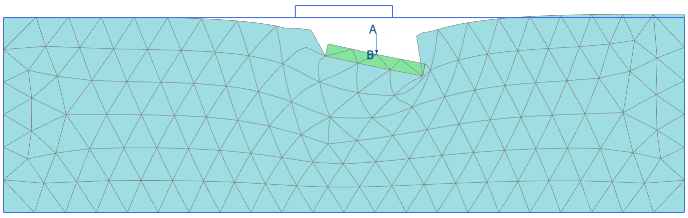
# ELASTOPLASTIC ANALYSIS OF A FOOTING



# INTRODUCTION

One of the simplest forms of a foundation is the shallow foundation. In this exercise we will model such a shallow foundation with a width of 2 meters and a length that is sufficiently long in order to assume the model to be a plane strain model. The foundation is put on top of a 4m thick clay layer. The clay layer has a saturated weight of 18 kN/m$^3$ and an angle of internal friction of 20°.

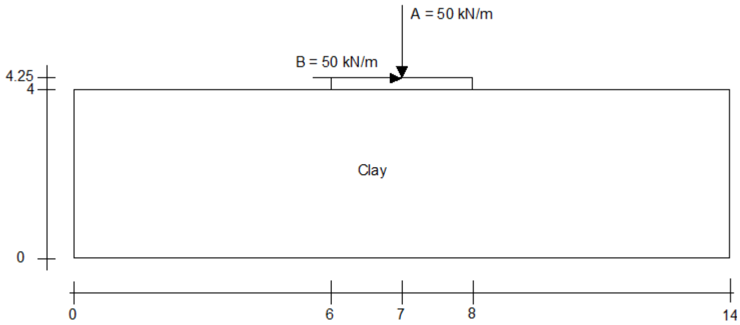



Figure 1: Geometry of the shallow foundation.

The foundation carries a small building that is being modelled with a vertical point force. Additionally a horizontal point force is introduced in order to simulate any horizontal loads acting on the building, for instance wind loads. Taking into account that in future additional floors may be added to the building the maximum vertical load (failure load) is assessed. For the determination of the failure load of a strip footing analytical solutions are available from for instance Vesic, Brinch Hansen and Meyerhof:

$$\frac{Q_f}{B} = c * N_c + \frac{1}{2}\gamma' B * N_\gamma$$
$$N_q = e^{\pi \tan\varphi'} \tan^2(45 + \tfrac{1}{2}\varphi')$$
$$N_c = (N_q - 1)\cot\varphi'$$
$$N_\gamma = \begin{cases} 2(N_q+1)\tan\varphi' & (Vesic) \\ 1.5(N_q-1)\tan\varphi' & (Brinch\,Hansen) \\ (N_q-1)\tan(1.4\,\varphi') & (Meyerhof) \end{cases}$$

This leads to a failure load of 117 kN/$\mathrm{m}^2$ (Vesic), 98 kN/$\mathrm{m}^2$ (Brinch Hansen) or 97 kN/$\mathrm{m}^2$ (Meyerhof) respectively.

# SCHEME OF OPERATIONS

This exercise illustrates the basic idea of a finite element deformation analysis. In order to keep the problem as simple as possible, only elastic perfectly-plastic behaviour is considered. Besides the procedure to generate the finite element mesh, attention is paid to the input of boundary conditions, material properties, the actual calculation and inspection of some output results.

## Aims

- Geometry input
- Initial stresses and parameters
- Calculation of vertical load representing the building weight
- Calculation of vertical and horizontal load representing building weight and wind force
- Calculation of vertical failure load.

### A) Geometry input

- General settings
- Input of geometry lines
- Input of boundary conditions
- Input of material properties
- Mesh generation

### B) Calculations

- Initial pore pressures and stresses
- Construct footing
- Apply vertical force
- Apply horizontal force
- Increase vertical force until failure occurs

### C) Inspect output

# GEOMETRY INPUT

Start PLAXIS by double-clicking the icon of the PLAXIS Input program. The *Quick select* dialog box will appear in which you can select to start an new project or open an existing one. Choose *Start a new project* (see Figure 2). Now the *Project properties* window appears, consisting of the two tabsheets *Project* and *Model* (see Figure 3 and Figure 4).

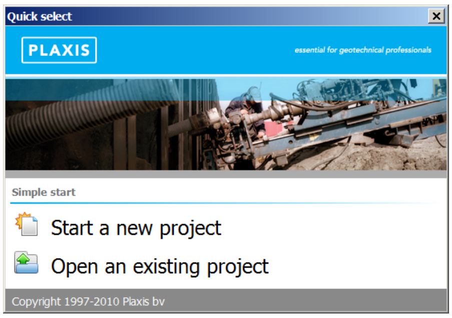



Figure 2: Quick select dialog

## Project properties

The first step in every analysis is to set the basic parameters of the finite element model. This is done in the *Project properties* window. These settings include the description of the problem, the type of analysis, the basic type of elements, the basic units and the size of the drawing area.

### The Project tabsheet

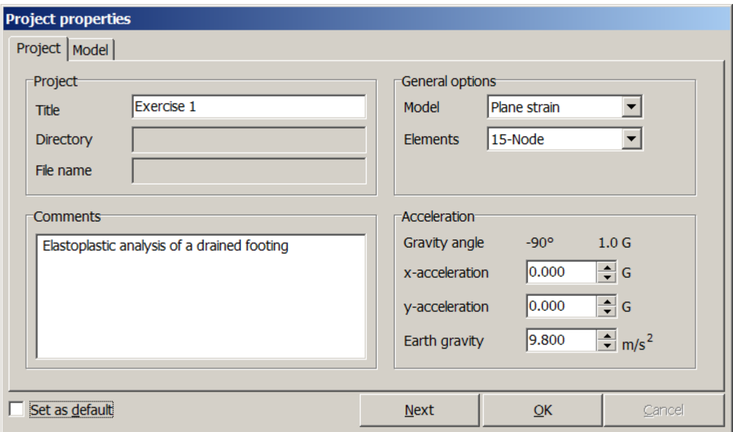



Figure 3: Project tabsheet of the Project Properties window

In order to enter the proper settings for the footing project, follow these steps:

- In the *Project* tabsheet, enter “Exercise 1” in the *Title* box and type “Elasto-plastic analysis of drained footing” or any other text in the *Comments* box.
- In the *General options* box the type of the analysis (*Model*) and the basic element type *(Elements*) are specified. As this exercise concerns a strip footing, choose *Plane strain* from the *Model* combo box. Select *15-node* from the *Elements* combo box.
- The *Acceleration* box indicates a fixed gravity angle of -90°, which is in the vertical direction (downward). Independent acceleration components may be entered for pseudo-dynamic analyses. Leave these values zero and click on the *Next* button below the tabsheets or click on the *Model* tabsheet.

### The Model tabsheet

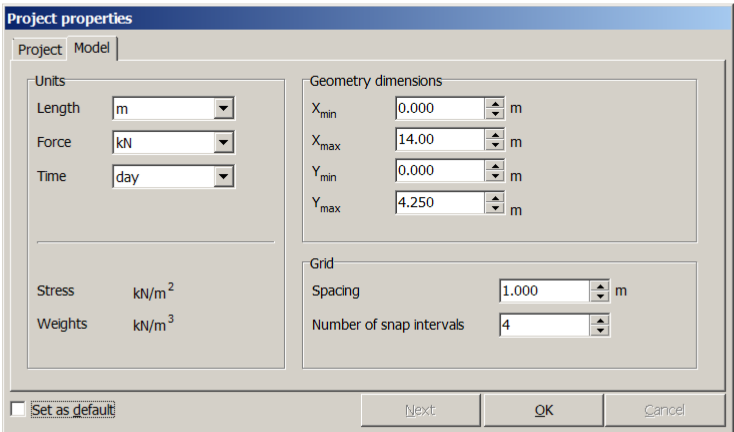



Figure 4: Model tabsheet of the Project properties window

- In the *Model* tabsheet, keep the default units in the *Units* box (Length = **m**; Force = **kN**; Time = **day**).
- In the *Geometry dimensions* box the size of the considered geometry must be entered. The values entered here determine the size of the draw area in the Input window. PLAXIS will automatically add a small margin so that the geometry will fit well within the draw area. Enter $X_{min}$=0.00, $X_{max}$=14.00, $Y_{min}$=0.00 and $Y_{max}$=4.25.
- The *Grid* box contains values to set the grid spacing. The grid provides a matrix of dots on the screen that can be used as reference points. It may also be used for snapping to regularly spaced points during the creation of the geometry. The distance of the dots is determined by the *Spacing* value. The spacing of snapping points can further be divided into smaller intervals by the *Number of snap intervals* value. Enter 1.0 for the spacing and 4 for the intervals.
- Click on the *Ok* button to confirm the settings. Now the draw area appears in which the geometry model can be drawn.

| | |
|---|---|
| **Hint:** | In the case of a mistake or for any other reason that the project properties should be changed, you can access the *Project properties* window by selecting the *Project properties* option from the *File* menu. |

## Creating the geometry

Once setting the project properties have been completed, the draw area appears with an indication of the origin and direction of the system of axes.

The cursor is automatically switched in the Geometry line drawing mode. If not, the user can change the drawing mode to Geometry line by clicking the geometry line button .

In order to construct the contour of the proposed geometry as shown in Figure 5, follow these steps. (Use Figure 5 for orientation, it represents the completed geometry).

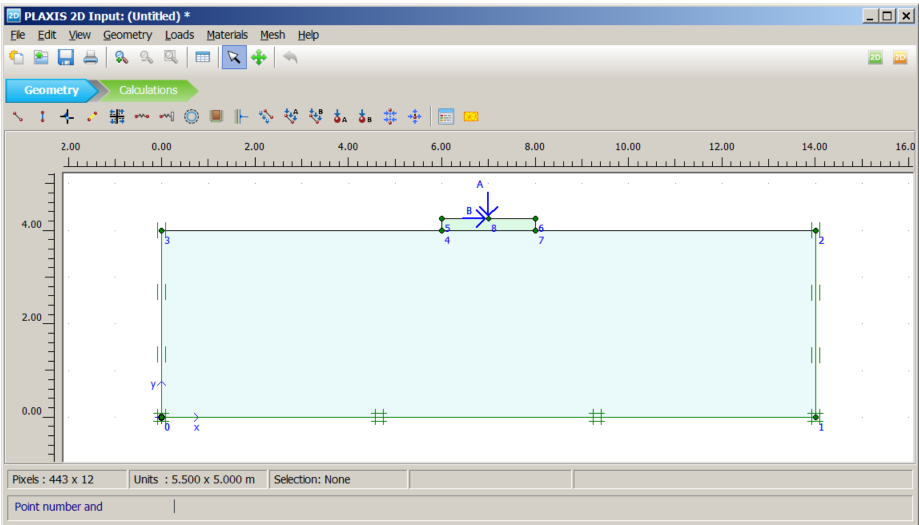



Figure 5: Geometry model

### Create sub-soil

- Position the cursor (now appearing as a pen) at the origin (point 0) of the axes (0.0; 0.0). Click the left mouse button once to start the geometry contour.
- Move along the x-axis to (14.0; 0.0). Click the left mouse button to generate the second point (number 1). At the same time the first geometry line is created from point 0 to point 1.
- Move upward to point 2 (14.0; 4.0) and click again.
- Move to the left to point 3 (0.0; 4.0) and click again.
- Finally, move back to the origin (0.0; 0.0) and click the left mouse button again. Since the latter point already exists, no new point is created, but only an additional geometry line is created from point 3 to point 0. PLAXIS will also automatically detect a cluster (area that is fully enclosed by geometry lines) and will give it a light colour.

- Click the right mouse button to stop drawing.

This action created the sub-soil cluster. The next step is to introduce the footing.

**Create footing**

- Position the cursor at point 4, (6.0, 4.0) and click the left mouse button once.
- Move vertical to point 5, (6.0; 4.25). Click the left mouse button to generate a vertical line.
- Move horizontal to point 6, (8.0; 4.25). Click the left mouse button to generate a horizontal line.
- Generate a second cluster by clicking the left mouse button on coordinate (8.0; 4.0).
- Click the right mouse button to stop drawing.

This action created the footing.

The proposed geometry does not include plates, hinges, geogrids, interfaces, anchors or tunnels. Hence, you can skip the corresponding buttons in the second toolbar.

| | |
|---|---|
| **Hints:** | Mispositioned points and lines can be modified or deleted by first choosing the *Selection* button  from the toolbar. To move a point of line, select the point or the line and drag it to the desired position. To delete a point or a line, select the point or the line and press the *Delete* key on the keyboard. |
| > | Undesired drawing operations can be restored by pressing the *Undo* button  from the toolbar or by selecting the *Undo* option from the *Edit* menu or by pressing <Ctrl><Z> on the keyboard. |

| | |
|---|---|
| **Hint:** | The full geometry model has to be completed before a finite element mesh can be generated. This means that boundary conditions and model parameters must be entered and applied to the geometry model first. |

| | |
|---|---|
| **Hint:** | During the input of geometry lines by mouse, holding down the *Shift* key will assist the user to create perfect horizontal and vertical lines. |

## Input of boundary conditions

Boundary conditions can be found in the second block of the toolbar and in the *Loads* menu. For deformation problems two types of boundary conditions exist: Prescribed displacements and prescribed forces (loads). In principle, all boundaries must have one boundary condition in each direction. That is to say, when no explicit boundary condition is given to a certain boundary (a free boundary), the so-called 'natural condition' applies, which is a prescribed force equal to zero and a free displacement. In order to avoid the situation where the displacements of the geometry are 'undetermined', some points of the geometry must have prescribed displacements. The simplest form of a prescribed displacement is a fixity (zero displacement), but non-zero prescribed displacements may also be given.

To create the boundary conditions for this exercise, follow the steps below.

### Prescribed displacements

Click on the *Standard fixities* button on the toolbar or choose the *Standard fixities* option from the *Loads* menu to set the standard boundary conditions. As a result PLAXIS will automatically generate a full fixity at the base of the geometry and roller conditions at the vertical sides ($u_x$=0; $u_y$=free). A fixity in a certain direction is presented as two parallel lines perpendicular to the fixed direction. Hence, the rollers appear as two vertical parallel lines and the full fixity appears as cross-hatched lines.

| | |
|---|---|
| **Hint:** | The *Standard fixities* option is suitable for most geotechnical applications. It is a fast and convenient way to input standard boundary conditions. |

### Vertical load

Click on the *Point load - load system A* button on the toolbar or choose the *Point load - static load system A* option from the *Loads* menu to enter another point force. Click on the coordinate (7.0, 4.25) to enter a point force. As a result PLAXIS will automatically generate a vertical point force on the indicated point with a unity force (f = 1).

### Horizontal load (see also next step "Changing direction .....")

Click on the *Point load - load system B* button on the toolbar or choose the *Point load - static load system B* option from the *Loads menu* to enter a point force. Click on the coordinate (7.0, 4.25) to enter a point force. As a result PLAXIS will automatically generate a vertical point force on the indicated point. As a horizontal force is needed, the direction of load B needs to be changed.

### Changing direction and magnitude of loads

Choose the *Selection* button from the toolbar. Double click on the geometry point 8 with coordinate (7.0, 4.25) which will display a box as indicated in Figure 6. Select *Point Load - load system B*, click *OK* and enter 1.0 as x-value and 0.0 as y-value. These values are the input load of point force B. Click *OK* to close the window.

## Input of material properties

In order to simulate the behaviour of the soil, a proper soil model and corresponding parameters must be applied to the geometry. In PLAXIS, soil properties are collected in material data sets and the various data sets are stored in a material database. From the database, a data set can be assigned to one or more clusters. For structures (like walls, plates, anchors, geogrids, etc.) the system is similar, but obviously different types of structures have different parameters and thus different types of data sets. PLAXIS distinguishes between material data sets for *Soil*

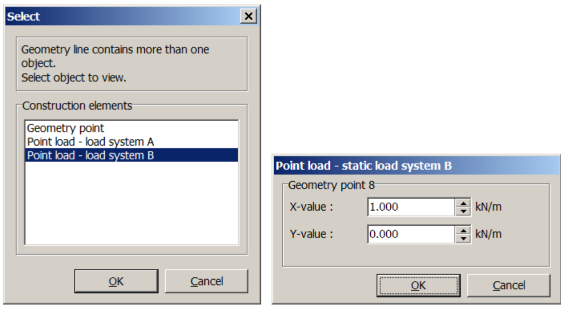



Figure 6: Select window and Point load window

*& Interfaces*, *Plates*, *Anchors* and *Geogrids*. The creation of material data sets is generally done after the input of boundary conditions. Before the mesh is generated, all material data sets should have been defined and all clusters and structures must have their appropriate data set.

Table 1: Material properties of the clay layer and the concrete footing.

| Parameter | Symbol | Clay | Concrete | Unit |
|---|---|---|---|---|
| Material model | Model | Mohr-Coulomb | Linear elastic | — |
| Type of behaviour | Type | Drained | Non-porous | — |
| Weight above phreatic level | $\gamma_{unsat}$ | 16.0 | 24.0 | kN/m$^3$ |
| Weight below phreatic level | $\gamma_{sat}$ | 18.0 | — | kN/m$^3$ |
| Young's modulus | $E_{ref}$ | $5.0 \cdot 10^3$ | $2.0 \cdot 10^7$ | kN/m$^2$ |
| Poisson's ratio | $\nu$ | 0.35 | 0.15 | — |
| Cohesion | c | 5.0 | — | kN/m$^2$ |
| Friction angle | $\varphi$ | 20 | — | ° |
| Dilatancy angle | $\psi$ | 0 | — | ° |

The input of material data sets can be selected by means of the *Material Sets* button  on the toolbar or from the options available in the *Materials* menu.

### Create material data sets

To create a material set for the clay layer, follow these steps:

- Select the *Material Sets* button on the toolbar.
- Click on the <New> button at the lower side of the *Material Sets* window. A new dialog box will appear with five tabsheets: *General*, *Parameters, Flow parameters, Interfaces* and *Initial* (see figure 7).
- In the *Material Set* box of the *General* tabsheet, write “Clay” in the *Identification* box.
- Select Mohr-Coulomb from the *Material model* combo box and *Drained* from the *Material type* combo box.

- Enter the proper values for the weights in the *General properties* box according to the material properties listed in table 1
- See also figure 8 and figure 9. In these figures the *Advanced* parameters part has been collapsed.

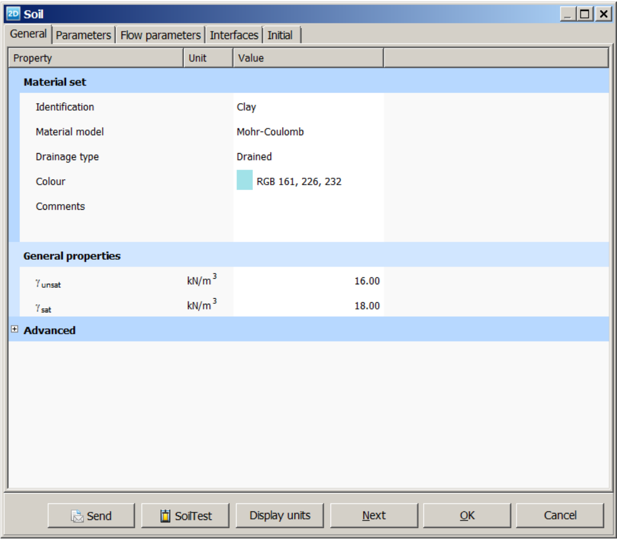



Figure 7: General tabsheet of the soil and interface data set window for Clay

- Click on the *Next* button or click on the *Parameters* tabsheet to proceed with the input of model parameters. The parameters appearing on the *Parameters* tabsheet depend on the selected material model (in this case the Mohr-Coulomb model).
- Enter the model parameters of table 1 in the corresponding edit boxes of the *Parameters* tabsheet. The parameters in the *Alternatives* and *Velocities* group are automatically calculated from the parameters entered earlier.
- Since the geometry model does not include groundwater flow or interfaces, the third and fourth tabsheet can be skipped. Click on the *OK* button to confirm the input of the current material data set.
- Now the created data set will appear in the tree view of the *Material Sets* window.

- For the concrete of the footing repeat the former procedure, but choose a *Linear Elastic* material behaviour and enter the properties for concrete as shown in table 1 (see also figures 9 and 10).

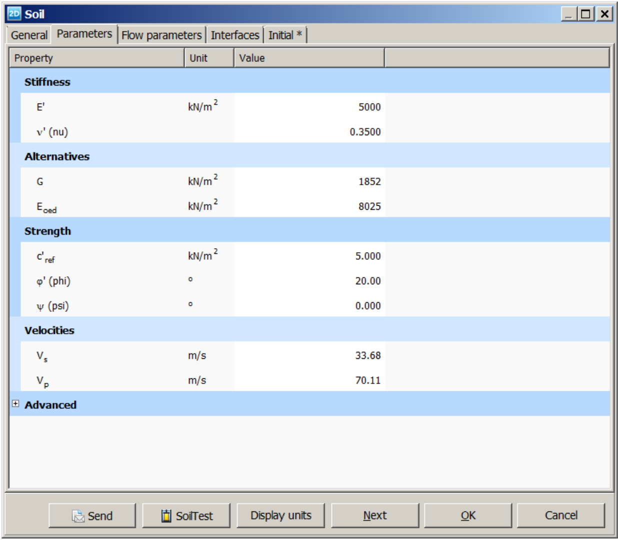


Figure 8: Parameters tabsheet of the soil and interface data set window for Clay

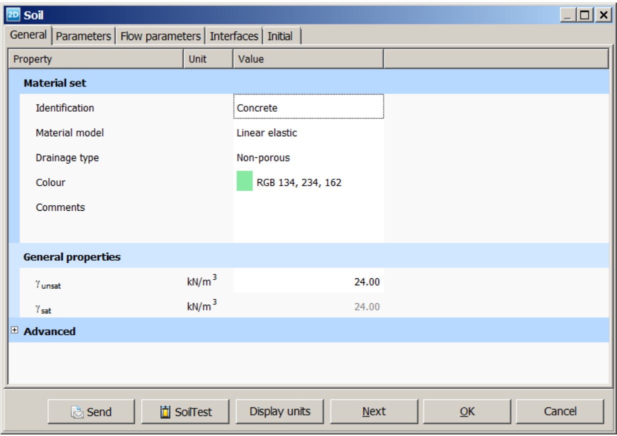


Figure 9: General tabsheet of the soil and interface data set window for Concrete

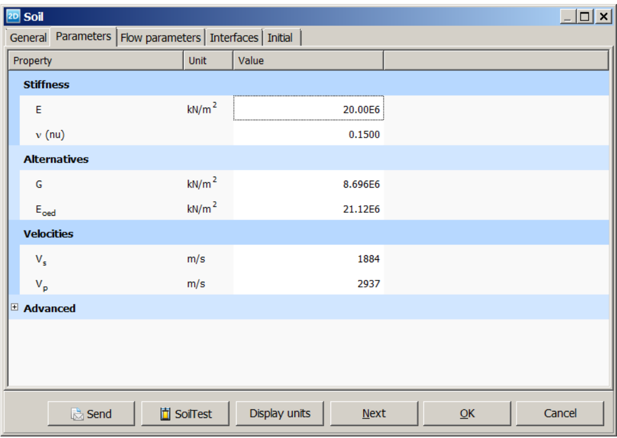


Figure 10: Parameters tabsheet of the soil and interface data set window for Concrete

### Assigning material data sets to soil clusters

- Drag the data set “Clay” from the *Material Sets* window (select it and keep the left mouse button down while moving) to the soil cluster in the draw area and drop it there (release the left mouse button). Notice that the cursor changes shape to indicate whether or not it is possible to drop the data set. When a data set is properly assigned to a cluster, the cluster gets the corresponding colour. Drag the concrete material set to the footing and drop it there.
- Click on the *OK* button in the *Material Sets* window to close the database.

| | |
|---|---|
| **Hint:** | PLAXIS distinguishes between a project database and a global database of material sets. Data sets may be exchanged from one project to another using the global database. In order to copy such an existing data set, click on the *Show global* button of the *Material Sets* window. Drag the appropriate data set (in this case “Clay”) from the tree view of the global database to the project database and drop it there. Now the global data set is available for the current project. Similarly, data sets created in the project database may be dragged and dropped in the global database. |

| | |
|---|---|
| **Hints:** | Existing data sets may be changed by opening the material sets window, selecting the data set to be changed from the tree view and clicking on the *Edit* button. As an alternative, the material sets window can be opened by double clicking a cluster and clicking on the *Change* button behind the *Material set* box in the properties window. A data set can now be assigned to the corresponding cluster by selecting it from the project database tree view and clicking on the *OK* button. |
| > | The program performs a consistency check on the material parameters and will give a warning message in the case of a detected inconsistency in the data |

### Mesh generation

When the geometry model is complete, the finite element model (mesh) can be generated. PLAXIS includes a fully automatic mesh generation procedure, in which the geometry is automatically divided into elements of the basic element type and compatible structural elements, if applicable. The mesh generation takes full account of the position of points and lines in the geometry model, so that the exact position of layers, loads and structures is reflected by the finite element mesh. The generation process is based on a robust triangulation principle that searches for optimised triangles, which results in an unstructured mesh. This may look disorderly, but the numerical performance of such a mesh is usually better than for regular (structured) meshes. In addition to the mesh generation itself, a transformation of input data (properties, boundary conditions, material sets, etc.) from the geometry model (points, lines and clusters) to the finite element mesh (elements, nodes and stress points) is made.

In order to generate the mesh, follow these steps:

- Click on the *Generate mesh* button  in the toolbar or select the *Generate* option from the *Mesh* menu. After the generation of the mesh a new window is opened (PLAXIS Output window) in which the generated mesh is presented (see Figure 11).

- Click on the *Close* button  to return to the geometry input mode.

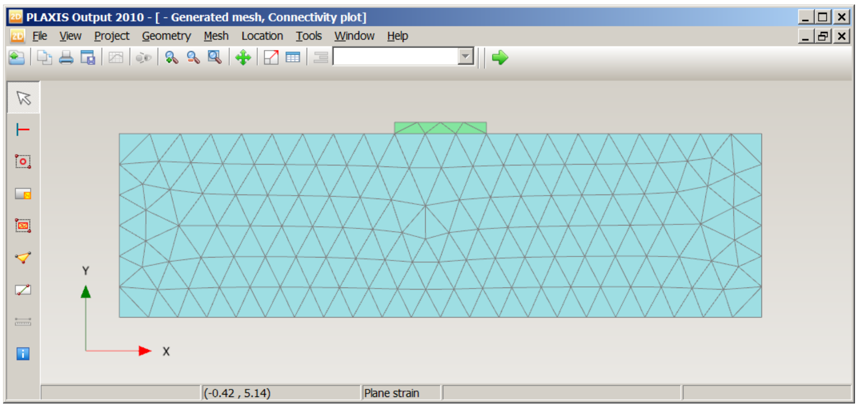



Figure 11: Generated finite element mesh of the geometry around the footing

If necessary, the mesh can be optimised by performing global or local refinements. Mesh refinements are considered in some of the other exercises. Here it is suggested to accept the current finite element mesh.

| | |
|---|---|
| **Hints:** | By default, the *Global coarseness* of the mesh is set to $Medium$, which is adequate as a first approach in most cases. The *Global coarseness* setting can be changed in the $Mesh$ menu. In addition, there are options available to refine the mesh globally or locally. |
| > | At this stage of input it is still possible to modify parts of the geometry or to add geometry objects. In that case, obviously, the finite element mesh has to be regenerated. |

Press the close button  to close the output program and return to PLAXIS input.

Creating the input for this project now finished. Press the green *Calculation* button on the toolbar to continue with the definition of the calculation phases.

# CALCULATION

After the finite element model has been created, the calculation phases need to be defined. This analysis consists of four phases. In the initial phase the initial pore pressures and stresses are generated, in the first phase the footing is constructed, during the second phase the vertical load is applied and in the third phase the horizontal load is applied.

When starting the PLAXIS Calculation program the *Calculation mode* window appears. In this window the user can choose how he wants PLAXIS to handle pore pressures during the calculation. This is important when calculating with undrained behaviour and/or groundwater flow. In this first exercise this is not important and so the default setting of *Classical mode* is chosen. Press <OK> to close the *Calculation mode* window. PLAXIS now shows the *General* tabsheet of the initial phase (see Figure 12).

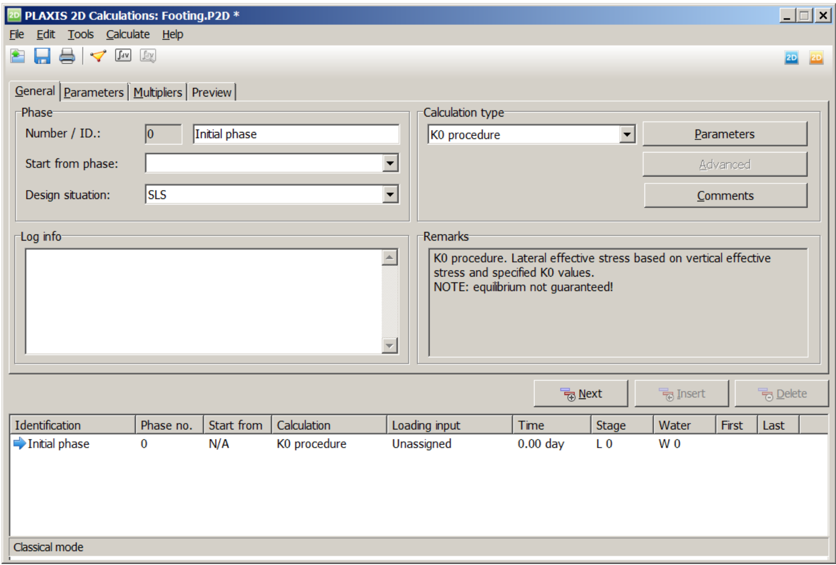
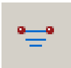



Figure 12: General tabsheet of the initial calculation phase

## Initial phase (generation of initial conditions)

Before starting the construction of the footing the initial conditions must be generated. In general, the initial conditions comprise the initial groundwater conditions, the initial geometry configuration and the initial effective stress state. The clay layer in the current footing project is fully saturated with water, so groundwater conditions must be specified. On the other hand, the situation requires the generation of initial effective stresses. As we want to include the footing construction in the simulation process, the footing should not be present in the initial situation (prior to construction). In PLAXIS it is possible to switch off clusters in order to calculate correct initial effective stresses. The initial stresses in this example case are generated using the $K_0$-procedure. The initial conditions are entered in separate modes of the Input program. In order to generate the initial conditions properly, follow these steps:

- In the phase list select the initial phase

- Make sure the *Calculation type* is set to $K_0$-procedure on the *General* tabsheet. This is the default setting.
- Go to the *Parameters* tabsheet by clicking the *Parameters* button or by directly selecting the tabsheet.
- On the *Parameters* tabsheet press the *Define* button located in the *Loading input* box. This will start a window presenting the problem in *Staged construction* mode. In *Staged construction* mode it is possible to switch on and off various parts of the geometry, change loads, apply strains etc.
- In the initial condition of this exercise, that is the situation before we start constructing our project, the footing is not present. Therefore the footing has to be deactivated. In order to do so, click on the area that represents the footing so that it will change color from the material set color to white. The footing is now disabled.
- Click on *Water conditions* in the button bar in order to move to the *Water conditions* mode of the program.
- Select the *Phreatic level* button .
- Position the cursor (appearing as a pen) at coordinate (0.0, 4.0) and click the left mouse button to start the phreatic level.
- Move along the x-axis to position (14.0, 4.0). Click the left mouse button to enter the second point of the phreatic level.
- Click the right mouse button to stop drawing.
- Press the *Water pressures* button  to view the pore pressures.

The pore pressures are generated from the specified phreatic level and the water weight. Directly after the generation, a PLAXIS Output window is opened, showing the pore pressure as presented in Figure 13. The colors indicate the magnitude of pore pressure. The pore pressures vary hydrostatically, ranging from 0 kN/m$^2$ at the top to -40 kN/m$^2$ at the bottom.

- Close the output program in order to return to the input program.
- Click on *Update* in order to save the changes made and return to the PLAXIS Calculations program. This completes the definition of the initial conditions.

| | |
|---|---|
| **Hints:** | For the generation of initial stresses based on the $K_0$procedure it is necessary to specify the coefficient of lateral earth pressure, $K_0$. This $K_0$value is defined per material set and therefore has to be set when entering material set data. If the $K_0$value is not explicitly set PLAXIS uses a value according to Jaky's formula ($K_0$ = 1-sin($\varphi$)). |
| **>** | The $K_0$ procedure may only be used for horizontally layered geometries with a horizontal ground surface and, if applicable, a horizontal phreatic level. |

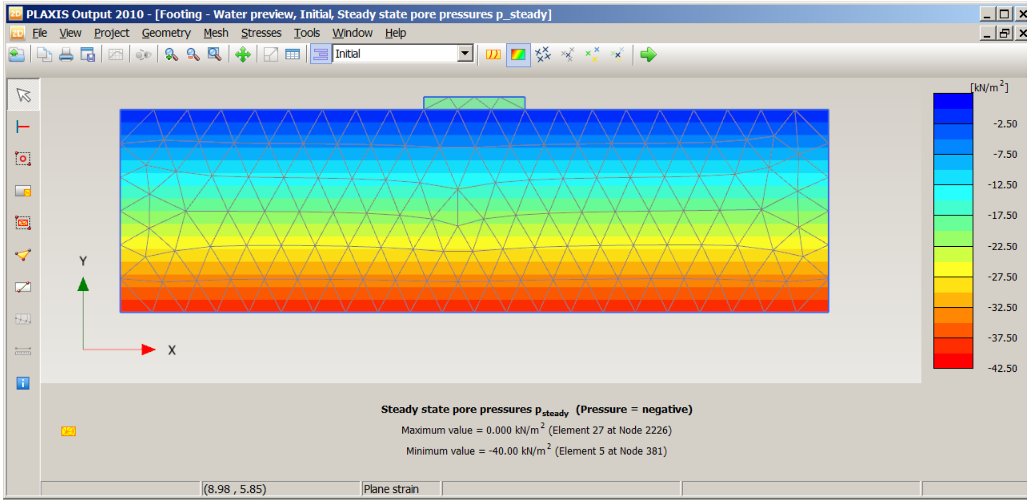
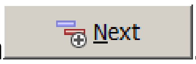



Figure 13: Initial pore pressures

## First calculation phase (construction of footing)

- Click on the *Next* button 

. This will introduce a new calculation phase and present the corresponding tabsheets for the first calculation stage. Enter a suitable name in the *Number/ ID* box (e.g. 'Construction of footing').
- Select the second tabsheet called *Parameters*. On this sheet *Staged construction* is selected by default in the *Loading input* combo box. Click the *Define* button. This will open the window presenting the problem in *Staged construction* mode.
- Click on the cluster that represents the strip footing, in order to switch on the footing (original colour should reappear).
- Click on *Update* to conclude the definition of the first calculation phase. Updating will automatically present the calculation window.

## Second calculation phase (apply vertical load)

- Click on the *Next* button . This will introduce a new calculation phase and present the corresponding tabsheets for the second calculation stage. Enter a suitable name in the *Number/ ID* box (e.g. 'apply vertical load').
- Select the *Parameters* tabsheet. On this tabsheet accept the selection *Staged construction* in the *Loading input* combo box. Click on the *Define* button. This will open the window presenting the problem in *Staged construction* mode.
- Click on the point forces in the middle of the footing, a *Select items* window comes up. Select the *Point load - Load System A* to activate point load A and press the *Change* button to change the load value. Change the y-value to -50 kN/m and press the *Ok* button.

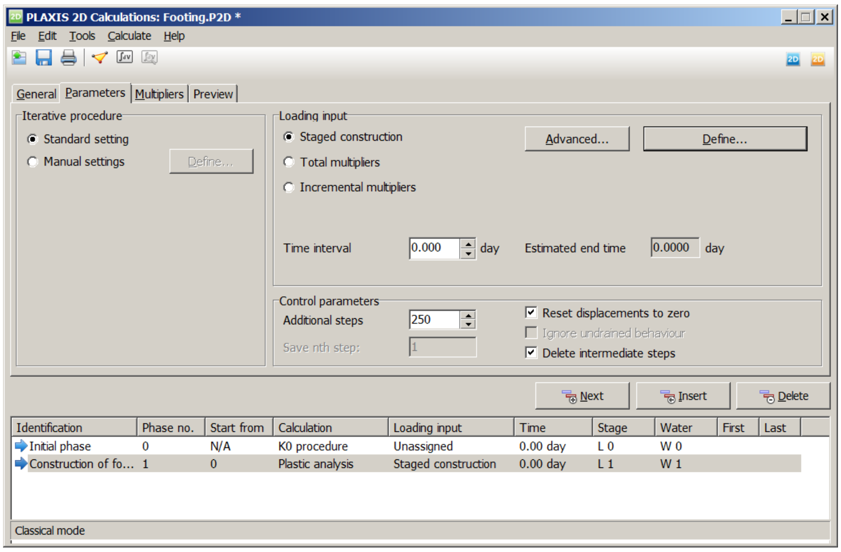



Figure 14: Parameters tabsheet of the first calculation phase

- The point load A is now active (blue) and has a load value of 50 kN/m.
- Press *Update.*

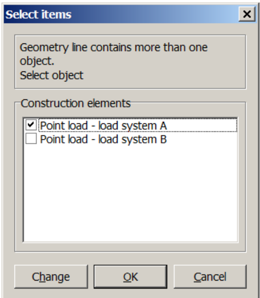



Figure 15: Select items window

## Third calculation phase (add horizontal load)

- Click on the *Next* button to add another phase. This will present the tabsheets for the third calculation stage. Enter a suitable name in the *Number/ID* box (e.g. 'apply horizontal load').
- Select the second tabsheet called *Parameters*. On this sheet accept the selection *Staged construction* in the *Loading input* combo box. Click on the *Define* button.
- Click on the point forces in the middle of the footing, select the *Point load - load system B* to activate point load B and press the *Change* button to change the load value. Change the load x-value to 20 kN/m2 and press the *Ok* button.
- Press the *Ok* button to closed the Select items window.
- Press *Update.*

## Fourth calculation phase (vertical load to failure)

- Click on the *Next* button . This will present the tabsheets for the fourth calculation stage. Enter a suitable name in the *Number/ID* box (e.g. 'vertical load – failure').
- Directly below the *Number/ID* box select from the *Start from phase* dropdown list the second calculation phase. By selecting this the 4th phase will be a continuation of the 2nd phase, hence we will continue to apply the vertical load without having the additional horizontal load that was applied in phase 3.
- Select the second tabsheet called *Parameters*. On this sheet choose the selection *Total multipliers* in the *Loading input* group box. Select the third tabsheet called *Multipliers* by either clicking on the *Define* button or directly selecting the tabsheet.
- Enter a ΣMloadA of 10. In this way the working force is increased to a maximum load of 10 x 50 = 500 kN/m.

In PLAXIS two methods exist to increase an active load. The magnitude of the activated load is the input load multiplied by the total load multiplier. Hence, in this excersise ΣMloadA x (input load of point load A) = Active load A
The value of the input load A can be changed using *Staged construction* as *Loading input* while using *Total multipliers* as *Loading input* may be used to change the load multiplier.

## Define load displacement points

After the calculation it is possible to create load-displacement curves. These can be used to inspect the behaviour in a node during the calculation steps. In order to create load-displacement curves it is first necessary to indicate for which node(s) the displacements should be traced.

- Click on the *Select points for curves* button  in the toolbar. This will result in a plot of the mesh, showing all generated nodes. Click on the node, located in the centre directly underneath the footing. For a correct selection of this node it may be necessary to use the zoom option  . After selection of the node it will be indicated as point A. Press the *Update* button  to proceed to calculations.

## Start the calculation

After definition of the last calculation phase, the calculation process is started by clicking the *Calculation* button . This will start the calculation. During the calculation a calculation window appears showing the status and some parameters of the current calculation phase.

# INSPECT OUTPUT

After each successful execution of a calculation phase PLAXIS will indicate the phase with a green check mark ( ). This indicates a successful calculation phase. If during execution either failure or an error occurs, PLAXIS marks the stage with a red cross ( ).

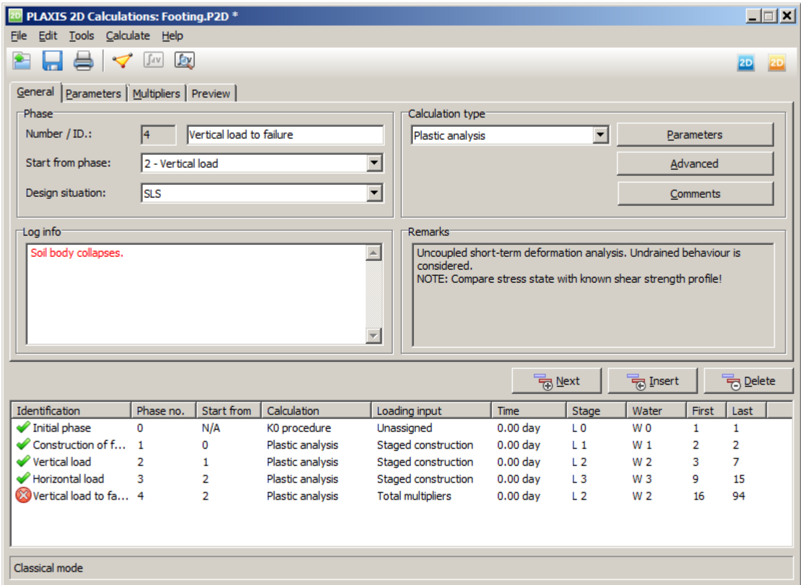



Figure 16: Calculation window with all phases calculated

- While phase 3 is highlighted, press the *View calculation results* button that will start the output program, showing the deformed mesh for the situation with both horizontal and vertical load applied, as presented in figure 17.

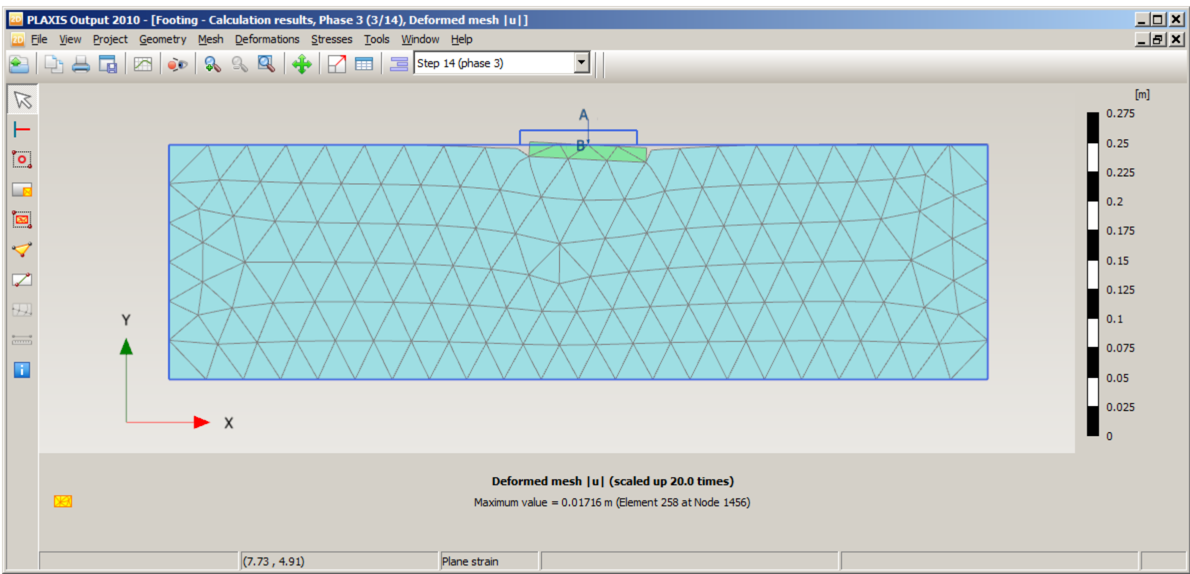



Figure 17: Deformed mesh at the end of phase 3

- Check the various types of output, such as the deformed mesh, displacement contours, effective (principal) stresses etc. These can be found from the *Deformations* and *Stresses* menus.

- Still in the Output program, select from the dropdown list at the right of the toolbar the output step belonging to phase 4.

- From the Displacements menu in the Output program now select *Incremental displacements* and then the option $|\Delta u|$. Display the incremental displacements as contours or shadings. The plot clearly shows a failure mechanism (see Figure 18).

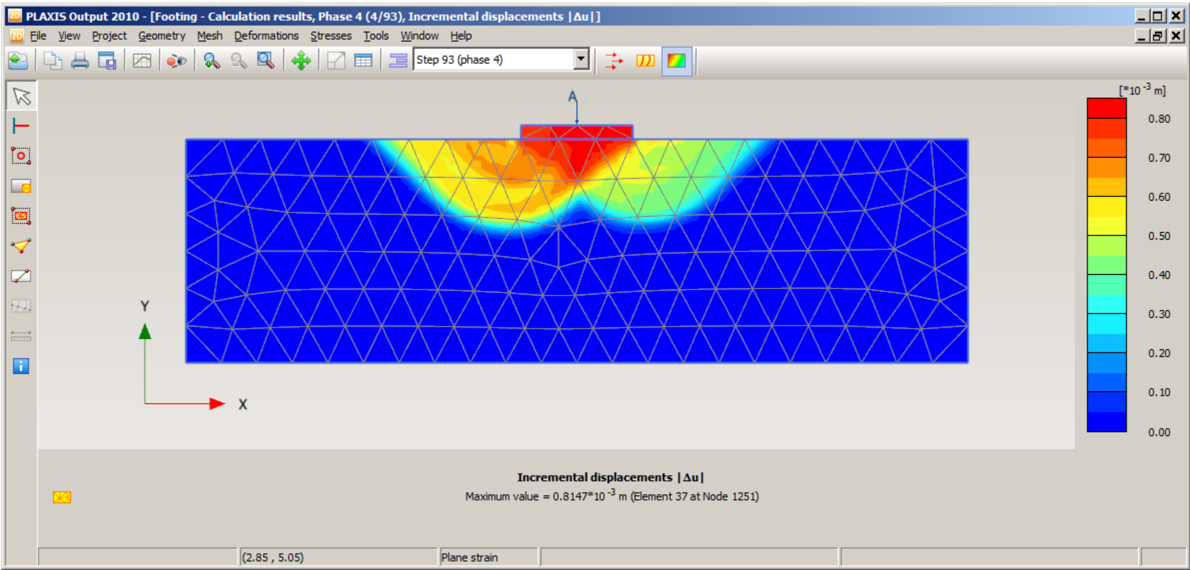



Figure 18: Shadings of displacement increments after phase 4

## Load displacement curves

- In the Output program, select the *Curves manager* from the *Tools* menu. The *Curves manager* has 2 tabsheets, one for the curves defined in this project (currently none) and one for the points selected to make load-displacement curves (currently 1 node that was pre-selected, that is before the calculation).

- In the *Curves manager* select the button *New* to define a new curve. Now the *Curve generation* window opens.

- On the x-axis we want to plot the settlement of our chosen point in the middle of the footing. In the x-axis box choose point *A* from the dropdown list and then below in *Deformations* and then *Total displacements* choose $|u|$.

- On the y-axis we want to plot the force applied on the footing, which is a global value not connected to a specific node or stress point. In y-axis box choose *Project* from the dropdown list to indicate we want to plot a global value, and then in *Multipliers* choose $\Sigma$*MLoadA*.

- Figure 19 shows the *Curve generation* window after applying the steps mentioned.
- Press *OK* to show the resulting curve. See also figure 20.

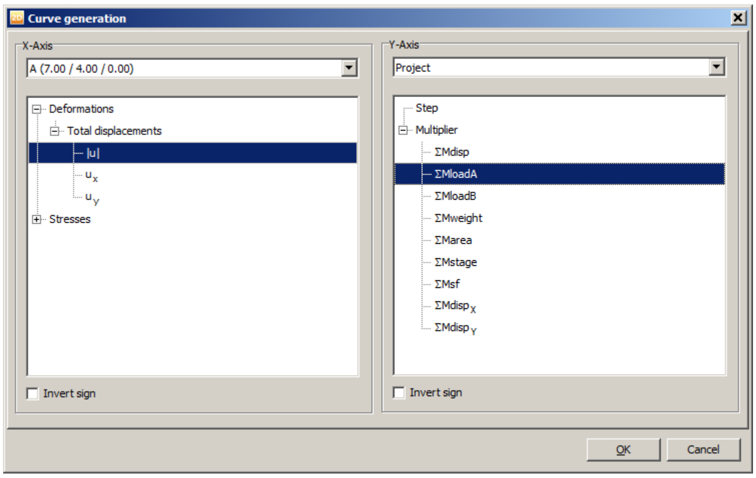



Figure 19: Curves generation window

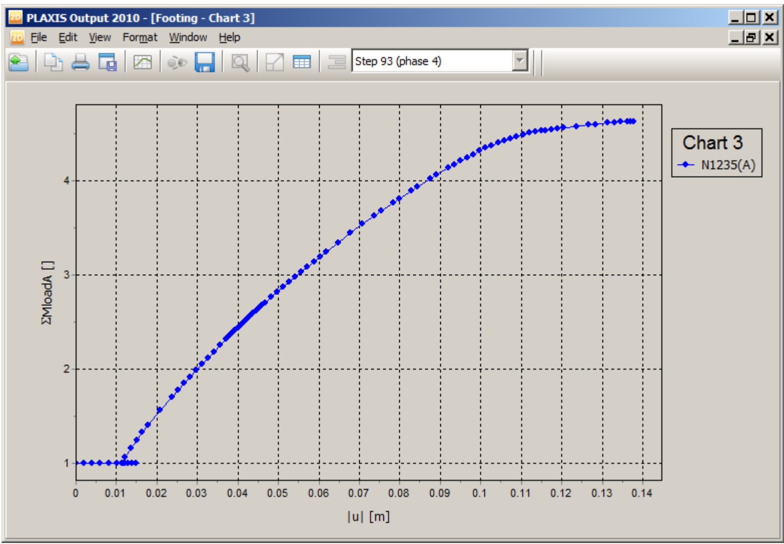



Figure 20: Load displacement curve for the footing

The input value of point load A is 50 kN/m and the load multiplier ΣMloadA reaches approximately 4.6. Therefore the failure load is equal to 50 kN/m x 4.6 = 230 kN/m. You can inspect the load multiplier by moving the mouse cursor over the plotted line. A tooltip box will show up with the data of the current location.

# RESULTS DRAINED BEHAVIOUR

In addition to the mesh used in this exercise calculations were performed using a very coarse mesh with a local refinement at the bottom of the footing and a very fine mesh. Fine meshes will normally give more accurate results than coarse meshes. In stead of refining the whole mesh, it is generally better to refine the most important parts of the mesh, in order to reduce computing time. Here we see that the differences are small (when considering 15-noded elements), which means that we are close to the exact solution. The accuracy of the 15-noded element is superior to the 6-noded element, especially for the calculation of failure loads.

| | |
|---|---|
| **Hint:** | In plane strain calculations, but even more significant in axi-symmetric calculations, for failure loads, the use of 15-noded elements is recommended. The 6-noded elements are known to overestimate the failure load, but are ok for deformations at serviceability states. |

The results of fine/coarse and 6-noded/15-noded analyses are given below.

Table 2: Results for the maximum load reached on a strip footing on the drained sub-soil for different 2D and 3D meshes

| Mesh size | Element type | Nr. of elements | Max. load | Failure load |
|---|---|---|---|---|
| | | | [kN/m] | [kN/m$^2$] |
| very coarse mesh with local refinements under footing | 6-noded | 79 | 281 | 146 |
| coarse mesh | 6-noded | 121 | 270 | 141 |
| very fine mesh | 6-noded | 1090 | 229 | 121 |
| very coarse mesh with local refinements under footing | 15-noded | 79 | 236 | 124 |
| coarse mesh | 15-noded | 121 | 248 | 130 |
| very fine mesh | 15-noded | 1090 | 220 | 116 |
| **Analytical solutions of:** | | | | |
| - Vesic | | | | 117 |
| - Brinch Hansen | | | | 98 |
| - Meyerhof | | | | 97 |

In this table the failure load has been calculated as:

$$\frac{Q_u}{B} = \frac{Maximum\,force}{B} + \gamma_{concrete} * d = \frac{Maximum\,force}{2} + 6$$

From the above results it is clear that fine FE meshes give more accurate results. On the other hand the performance of the 15-noded elements is superior over the performance of the lower order 6-noded elements. Needless to say that computation times are also influenced by the number and type of elements.

# ADDITIONAL EXERCISE:

# UNDRAINED FOOTING

# INTRODUCTION

When saturated soils are loaded rapidly, the soil body will behave in an undrained manner, i.e. excess pore pressures are being generated. In this exercise the special PLAXIS feature for the treatment of undrained soils is demonstrated.

# SCHEME OF OPERATIONS

In PLAXIS, one generally enters effective soil properties and this is retained in an undrained analysis. In order to make the behaviour undrained one has to select 'undrained' as the *Type of drainage*. Please note that this is a special PLAXIS option as most other FE-codes require the input of undrained parameters e.g. $E_u$ and $\nu_u$.

## Aims

- The understanding and application of undrained soil behaviour
- How to deal with excess pore pressures.

### A) Geometry input

- Use previous input file
- Save as new data file
- Change material properties, undrained behaviour for clay
- Mesh generation, global mesh refinement

### B) Calculations

- Re-run existing calculation phases
- Construct footing
- Apply vertical force
- Apply horizontal force

### C) Inspect output

- Inspect excess pore pressures

# GEOMETRY INPUT

## Use previous input file

- Start PLAXIS by clicking on the icon of the Input program.
- Select the existing project file from the last exercise (drained footing).
- From the File menu select Save As and save the existing project under a new file name (e.g. 'exercise 1b')

## Change material properties

Change material properties by selecting the item *Soils & Interfaces* from the *Materials* menu or click on the Material sets button . Select the 'clay' from the *Material sets* tree view and click on the *Edit* button. On the first tab sheet, *General*, change the *Drainage type* to "*Undrained A*" and close the data set.

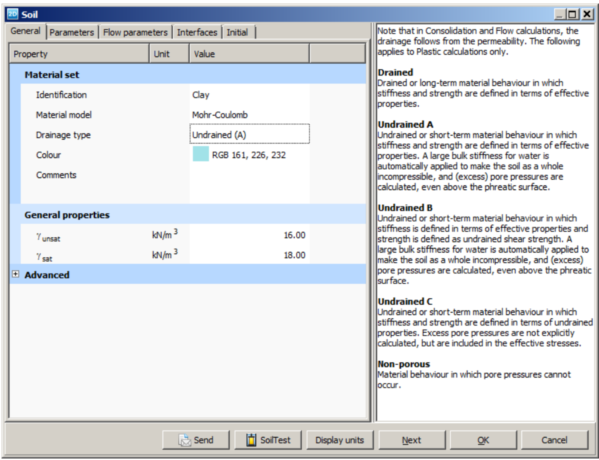



Figure 21: Set drainage type to "Undrained A"

## Mesh generation

The mesh generator in PLAXIS allows for several degrees of refinement. In this example we use the *Refine global* option from the *Mesh* menu, which will re-generate the mesh, resulting in an increased number of finite elements to be distributed along the geometry lines. Notice the message that appears about staged being reconstructed: the program will take into account the newly generated mesh for the previously generated initial conditions and staged construction phases. From the output window, in which the mesh is shown, press the continue button  to return to the Input program.

| | |
|---|---|
| **Hint:** | After generation of a finer mesh, the geometry may be refined until a satisfactory result appears. Besides the option *Refine global* several other methods of refinement can be used. |

**Hint:** After re-generation of the finite element mesh new nodes and stress points exists. Therefore PLAXIS has to regenerate pore water pressures and initial stresses. This is done automatically in the background when regenerating the mesh. Also, the new mesh is taken into account for any change to calculation phases with the exception of ground water flow analysis.

After generating the mesh one can now continue to the calculation program. Click on the *Caculations* button to proceed to the calculations program. Click 'yes' to save the data.

# CALCULATIONS

## Re-run existing calculation list

The calculation list from example 1 appears, as indicated below. All phases are indicated by (blue arrows). After mesh (re)generation, staged construction settings remain and phase information has been rewritten automatically for the newly generated mesh. However, this is not the case for points for load displacement curves due to the new numbering of the mesh nodes.

- Click on the *Select points for curves* button in the toolbar. Reselect the node located in the centre directly underneath
- Click on the *Calculate* button to recalculate the analysis. Due to undrained behaviour of the soil there will be failure in the 3rd and 4th calculation phase.

# INSPECT OUTPUT

As mentioned in the introduction of this example, the compressibility of water is taken into account by assigning 'undrained' behaviour to the clay layer. This results normally, after loading, in excess pore pressures. The excess pore pressures may be viewed in the output window by selecting:

- Select in the calculation program the phase for which you would like to see output results.
- Start the output program from the calculation program by clicking the *View output* button .
- Select from the *Stresses* menu the option *Pore pressures* and then $p_{excess}$, this results in Figure 22 .

The excess pore pressures may be viewed as stress crosses ( ), contour lines ( ), shadings ( ) or as tabulated output ( ). If, in general, stresses are tensile stresses the principal directions are drawn with arrow points. It can be seen that after phase 3 on the left side of the footing there are excess pore tensions due to the horizontal movement of the footing. The total pore pressures are visualised using the option of active pore pressures. These are the sum of the steady state pore pressures as generated from the phreatic level and the excess pore pressures as generated from undrained loading.

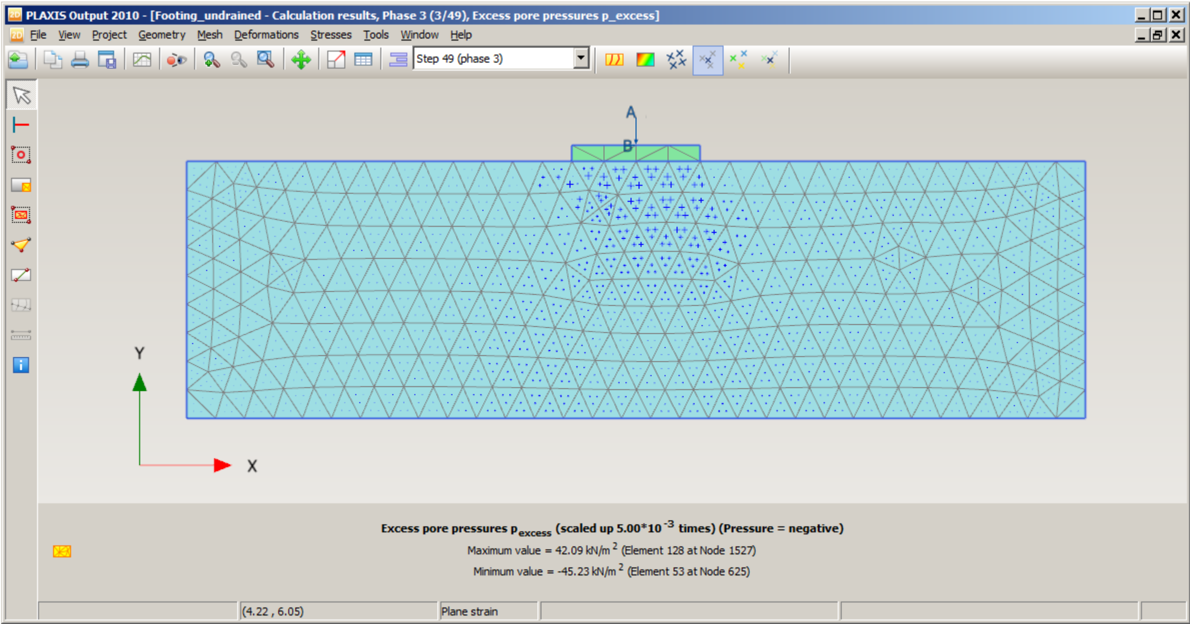



Figure 22: Excess pore pressures at the end of the 3rd phase

- Select from the *Stresses* menu the option *Pore pressures* and then $p_{active}$. The results are given in Figure 23.

From the load displacement curve it can be seen that the failure load in the last phase is considerably lower for this undrained case compared to the drained situation, as expected. For the undrained case the failure load is approx. 70 kPa.

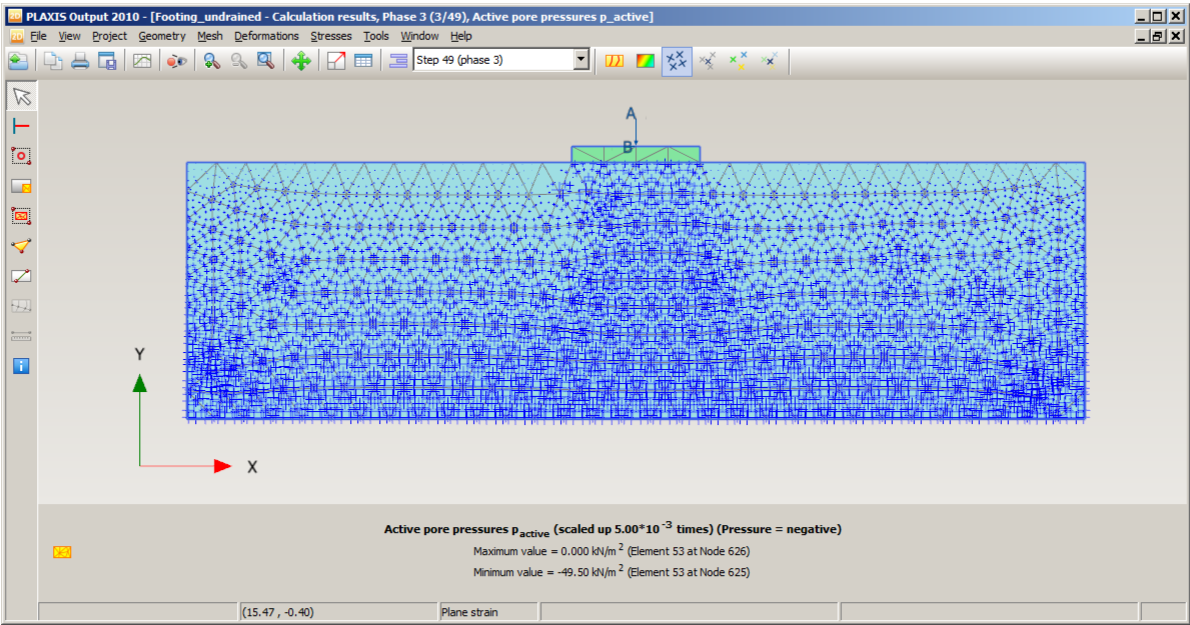



Figure 23: Active pore pressures at the end of phase 3

# APPENDIX A: BEARING CAPACITY CALCULATION

Given the formula for bearing capacity of a strip footing:

$$
\begin{aligned}
\frac{Q_f}{B} &= c \cdot N_c + \tfrac{1}{2}\gamma' B \cdot N_\gamma \\
N_q &= e^{\pi \tan \varphi'} \tan^2(45 + \tfrac{1}{2}\varphi') \\
N_c &= (N_q - 1) \cot \varphi' \\
N_\gamma &= \begin{cases} 2(N_q + 1) \tan \varphi' & (Vesic) \\ 1.5(N_q - 1) \tan \varphi' & (Brinch\,Hansen) \\ (N_q - 1) \tan(1.4\,\varphi') & (Meyerhof) \end{cases}
\end{aligned}
$$

Filling in given soil data:

$$
\begin{aligned}
N_q &= e^{\pi \tan(20)} \tan^2(55) = 6.4 \\
N_c &= (6.4 - 1) \cot(20) = 14.84 \\
N_\gamma &= \begin{cases} 2(6.4 + 1) \tan(20) = 5.39 & (Vesic) \\ 1.5(6.4 - 1) \tan(20) = 2.95 & (Brinch\,Hansen) \\ (6.4 - 1) \tan(28) = 2.97 & (Meyerhof) \end{cases}
\end{aligned}
$$

The effective weight of the soil:

$$\gamma' = \gamma_w - 10\,kN/m^3 = 18 - 10 = 8\,kN/m^3$$

For a strip foundation this gives:

$$
\frac{Q_f}{B} = c \cdot N_c + \tfrac{1}{2}\gamma' B \cdot N_\gamma = \begin{cases} 5 * 14.83 + \frac{1}{2} * 8 * 2 * 5.39 \approx \mathbf{117\,kN/m^2} & (Vesic) \\ 5 * 14.83 + \frac{1}{2} * 8 * 2 * 2.95 \approx \mathbf{98\,kN/m^2} & (Brinch\,Hansen) \\ 5 * 14.83 + \frac{1}{2} * 8 * 2 * 2.87 \approx \mathbf{97\,kN/m^2} & (Meyerhof) \end{cases}
$$