Embedded Retaining Wall

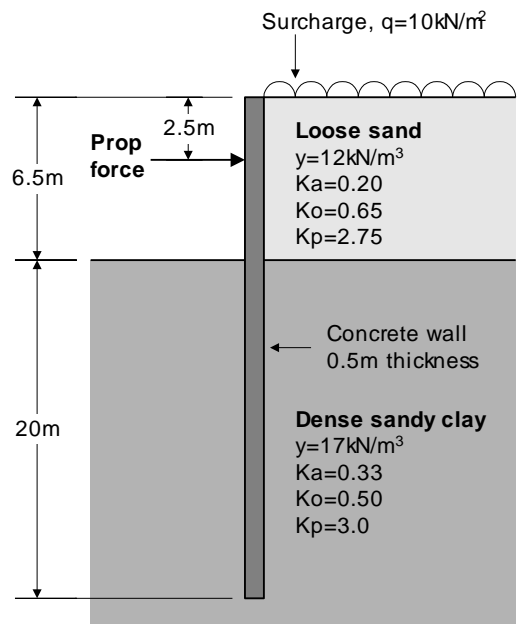| For LUSAS version:	18.0
For software product(s):	LUSAS *Civil & Structural* or LUSAS *Bridge.*
With product option(s):	Nonlinear.

Description

An embedded retaining wall is required to retain 6.5m of loose sand along with an additional surcharge of $10kN/m^2$. A 2D analysis is to be carried out with the soil represented by tri-linear earth pressure joint material for both an unpropped and a propped wall condition.

The units of the analysis are kN, m, t, s, C throughout.

Objectives

❑ To calculate bending moment and shear forces in the embedded wall and ensure that deflections are within serviceable limits.

❑ To investigate the effects of installing a prop and reducing the pile length.

Surcharge, $q=10kN/m^2$

Prop force

2.5m

6.5m

Loose sand
$y=12kN/m^3$
Ka=0.20
Ko=0.65
Kp=2.75

Concrete wall
0.5m thickness

Dense sandy clay
$y=17kN/m^3$
Ka=0.33
Ko=0.50
Kp=3.0

20m

Keywords

2D, Inplane, Embedded, Retaining, Wall, Trilinear Earth Pressures, Modulus of Subgrade Reaction, Soil-Structure Interaction, SSI, Geotechnics

Associated Files

❑ **embedded_wall_modelling.vbs** carries out the modelling of this 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. Modeller will prompt for any unsaved data and display the New Model dialog.

Creating a new model

- Enter a File name of **Embedded 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 **Embedded Retaining Wall Example**

- Click the **OK** button.

Note. Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.

Defining the Wall Geometry

Geometry
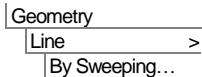Point   >
  By Coords...

- Enter coordinate (**0, 0, 0**) to define ground level and the top point of the wall. Click the **OK** button to create a single point at the origin.

Note. Separate lines must be defined to represent each change of soil type.

- Select the newly created Point.

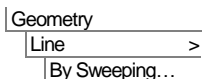Geometry
| Line | > |
| By Sweeping… | |

⬒ Sweep the selected point with a Y translation of **-2.5** and click the **OK** button to create the new Line.

- Select the newly created lower point

Geometry
| Line | > |
| By Sweeping… | |
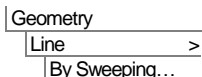
⬒ Sweep the selected point with a Y translation of **-4.0** and click the **OK** button to define the new Line.

- Select the newly created bottom point

Geometry
| Line | > |
| By Sweeping… | |

⬒ Sweep the selected point with a Y translation of **-20.0** then click the **OK** button to define the new Line.

Defining the Earth Geometry

The earth pressure acting on the wall is modelled using joint elements connected between the wall itself and fixed supports. Additional lines are required to assign the fixed sides of the joint elements to.

Note. Joint elements should normally have a zero length, however to reduced confusion from overlapping features during definition they are usually initially located with a gap that is later closed once the model has been setup.

- Select all lines and points (Press **Ctrl** + **A** keys)
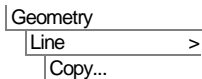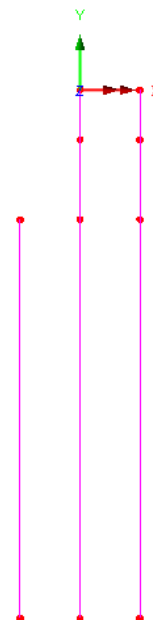
Geometry
| Line | > |
| Copy... | |

⬒ Copy all the geometry by **3.0** in the **X** direction and click **OK** to define the retained soil.

- With the wall structure still selected:
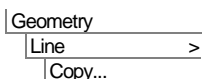
Geometry
| Line | > |
| Copy... | |

⬒ Copy all the geometry **by -3.0** in the **X** direction and click **OK** to define the retained soil.

Geometry
Line >
Delete...
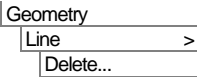
✗ Select the upper two lines of the excavated side and delete them.

The geometry is now complete. The central series of lines represent the wall. The lines to either side represent the soil. A load which represents the propping force applied to the upper part of the wall will be added to the model later in the example.

File
Save

💾 Save the model file.

Meshing

Note. Due to the underlying implementation of the tri-linear earth pressure material model, the joint elements that the soil material will be assigned to must be orientated in a specific way:

- For both 2D and 3D joint elements the joint element **local x** axis must align with the horizontal and connect the support to the structure.

- For 2D joint elements the **local y** axis must be aligned to the model vertical axis

- For 3D joint elements the **local z** must be aligned to the model vertical axis.

Note that the model vertical axis can be seen and changed on the model properties dialog, which can also be accessed by the **Tools > Vertical Axis** menu item.

Defining local coordinate systems

To ensure the joint elements are orientated correctly two local coordinate systems are needed: one in the direction of the main model X axis, and one in an opposing direction.

Attributes
Local Coordinates...

- Create a local **Cartesian** coordinate system rotated **0** degrees about the **Z-axis** with an Origin of **(-3,0,0)**, enter the dataset name as **Excavated** and click **Apply**

- Create a second local **Cartesian** coordinate system rotated **180** degrees about the **Z-axis** with an Origin of **(3,0,0)**, enter the dataset name as **Retained** and click **OK.**

- ⊕ Press the Isometric button to see the local coordinate systems created.
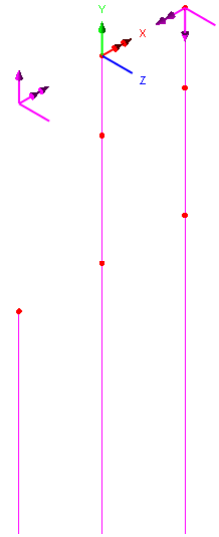
Defining element types

The joint elements are required to have displacements in the u and v directions to represent horizontal and vertical pressures. For this 2D problem, using a 2D joint element for bars, plane stress and plane strain, which connects two nodes by two springs in the local x and y-directions is sufficient.

Attributes	
Mesh	>
Line...	

- Select the element type as **Joint no rotational stiffness** and the interpolation order as **Linear**. Select an **Element length** of **1.0**. Enter the dataset name as **Soil Mesh** and click the **OK** button.

The wall requires thick beam elements:

Attributes	
Mesh	>
Line...	

- Select the element type as **Thick beam** and the interpolation order as **Linear**. Select an **Element length** of **1.0**. Enter the dataset name as **Thick Beam** and click the **OK** button.

Assigning the elements

The wall is meshed first, and the joint elements are then assigned between the lines representing the soil and the wall.

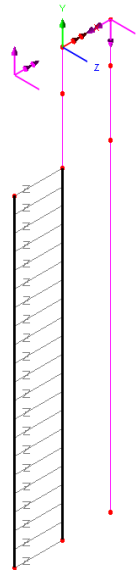### Wall

- Select the three central lines representing the wall and drag the dataset **Thick Beam** the 🟤 Treeview onto the model.

Exacavated side

- Select the left-hand line representing the excavated earth and then (by holding down the **Shift** key) also select the corresponding adjacent line of the wall.

- Drag the dataset **Soil Mesh** from the 🍀 Treeview onto the model. In the Line Mesh Assignment dialog, select **Excavated** from the specified local coordinates list and ensure **Mesh from master to slave** is selected. Click **OK** to complete the assignment

Note. The first line that is selected will be the 'Master' and the second line selected becomes the 'Slave'

Retained side

- Click in a blank part of the view window to deselect the two lines previously selected, then select (from top to bottom) the three central lines representing the wall.

Note. To make joint mesh assignments to sets of multiple lines the selection memory must be used:
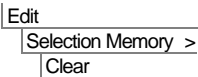
Edit
  Selection Memory >
    Set

This places the lines in the selection memory.

- Click in a blank part of the view window to deselect the three lines previously selected. Note that they will remain 'highlighted' because they are in selection memory.

- Select (from top to bottom) the three lines on the far right representing the retained soil.

- Drag the dataset **Soil Mesh** from the 🍀 Treeview onto the model. In the Line Mesh Assignment dialog, select **Retained** from the specified local coordinates list and ensure **Mesh from master to slave** is selected. Click **OK** to complete the assignment.

Note. In this instance the lines selected in the View window are the 'Masters' and the lines present in the selection memory are the 'Slaves'. Selecting the lines from top to bottom ensure each 'master' is mapped to the correct 'slave'.

Edit
Selection Memory >
Clear
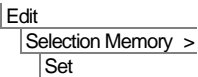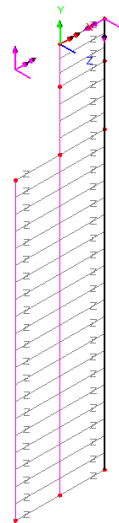
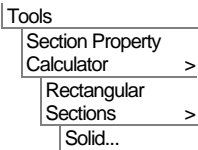This clears the selection memory.

The previous image shows the model after selection memory has been cleared with the joints correctly assigned.

Note. If necessary the joint element axes can be visualised and checked by editing the Mesh layer properties (double-click the **Mesh** layer name in the Treeview) and selecting Show element axes.

Note. If necessary line mesh assignments can be checked by selecting a line, viewing its **Properties**, and selecting the **Details** option for the corresponding mesh entry.

Geometric Properties

Tools
Section Property
Calculator >
Rectangular
Sections >
Solid...

| Attributes |
|---|
| Geometric > |
| Section Library... |

- Enter a depth **D** of **0.5** and a breadth **B** of **1.0**. Enter a dataset name of **Wall**, ensure **Add to local library** is selected and click the **OK** button.

To pull the section out of the library for use on the model:

- In the Geometric Line dialog select **2D Thick Beam** for the usage, select **User Sections** in the top right list and dataset **Wall** from the available **Local** sections. Enter the dataset **Wall 1m run** and click **OK** to create the geometric attribute.

- Select the three lines representing the wall and drag the geometric attribute **Wall 1m run** from the Treeview onto the model.

Geometric properties are visualised by default.

- In the Treeview drag and drop the layer names so that the **Attributes** layer is at the top of the Treeview, followed by the **Mesh** layer, followed by **Geometry** layer.

- Select the fleshing on/off button to turn-off the geometric visualisation.
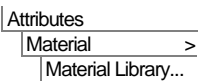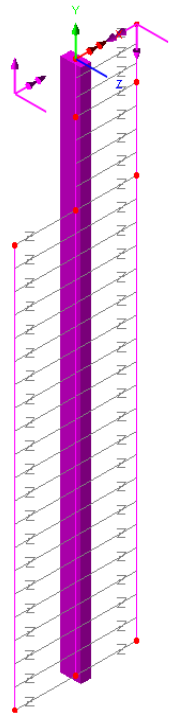
Material Properties

Attributes
Material >
Material Library...

- From the drop-down lists select material type **Concrete,** region Europe, Standard grade **EN1992-1-1:2004/2014** and **Grade C40/50** and click **OK** to add the material dataset to the Treeview.

- Select the three lines representing the wall and drag and drop the geometry attribute **Iso1 (C40/50 | Concrete | EN1992-1-1:2004/2014)** from the 🔗 Treeview onto the selected lines.

Attributes
  Material   >
    Joint...

- In the Joint Material dialog select **Trilinear Earth Pressure** and select **Next**. Three earth pressure material attributes are required: one for the Loose Sand, and two for the Dense Sandy Clay (where one is for the retained side and one is for the excavated side).

- All materials are to be defined for use with **Two dimensional joint elements** (which is set by default) with the defining geometry being **Lines**.
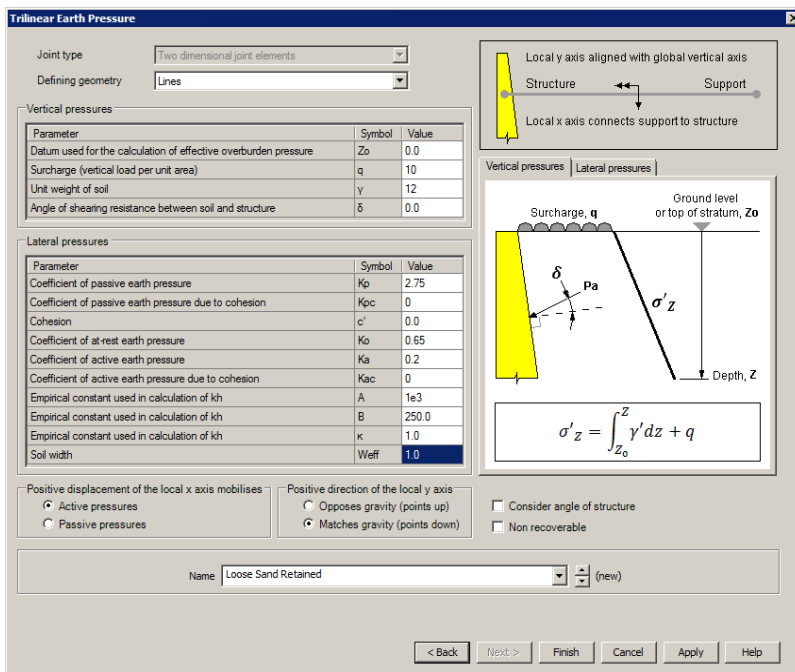
To create the Loose Sand material, enter the following properties:

Datum used for the calculation of effective overburden pressure	**Zo**	0.0
Surcharge (vertical load per unit area)	**q**	10.0
Unit weight of soil	**γ**	12.0
Angle of shearing resistance between soil and structure	**δ**	0.0

Coefficient of passive earth pressure	**Kp**	2.75
Coefficient of passive earth pressure due to cohesion	**Kpc**	0.0
Cohesion	**c'**	0.0
Coefficient of at-rest earth pressure	**Ko**	0.65
Coefficient of passive earth pressure due to cohesion	**Ka**	0.20
Coefficient of active earth pressure due to cohesion	**Kac**	0.0
Empirical constant used in calculation of kh	**A**	1E3
Empirical constant used in calculation of kh	**B**	250.0
Empirical constant used in calculation of kh	**κ**	1.0
Soil width	**Weff**	1.0

- Observing the local coordinate system established for the retained side, select **Active pressures** to define just what the positive displacement of the local x axis mobilises (i.e towards the wall) and **Matches gravity (points down)** to define just what the positive direction of the local y axis represents.

- Deselect **Consider angle of structure**

- Enter a dataset name of **Loose Sand Retained** and click **Apply** to create the attribute.

The dialog should look like this:

Then, to define dense sand, change the parameters to the following:

Datum used for the calculation of effective overburden pressure	**Zo**	-6.5
Surcharge (vertical load per unit area)	**q**	88.0
Unit weight of soil	**γ**	17.0
Angle of shearing resistance between soil and structure	**δ**	0.0

Coefficient of passive earth pressure	**Kp**	3.0
Coefficient of passive earth pressure due to cohesion	**Kpc**	0.0
Cohesion	**c'**	0.0
Coefficient of at-rest earth pressure	**Ko**	0.5
Coefficient of passive earth pressure due to cohesion	**Ka**	0.33
Coefficient of active earth pressure due to cohesion	**Kac**	0.0
Empirical constant used in calculation of kh	**A**	1E3
Empirical constant used in calculation of kh	**B**	250.0
Empirical constant used in calculation of kh	**κ**	1.0
Soil width	**Weff**	1.0

- Enter a dataset name of **Dense Sand Retained** and click **Apply** to create the attribute.

Finally, to define dense sand excavated, change the Surcharge value to **0.0** and observing the local coordinate system established for the excavated side, ensure **Active pressures** is set to define just what the positive displacement of the local x axis mobilises and select **Opposes gravity (points up)** to define just what the positive direction of the local y axis represents. Leave any other settings unchanged. Change the dataset name to **Dense Sand Excavated** and click **Finish** to create the final soil material attribute.

Note. When creating soil layers, if a different soil weight is used for each layer the overburden pressure from any soil layers above must be calculated manually and entered as a surcharge. Where all layers have the same unit weight the datum for each layer can be the same (that is, ground level) remembering that any additional surcharge (q) at ground level needs to be included in the definition of all lower layers. In this example the dense sandy clay on the retained side was given a surcharge of 88kN/m². This was calculated from the depth of sand above plus the applied surcharge such that q = 6.5x12 + 10 = 88

- Select the upper two lines representing the loose sand on the retained side of the wall (not the wall itself) and drag the material attribute **Loose Sand Retained** from the ⚙ Treeview onto the model.

- Select the bottom line representing the dense sand on the retained side of the wall and drag the material attribute **Dense Sand Retained** from the ⚙ Treeview onto the model.

- Select the line representing the dense sand on the excavated side of the wall and drag the material attribute **Dense Sand Excavated** from the ⚙ Treeview onto the model.

Note. Earth pressure joint material attributes are held in the ⚙ Treeview. Context menu entries named Edit Definition... and Edit Attribute... can also be seen by right clicking on an attribute.

❑ Selecting the **Edit Definition...** menu entry or double clicking the attribute name displays the original definition dialog with all the original input data for viewing or editing.

❑ Selecting the **Edit Attribute...** menu entry displays the joint type and stored piecewise linear joint material values that are used to define the earth pressure joint material. Note that whilst these values may be changed, the link to the original definition dialog will be broken.

Supports

For the wall it is assumed that a supported end condition exists.

Attributes
  Support...

- Select a **Fixed** translation for the **Y** axis, enter a dataset name of **Fixed in Y** and click **OK** to create the attribute.

- Select the point representing the bottom of the wall then drag the support attribute **Fixed Y** from the the 🔴 Treeview onto the model for all analysis loadcases.

The Trilinear earth pressure material generates forces in the joint elements appropriate to the position (depth) of the joint element in the model. These forces must be resisted by supporting the free end of the joint elements.

Attributes
  Support...

- Select **Fixed** translations for the **X** and **Y** axes, enter a dataset name of **Fixed** and click **OK** to create the attribute.

- Select the lines representing the excavated and retained soils and drag the support attribute **Fixed** from the 🔴 Treeview onto the model. Accept the defaults in the assignment dialog.
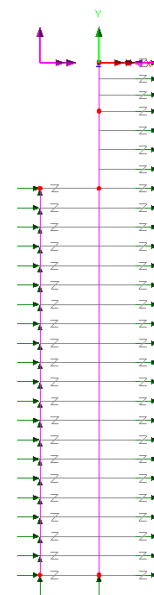
Setting the Analysis

The Trilinear earth pressure material is a nonlinear material and therefore a nonlinear analysis is required.

- In the Analyses 🕒 Treeview, right click on **Loadcase 1**. From the Context menu select **Controls > Nonlinear and Transient**

- In the Nonlinear & Transient dialog check the **Nonlinear** check box (leaving all setting and values as their defaults) and select the **OK** button to return to the model.

Saving the model

File
  Save

💾 Save the model file.

⚠️ **Caution.** For ease of definition the joint elements used have been inserted into the model a set distance away from where they will be acting and, as a result, they have a non-zero length. Note that, by default, Modeller ignores the joint length. In other situations if it is required to solve the model considering the joint length, a geometric attribute should be defined using the Attributes > Geometric > Joint menu item, and selecting the 'Use joint length' option, before assigning the attribute to the three lines representing the wall that were defined to be the 'master; for the assigned line mesh attributes earlier in the example.

Running the Analysis

With the model loaded:

 Select the **Solve Now** button from the toolbar and click **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.

Note. For this analysis the use of non-coincident joints (i.e. omitting the merge step) and linear beam elements within a nonlinear analysis will cause warnings to be written to the text output window. These warnings can be ignored.

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 where the model file resides:

❑ **Retaining Wall~Analysis 1.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.

❑ **Retaining Wall~Analysis 1.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, information relating to the nature of the error encountered can be written to an output file in addition to the text output window. Any errors listed in the text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis.

Rebuilding a Model

If it proves impossible for you to correct the errors reported a file is provided to enable you to re-create the model from scratch and run an analysis successfully.
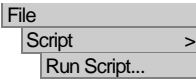
❑ **embedded_wall _modelling.vbs** carries out the modelling of the example.

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 **Embedded Wall**

File	
Script	>
Run Script...	

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

Rerun the analysis to generate the results.

Viewing the Results

Analysis loadcase results are present in the 🕒 Treeview.

A Deformed mesh layer will be added to the Layers 🔲 Treeview:

- Double-click on the **Deformed Mesh** layer in the 🔲 Treeview, and in the **Mesh** tab set the pen to be **blue**

- Turn on the **Diagrams** layer in the Layers 🔲 Treeview. In the Diagrams Properties dialog set the Entity to **Force/Moment** and the Component to **Mz**

- In the panel at the bottom of the Layers 🔲 Treeview tick **Window Summary**.
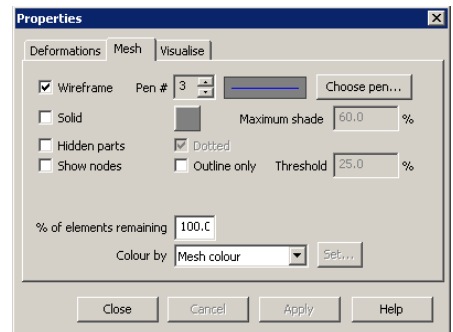
A maximum Bending Moment of 343kNm can be seen to occur in the retaining wall.

Double click the **Diagrams** item in the Layers 🔲 Treeview. In the Diagrams Properties dialog set the Entity to **Force/Moment** and the Component to **Fy**

A maximum Shear Force of 78kN is seen to occur in the retaining wall.

13

- Turn on the **Values** layer in the Layers ⬜ Treeview. In the Values Properties dialog set the Entity to **Displacement** and the Component to **DX**

- On the **Values Display** tab check **Show values of Selection,** check the **Deform** option, set a font **Angle of 45º**, click the choose font button and select a **font size of 16**, click the **OK** button to return to the Properties dialog and click the **OK** button again to return to the model.

- Select the top-most node of the deformed mesh to display the maximum deflection. A maximum deflection of 100mm is seen to occur.

File
Save

- 💾 Save the model file.

- In the Layers ⬜ Treeview turn-off the display of the **Diagrams**, **Values** and **Deformed mesh** layers.

Loadcase 2: Adding a propping force

For this example the previously calculated deflection of the wall has been considered to be too large and the wall is to be propped at a depth of 2.5m from the top of the wall. The propping force will be modelled using a concentrated load that will be increased incrementally to calculate the propping force required to reduce the deflection to 30mm. The calculated propping force can then be used to design the anchoring system.

Attributes
Loading...

- Select **Concentrated** and click **Next**

- Enter a value of **10** in the **X direction,** enter a dataset name of **Prop Force** and click the **Finish** button to create the attribute

Analyses
Loadicase...

- Click **OK** to create a second loadcase with the default name Loadcase 2
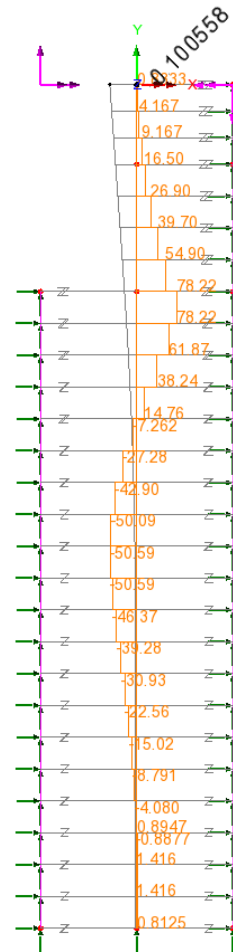
- Select the prop point on the wall at (0,-2.5).

- Drag and drop the loading dataset **Prop Force** from the 🔗 Treeview onto the model

- In the Loading Assignment dialog select **Single loadcase**, select **Loadcase 2** from the Loadcase list and ensure that **Set loadcase active** is checked.

- Click the **OK** button to make the assignment

- In the Analyses 🕐 Treeview, right click on loadcase 2. From the Context menu select **Controls > Nonlinear and Transient**

- In the Nonlinear & Transient dialog check the **Nonlinear** check box and select **Automatic** Incrementation.

- Set the **Starting load factor** to **1.0**, set the **Max change in load factor** to **1.0** and the **Max total load factor** to **10.0**

- Click the **OK** Button to make the changes and return to the model.

- ⬛ Select the **Solve Now** button from the toolbar and click **OK** to run the analysis.
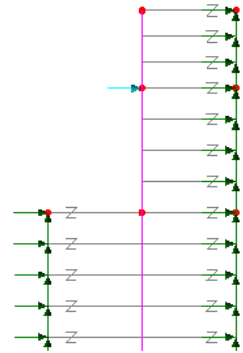
Loadcase results for Loadcase 2 will be added to the 🕐 Treeview. The last solved nonlinear increment (Load factor = 10.0) will be set active by default.

Designing the wall

- In the Layers 🔲 Treeview ensure the display of the **Diagrams**, **Values** and **Deformed mesh** layers are turned on.

- Select the top node of the deformed wall. See following image.

- With the values layer still set to display the horizontal displacement (DX), set each of the load increments (load factors) of **Loadcase 2** active in turn to investigate the effect of applying different prop forces to the top of the wall.

From examination it can be seen that a 30mm top of wall displacement is achieved at **Increment 6**. This increment has a load factor of 5.0 showing a required propping force of 5x10 = 50kN.

- Turn-on the **Diagrams** layer in the Layers Treeview. Double click the Diagrams entry and in the Diagrams Properties dialog ensure that entity **Force/Moment** and component **Mz** are selected.

The moments in the wall can now be seen to be significantly reduced, with the maximum being around 78kNm

Note. This example shows a prop force applied externally to the wall. If a soil anchoring system were to be used to provide the restraint, a different modelling approach would be necessary. In addition, the propping force required to reduce the wall deflections may not be achievable in practice due to the soil type. In this case, the design may require some iteration and re-running of the analysis to achieve acceptable forces, moments and deflections.
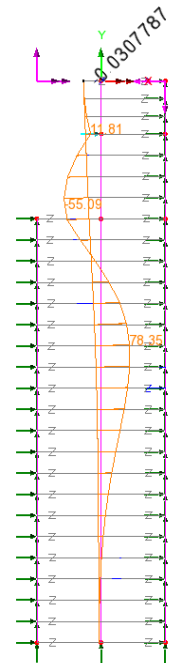
Looking to reduce the pile length

Once suitable deflections and design forces have been established the required length of pile can be determined by reducing the embedded length and ensuring the effect on the required deflection is acceptable.

An initial estimate of the reduction can be sought by considering where wall forces become negligible. To do this a diagram of Shear force in the wall is to be plotted.

- In the Diagrams Properties dialog set the Entity to **Force/Moment** and the Component to **Fy**

By inspection the shear within the wall for the bottom 5m length is negligible and can be removed.

Geometry
Point   >
  Move...

- Select the bottom three points of the model and enter **5** in **Y translation**. Click **OK** to reduce the wall depth.
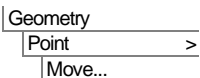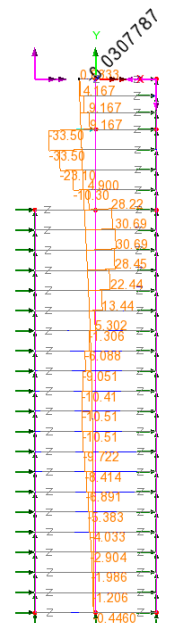
Note. Because results are currently loaded whilst the geometry has been modified, the mesh will have been locked and the updated joint mesh arrangement will not be immediately visible. The mesh assignments will still be present for those lines and the model will solve, but to see the updated mesh arrangement the mesh can be unlocked prior to a re-solve taking place:

Tools	
Mesh	>
Mesh lock...	
File	
Save As…	
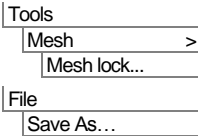
This will display the updated mesh arrangement. If it is required to see the joint element mesh, the Mesh layer would need to be turned-on.
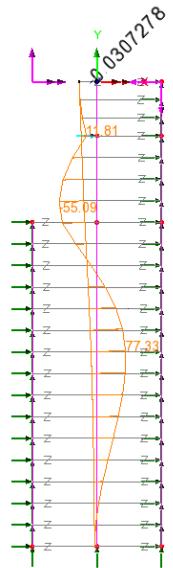
Save the model file with a different name, such as **embedded_wall_reduced.mdl**

- Select the **Solve Now** button from the toolbar and click **OK** to re-run the analysis.

- With the top node of the wall selected, and the values layer still set to display the horizontal displacement (DX), and the diagrams layer set to show Force / Moment of My, set each of the load increments (load factors) of Loadcase 2 active in turn to investigate the effect of applying different prop forces to the top of the wall.

From examination it can be seen that a 30mm top of wall displacement is still achieved at **Increment 6** (with a load factor 5.0) so the reduction in length has not altered the top of wall displacement by any appreciable amount.

Maximum bending moment in the wall is similar to that previously achieved at 77kNm.

Further reduction in embedded length maybe possible by repeating the previous steps and ensuring deflections remain suitable.

This concludes the example.