Embedded Retaining Wall

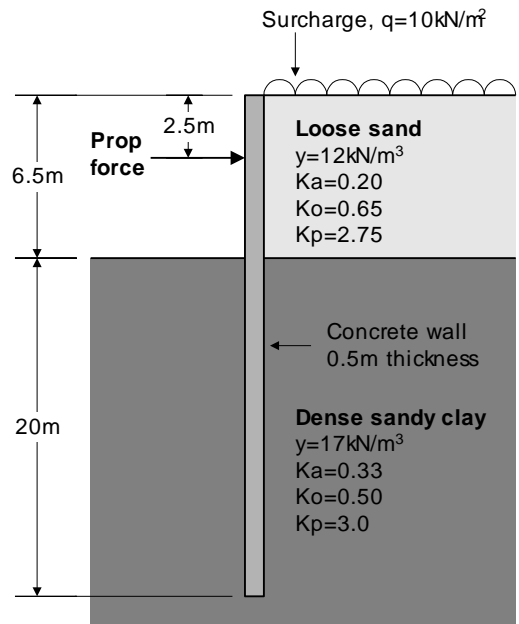| For LUSAS version:	16.0
For software product(s):	LUSAS *Civil & Structural* or LUSAS *Bridge.*
With product option(s):	Nonlinear.

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/m²

Prop force 2.5m

6.5m

Loose sand
y=12kN/m³
Ka=0.20
Ko=0.65
Kp=2.75

Concrete wall
0.5m thickness

20m

Dense sandy clay
y=17kN/m³
Ka=0.33
Ko=0.50
Kp=3.0

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.

- Select the newly created Point.

Geometry	
Surface	>
Sweep…	

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	
Surface	>
Sweep…	

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	
Surface	>
Sweep…	

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

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

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 (Ctrl + A)

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 model the wall. The lines to either side are used to model the soil. A load to model 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.
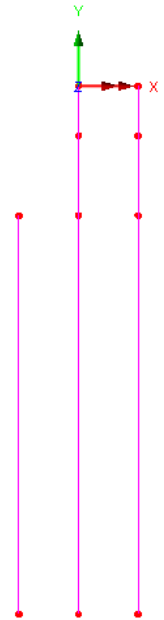
Meshing

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 is defined in the new model dialog and can be accessed subsequently by the Utilities > 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**
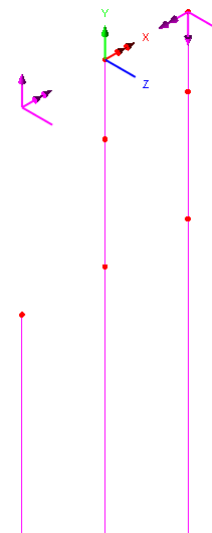
Defining element types

The joint elements are required to have displacements in the u and v directions to represent horizontal and vertical pressures. For this 2D problem the JNT3 element with only translational freedoms is sufficient.

Attributes	
Mesh	>
Line...	

- Select the element type as **Joint no rotational stiffness**, the number of dimensions as **2 dimensional** 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**, the number of dimensions as **2 dimensional** 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.

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

- Select the left-hand line representing the excavated earth and then, at the same time, 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

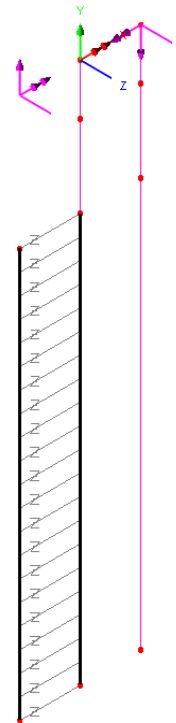To assign the mesh for the retained side:

- Select 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

Add the lines to the selection memory.

- Select 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

Edit
 Selection Memory >
  Clear

This clears the selection memory.

> **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.
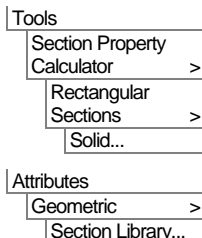
Geometric Properties

- 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.

- 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.
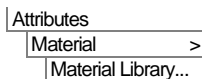
Geometric properties are visualised by default.

- In the 🔲 Treeview re-order the layers 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.
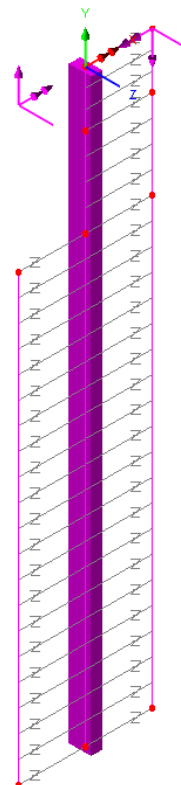
Material Properties

- From teh 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.

- 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** with the defining geometry being **Lines**.
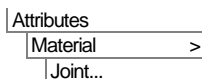
To create the Loose Sand material, enter the following properties:

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.

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.

Note. 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.

Note. 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. These values may be changed but the link to the original definition dialog will be broken.

Supports

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

- 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.

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.

- Select **Fixed** translations for the **X**, **Y** and **Z** 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.
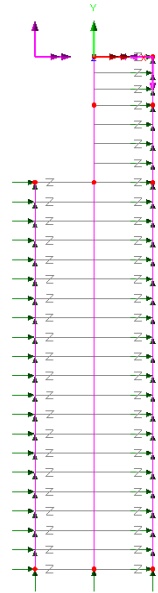
Setting the Analysis

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

💾 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. Prior to an analysis taking place, the lines <u>and</u> points representing the soil should normally be set to be unmergeable prior to being moved to <u>overlay</u> the line representing the wall. For some modelling situations the use of joint elements with a

non-zero length will produce unreliable results, and in general joint elements should always be modelled with a zero-length. For clarity in this example merging has been omitted, as there is no significant difference in the results obtained.

To model joints with zero-length

To correctly model the soil joints the joint elements would normally be modelled with zero-length. For this example this would be done as follows:

- Select the lines on either side of the wall that represent the soil.

- Select **Geometry > Line> Make unmergable**

- Select **Geometry > Point> Make unmergable** (because making a line unmergable does not make the points defining it unmergable too.)

- Select the left-hand line representing the soil and select **Geometry > Point> Make unmergable** and move the line **3** metres in the **X** direction

- Select the right-hand lines representing the soil and select **Geometry > Point> Make unmergable** and move the line **-3** metres in the **X** direction

- Save the model with an appropriate filename.

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.
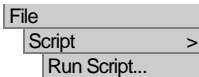
❑ **embedded_wall _modelling.vbs** carries out the modelling of the example.
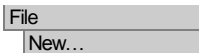
File
New…

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 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 – Thick 2D Beam** and the Component to **Mz**

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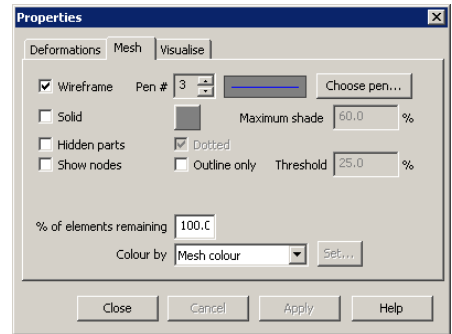
A maximum Bending Moment of 343kNm can be seen to occur in the retaining wall.

Double click the **Diagrams** item in the Layers Treeview. In the Diagrams Properties dialog set the Entity to **Force/Moment – Thick 2D Beam** and the Component to **Fy**

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

- 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 45 º**, 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.
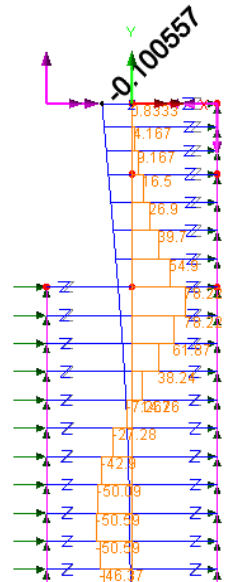
 Save the model file.

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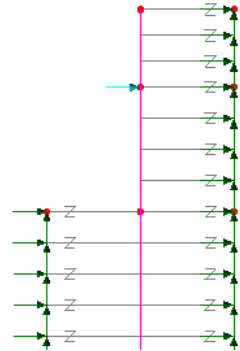
- In the Nonlinear & Transient dialog check the **Nonlinear** check box and select **Automatic** Incrementation.

- 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.

- With the top node of the wall selected (and the values still set to display the horizontal displacement), 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 leading to a required propping force of 5x10 = 50kN.

- Double click the **Diagrams** layer in the Layers Treeview. In the Diagrams Properties dialog set the Entity to **Force/Moment – Thick 2D Beam** and the Component to **Mz**

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

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 – Thick 2D Beam** 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...	
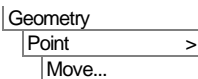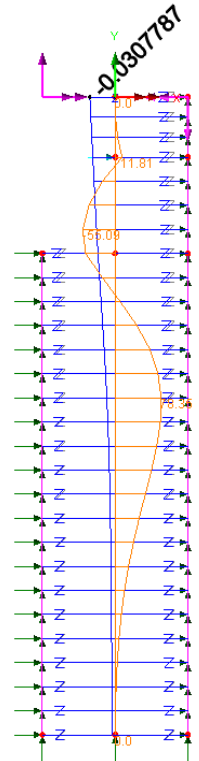
 Select the bottom three points of the model and enter **5** in **Y translation**. Click **OK** to reduce the wall depth.

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:
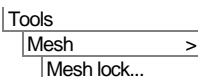
This will display the updated mesh arrangement.

Tools	
Mesh	>
Mesh lock...	

Save the model file with a different name, such as embedded_wall_reduced.mdl if desired.

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 displacement in X, 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** so the reduction in length has not altered the top of wall displacement. Maximum bending moment in the wall is similarly unaltered at 77kNm.

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

This concludes the example.