Drained Nonlinear Analysis of a Retaining Wall

For LUSAS version:	16.0
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.	

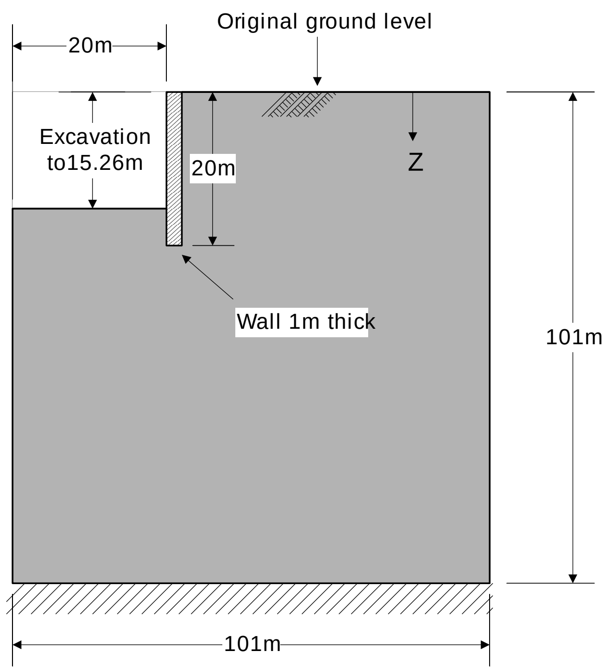
Description

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' in 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

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).

Attributes
Mesh >
Surface...

- 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.

General Field Variations

Utilities
Variation...

The Young's Modulus of the soil and the in situ stresses vary with depth and therefore general field variations are required.

- Select **General Field Variation** and press **Next**.
- Enter the function as **-6000*Y** and the attribute name as **y_mod** for the definition of Young's Modulus with depth and click **Apply** to add the attribute to the Treeview and keep the dialog active.
- Enter a new function as **20*Y** and the attribute name as **sig_v** for the definition of vertical stress and click **Apply.**
- Enter a new function as **2*20*Y** and the attribute name as **sig_h** for the definition of horizontal stress and click **Finish.**

Note. The LUSAS convention assumes that negative stresses are compressive. The Y coordinate datum is at ground level and so increasing depth will lead to increasingly negative Y coordinates. The variations as entered will ensure that stresses increase negatively with depth while the Young's Modulus increases positively with depth.

Feature Geometry

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

Geometry > Line > By Coords...

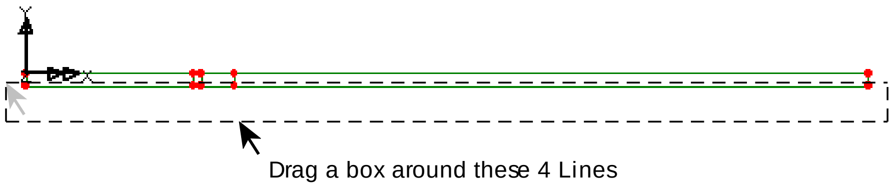
Enter coordinates of (**0, 0**), (**20, 0**), (**21, 0**), (**25, 0**) and (**101, 0**) to define four lines representing the original ground level (all Z coordinates should be zero or left blank). Use the Tab key to move to the next entry field on the dialog. When all coordinates have been entered click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

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

Geometry > Surface > By Sweeping...

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.



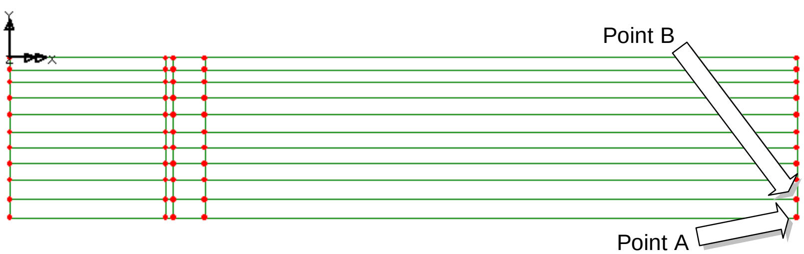
- Select the bottom four horizontal Lines by dragging a box around them.

Geometry > Surface > By Sweeping...

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...

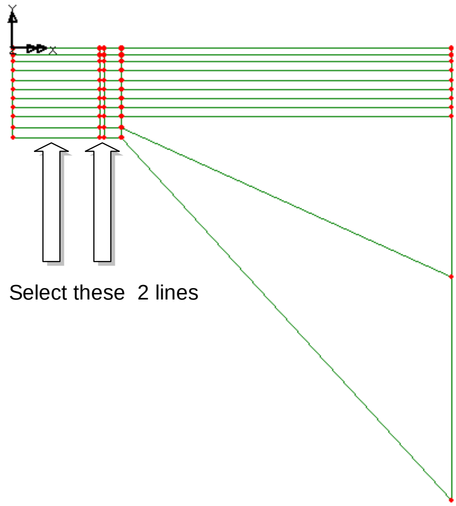
Ensure the **Translate** option is selected and enter a value of **–81** in **Y.**

- 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.

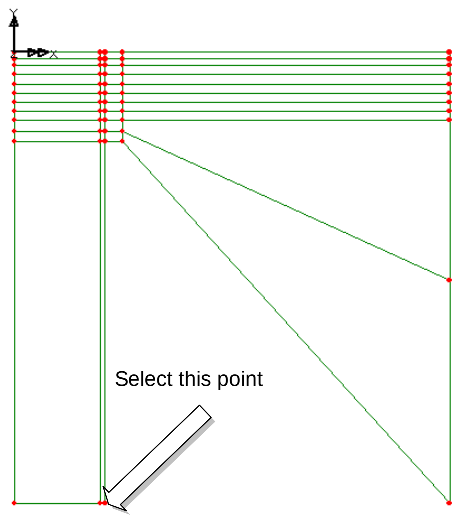


Geometry
Surface >
By Sweeping...

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 the Point shown.



Geometry
Point >
Move...

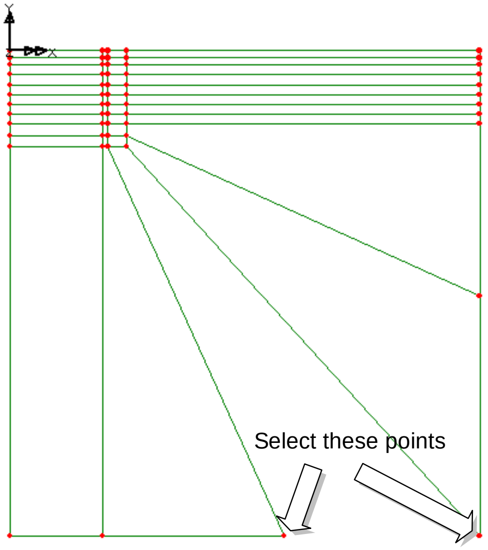
Ensure the **Translate** option is selected and enter a value of **38** in **X.**

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

Geometry
Line >
By Points...

- Select the two Points shown.

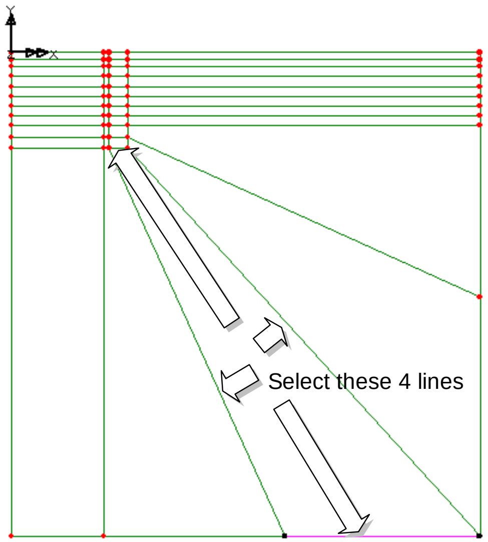
Create a Line between the two selected Points.



Geometry
Surface >
By Lines...

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

Create a Surface from the four selected Lines.



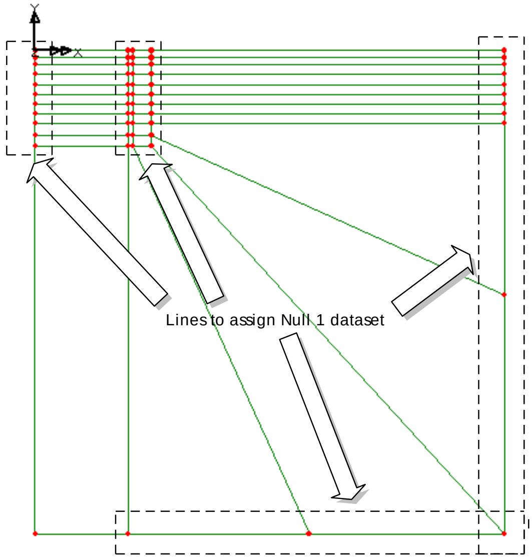
Mesh Grading

Attributes
Mesh >
Line...

- 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.**

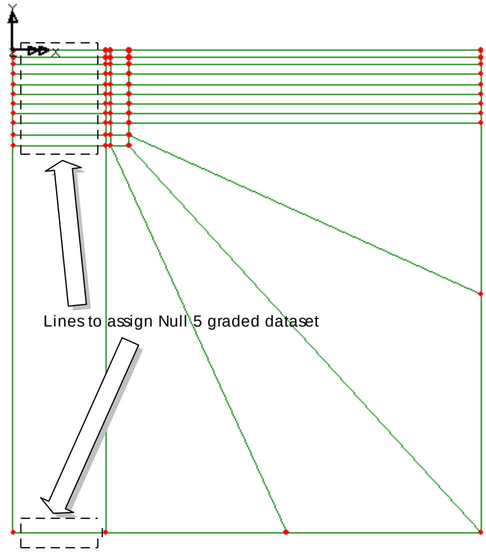
Note. The selection of some of the sets of lines as shown in the next two images is best done by changing the selection cursor and additionally using a special keyboard shortcut which causes any lines passing through a box-selection to be selected.

To prepare for this:

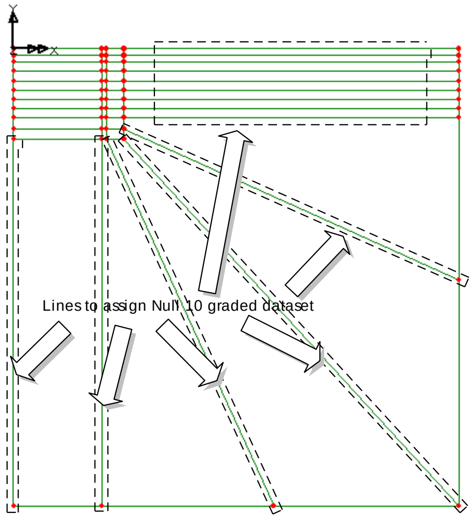
 Change the cursor to the Select Lines cursor.

- 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

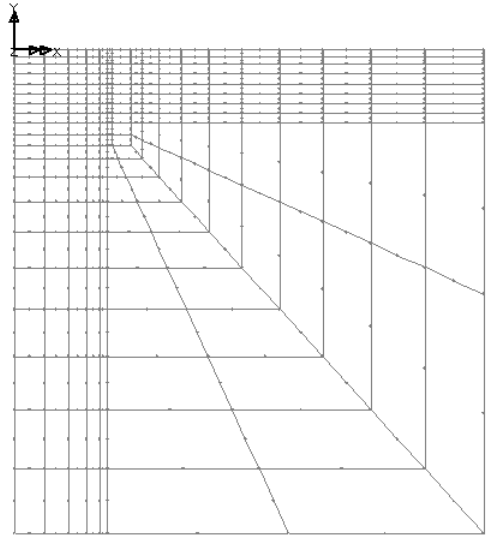


- Select the remaining lines as shown and assign the mesh attribute **Null 10 Graded.**



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.

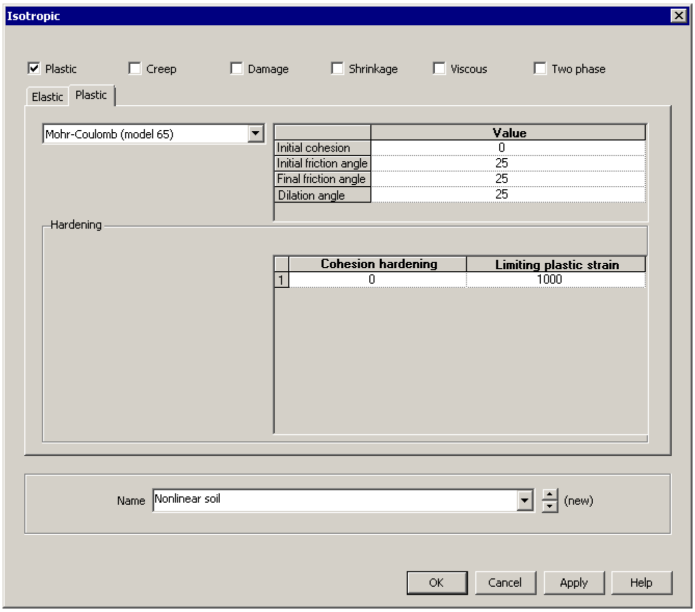
Attributes
Material >
Isotropic...

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

- Enter the attribute name as **Concrete Wall.**
- 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*y_mod**, Poisson's Ratio **0.2** and Mass Density **2.03874** tonne/m^3.

Note. The **1*y_mod** indicates that the Young's Modulus is defined by a variation scaled by a factor of **1**. As an alternative to typing in the formula, enter **1** in the grid location and click on the button. Select the **y_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**
- Enter the attribute name as **Nonlinear Soil**
- Click the **OK** button to add the attribute to the Treeview.

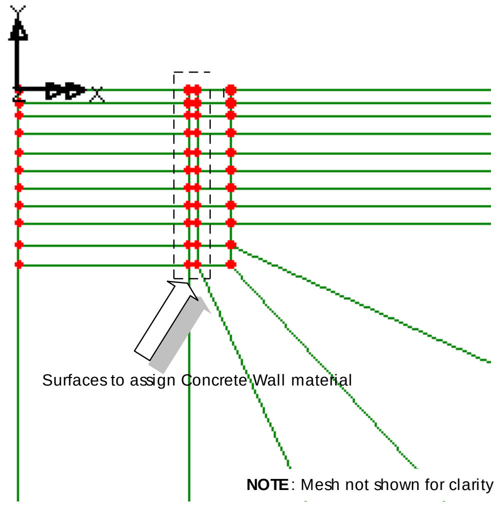


Assigning material properties

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.



Geometry
Group >
New Group...

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.

Geometry
Group >
New Group...

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

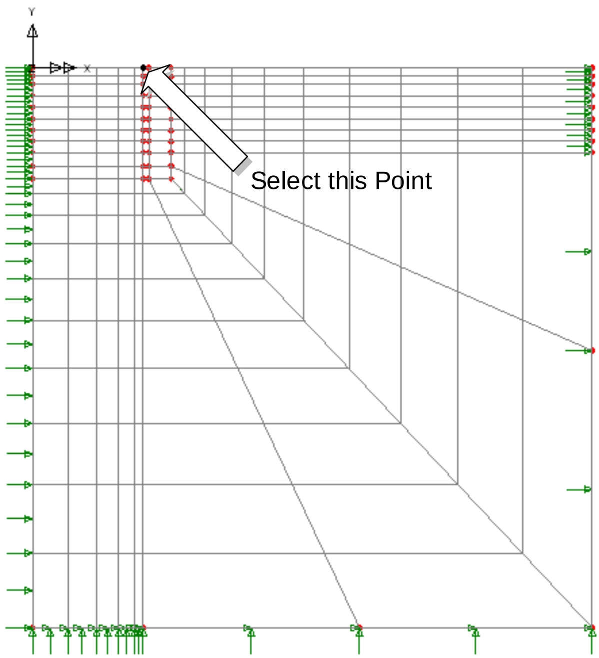
Define supports to prescribe zero horizontal displacement as well as zero horizontal and vertical displacement.

Attributes
Support…

- 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.



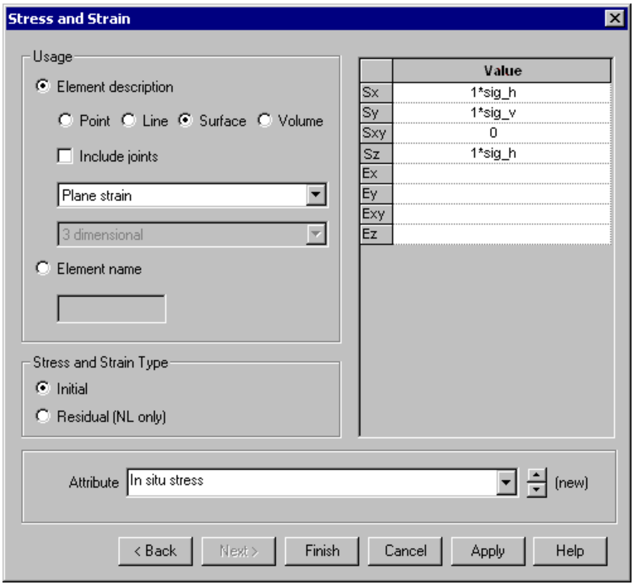
- Select the 3 horizontal Lines representing the bottom limit of the soil and drag and drop the support attribute **Fixed in X and Y** from the Treeview onto the selection. With the **Assign to lines** option selected click **OK** to assign the support attribute to **All analysis loadcases.**

Loading

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

Attributes
Loading…

- Select the **Stress and Strain** option and click **Next**
- On the Stress and Strain dialog ensure the Stress And Strain Type is set to **Initial**



- Select a **Surface** element description
- Select **Plane strain** from the drop down list
- Enter **1*sig_h** for Sx. (This can be done most easily by clicking in the right-hand side of the Sx field and selecting the variation attribute from the drop-down list)
- Enter **1*sig_v** for Sy.
- Enter **0** for Sxy
- Enter **1*sig_h** for Sz. The remaining fields may be left blank (to signify zero)
- Enter the attribute name as **In situ stress** and click **Finish.**

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.

The in-situ stress is to be assigned to the whole model:

- Select the whole model by pressing **Ctrl-A** and drag and drop the loading attribute **In situ stress** from the Treeview onto the selection. With the **Assign to Surfaces** option selected rename the loading attribute to **Loadcase 1** and click OK.

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

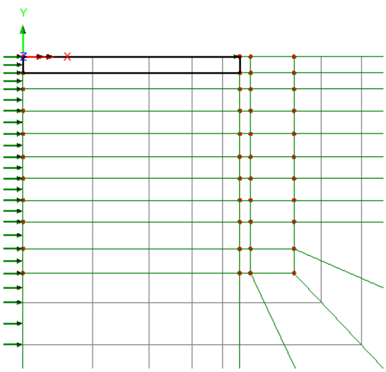
The **Birth and Death** feature must be used to simulate the removal of soil during excavation in front of the wall. Excavation will take place over eight loadcases.

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.

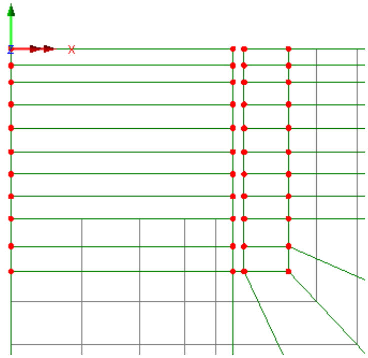
Assign the deactivation to the model.

- 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.

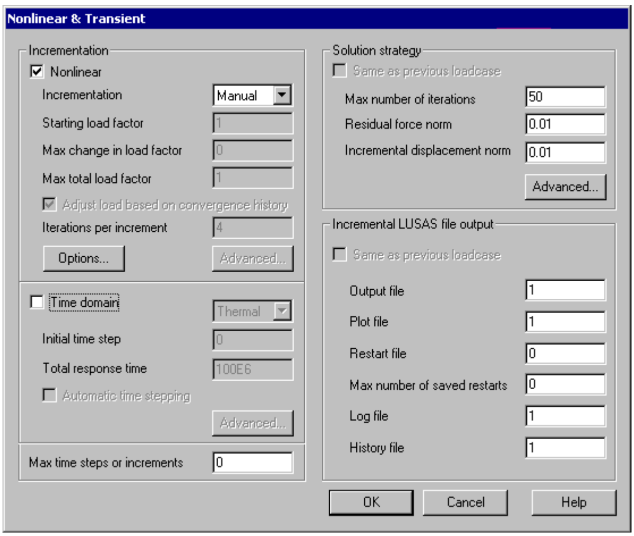


Note. Deactivated surfaces (and hence elements) can be visualised and checked by setting each loadcase active in the Treeview.

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**.

- 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.

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 **drained_wall**

File
Script >
Run Script...

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.

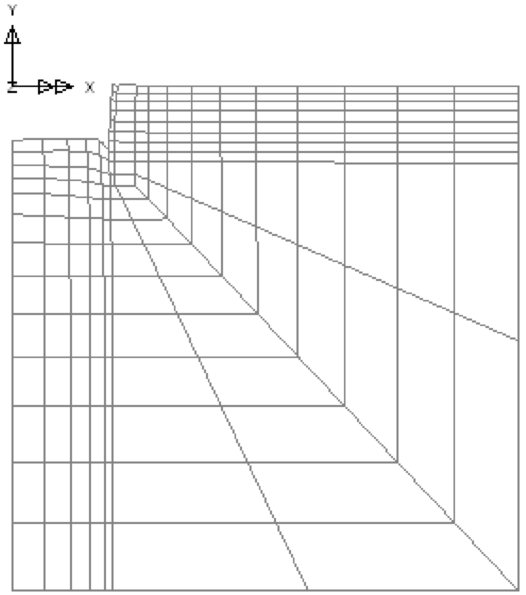
Wall Displacement

- 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.

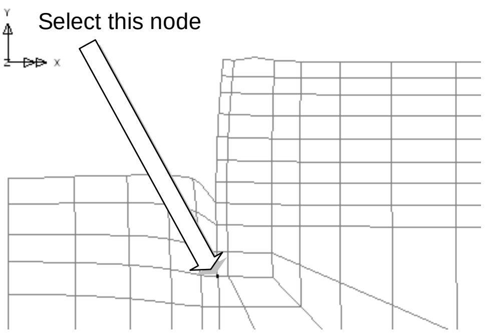


A graph of the deformation of the wall toe over a range of loadcases will now be created using the graph wizard.

- Zoom into the top left-hand side of the model and select the node at the wall toe.

Utilities
Graph Wizard...

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



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

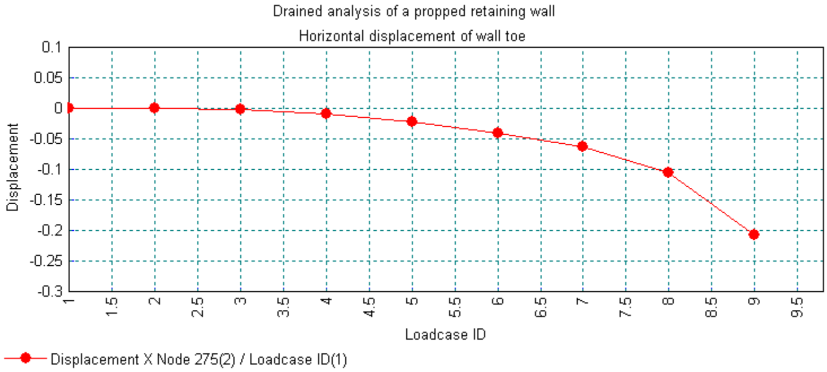
- Selected the **Named** option and click **Next.**
- Select **Loadcase ID** from the drop down list and click **Next.**

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 **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.

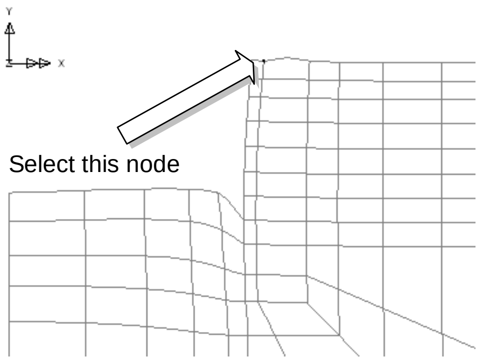
-  Maximise the graphics window.

Surface Heave

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

Utilities
Graph Wizard...

- 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.**

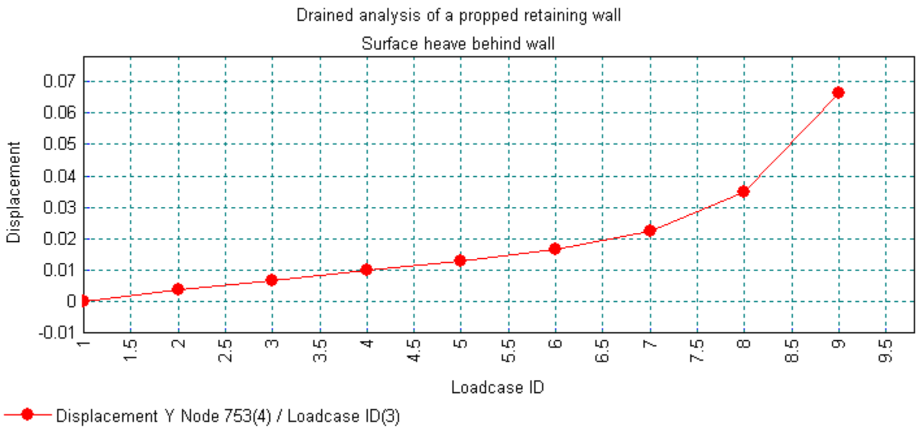
- Select **Loadcase ID** from the drop down list and click **Next.**

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.

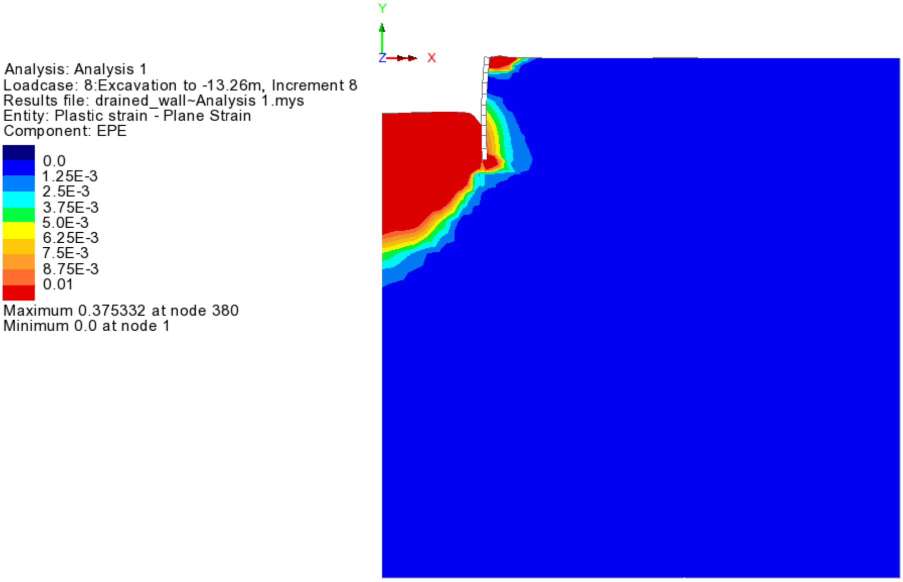


- Close the graph window then maximise the graphics window.

Plotting contours of plastic strain in the soil

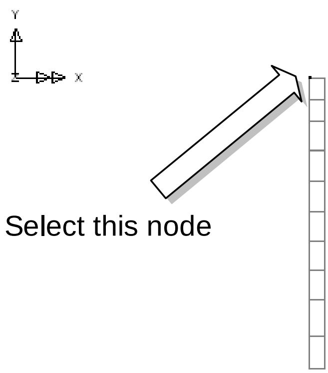
Because the model contains elements with two different material properties the active set on which results are to be plotted must be selected.

- 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.



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.



Utilities
Graph Wizard...

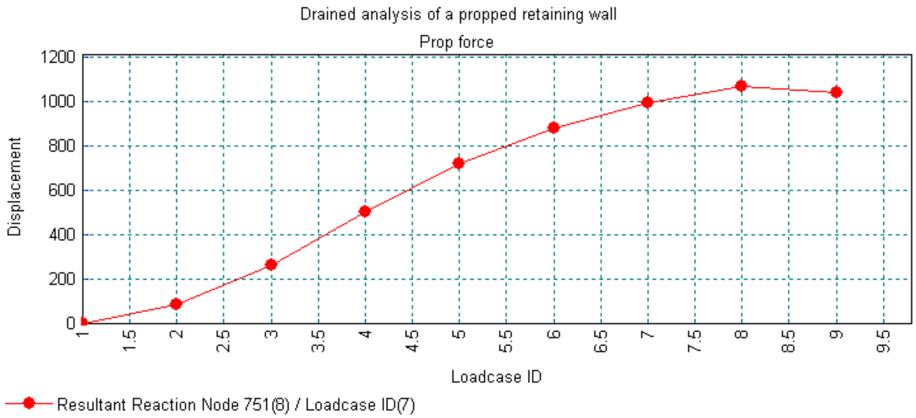
- With the **Time history** option selected click **Next.**

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

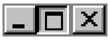
- Selected the **Named** option and click **Next.**
- Select **Loadcase ID** from the drop down list and click **Next.**

Secondly define the data for the Y axis.

- With the **Nodal** option selected and click **Next.**
- Select **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

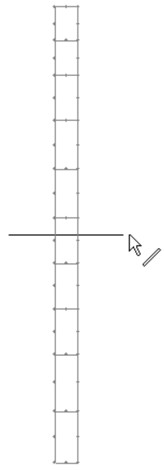
Because the model contains elements with two different material properties the active set on which results are to be plotted must be chosen.

- 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**.

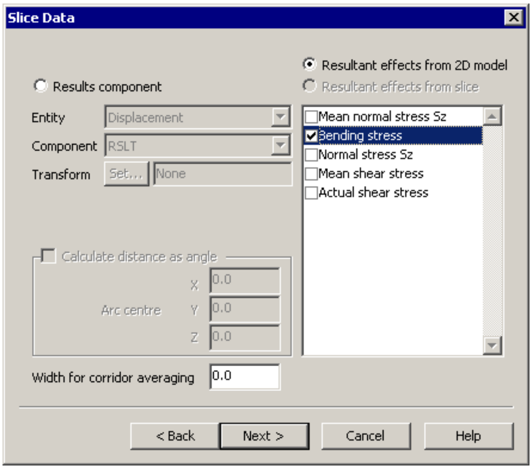
Utilities
Graph Through 2D...

- Ensure the **Snap to grid** option is selected and enter a value of **1** for **Grid size**.
- Click on **OK**.

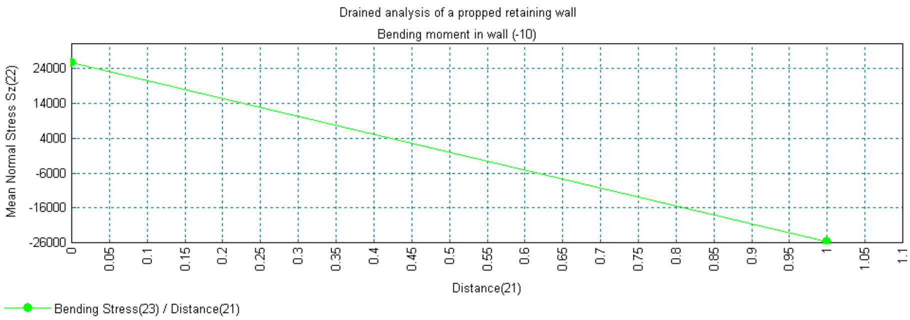
- With reference to the ruler on the edge of the view wiondow, draw a line with the mouse through the wall at 10m depth as shown.



- 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.




```
2D Forces/moments
Loadcase "8:Increment 8"
Results File "drained_wall~Analysis 1.mys" (ID=1)
Axial force per unit width = -49.9059
Shear force per unit width = -148.842
Moment per unit width = 4.2793E3
Section Depth 1.0
Mean normal stress Sz = -49.9059
Nominal bending stress = 25.6758E3
```

The bending moment reported is 4280 kN/m^2 per metre run of the wall.

This completes the example.