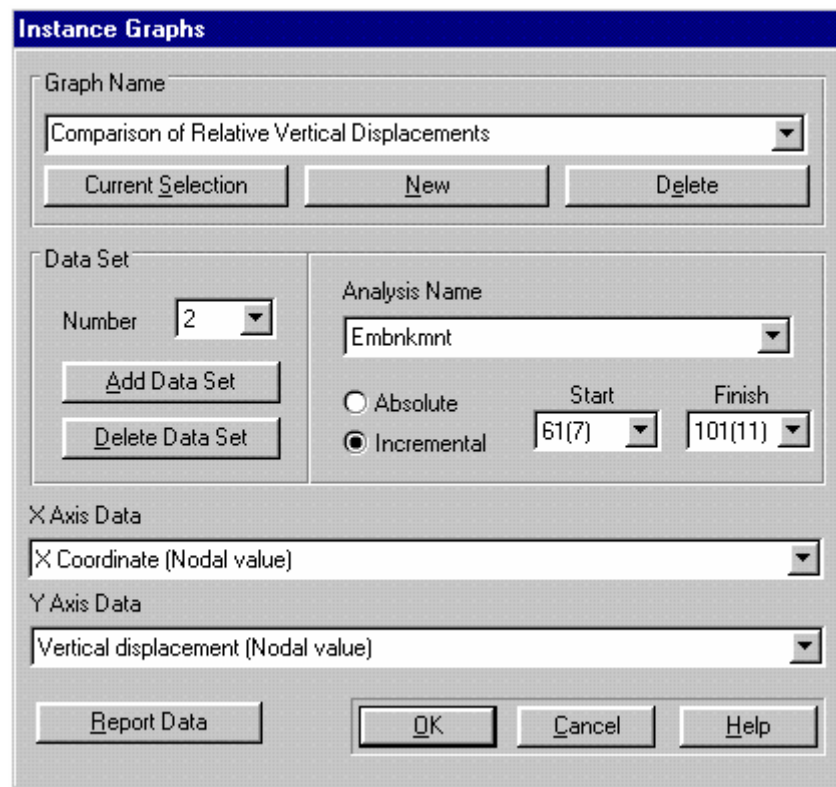
Figure 6-38: Setting Up An Instance Graph

Step 5   Click on the New button to create a new graph whose data is taken from the selected nodes. Change the title to "Comparison of Relative Vertical Displacements".

Step 6   Click on Incremental and enter Start and Finish values of 1 and 61.

Step 7   Choose 'X Co-ordinate' as X Axis Data and 'Vertical displacement' as Y Axis Data.

Step 8   Click on the Add Data Set button and select Data Set 2 from the Data Set Number list.

Step 9   Click on Incremental and enter Start and Finish values of 61 and 101. The Instance Graph Dialogue box should look like Figure 6-38.

Step 10   Click on OK to exit, once you have finished. You should see a graph similar to that displayed in Figure 6-39.

Step 11   The appearance of the graph can be altered in the in the Graph Configuration dialogue box. This is accessed via the Options menu, or by double-clicking on the graph itself. The options used in Figure 6-39 were 7 X axis divisions with minima and maxima of 0 and 70 respectively, 10 Y axis divisions with minima and maxima of -1.0 and 0. The legend, axis labels, graph title and grid lines can also be changed in this dialogue box.
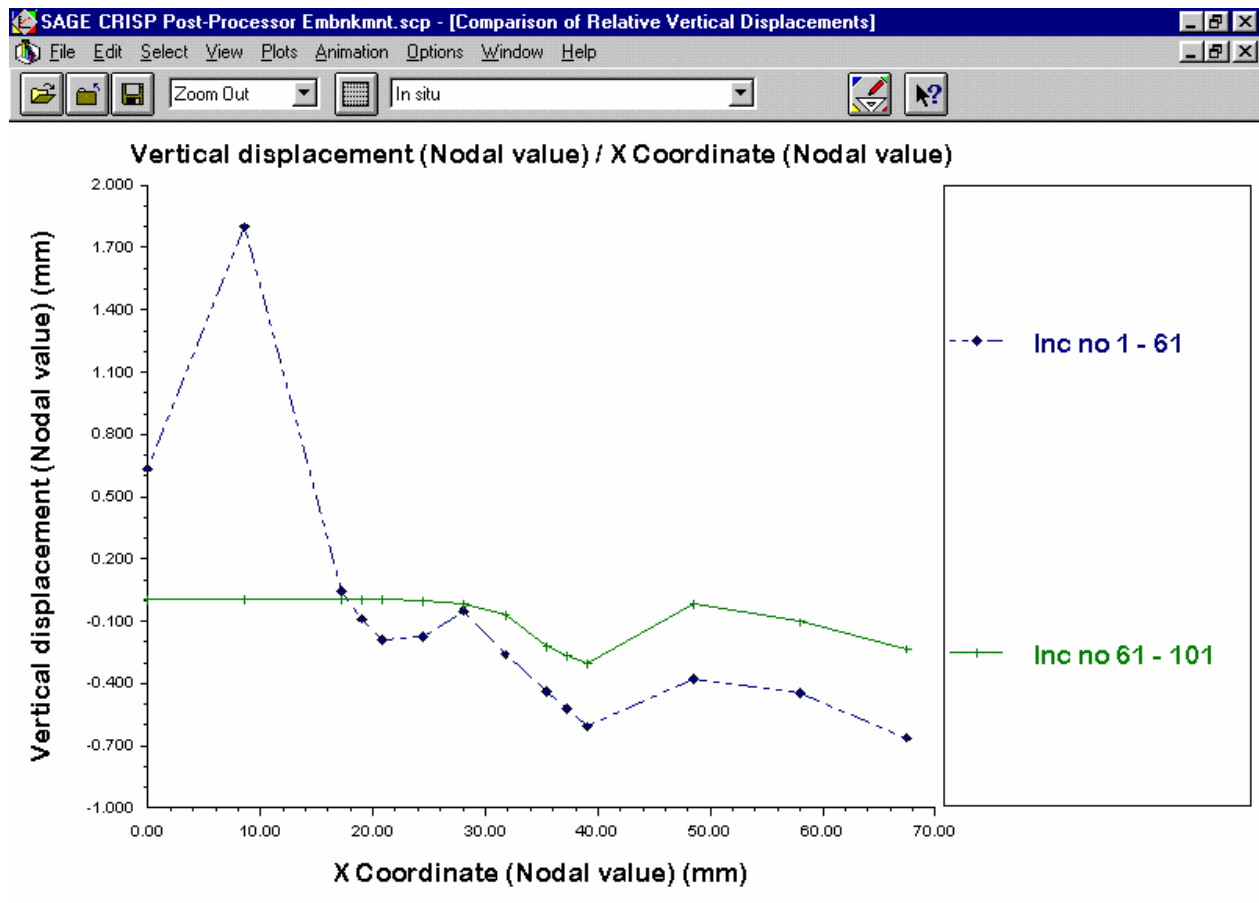
Figure 6-39:Comparison Of Relative Vertical Displacements

Step 12   The resulting graph indicates that the vertical displacements prior to the fourth layer being applied amount approximately 80% of the total displacements.

6.4.4   Duration Graphs

Duration graphs allow you to investigate the variation of output from individual integration points or nodes over the course of a specified range of increments.

Up to 10 data sets (each one representing a different integration point or node) can be viewed on the same graph. For this example, two integration points beneath the footing will be used.

Step 1   To reactivate the Graphics Window, click on Cascade, Tile Vertically or on Tile Horizontally in the Window menu. These options allow you to switch between the Graphics Window and any graphs that are on display.

Step 2   In the Select menu, click on Integration Points and then on Clear Selection. Select an integration point at the centroid of one of the finite elements in the Kaolin clay layer beneath the middle of the embankment. (This graph was drawn from centroid of the fifth triangle up, in the second column in from the right of the mesh).

Step 3   In the Plots menu, click on Duration Graphs... to display the Duration Graphs dialogue box.

Figure 6-40:Setting Up A Duration Graph

Step 4   Click on the New button to create a new Duration graph and change the title to "Stress Path Beneath Embankment".

Step 5   Ensure that the Increment Range starts at 1 and finishes at 101.

Step 6   Check each of the Critical State Lines check boxes.

Step 7   Click on the top left Stress Path Graph button (i.e. the q - p' button). The Duration Graph dialogue box should now look like Figure 6-40.

Step 8   Click on OK to exit, once you have finished. The graph displayed should look similar to Figure 6-41.

Step 9   You can configure the way the graph looks (axis labels, titles and legend) by double-clicking on the graph window and changing the set-up in the Graph Configuration dialogue box. The options used in this case are 9 X axis divisions with minima and maxima of -200 and 1600 respectively, 8 Y axis divisions with minima and maxima of 0 and 800. The line styles, symbol styles and legend have all been modified.
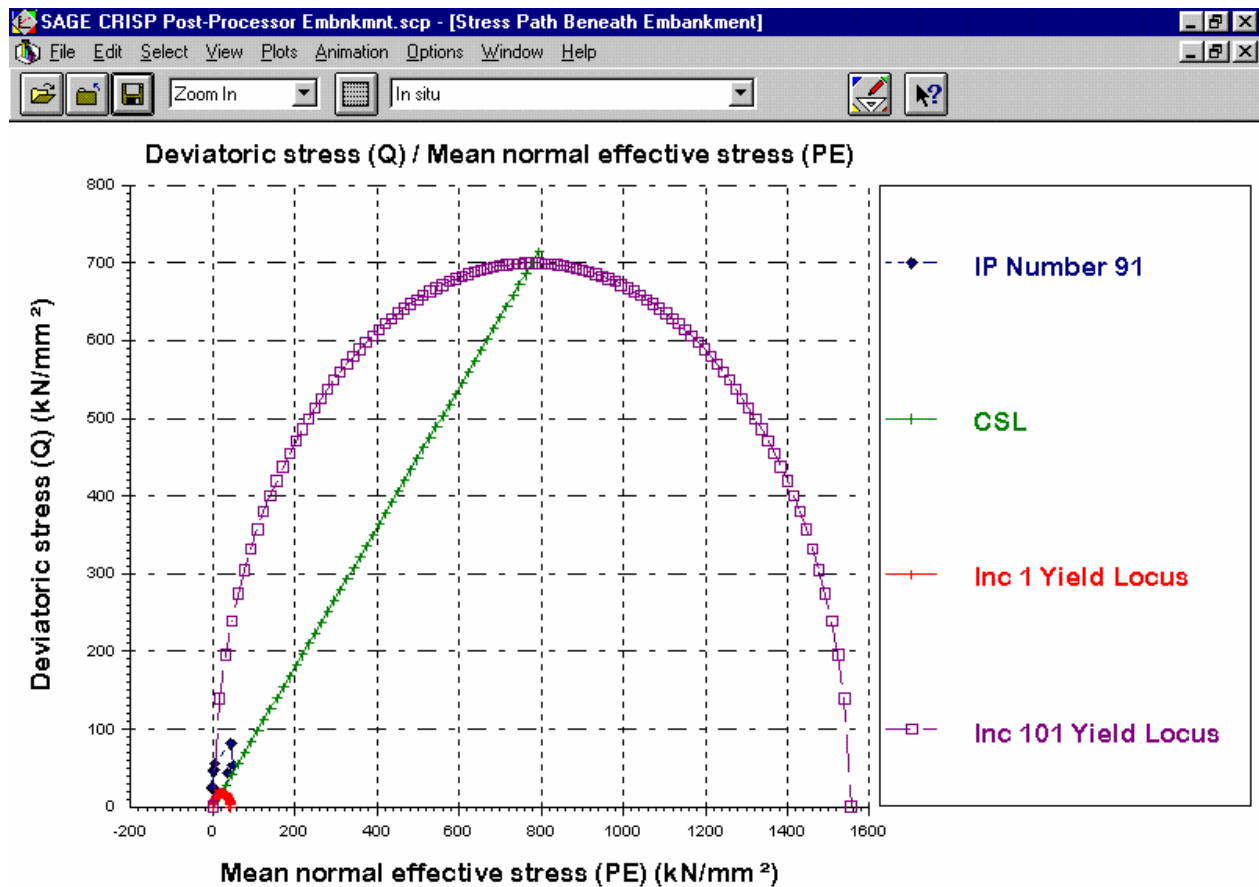
Figure 6-41:Stress Path Plot for a Point Beneath the Embankment

6.4.5 Status Plots

The Status Plot option covers a variety of different kinds of plot that allow you to represent, by use of vectors and coloured blocks, the patterns of variation of particular quantities across the entire mesh. This is useful when you want to look at the formation of failure mechanisms or, if you are assessing the validity of your analysis. It is this latter application to which the status plot is being applied in this example.

Here, a stress state code plot is being used to determine whether or not the stresses in the soil are plausible.

Step 1   To reactivate the Graphics Window, click on Cascade, Tile Vertically or on Tile Horizontally in the Window menu. Click on the Graphics Window to activate it and maximise it by clicking on the Maximise arrow in the top right hand corner of the window.

Step 2   In the Plots menu, click on Status Plots... to display the Status Plot dialogue box.
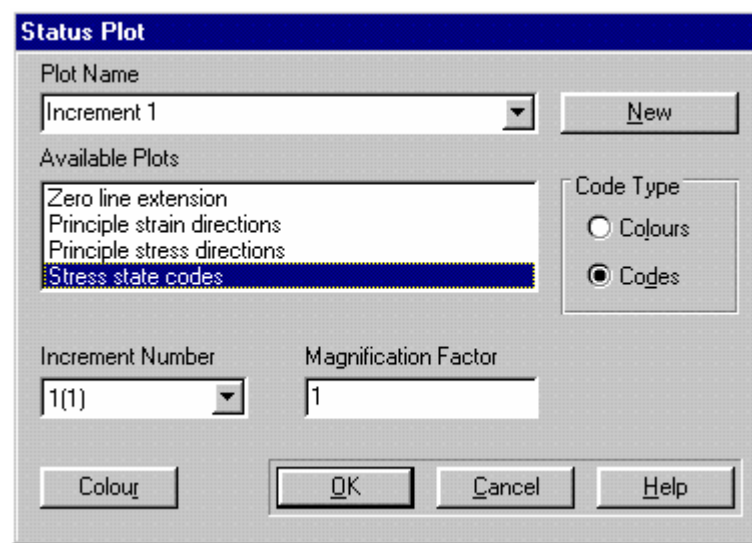
Figure 6-42:Setting Up A Stress State Code Status Plot

Step 3   Click on the New button to create a new Status plot and change the title to "Increment 1".

Step 4   Select 'Stress state codes' from the Available Plots list.

Step 5   Enter '1' into the Increment Number text box.

Step 6   Magnification Factor has no effect on a stress state code plot, so this can be ignored.

Step 7   The Status Plot dialogue box should now look like Figure 6-42.

Step 8   Click on OK to exit.

Step 9   In the View menu, click on Legend. The Graphics Window should now look similar to Figure 6-43. If the colour scheme is not clear, you can change it by going back into the Status Plot dialogue box, selecting the plot from the Plot Name list and clicking on the Colour button. This displays the Stress State Colours dialogue box which allows you to change the colour used to represent each stress state.
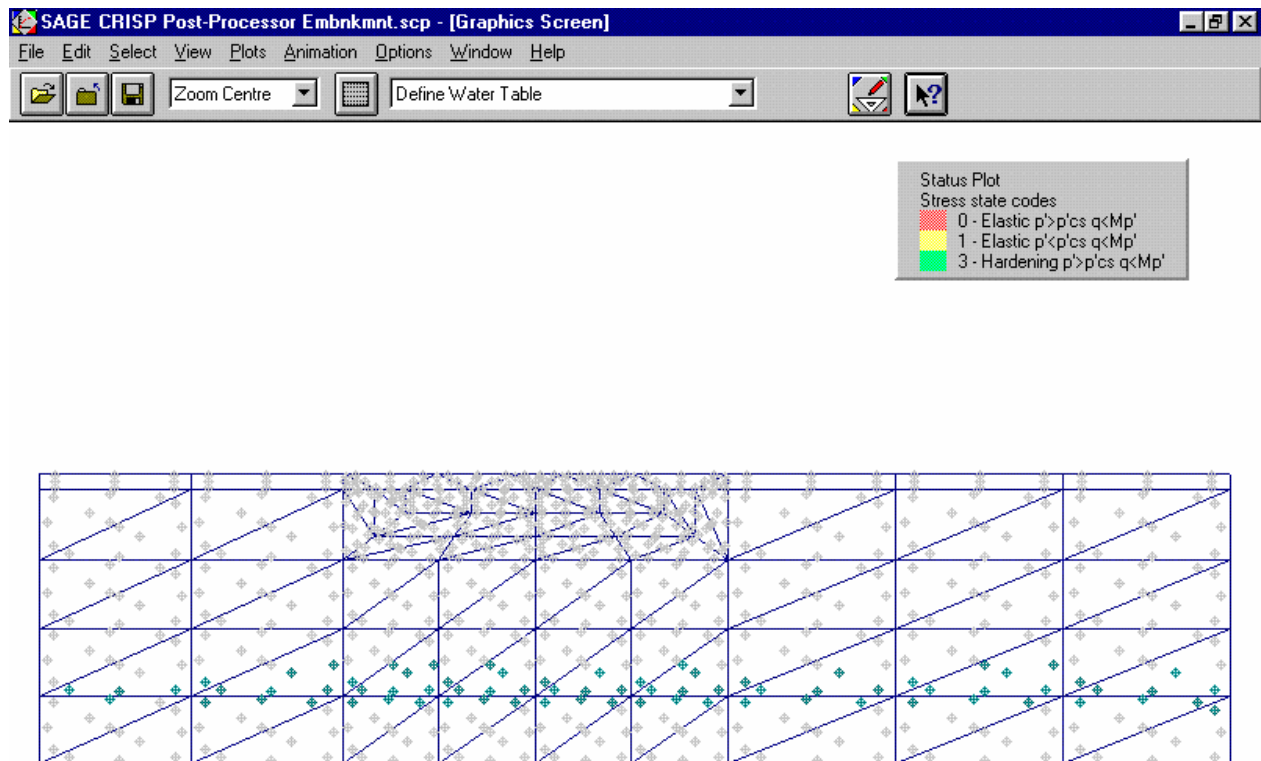
Figure 6-43:Stress State Code Plot for Increment 1

Step 10   By reference to the Cam clay yield locus described in section 4.5 of the Technical Reference Guide, this plot clearly shows the difference in strength between the two clay layers, with the top (Gault clay) layer being the stiffer of the two. This plot is useful as it correctly predicts that before any loading occurs, the soil is elastic.

Step 11   Any anomalies at this stage are caused by numerical conditions- in Figure 6-43, a row of integration points undergoing strain hardening can be seen close to the bottom of the mesh. If this is a cause for concern, it can be easily remedied by using a slightly finer mesh and re-running the analysis.

Step 12   In the Plots menu, click on Status Plots to again display the Status Plot dialogue box.

Step 13   Click on New to create a new Status plot and change the title to "Increment 21".

Step 14   Select 'Stress state codes' from the Available Plots list.

Step 15   Enter '21' into the Increment Number text box.

Step 16   Click on OK to exit.

Step 17   The Graphics Window should now look similar to Figure 6-44.
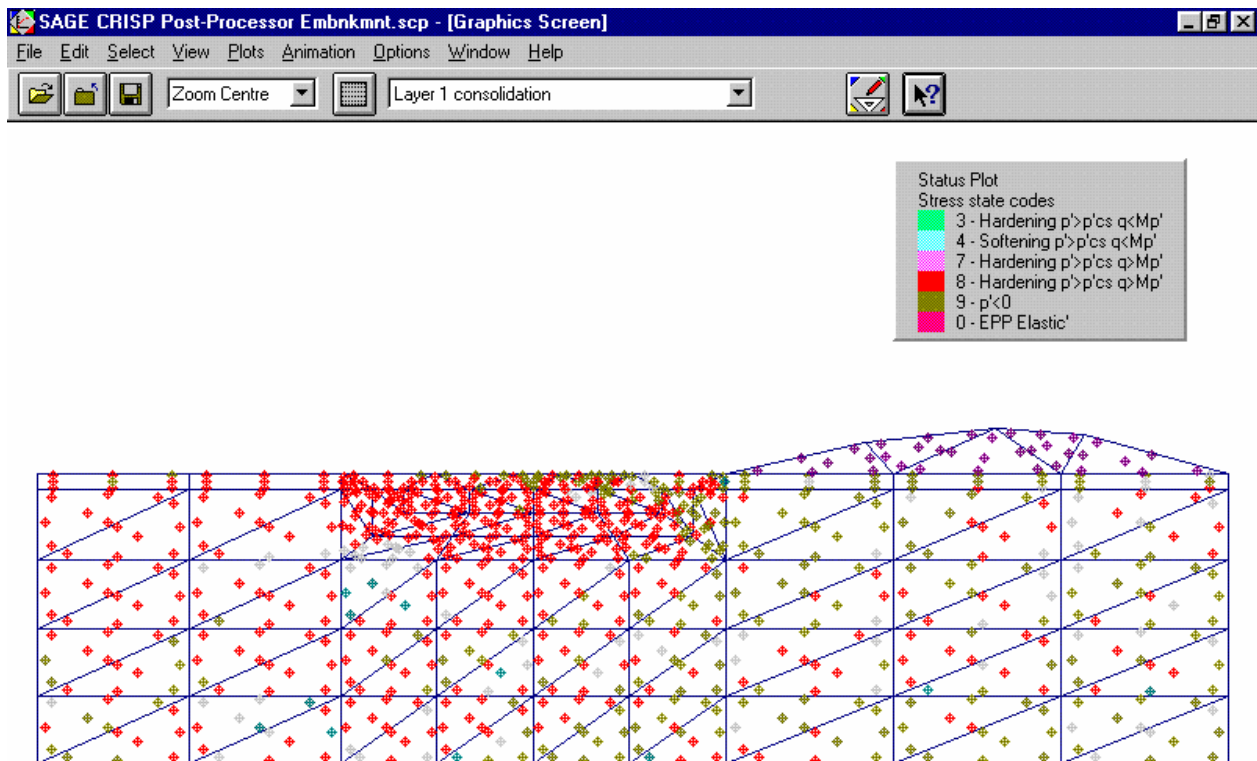
Figure 6-44:Stress State Code Plot for Increment 21

Step 18   This plot shows the softer Kaolin clay beneath the embankment undergoing significant strain hardening. Away from the embankment, the soil remains elastic. The Gault clay beneath the embankment is variously undergoing strain hardening and strain softening, suggesting that its behaviour is not being accurately captured. Use of smaller elements in this region may be advisable.

6.5   Summary

By the end of this example you will have made use of the following features:

- ■   The Structured Mesh Generator

- ■   Creating finite elements manually

- ■   Cam clay

- ■   Consolidation analysis

- ■   Construction sequences

- ■   Displacement plots

- ■   Instance graphs

- ■   Duration graphs

- ■   Status plots

6.6   Things To Try Next

- ■   Create status plots for increments 41, 61, 71, 81, 91 and 101. These plots suggest that the region of greatest activity is in the Kaolin clay beneath the embankment.

- ■   Return to the Pre-Processor and save the Project under a different name. Try modifying the super mesh so that the elements beneath the toe of the embankment are slightly larger and the elements directly beneath the embankment are significantly smaller. Run the analysis again and see how the status plots change. The ideal mesh should exhibit no anomalous behaviour.

TUNNEL EXAMPLE

7.1 Introduction

The following steps will be covered:

- Adding and removing elements

- Using layers

7.2 The Problem
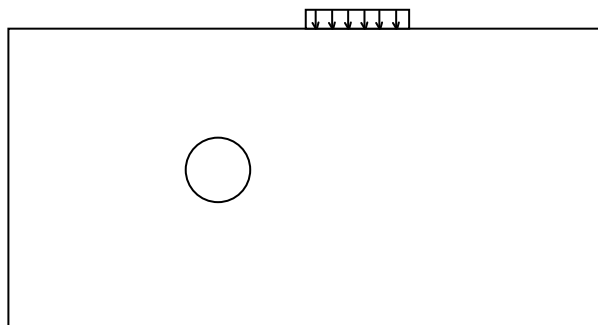
A 6 meter diameter tunnel is to be constructed at a depth of 25 meter below ground level. A pressure load is to be applied representing a new building.

7.3 Setting up the mesh

Step 1   Start CRISP2D and choose Structured Mesh from the project setup menu. Click on All other Element types as we do not need the high order CuST element here.

Step 2   Click on Super Node List and enter the following nodes in the list

Super Node List

Super Node List Units in m

Node	X Co-ord	Y Co-ord
1	0.000	0.000
2	60.000	0.000
3	60.000	50.000
4	0.000	50.000
5	20.000	30.000
6	30.000	30.000
7	30.000	20.000
8	20.000	20.000
9		

OK   Cancel   Preview   Help

Step 3   Next, make sure that the Create Elements in the Mesh menu is ticked, and create the following four elements (leave an empty box in the middle as shown in the figure below):
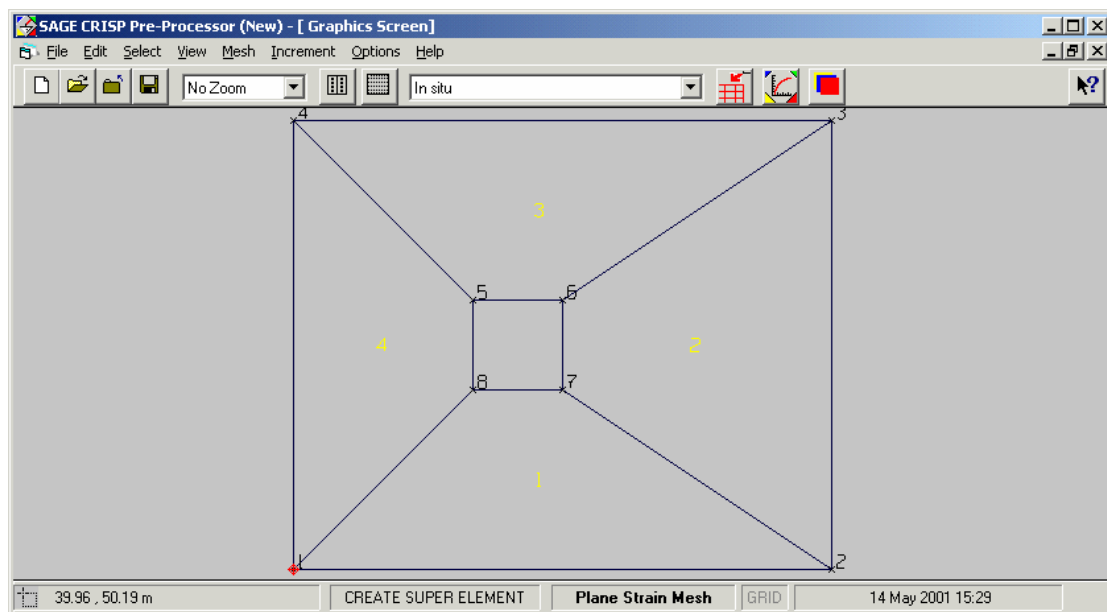
Step 4   Click on Select>Super Edges and click on each outer edge of the super mesh. Enter 4 divisions and use the Equal option.

Step 5   Select the diagonal super edges (click nearer the middle box on each diagonal edge), then enter 4 for the number of divisions, use a Factor of 5 and click on the option End 1 is K times End 2. You should get the super egde grading as shown below:
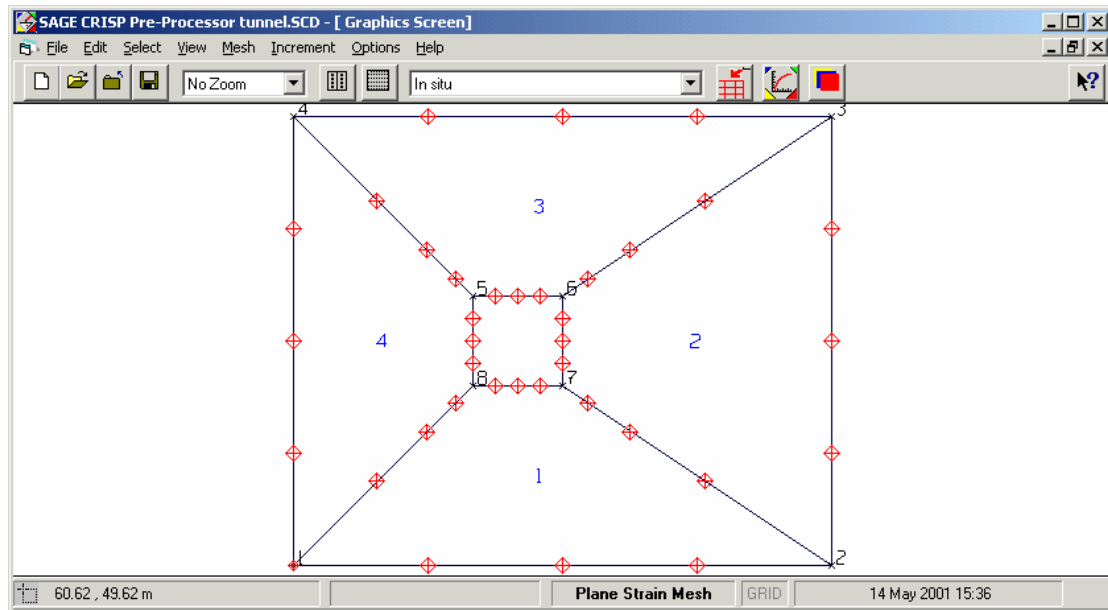
Step 6   Next, click on Mesh>Material Properties and enter the following material parameters for the soil
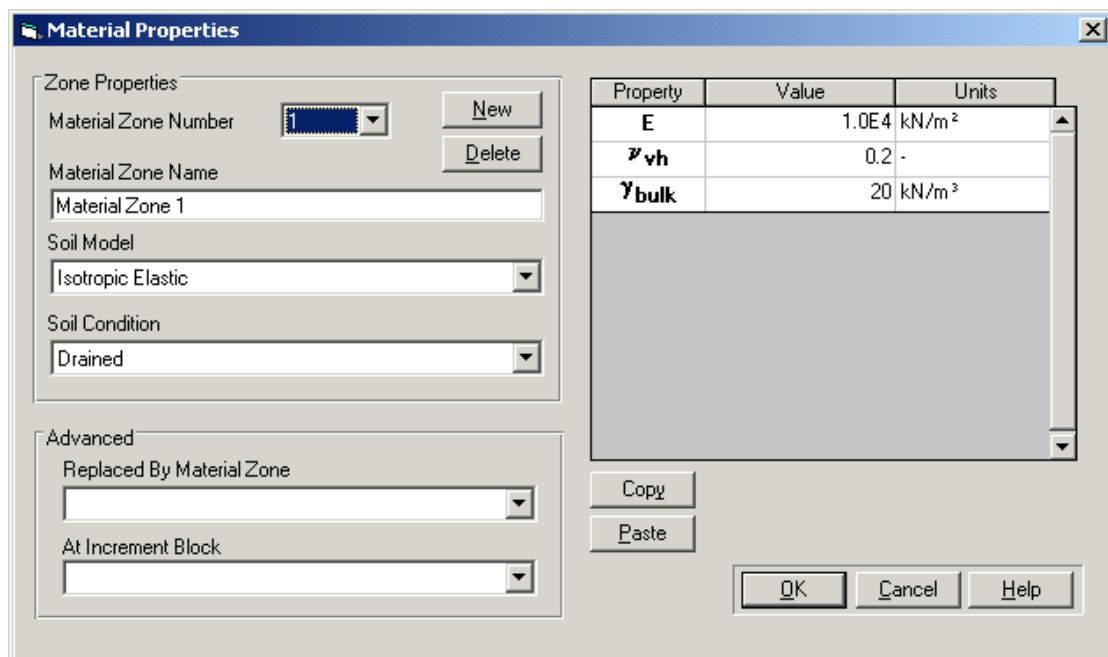
Step 7   Enter another material zone by clicking on New in the Material Properties dialogue box and enter the following material parameters for the lining:

Step 8   Click on Select>Super Elements in the Super mesh and select all the elements in the super mesh.

Step 9   Click on Mesh>Element Properties and fill in the properties for the super elements as shown below. Ensure that you use the material zone you have created for the soil

Step 10   Click on Mesh>Generate Finite Element Mesh to create the mesh. This should like as shown below.
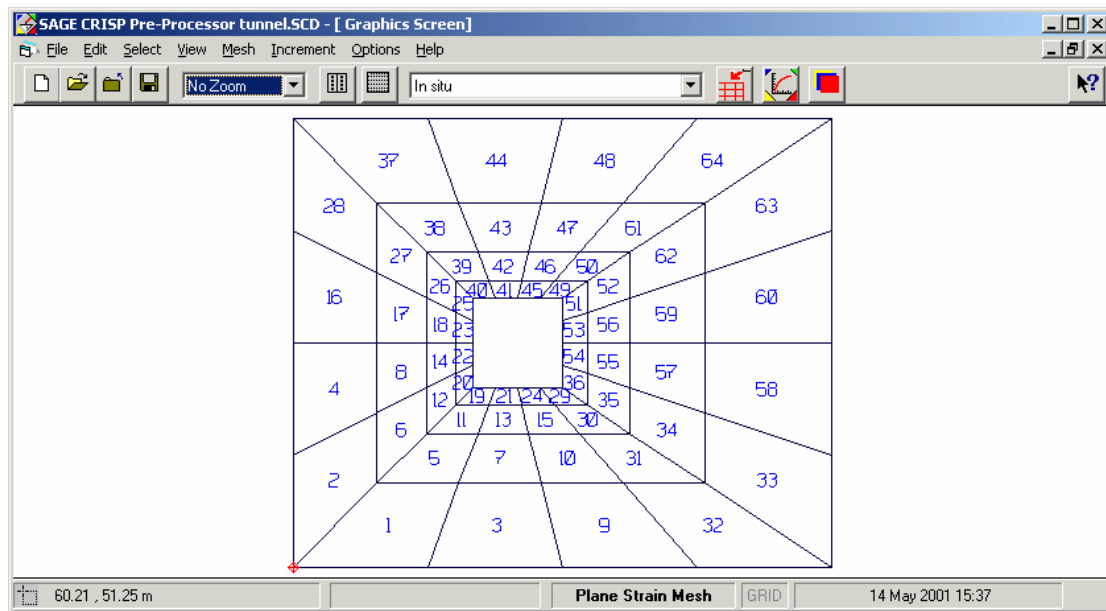
Step 11   The next step involves manually creating the elements for the tunnel and the tunnel lining. First we create tunnel nodes. Click on Mesh>Tunnel Nodes and fill in the following values

Step 12   Click on Mesh>Tunnel nodes again to create another set of tunnel nodes to represent the outer limit of the linning. Fill in the following

Tunnel Nodes Setup

Centre Point

X Co-ordinate
25

Y Co-ordinate
25

Radius
3.3

Divisions per quarter
4

OK   Cancel   Preview   Help

Step 13   Create an extra node for the centre of the tunnel by clicking on Mesh>Node List and enter node coordinates of 25,25 for an extra node at the end of the list.

Step 14   Next, click on Mesh>Create Elements and create elements to fill in the empty box where the tunnel is to be dug out as shown below. Remember that we need to create all the elements before the tunnel is dug out.
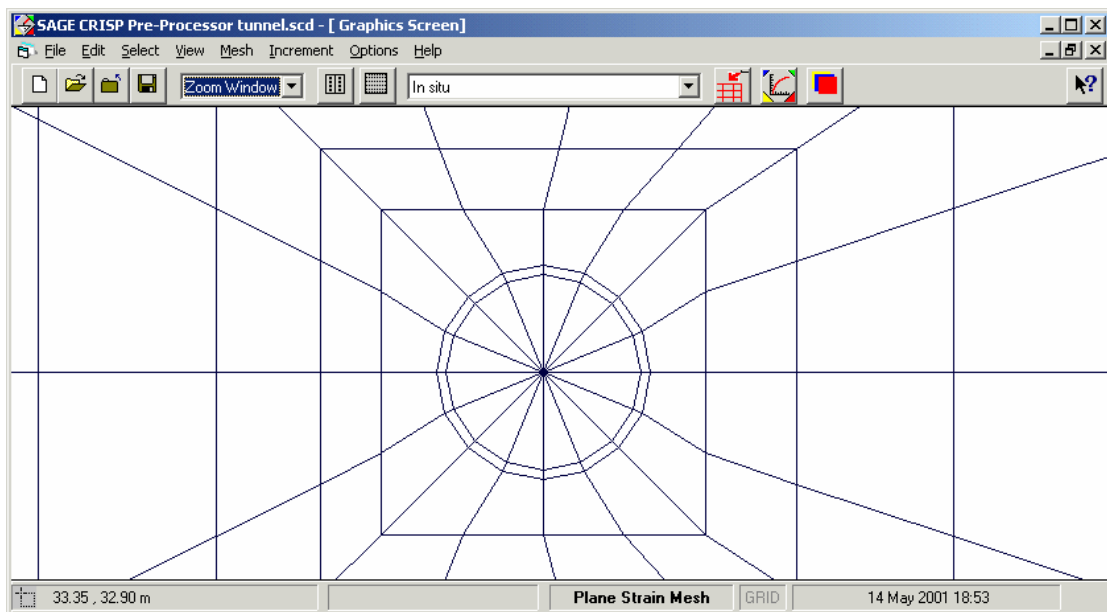
Step 15   Click on Select>Domain Elements and select triangular elements you have just created manually, then select Mesh>Element Properties and assign these elements the following properties
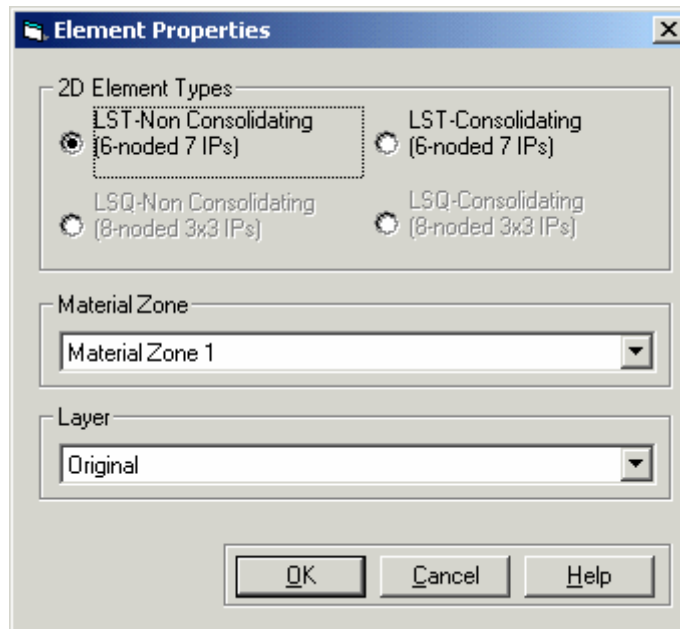
Step 16   Do the same step above for the rectangular elements representing the lining and assign them the soil material zone but using LSQ (Quadrilateral elements). Remember that at the in-situ stage the lining does not exists and that these elements represent the soil.

Step 17   Click on Increment>In-situ stresses and enter the following stresses which corresponds to an elastic material with Ko=1 and a unit weight of 20KN/m3

	Height (m)	σ'_{xx} (kN/m²)	σ'_{yy} (kN/m²)	σ'_{zz} (kN/m²)	τ_{xy} (kN/m²)	PWP (kN/m²)	$\bar{U}$ (kN/m²)	P'c (kN/m²)
1	50	0	0	0	0	0	0	0
2	0	1000	1000	1000	0	0	0	0

Step 18   Next click on the edges of the mesh and apply fixties as shown in the figure below
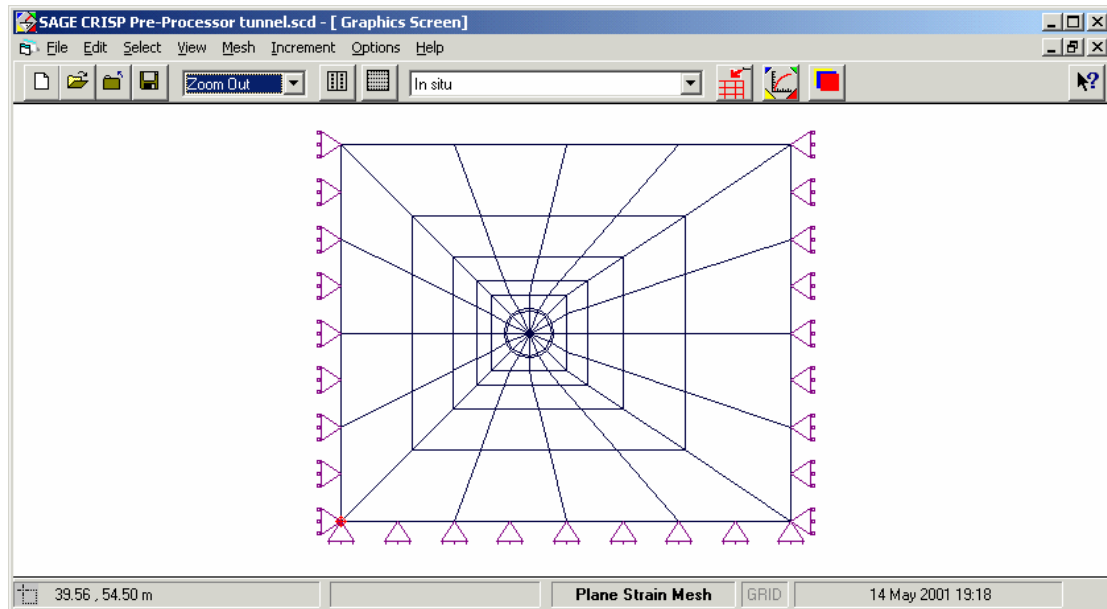
By now the in-situ mesh should be completed. We now need to introduce the elements representing the lining.

Step 19   Select the rectangular elements representing the lining and click on Mesh>Super Impose Elements and fill in the detail below.
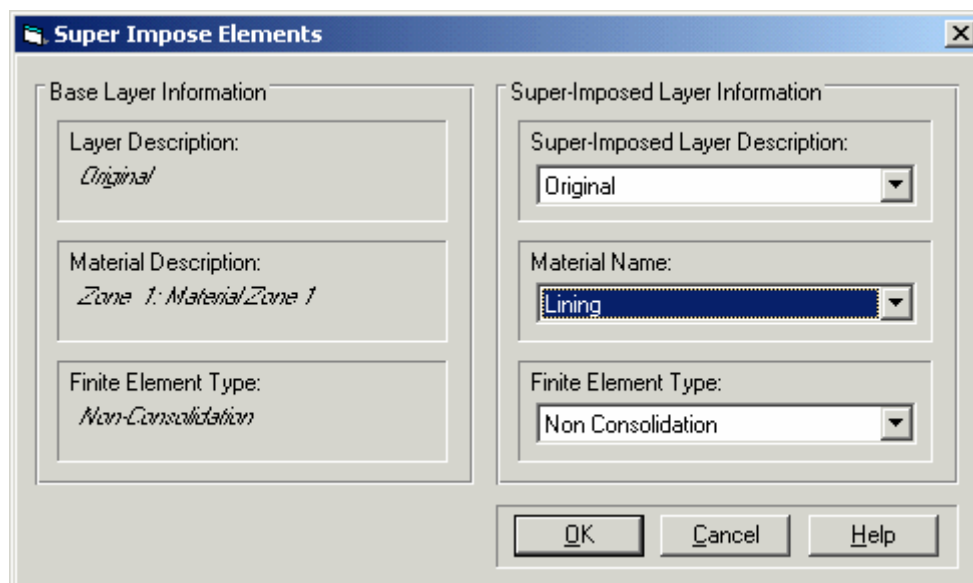
This will create a new set of elements on top of the selected elements but having the lining properties.

You should now remove these new elements from the in-situ stage. They will be introduced when the lining is intalled.

Step 20 To remove these elements, click on Select>Domain Elements and select these lining elements (not the soil elements), then click on Increment>Remove Elements.

Step 21 Next, click on Increment>Define Increment Block Parameters and enter the following three load blocks

Step 22 Click OK and in the pre-processor click on the Excavate load block on the combo box of the current increment.

Step 23 Click on Select>Domain Elements and click on all the tunnel elements including the rectangular ones around the tunnel. Click on Increment>Remove Elements. This will complete the excavation stage.

We now want to add the elements representing the lining.

Step 24 Select the Install Lining load block from the compo box of the Current Increment. The lining (concrete) elements cannot be seen as they are invisible at prersent. Click on View>Removed Elements and select the elements representing the lining (not the removed soil elements). Once you have completed the selection, click on Increment>Add Elements. Click on View>Present Elements to go back to the active (present) mesh. You should be able to see the lining elements present.

Step 25 Finally, select the last load block of Apply Pressure. Click on Select>Edges and select the edge shown below
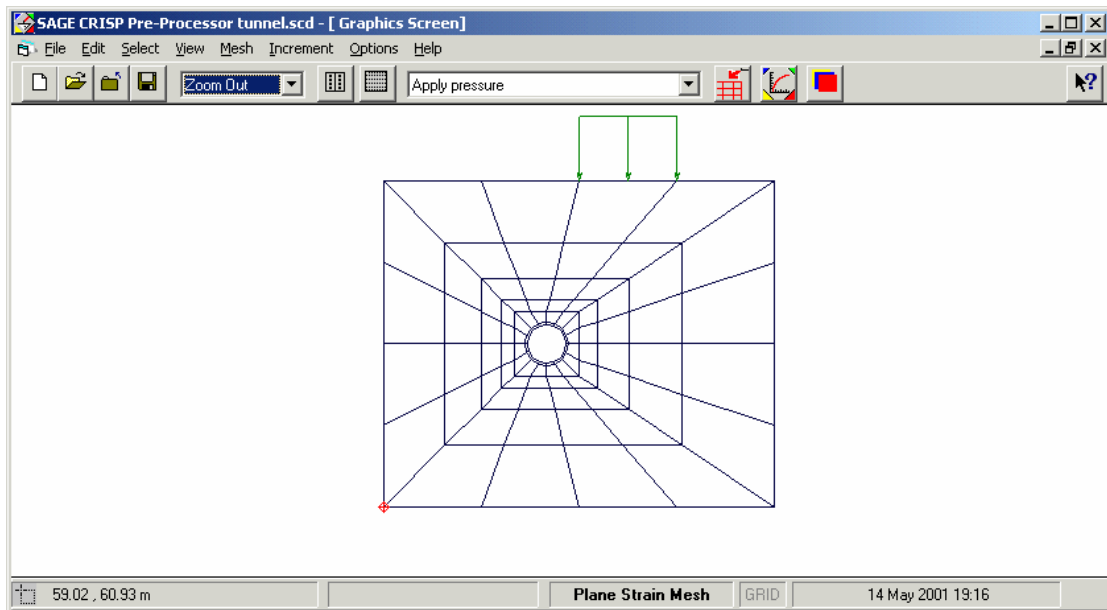
Step 26   Click on Increment>Loads and enter the following values:

RETAINING WALL EXAMPLE

8

8.1   Introduction

This example lecture is designed to introduce you to the excavation sequences and layer control in CRISP2D.

The following steps will be covered:

- ■   Adding and removing elements

- ■   Using layers

- ■   Animation Sequences

- ■   Anamation plots

8.2   The Problem

The dimensions and loading conditions of the problem are shown below.
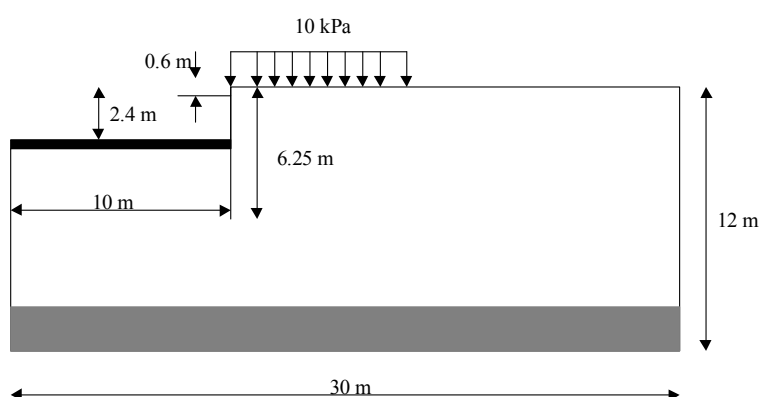
Figure 1:Problem dimensions

8.3   Generating a Mesh with the Structured Mesh Generator

Step 1   Start the CRISP2D pre-processor

Step 2   Click on File-New Project and select Structured Super Mesh

Step 3   In the project setup select All Other Elements in the Element Type option, then click OK

Step 4   Click on Mesh-Super Node List and enter the following node numbers and coordiantes

Node	X-Coord	Y-Coord
1	0	0
2	30	0
3	30	12
4	10	12
5	10	5.75
6	10	11.4
7	0	12
8	0	9.6
9	10	9.6
10	0	9
11	10	9
12	10	0
13	0	
	0	
15	30	5.75
16	30	9
	30	
18		11.4
19	30	11.4

Step 6   Once you have completed creating the super mesh, click on **Super Node Numbers**

Step 7   Click on Select Super Edges and click on the top edge between nodes 7 and 4 as shown above. Make sure you click nearer to node 4 than to node 7 so the pink square marker is on node 4

Step 8   Click on Mesh-Super Edge Grading and enter 4 for No. Divisions and 3 for Factor(k) and click on the option End 1 is k times End 2 as shown below
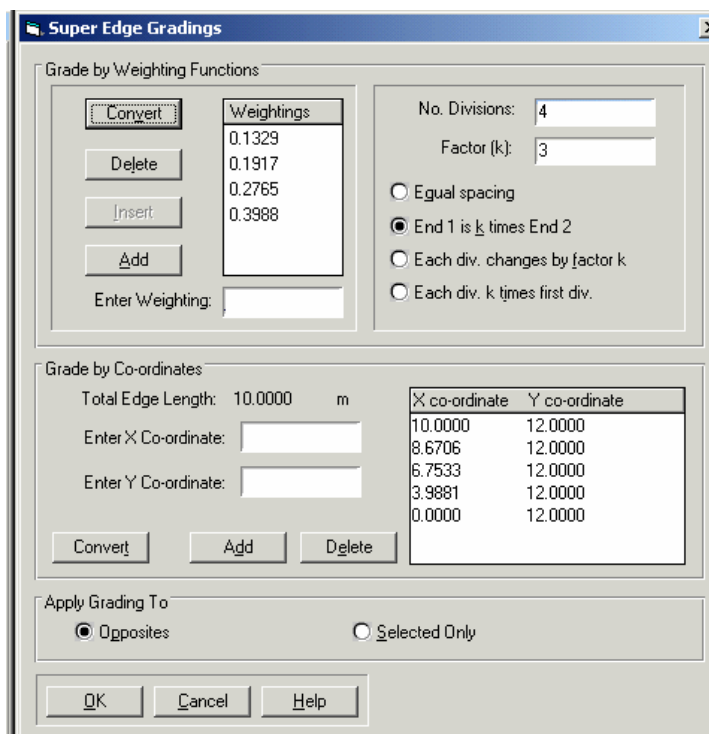
These divisions will be the same for all edges parallel to the edge you selected.

Step 9   Next clear the selected edge by clicking Select-Clear Selection, or use the short cut CTRL+L to clear the selection.

Step 10   Select the edge between nodes 4 and 3 and make sure you click nearer to node 4. Enter 5 for No. Divisions and 3 for Factor(k) and click on the option End 1 is k times End 2 as shown below.

Step 11   Clear the currently selected edge (Select-Clear Selection)

Step 12   Select the edge between nodes 7 and 18 and enter two 1 division and click on Equal Spacing

Remember you have to enter edge divisions even if you have only one division on the side of the super element.

Step 13   Next, clear the currently selected edge, and click on the edge between nodes 8 and 18.

Step 14   Enter 2 for No. Divisions and click on the option Equal Spacing.

Step 15   Clear the current edge selection and click on the edge between the nodes 14 and 1. Click nearer to the node 14 so that the pink marker is on node 14. Enter 3 for No. Divisions and 3 for Factor(k) and click on the option End 1 is k times End 2 as shown below
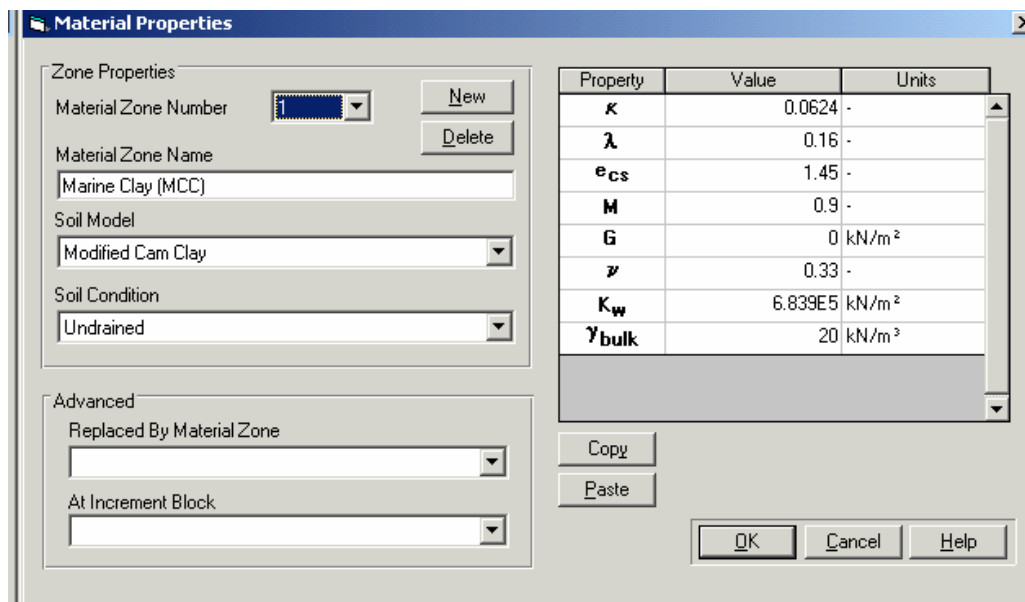
All edges should by now have been sub-divided correctly.

Step 1   Click on Mesh-Material Properties and enter the first material zone as shown below
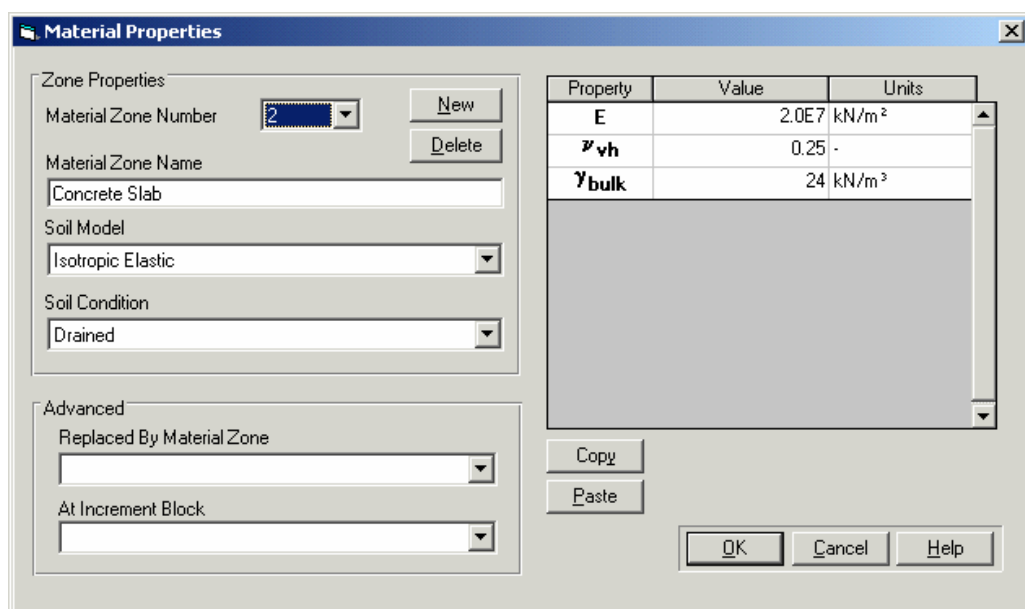
Step 2   Click on **New** in the **Material Properties** dialogue box and enter details for the **Concrete Slab** as shown below
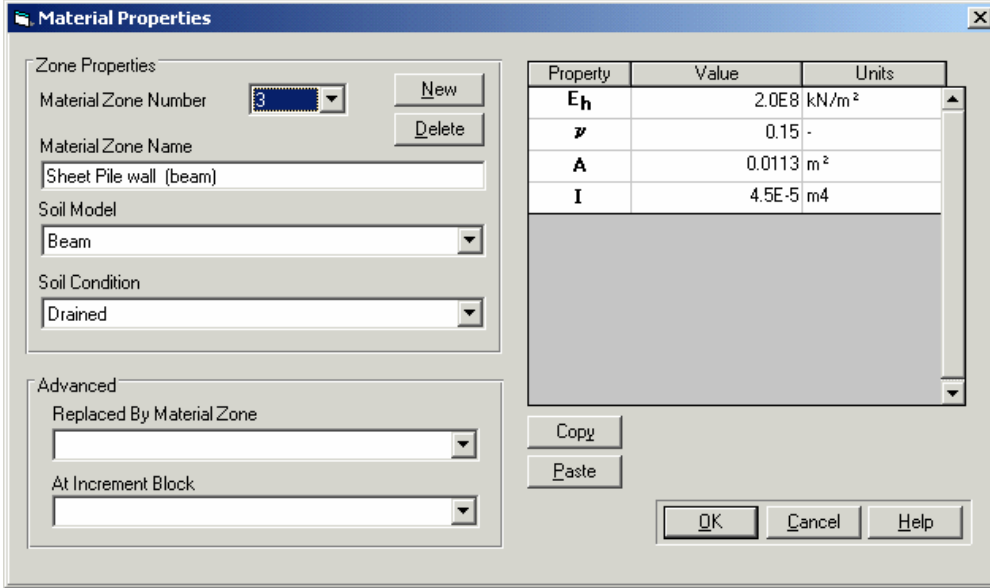
Step 3   Click on **New** in the **Material Properties** dialogue box and enter details for the **sheet pile wall**. This will make use of the **Beam** model in CRISP, which solves bending moments directly
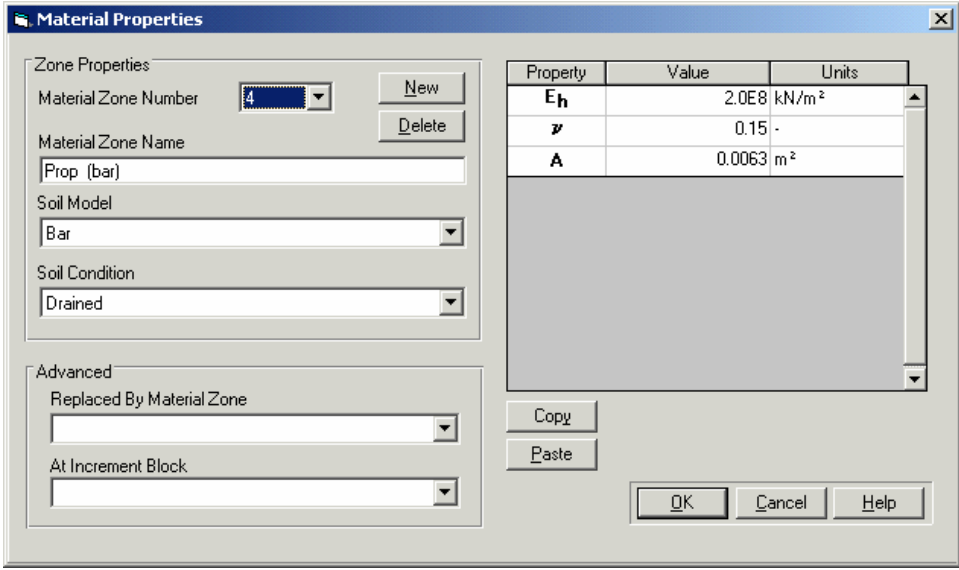
Step 4   Click on New in the Material Properties dialogue box and enter details for the horizontal prop. This also makes use of this will make use of the bar element which allows for transfer of axial forces

Click on New in the Material Properties dialogue box and enter details for the bedrock as shown.

8.4.1   Assigning Element Properties

We have so far defined material zones but we have not associated these material zones with elements. We also need to specify the type of finite element to be generated (quadrilaterals or triangles, consolidation or non-consolidation).

Step 1   Click on Select-Domain Super Elements and click on all the elements in the super mesh.

Step 2   Click on Mesh-Element Properties and choose LSQ non-consolidation. For the material Zone, select Marine Clay (MCC). For the time being we will give the same properties for all the elements but would change that later

8.4.2   Generating the Finite Element Mesh

In the mesh menu click on **Generate Finite Element Mesh**. Your mesh should like as shown below. You can view the element numbers by clicking on **View-Element Numbers**.
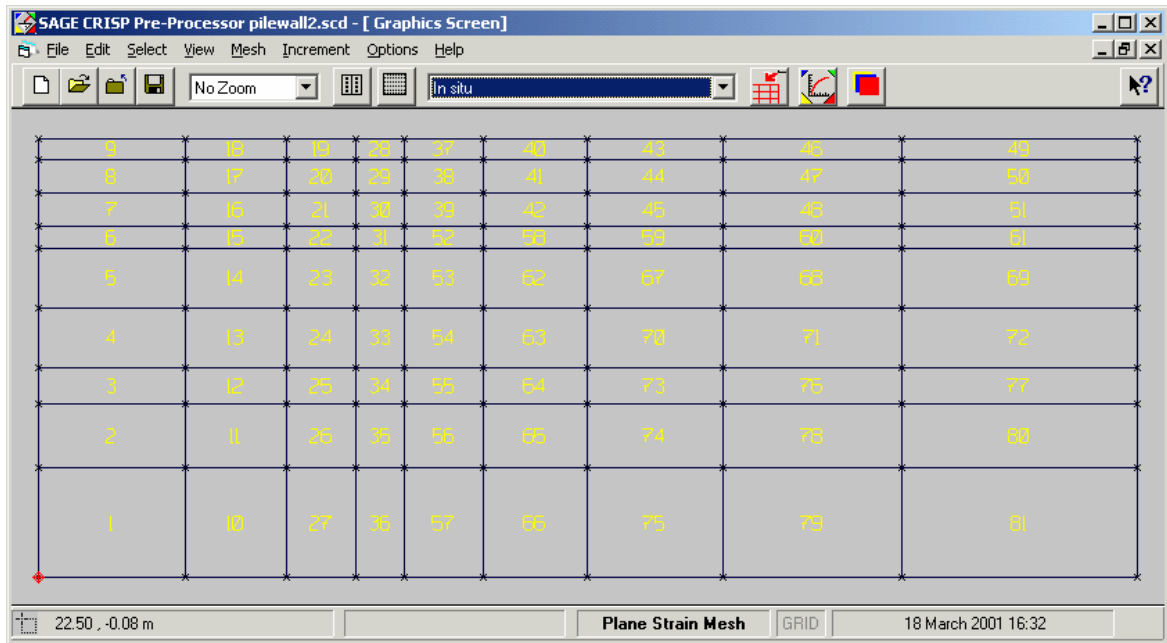
8.4.3 Defining the elements for the concrete slab

We now need to define the elements which form the concrete slab which is to be added after the excavation. There are two way of doing this:

Method 1

Create new elements on top of elements 6, 15 and 22 (see above mesh). You do that as follows

Step 1   click on Mesh-Create Elements and click on the four corner nodes of each of the elements 6,15 and 22. This will create a new element on top of the existing soil element

Step 2   Then select these new elements, assign element properties for them using Mesh-Element Properties. Then select these newly added elements and remove them from the in-situ stage using Increment-Remove Elements

Method 2

You can create a new layer of elements using Layers.

Step 1 First click on View-Layers and click on Add. Enter a layer name. Lets say Concrete Slab.
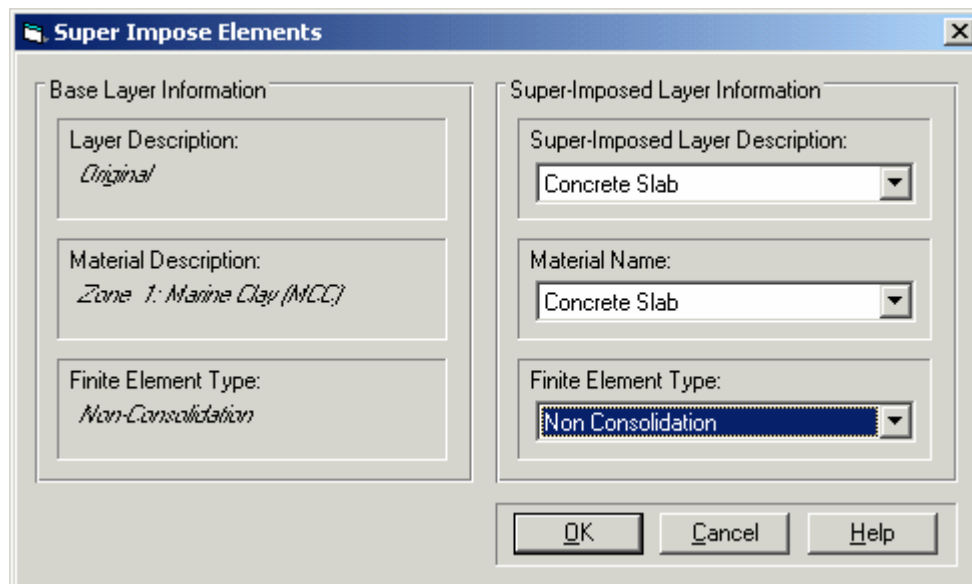
Step 2 Next click on Mesh-Super Impose Elements and fill in details as below. The super-imposed layer description should be the new layer (Concrete Slab). Its associated material zone should be the Concrete Slab which you have defined earlier on. We will use non-consolidation elements for the concrete slab.

Step 3 Click OK and you will see a new set of elements created on top of the existing soil elements.

Step 4 Now we need to remove these elements from the in-situ stage. You can either click on these elements and select the concrete slab elements from the multi-selection menu, or you can switch off the original soil layer so that you can select the concrete elements more easily. To do that you click on View-Layers and untick the original layer. Then select the concrete elements and click on Increment Remove Elements.

Step 5 Then click on View-layers and tick on original layer so as to be able to see it again.

8.4.4   Defining the elements for the wall and prop

Step 1   Click on **Mesh- Create** elements and start creating the wall elements. We will use beam elements for this which will produce the bending moments for us.

Step 2   To create the element between nodes 29 and 34, you click on 29, then 34 then 29 again, then you commit the element. Proceed in the same way to create the wall elements between nodes 34-4 , 4-9, 9-74, 74-79, and 79-82

Step 3   Once you have completed creating the wall elements, you would need assign element properties for them and then remove them from the in-situ stage.

Step 4   To assign element properties click on **Select-Line Elements** and click on the wall elements, then click on **Mesh-Element Properties** and chose the material zone **Sheet Pile Wall (Beam).**

Step 5   Next select these line elements and click on **Increment-Remove Elements** so that they are not present in the in-situ stage.

Step 6   Do the same procedure for the prop which is between nodes 79 and 78
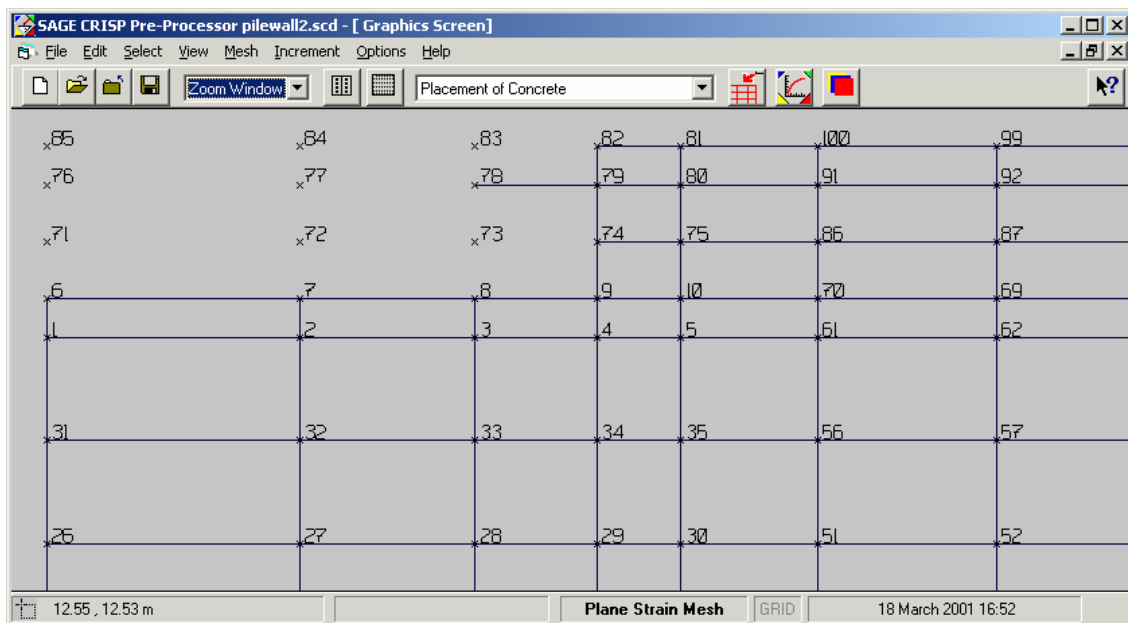
8.4.5   Assigning Element Properties for other material zones

Select on the bottom row of the quadrilateral elements and click on **Mesh-Element properties**. Make sure that these elements have the material zone **Bedrock (Elastic)**

8.4.6   Defining Boundary Conditions

Step 1   Click on Select-Edges, then select all the vertical edges on the left and right sides of the mesh.

Step 2   Click on Increment-Fixities and make sure that the edges have zero fixity in the x-direction.

Step 3   Clear the selection and select the bottom edges. Click on Increment-Fixities and make sure that these edges have zero fixities in both x and y directions.

8.4.7   Defining insitu stresses

Click on Increment-Define In situ stress conditions and enter the following:

	Height (m)	σ'_{xx} (kN/m²)	σ'_{yy} (kN/m²)	σ'_{zz} (kN/m²)	τ_{xy} (kN/m²)	PWP (kN/m²)	$\bar{U}$ (kN/m²)	P'c (kN/m²)
1	12	0	0	0	0	0	0	0
2	11	6.2	10.19	6.2	0	9.81	0	10.19
3	3	55.94	91.71	55.94	0	88.29	0	91.71
4	3	53.2	91.71	53.2	0	88.29	0	91.71
5	0	74.4	128.8	74.4	0	117.72	0	128.28

Plot □ □ □ □ □ □ □

Insert   Delete   OK   Cancel   Plot   Convert Height

8.4.8   Installing the wall

Step 4   Select the "Install Sheet Pile Wall" increment from the increment drop down list.

Step 5   Ensure that you are currently viewing the removed elements and select the 1D elements that represent the wall.

Step 6   Select "**Add elements**" from the increment menu.

Step 7   The elements will now disappear from this view and will be added to the analysis during the first increment block

8.4.9   Apply overburden pressure load

Step 1   Create another increment block by clicking on **Increment-Define Increment Blocks** and enter a new increment block named Initial Excavation

Step 2   Select the edges between nodes 81-100-99-98, then select **Increment-Load** and apply a normal load of 10 Kpa.

Create another increment block and call it Excavate Stage 1

> *Step 1*   Select the "Excavate Stage 1" block from the increment drop down list.

> *Step 2*   Select the two top rows of elements to the left of the wall

> *Step 3*   Select "**remove elements**" from the "**increment**" menu.

> *Step 4*   The excavation stage is now complete and the screen should look like figure 5
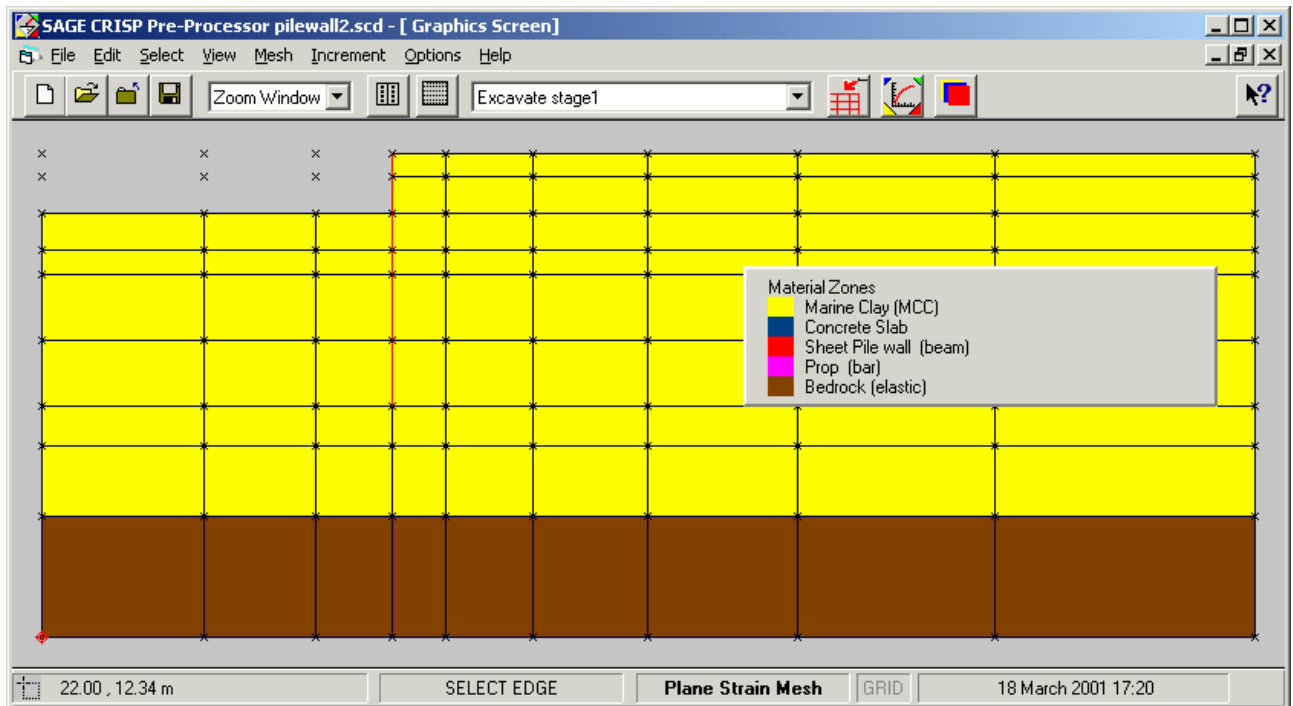
8.4.11   Installing the crest prop

Create another increment block and call it Install prop

> *Step 1*   Select the "Install prop" increment from the increment drop down list.

> *Step 2*   View the removed elements by selecting "removed elements" from the "view" menu

> *Step 3*   Select the 1D element that represents the Strut.

> *Step 4*   Select "add elements" from the "increment" menu.

> *Step 5*   Introduce a nodal fixity to the free end of the prop so that it is held in both x and y directions. You do that by clicking on Select-Nodes, then select the node on the free end of the prop. Select Increment Fixities and introduce zero fixities in the x and y directions.

Step 6   View present elements by selecting "present elements" from the "view" menu. The analysis should now look like figure 6.
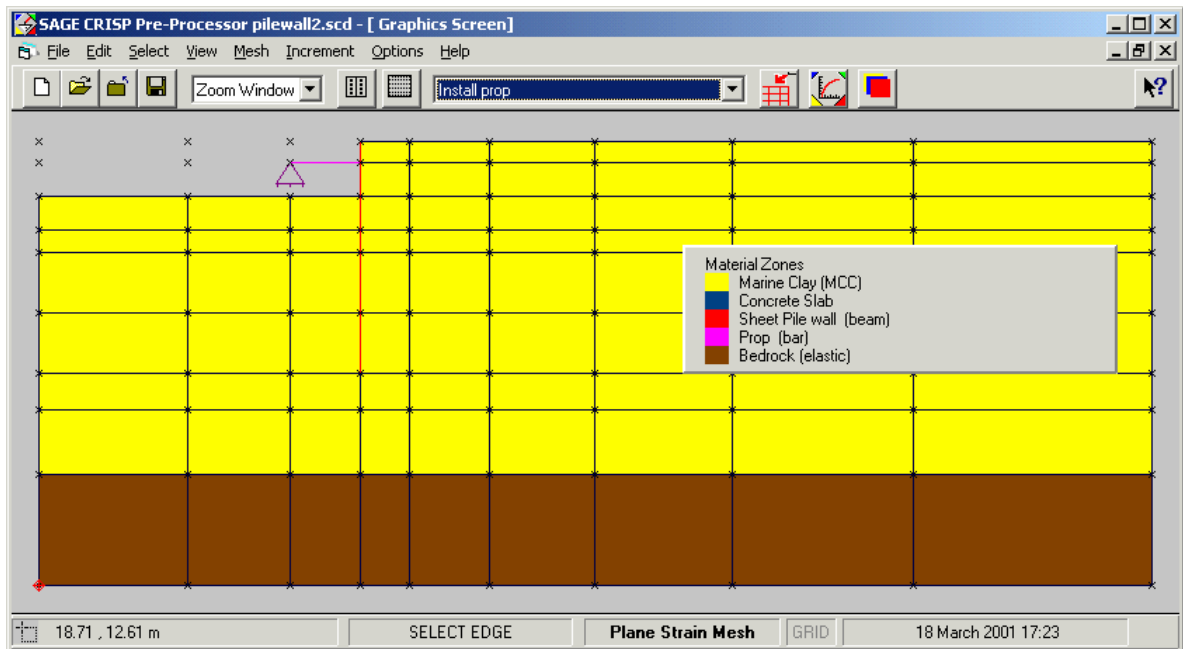

Figure 6 - Strut installed

8.4.12 Continue excavation

Create another increment block and call it "Excavate final stage"

Step 1   Select the "**Excavate final stage**" increment from the increment drop down list.

Step 2   Select the top three rows of elements to the left of the wall

Step 3   Select "**remove elements**" from the "**incremen**t" menu.

Step 4   The excavation stage is now complete and the screen should look like figure 7
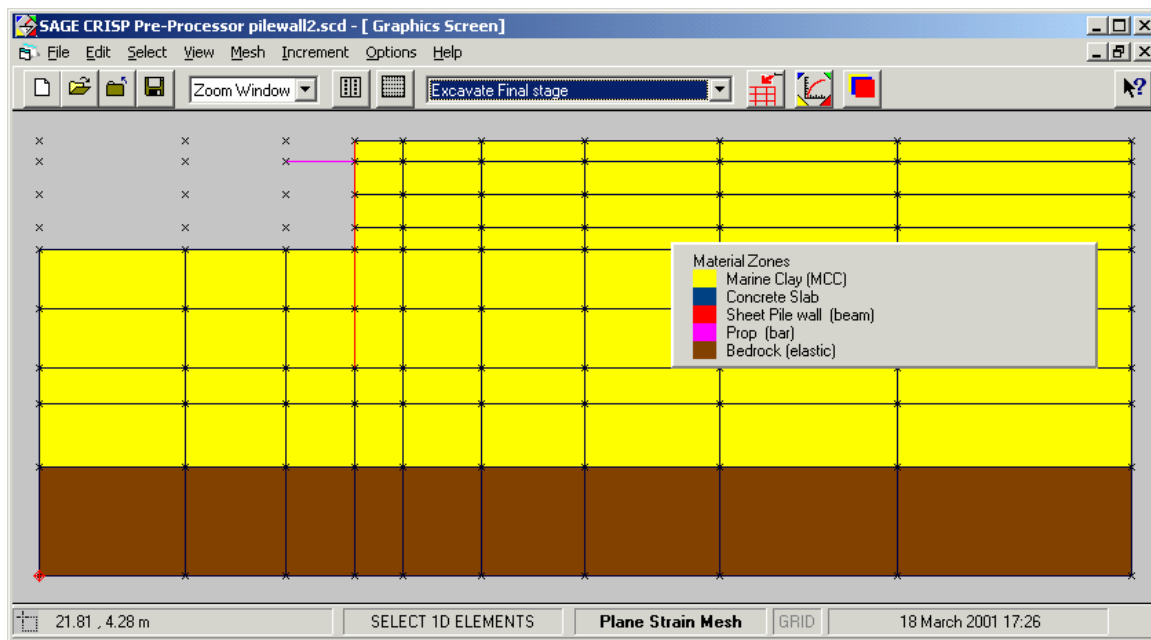
Figure 7 - excavation complete

8.4.13 Installing base slab

The final stage of constuction is to install the base slab. Create another increment block and call it "Placement of Concrete"

Step 1   Select the "Placement of concrete" increment from the increment drop down list.

Step 2   View the removed elements by selecting "**removed elements**" from the "**view**" menu.

Step 3   Select the slab elements. (You will find it easier to select the slab element if you turn the original layer off)

Step 4   Select "**add elements**" from the "**increment**" menu.

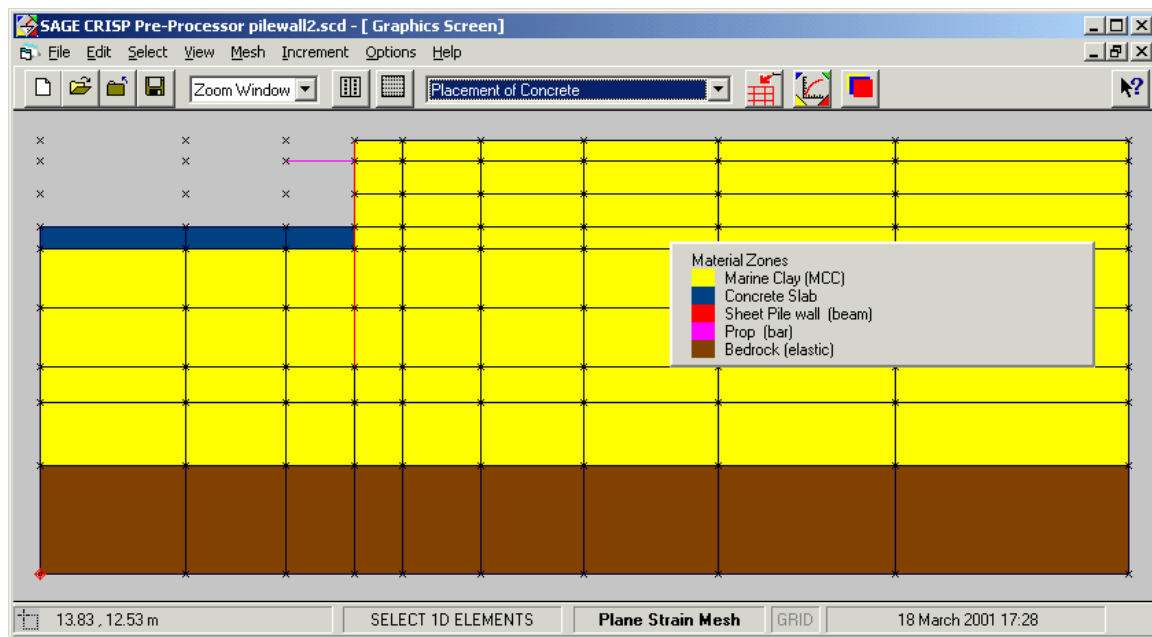*Step 5*   Your excavation stages are now complete and you are ready to run the analysis.

Figure 8 - Completed analysis.

8.4.14 Running The Analysis

You should save the Project now. This concludes the pre processing for this particular problem so all that remains is for you to run the finite element analysis program.

> **Step 6**   Select Save from the File Menu.
>
> **Step 7**   In the File menu, click on Run Analysis... to run the finite element analysis. This displays the Run Analysis dialogue box.
>
> **Step 8**   Click on Create CRISP Files Now. This will write two files with the extensions .GPD and .MPD to disk. These files contain the input data required by the finite element analysis program.
>
> **Step 9**   Click on Run Analysis Now. This will task switch you to a DOS box and will run the finite element program using the data files just created.
>
> **Step 10**   When the analysis is finished, you will be returned to Windows and the output from the analysis is converted into a Microsoft Access database. A percentage bar informs you as to how long you have left to wait.

8.5   Post Processing

In this section you will be introduced to the following features:

■   Bending moment plots

■   Stress state plots
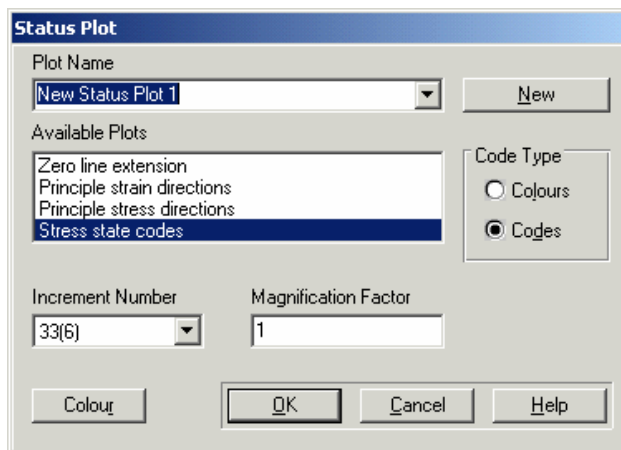
■   Animation sequences

8.5.1   Bending Moment plots

Step 1   Click on View-Material zones

Step 2   Go to the last Increment block (Placement of Concrete)

Step 3   Click on Select -Line Elements and select the element representing the wall.

Step 4   Click on Plots-Bending Moment Plot

Step 5   Fill in the Bending Moment plot as shown below. Remember to choose the last increment in the Start box so the that bending moment for the last increment could be presented

Step 6   Try doing a similar plot for the 2D quadrilateral elements which represent the concrete slab.

8.5.2   Stress State plot

Step 1   Clear the previous bending moment plot by clicking on Plots-Clear Current plot

Step 2   Click on Plots-Status plot and fill in the dialogue box as shown below. Remember to choose the last increment in the Increment Number box

> *Step 3*   Click on View-Legend to see what the stress codes mean

8.5.3   Animation plots

8.5.4   Setting up an animation sequence.

Before you can run an animation you must define a sequence of increments to display in the animation.

> *Step 2*   Click on the "new" button to define a new sequence. This will display all the
> increments that have output defined in the left hand list.
>
> *Step 3*   Select all the increments by selecting the first and then holding down the shift
> key while selecting the last.
>
> *Step 4*   Click the ">>" button to add the selected increments to the list on the right hand
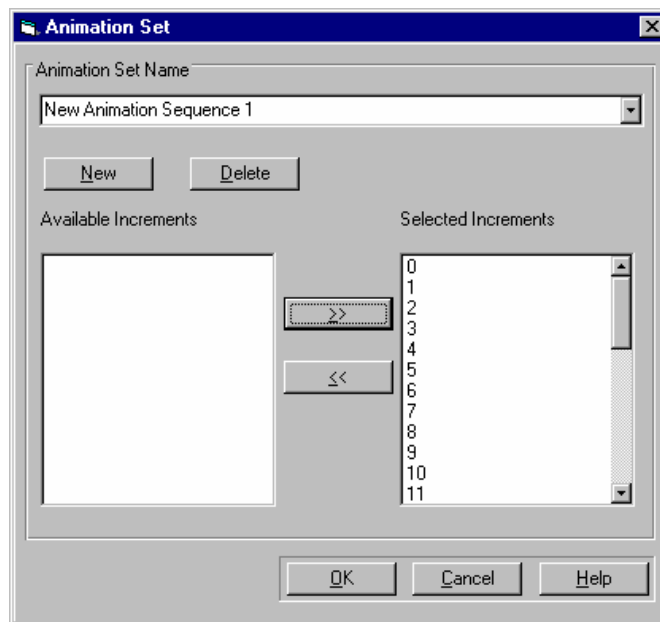> side. See figure 10

Figure 10 - Setting up an anlaysis sequence

8.5.5   Running an animation

Step 1   Select deformed mesh plots from the animation menu

Step 2   Click the new button to define a new plot

Step 3   Select "New animation sequence 1" from the Animation drop down list

Step 5   Set the scale factor = 1. The animation plot form should not appear as figure 11

Step 6   Click OK to display the animation plot.

Figure 11 - deformation plot animation form.

8.5.6   Benefits of Animation plots.

In previous examples we have seen how you can plot the deformed mesh at any increment block. If you have a large analysis it can be a time consuming process to view a deformed mesh for each stage. The animation plots allow you to view all stages of the analysis at once and locate areas of problems in you analysis faster.

8.5.7   Things to try next

With the experience you have gained from this and previous analysis try the follow;

- Run an animation plot for stress state codes (you can use the sequence set up for the deformation plot)
- Plot a graph showing the deformation of the wall at the end of each construction stage.