

# Drained Nonlinear Analysis of a Retaining Wall

For LUSAS version:	17.0
For software product(s):	Any Plus version.
With product option(s):	Nonlinear
Note: The example exceeds the limits of the LUSAS Teaching and Training Version.	

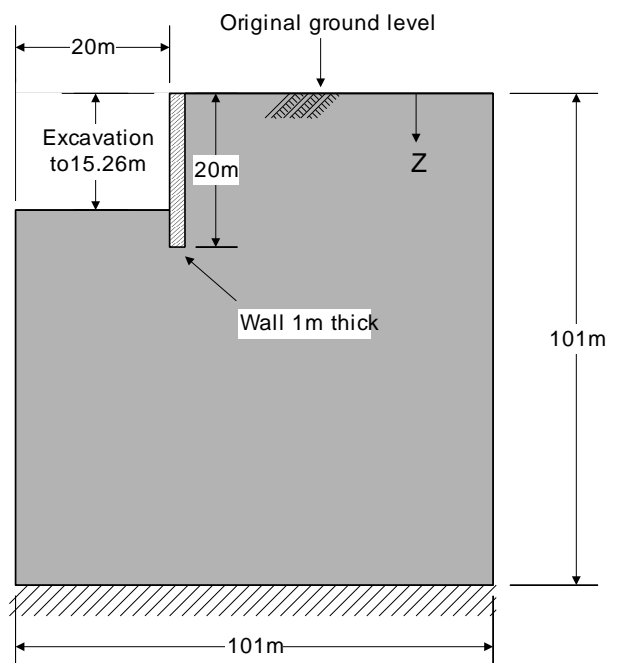
## Description

The behaviour of a single propped retaining wall is to be investigated in the long term. Drained soil parameters are used throughout.

Horizontal displacement is restrained at the left and right hand edge and both horizontal and vertical displacement is restrained at the base of the soil as the soil rests on solid rock.

The wall is 'wished-in-place' meaning that construction of the wall is not modelled.

Units of kN, m, t, s, C are used throughout.



### Objectives

The required output from the analysis consists of:

- ☐ Displacement of the wall toe towards the excavation
- ☐ Surface heave immediately behind the wall
- ☐ Contours of plastic strain around the excavation
- ☐ Prop force per m length of wall
- ☐ Bending moment in the wall at 10m depth

### Keywords

**2D, Inplane, Retaining wall, Drained, Nonlinear, Excavation, Surface heave, Plane strain, Prop force, Bending Moment, Profile variation, Initial Soil stress,  $K_0$  Initialisation.**

### Associated Files



- ☐ **drained\_wall\_modelling.vbs** carries out the modelling of the example.

## Modelling

### Running LUSAS Modeller

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



**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.

### Creating a new model

- Enter a File name of **drained\_wall**
- Use the default User-defined working folder.
- Ensure an Analysis type of **Structural** is set.
- Select an Analysis Category of **2D Inplane**
- Set Model units of **kN,m,t,s,C**
- Leave the Timescale units as **Seconds**
- Select a Startup template of **None**

- Ensure the Layout grid is set as **None**
- Enter a Title of **Drained analysis of a propped retaining wall**
- Click the **OK** button.



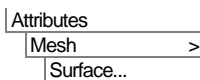
**Note.** Save the model regularly as the example progresses. Use the undo command to undo mistakes as far back as the last save.



**Note.** For a 2D Inplane analysis category the vertical axis is set to be the Y-axis.

## Mesh Definition

The surfaces are to be meshed using plane-strain quadrilaterals with a quadratic interpolation order (QPN8).



- Select the **Plane Strain, Quadrilateral, Quadratic** elements. Ensure the **Regular mesh** option is selected with **Automatic divisions** so that Modeller uses the mesh divisions assigned to each line.
- Give the attribute the name **Plane Strain quads** and click the **OK** button to add the mesh attribute to the Treeview.
- In the Treeview click on the mesh attribute **Plane Strain quads** with the right-hand mouse button and select the **Set Default** option. This will ensure all newly created surfaces will be assigned the elements defined in this mesh attribute.


## Profile Variation



The Young's Modulus of the soil varies with depth and is described by a Profile Variation. It varies linearly from a value of  $0.0\text{kN/m}^2$  at the surface to  $6.06\text{E}5\text{kN/m}^2$  at a depth of 101m.

- Select **Profile Variation** and press **Next**.
- Select **New...** and enter the values of Distance (x) **0.0** Value (y) **0.0**, Distance (x) **101.0** Value (y) **6.06E5** in the Line Profile Variation dialogue box which appears. Enter **E\_LineP** for the line profile name and click **OK**.
- Select the line profile named **E\_LineP** by left clicking on it and then click on the 'Add to' button >> to transfer it to the profile set. The origin of the profile line is by default x origin 0.0, y origin 0.0 and so does not need to be changed. Similarly, the positive direction of the line profile is, by default, the negative vertical and again does not need modification.
- Enter the name **E\_mod** for profile variation and click **Finish**.

### Feature Geometry

- Turn off the display of the **Mesh** layer in  Treeview.



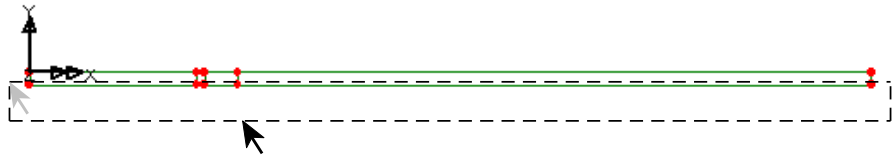
Enter coordinates of **(0, 0)**, **(20, 0)**, **(21, 0)**, **(25, 0)** and **(101, 0)** to define four lines representing the original ground level (all Z coordinates should be zero or left blank). Use the Tab key to move to the next entry field on the dialog. When all coordinates have been entered click the **OK** button.

- Press **CTRL-A** to select all the Lines just drawn.



Ensure the **Translate** option is selected and enter a value of **-1.5** in **Y**.

- Click **OK** to sweep the Lines to create the Surfaces.



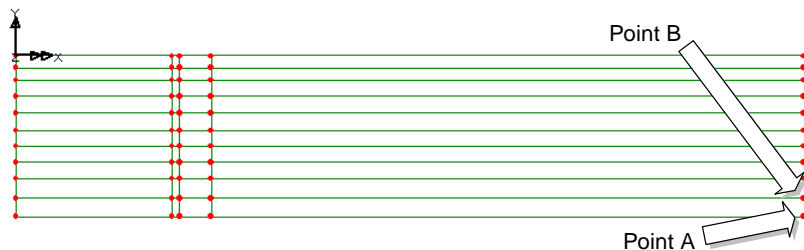
- Select the bottom four horizontal Lines by dragging a box around them.



Ensure the **Translate** option is selected and enter a value of **-1.5** in **Y**.

- Click **OK** to sweep the Lines to create the Surfaces.

Repeat the preceding commands eight times, selecting the bottom four lines of the model each time and sweep the lines by **-2.0**, **-2.13**, **-2.13**, **-2.0**, **-2.0**, **-2.0**, **-2.49** and **-2.25** metres in **Y** as shown above.



- Select Point A shown.

Geometry  
Point >  
Move...



Ensure the **Translate** option is selected and enter a value of **-81** in **Y**.

- Click **OK** to move the Point.
- Select Point B shown.

Geometry  
Point >  
Move...



Ensure the **Translate** option is selected and enter a value of **-33.25** in **Y**.

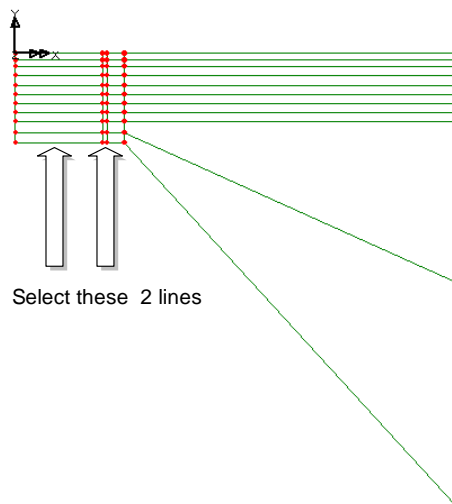
- Click **OK** to move the Point.
- Select the two Lines shown. If necessary zoom in to make the selection easier.

Geometry  
Surface >  
By Sweeping...



Ensure the **Translate** option is selected and enter a value of **-81** in **Y**.

Click **OK** to sweep the Lines to create the Surfaces.



Select these 2 lines

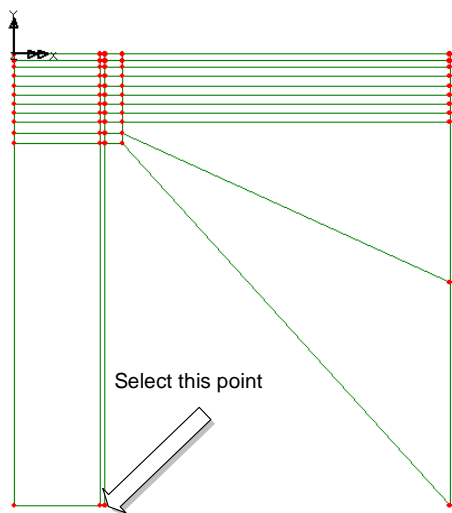
- Select the Point shown.

Geometry  
Point >  
Move...



Ensure the **Translate** option is selected and enter a value of **38** in **X**.

- Click **OK** to move the Point.



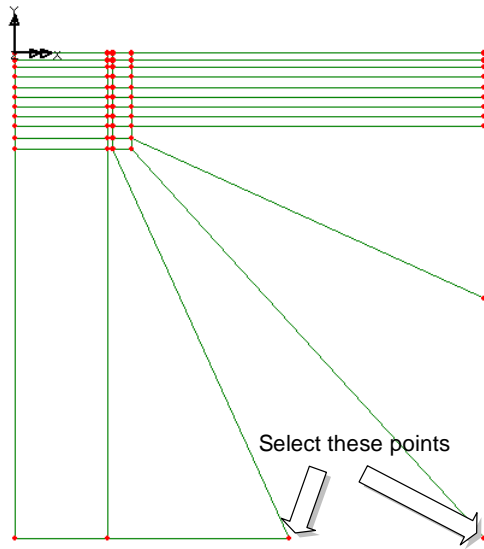
Select this point

## Drained Nonlinear Analysis of a Retaining Wall

- Select the two Points shown.



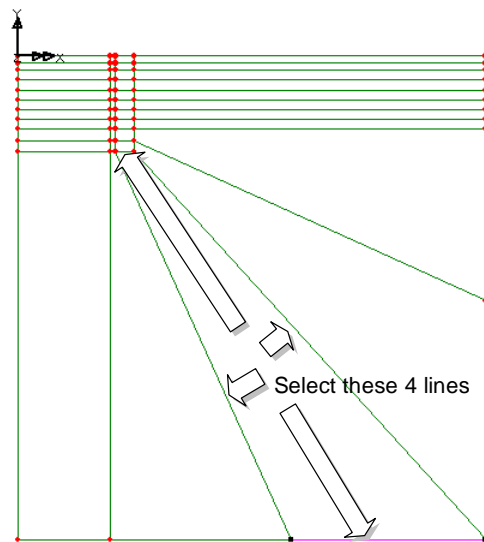
Create a Line between the two selected Points.



- Select the Line just drawn along with the three Lines as shown.



Create a Surface from the four selected Lines.

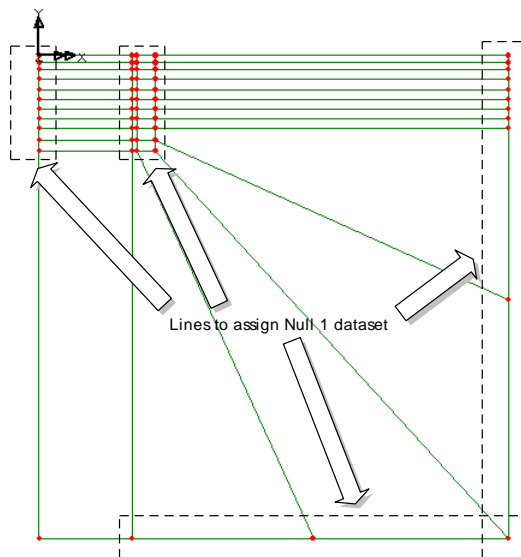


## Mesh Grading

- Define a Line mesh with element type **None** and **1** division.
- Enter the attribute name as **Null 1** and click the **Apply** button to add the attribute to the Treeview and keep the dialog active.

- Define a Line mesh with **5** divisions.
- Select the **Spacing** button.
- Select the **Uniform transition** option and set the **Ratio of first to last element** to **4**
- Click **OK** to accept the spacing properties.
- Give the attribute name as **Null 5 Graded** and click **Apply** to add the attribute to the Treeview and keep the dialog active.
- Define a Line mesh with **10** divisions.
- Select the **Spacing** button.
- Select the **Uniform transition** option and set the **Ratio of first to last element** to **0.2**
- Click **OK** to accept the spacing properties.
- Give the attribute name as **Null 10 Graded** and click **OK** to add the attribute to the Treeview.

## Assigning the mesh attributes



- Box-select the lines as shown and assign the mesh attribute **Null 1**.



**Note.** The selection of some of the sets of lines as shown in the next two images is best done by changing the selection cursor and additionally using a special keyboard shortcut which causes any lines passing through a box-selection to be selected.

To prepare for this:

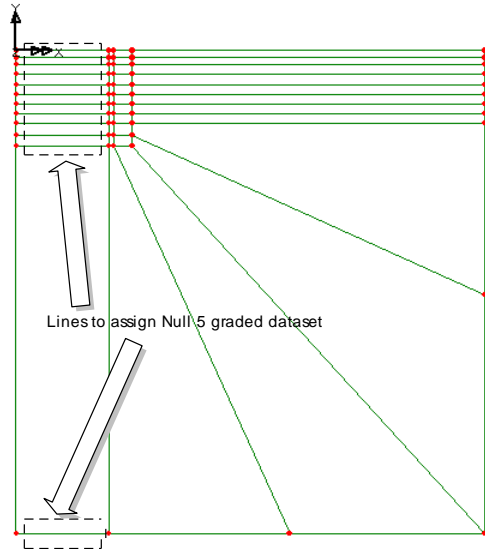


Change the cursor to the Select Lines cursor.

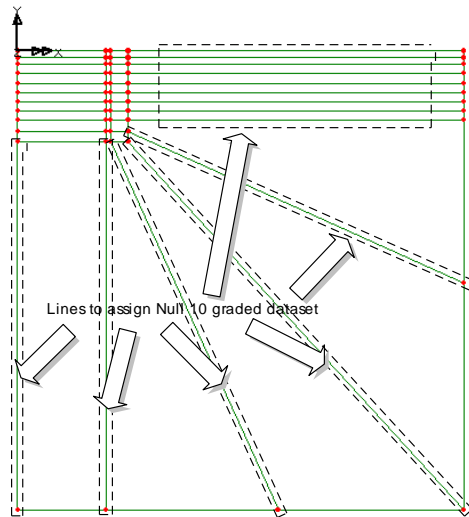
- Hold down the **ALT** key and box-select each set of horizontal lines as shown in the preceding image and assign the mesh attribute **Null 5 Graded**.



**Note.** Pressing the **Shift** key at the same time as the **ALT** key will allow extra lines to be added to a selection




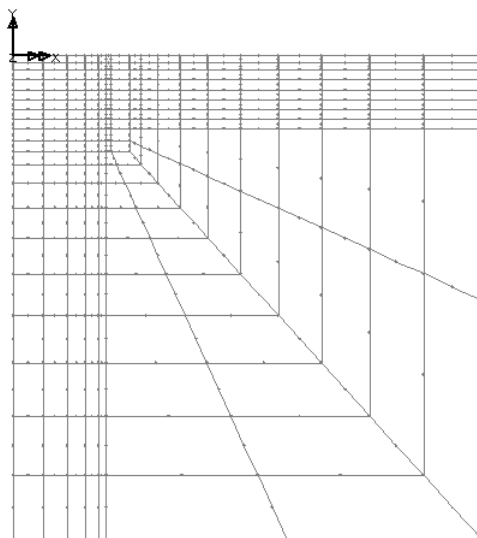
- Select the remaining lines as shown and assign the mesh attribute **Null 10 Graded**.



**Note.** See the *Keyboard Shortcut Guide* for more details on the use of keyboard shortcuts.



- Turn on the **Mesh** layer in the  Treeview to see the mesh.




**Note.** If the mesh is graded with the smaller elements at the wrong end of a line reverse the line by selecting the line using the **Geometry>Line>Reverse** menu selection. If errors have been made in mesh assignments simply re-select any lines with an incorrect mesh assignment and re-assign the correct line mesh.

## Material Properties


An isotropic elastic material will be used for the retaining wall while an isotropic nonlinear material utilising the Mohr-Coulomb failure surface will be used for the soil. The initial stress state is defined by assigning a lateral earth pressure,  $K_0$ , of 2 to both the soil and concrete wall.


Attributes	
Material	>
Isotropic...	

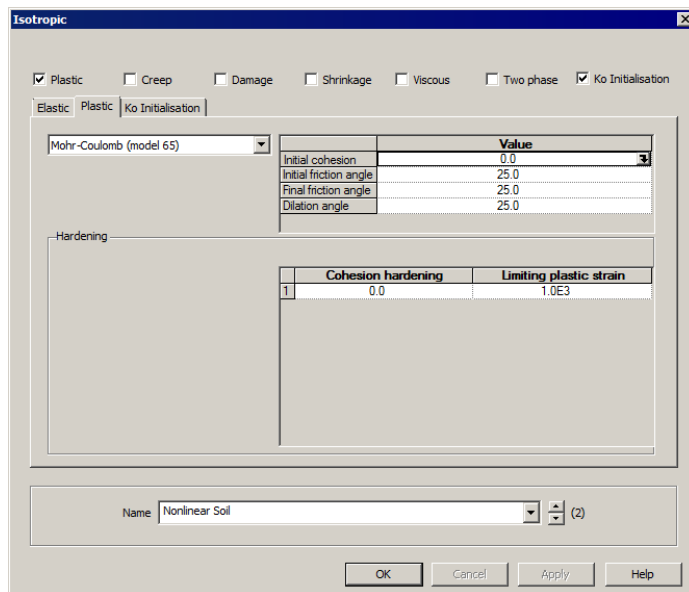
With the **Elastic** tab selected enter the isotropic material properties for the wall as Young's Modulus **28E6** kN/m<sup>2</sup>, Poisson's Ratio **0.15** and Mass Density **2.03874** tonne/m<sup>3</sup>.

- Enter the attribute name as **Concrete Wall**.
- Click the **Ko Initialisation** option and enter a **Coefficient of lateral earth pressure** of **2**
- Click the **Apply** button to add the attribute to the  Treeview and keep the dialog active.
- With the **Elastic** tab selected enter the isotropic material properties for the soil as Young's Modulus **1\*E\_mod**, Poisson's Ratio **0.2** and Mass Density **2.03874** tonne/m<sup>3</sup>.



**Note.** The **1\*E\_mod** indicates that the Young's Modulus is defined by a Profile variation scaled by a factor of **1**. Enter **1** in the grid location and click on the  button. Select the **E\_mod** variation from the drop down list.

- Click the **Plastic** option and from the Model drop-down list select the **Mohr-Coulomb (model 65)** entry.
- Enter an **Initial cohesion** value of **0**
- Enter an **Initial friction angle** value of **25**
- Enter an **Final friction angle** value of **25**
- Enter a **Dilation angle** value of **25**
- In the tension section enter a **Cohesion hardening** value of **0**
- In the tension section enter a **Limiting plastic strain** value of **1000**
- Click the **Ko Initialisation** option and enter a **Coefficient of lateral earth pressure** of **2**.
- Enter the attribute name as **Nonlinear Soil**
- Click the **OK** button to add the attribute to the  Treeview.



**Isotropic**

☒ Plastic ☐ Creep ☐ Damage ☐ Shrinkage ☐ Viscous ☐ Two phase ☒ Ko Initialisation

Elastic Plastic Ko Initialisation

Mohr-Coulomb (model 65)

	Value
Initial cohesion	0.0
Initial friction angle	25.0
Final friction angle	25.0
Dilation angle	25.0

Hardening

	Cohesion hardening	Limiting plastic strain
1	0.0	1.0E3

Name: Nonlinear Soil (2)

OK Cancel Apply Help

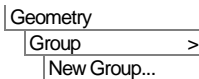
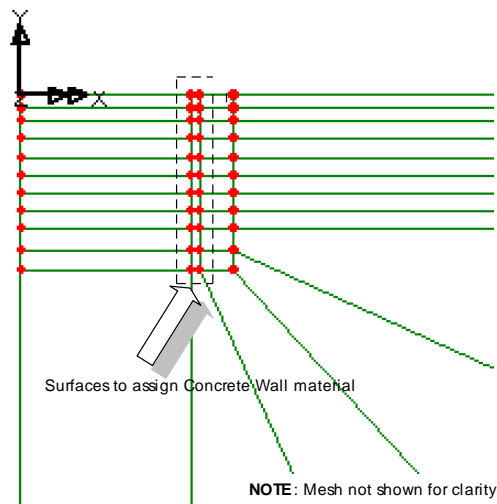
## Assigning material properties

To assign the material properties groups will be used. Groups are very useful in allowing features of the model to be turned-on and off to aid the assignment of attributes or the viewing of results on selected features of the model.



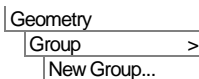
Change the cursor back to the normal cursor.

- Select the ten surfaces that form the retaining wall as shown.




Create a group containing the wall elements only and rename it **Wall**. Click **OK**.

- With the ten surfaces of the wall still selected drag and drop the material attribute **Concrete Wall** from the Treeview onto the selection. With the **Assign to surfaces** option selected click **OK** to assign the material attribute.
- In the Treeview right-click on the **Wall** group and select **Invisible**.
- Select the whole visible model by pressing the **Ctrl-A** keys.





Create a group containing the soil elements only and rename it **Soil**. Click **OK**.

- Drag and drop the material attribute **Nonlinear Soil** from the Treeview onto the selection. With the **Assign to surfaces** option selected click **OK** to assign the material attribute.



- In the  Treeview right-click on the **Wall** group and select **Visible**.

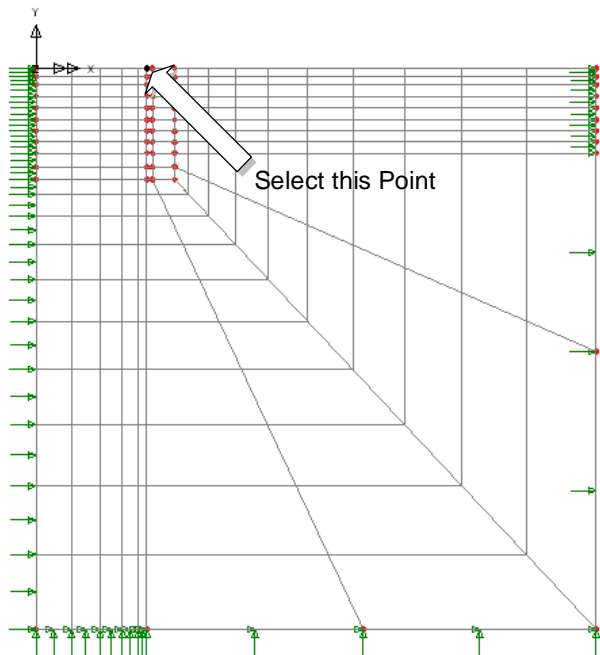
### Supports


Define supports to prescribe zero horizontal displacement as well as zero horizontal and vertical displacement.

- Click on the option to fix the **Translation in X**.
- Enter the attribute name as **Fixed in X** and click the **Apply** button to add the attribute to the  Treeview and keep the dialog active.
- Click on the option to fix the **Translation in Y**.
- Enter the attribute name as **Fixed in X and Y** and click the **OK** button to add the attribute to the  Treeview.

Assign the supports to the model.

- Select the vertical Lines on the left-hand side and right-hand side of the model and drag and drop the support attribute **Fixed in X** from the  Treeview onto the selection. With the **Assign to lines** option selected click **OK** to assign the support attribute to **All loadcases**.
- Select the Point which represents the top-left corner of the concrete wall as shown and drag and drop the support attribute **Fixed in X** from the  Treeview onto the selection. With the **Assign to Points** option selected click **OK** to assign the support attribute to **All analysis loadcases**. This support represents the prop.
- Select the 3 horizontal Lines representing the bottom limit of the soil and drag and drop the support attribute **Fixed**



in **X and Y** from the  Treeview onto the selection. With the **Assign to lines** option selected click **OK** to assign the support attribute to **All analysis loadcases**.

## Loading



The initial in situ ground conditions must be established in the first loadcase of the analysis. This is carried out using a gravity (body force) load.

Attributes  
Loading...

- Select the **Body force** tab from the Structural Loading dialog and click **Next**
- Enter **-9.81** in **Linear acceleration in Y Dir**.
- Enter the attribute name as **Gravity** and click **Finish**.

### Assign the loads to the model.


Gravity is assigned to the whole model.

- With the whole model selected drag and drop the loading attribute **Gravity** from the  Treeview onto the selection. With the **Assign to Surfaces** option selected click **OK** to assign the loading attribute to **Loadcase 1**
- In the  Treeview drag and drop the **Geometry** layer name beneath the Attributes layer name.


## Birth and Death



The **Birth and Death** feature must be used to simulate the removal of soil during excavation in front of the wall. Excavation will take place over eight loadcases.

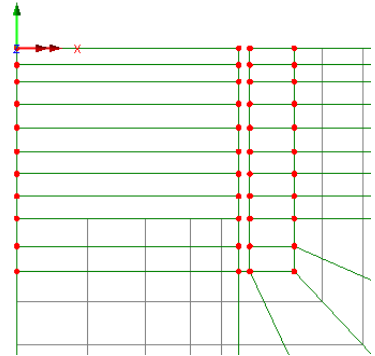
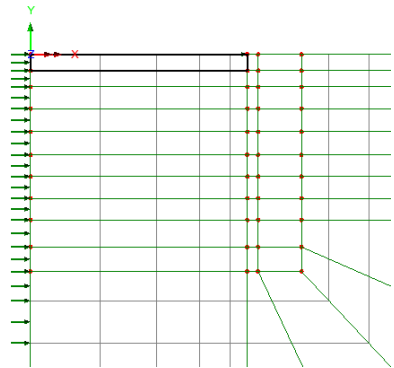
Attributes  
Birth and Death...


- Select the **Deactivate** tab and click **Next**. Select **Percentage to Redistribute** and ensure the value is set to **100%**. Enter the attribute name as **Excavation**.
- Click **Finish** to add the attribute to the  Treeview.

Assign the deactivation to the model.

- Use the zoom-in button to enlarge the view of the top-left corner of the model.
-  If necessary, for clarity, turn -off the display of the loading by pressing the loading on/off button.



- Select the top surface representing the first layer of soil to be removed in front of the retaining wall and drag and drop the deactivation attribute **Excavation** from the  Treeview onto the selection. With the **Assign to surfaces** option selected enter the Loadcase name as **Excavation to -1.50m** and click **OK** to assign the attribute.
- Select the next surface down in front of the wall and drag and drop the deactivation attribute **Excavation** from the  Treeview onto the selection. With the **Assign to surfaces** option selected enter the Loadcase name as **Excavation to -3.00m** and click **OK** to assign the attribute.
- Repeat this procedure, selecting the next surface down each time and renaming the loadcases **Excavation to -5.00m**, **Excavation to -7.13m**, **Excavation to -9.26m**, **Excavation to -11.26m**, **Excavation to -13.26m** and **Excavation to -15.26m** respectively. The adjacent image shows the final excavation assignment made.



**Note.** Deactivated surfaces (and hence elements) can be visualised and checked by setting each loadcase active in the  Treeview.

### Analysis Control

The use of a Mohr-Coulomb nonlinear material in this example dictates that a nonlinear analysis control must be specified.

- In the  Treeview right-click on **Loadcase 1** and rename it to **In situ**
- In the  Treeview right-click on **In situ** and select **Nonlinear & Transient** from the **Controls** menu.

In the **Incrementation** section:

Select the **Nonlinear** option and choose **Manual** incrementation.

In the **Solution strategy** section:

- Set the **Max number of iterations** to **50**.
- Set the **Residual force norm** to **0.01**.
- Set the **Incremental displacement norm** to **0.01**.

**Nonlinear & Transient**

**Incrementation**

☒ Nonlinear

Incrementation: **Manual**

Starting load factor: 1

Max change in load factor: 0

Max total load factor: 1

☒ Adjust load based on convergence history

Iterations per increment: 4

Options... Advanced...

☐ Time domain: **Thermal**

Initial time step: 0

Total response time: 1.00E6

☐ Automatic time stepping

Advanced...

Max time steps or increments: 0

**Solution strategy**

☐ Same as previous loadcase

Max number of iterations: 50

Residual force norm: 0.01

Incremental displacement norm: 0.01

Advanced...

**Incremental LUSAS file output**

☐ Same as previous loadcase

Output file: 1

Plot file: 1

Restart file: 0

Max number of saved restarts: 0

Log file: 1

History file: 1

OK Cancel Help

- Click **OK** to set the loadcase control.

## Saving the model

File  
Save



Save the model file.

## Running the Analysis



Open the **Solve Now** dialog. Press **OK** to run the analysis.

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, 2 files will be created in the Associated Model Data directory where the model file resides:



- ☐ **drained\_wall.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- ☐ **drained\_wall.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 the analysis fails, the output file will provide information relating to the nature of the error encountered. A common mistake made when using LUSAS Modeller for the first time is to forget to assign particular attribute data (geometry, mesh, supports, loading etc.) to the model. Any errors listed in the output file should be fixed in LUSAS Modeller before saving the model and re-running the analysis.

### Rebuilding a Model

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



-  **drained\_wall\_modelling.vbs** carries out the modelling of the example.



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 **drained\_wall**




To recreate the model, select the file **drained\_wall\_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.





Rerun the analysis to generate the results

## Viewing the Results



Loadcase results can be seen in the  Treeview, and for a nonlinear analysis the load case results for the last solved loading increment are set active by default.

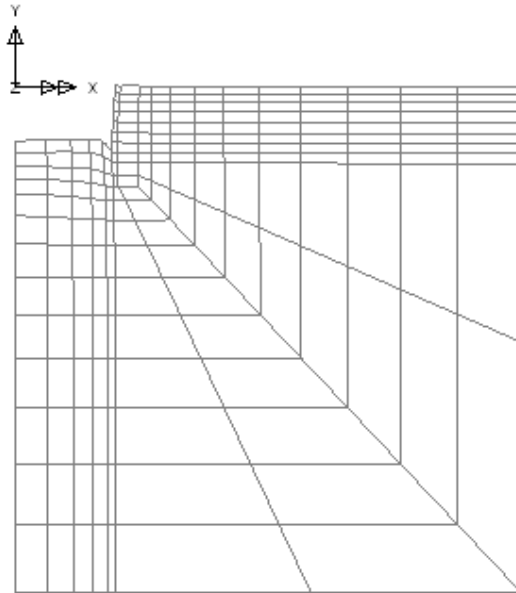
- If present, turn off the **Geometry**, **Mesh**, **Utilities** and **Attributes** in the  Treeview.

### Wall Displacement

- In the  Treeview right-click on the results for **Excavation to -13.26m** and **Set Active**.
- If not already on, add the **Deformed mesh** to add the deformed mesh layer to the  Treeview.



- In the panel at the bottom of the  Treeview press the **Deformations...** button, select the **Specify factor** option and specify a factor of **5**. Press **OK** to return.
- Double click the **View properties** control in the  Treeview. On the **View** tab ensure the option to **Show only activated elements** is selected. Click **OK** to visualise the deformed mesh for the active elements at increment 8, magnified by a factor of 5.



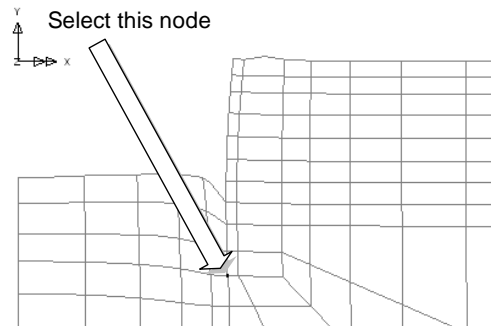
A graph of the deformation of the wall toe over a range of loadcases will now be created using the graph wizard.

- Zoom into the top left-hand side of the model and select the node at the wall toe.
- With the **Time history** option selected click **Next**.

Firstly define the data to be used for the X axis.

- Selected the **Named** option and click **Next**.
- Select **Loadcase ID** from the drop down list and click **Next**.

Secondly define the data for the Y

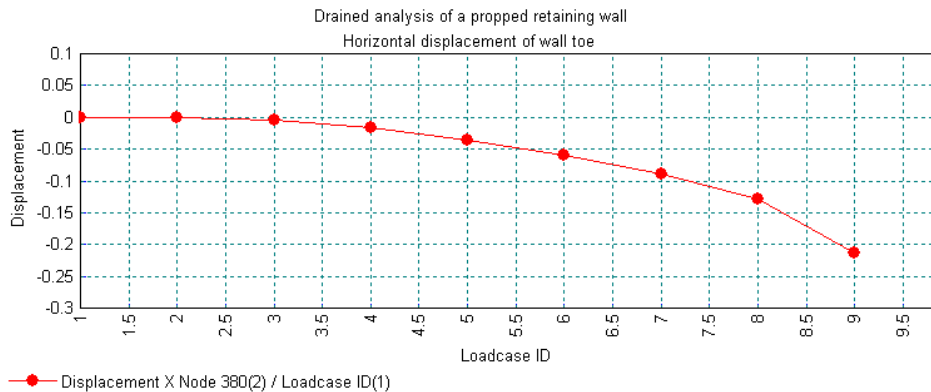




Utilities

Graph Wizard...

axis.

- With the **Nodal** option selected and click **Next**.
- Select **Displacement** from the Entity drop down list and **DX** from the Component drop down list.
- The selected node number will appear in the **Specify node** drop down list. Click **Next**.
- Enter suitable text for the graph titles and click **Finish** to display the graph of displacement of the wall toe towards the excavation.



-  Close the graph window.
-  Maximise the graphics window.

## Surface Heave

- Select the node at the top of the back of the wall.

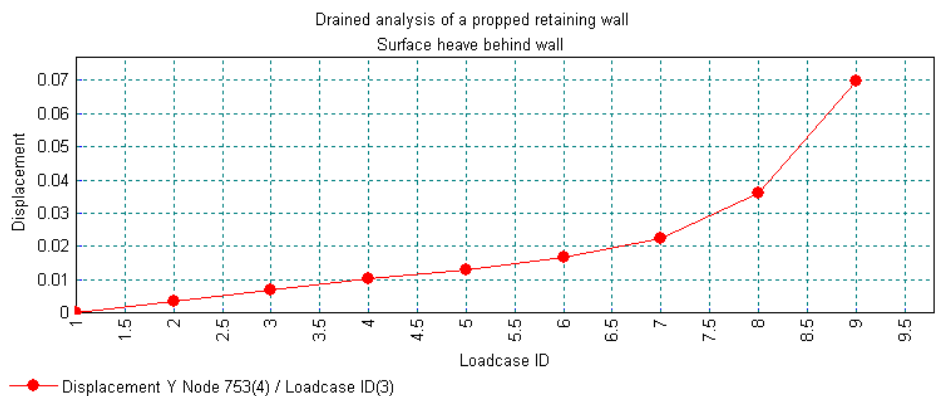
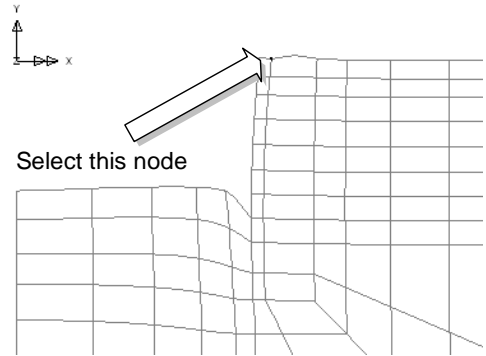
- With the **Time history** option selected click **Next**.

Firstly define the data to be used for the X axis.

- Selected the **Named** option and click **Next**.
- Select **Loadcase ID** from the drop down list and click **Next**.

Secondly define the data for the Y axis.



- With the **Nodal** option selected and click **Next**.
- Select **Displacement** from the Entity drop down list and **DY** from the Component drop down list.
- The selected node number will appear in the **Specify node** drop down list. Click **Next**.
- Enter suitable text for the graph titles and click **Finish** to display the graph of heave behind the wall.



- Close the graph window then maximise the graphics window.

### Plotting contours of plastic strain in the soil

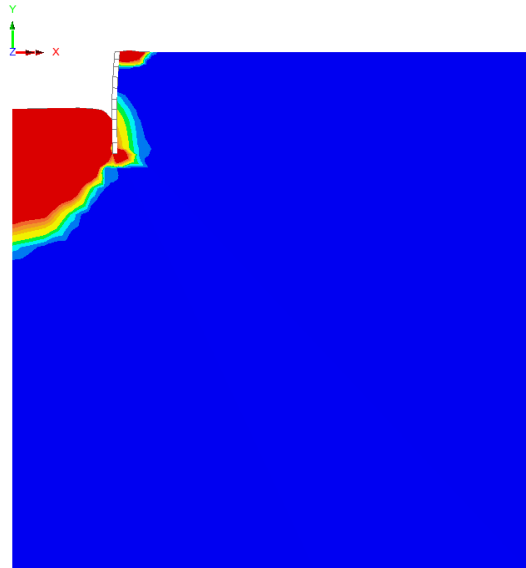
Because the model contains elements with two different material properties the active set on which results are to be plotted must be selected.

- In the  Treeview right-click on the group name **Soil** and select **Show Results > Results Plots> Show Results Only On This Group**
- With no features selected click the right-hand mouse button in a blank part of the graphics window and select **Contours** to add the contours layer to the  Treeview.
- Select **Plastic strain – Plane strain** from the Entity drop down list and **EPE** from the component drop down list.
- Select the **Contour Range** tab and set the **Maximum** to **0.01** (1% strain).
- Click the **OK** button to display contours of the plastic strain distribution around the wall.



Analysis: Analysis 1  
Loadcase: 8:Excavation to -13.26m, Increment 8  
Results file: drained-Analysis 1.mys  
Entity: Plastic strain - Plane Strain  
Component: EPE



0.0
1.25E-3
2.5E-3
3.75E-3
5.0E-3
6.25E-3
7.5E-3
8.75E-3
0.01

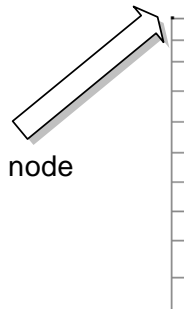
Maximum 0.306056 at node 380  
Minimum 0.0 at node 1



### Prop Force

- Turn off the display of the **Deformed Mesh** and **Contours** layers in the  Treeview.
- Turn on the display of the **Mesh** layer in the  Treeview.

- In the  Treeview right-click on the group name **Wall** and select **Show Results > Results Plots> Show Results Only On This Group**
- Right-click on the group **Wall** in the  Treeview and select **Set as Only Visible**.
- Select the node at the top-left of the wall that is supported by the prop in the X direction.
- With the **Time history** option selected click **Next**.



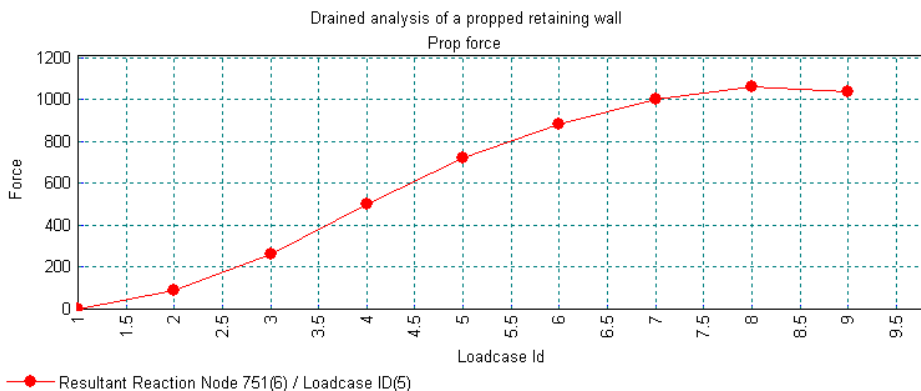
Firstly define the data to be used for the X axis.

- Selected the **Named** option and click **Next**.
- Select **Loadcase ID** from the drop down list and click **Next**.

Select this node



Secondly define the data for the Y axis.

- With the **Nodal** option selected and click **Next**.
- Select **Reaction** from the Entity drop down list and **RSLT** from the Component drop down list.
- The selected node number will appear in the **Specify node** drop down list. Click **Next**.
- Enter suitable text for the graph titles and click **Finish** to display the graph of heave behind the wall.






**Note.** The resultant reaction has been graphed in the above figure. Since the wall is propped only in the horizontal (X) direction at this location (see Supports section) there is no reaction in the vertical direction and therefore the resultant reaction is identical to the horizontal reaction.

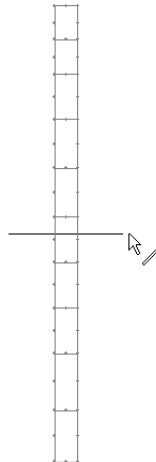
-  Close the graph window.
-  Maximise the graphics window.

### Bending moment in the wall at 10m Depth

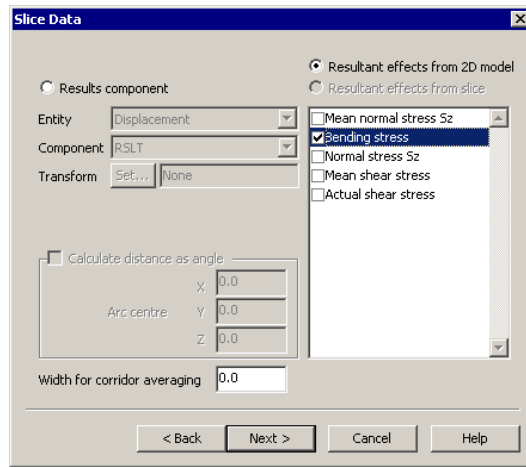
Because the model contains elements with two different material properties the active set on which results are to be plotted must be chosen.

- Turn off the **Deformed mesh** layers in the  Treeview, and restore the **Geometry** layer by right-hand clicking on a blank area of the model view and selecting **Geometry**.
- Ensure the **Snap to grid** option is selected and enter a value of **1** for **Grid size**.
- Click on **OK**.
- With reference to the ruler on the edge of the view window, draw a line with the mouse through the wall at 10m depth as shown.

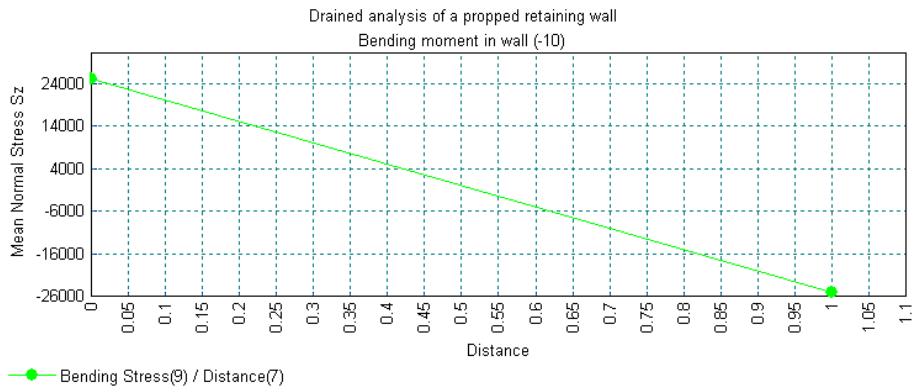
Utilities  
Graph Through 2D...



- Ensure the **Active** loadcase radio button is selected, and press **Next**:
- Select **Resultant effects from 2D model** and ensure that only the **Bending stress** option is selected in the list and click **Next**.



- Enter suitable text for the graph titles and click **Finish**. A graph is drawn and the bending moment (and other information) is displayed in the Text output pane.



```
Dataset 17 , no. of values 2 range -170.092 to -170.092 (mean shear)
Dataset 18 , no. of values 2 range -180.236 to -159.948 (actual shear)
2D Forces/moments
Loadcase "8:Increment 8"
Results File "drained_new~Analysis 1.mys" (ID=1)
Axial force per unit width = -48.2826
Shear force per unit width = -170.092
Moment per unit width = 4.17332E3
Section Depth 1.0
Mean normal stress Sz = -48.2826
Nominal bending stress = 25.0399E3
```

The bending moment reported is  $4173 \text{ kN/m}^2$  per metre run of the wall.

This completes the example.