COLLAPSE ANALYSIS OF FOOTING ON COHESIVE SOIL

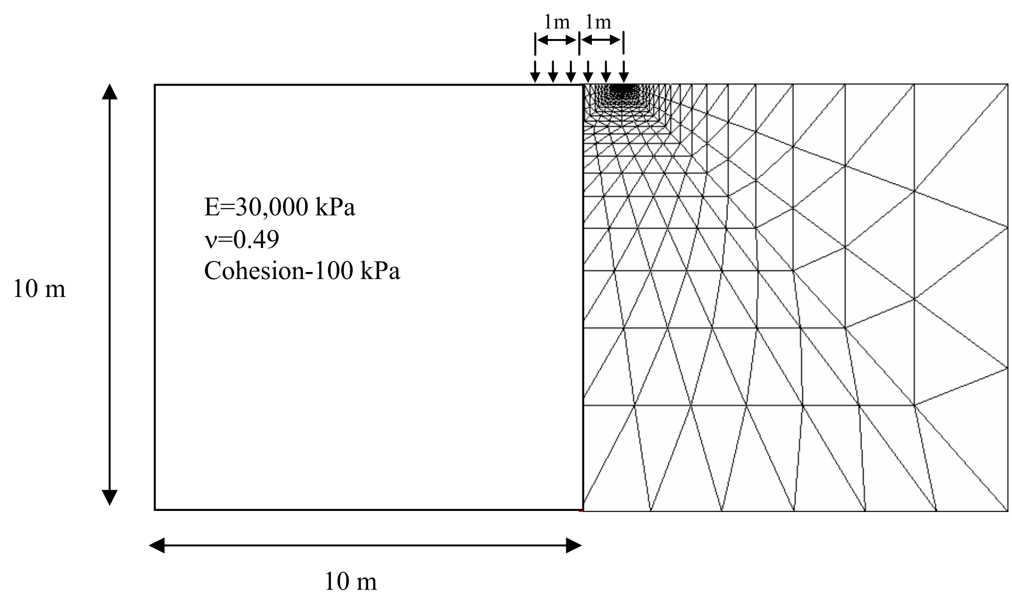


Figure 1 Strip Footing

Figure 1 shows the mesh and the soil data for a plane strain strip footing on purely cohesive soil with constant strength.

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.

The analytic solution is exact only for an infinite half-space, whereas the solution obtained here is for a layer of finite depth. The analytical solution for a strip footing gives a bearing capacity factor Nc of 5.1415 (ie $Nc=2+\pi$)

1. 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).

1.1 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

1.2 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 SAGE CRISP Pre-Processor, select New Project... from the File menu. This displays the Mesh Mode dialogue box:

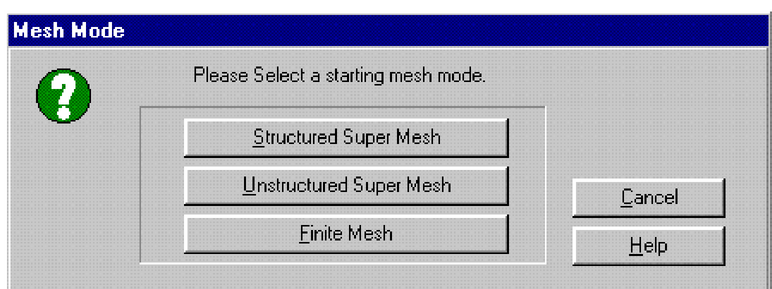


Figure 2 Mesh Mode dialogue box

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.

1.3 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 3.

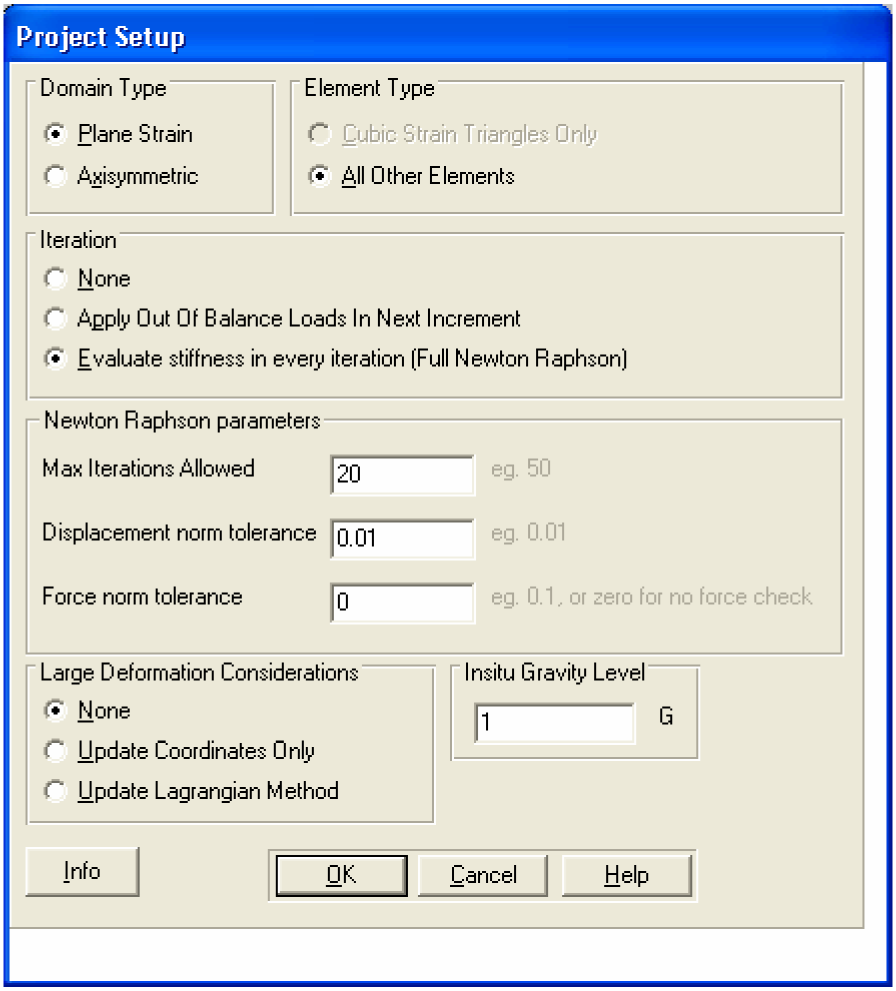


Figure 3 Project Setup dialogue box

1.4 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.

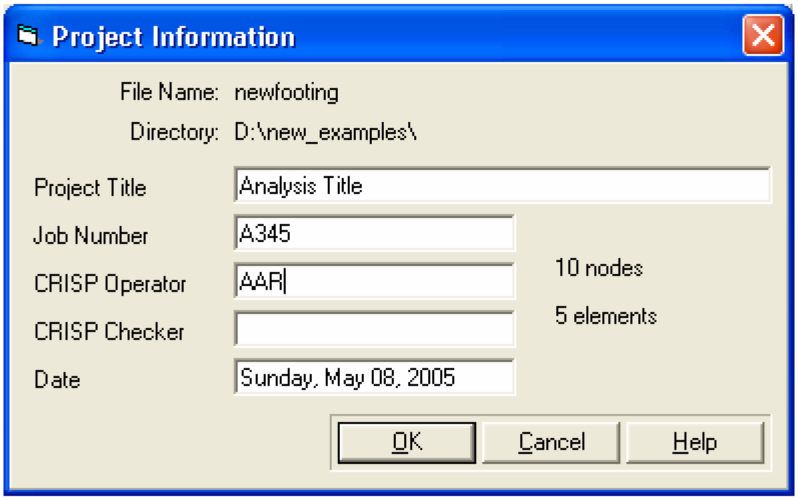


Figure 4 Project Information dialogue box

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

Step 4 Click on OK to exit.

1.5 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.

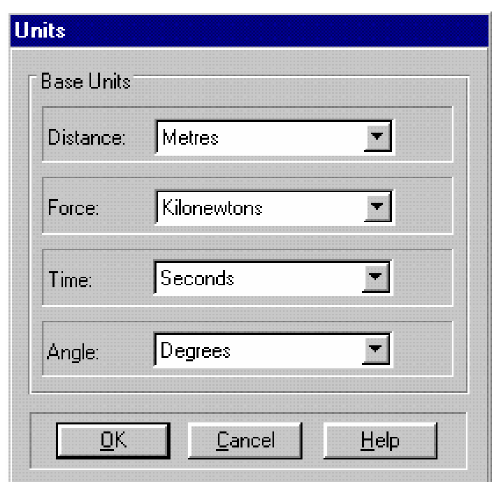


Figure 5 Units dialogue box

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

1.6 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 we use a special structured mesh which aims to create the finest elements in the area of maximum stress activity or shear stress. This is around the edge of the foundation and in the corner of the edge of the foundation.

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:

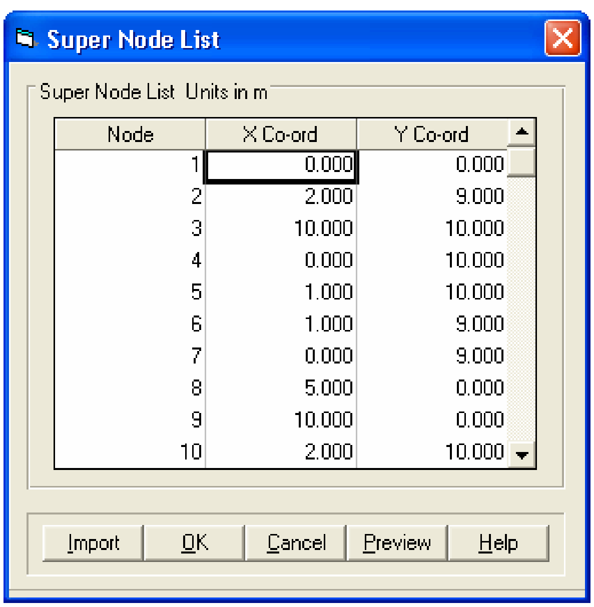
- Drawing A Super Mesh
- Defining Edge Gradings
- Defining A Material Zone
- Assigning Element Properties
- Generating The Finite Element Mesh

1.7 Drawing The Super Mesh

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 Click on Mesh>Super Node List.. and fill in .the nodal coordinates as shown in the figure below.



Step 3

Create the super elements by going through each corner node. So for example, to create super element 1 as in the figure below, click on super nodes 7,6,5,4 and back on 7 again. Make sure you click In the Edit menu, click on Commit Super Element Creation. Repeat the process for all the remaining four super elements.

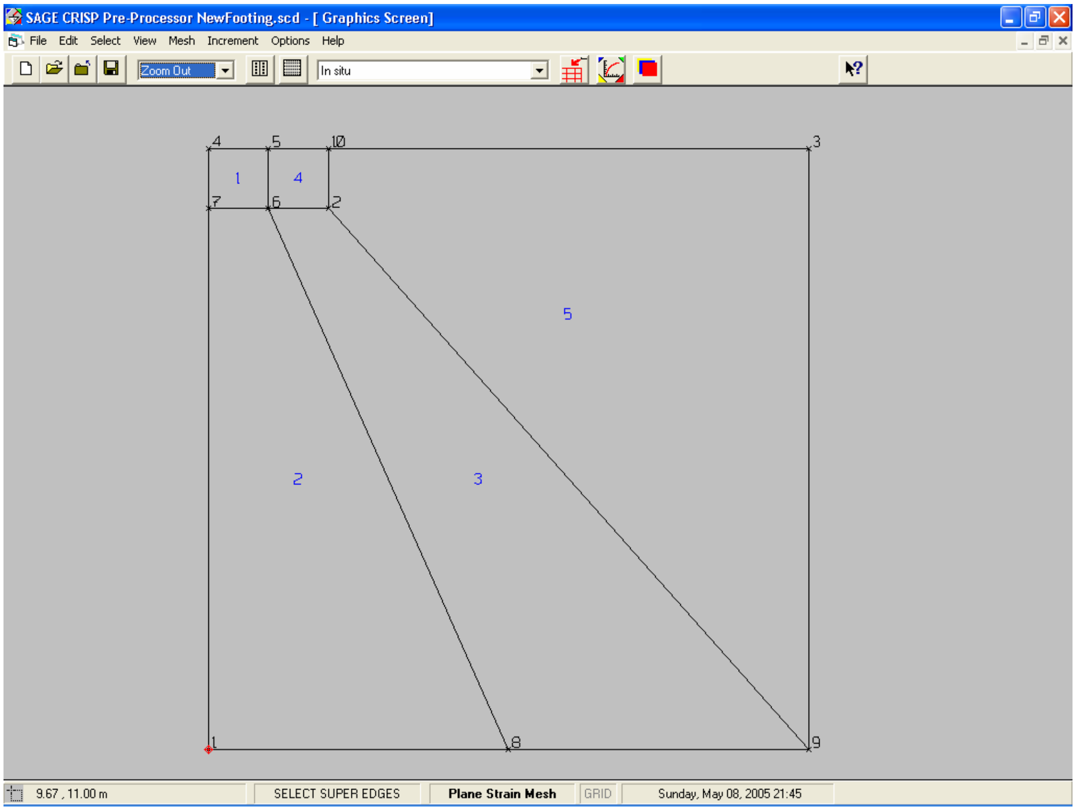


Figure 6 Mesh of super elements

1.8 Defining Edge Gradings

In this example we will use the Macro Element Type M3 (Two-Way) in the structured mesh. For full details of this type of structured mesh generation method, refer to the mesh generator document SWAN, "Macro Element Specification For Structured Mesh Generation" by Johann Sienz, University of Wales, Swansea.

The macro element type M3 is divided into three sub-macro elements before meshing starts. A type M3 macro element is shown in Figure below. The name Two-Way macro element is derived from the fact that it allows the transition from a fine mesh to a coarse mesh in both direction.

The macro element type M3 is characterised by:

- The difference between the number of subdivision on opposite sides must be the same and must be more than zero. In other word, number of subdivisions: $n1 - n3 = n2 - n4$, and $n1 - n3 > 0$ and $n2 - n4 > 0$.

- The orientation of the local ξ-η coordinate system with respect to the global x-y coordinate system is given by the number of subdivisions. The element is rotated until the conditions above are fulfilled.

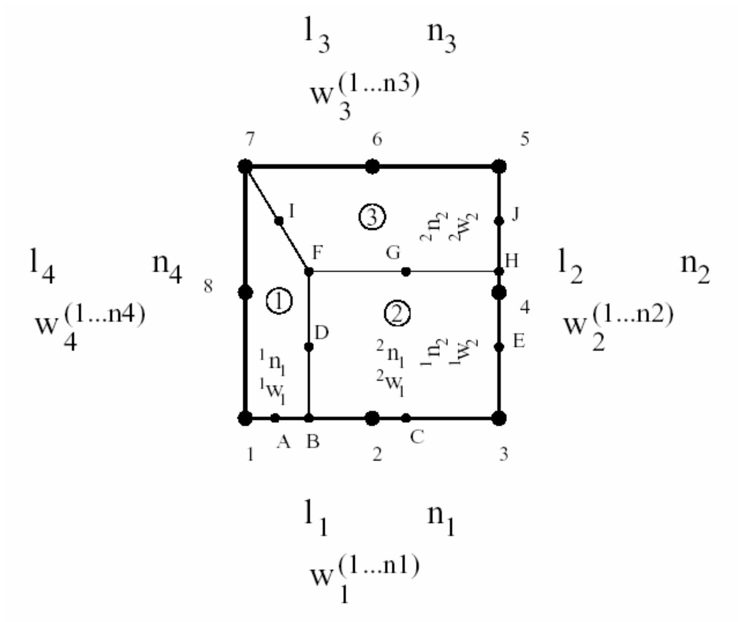


Figure 7 Two Way Macro Element weights and subdivisions

Example 1:

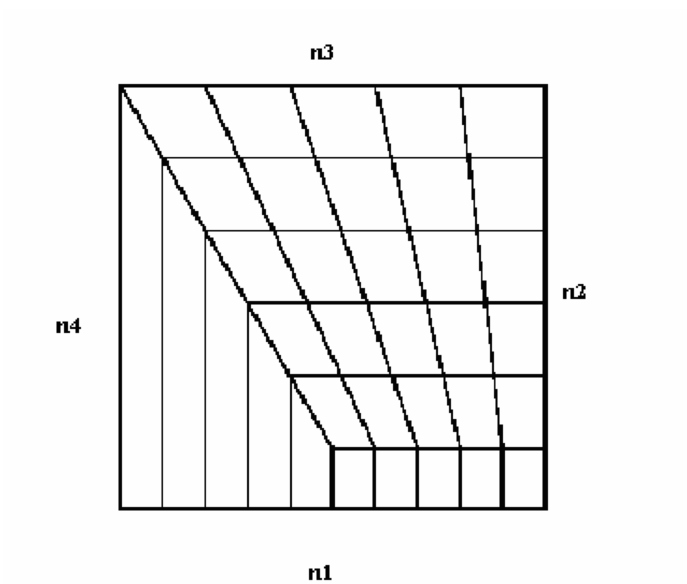


Figure 8 Example of a two way macro element

In this example, n1=10, n2=6, n3=5 and n4=1

The conditions n1 - n3 = n2 - n4, n1-n3 > 0 and n2 - n4 > 0 are satisfied

Example 2

This is our example. We have the following arrangement:

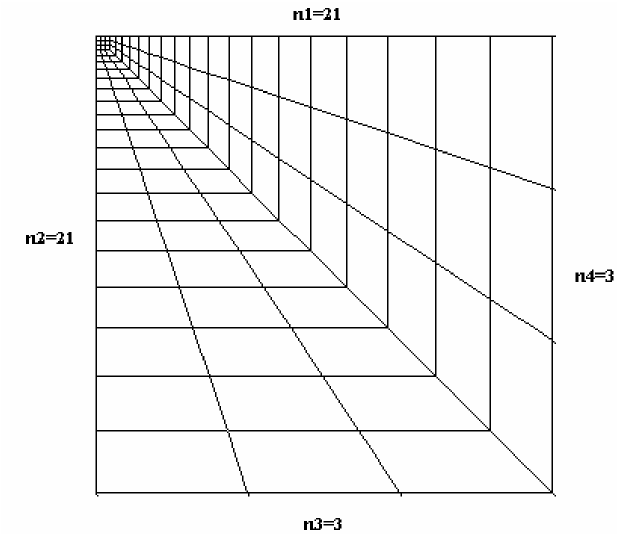


Figure 9 Another example of a Two Way Macro Element weights and subdivisions

Again, the conditions n1 - n3 = n2 - n4, n1 - n3 > 0 and n2 - n4 > 0 are satisfied

Start defining the edge gradings as follows:

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. When an edge is selected, one or other of that edge's nodes is also highlighted. Ensure that the highlighted node (with a pink square) is the one on the right-hand end of the edge by clicking closer to this end than the other when you select the edge (ie click closer to node 5 in the figure below). If the wrong node is highlighted, your gradings will appear the wrong way around.

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

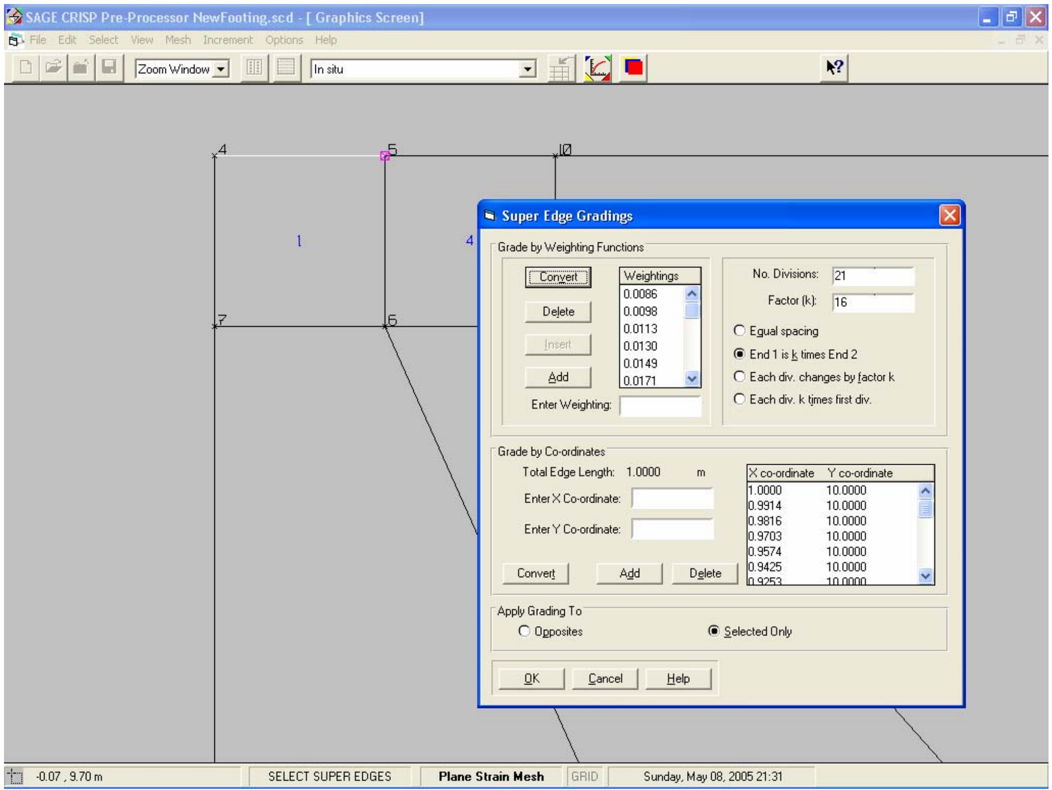


Figure 10 Defining Edge Gradings for the Top Left-Hand Edge which will form the footing

Step 3 Enter '21' into the Number of Divisions text box and click on the End 1 is k times End 2. option. Enter '16' into the Factor (k) text box. Weighting values will be automatically entered and the dialogue box should now look like Figure 10.

Step 4 Make sure you click on option Selected Only. This is very important, or else you would not be able to produce the desired fine mesh.

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

Step 6 In the Select menu, click on Clear Selection, or click again on the selected edge to de-select that edge. Click on the next top left -hand (horizontal) edge as shown below. Ensure that the highlighted node (with a pink square) 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 (ie click closer to node 5 in the figure below). If the wrong node is highlighted, your gradings will appear the wrong way around.

Step 7 In the mesh menu, click on Super Edge Grading... to again display the Super Edge Gradings dialogue box.

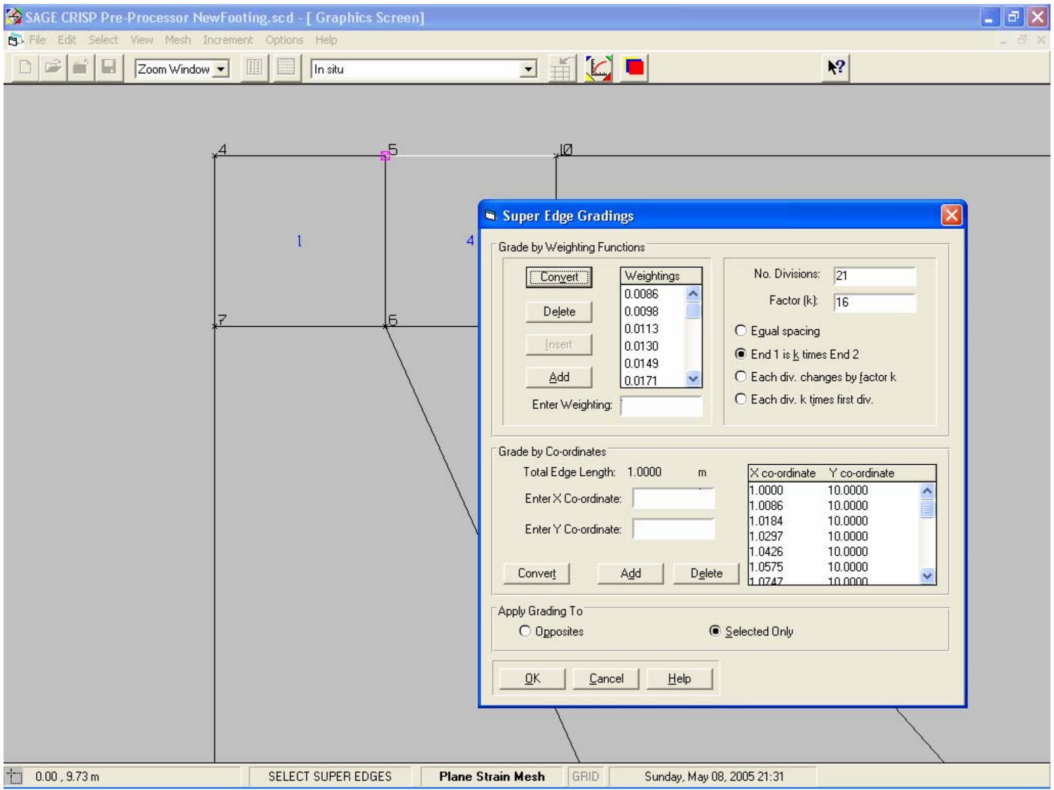


Figure 11 Defining Edge Gradings for the Top Left-Hand Edge which will form the area to the right of the footing

Step 8 Enter '21' into the Number of Divisions text box and click on the End 1 is k times End 2 option.. Enter '16' into the Factor (k) text box. Weighting values will be automatically entered and the dialogue box should now look like Figure 11.

Step 9 Make sure you click on option Selected Only.

Step 10 In the Select menu, click on Clear Selection, or click again on the selected edge to de-select that edge. Click on the vertical edge near the top left corner as shown below. Ensure that the highlighted node (with a pink square) is the one on the top end of the edge by clicking closer to this end than the other when you select the edge (ie click closer to node 5 in the figure below).

Step 11 In the mesh menu, click on Super Edge Grading... to again display the Super Edge Gradings dialogue box.

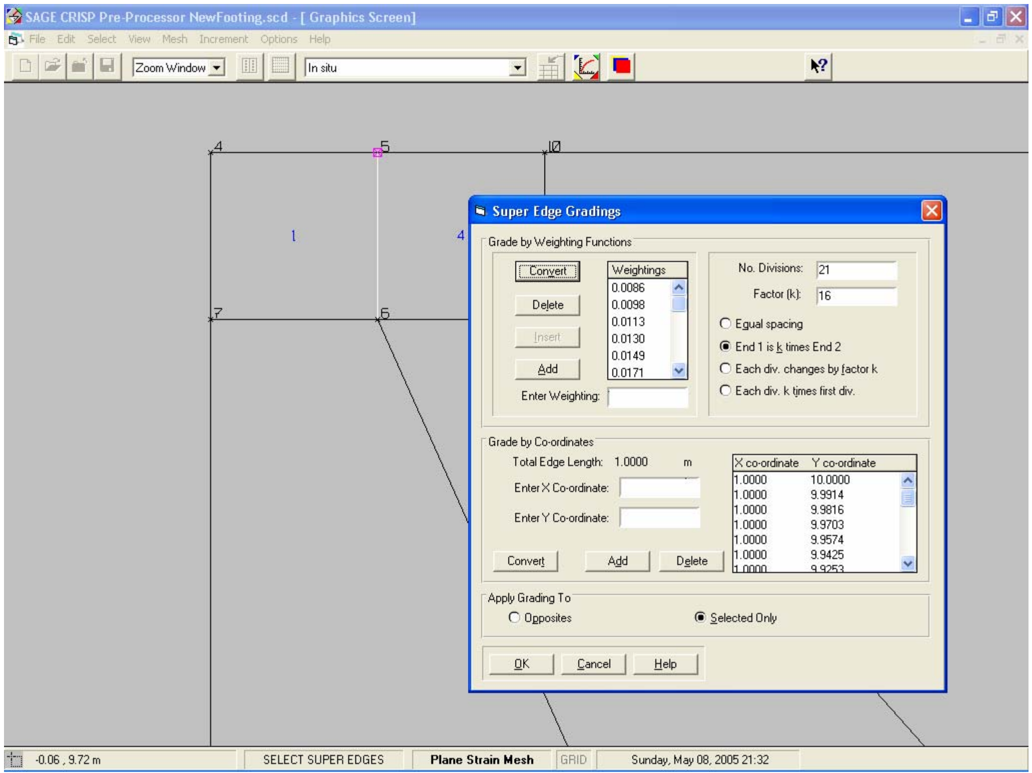


Figure 12 Defining Edge Gradings near the footing corner

Step 12 Enter '21' into the Number of Divisions text box and click on the End 1 is k times End 2 option. Enter '16' into the Factor (k) text box. Weighting values will be automatically entered and the dialogue box should now look like Figure 12

Step 13 Make sure you click on option Selected Only.

Step 14 In the Select menu, click on Clear Selection. Click on the vertical edge near the top left corner as shown below. It doesn't matter if you click near any of the edge nodes on this edge as we intend to have equal spacing for this edge.

Step 15 In the mesh menu, click on Super Edge Grading... to again display the Super Edge Gradings dialogue box.

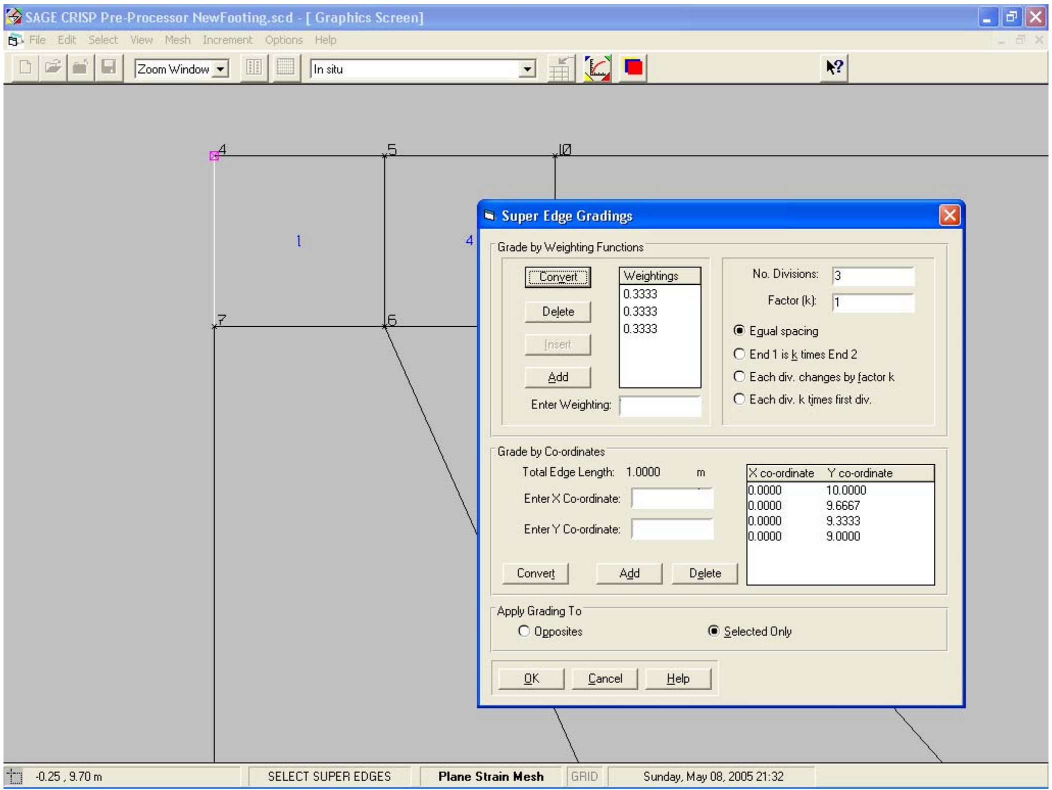


Figure 13 Defining Edge Gradings near the footing corner

Step 16 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 13.

Step 17 Make sure you click on option Selected Only.

Step 18 In the Select menu, click on Clear Selection, or click again on the selected edge to de-select that edge. Click on the horizontal edge near the top left corner as shown below. It doesn't matter if you click near any of the edge nodes on this edge as we intend to have equal spacing for this edge.

Step 19 In the mesh menu, click on Super Edge Grading... to again display the Super Edge Gradings dialogue box.

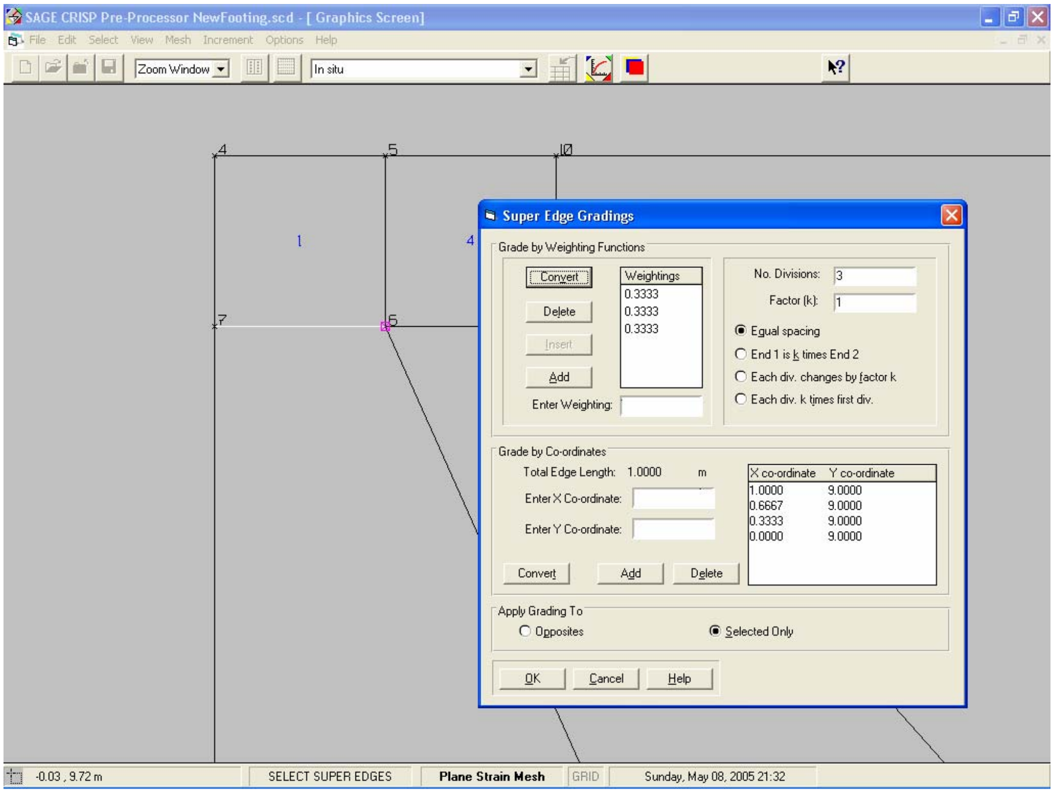


Figure 14 Defining Edge Gradings near the footing corner

Step 20 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 14.

Step 21 Make sure you click on option Selected Only.

Step 22 In the Select menu, click on Clear Selection. Click on the next horizontal corner as shown below. It doesn't matter if you click near any of the edge nodes on this edge as we intend to have equal spacing for this edge.

Step 23 In the mesh menu, click on Super Edge Grading… to again display the Super Edge Gradings dialogue box.

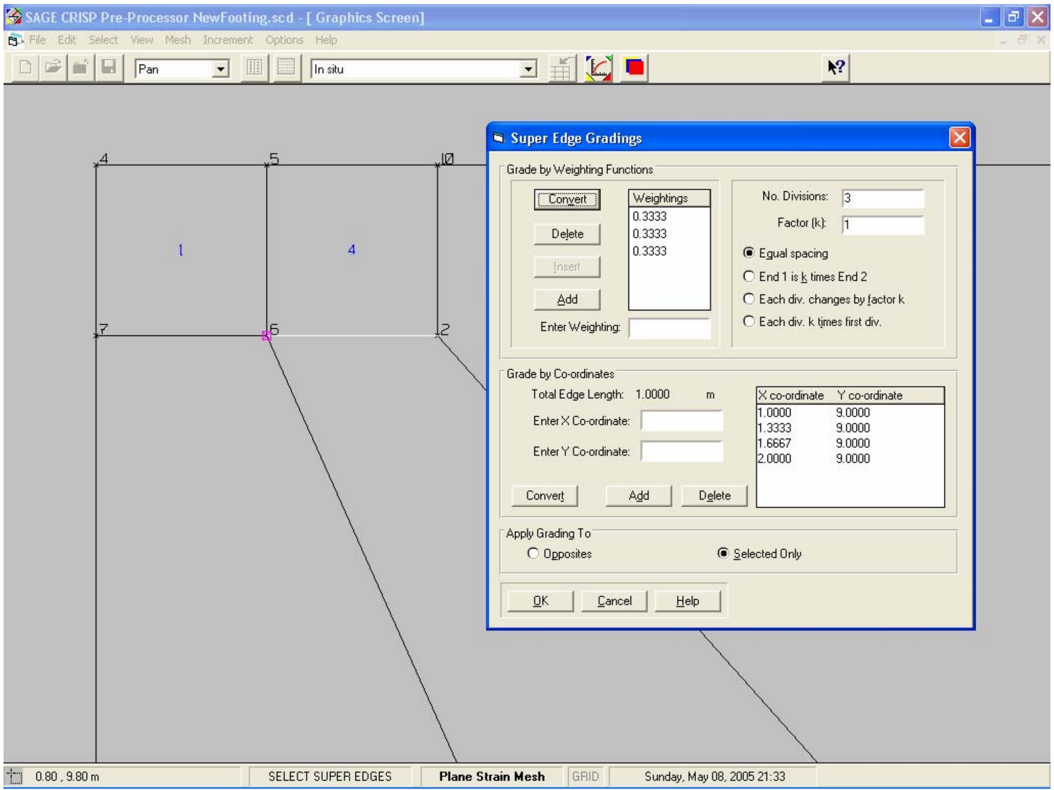


Figure 15 Defining Edge Gradings near the footing corner

Step 24 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 15.

Step 25 Make sure you click on option Selected Only.

Step 26 In the Select menu, click on Clear Selection.. Click on the vertical edge on the left side as shown below. Ensure that the highlighted node (with a pink square) is the one on the top end of the edge by clicking closer to this end than the other when you select the edge (ie click closer to node 5 in the figure below).

Step 27 In the mesh menu, click on Super Edge Grading... to again display the Super Edge Gradings dialogue box.

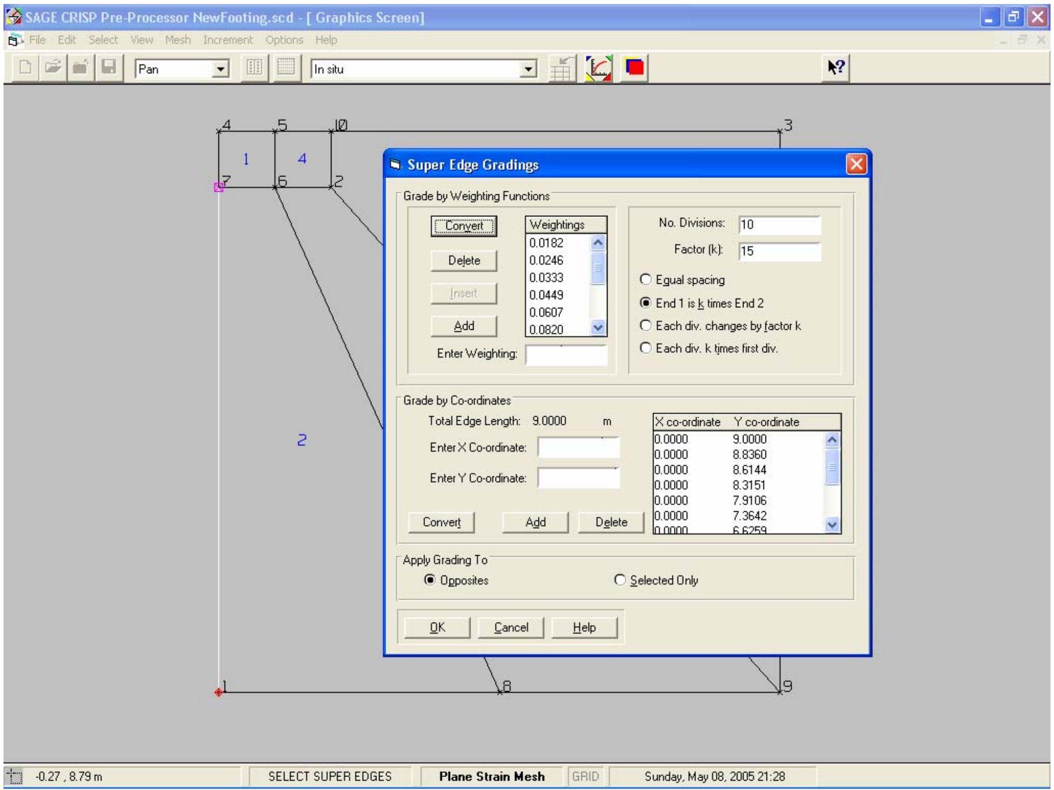


Figure 16 Defining Edge Gradings on vertical side and opposite sides

Step 28 Enter '10' into the Number of Divisions text box and click on the End 1 is k times End 2. option. Enter '15' into the Factor (k) text box. Weighting values will be automatically entered and the dialogue box should now look like Figure 16.

Step 29 Make sure you click on option Opposites. Notice that we have changed this option now as we intend to give equal subdivisions to opposite sides facing this vertical side.

Step 30 In the Select menu, click on Clear Selection,. Click on the horizontal bottom edge on the left as shown below. It doesn't matter if you click near any of the edge nodes on this edge as we intend to have equal spacing for this edge.

Step 31 In the mesh menu, click on Super Edge Grading... to again display the Super Edge Gradings dialogue box.

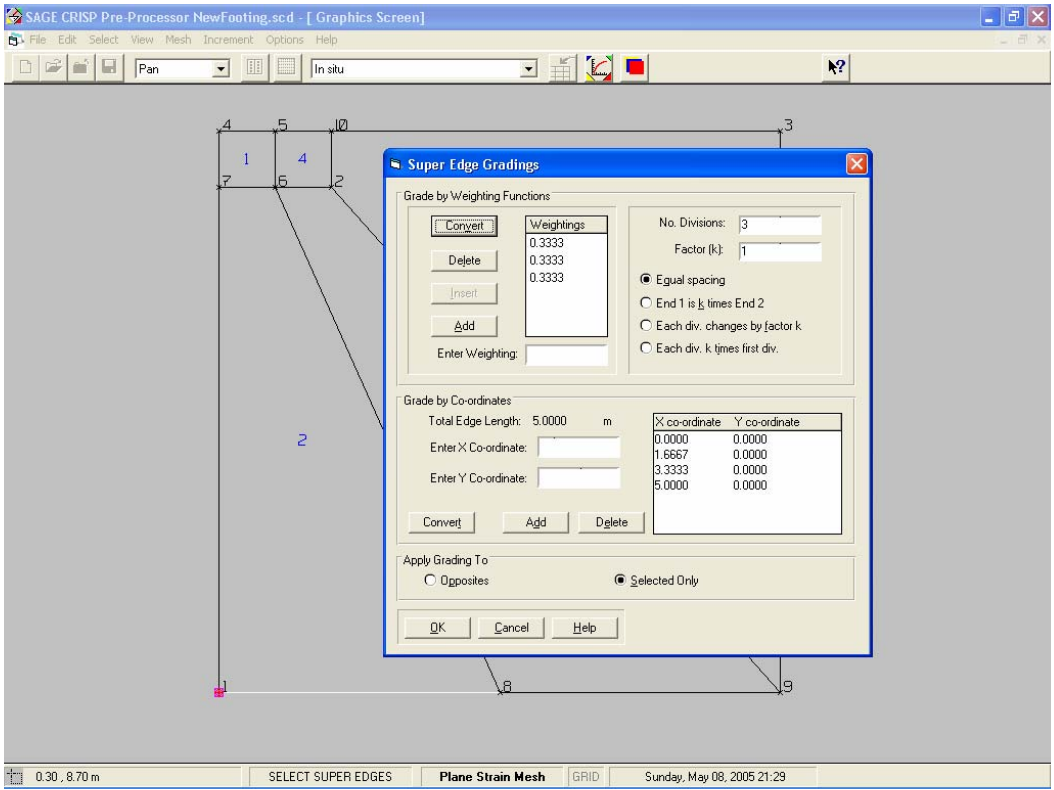


Figure 17 Defining Edge Gradings on bottom side

Step 32 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 .

Step 33 Make sure you click on option Selected Only. This is very important, or else you would not be able to produce the desired fine mesh.

Step 34 In the Select menu, click on Clear Selection. Click on the next bottom edge on the right side as shown below. It doesn't matter if you click near any of the edge nodes on this edge as we intend to have equal spacing for this edge.

Step 35 In the mesh menu, click on Super Edge Grading... to again display the Super Edge Gradings dialogue box.

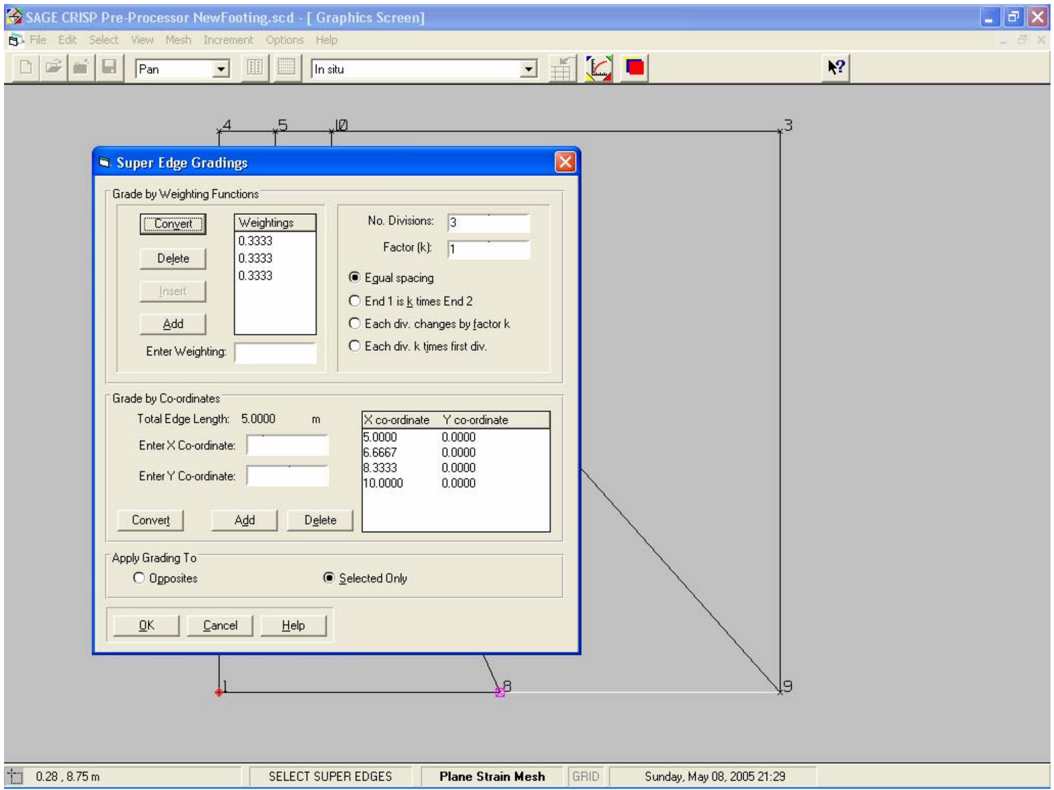


Figure 18 Defining Edge Gradings on bottom side

Step 36 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 18.

Step 37 Make sure you click on option Selected Only.

Step 38 In the Select menu, click on Clear Selection. Click on the vertical edge on the right as shown below. It doesn’t matter if you click near any of the edge nodes on this edge as we intend to have equal spacing for this edge.

Step 39 In the mesh menu, click on Super Edge Grading... to again display the Super Edge Gradings dialogue box.

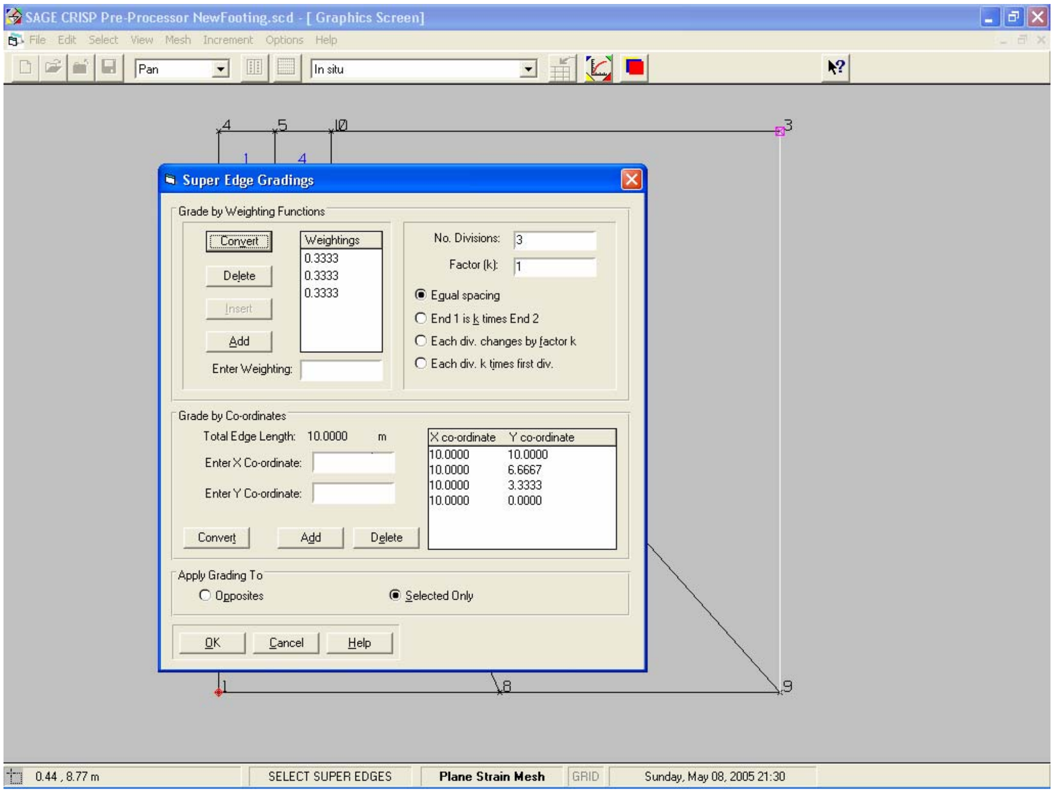


Figure 19 Defining Edge Gradings on right side

Step 40	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 19.
Step 41	Make sure you click on option Selected Only.
Step 42	Click on OK to exit, once you have finished.
Step 43	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 20.

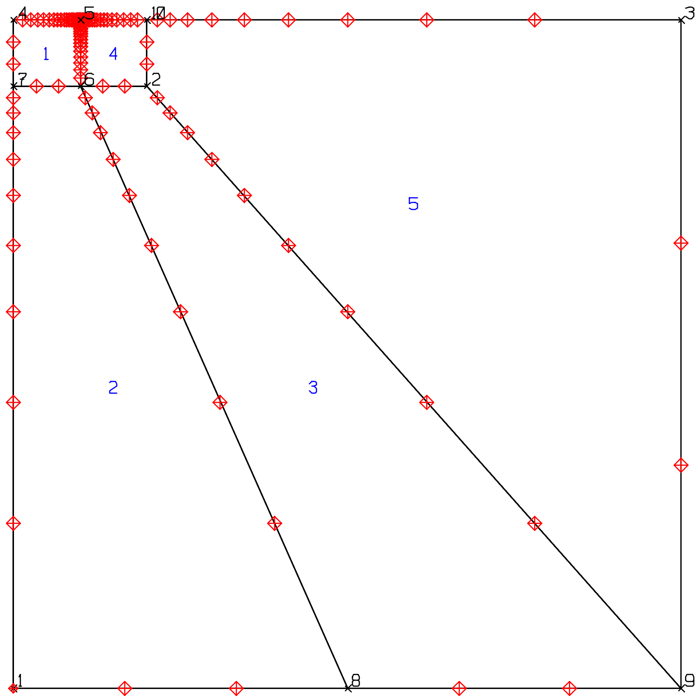


Figure 20 Fully Assigned Edge Gradings

1.9 Defining A Material Zone

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

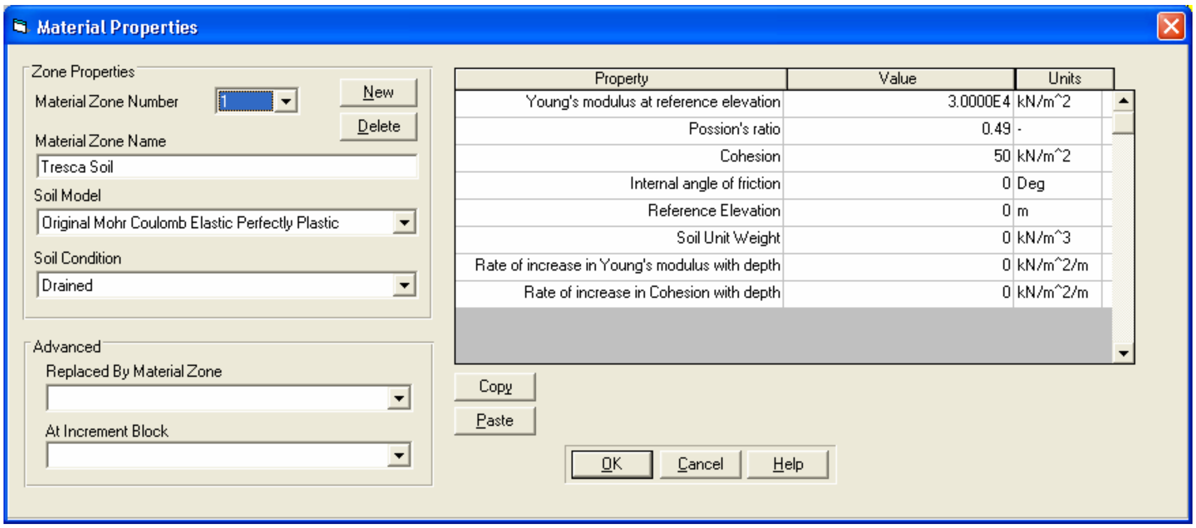


Figure 21 Material Properties dialogue box

Step 2 Enter the name 'Tresca soil' and select 'Original Mohr Coulomb Elastic-perfectly plastic' from the Soil Model list box..

Step 3 Enter the material property values as shown above.

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.

1.10 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.

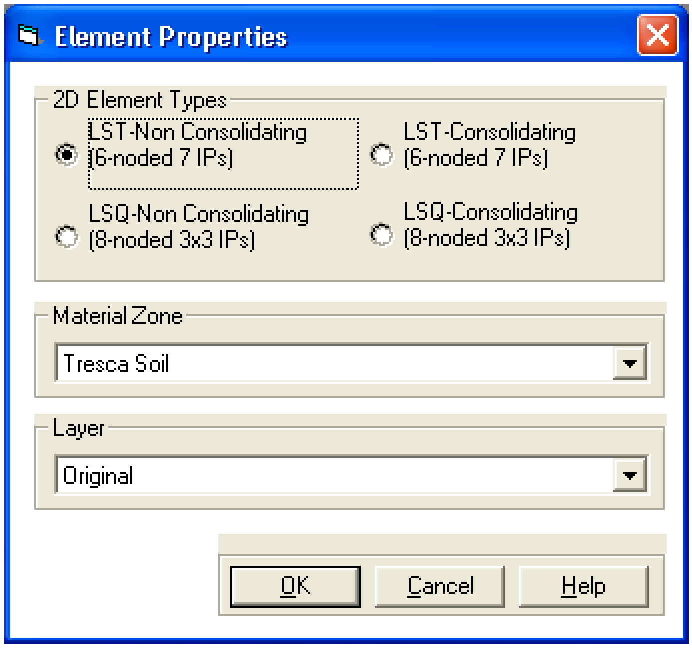


Figure 22 Defining Element Type and Zone

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

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

The Element Properties dialogue box should now look like Figure 22. Click on OK to exit, once you have finished.

1.11 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 23 . The menu bar will now allow you access to the additional options available in Finite Element Mesh mode.

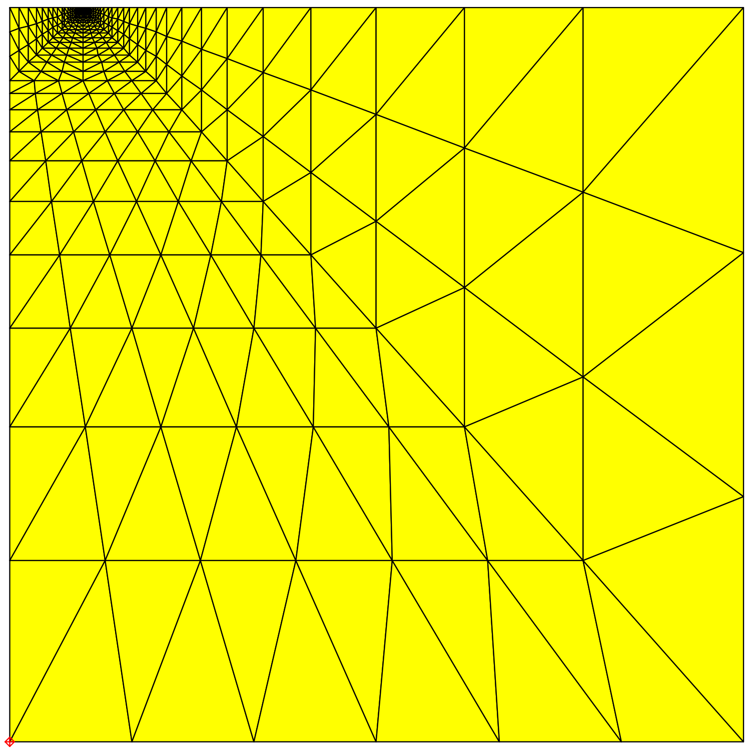


Figure 23 Automatically Generated Finite Element Mesh

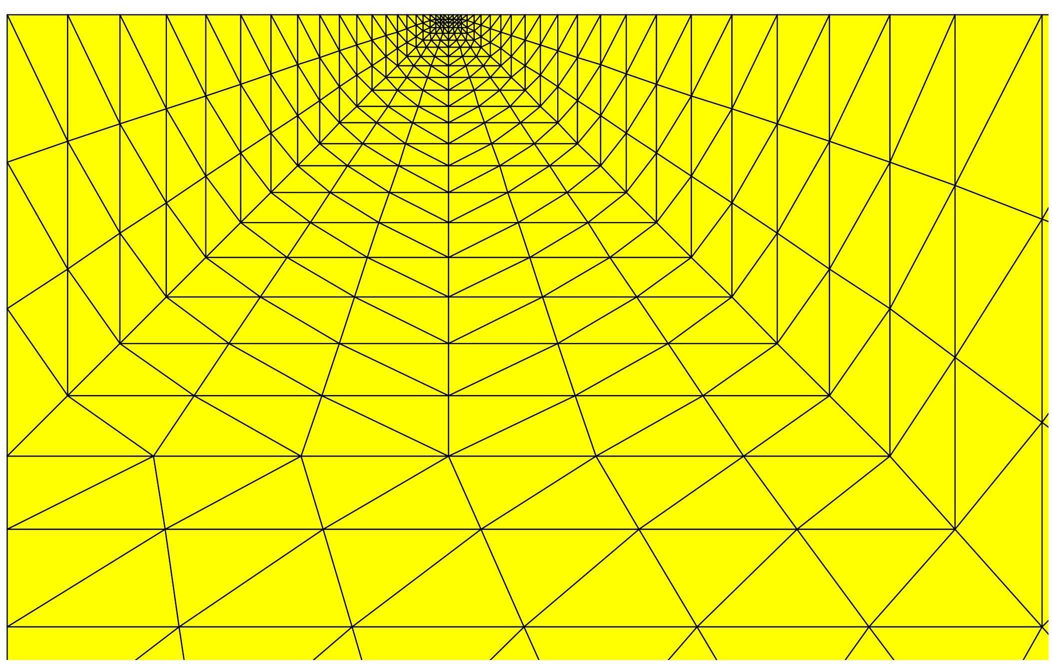


Figure 24 Finite Element Mesh showing the fine elements area around the edge of the footing (footing not shown)

1.12 Defining In Situ Stresses

This example assumes weightless soil as we are interested in the collapse response which is not usually influenced by the in-situ stresses. But we must always defined in-situ stresses, even when they are equal to zero.

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

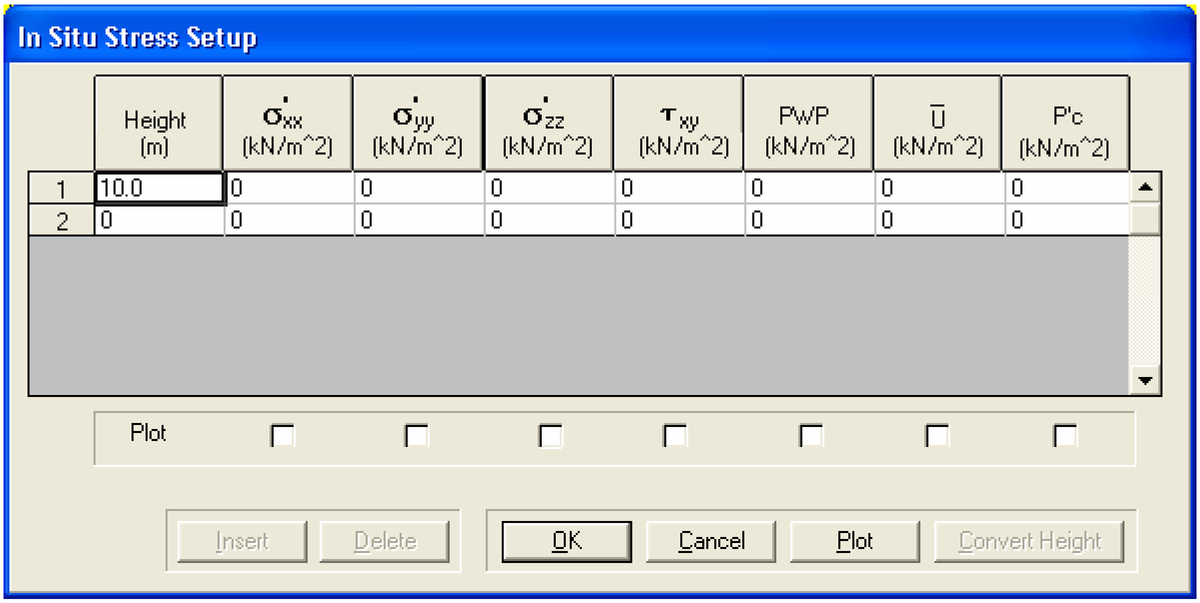


Figure 25 In Situ Stress Setup dialogue box

Step 2 Enter zero values as shown above.

1.13 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.

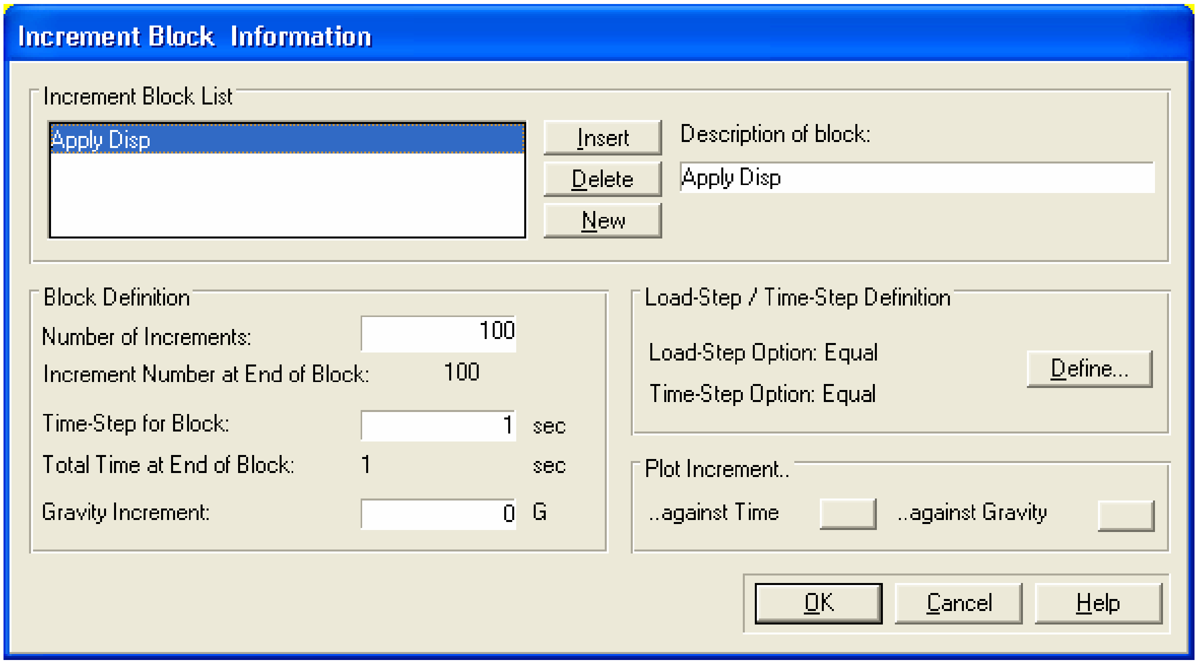


Figure 26 Defining Increment Blocks

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 100 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.

1.14 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.

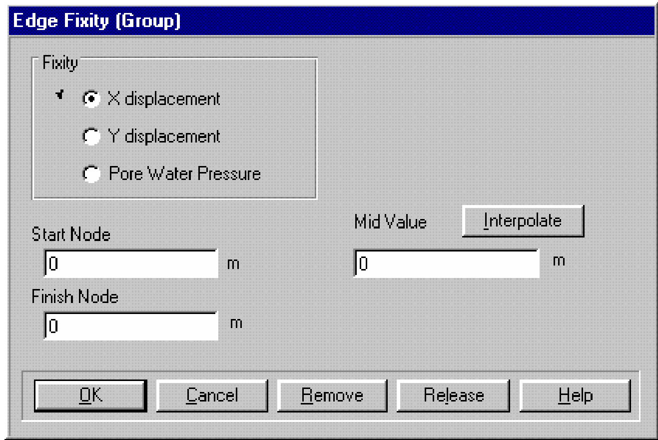


Figure 27 Defining Edge Fixities

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 Graphics Window should look similar to Figure 28. 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).

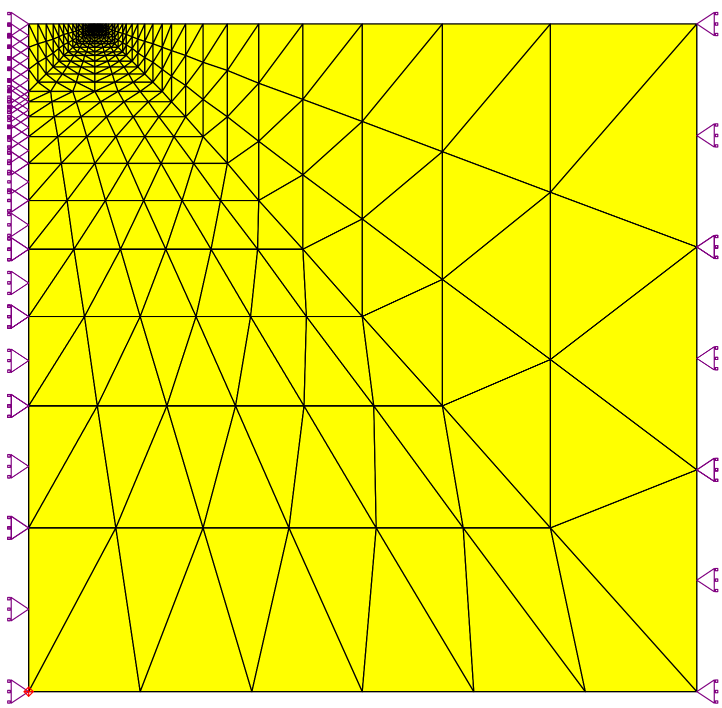


Figure 28 Fixities Defined for the Left and Right Vertical Sides

Step 8 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 9 In the Increment menu, click on Fixities... and define zero X displacement fixities as before.

Step 10 Click on the Y displacement option button.

Step 11 Specify Y direction fixities of zero in the Start Node, Finish Node and Mid-Value text boxes.

Step 12 Click on OK to exit, once you have finished. The Graphics Window should look similar to Figure 29.

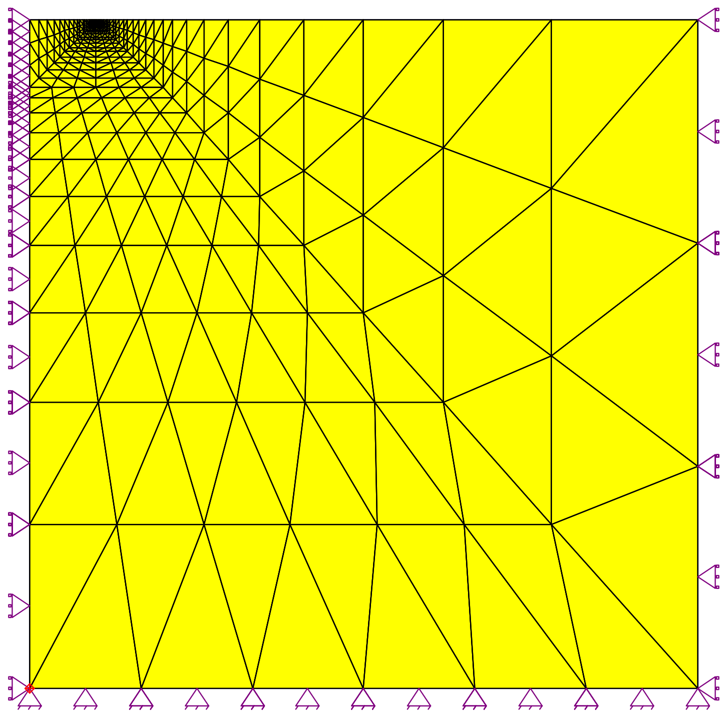


Figure 29 In Situ Fixities Fully Defined

1.15 Defining Applied displacement

This problem assumes a fully smooth footing. Therefore, in order to find the collapse load, we applied a set of specified vertical displacements directly to the soil nodes which were meant to interface with the foundation. The foundation need not be modelled in this case.

Step 1 Select 'Apply Displacements' 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. These can be seen in the figure below on the finite element mesh. You may need to zoom in to select the edges in the very fine elements area at the edge of the footing.

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

Step 5 Enter a value of -0.2m in the Y displacement Start node and Normal Finish node text boxes. Click on the Interpolate button and the Normal Mid Value will automatically be entered as -0.2 m. Alternatively the value can be entered manually.

Step 6 The Edge Load dialogue box should now look like Figure 30. Click on OK to exit, once you have finished.

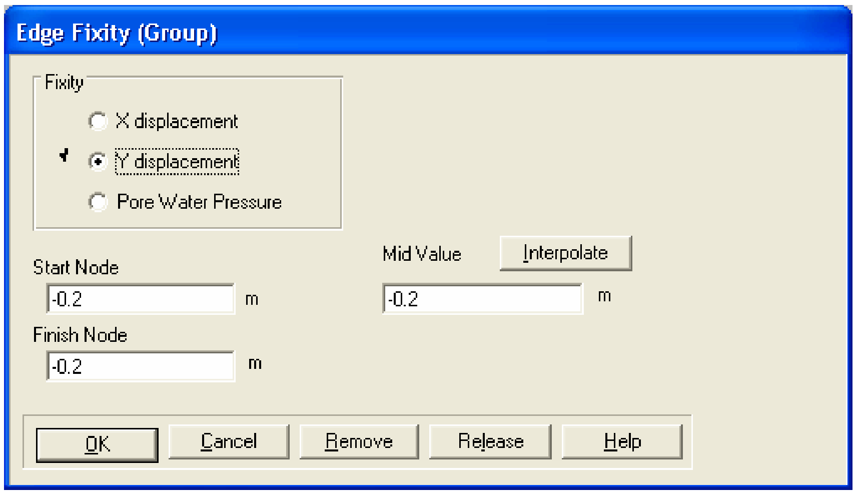


Figure 30 Defining the Applied Displacement due to strip footing

Step 7 Once you have finished, the Graphics Window should look something like Figure 31.

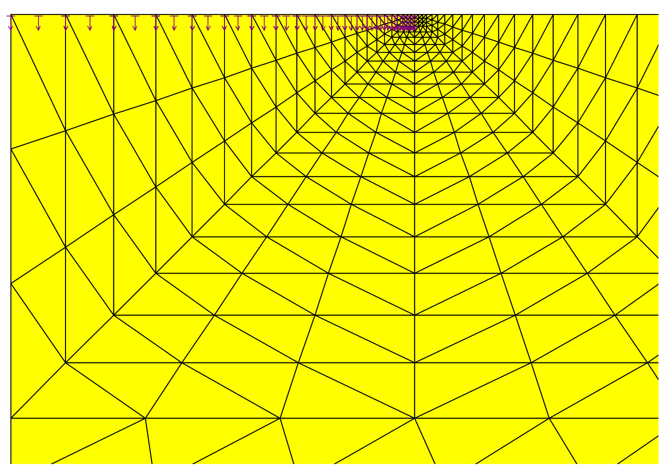


Figure 31 Strip vertical displacement applied at the Top of the Mesh

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.

2.1 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.

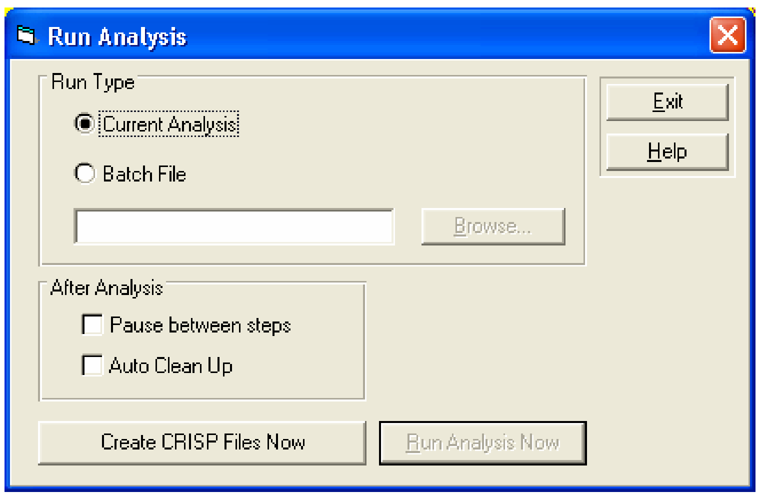


Figure 32 Run Analysis dialogue box

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.

2.2 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 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 3 Once 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.

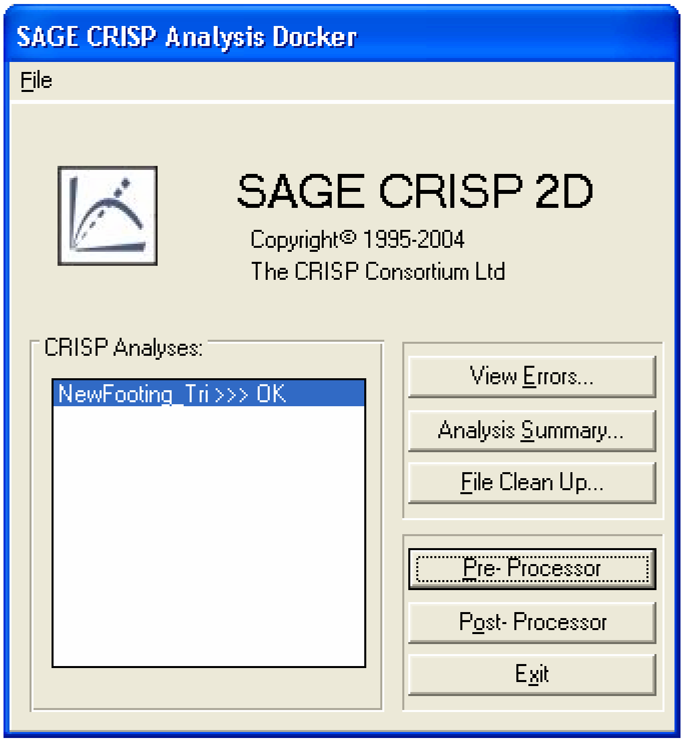


Figure 33 Analysis Docking Module

Step 4 Click on the Post-Processor button. This loads the Project into the SAGE CRISP Post-Processor.

POST PROCESSING

In this section you will check the load displacement response:

- Displacement Plots
- Instance Graphs
- Duration Graphs

3.1 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 SAGE CRISP 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.

3.2 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.

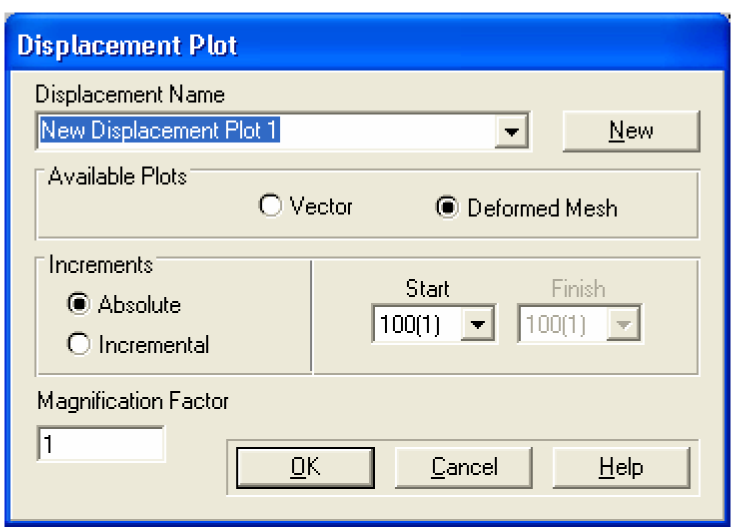


Figure 34 Setting Up A Displaced Plot

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.

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 35.

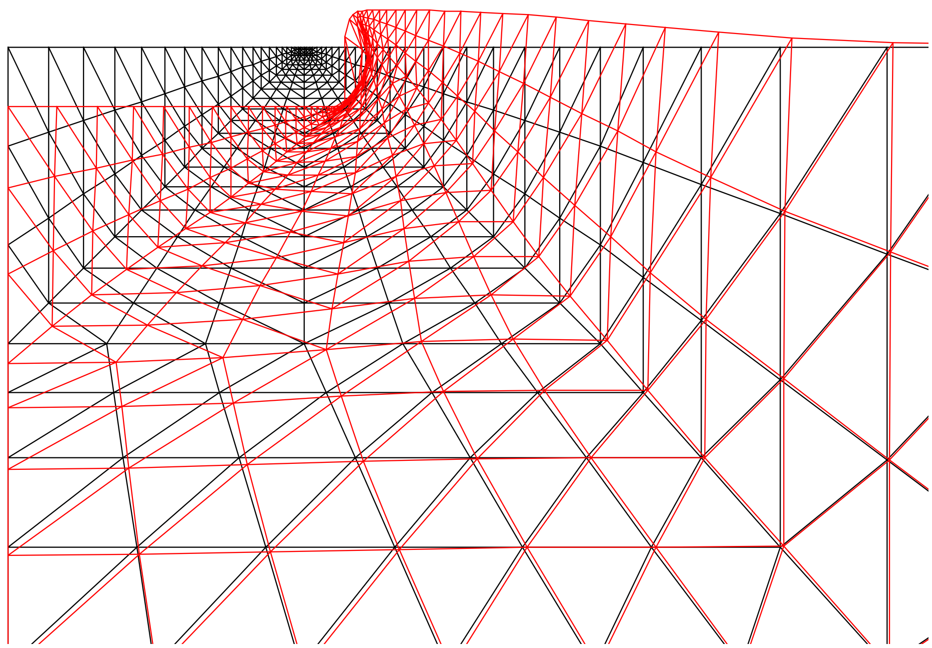


Figure 35 Deformed Mesh

3.3 Contour Plots

You can do contour plots for the deviatoric stress or the shear stress. Select Contour Plots... from the Plots menu and select Deviatoric stress contour Q

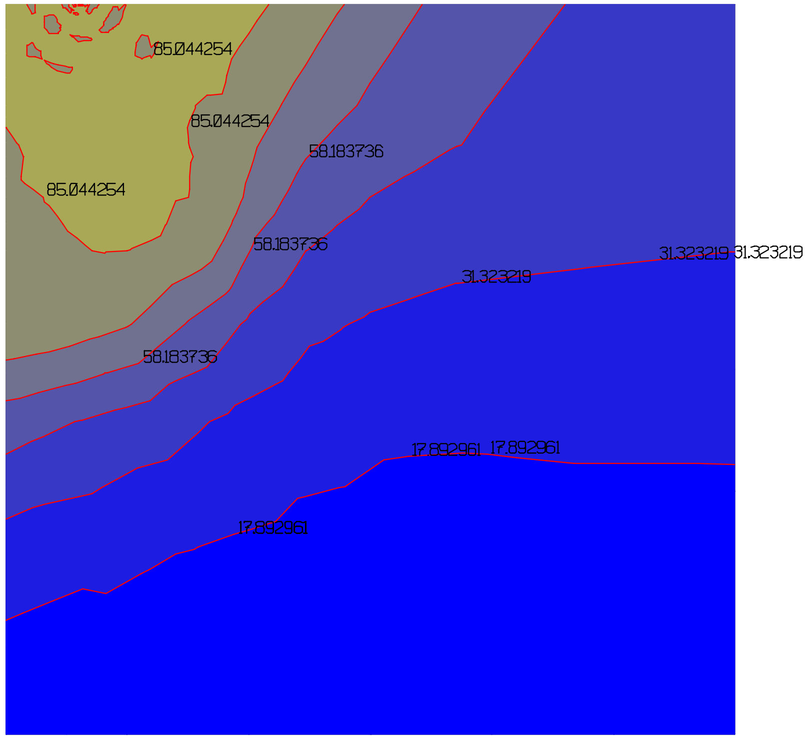


Figure 36 Contour plot of Deviatoric Stresses

3.4 Load Displacement plot from output text file

The applied displacements have been given to the edges forming the interface with the footing. These displacements are applied to the edge of each element. Each edge contains three nodes, two on the ends and one mid-side. The post processor would not show the reactions of mid-side nodes directly. But it is possible to obtain these from the _REC.TXT file produced at the end of the analysis. This file is produced when the parameter IRAC is set to equal to 3 in record C2 of the MPD file. This is set to three by SAGE-CRISP pre-processor by default. The reactions text file is only produced when an analysis contains a load block with specified displacement as we have in our example. The total reaction is given at the end of each increment and it is the sum of all the reactions of the nodes with specified (applied) displacements, including the mid-side nodes.

To plot the displacement, reaction response, we start MS Excel, open existing file and click on .txt for the File Type, then point to our reactions file which should have the extension _REC.TXT

Plot the columns of Displacement_y against the column of Sum_of_y reactions using an x-y graph in Excel.

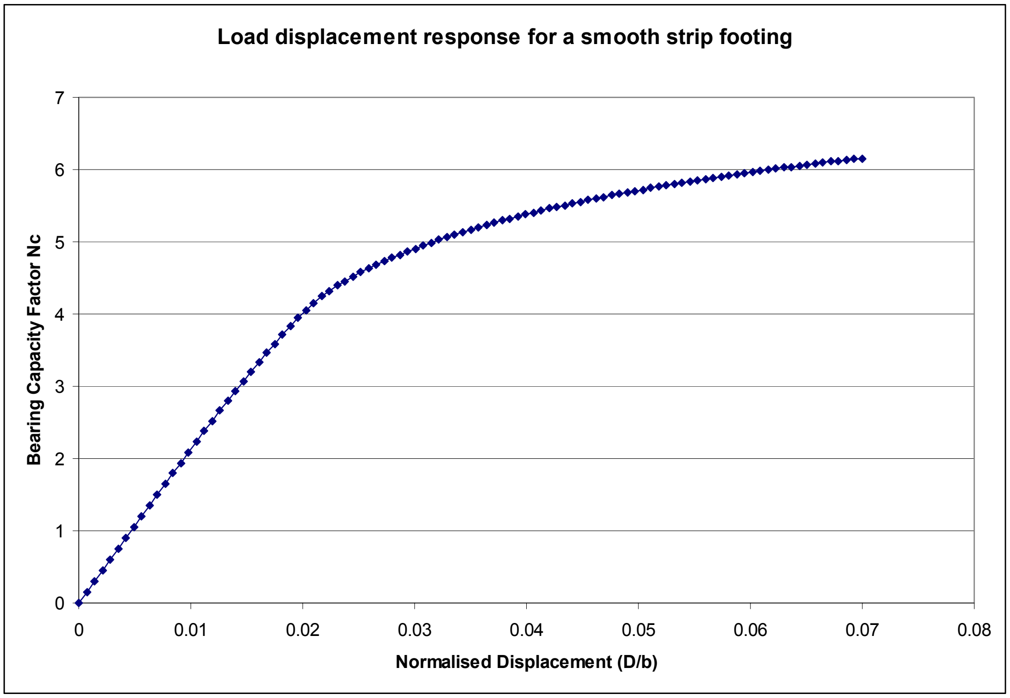


Figure 37 Graph of bearing capacity factor against normalized displacement (disp / footing width)

SUMMARY

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

- Autosave
- Units
- The Structured Mesh Generator
- The In Situ Stress converter
- Displacement plots
- Contour plots
- Load displacement response using output text file

THINGS TO TRY NEXT

You can re-run the analysis for a circular footing by switching to axi-symmetric mesh in the Project Setup menu. You can also generate a mesh with Linear Strain Quadrilateral elements to check the performance of quads against triangles. Generally, triangular elements perform better in collapse analyses, especially with axi-symmetric (circular footing) problems