Submerged excavation

For LUSAS version:	21.0
For software product(s):	LUSAS Bridge plus or LUSAS Civil&Structural plus
With product option(s):	Geotechnical, Nonlinear

Problem Description

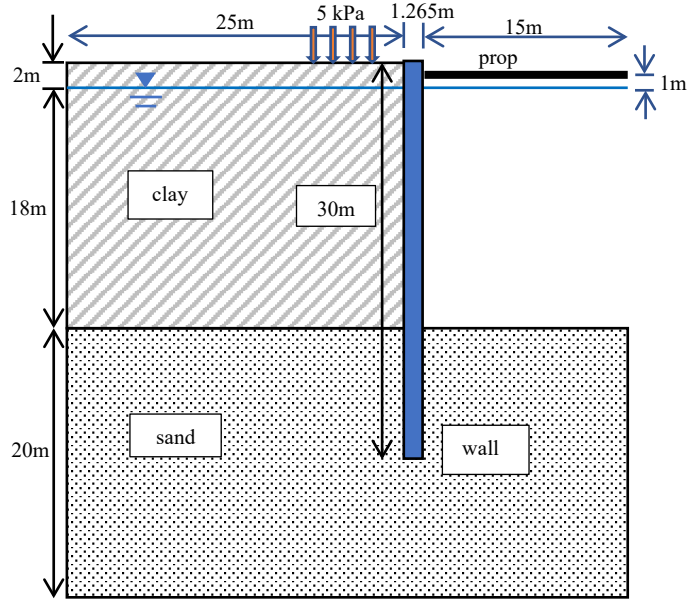


Figure 1: Problem geometry

A submerged excavation is analysed in which water in the excavation is not pumped out, maintaining a constant level throughout. The upper clay layer of 20m is to be excavated down to the sand layer beneath. Diaphragm walls are installed and propped, and the soil

in between excavated, with a first excavation of 2m down to the phreatic surface, followed by further two excavations of 9m each. A surface load of 5kPa beside the wall is also included. The props are at a spacing of 5m. Using symmetry only half of the problem is analysed. Geometry is shown in figure 1.

Keywords

Two phase elements, activation, deactivation, interface elements, calculation of bending moment and axial and shear forces from 2D stresses.

Associated Files

Associated files can be downloaded from the user area of the LUSAS website.

- **submerged_excavation.lvb** carries out automated modelling of the example.

- Use **File > New** to create a new model of a suitable name in a chosen location.
- Use **File > Script > Run Script** to open the lvb file named above that was downloaded and placed in a folder of your choosing.

Objectives

The objective is to calculate the bending moment and shear and axial forces in the wall and the axial force in the prop.

Preparing the Model Features

It is assumed that the user has basic knowledge about the definition of attributes and how to assign them to the model and so on. Other operations specific to this problem are covered in detail.

The user has to create a new model, set the Analysis category as 2D, and specify the model units as kN,m,t,s,C.

Feature Geometry

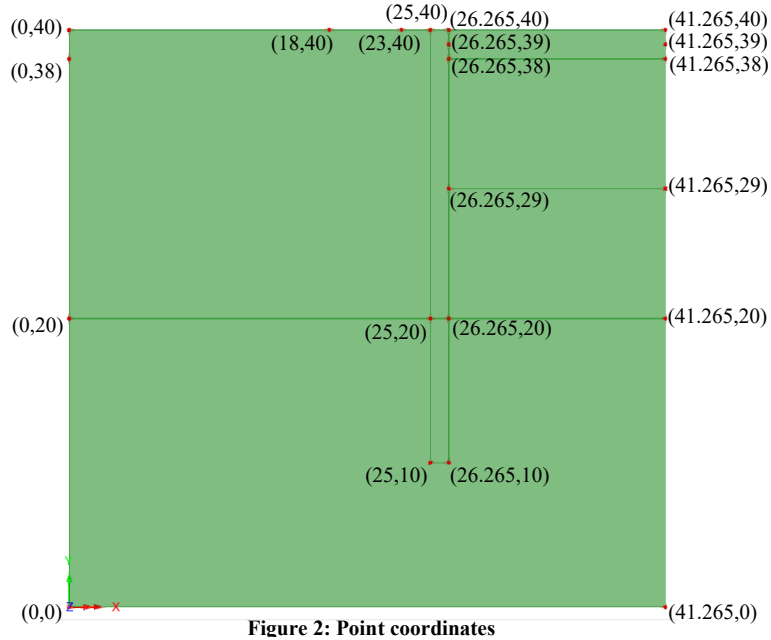


Figure 2: Point coordinates

Enter the coordinates for the points shown in figure 2 using **Geometry>Point>By Cords…** Then select the points to form the surfaces shown in figure 2 **Geometry> Surface > By Points..**. Details of the surface for the main sand layer, which is irregular, is shown in figure 3. For this surface, the points defining it must be selected in order around the perimeter in either a clockwise or anticlockwise direction.

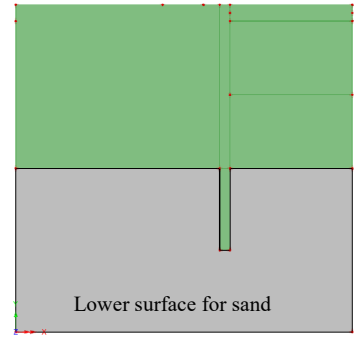


Figure 3: Details for the formation of sand surface

Preparing the Model Attributes

Defining the geometry

The prop is modelled by a solid circular section beam of diameter 44cm. Select **Attribute>Geometric>Line…** Click on the **From library/calculator** radio button. Ensure that **Parametric Sections** and **Circular Sections** are visible in the dropdown lists and then click on the down arrow in the **<Select>** dropdown list and then click on **New…** Enter .44 for the diameter **D** and 'prop' in the **Name** field. Then click **OK**. Enter 'prop' again in the **Name** field of the **Geometric Line** dialog and click **OK** (figure 4).

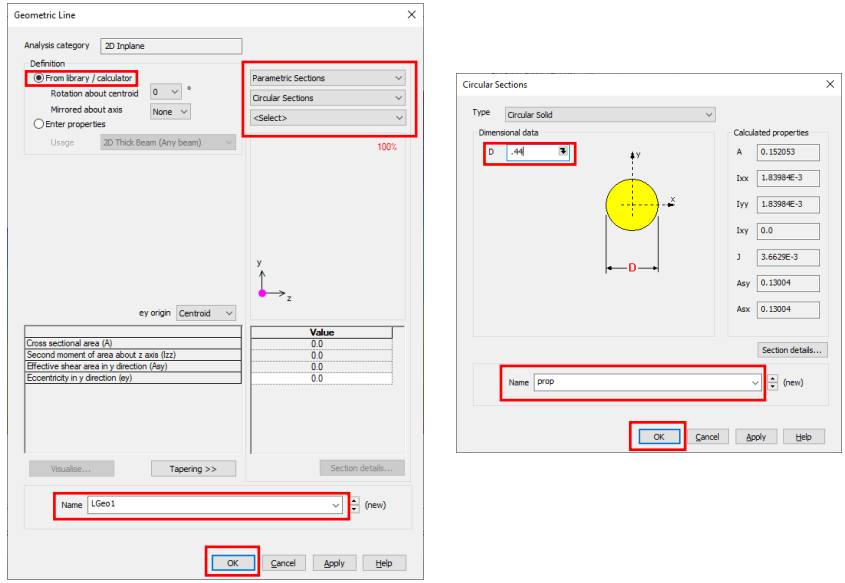


Figure 4: Dialogs defining prop geometric properties

Defining the Mesh

Select **Attributes>Mesh>Surface…** and define the soil and wall meshes shown in figure 5. Two-phase plane strain quadrilateral elements are used for the soil and plane strain quadrilateral elements for the wall. An element size of 1m is set for both.

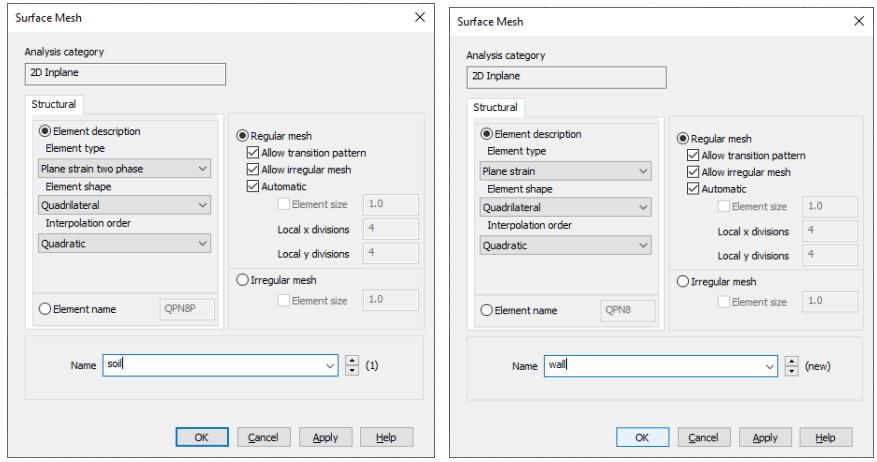


Figure 5: Details of soil and wall meshes

Now select **Attributes>Mesh>Line…** and define the mesh for the prop, which consists of four thick beams, as well as the mesh for the soil/wall interface. The interface elements are quadratic with an element length of 1m and are to be assigned manually. Manual meshing is used to allow us to connect the prop to the wall. Enter the name 'prop' for the prop mesh and 'Intf=1' for the interface elements. Dialogs are shown in figure 6.

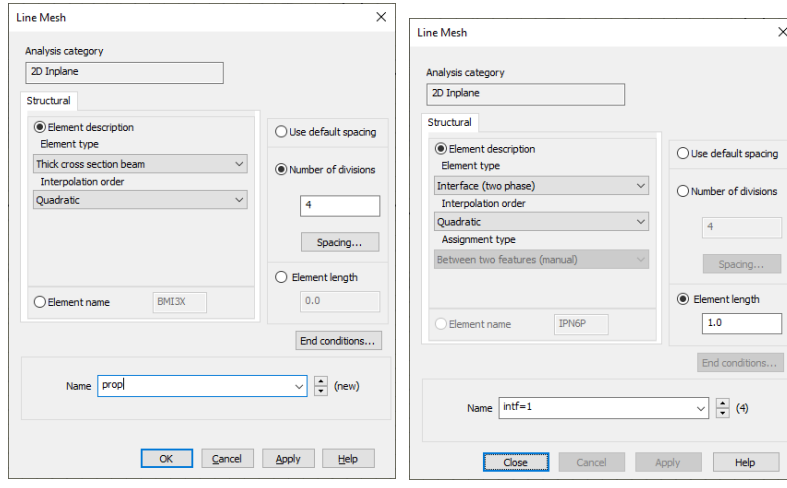


Figure 6: Details of prop and soil/wall interface meshes

Select all the surfaces and assign the 'soil' mesh to them. Now select just the soil column which is to be replaced by the wall. Copy the surfaces comprising the soil column to above the model using **Geometry>Surface>Copy**, with a simple translation of -5m in the x-direction and 35m in the y, as shown in figure 7.

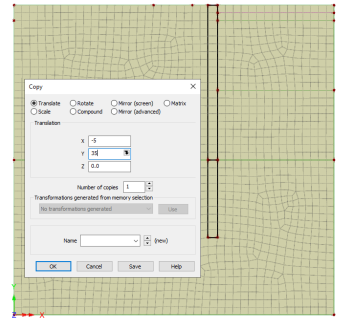


Figure 7: Make a copy of the soil column

Click on the Home button to see the new geometry. Now select the surfaces which are to form the wall and assign the 'wall' mesh to them (figure 8).

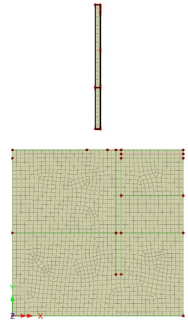
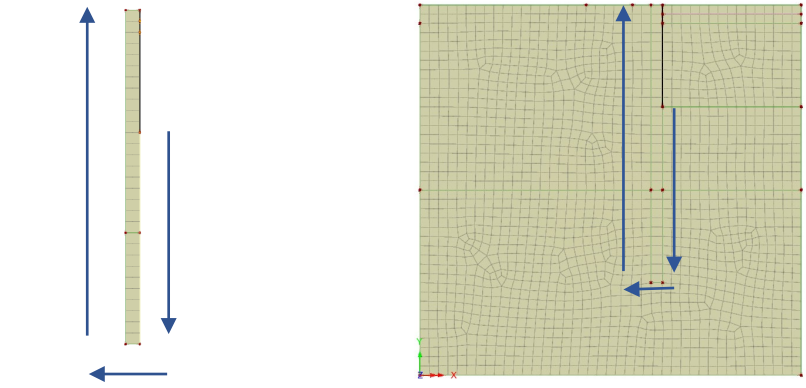


Figure 8: Select surfaces forming wall

Starting at the top right-hand side of the wall, select the lines in order one by one proceeding down to the bottom of the wall and then up the other side as shown in figure 9. Once selected, right-click in the display window and **Selection Memory>Set** to store the selected lines in the order of selection. Now click the corresponding lines of the soil column starting at the top right-hand side and proceed to select them in the same order.



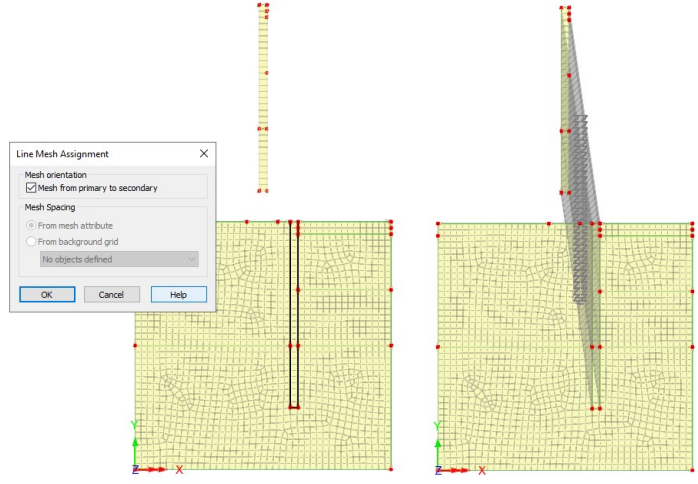
1) Select surfaces on wall

2) Save to selection memory

3) Select surface on soil column

Figure 9: Select of Primary and Secondary surfaces for interface elements

The interface elements are then assigned to the selected lines as shown in figure 10. The Secondary assignments are the lines which were selected first, ie those of the wall and the Primary assignments are the lines of the soil column.



Assignment of interface mesh

Generated interface elements

Figure 10: Generation of interface elements

The prop is now connected to the wall by selecting points on the wall and the soil and running a line between them, as shown in figure 11.

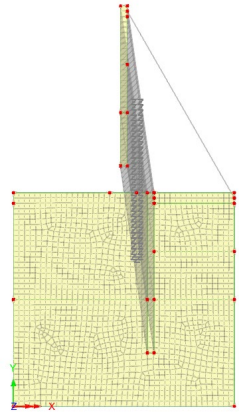


Figure 11: Connection of line to model prop

The line is selected and the prop mesh and geometric properties are assigned to it.

Defining the Materials

The clay and sand are modelled by the Modified Mohr-Coulomb with Rankine cutoff material. The concrete wall and steel prop by linear elastic materials. Properties are given in table 1.

Parameter	Clay	Sand	Concrete	Prop
Material	Mohr-Coulomb	Mohr-Coulomb	Linear elastic	Linear elastic
Young's Modulus	10,000 kPa	40,000 kPa	5.930 GPa	40* GPa
Poisson's ratio	0.35	0.3	0.3	0.0
Friction angle	25º	32º	-*-	-*-
Dilantancy angle	0.0º	2.0º	-*-	-*-
Cohesion	5.0 kPa	1.0 kPa	-*-	-*-
Tensile cutoff stress	0.0 kPa	0.0 kPa	-*-	-*-
Compressive cutoff stress	1E30 kPa	1E30 kPa	-*-	-*-
Saturated density	1.8 Tonnes/m^3	2.0 tonnes/m^3	-*-	-*-
Density	-*-	-*-	2.4 tonnes/m^3	1. 57* tonnes/m^3

Porosity	0.2	0.3	-*-	-*-
K_0	0.577	0.470	-*-	-*-

Table 1: material properties

* The values of the prop's Young's modulus and density have been modified by dividing by 5, the spacing between the props in metres. Plane strain elements model a 1m length of the excavation, so the loads applied to the prop by the wall are only 1/5 those actually applied. As the loads are only 1/5, the prop's Young's modulus is reduced by a factor of 5 to correctly model the displacements and similarly the density is also reduced by a factor of 5 to model the effect of gravity on the prop.

Additional two-phase properties are required, although the hydraulic conductivity is arbitrary as there is no water flow in this problem. The two-phase properties in table 2 are assigned to both the clay and sand.

Partially saturated parameters	Bulk modulus water	Hydraulic conductivity	Density fluid	Saturation at residual water content	Saturation at full water content
Constant water content	2.1 MPa	1.0 m/s	1 tonne/m^3	0.0	1.0

Table 2: Additional hydraulic properties

In addition, we need the properties at the concrete/soil interfaces. Two-phase interface elements are used because we want to use the effective rather than total stress in calculating the normal stress across the interface.

Interface	Angle of friction	Cohesion	Hydraulic conductivity
Concrete/clay	12º	0 kPa	1.0 m/s
Concrete/sand	22º	0 kPa	1.0 m/s

Table 3: Tangential slip interface properties

The modified Mohr-Coulomb and interface friction models are found in **Attribute>Materials>Geotechnical**

The sand, clay and concrete materials are assigned by selecting the appropriate surfaces of the model. The steel is assigned to the line meshing the prop. Interface material properties are assigned to the primary sides of the interface. To confirm which are the primary sides before assigning the interface materials right-click on Mesh/Line attribute 'Intf=1' and then on **Select Primary Assignments** as shown in figure 12.

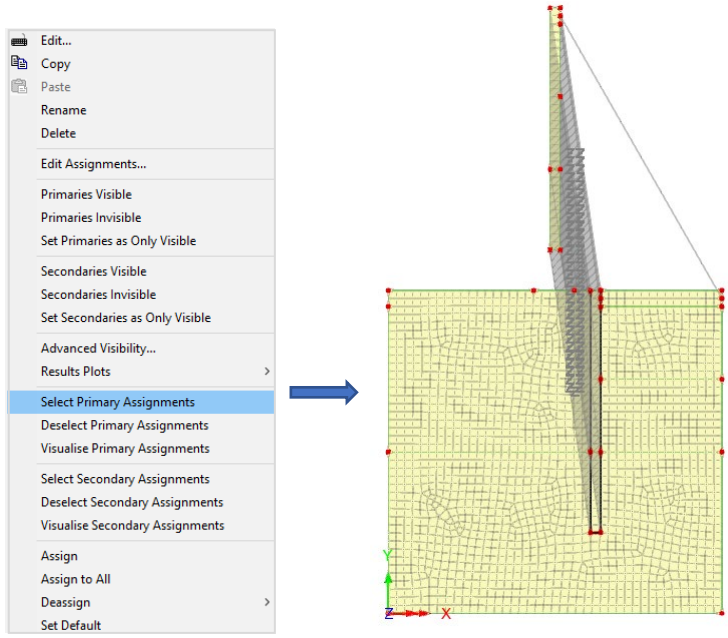


Figure 12: Identification of interface Primary Assignments

Defining the Supports

- The base of the model is constrained in both X and Y directions, while the lateral sides are limited in the X direction (Figure 13).

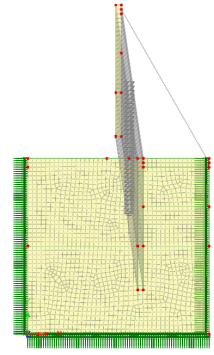
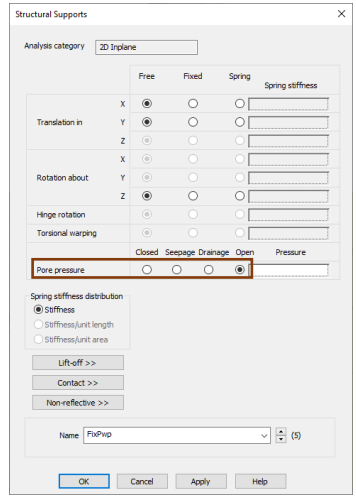


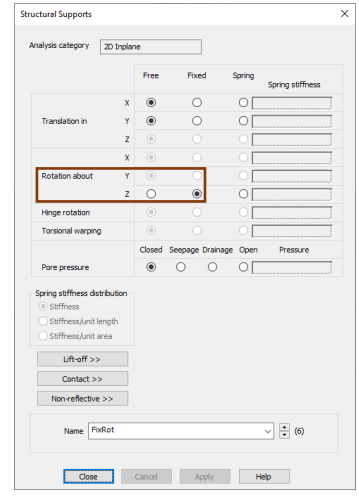
Figure 13: Displacement boundary conditions

- To establish the position of the phreatic surface, the pore pressure at a point is set to **Open** (figure 14). At the start of the analysis all pore water pressures are initialised to zero. Therefore, the **Open** boundary condition applied on the first increment fixes the pressure at that point to the initialisation value zero.

- A further boundary condition to restrain the rotation of the prop is required (figure 14).



Definition of open boundary condition



Definition of rotation boundary condition

Figure 14: Definition of point boundary conditions

- The displacement restraints in the x and y directions are applied to the lines along the edge of the model. The open pore water pressure and fixed rotation boundary conditions are assigned to the points shown in figure 15. The point boundary conditions combine with the displacement boundary conditions already applied to the lines.

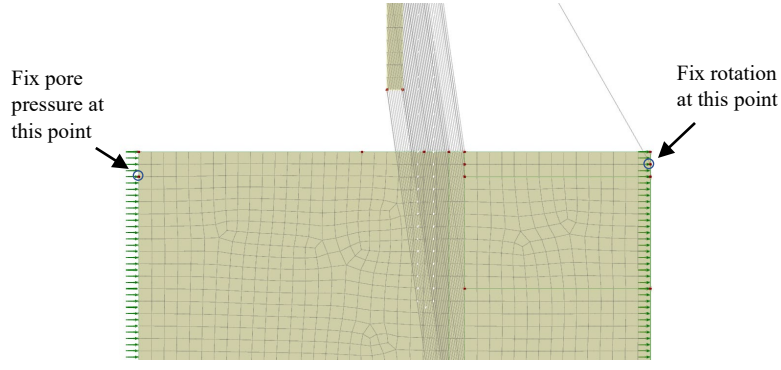


Figure 15: Apply boundary conditions at points

Defining the Loads

The 5 kPa surface load is defined using **Attributes>Loading….** and clicking the radio button **Distributed loads** followed by the radio button **Face** and then **Next**. Face loadings are defined in the local coordinate direction of the element face. In this case, the **x Direction** applies a shear stress along the face and the **y Direction** an inwards stress normal to the face. Enter 5 in the y-direction field and 'surf.load 5kPa' in the name field and click **Apply** (figure 16).

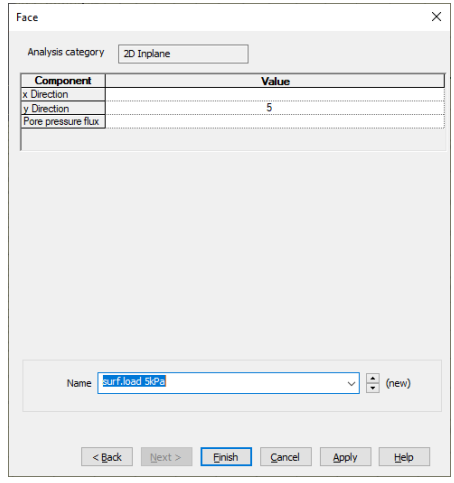
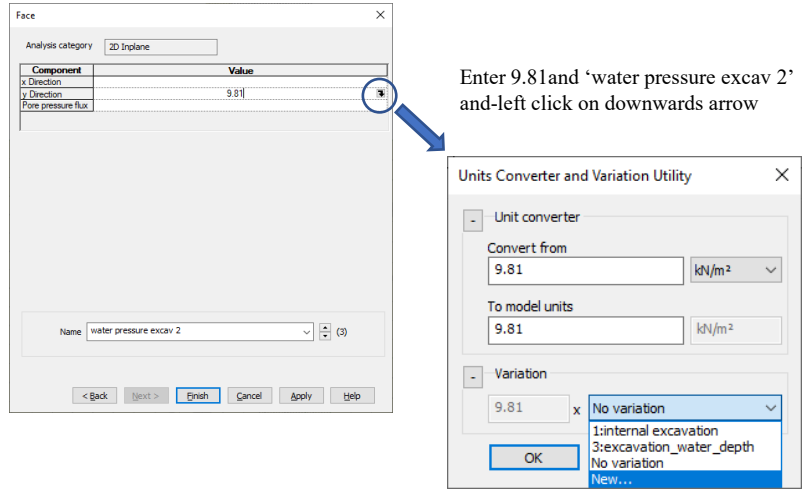


Figure 16: Definition of surface load attribute

In addition to the surface load, we need to define a water pressure load to apply to the inside of the excavation as the saturated soil is removed. Again, we define a face load to act normal to the surface. The water pressure p_w is defined

$$p_w = \rho g h$$

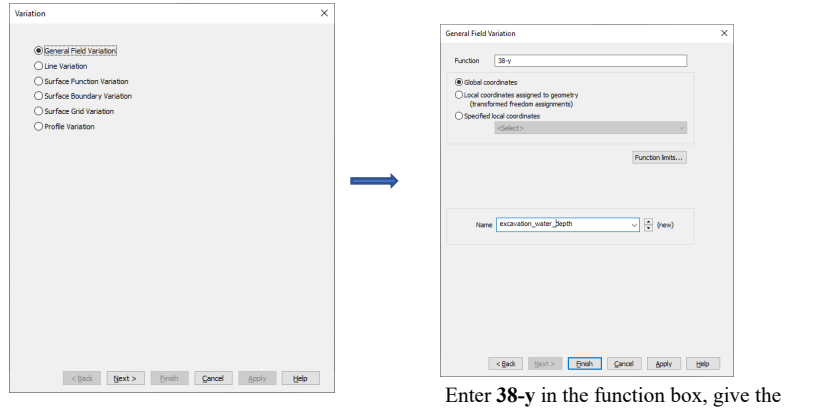
where ρ is the water density, 1 tonne/m^3, g is the gravitational acceleration and h the distance below the water surface. In figures 17, 18 and 19 the appropriate load attributes are generated. Change 5 to 9.81 and update the name field to 'water pressure excav 2'.



Enter 9.81and 'water pressure excav 2' and-left click on downwards arrow

Click on **New** in drop down list

Figure 17: Generate new load and new variation



Select radio button **General Field Variation** and then **Next**

Enter **38-y** in the function box, give the function the name 'excavation_water_depth' and then **Finish**

Figure 18: Generate function for depth of water in excavation

The function 38-y is the depth of water measured from the water table. This variation is to be multiplied by 9.81, ie gravity times the water density as shown in figure 19. Select **Apply** to save the attribute. Change the name to 'water pressure excav 3' to generate a second water pressure load attribute and then **Finish**.

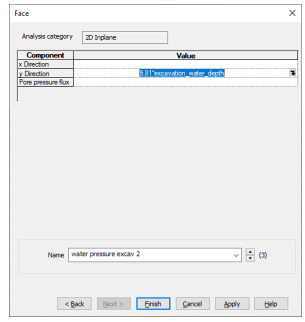


Figure 19: Definition of pressure attribute with variation

Deactivation Attributes

Various deactivation attributes are used during the analysis. Click on **Attributes > Activate and deactivate …** Click on **Next** and enter 'wall and prop' in the name field and then select the **Custom inactive treatment** radio button and click on the three dots **…** (figure 20)

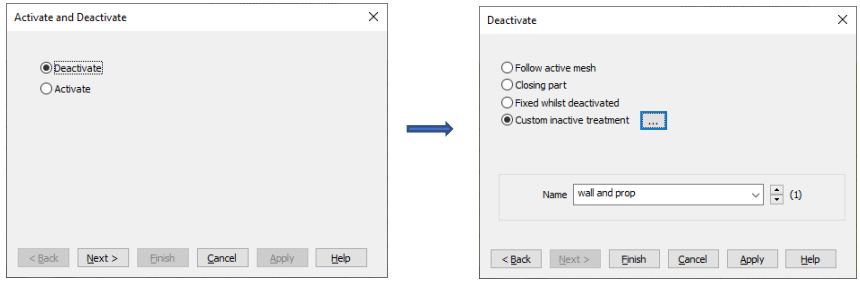


Figure 20: Definition of deactivation attribute

Click on the **Distribute over stage** radio button and **OK**. (Figure 21)

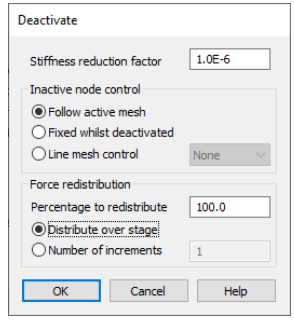


Figure 21: Set custom deactivation values

Distribute over the stage allows the residual forces to be distributed gradually over the load stage and improves convergence when the residual forces to be redistributed are large.

Click **Apply** to save the attribute. For clarity in the assignments, we will generate identical attributes but with different names. So, enter ‘soil column’ in the name field and click **Apply**, followed by ‘1st excavation’ and click **Apply**, ‘2nd excavation’ and click **Apply**, and finally ‘3rd excavation’ after which you click **Finish** to save the final attribute and close the dialog.

We also need a couple of activate attributes for the wall and prop. Click on **Attributes > Activate and deactivate …** and select the radio button **Activate**.

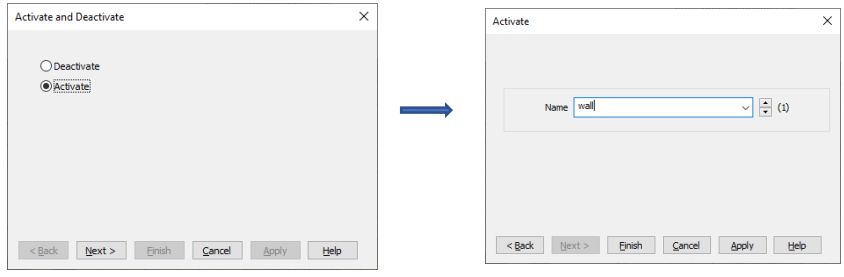


Figure 22: Definition of activate attributes

Enter ‘wall’ into the name field and click **Apply**. Change the name field to ‘prop’ and click **Finish** (figure 22).

Running the Analysis

We consider the following stages.

Initial Phase

The first stage establishes the initial stress and water pressure distributions. Right-click on **Loadcase** in the Treeview and rename to ‘init.conds’. Now, select the wall surfaces and assign the deactivation attribute ‘wall and prop’ to them (figure 23).

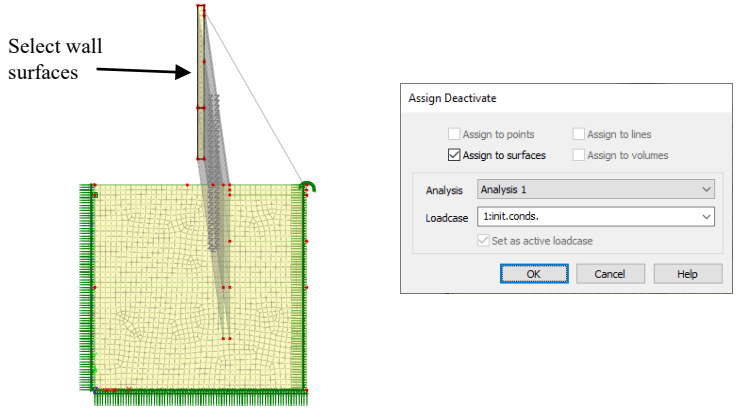


Figure 23: Deactivate wall

To deactivate the interface elements, the primary assignments must be selected, so now select the lines around the soil column and at the same time select the prop. Assign the Deactivate Elements attribute 'wall and prop' (figure 24).

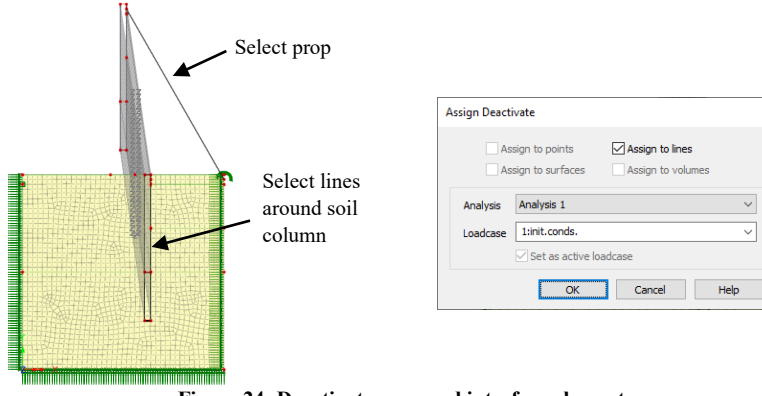
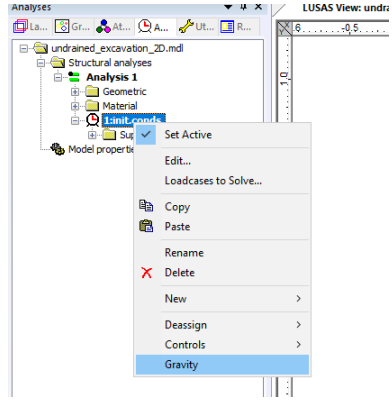
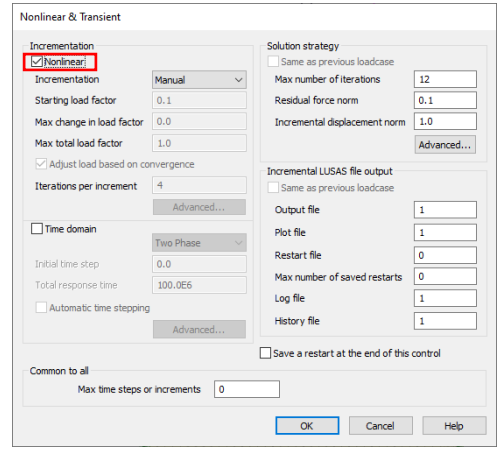


Figure 24: Deactivate prop and interface elements

Right-click on the loadcase name 'init.coords.' in the Treeview and select **gravity** to add gravity to this loading stage (figure 25). Right-click again on 'init.coords' and select **Controls> Nonlinear and transient…** .Check the tickbox **Nonlinear** to inform Modeller than the analysis is nonlinear (figure 25). The first stage is to be solved in a single increment so there are no other changes to make to the dialog.



Setting gravity for the stage



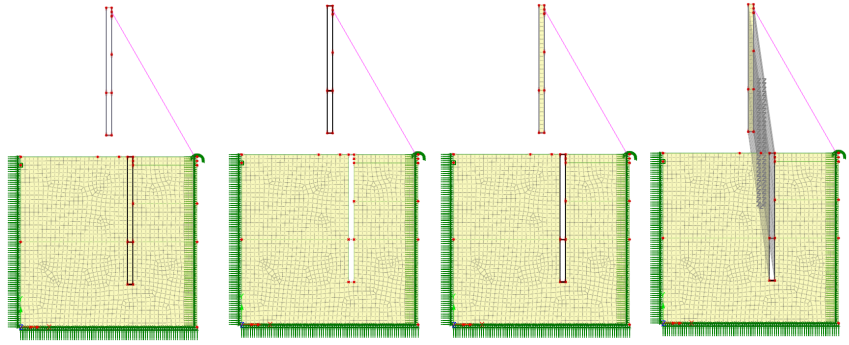
Setting nonlinear manual loading for stage

Figure 25: Setting values for stage

Installation of wall

In this stage the column of soil is replaced by the wall. Right-click on ‘init.conds’ in the Treeview and select **New>Loadcase…** In the dialog enter ‘install wall’ in the name field and check the tickbox **automatically add gravity to this loadcase**, and then the **OK** button.

Select surfaces forming the soil column and assign the **Deactivate Elements** attribute ‘soil column’ (figure 26). Then select the wall surfaces and assign the **Activate Elements** attribute ‘wall’. Finally select the lines around the soil column, the primary assignment of the interface elements, and assign **Activate Elements** attribute ‘wall’ to activate the interface elements.



Select soil column and assign Deactivation attribute

Select wall and assign Activation attribute

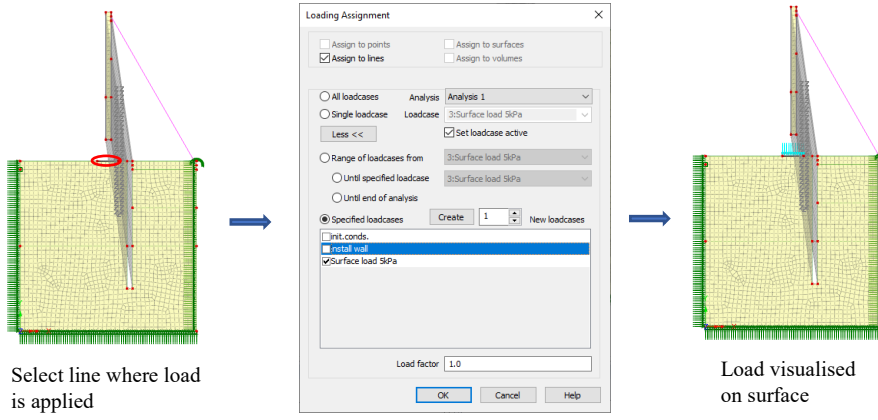
Select lines around soil column and assign Activation attribute

Final state: Soil inactive, wall and interface elements active

Figure 26: Installation of wall

Application of surface load

In this stage the surface load of 5 kPa is applied. Right-click on 'install wall' in the Treeview and select **New>Loadcase…** In the dialog enter 'surface load 5kPa' in the name field and check the tickbox **automatically add gravity to this loadcase**, and then the **OK** button. Select the line on the surface next to the excavation and assign the loading attribute 'surf.load 5kPa'(figure 27). In the loading assignment dialog set the radio button for **Specific loadcases**. The surface load is always applied when the loading is manual. Later, we will select in which load stages to include the surface load.



Select line where load is applied

Select **Specified loadcases**

Load visualised on surface

Figure 27: Application of surface load

1st Excavation

In this stage the first two metres of clay are excavated. Right-click on ‘surface load 5kPa’ in the Treeview and select **New>Loadcase…** In the dialog enter ‘1st excavation’ in the name field and then the **OK** button. Note: **automatically add gravity to this loadcase** is not used during this stage because we are going to use automatic loading to excavate the soil and we do not want to factor gravity at the same time. Select the surface of the first excavation and assign the **Deactivate Elements** attribute ‘1st excavation’ (figure 28). Now select the lines of the surface next to wall and assign the **Deactivate Elements** attribute ‘1st excavation’ to deactivate the interface elements. This is important to stop them flapping around and sabotaging the solution when the soil is removed and the residual forces are reduced to zero.

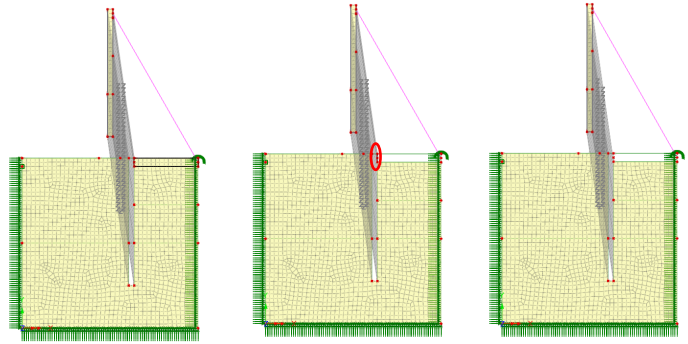


Figure 28: 1st Excavation

The excavation will be done in small increments over the stage using the automatic loading facility. Right-click on ‘1st excavation’ in the Treeview and select **Controls > Nonlinear and transient…**. On the dialog ensure that the tickbox **Nonlinear** is checked and select **Automatic** from the **Incrementation** drop-down list. Set the **Max change in load factor** to 1.0 and finally the **Max time steps or increments** to 20 (figure 29).

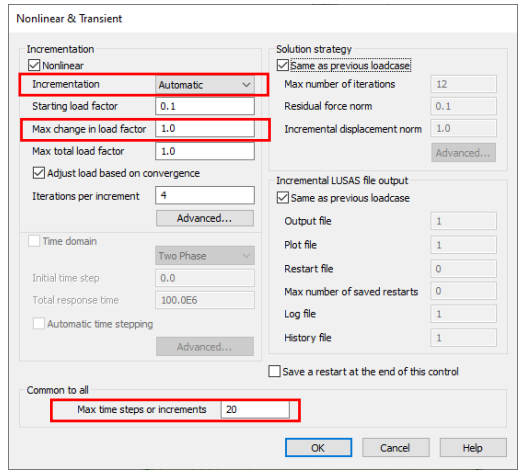


Figure 29: Nonlinear parameter settings

The gravity load and 5 kPa surface load from the previous manual loading stage 'Surface load 5 kPa' are maintained at a constant value during this stage, whilst the residual forces from the deactivation of the soil are gradually reduced.

Install prop

In this stage the prop is activated. Right-click on '1st excavation' in the Treeview and select **New>Loadcase…** In the dialog enter 'install prop' in the name field and check the tickbox **automatically add gravity to this loadcase**, and then the **OK** button. We will come back and add the surface load to this stage after the definition of all the other stages is completed. Select the line representing the prop and assign the **Activate Elements** attribute 'prop' (figure 30).

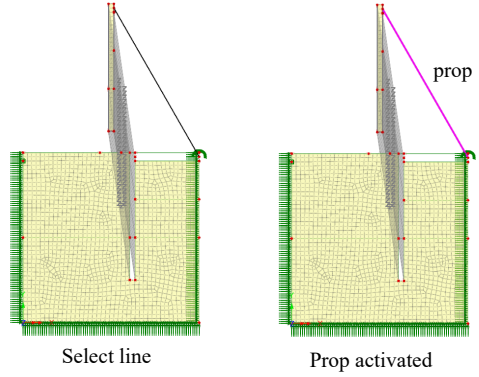


Figure 30: Installation of prop

It is not necessary to add a nonlinear control as the solution will revert back to manual loading.

2nd Excavation

In this stage, the next nine metres of clay are excavated. Right-click on 'install prop' in the Treeview and select **New>Loadcase…** In the dialog enter '2nd excavation' in the name field and then click the **OK** button. Select the surface of the second excavation and assign the **Deactivate Elements** attribute '2nd excavation'. Now select the line of the surface next to wall and assign the **Deactivate Elements** attribute '2nd excavation' to deactivate the interface elements (figure 31).

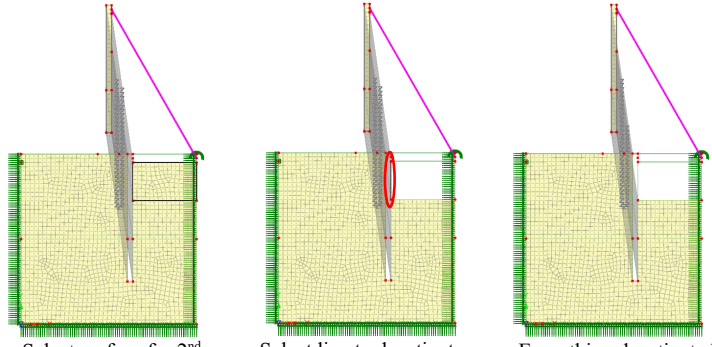
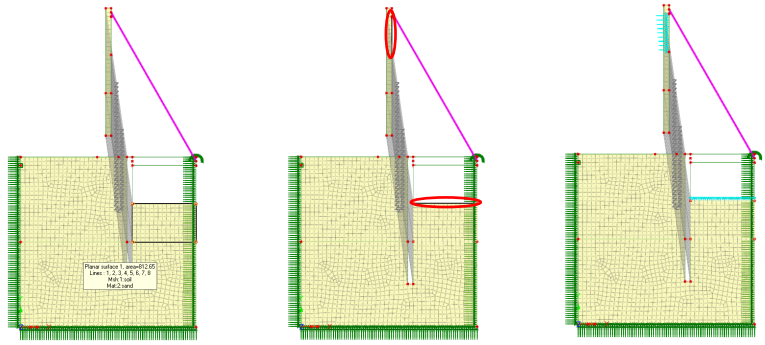


Figure 31: 2nd excavation

In removing the saturated soil, we also remove the water from the excavation. To put it back at the same rate as it is removed during the reduction of residual forces, we use the load attribute 'water pressure 2nd excavation' assigned to the wall and the floor of the excavation. First, select the surface directly below the excavation floor and save it in the selection memory by right-clicking in the display window and choosing **Selection Memory > Set** (figure 32). This is done as the line marking the excavation floor separates two surfaces and it is necessary to identify to which surface the pressure load is to be applied to. Now, select the lines on the wall and the floor of the excavation and assign the **Loading Attribute** 'water pressure excav 2'. Again, choose the option to assign load to **Specified Loadcases** (figure 33). Note that the visualised pressure loading on the wall is in the wrong direction. This is because the wall is above 38m and has a negative loading. When the wall is move into its final position the load is correctly applied.



Select surface below excavation and save in Selection Memory

Select lines and assign load attribute

Loading applied

Figure 32: Application of water pressure loading for 2nd excavation

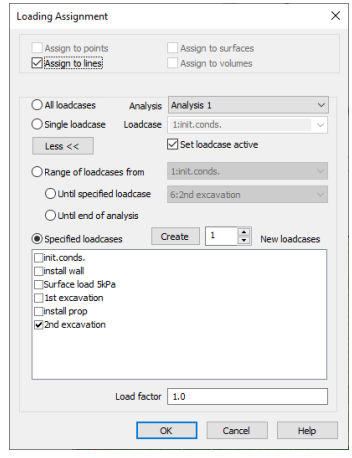


Figure 33: Loading assignment of 'water pressure excav 2'

The removal of the residual forces from the soil after excavation is done in small increments to allow the solution to converge. We again need automatic loading. Right-click on the '2nd Excavation' in the Treeview and **Controls > Nonlinear and Transient**. Check the **Nonlinear** tickbox. Set **Incrementation** to automatic, **Max change in load factor** to 1.0 and **Max time steps or increments** to 20 and click **OK** (figure 34).

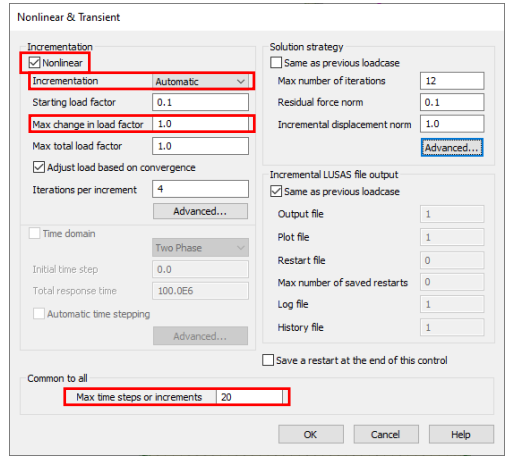


Figure 34: Nonlinear controls for 2nd excavation

2nd Excavation lock-in

Because the last stage used automatic loading and we applied a pressure load to represent the freestanding water in the excavation we need to lock that freestanding water into the constant loading. This is done by doing a lock-in stage. Right-click on '2nd excavation' in the Treeview and select **New>Loadcase…** In the dialog enter '2nd excavation lock-in' in the name field and check the tickbox **automatically add gravity to this loadcase**, and then click the **OK** button. That is all for now. We will return later to add the surface load and the freestanding water load.

3rd Excavation

In this stage, the final nine metres of clay are excavated. This is a repeat of the 2nd excavation. Right-click on '2nd excavation lock-in' in the Treeview and select **New>Loadcase…** In the dialog enter '3rd excavation' and then click on the **OK** button. Select the surface of the third excavation and assign the **Deactivate Elements** attribute '3rd excavation' (figure 35). Now select the lines of the surface next to wall and assign the **Deactivate Elements** attribute '3rd excavation' to deactivate the interface elements as well.

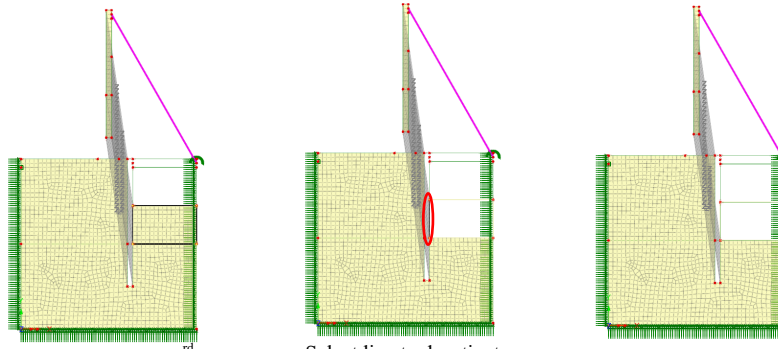


Figure 35: 3rd Excavation

Again, the freestanding water must be added back in as the residual forces are reduced. Select the surface directly below the excavation floor and save it in the selection memory by right-clicking in the display window and choosing **Selection Memory > Set** (figure 36). Now, select the lines on the wall and the floor of the excavation and assign the **Loading Attribute** 'water pressure excav 3'. Choose the option to assign load to **Single loadcase** (figure 37) as there are no more stages to consider.

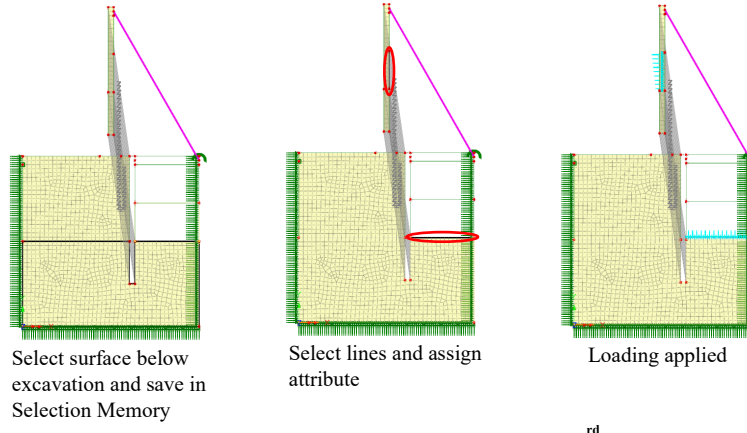


Figure 36: Application of water pressure loading for 3rd excavation

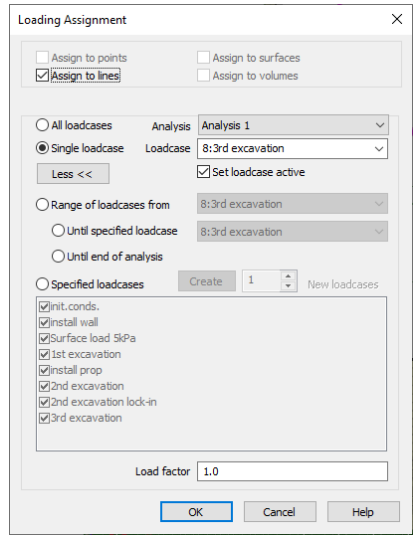


Figure 37: Load assignment for water pressure loading for 3rd excavation

Right-click on '3rd Excavation' in the Treeview and **Controls>Nonlinear and Transient**. Check the **Nonlinear** tickbox. Set **Incrementation** to automatic, **Max change in load factor** to 1.0 and **Max time steps or increments** to 20 and click **OK** (figure 38).

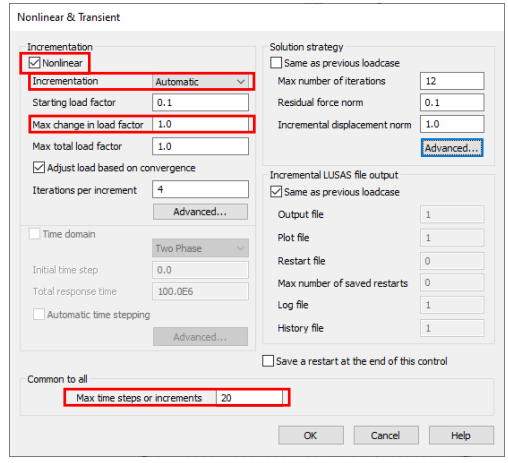
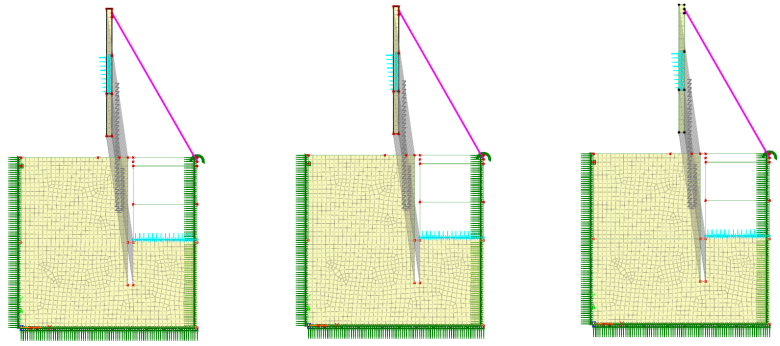


Figure 38: Nonlinear controls for 3rd excavation

It is now time to put the wall into position. To avoid the wall geometry merging with that of the soil pile after moving, select the surfaces of the wall and then click on **Geometry > Surface > Make Unmergeable**. Then select all the wall lines and click on **Geometry**

> Line > Make Unmergeable. And finally select the points and click on **Geometry > Point > Make Unmergeable** (figure 39).



Select surfaces on wall and make unmergeable

Select lines on wall and make unmergeable

Select points on wall and make unmergeable

Figure 39: Make all geometry unmergeable before moving wall into place

Now select the wall surfaces and click on **Geometry > Surface > Move…**. Enter **x**=5 and **y**=-35 and click **OK** (figure 40).

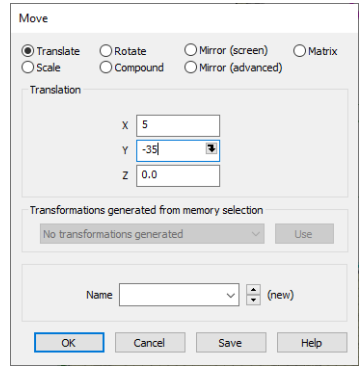


Figure 40: Move wall into final position

If everything went well it will look like figure 41.

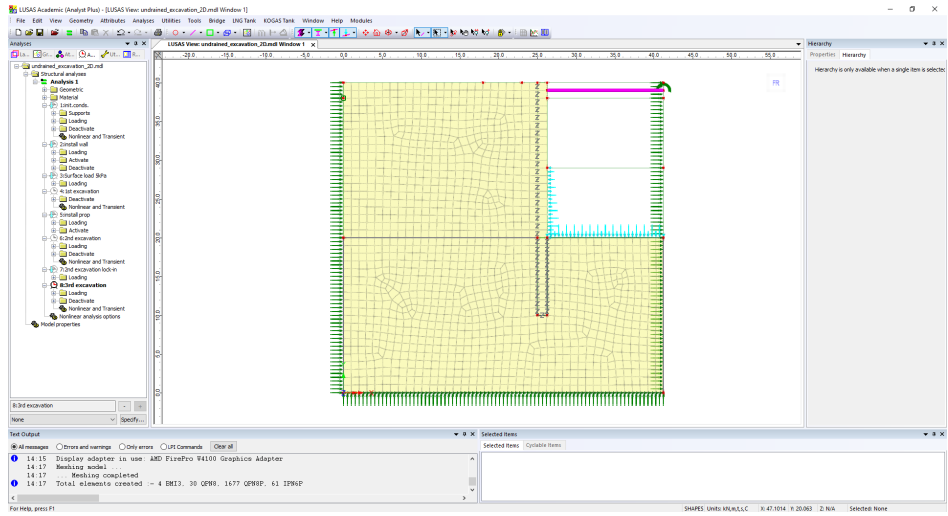


Figure 41: State after definition of 3rd excavation

Before solving we need to apply the missing constant loads to the manual stages of the solution. Left-click on the **Attributes** tab and in **Loading** right-click on 'surf.load 5kPa' and then click on **Edit Assignments…**. The manual load stages following the application of the load are 'install prop' and '2nd excavation lock-in'. Check the tickboxes corresponding to these stages and then **OK** (Figure 42). Similarly, right-click on 'water pressure excav 2' and then click on **Edit Assignments…**. Check the tickbox for '2nd excavation lock-in' and then **OK**.

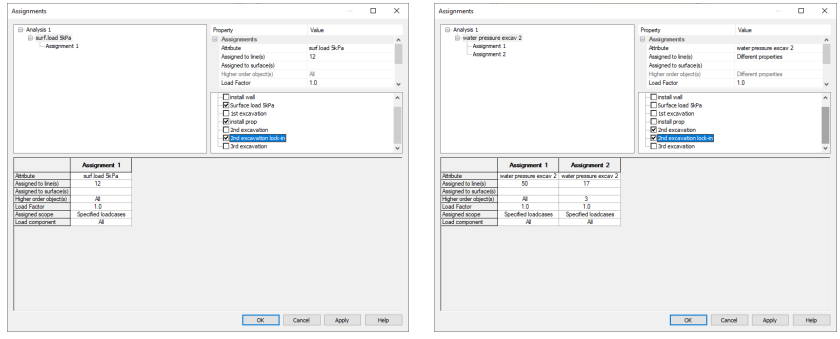


Figure 42: Loading assignments for manual load stages

The gravity load is indicated by an arrow and should be visible in the Treeview. It should only appear in the manual loading stages (figure 43).

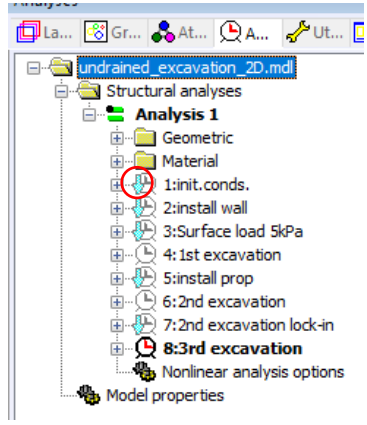


Figure 43: Manual load Stages with gravity loading

The analysis is now run by clicking on the equals icon (figure 44).

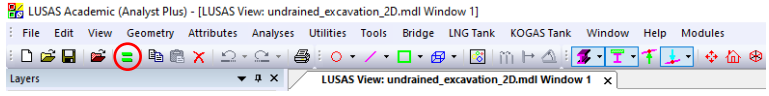


Figure 44: To run the analysis

Then check the tickbox **Analysis 1** and click on **OK**, (figure 45)

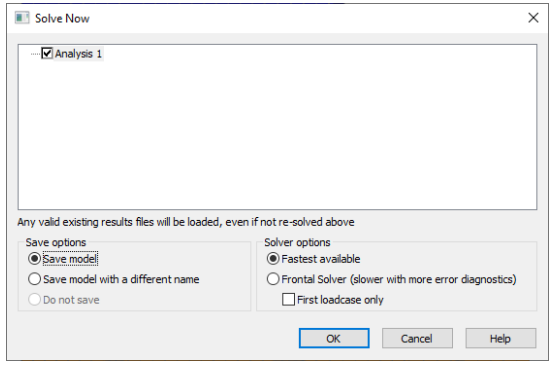


Figure 45: Solve Now dialog

Viewing the Analysis

Analysis loadcase results are presented in the Treeview.

In the Treeview, right-click on the final increment and then click on **Set Active** (figure 46).

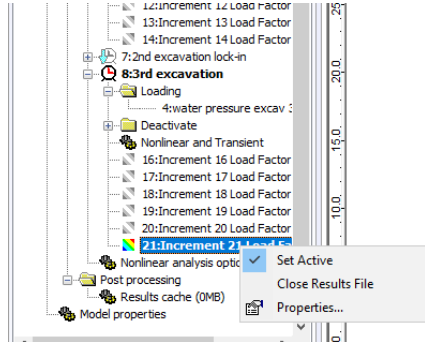


Figure 46: Setting the active results

For the purposes of the display of the results, the 'Z's marking the position of the interface elements can be switched off. In the **Layers** tab right-click on **Mesh** and then click on **properties…** On the **Visualise** tab of the properties dialog uncheck the **Joint and Interface elements** tickbox. The process needs to be repeated for the **Deformed mesh** as well.

To hide the supports and water pressure loading click on the icons shown in figure 47.

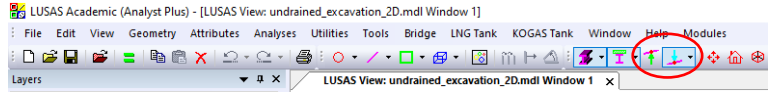


Figure 47: Hide supports and loads

Additionally, we will switch off the **Geometry** and **Deformed mesh** layers. Click on the **Layers** tab then right-click on **Geometry** and untick **Display** (figure 48).

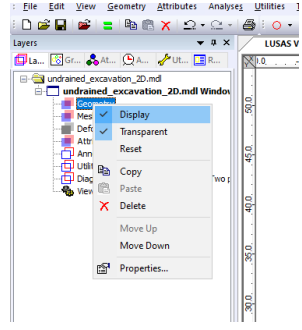


Figure 48: Hiding Geometry layer

Do likewise with the **Deformed mesh** layer.

The contours we will check are those of the water pressure distribution. Right-click in the display window and click on **Contours**. Click on the downwards arrow of the **Component** field and select **Pres** and then click **OK** (figure 49).

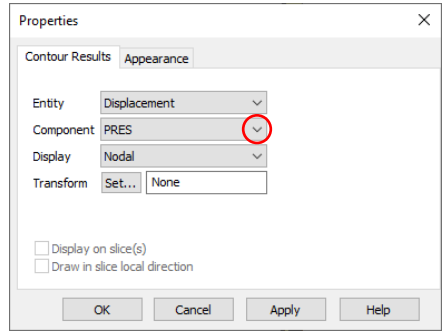


Figure 49: Contour dialog

As required the water pressure decreases with depth and the contours are horizontal (figure 50).

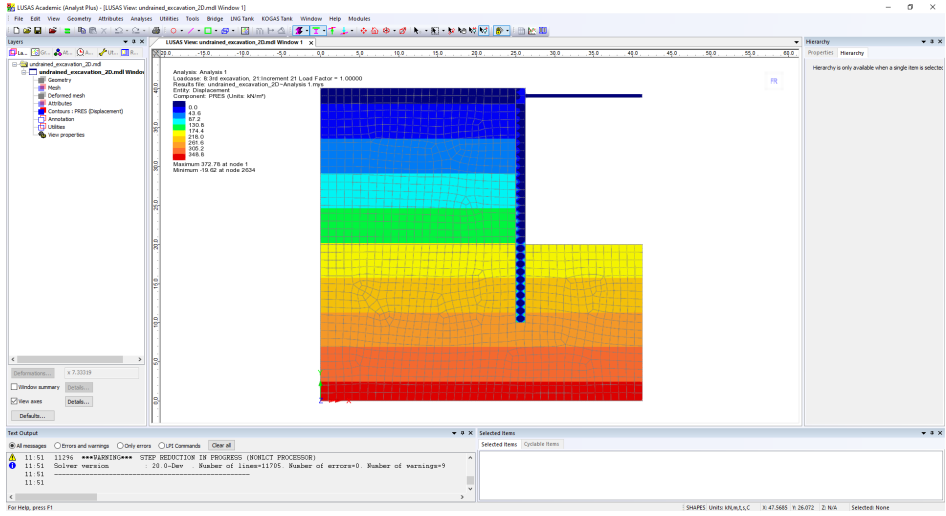


Figure 50: Water pressure contours at end of excavation

Next, we will plot the effective strain on the deformed mesh (figure 51). In the **Layers** tab right-click on **Contours** and then click on **Properties…** Select **Strain -Two phase plane strain** for the **Entity** and **EE** as the **Component** from the dropdown lists. The

Mesh can be hidden and **Deformed Mesh** displayed by clicking on each item's properties in turn in the **Layers** tab.

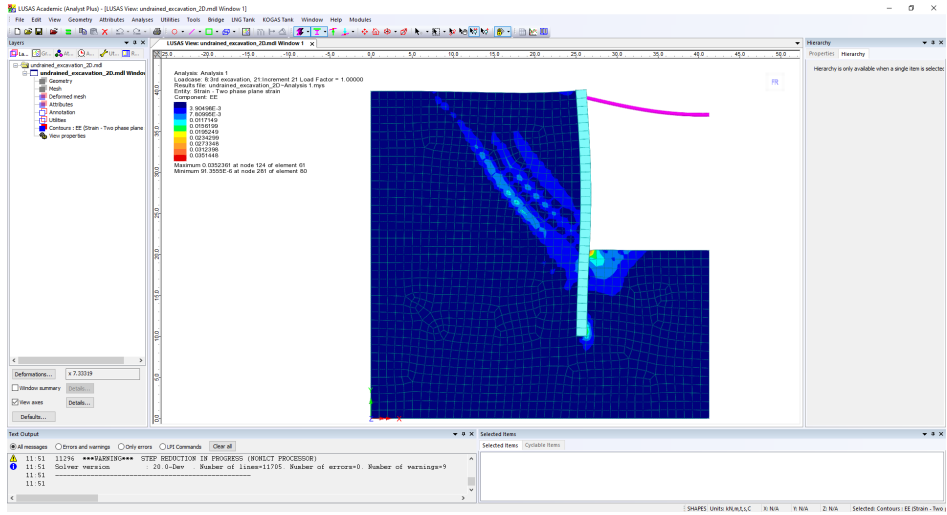


Figure 51: Effective strain at end of excavation

We can plot contours of the resultant displacements by again right-clicking on **Contours** in the **Layers** tab and then **Properties…**. Selecting **Entity > Displacement** and **Component > RSLT** from the dropdown lists, we get (figure 52). The displacements are dominated by the prop which has deflected 26cm downwards under self-weight. If we select all the elements using the cursor and then deselect the bar elements modelling the prop by pressing the CTRL key whilst selecting the beams again, we are left with just the soil elements selected (figure 53). Right-click in the display window and select **Keep as Only Visible**. The prop is hidden and the contours are redrawn with just the soil and wall (figure 54).

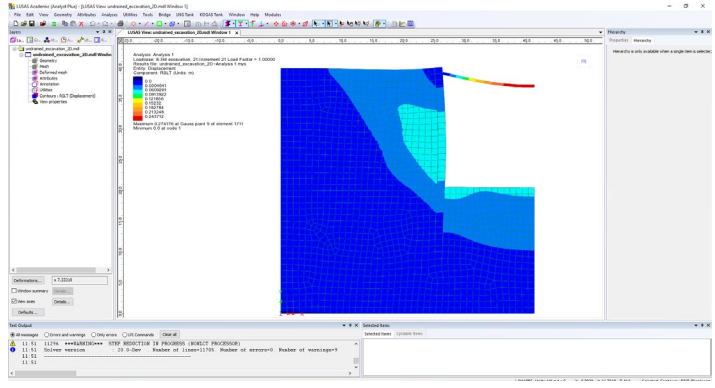


Figure 52: Resultant displacement after excavation

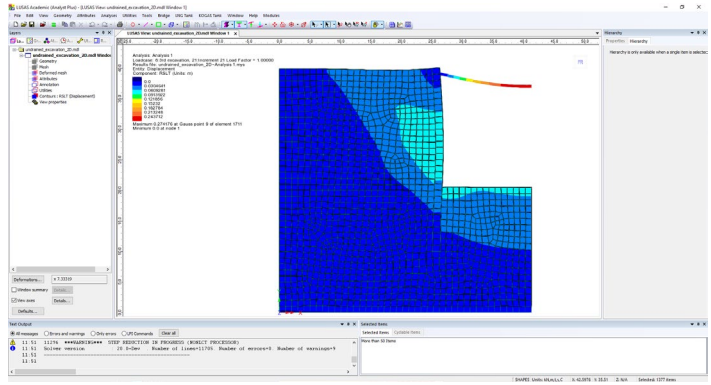


Figure 53: Select just soil and wall elements

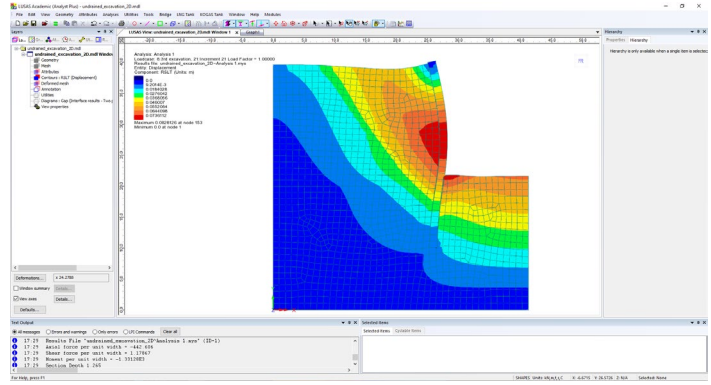


Figure 54: Contours and deformed mesh redrawn without prop

Next, we will evaluate the bending moment and shear and axial forces in the diaphragm wall. Hide the contours by clicking on the **Layers** tab and then right-clicking on **Contours** and unticking **Display**. On the **Attributes** tab, right-click on the **Material** 'concrete' and then click on **Set as Only Visible** to display just the wall. The original mesh is then made visible by clicking on the **Layers** tab, right-clicking on **Mesh** and then ticking **Display**. As we have used membrane elements rather than beams to model the wall, the bending moments and shear and axial forces it sustains are not readily available and must be calculated from the membrane element stresses. Use the magnify bottom to zoom in on the wall. Then click on **Utilities > Graph through 2D…** Accept the default option to use a grid by clicking **OK** (figure 55).

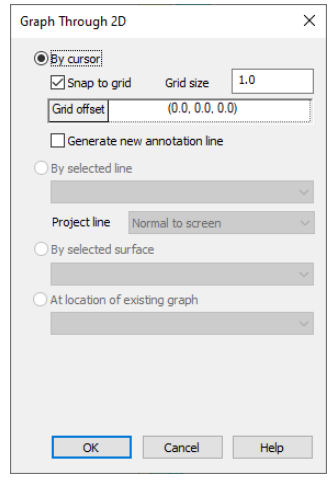


Figure 55: Setting line to cut model

Now click on one of the dots of the grid on the left side of the wall and whilst keeping the mouse button pressed move the cursor to the right. Release the button on the second dot to the right (figure 56).

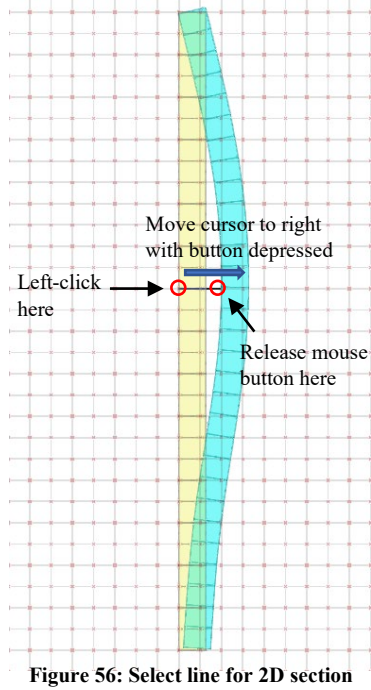


Figure 56: Select line for 2D section

Accept the defaults on the next dialog to act on the current loadcase by clicking on **Next** (figure 57).

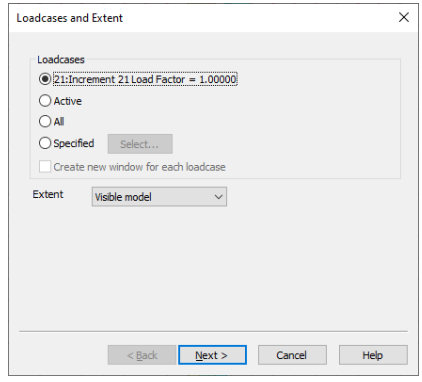


Figure 57: Result to use in 2D section

On the following dialog select the **Resultant effects from 2D model** radio button and click **Next** (figure 58).

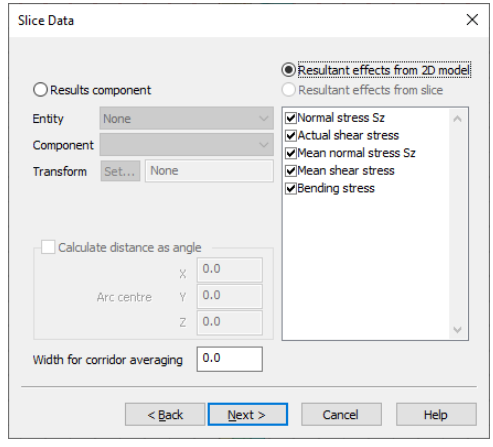


Figure 58: Results to calculate across 2D section

And finally, the graph dialog appears. Click **Finish** (figure 59).

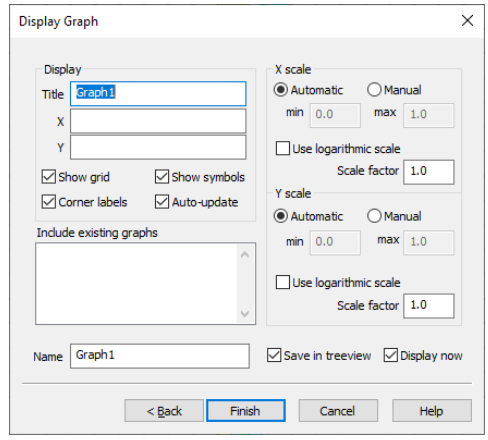


Figure 59: Graph dialog for plotting of section results

The results from the section are presented in graphical form (figure 60). The results are calculated by transforming the 2D membrane stresses to the section line. Stresses at right angles to line are integrated to give the axial force and bending moment, whilst the shear force is evaluated parallel to the line. Data values can be extracted from the graph datasets and also from the Text Output window.

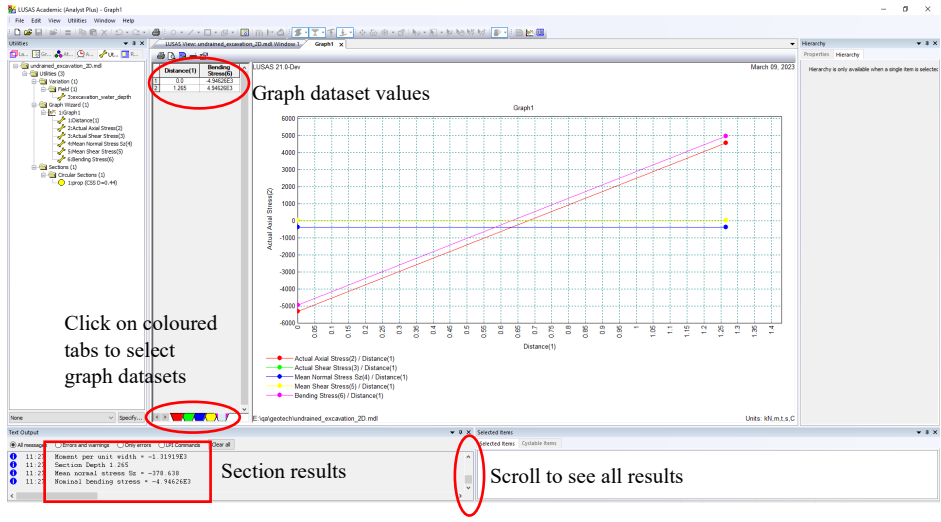


Figure 60: Results from 2D section across wall

At the cross section selected in figure 56, the moment/unit width is -1319.2 kNm/m, the shear force 32.2 kN/m and the axial force -479.0 kN/m. By evaluating the resultant values at different sections along the wall length the bending moment and shear and axial force

distributions along the wall are found. Results are plotted using 3rd party software as shown in figure 61.

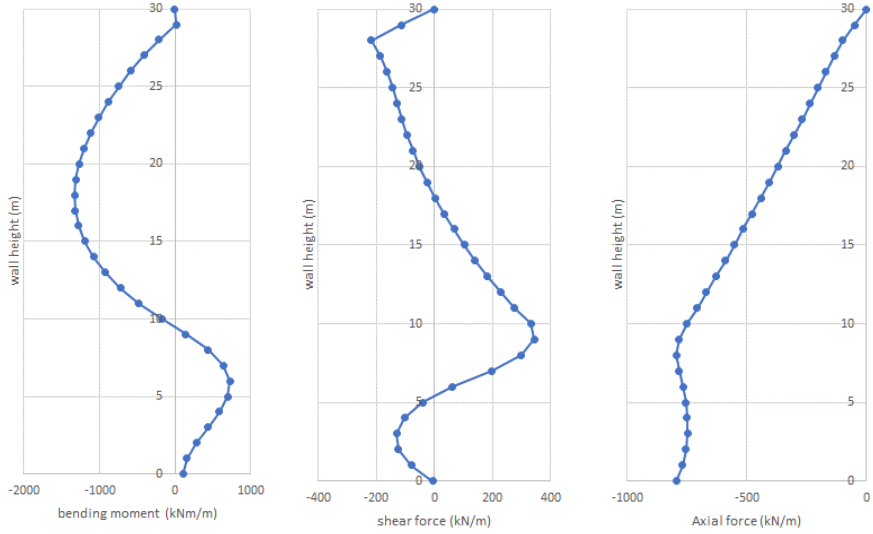


Figure 61: Bending moment, shear and axial force distribution in wall

Right-click in the display window and **Cancel cursor mode** (figure 62).

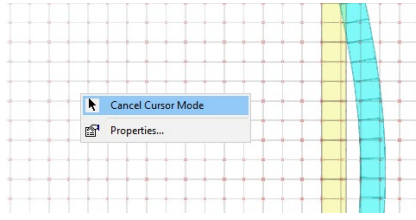


Figure 62: Return to normal cursor

Now, in the **Attributes** layer click on **Material** 'prop' and then on **Set as Only Visible**. Centre the prop in the display screen. Right-click on the display screen and select **Diagrams**. Click on the tab **Diagram Display** and untick **Peaks Only** to display all the nodal values (Figure 63).

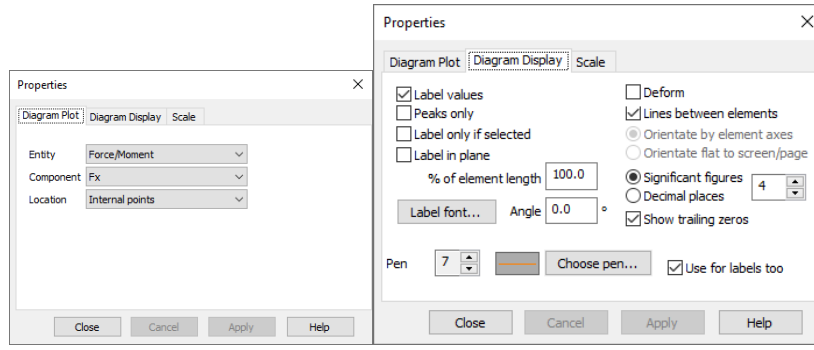


Figure 63: Diagram plot dialogs

Plots for both the axial force and bending moment are shown in Figures 64 and 65. Remember that we reduced both the prop stiffness and its density by a fifth to account for the reduced load applied by the wall, so the actual prop force and bending moments are calculated by multiplying the values by 5, the distance between props.

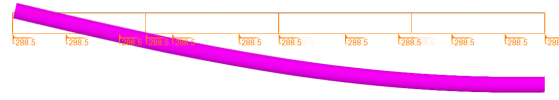


Figure 64: Axial force distribution in prop

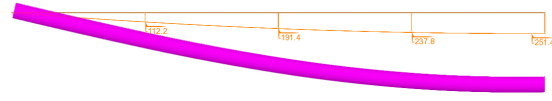


Figure 65: Bending moment distribution in prop