Dry excavation

For LUSAS version:	23.0
For software product(s):	LUSAS Bridge plus or LUSAS Civil & Structural plus
With product option(s):	Geotechnical, Nonlinear, Dynamic

Description

A dry excavation is analysed in which water is pumped out as the excavation progresses. Diaphragm walls are installed, surface loads activated and 3m metres of fill material excavated. Ground anchors, horizontally spaced at 3m intervals, are then installed and the water level lowered by 4m before excavating 4m of sand. Further anchors, again spaced at 3m intervals, are installed, the water level lowered by a further 3m and the final 3m of sand excavated. Using symmetry only half of the problem is analysed. Figure 1 shows details of the excavation to be modelled.

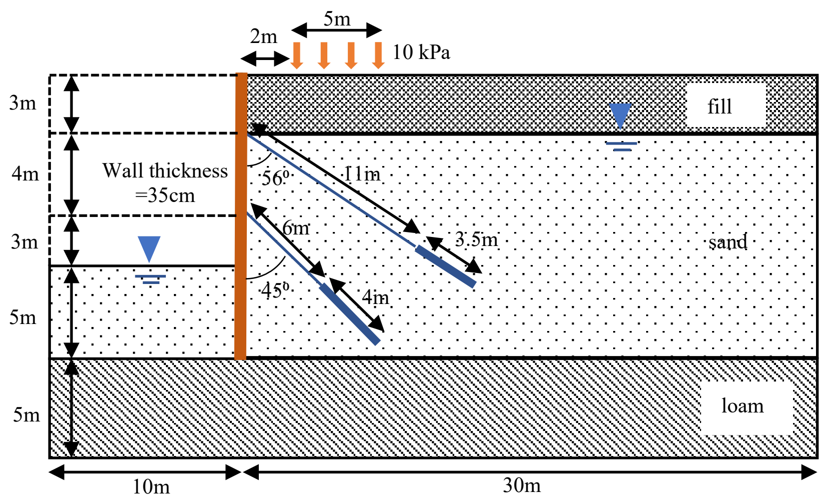


Figure 1: Excavation to be modelled

Objectives

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

Keywords

2D, Inplane, Geotechnical, Excavation, Staged Construction, Two phase elements, Activation, Deactivation, Interface elements, Water pressure, Displacement, Calculation of bending moment, axial and shear forces from 2D stresses.

Associated Files

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

- ❑ **dry_excavation.lvb** carries out the modelling of the example.

Preparing the Model Features

For this example, it is assumed that you have a basic knowledge of how to define attributes and assign them to the model. Other operations specific to this problem are covered in detail.

Running LUSAS Modeller

For details of how to run LUSAS Modeller, see the heading ***Running LUSAS Modeller*** in the *Introduction to LUSAS Worked Examples* document.

Note. This example is written assuming a new LUSAS Modeller session has been started. If continuing from an existing Modeller session select the menu command **File > New** to start a new model file. Modeller will prompt for any unsaved data and display the New Model dialog.

Creating a new model

- Enter the file name as **dry_excavation**
- Use the default user-defined working folder.
- Select an Analysis Category of **2D Inplane**
- Set the model units to **kN,m,t,s,C**
- Ensure the timescale units are **Days**
- Enter the title as **Dry_excavation**
- Click the **OK** button.

> **Note.** In this guided example, menu selections are shown within the body of the text like this: **Geometry > Points > By Coords…** as opposed to being shown in the left-hand side of each page as seen in other worked examples.

Feature Geometry

- Use the menu item **Geometry > Points > By Coords…** to define the points along the centreline that represent the soil and exaction levels (figure 2).

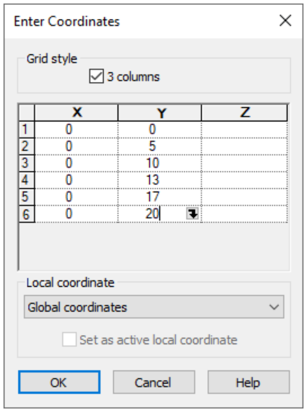


Figure 2: Input of coordinates

- Once defined, select all the points.
- Select the menu item **Geometry > Point > Copy…** and enter an X translation of **10** and click **OK**.
- Then without deselecting the points, re-select the same menu item, enter an X translation of **40** and click **OK**. (figure 3).

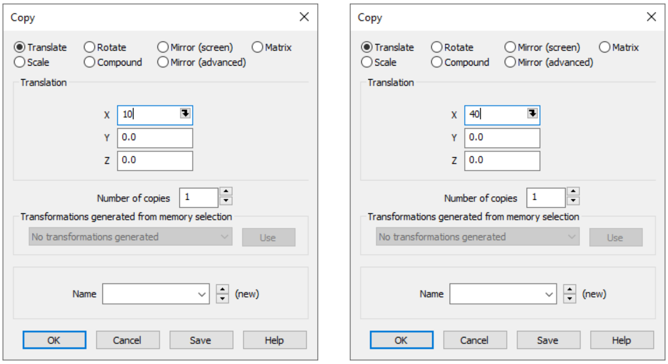


Figure 3: Make copies of points on the centreline

The ground anchors are defined using local coordinate systems.

- Select the point at (10,17) to define the origin of the new coordinate system,
- Select **Attributes > Local Coordinate…** then select the radio button **Rotate**, enter an angle of **-34** in the angle field (the angle is measured anticlockwise about the z-axis) and enter a name of **upper anchor** (figure 4). Then click **OK**.

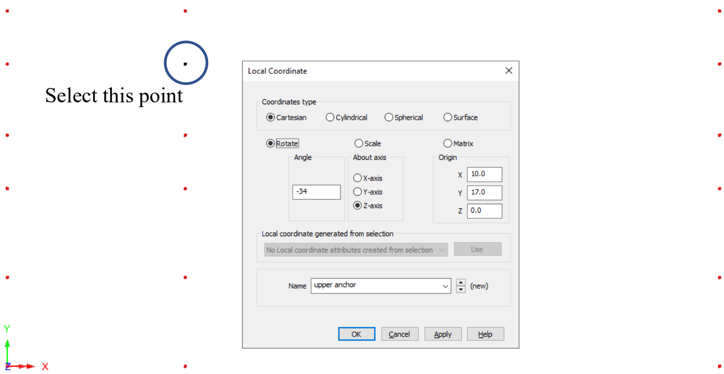


Figure 4: Definition of local coordinate system for upper anchor

The coordinates of the grouted part of the anchor are entered in this local coordinate system.

- Select **Geometry > Point > By Coords...** Click on the dropbox containing **Global Coordinates** and select **upper anchor**. Enter the local coordinates (**11,0**) and (**14.5,0**) and click **OK** (figure 5).

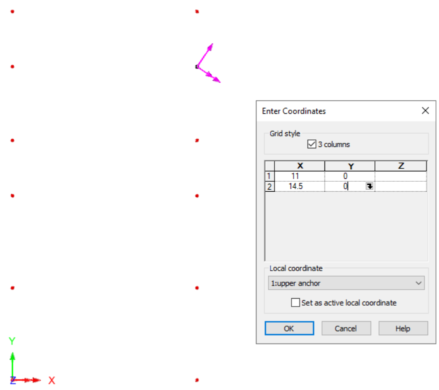


Figure 5: Input of coordinates of upper anchor

The process is repeated for the lower anchor.

- Select the point (**10,13**)
- Select **Attributes > Local Coordinate…** and choose the radio button **Rotate**, enter an angle of **-45** in the angle field, a name of **lower anchor** (figure 6). Then click **OK**.

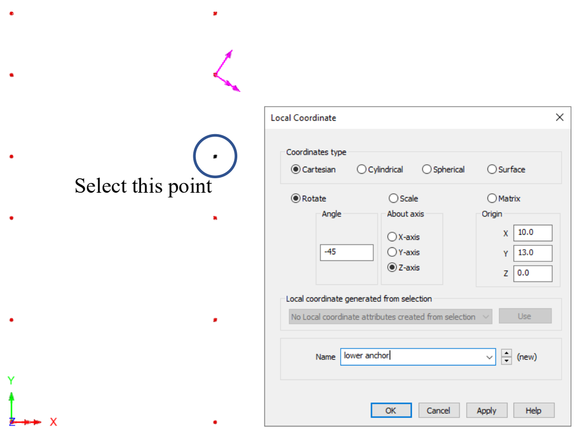


Figure 6: Definition of local coordinate system for lower anchor

The coordinates of the grouted part of the anchor are entered using the local coordinate system.

- Select **Geometry > Point > By Coords…** then click on the dropbox containing **Global Coordinates** and select **lower anchor**. Enter the local coordinates (**6,0**) and (**10,0**) and click **OK**.

- In the Attributes treeview, right-click on **Local Coordinate** and select **Hide all definitions** (figure 7) to hide the local coordinate axes.

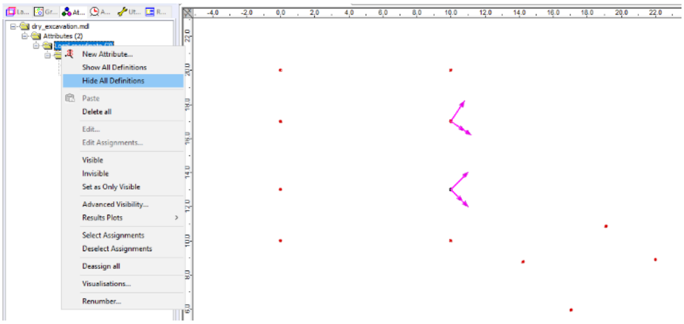


Figure 7: Hide local coordinate systems

The coordinates for the wall and edge of the soil in the excavation are to be defined next.

Note. To aid with the assignment of attributes between the wall and the soil mass, the points at the interface between these features will be modelled out of position 'expanded' and then after all assignments have been made, moved back into their correct location.

- Select the four nodes shown in figure 8.

- Select **Geometry > Point > Copy…** and enter a distance of **-1** in X and set **Number of copies** to **2**. Click **OK**.

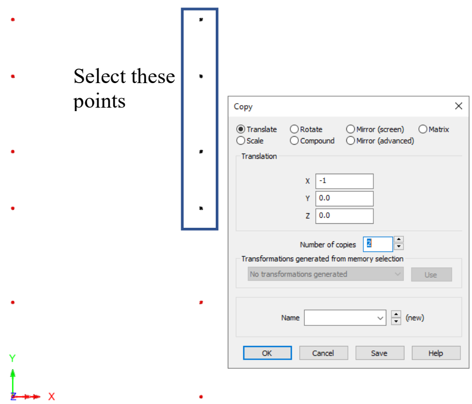


Figure 8: Definition of points for wall and excavation soil

It would be convenient to define the points for the surface load at this stage, but this is done later to show how to split a line using points.

-  Set the cursor to select just points.
- Select **Geometry** > **Surface** > **By Points…** to define the soil surfaces shown in figure 9. Note that an interior surface is generated from the points which define the anchor grout.

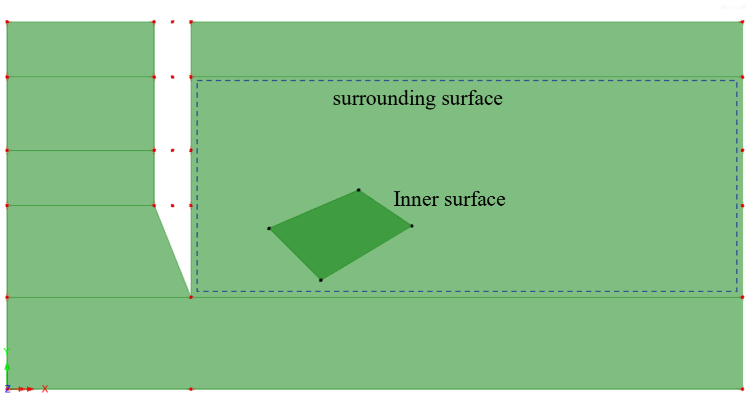


Figure 9: Definition of surfaces

-  Change the cursor to select surfaces,
- Select both the inner surface and surrounding surface. (figure 10)

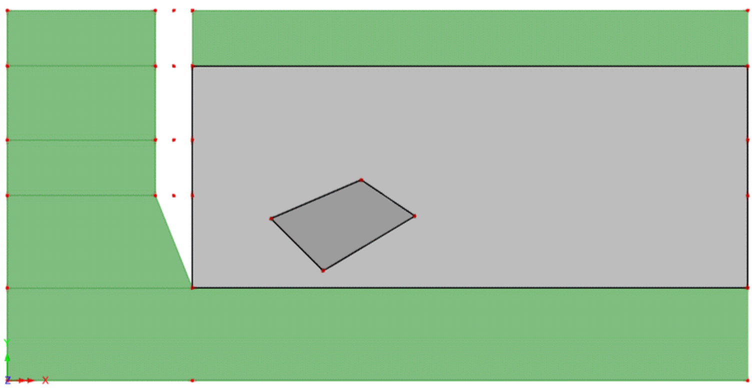


Figure 10: Definition of hole

- Select **Geometry** > **Surfaces** > **Holes** > **Create…** and uncheck the tickbox **Delete geometry defining holes** and click **OK**.

-  Set the cursor to select just points.
- Define the lines for the wall and anchor rods using **Geometry > Line > By Points…** (figure 11). Note the anchor rods are connected to the wall.

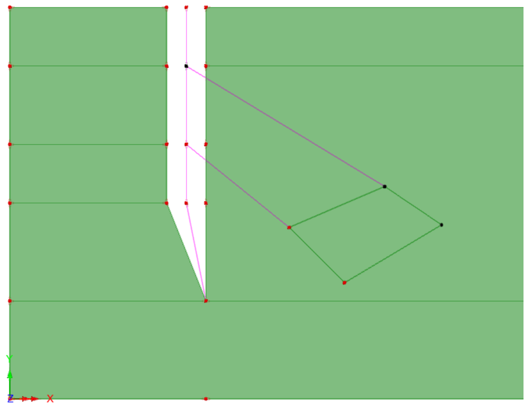


Figure 11: Lines defining wall and anchor rods

- To define the points between which the surface load is applied, select the point (10,20) and copy it **2**m to the right using **Geometry > Point > Copy…** (figure 12). Copy again to generate a point **7**m to the right.

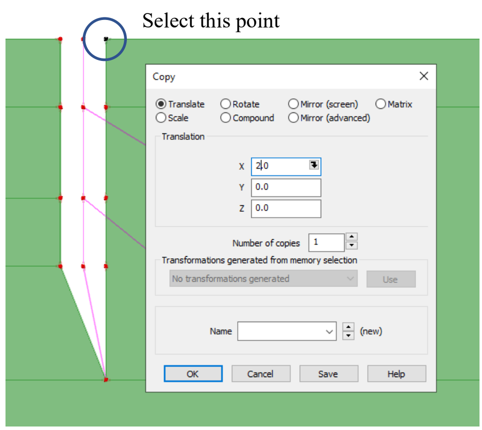


Figure 12: Copy point for load position

The points are now used to break the surface boundary line into three lines.

- Select the line and the two points, then use **Geometry > Line > By Splitting> At a Point..**. Leave the defaults and click **OK** (figure 13).

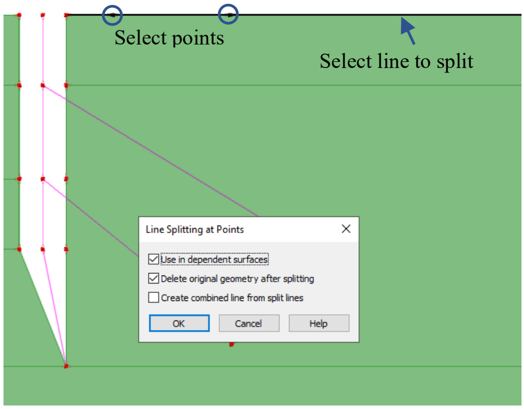


Figure 13: Splitting line for surface loading

The position of phreatic surfaces can be defined by points or lines in two dimensional problems. In this case, three phreatic surfaces are required to define the flow boundary conditions. The first to position the ground water, the second to lower the water before excavating the sand and the third for lowering the water before the final excavation. The use of two phreatic surfaces to model the lowering of the water level in the excavation is required to simplify the application of automatic loading to the problem.

- Select points on the centreline at (0,13) and (0,17).
- Using **Geometry > Point > Copy…** set the X field to **-1** and click **OK** (figure 14).
- Copy the points again, this time setting the X field to **-3**.
- Select the point at (40,17).
- Using **Geometry > Point > Copy…** set the X field to **1** and click **OK** (figure 15).
- Copy the point again this time setting the X field to **3**.

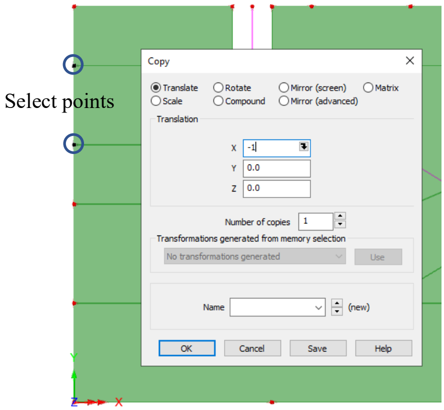


Figure 14: Copy points for phreatic surfaces in excavation

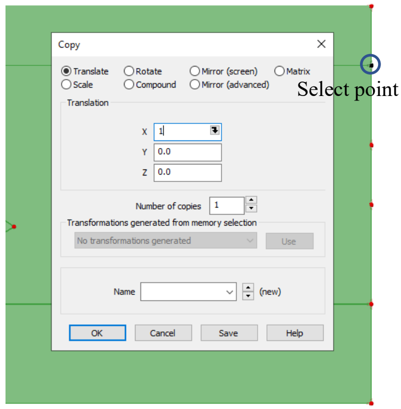


Figure 15: Copy point for phreatic surface defining groundwater

- Finally, select pairs of points and form the three phreatic surfaces using **Geometry > Line > By Points…**. Figure 16 shows the final geometry.

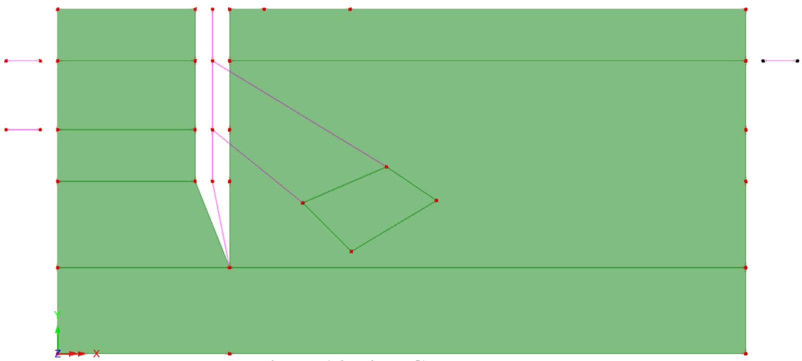


Figure 16: Final Geometry

Defining the geometric properties

The anchor rod has a diameter of 3cm and the grout a diameter of 60cm.

- Using **Attributes > Geometry > Line…** Select **Bar** from the **Usage** dropdown list. Enter **7.07x10^{-4}** for the area and **Anchor Rod** in the name field. Then click **Apply** (figure 17).

- Change the area to be **0.28** and change the name to **Anchor Grout** and click **Apply** (figure 18).

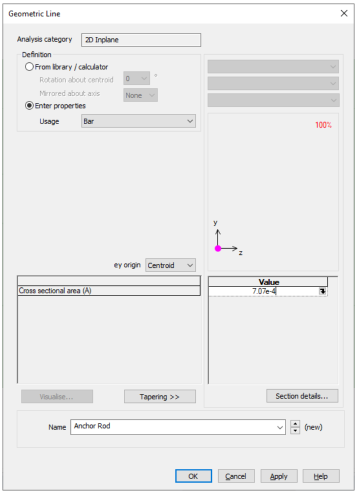


Figure 17: Anchor Rod geometric properties

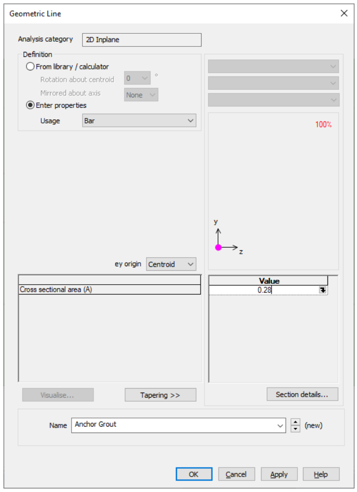


Figure 18: Anchor grout geometric properties

- Now, change the **Usage** dropdown list selection to **Plane Strain Beam** and enter **0.35** for the inplane thickness.

- Change the name to **Wall**, then click **OK** (figure 19).

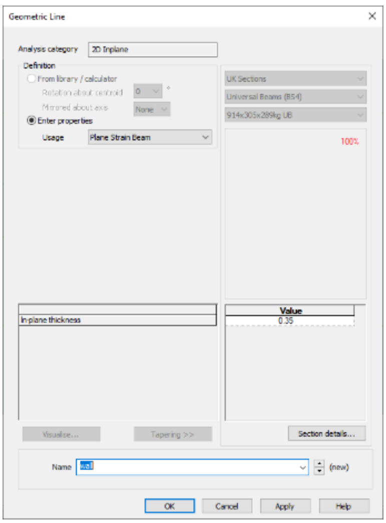


Figure 19: Wall geometric properties

- Assign the **Wall**, **Anchor Rod** and **Anchor Grout** properties in turn to the relevant selected features in the model, (figure 20).

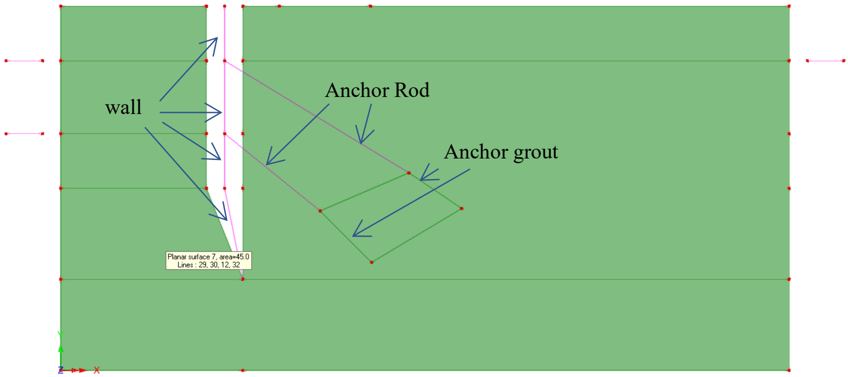


Figure 20: Assignment of geometric properties to lines

Defining the Mesh

Soil

Two-phase plane strain quadrilateral elements are used for the soil with an element size of 1m.

- Select **Attributes > Mesh > Surface…** to define the mesh for the soil.
- Choose **Plane strain two phase** from the Element type dropbox and **Quadrilateral** for the Element shape.
- Set the **Irregular mesh radio button**, check the **Element size** tickbox, enter **1.0** into the element size field. Enter the name to be **soil** and click **OK** (figure 21).

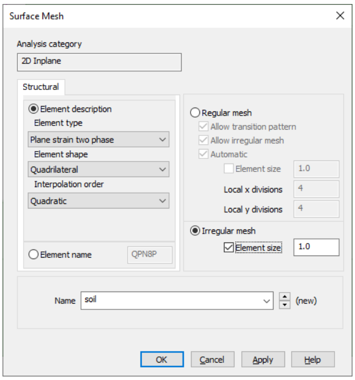


Figure 21: Definition of soil mesh

Wall

- Now select **Attributes > Mesh > Line…** to define the mesh for the wall. Define a **Plain strain beam** of **Quadratic** order with an element length of **1** and click **OK** (figure 22).

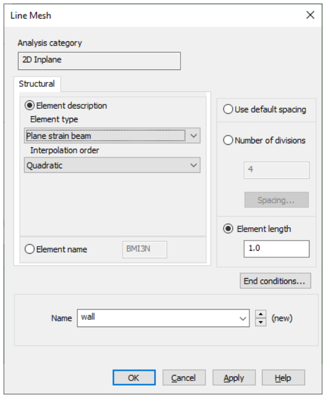


Figure 22: Definition of wall mesh

Anchors

Note. A single two noded bar element is used to model the anchor rod: it is not possible to use three-noded bars or more than one bar element without forming a mechanism. 1m long three-noded bars are used for the grout. The mesh details are shown in figures 23 and 24.

- For the anchor rod, select **Attributes > Mesh > Line…** Define a **Bar** of **Linear** order with a single division and click **Apply**.

- For the anchor grout, change to a **Quadratic** order and specify an element length of **1.0** and click **OK** (figure 22)

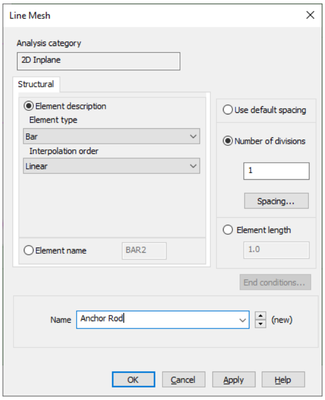


Figure 23: Anchor Rod mesh

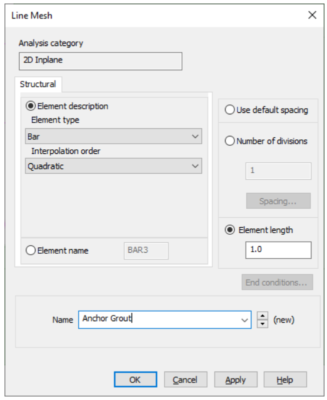


Figure 24: Anchor Grout mesh

Manually assigned two phase interfaces of length 1m also need to be defined.

- Select **Attributes** > **Mesh** > **Line…** to define the interface mesh. Define an **Interface (two phase)** element type of **Quadratic** order with an element length of **1** (figure 25)

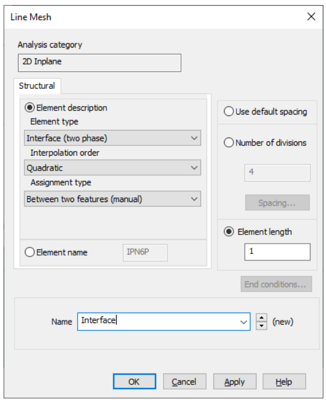


Figure 25: Interface mesh

- Select all the surfaces and assign the **soil** mesh attribute.

- Then, in turn, assign the **wall**, **Anchor Rod** and **Anchor Grout** line mesh attributes to the appropriate features (figure 26).

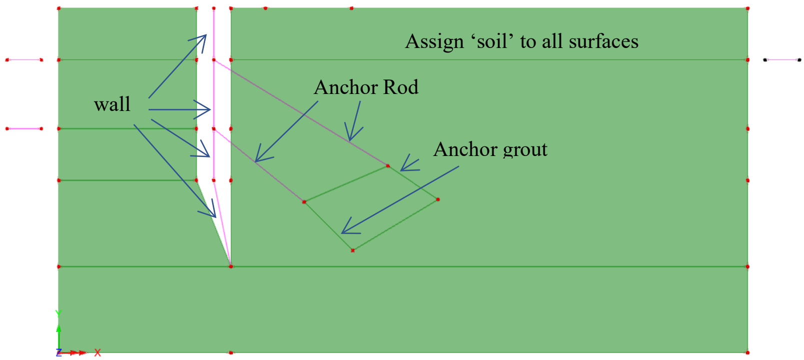


Figure 26: Assignment of mesh attributes

To mesh the interface elements between the soil and the wall, the Secondary Assignment of the interface is selected first.

- Select the four lines representing the wall in the order shown in figure 27, then, right-click in the model view window and choose **Selection Memory > Set**.
- Now select the four lines marking the boundary of the soil, the Primary Assignment, in the order shown in figure 27. Then assign the **Interface** mesh.

Note. If the Selection Memory is not used, the Primary Assignment is selected first, and each pair of lines must be processed one at a time.

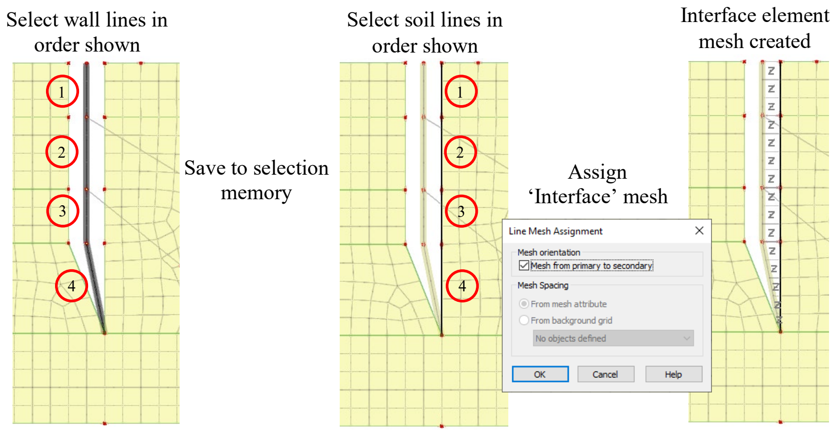


Figure 27: Assignment of interface mesh 1

Now, select the four lines marking the boundary of the soil on the excavation side of the wall in the order shown in figure 28 and assign the **Interface** mesh.

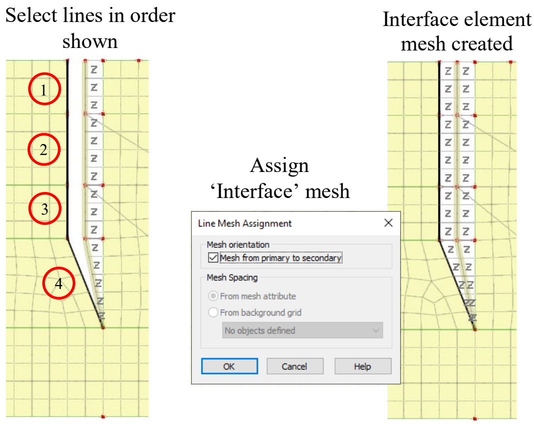


Figure 28: Assignment of interface mesh 2

Note. The lines defining the wall are chosen as Secondary Assignments because they already have the beam elements and wall thickness assigned to them. The lines on the soil side have no assignments, so the interface material properties can be assigned to them. They are designated as the Primary Assignments for the interface mesh.

Defining the materials

Fill, sand and loam

The fill, sand and loam are modelled by the Modified Mohr-Coulomb with Rankine cutoff material.

- For each soil material select **Attribute** > **Materials** > **Geotechnical** > **Mohr-Coulomb…** and make the appropriate selections according to table 1.

Dialog tab	**Parameter**	**Fill**	**Sand**	**Loam**
Elastic	Young's Modulus	**8,000** kPa	**30,000** kPa	20,000 kPa
Elastic	Poisson's ratio	**0.3**	**0.3**	0.33
Elastic	Mass density (Fully Saturated)	**2.0**	**2.0**	**1.9**
Plastic	Friction angle	**30.0º**	**34.0º**	**29.0º**
Plastic	Dilation angle	**0.0º**	**4.0º**	**0.0º**
Plastic	Cohesion	**1.0** kPa	1.0 kPa	8.0º
Plastic (Rankine)	Tensile cut-off stress	**0.0** kPa	**0.0** kPa	**0.0** kPa

Dialog tab	Parameter	Fill	Sand	Loam
Plastic (Rankine)	Compressive cut-off stress	**1E30** kPa	**1E30** kPa	1E30 kPa
Plastic (Rankine)	Damping factor	**0**		
Elastic (once Two-phase tab is selected)	Fully saturated soil density	**2.0** tonnes/m^3	**2.0** tonnes/m^3	**1.9** tonnes/m^3
Two phase (Partially saturated, Saturation)	Bulk modulus of fluid phase	**2.1** MPa	**2.1** MPa	**2.1** MPa
	Porosity of medium	**0.4**	**0.3**	**0.2**
	Hydraulic conductivity in global X, Y, Z	**0.5** m/day	**1.0** m/day	**0.1** m/day
	Density of fluid	**1** tonne/m^3	**1** tonne/m^3	**1** tonne/m^3
	Saturation at residual water content	**0.0**	**0.0**	**0.0**
	Saturation at full water content	**1.0**	**1.0**	**1.0**
(Constant water content)	Permeability factor in partially saturated zone	**1x10^{-3}**	**1x10^{-3}**	**1x10^{-3}**
Ko Initialisation	Coefficient of lateral earth pressure K_0	**0.5**	**0.441**	**0.515**

Table 1: soil material properties

Wall and ground anchor materials

The wall and anchors are modelled by linear elastic materials.

- For each material select **Attributes > Material > Isotropic…** and make the appropriate selections according to table 2.

Dialog tab	Parameter	Wall	Anchor Rod	Anchor Grout
Elastic	Young's Modulus	**35** GPa	**66.7** GPa -See note	**83,000** kPa - See note
Elastic	Poisson's ratio	**0.15**	**0.0**	**0.0**
Elastic	Density	**2.4** tonnes/m^3	-	-

Table 2: wall and anchor material properties

Note. The values of the ground anchors' Young's moduli have been modified by dividing by 3, the horizontal distance between the anchors in metres. Plane strain elements model a 1m length of the excavation, so the loads carried by the anchors from the wall are only one third of those actually applied. As the loads are only one third, the anchor rod's Young's modulus is reduced by a factor of 3 to correctly model the displacements.

Wall / Soil interface properties

In addition, we need the properties at the wall/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.

- For each interface material select **Attributes > Material > Geotechnical > Interface…** and select **Mohr-Coulomb Friction Interface** and click **Next**. Select **Two-phase** and make the appropriate selections according to table 3.

Dialog tab	Parameter	wall/fill	wall/sand
Soil structure interface (tangential slip)	Angle of friction	**20º**	**24º**
Soil structure interface (tangential slip)	Dilatency angle	0	0
Soil structure interface (tangential slip)	Cohesion	**0** kPa	**0.0** m/s
Two Phase	Hydraulic conductivity	**0** kPa	**0.0** m/s

Table 3: Tangential slip interface properties

Assigning the materials

- Assign the sand, fill and loam materials, in turn, by selecting the appropriate surfaces of the model (figure 29).
- Assign the anchor properties, in turn, by selecting the lines representing the rods and grout.

- Assign the interface material properties to the primary sides of the interface. To confirm which are the primary sides before assigning the interface materials, in the Attributes treeview, right-click on the Mesh/Line attribute **Interface** and then on **Select Primary Assignments** as shown in figure 30.

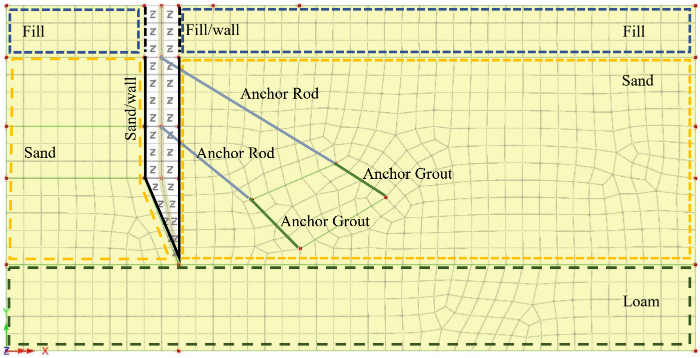


Figure 29: Material Assignments

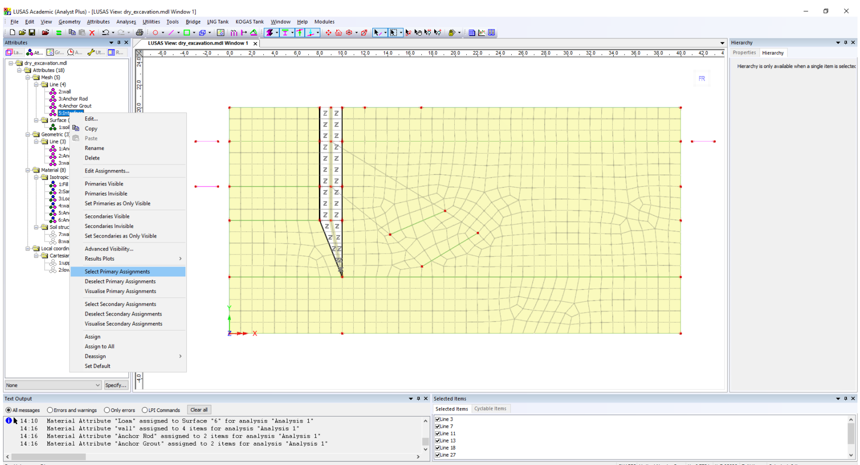


Figure 30: Interface Primary Assignments

Defining the Supports

The base of the model is to be constrained in both X and Y directions, while the lateral sides are limited in the X direction (Figure 31).

- Define the appropriate supports using **Attributes > Support…** and assign to appropriate features in the model.

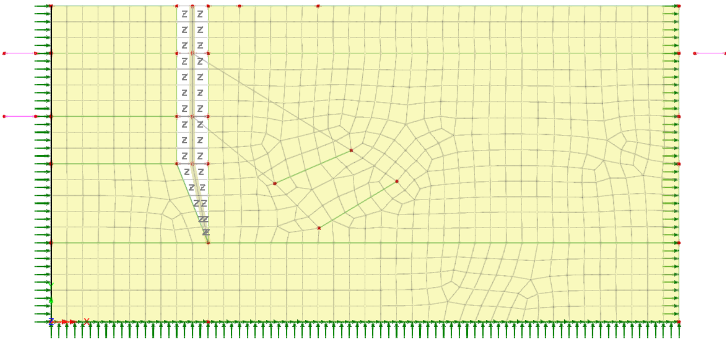


Figure 31: Displacement boundary conditions

Defining the Loads

Surface loading

- The 10 kPa surface load is defined using **Attributes** > **Loading…** and clicking the radio button **Distributed loads** followed by the radio button **Face** and then click **Next**. Enter **10** in the y-direction field. Enter a name of **Surf.load 10kPa** and click **Finish** (figure 32).

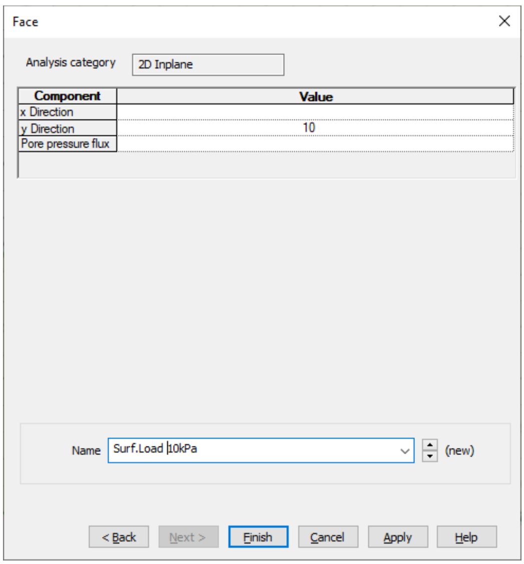


Figure 32: Definition of surface load attribute

Note. 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.

Fluid flow conditions

In addition to the surface load, we need to define the fluid flow conditions. This is done using phreatic surfaces. Three attributes are required.

- Select **Attributes > Pore Water Pressures > Phreatic Surface…** to define a phreatic surface attribute. Enter a name of **Groundwater** and click **Apply** (figure 33), then change the name to **1st de-watering** and click **Apply**, then change the name to **2nd de-watering** and click **OK**.

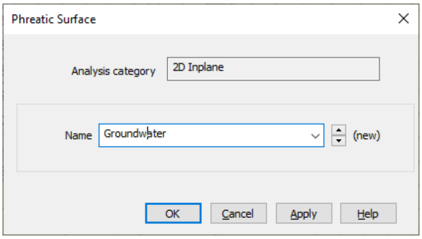


Figure 33: Definition of phreatic surface attribute

- Assign each phreatic surface attribute, in turn, to the relevant lines defined earlier (figure 34).

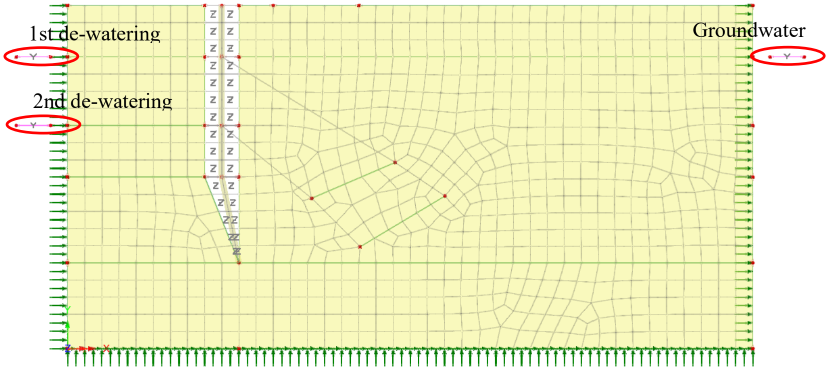


Figure 34: Phreatic surface assignments

Total prescribed displacements are used to lower the level of the phreatic surfaces to model the de-watering.

- Select **Attributes > Loading…** then the radio button **Displacement, velocity, acceleration, body force** followed by the radio button **Prescribed Displacement** and then click on **Next**. Click the radio button **Total**, then the radio button **Translation in Y** and enter **-4** in the **Displacement** field. Enter a name of **1st de-water** and click **Apply** (figure 35).

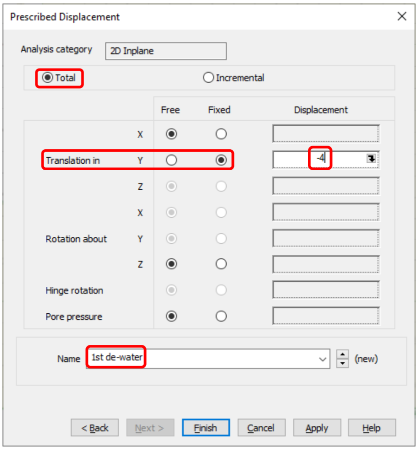


Figure 35: Definition of prescribed displacement attribute

- Change the **Displacement** to **-3** and the name to **2nd de-water** and click **Finish**.

The phreatic surfaces are used to set the pore water pressures at different parts of the model via phreatic loads.

- Select **Attributes > Loading…** then the radio button **Distributed Loads** followed by the radio button **Water Pressure Distribution** and then click on **Next**. Click on the radio button **Calculated from phreatic surface** and select **Groundwater** from the dropbox. Click on the radio button **Assigned to faces** and uncheck the **Include face pressure** tickbox. Enter a name of **Groundwater pwp** then click **Apply** (figure 36).

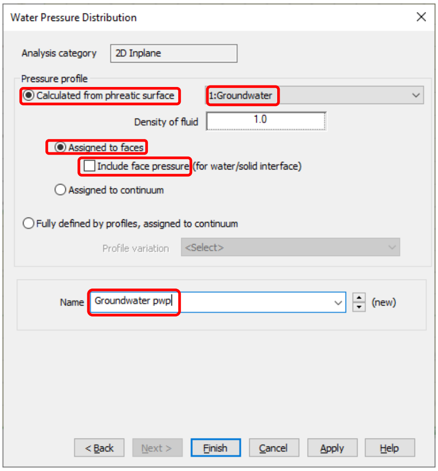


Figure 36: Definition of pressure attribute

- In the dropbox select **1st de-watering**, then change the name to be **1st de-watering pwp** and click **Apply**.
- In the dropbox select **2nd de-watering**, then change the name to be **2nd de-watering pwp** and click **Finish**.

Anchor forces

The anchor forces are 360kN in the upper anchor and 600kN in the lower one. As we reduced the Young's moduli of the rod and grout by a factor of 3, the forces must also be reduced by a factor of 3, to give 120kN and 200kN respectively.

The anchor forces are defined using **Attributes > Loading….**

- Click the radio button **Strain and stress** followed by the radio button **Stress and strain** and then **Next**. In the dialog ensure that the **Element Description** radio button is on, that the **Line** radio button is on, that **Bar** is visible in the dropdown list and that the **Target** radio button is on. Check the **Fx** tickbox and enter **120** in the value field. Enter a name of **U.Anchor=120kN** and click **Apply** (figure 37).
- Then, in the **Fx** value field enter **200**. Change the name to be **L.Anchor=200kN** and click **Finish**.

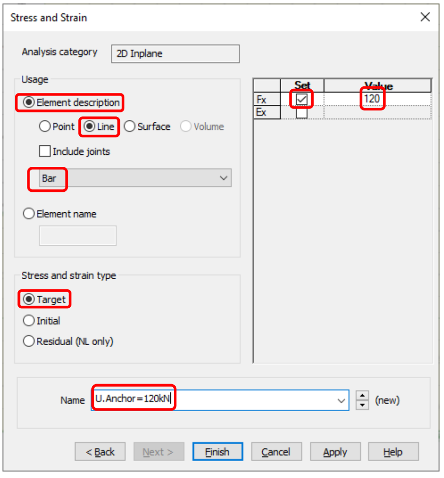


Figure 37: Definition of anchor forces

A final, dummy load is required for use during the excavations.

- Select **Attributes > Loading** then the radio button **Point Loads** followed by the radio button **Concentrated** and then **Next**. Enter a name of **dummy load for excavation** and click **Finish**. No other values are entered.

Deactivation Attributes

Various deactivation attributes are used during the analysis.

- Select **Attributes > Activate and deactivate …** Click on **Next** and enter a name of **wall and anchors**. Then select the **Custom inactive treatment** radio button and click on the ellipsis button **…** (figure 38)

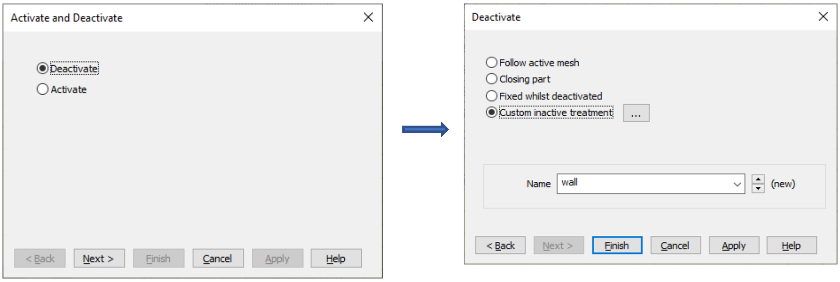


Figure 38: Definition of deactivation attribute

- Click on the **Distribute over stage** radio button and **OK**. (Figure 39)

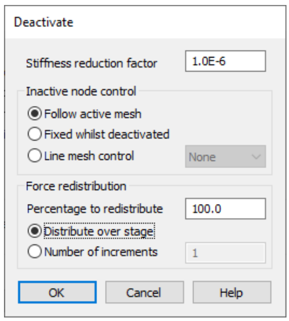


Figure 39: Set custom deactivation values

The option '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.

- On the parent dialog, click **Apply** to save the attribute.

For clarity in the assignments, we will generate identical attributes but with different names. So, with the dialog still displayed,

- Enter a name of **1st excavation** and click **Apply**, change it to **2nd excavation** and click **Apply,** and change it again to **3rd excavation** and click **Finish** to save the final attribute and close the dialog.

Activation attributes

We also need to create activate attributes for the wall and anchors.

- Select **Attributes > Activate and deactivate …** and select the radio button **Activate** then **Next**.

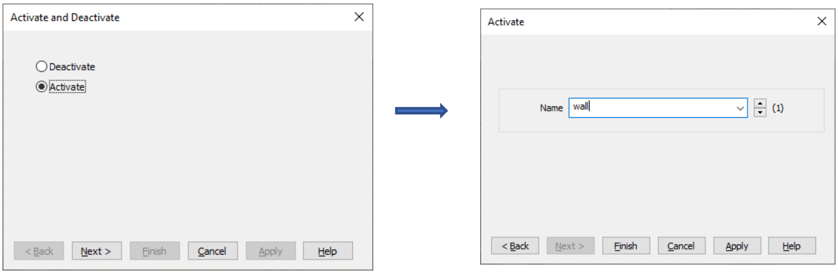


Figure 40: Definition of activate attribute

- Enter a name of **wall** and click **Apply** (figure 40).
- Change the name to be **upper anchor** and click **Apply**, then change it again to be **lower anchor** and click **Finish**.

Analyses to be considered

We consider the following stages.

Initial Phase

The first stage establishes the initial stress and water pressure distributions.

- In the Analyses  treeview, right-click on **Loadcase 1** and rename to it to be **Init.conds**.
- Now, select the wall and anchors and assign the deactivation attribute **wall and anchors** to them (Figure 41).

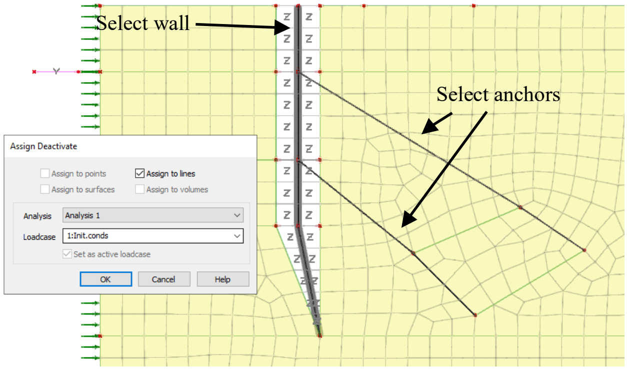


Figure 41: Deactivate wall and anchors

The interface elements are not deactivated as they are required to transfer the horizontal stresses across the gap made in the mesh by the wall.

- Now select all the lines on the right-hand boundary of the model and assign the **Groundwater pwp** attribute to them. Click on **More >>** and then the radio button **Specified loadcases** and then **OK** (figure 42).

The groundwater is applied throughout the solution but is not active in loadcases which use automatic loading. We will return once all the loadcases have been defined to select the manual loadcases in which the load is applied.

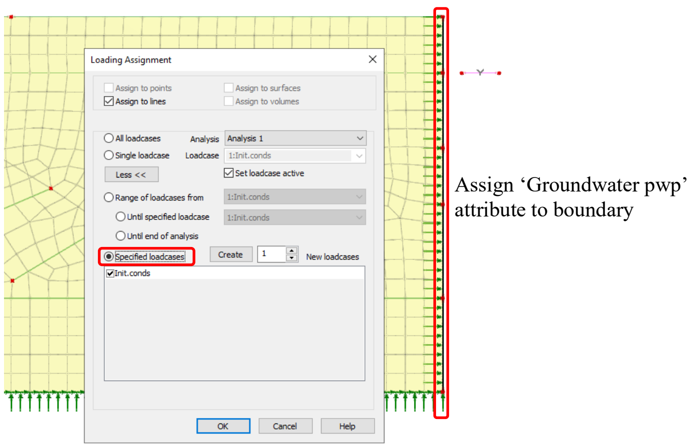
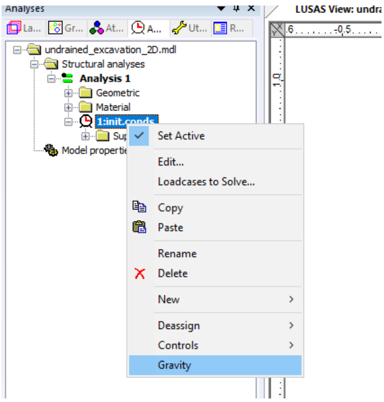


Figure 42: Setting groundwater conditions

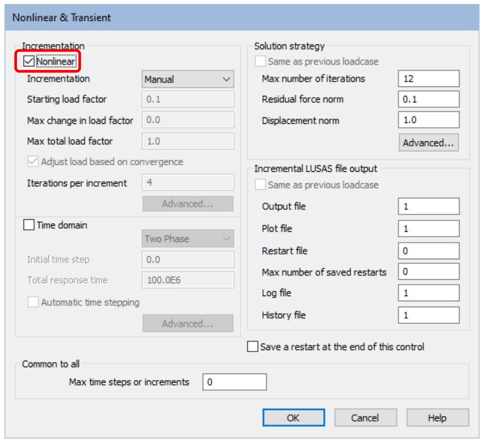
- In the Analyses treeview, right-click on the loadcase name **Init.conds** and select **Gravity** to add gravity to this loading stage (figure 43).

- Right-click again on loadcase **Init.conds** and select **Controls > Nonlinear and transient…**. Check the tickbox **Nonlinear** to inform Modeller than the analysis is nonlinear (figure 43).

The first stage is 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 43: Setting gravity and nonlinear control for the first stage

Installation of wall

In this stage the wall is activated.

- In the Analyses treeview, right-click on loadcase **Init.conds** and select **New > Loadcase…** Enter a name of **Install wall** and check the tickbox **Automatically add gravity to this loadcase**, and then the **OK** button.

- Select the wall and assign the **Activate Elements** attribute **wall** (figure 44).

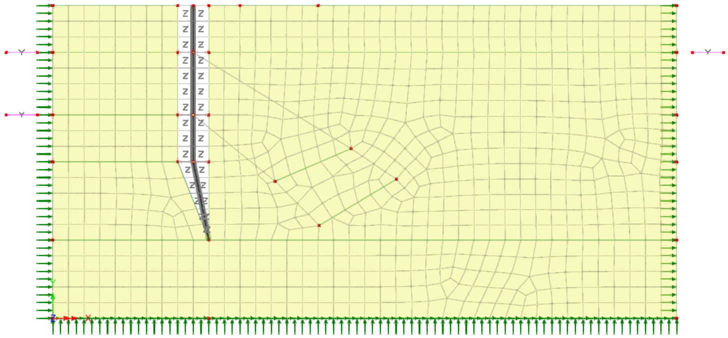


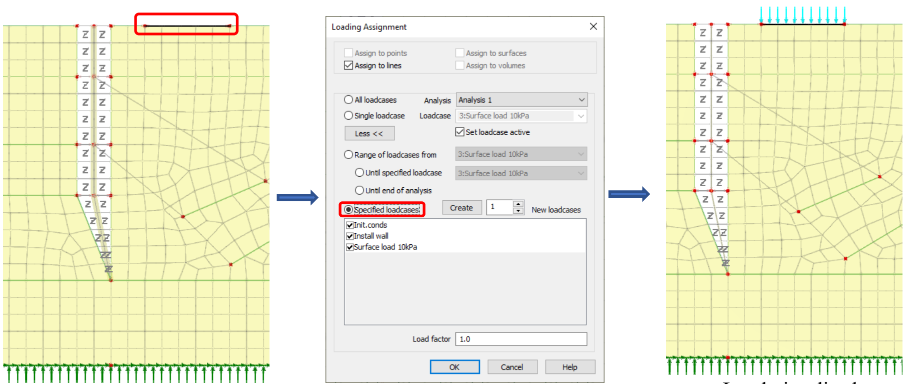
Figure 44: Installation of wall

Application of surface load

In this stage the surface load of 10 kPa is applied.

- In the Analyses treeview, right-click on **Install wall** and select **New > Loadcase…** In the dialog enter a name of **Surface load 10kPa** 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 10kPa** (figure 45). In the loading assignment dialog set the radio button for **Specified loadcases**. Click **OK**.



Select line where load is applied

Select **Specified loadcases**

Load visualised on surface

Figure 45: Application of surface load

1st Excavation

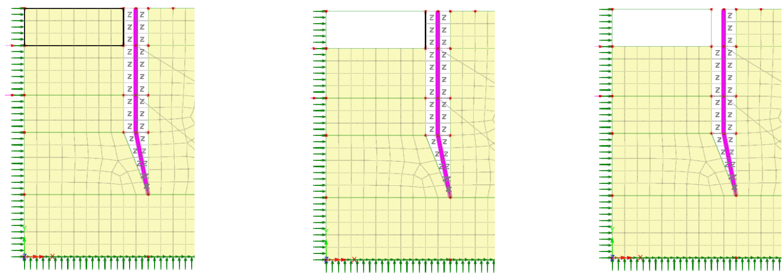
In this stage the first three metres of fill are excavated.

- In the Analyses treeview, right-click on **Surface load 10kPa** and select **New > Loadcase…** Enter a name of **First excavation** and click the **OK** button.

Note: The option **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 deactivation attribute **1st excavation** (figure 46).

- Now select the lines of the surface next to wall and assign the deactivation attribute **1st excavation** to deactivate the interface elements. This is important to stop unwanted mesh deformations when the soil is removed and the residual forces are reduced to zero.



Select surface to be deactivated | Select line to deactivate interface elements | Everything deactivated for 1st excavation

Figure 46: 1st Excavation

The excavation is done in increments over the stage using the automatic loading facility.

- In the Analyses treeview, right-click on **1st excavation** 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 47).

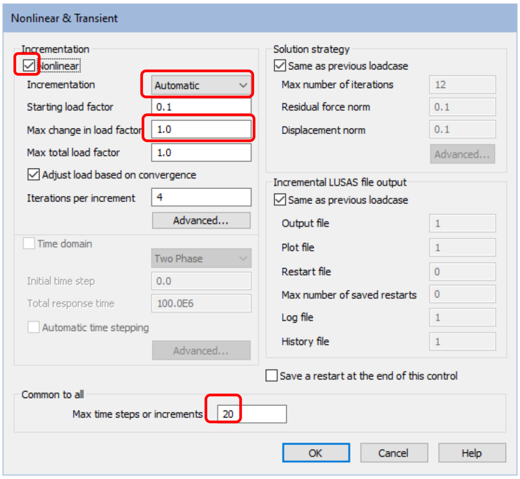
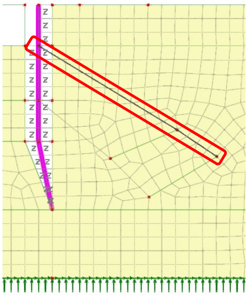


Figure 47: Nonlinear parameter settings

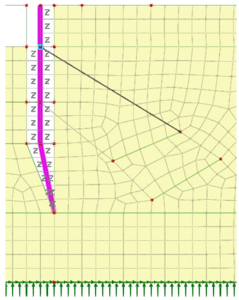
Install upper anchor

In this stage the upper anchor is activated.

- In the Analyses treeview, right-click on **1st excavation** and select **New > Loadcase…** Enter a name of **Install upper anchor** and click the **OK** button.
- Select the lines representing the upper anchor rod and grout and assign the **Activate Elements** attribute **upper anchor** (figure 48).
- Now, select the anchor rod and assign the load attribute **U.Anchor=120k**. In the load assignment ensure that the **Single loadcase** radio button is set. Click **OK**.



Select lines and assign ‘upper anchor’ activate attribute



Select line and assign ‘U.Anchor=120kN’ load attribute

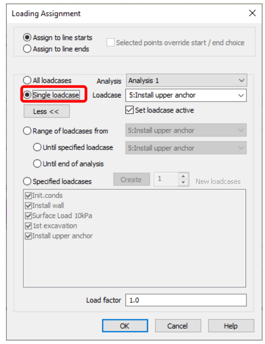


Figure 48: Installation of upper anchor

The anchor load is applied in increments.

- In the Analyses treeview, right-click **Install upper anchor** 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 47).

Set water pressure at base of 2nd excavation

The water level at base of the 2nd excavation is set as a prelude to de-watering.

- In the Analyses treeview, right-click on **Install upper anchor** and select **New > Loadcase…** In the dialog enter a name of **Set water level excav.2**, check the tickbox **Automatically add gravity to this loadcase**, and then click the **OK** button.

- Select the line at the bottom of the 2nd excavation and assign the Water pressure distribution Load attribute **1st de-watering pwp**. Ensure that the **Single loadcase** radio button is selected (figure 49).

The attribute causes the water pressure along the line to be calculated from the upper phreatic surface marked on the left of the model.

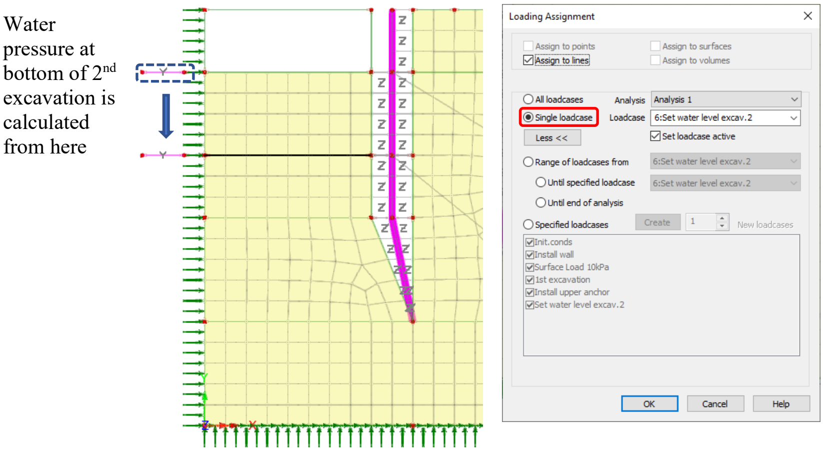


Figure 49: Setting water pressure at bottom of 2nd excavation

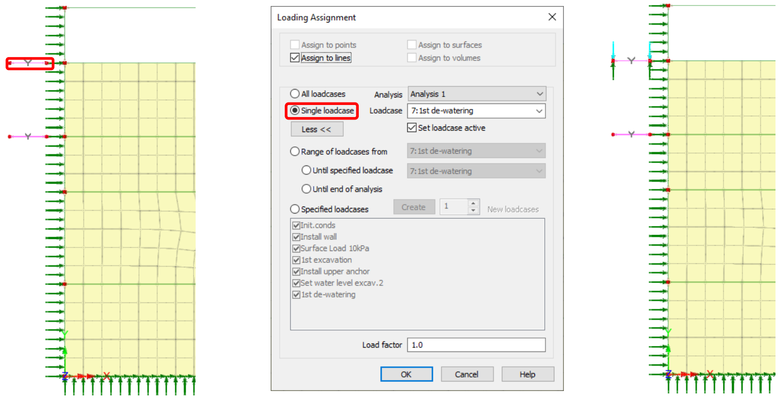
1st de-watering – lower water level 4m

The water level is lowered using automatic loading. If we use automatic loading on the water pressure load attribute set in the previous phase, we would scale the water density. In this phase we move the phreatic surface down by 4m.

- In the Analyses treeview, right-click on **Set water level excav.2** and select **New > Loadcase…** In the dialog enter a name of **1st de-watering** and click the **OK** button.

- Select the upper phreatic surface on the left and assign structural loading attribute **1st de-water**. Ensure that the **Single loadcase** radio button is selected and click **OK** (figure 50).

- In the Analyses treeview, right-click on **1st de-watering** 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 47).



Select phreatic surface

Assign '1st de-water' attribute

Total prescribed displacement is used to lower water level 4m

Figure 50: Lower water level in excavation

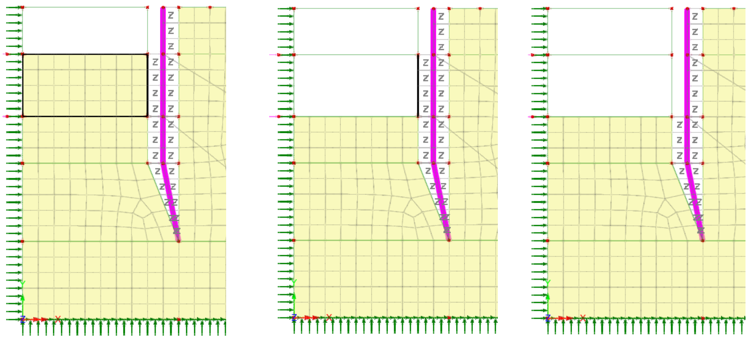
2nd Excavation

In this stage, the first four metres of sand are excavated.

- In the Analyses treeview, right-click on **1st de-watering** in the Treeview and select **New > Loadcase…** In the dialog **Name** field enter **2nd excavation** and then click the **OK** button.

- Select the surface of the second excavation and assign the deactivation attribute **2nd excavation**.

- Now select the line of the surface next to wall and assign the deactivation attribute **2nd excavation** to deactivate the interface elements (figure 51).



Select surface for 2nd excavation and assign Deactivate attribute

Select line to deactivate interface elements and assign Deactivate attribute

Everything deactivated for 2nd excavation

Figure 51: 2nd excavation

- Now select the upper righthand point (40,20) and assign the load **dummy load for excavation** and click on the radio button for **Specified loadcases** then **OK** (figure 52).

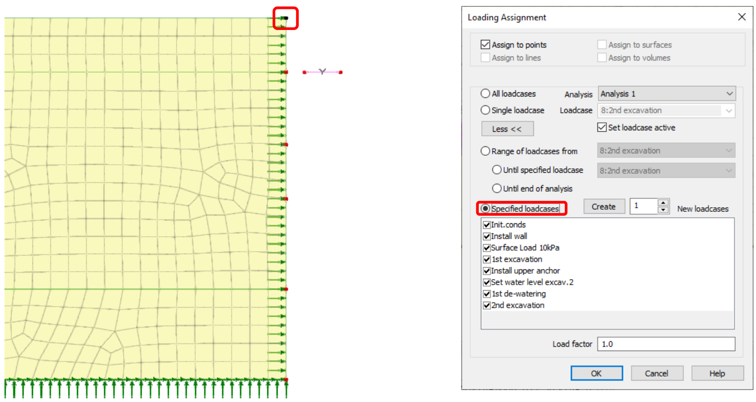


Figure 52: Assignment of dummy load for excavation

Note. Automatic loading is used to reduce the residual forces of the excavated soil. The deactivate attribute is not a load so the prescribed displacement used to lower the phreatic surface in the previous stage is still active and will continue to be factored by the automatic load scale parameter. The dummy load is used to replace the prescribed displacement in the automatic loading. An alternative to using a dummy load, would be to use a lock-in stage to reset the manual loads and zero the automatically scaled loads.

For the first excavation, the previous stage applies the surface load using manual loading and the dummy load is not required. The dummy load can be applied to any feature in the model that is active during the excavation to serve its purpose of clearing the existing automatic loading.

- In the Analyses treeview, right-click on the **2nd Excavation** and select **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 47).

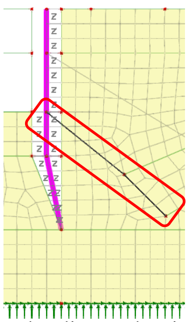
Install lower anchor

In this stage the lower anchor is activated.

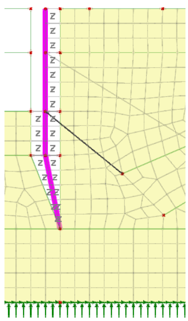
- In the Analyses treeview, right-click on **2nd excavation** and select **New > Loadcase…** In the dialog enter a name of **Install lower anchor** and then click the **OK** button.

- Select the lines representing the lower anchor rod and grout and assign the Activate Elements attribute **lower anchor** (figure 53).

- Now, select the anchor rod and assign the load attribute **L.Anchor=200kN**. In the load assignment ensure that the **Single loadcase** radio button is set and then click **OK**.



Select lines and assign ‘lower anchor’ activate attribute



Select line and assign ‘L.Anchor=200kN’ load attribute

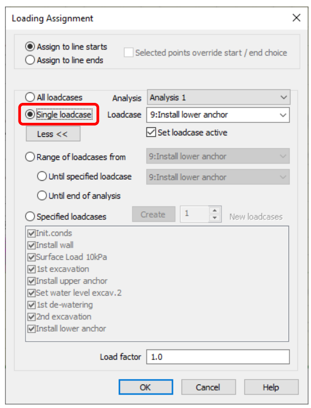


Figure 53: Installation of lower anchor

The anchor load is applied in increments.

- In the Analyses treeview, right-click on **Install lower anchor** 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 47).

Set water pressure at base of 3rd excavation

The water level at the base of the 3rd excavation is set as a prelude to de-watering.

- In the Analyses treeview, right-click on **Install lower anchor** and select **New > Loadcase…** In the dialog enter a name of **Set water level excav.3**, check the tickbox **Automatically add gravity to this loadcase**, and then click the **OK** button.

- Select the line at the bottom of the 3rd excavation and assign the Water pressure distribution Load attribute **2nd de-watering pwp**. Ensure that the **Single loadcase** radio button is selected (figure 54).

The attribute causes the water pressure along the line to be calculated from the lower phreatic surface marked on the left of the model.

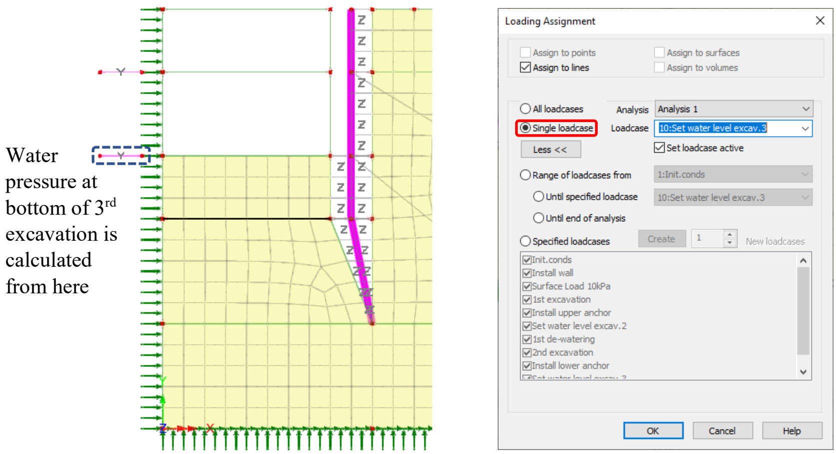


Figure 54: Setting water pressure at bottom of 3rd excavation

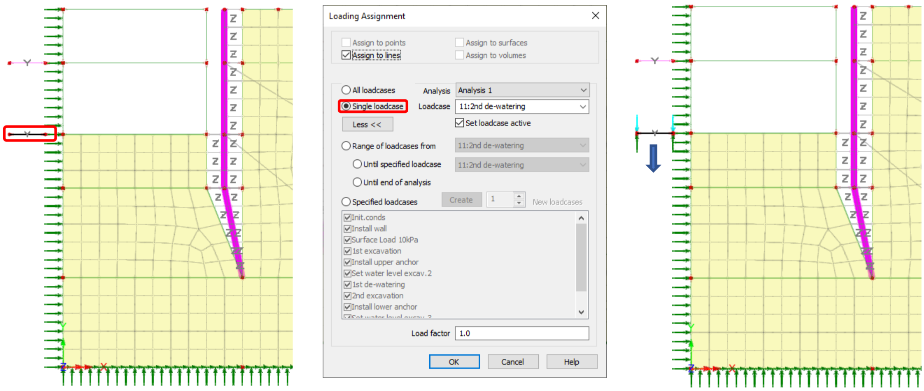
2nd de-watering – lower water level further 3m

In this phase we move the phreatic surface down by a further 3m.

- In the Analyses treeview, right-click on **Set water level excav.3** and select **New > Loadcase…** Enter a name of **2nd de-watering** and click the **OK** button.

- Select the lower phreatic surface on the left and assign structural loading attribute **2nd de-water**. Ensure that the **Single loadcase** radio button is selected (figure 55).

- In the Analyses treeview, right-click on **2nd de-watering** 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 47).



Apply prescribed displacement to phreatic surface

Assign '2nd de-water' attribute

Phreatic surface moves down

Figure 55: Lowering of phreatic surface

3rd Excavation

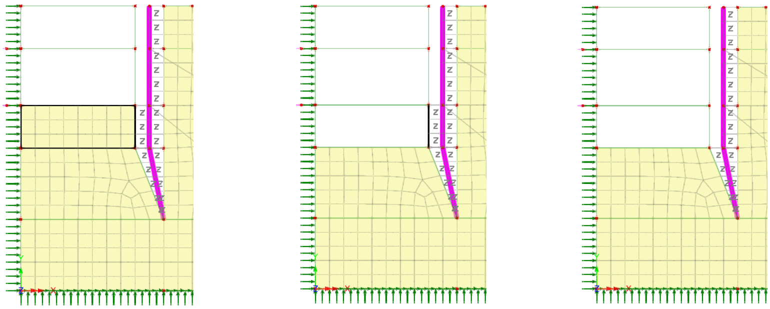
In this stage, the final three metres of sand are excavated.

- In the Analyses treeview, right-click on **2nd de-watering** and select **New > Loadcase…** In the dialog enter a name of **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 56).

- 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.

- In the Analyses treeview, right-click on **3rd excavation** 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 47).



Select surface for 3rd excavation and assign Deactivate attribute

Select line to deactivate interface elements and assign Deactivate attribute

Everything deactivated for 3rd excavation

Figure 56: 3rd Excavation

Final Preparations

All the analysis stages have now been defined. Now it's time to set the loads which appear at different stages of the analysis.

- In the Attributes treeview, right-click on the load attribute **Surf.Load 10kPa** and then click on **Edit Assignments…** Check the tick-boxes for loadcases **Surface Load 10kPa**, **Set water level excav.2** and **Set water level excav.3**. Make sure that the other tick-boxes are unchecked (figure 57). Click **OK**.

The surface load is applied in all the manual loadcases.

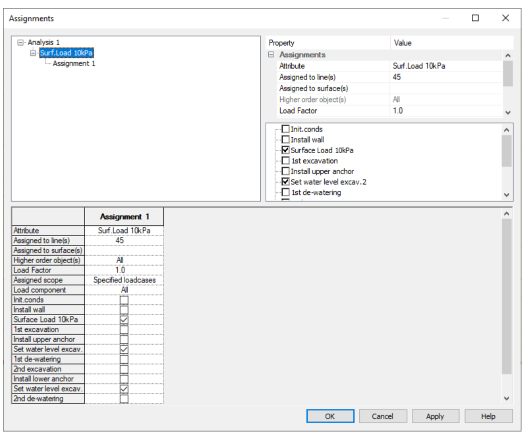


Figure 57: Selection of stages for the application of the surface load

- In the Attributes treeview, right-click on the load attribute **dummy load for excavation** and then click on **Edit Assignments…** Check the tick-boxes for loadcases **2nd excavation** and **3rd excavation**. Make sure that the other tick-boxes are unchecked (figure 58). Click **OK**.

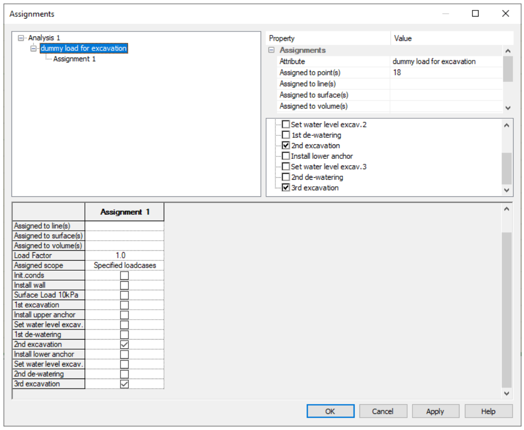


Figure 58: Selection of stages for the application of the dummy excavation load

- In the Attributes treeview, right-click on the load attribute **Groundwater pwp** and then click on **Edit Assignments…** Check the tick-boxes for loadcases **Init.conds**, **Install wall**, **Surface Load 10kPa**, **Set water level excav.2** and **Set water level excav.3**. Make sure that the other tick-boxes are unchecked (figure 59) then click **OK**.

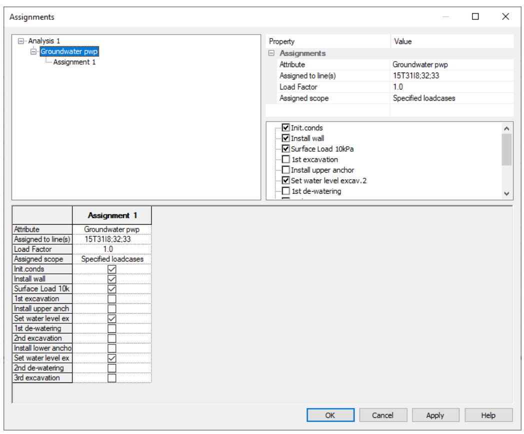


Figure 59: Selection of stages for the groundwater

Moving the wall into its correct position

The wall can now be moved into its correct position. First set the correct loadcase.

- In the Analyses treeview, right-click **on Init.conds** and click on **Set Active**.

To avoid the wall geometry merging with that of the soil after moving, the lines and points are made unmergeable.

- Select the 12 lines defining the wall and soil and then click on **Geometry** > **Line** > **Make Unmergeable**.
- Then select the 12 points as shown, and click on **Geometry** > **Point** > **Make Unmergeable** (figure 60).

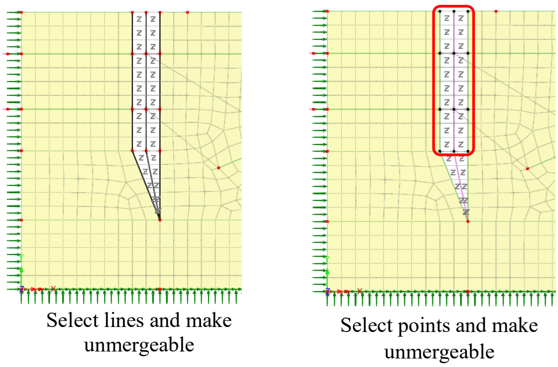
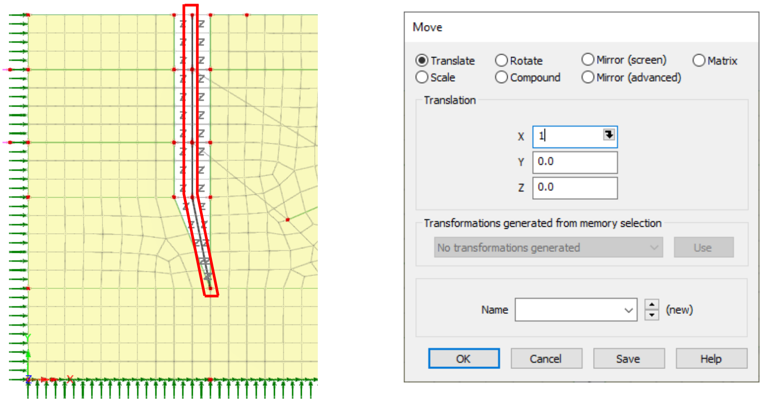


Figure 60: Make all geometry unmergeable before moving into place

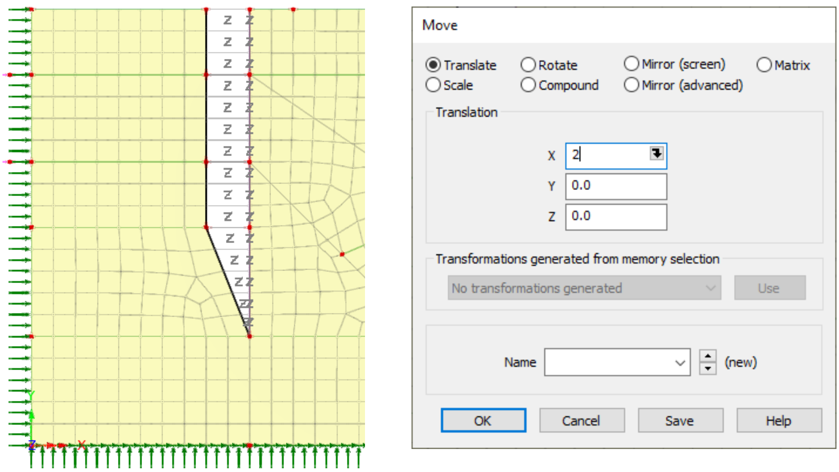
- Now select the four lines representing the wall and click on **Geometry** > **Line** > **Move…**. Enter a distance of **1** in the X direction and click **OK** (figure 61).



Select lines for wall and move 1m

Figure 61: Close gap between wall and soil

- Select the four lines representing the soil to the left of the wall and click on **Geometry > Line > Move…**. Enter a distance of **2** in the X direction and click **OK** (figure 62).



Select lines for soil and move 2m

Figure 62: Close gap between soil and wall

The model should now look as shown in figure 63.

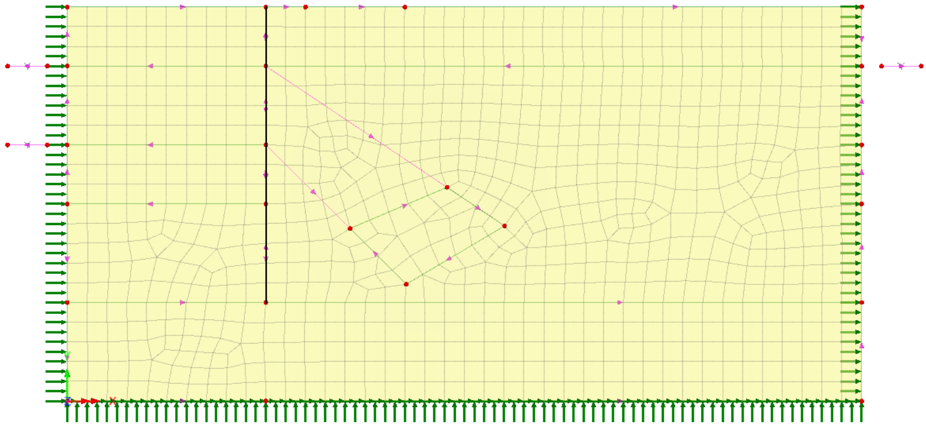


Figure 63: Model state for 'Init.conds'

Running the Analysis

 Open the **Solve Now** dialog.

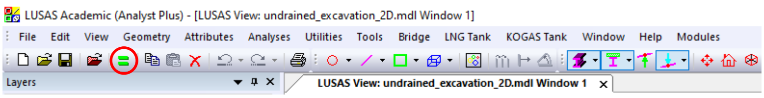


Figure 64: To run the analysis

- Then check the tickbox **Analysis 1** and click on **OK**, (Figure 65)

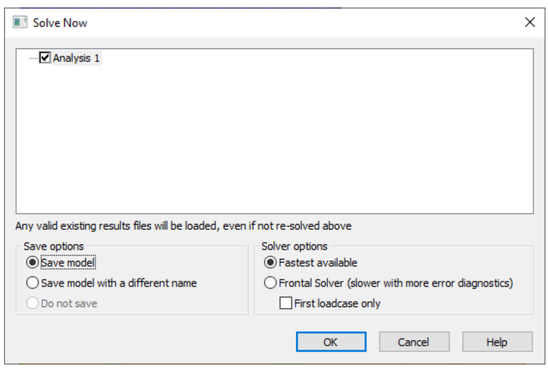


Figure 65: Solve Now dialog

A LUSAS Datafile will be created from the model information. The LUSAS Solver uses this datafile to perform the analysis.

If the analysis is successful...

Analysis loadcase results are added to the Treeview.

In addition, these files will be created in the LUSASFiles32\<model_name> folder:

- **dry_excavation.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- **dry_excavation.mys** this is the LUSAS results file which is loaded automatically into the Treeview to allow results processing to take place.

If the analysis fails...

If errors are listed that for some reason you cannot correct, a file is provided to re-create the model information correctly, allowing a subsequent analysis to be run successfully.

- **dry_excavation.lvb** carries out the complete modelling of the example.

File
New…

Start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **dry_excavation**
- Use the default **User-defined** working folder.
- Ensure an Analysis category of **2D Inplane** is set.
- Click the **OK** button.

Note. There is no need to enter any other new model details when a script is run to build a model, since the contents of the script will overwrite any other settings made.

File
Script >
Run Script...

To recreate the model, select the file **dry_excavation.lvb** that was downloaded and placed in a folder of your choosing.

Rerun the analysis to generate the results.

Viewing the Results

Analysis loadcase results are present in the Analyses Treeview.

- In the Analyses treeview, in the loadcase named **3rd excavation**, right-click on the final increment having a load factor of **1.0** and then click on **Set Active** (figure 66).

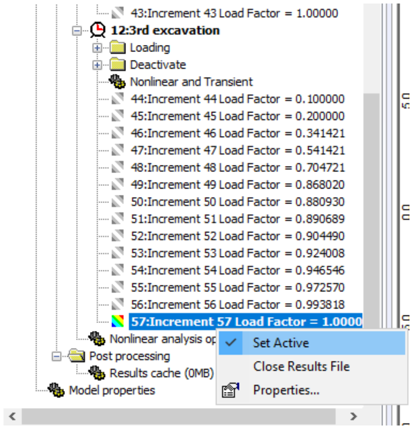


Figure 66: Setting the active results

Note. Results increments and hence results may differ marginally due to any slight differences in meshing.

To simplify the display, the 'Z' symbols marking the position of the interface elements can be switched off.

- In the Layers treeview right-click on **Mesh** and then click on **Properties…** On the **Visualise** tab of the properties dialog uncheck the **Joint and Interface elements** tickbox.
- If not present, add and then repeat the above steps for the **Deformed mesh** layer.
- To hide the supports and loading arrows click on the icons shown in figure 67.

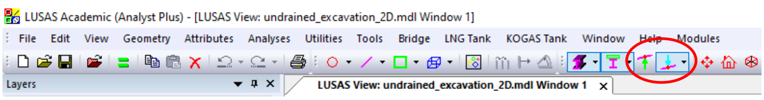


Figure 67: Hide supports and loads

Additionally, we will turn off the display of the **Geometry** layer.

- In the Layers treeview, right-click on **Geometry** and untick **Display** (figure 68).
- Repeat to turn off the display of the **Deformed mesh** layer.

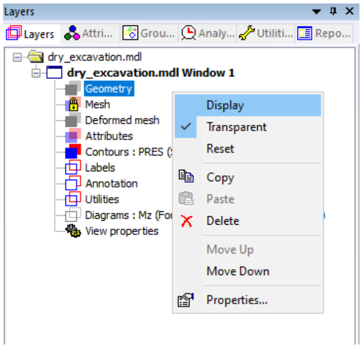


Figure 68: Hiding Geometry layer

Water pressure distribution

Contours of water pressure distribution will be displayed.

- Right-click in the model view window and click on **Contours**. Click on the entity **Displacement** and the component **Pres** and then click **OK** (figure 69).

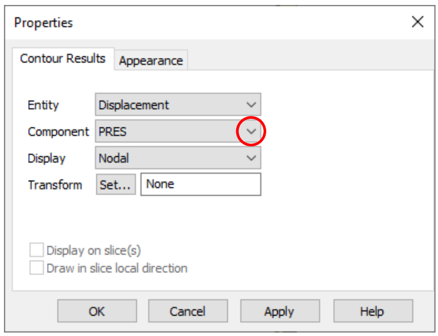


Figure 69: Contour dialog

The water pressure contours at the end of the excavation are shown in (figure 70).

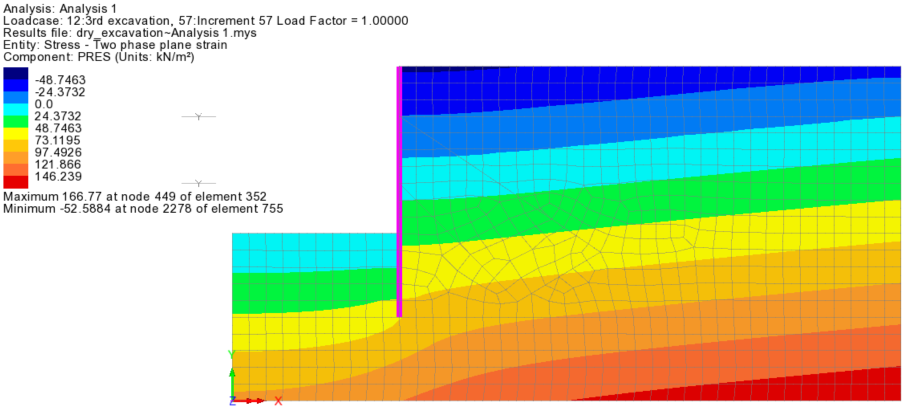


Figure 70: Water pressure contours at end of excavation

- Turn fleshing on to see the extent of the wall.

Horizontal displacements

- We can plot contours of the horizontal displacements on the deformed mesh by right-clicking on **Contours** in the **Layers** tab and then **Properties…**. Select entity **Displacement** and component **Dx** from the dropdown lists.
- In the Layers treeview, right-click on **Mesh** and then untick **Display**. Then click on **Deformed Mesh** and tick **Display**.

The largest displacement of the wall is 7.2cm into the excavation (figure 71).

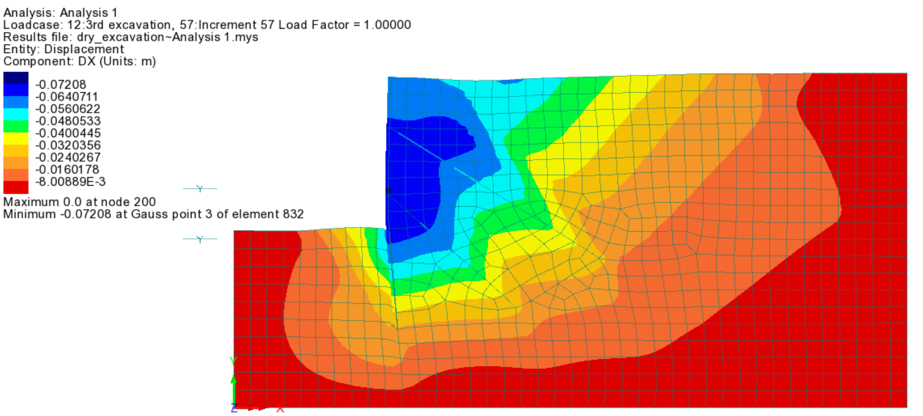


Figure 71: Horizontal displacement after excavation

Bending moment, shear and axial force in wall

Next, the bending moment, shear and axial forces in the diaphragm wall will be investigated.

- In the Layers treeview, untick **Display** for the **Deformed mesh** and **Contours**.
- In the Attributes treeview, right-click on the material **Wall** (tick the Analysis 1 check box in the dialog that appears) and then click on **Set as Only Visible** to display just the wall.
- The fleshing of the wall is still visible so switch the fleshing off by clicking the fleshing icon (figure 72).

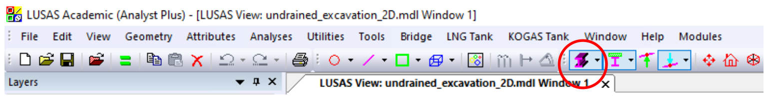


Figure 72: Toggle fleshing icon

- Right-click in the view window and select **Diagrams...** From the **Component** dropdown box select **Mz** and click **OK** (figure 73).

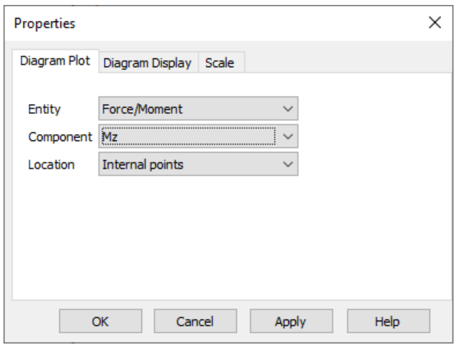


Figure 73: Diagram Properties Dialog

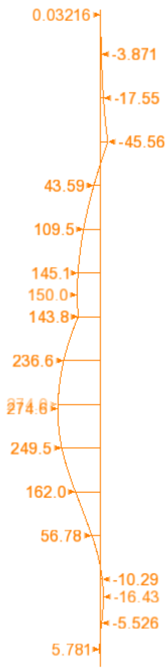


Figure 74: Wall bending moments

- To display the shear, right click on **Diagrams** in the layer tab and then **Properties**. Select **Sxy** from the **Component** dropdown box.

Figures 74 and 75 show the distribution of bending moment and shear along the wall respectively at the end of the 3rd excavation.

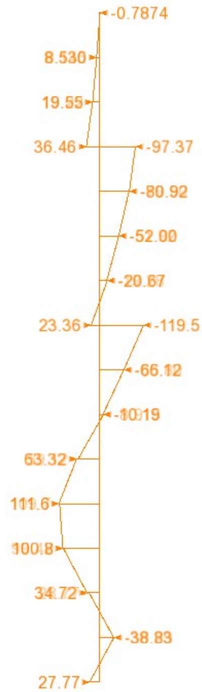


Figure 75: Wall shear

This completes the example.

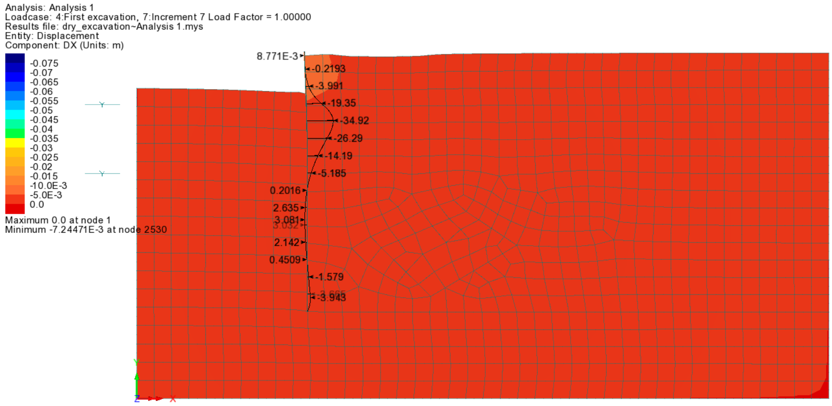
Optional additional viewing of results

With the whole model set to be 'All visible', and with the deformed mesh and contours layers re-displayed, the soil displacement and bending moments induced in the wall from the various loadcases can be seen in context.

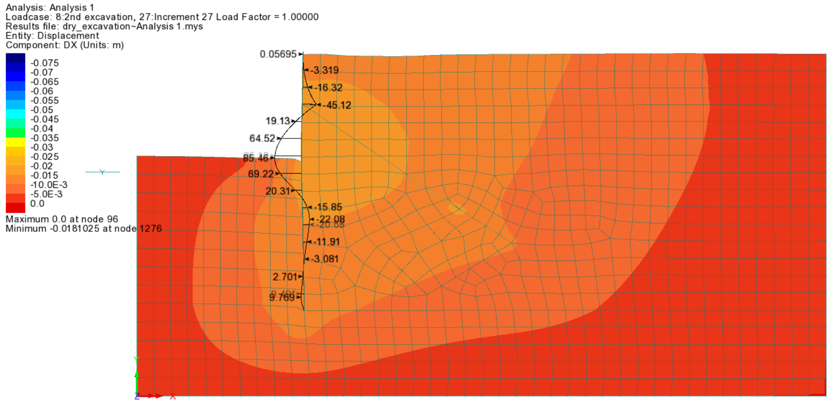
For this example, the use of a contour range with a maximum value of 0, a minimum values of -0.075 and a contour interval of 0.005 is used to ensure that consistent contours apply over a range of loadcases. In addition, changing the diagram display colour enables those values to be read more easily when overlaid on results contours.

Choosing the **Tools > Animation wizard** menu item and selecting 'load history' for final 'final increments' will animate all construction stages. Results for only the 3 excavation stages are shown below.

First excavation



Second excavation



Third excavation