

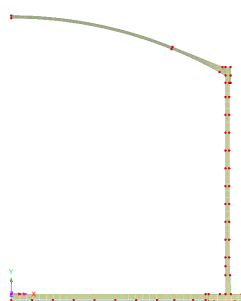
Storage Tank Modelling – 2D and 3D Concrete Models

For LUSAS version:	22.0
For software product(s):	LUSAS Tank
With product option(s):	Nonlinear, Dynamic, Thermal/Field, RC Slab design

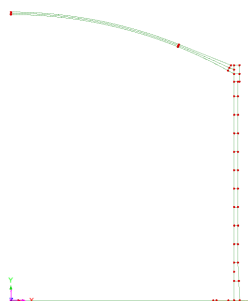
Description

The LUSAS Tank software product allows you to automatically create a range of 2D and 3D finite element models of above ground, circular, full containment, concrete or double-walled steel tanks from user-defined common tank definition data.

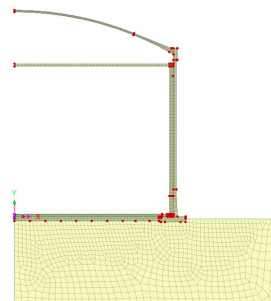
The range of concrete models include:



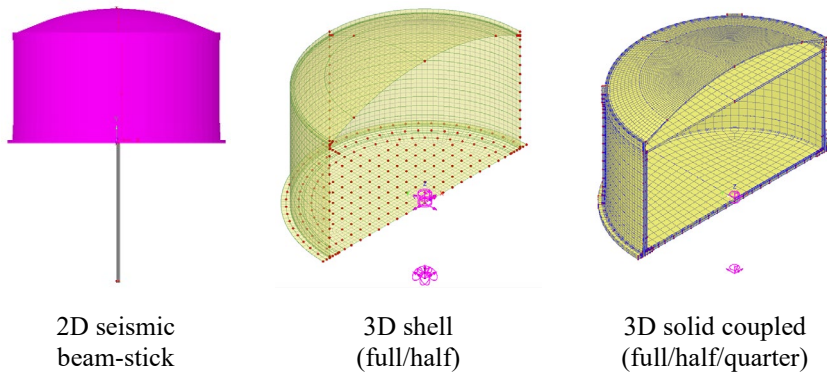
2D axisymmetric
structural



2D axisymmetric
staged construction



2D axisymmetric
coupled



For clarity, loading and supports have been omitted from the above models.

Using these models, a range of analyses can be performed, and design checks can be optionally carried out for specified load combinations and supported design codes.

Benefits of multiple models

The ability to define different types of models from common tank definition data allows for preliminary studies to be done using any of the 2D tank modelling options that are available, before moving on developing and investigating the suitability of designs in more detail using 3D shell or 3D solid models.

Comparative studies can be achieved by copying tank definitions and re-creating models from revised data. Design checks are performed using 3D shell models.

Objective

- ☐ To carry out illustrative 2D and 3D modelling and analysis of concrete tanks.
- ☐ To provide an overview of viewing design results from a 3D shell model.

Keywords

Tank, 2D, 3D, Axisymmetric, Shell, Solid, Concrete, Thermal, Coupled, Structural, Seismic, Staged Construction, Design, Spillage.

Running LUSAS Modeller

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

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

Creating a New Model

File
New...

- Enter a file name of **tank**
- Use the default **User-defined** working folder.
- Ensure an Analysis type of **3D** is set. This is somewhat irrelevant since the analysis type will be automatically set when each tank model is created by LUSAS Tank.
- Click the **OK** button.

Defining tank definition data

LUSAS Tank uses modelling units of 'N,m,kg,s,C'. If these are not in use when creating the Tank Definition, a statement advising that the units will be changed will be displayed.

Tank
Tank definition...

The Tank Definition dialog contains default values and settings to build an illustrative tank of a chosen tank type. For these worked examples, concrete tank models will be generated using default data.

Tank type

Material: Concrete

Elevation: Above ground tank

Target models to build

☒ 2D axisymmetric structural
 ☒ 2D axisymmetric coupled thermal/structural
 ☒ 3D solid

☒ 2D beam-stick seismic
 ☒ 3D shell structural
 ☐ 3D shell roof-only

Tank Definition

Load

Prestress

Insulation

Support (3D)

Seismic

Ground

Base Slab and Roof

Wall and Ring beam

Materials

Support (2D)

Base slab (Units: m)

Circular part length (L_{inner})

39.8

Circular part depth (D_{inner})

1.2

Tapered section length (W_t)

0.6

Annular part length (L_{outer})

6.7

Annular part depth (D_{outer})

1.5

Base heating ($D_{heating}$)

0.386

Base heating ($L_{heating}$)

46.5

Ground level (D_{ground})

0.9

Roof (Units: m)

Radius of inner roof ($R_{roof,i}$)

86.406

Radius of outer roof ($R_{roof,o}$)

86.906

Height from the top of the base slab to the topmost of the roof (R_{Height})

56.2545

Distance of tapered section 1 ($sl1$)

10.079

Distance of tapered section 2 ($sl2$)

0.6

Set zero

Set defaults

Name concrete tank (new)

OK

Cancel

Apply

Help

- Ensure the tank type material is set to **Concrete** and the elevation is set to **Above ground tank**



Note. For concrete tanks, in the ‘Target model to build’ panel, the ‘2D axisymmetric structural’ option is always ‘on’ by default since it requires the minimum amount of data in order to create a tank model. Other tank types may be selected alongside this option, requiring more data to be supplied.

To allow all types of model to be generated, in the ‘Target model to build’ panel:

- Select the **2D axisymmetric coupled/thermal structural** checkbox. This adds an ‘Insulation’ button to the dialog.
- Select the **2D beam-stick seismic** check box. This adds a ‘Seismic’ and ‘Ground’ button to the dialog.
- Select the **3D shell structural** check box. This adds a ‘Support (3D)’ button to the dialog.
- Select the **3D solid** check box. This then ‘greys-out’ the 2D thermal and 3D shell models checkbox options since all data required by these models will be provided by that defined for the 3D solid model.
- Beneath the check boxes **click on each button in turn**, and then on the tabs that appear for each button, to browse the example data that is provided, and hence see the type of data that is required to model each tank type.
- Enter a tank definition name of **concrete tank** and press the **OK** button.

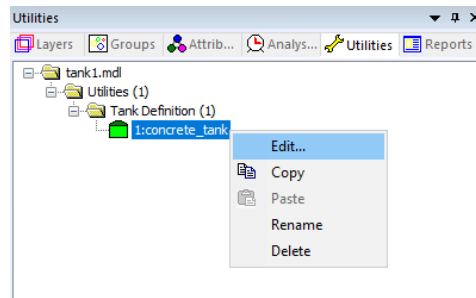
A concrete tank definition entry will be added to the Utilities  Treeview.



Note. Models of metallic tanks may also be created using **LUSAS Tank**. For these, no 3D solid model is available but all other target models as well as a 3D shell roof-only model can be created. These types of tanks are not covered by this example, but model creation after entering all relevant details is very similar.

Editing tank definition data

Changes to the tank definition data for this range of tank models can be made within the Utilities  Treeview by double-clicking on the created utility or using the ‘Edit’ option on the context menu provided.



Note. Editing of previously defined tank data can only be achieved by using this context menu.



Note. Re-selection of the menu item **Tank > Tank Definition** will generate a new tank definition each time.

Concrete tank model examples available

- ☐ [2D axisymmetric structural](#)
- ☐ [2D axisymmetric staged construction](#)
- ☐ [2D axisymmetric coupled thermal / structural](#)
- ☐ [2D beam-stick seismic](#)
- ☐ [3D shell structural](#)
- ☐ [3D solid coupled thermal / structural](#)

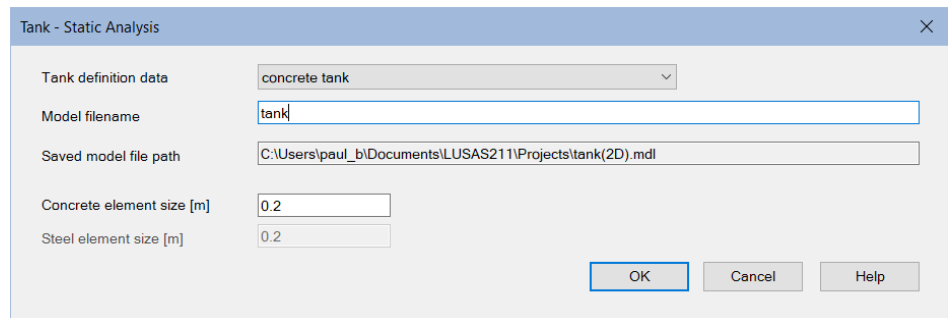
Whilst each example can be undertaken in isolation from the others, it is recommended to work through each example as listed as they are in order of complexity.

2D Axisymmetric Structural Tank Model

Creating the model

After having defined the relevant tank data, a 2D axisymmetric structural analysis tank model will be created first.

Tank
Create 2D Model
Structural



Tip. Press the **Help** button for more information about this dialog and the type of model that is created.

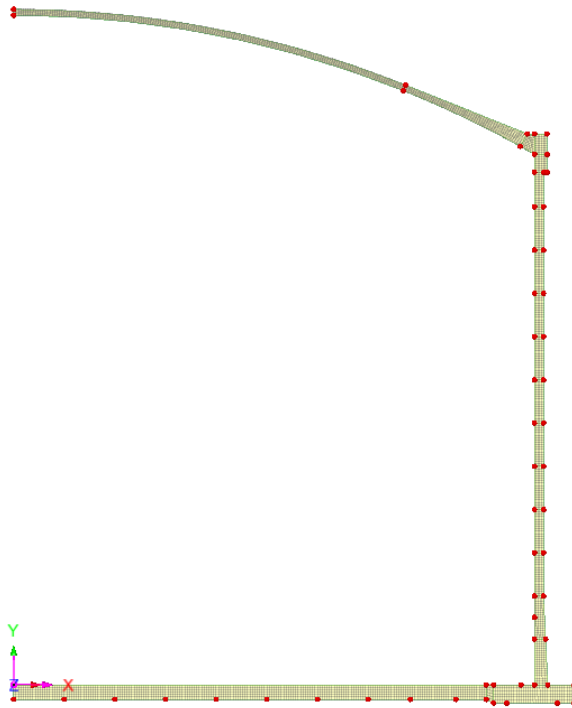
- Accepting the default values on the dialog, enter a model filename of **tank**



Note. The filename entered will be appended with '(2D)' to make a unique filename for a range of tanks that use the same base name.

- Click **OK**, and then click **OK** again to save any changes to the previously defined model. The saved tank definition will be copied in the created model.

The following model will be created.



Note. Browsing the Groups , Attributes  and Analyses  treeviews will show the groups of features added, the attributes created and assigned to the model, and the analyses and loadcases defined.



Note. Context menus for the items present can be used to locate features assigned to groups and to visualise attribute assignments.

Viewing assigned loadings



By ensuring the loading arrows are being drawn, and setting active each loadcase in turn, the loadings assigned to the model can be viewed and checked.

Running the analysis

With the model loaded:

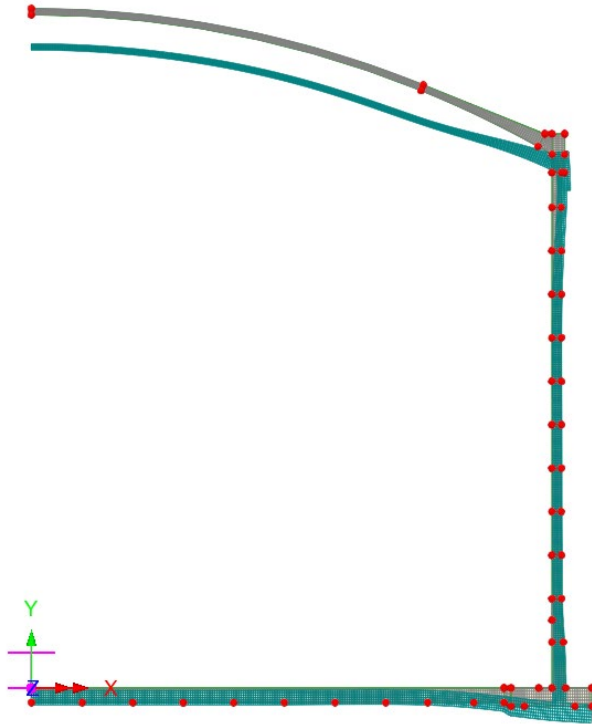


Select the **Solve Now** button from the toolbar and click **OK** to run the analyses listed.

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 Analyses  Treeview and a deformed mesh layer is added to the view window.



Viewing the results

- For clarity when viewing the contour results, in the Layers  Treeview, turn off the display of the **Mesh** layer and the **Deformed mesh** layer.
- In the Analyses  Treeview, ensure the loadcase **SelfWeight** is set active.

Contours

- With no features selected, click the right-hand mouse button in a blank part of the view window and select the **Contours** option to add the contours layer to the  Treeview.

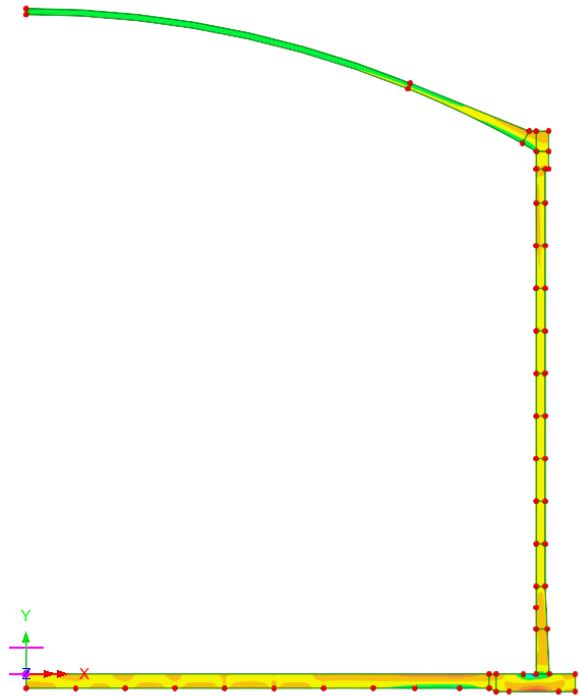
2D Axisymmetric Structural Tank Model

- Select entity **Stress – Axisymmetric Solid** and component **SX** and click **OK**.

Analysis: Analysis 1
Loadcase: 1:SelfWeight
Results file: tank_2D_~Analysis 1.mys
Entity: Stress - Axisymmetric Solid
Component (Averaged nodal): SX (Units: N/m²)

	-4.06271E6
	-3.25017E6
	-2.43762E6
	-1.62508E6
	-812.541E3
	0.0
	812.541E3
	1.62508E6
	2.43762E6

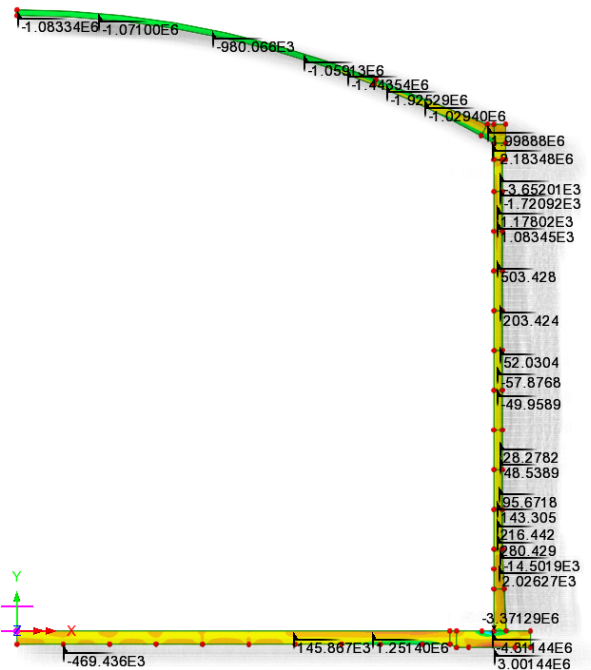
Maximum 3.00144E6 at node 1433 of element 1314
Minimum -4.31144E6 at node 1473 of element 1303



Values

- With no features selected, click the right-hand mouse button in a blank part of the view window and select the **Values** option to add the contours layer to the  Treeview.
- Select entity **Stress- Axisymmetric Solid** and component **SX** and click **OK**.

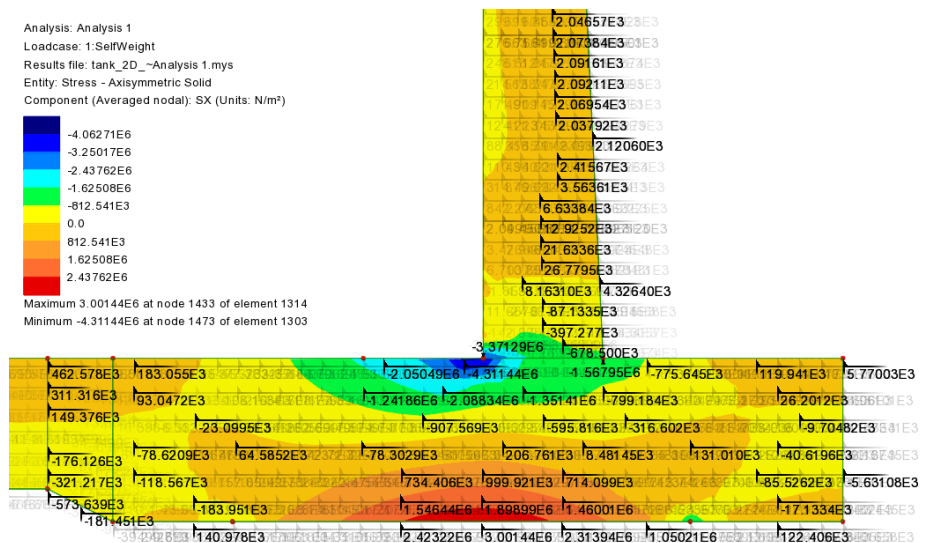
Maximum 3.00144E6 at node 1433 of element 1314
Minimum -4.31144E6 at node 1473 of element 1303

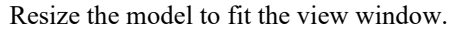


Zooming-in to regions of interest will provide more detail and values.

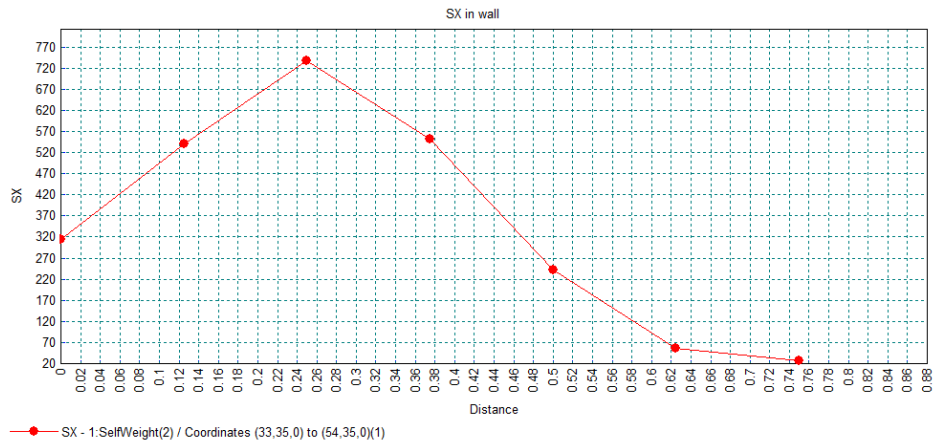
-4.06271E6
 -3.25017E6
 -2.43762E6
 -1.62508E6
 -812.541E3
 0.0
 812.541E3
 1.62508E6
 2.43762E6

Maximum 3.00144E6 at node 1433 of element 1314
 Minimum -4.31144E6 at node 1473 of element 1303





Graph through 2D



- Reselect the View window tab.

To plot a graph of resultant forces through the wall at the same location:

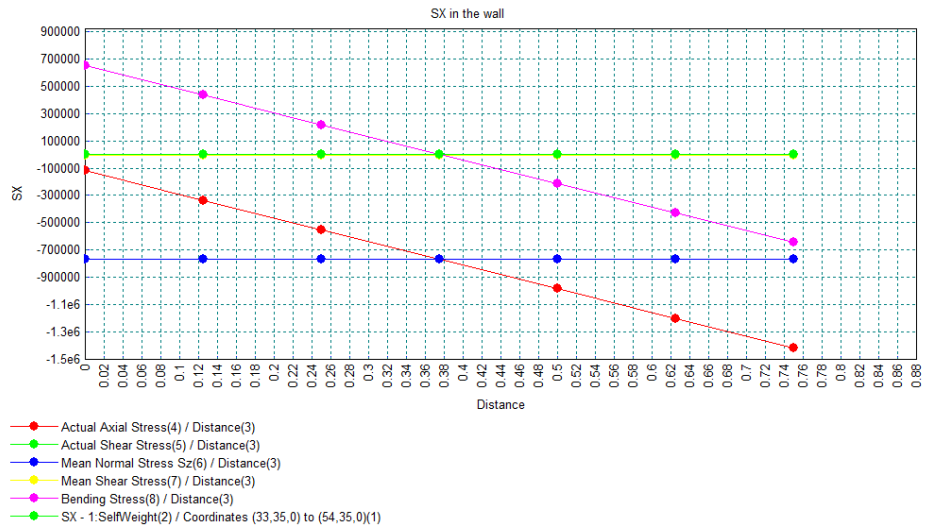
- Select **At location of existing graph** and click **OK**.
- On the Loadcases and Extent dialog ensure **SelfWeight** is selected and click **Next**.
- On the Slice data dialog select **Resultant effects from 2D model** option and click **Next**.
- On the Display Graph dialog enter a graph title of **SX in the wall**, an X axis name of **Distance**, and a Y axis name of **SX** and click **Finish**.

Resultant forces are computed and printed in the text output window, and a graph showing the variation of forces at the wall thickness is generated.

Utilities

Graph through 2D

2D Axisymmetric Structural Tank Model



- Reselect the View window.

Export Forces to Excel (2D)

Forces calculated for one or more model loadcases can be exported to an Excel spreadsheet.

Tank
Excel Tools
Export
Forces/Design
Results

Tank - Export Forces/Moments to Excel (2D)

Output filename:

Working folder: ☒ Current ☐ User Defined

Save in:

Target: ☐ Base slab ☒ Wall + Ringbeam ☐ Roof

Loadcases:

- 1: Self Weight
- 2: Dead loads of steel structure
- 3: Dead load of liner and steel roof
- 4: Dead load of steel structures on the roof
- 5: Dead load of insulation
- 6: Pressure on outer tank wall due to insulation
- 7: Wall piping loading
- 8: Liquid bottom (max)
- 9: Liquid bottom (min)
- 10: Gas pressure (max)
- 11: Gas pressure (min)
- 12: Live load
- 13: Snow load
- 14: Test load (liquid)

Range (Y Coord):

Start: m

Finish: m

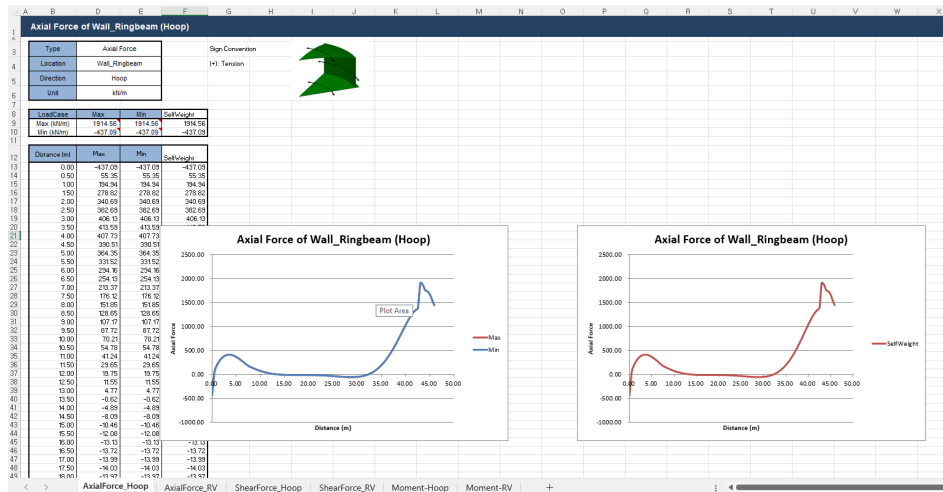
Interval: m

- Enter an output filename of **2d_axi_structural** (noting that the target name is appended to the name entered).
- For a target, choose **Wall + Ringbeam**
- In the Loadcases panel, ensure **Self weight** is selected and press **OK**

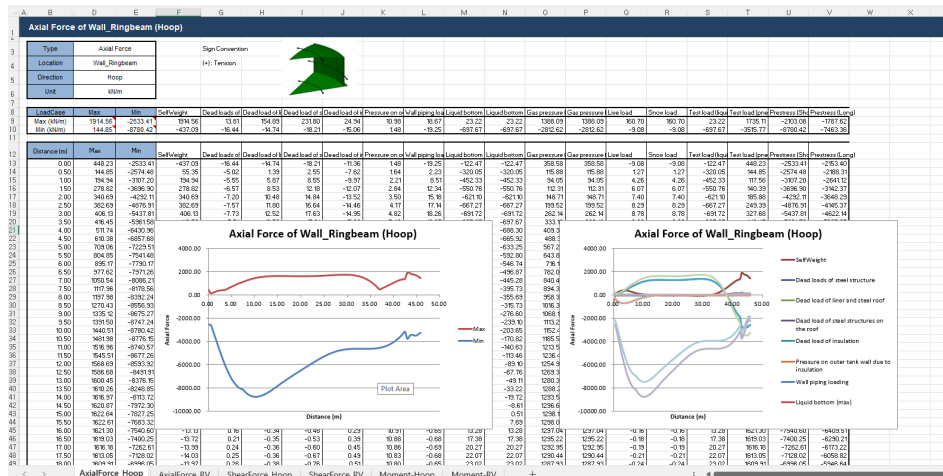
A spreadsheet containing section forces including axial force, shear force, moment force for Wall & RingBeam will be created.

- Open the working folder to view this file.

2D Axisymmetric Structural Tank Model



Note. If all the loadcases from the Loadcases panel were to be selected, the forces for all loadcases are computed, resulting in the following output.



This completes this tank example.

2D Axisymmetric Staged Construction Tank Model

Creating the model

To create a staged construction tank model from previously defined tank definition data:

Tank
Create 2D Model
Staged
Construction

2D Axisymmetric Staged Construction Tank Model

Tank - Staged Construction Analysis

Tank definition data: concrete tank

Model filename: tank

Saved model file path: C:\Users\paul_b\Documents\LUSAS220\Projects\tank(2D Staged).mdl

Modelling options

Concrete element size [m]: 0.2 Steel element size [m]: 0.2

Loads to apply

☒ Self weight ☒ Structural loadings (Variable Loads : ☒ Max ☐ Min)

Concrete Tank Options

Roof / Ringbeam

Roof construction plan: Layered roof option 1

Roof first stage thickness (ratio): 0.5

Initial prestress for ringbeam (ratio): 0.5

Initial prestress for slab (ratio): 0.7

* Roof frame loads are not considered

* Roof first stage wet concrete is not considered

Construction Scenario - Layered roof option 1

- 1 - Base / Wall / Ringbeam
- 2 - Ringbeam 1st PS
- 3 - Roof frame 1 / Inner work
- 4 - Roof frames 2,3
- 5 - Roof lower wet / Roof Lower complete
- 6 - Roof upper wet / Roof complete
- 7 - Ringbeam 2nd PS
- 8 - Wall vertical PS
- 9 - Wall horizontal PS

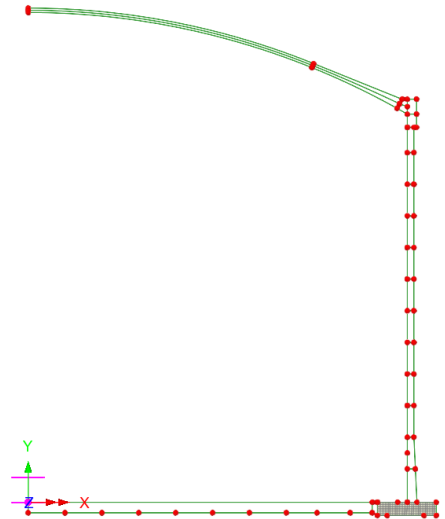
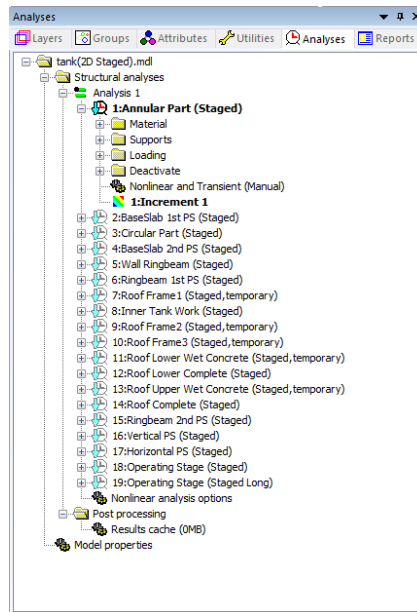
OK Cancel Help



Tip. Press the **Help** button for more information about this dialog and the type of model that is created.

- Accepting the default values on the dialog, enter a model filename of **tank** and click **OK**. The saved model name will include '(2D Staged)' to identify this model within the set of tank models that are being created.
- Click **OK** to save any changes to the previously defined model.
- Click **Yes** to accept that the wall will be constructed all at once and not in stages.

The following Analyses  Treeview and model will be created.



Note. Browsing the Groups , Attributes  and Analyses  Treeviews will show the groups of features added, the attributes created and assigned to the model, and the analyses and loadcases defined.



Note. Context menus for the items present can be used to locate features assigned to groups and to visualise attribute assignments.

Tank construction stages – an overview

Tank construction stages are created using activation and deactivation of elements and a nonlinear analysis sequence which inherits the stresses and strains from the previous stages. All materials are assumed to be linear elastic.

The total number of construction stages to be created will be as stated on the ‘Staged construction analysis’ dialog according to the chosen ‘Roof construction plan’, the ‘Roof 1st stage thickness (ratio)’, the ‘Initial Prestress for Ringbeam (ratio)’ and the ‘Initial Prestress for slab (ratio)’ plus any wall construction stages as defined within the Tank definition dialog itself.



Note. For this overview example no wall construction stages are going to be considered. If these were to be introduced as part of a construction check, they would be defined by visiting the ‘Wall and Ringbeam’ tab of the Tank Definition dialog and pressing the ‘Wall stages’ button.

2D Axisymmetric Staged Construction Tank Model

Tank Definition

Tank type
Material : Concrete
Elevation : Above ground tank

Target models to build
☒ 2D axisymmetric structural
☒ 2D axisymmetric coupled thermal/structural
☒ 3D solid
☒ 2D beam-stick seismic
☒ 3D shell structural
☐ 3D shell roof-only

Tank Definition | Load | Prestress | Insulation | Support (3D) | Seismic | Ground

Base Slab and Roof | Wall and Ring beam | Materials | Support (2D)

Wall and ring beam (Units: m)

Radius to inside face of outer tank wall (InsR)	43.2
Thickness of wall base (T_bottom)	1.1
Height of tapered wall (H_wall_t)	7.4
Thickness of wall top (T_top)	0.75
Height of wall (H_wall)	42.68
Height of ringbeam_2 (H_ringbeam_2)	1.7
Height of ringbeam_1 (H_ringbeam_1)	1.5
Thickness of ringbeam (T_ringbeam)	1.05
Slope height (R_sl_height)	0.0

Corner Protection (Units: m)

Corner protection start (H_bcp_s)*	0.617
Corner protection end (H_bcp_e)*	5.617
Corner protection thickness (T_bcp)*	0.155

Set zero | Set defaults | Wall stages | Openings

Name: concrete tank (1)

OK | Cancel | Apply | Help

* Guidance for corner protection inputs based on the current insulation data
- Corner protection start: 0.105,0.567,0.617,0.6915
- Corner protection end: 5.617
- Corner protection thickness: 0.155

This would result in this dialog being displayed (that shows example inputs).

Wall Construction Stages

Height / Stages

Wall stage ID	Height (H)	Stage Y/N
1	7	Y
2	7	Y
3	7	Y
4	7	Y
5	7	Y
6	7.68	Y

Set defaults
Clear grid
Add
Remove

OK | Cancel | Apply | Help

Viewing construction stages

Construction stages (and the features and elements that comprise those stages) may be viewed by setting active each loadcase in turn within the Analyses Treeview of the generated model.

Viewing assigned loadings



By ensuring the loading arrows are being drawn, and setting active each loadcase in turn, the loadings assigned to the model can be viewed and checked.

Nonlinear Analysis Controls

The geometry of the structure changes at each loadcase (each stage), so a Nonlinear Control is used (appearing as  Nonlinear and Transient (Manual) with the Analyses  Treeview).

When Nonlinear Control is set for the 1st loadcase (as in this case) it is applied to all the other subsequent loadcases unless otherwise defined separately for them.

'Manual' incrementation control is set in the model, which means that:

- ☐ Subsequent loadcases inherit the stress and strains from the previous loadcases
- ☐ Subsequent loadcases inherit the support conditions from the previous loadcases
- ☐ Loading is not inherited.
- In the Analyses  Treeview double click the  Nonlinear and Transient (Manual) entry to see the default settings used

Nonlinear & Transient

Incrementation

☒ Nonlinear

Incrementation

Manual

Starting load factor

0.1

Max change in load factor

0.0

Max total load factor

1.0

☒ Adjust load based on convergence

Iterations per increment

4

Advanced...

☐ Time domain

Two Phase

Initial time step

0.0

Total response time

100.0E6

☐ Automatic time stepping

Advanced...

Solution strategy

☐ Same as previous loadcase

Max number of iterations

12

Residual force norm

0.1

Displacement norm

1.0

Advanced...

Incremental LUSAS file output

☐ Same as previous loadcase

Output file

1

Plot file

1

Restart file

0

Max number of saved restarts

0

Log file

1

History file

1

☐ Save a restart at the end of this control

Common to all

Max time steps or increments

0

OK

Cancel

Help

Loading

Setting a 'Manual' incrementation means that any loading defined in a previous loadcase is not inherited by the following loadcase. As a result, all loading that applies to the current loadcase is and should be assigned separately.

Construction stages explained

With reference to the Analyses  Treeview, note that the loadcase id numbers relate to the stages as described below.

Stage 1: Annular Part + Stage 2: BaseSlab 1st PS

Self-weight is assigned using 'Gravity' loading. The initial prestress loading to the BaseSlab is added in Stage 2. If no prestress is defined for the slab, Stage 2 will be the same as Stage 1.

Stage 3: Circular Part + Stage 4: BaseSlab 2nd PS

The 2nd prestress loading to the BaseSlab is added in Stage 4. If no prestress is defined for the slab, Stage 4 will be the same as Stage 3.

Stage 5: Wall Ringbeam + Stage 6: Ringbeam 1st PS

At Stage 5 the Wall and Ringbeam are completed. The loading is the same as Stage 4.

Initial Horizontal Prestress for the RingBeam is added in Stage 6, but with a load factor of 0.5 (the 'Initial prestress for ringbeam (ratio)' from the model creation dialog) which means only 50% of the defined RingBeam prestress is applied at this stage.

Stage 7: Roof Frame 1 + Stage 8: Inner Tank Work

Stage 7 allows for an additional temporary load to be applied when the temporary frame is installed for preparing the roof 1st concrete pour. If required, the loading for Roof Frame 1 should be defined and assigned to the model manually.

Stage 8 allows for any additional loading to be applied while building the inner tank. If required, this loading should be defined and assigned to the model manually.

Stage 9: Roof Frame 2 + Stage 10: Roof Frame 3

Stage 9 and Stage 10 allows for temporary loads to be assigned when the temporary frame is installed for preparing the roof 2nd concrete pour. If required, the loading for 'Roof Frame 2' and 'Roof Frame 3' should be defined and assigned to the model manually.

Stage 11: Roof Lower Wet Concrete + Stage 12: Roof Lower Complete

Stage 11 assumes that the roof is in the process of being built and the poured concrete is acting as a loading on the ringbeam.

Stage 12 assumes that the lower part of the roof is completed. At this stage the roof lower wet concrete loading assigned at Stage 11 is removed and replaced with the body force of the lower part of the roof.

Stage 13: Roof Upper Wet Concrete + Stage 14: Roof Complete

Stage 13 assumes that the upper part of the roof is being built and the poured concrete is acting as a loading on the lower part of the roof.

Stage 14 assumes that the roof is completed. At this stage the roof upper wet concrete loading assigned at Stage 13 is removed and replaced with the body force of the roof. For this, the weight of the upper part of the roof and the area of the top surface of the Roof Lower Part are computed automatically by the LUSAS Tank to derive a global distributed load, which can be viewed from the entries in the Attributes  or Analyses  Treeviews. If required, this could be verified by assigning self-weight loading to the upper part of the Roof and checking the reaction.

Stage 15: Ringbeam 2nd PS + Stage 16: Vertical PS

The remaining RingBeam prestress is added at Stage 15. (with the load factor now changed from 0.5 to 1.0). The structure is fully built at Stage 15, and the additional loading of the Vertical Prestress is added to Stage 16.

Stage 17: Horizontal PS + Stage 18: Operating Stage

Horizontal Prestress is added to Stage 17. Stage 18 models the operating (in-service) stage.

All the loadings used in the 2D Axisymmetric Static Analysis Model are included in this stage.

The prestress loadings are defined with the values obtained from the tank definition dialog and only the short-term prestress is applied.

Stage 19: Operating Stage (Long)

Stage 19 models the operating (in-service) stage for long-term.

All the loadings used in the 2D Axisymmetric Static Analysis Model are included in this stage.

The prestress loadings are defined with the values obtained from the tank definition dialog and only the long-term prestress is applied.

Running the analysis

With the model loaded:

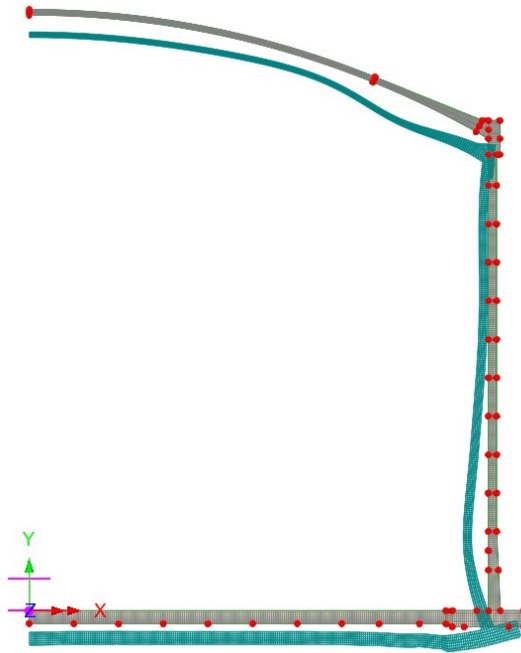


Select the **Solve Now** button from the toolbar and click **OK** to run the analyses listed.

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

Viewing the results

- The deformed mesh plot for the final construction stage is shown below.



- Results can be viewed in a similar manner to that described for the [2D Axisymmetric structural example](#).

Additional notes

Adding extra construction stages

If additional construction stages are required, the loadcase context menu items ‘Copy’ and ‘Paste’ can be used to duplicate construction stages (for renaming). Other loadcase-related attributes such as ‘Activate’ and ‘Loading’ are also copied.

Defining Load Combinations

Whilst load combinations can be defined using general combination facilities, they can be defined more quickly for LUSAS Tank created models by specifying data within a supplied template.

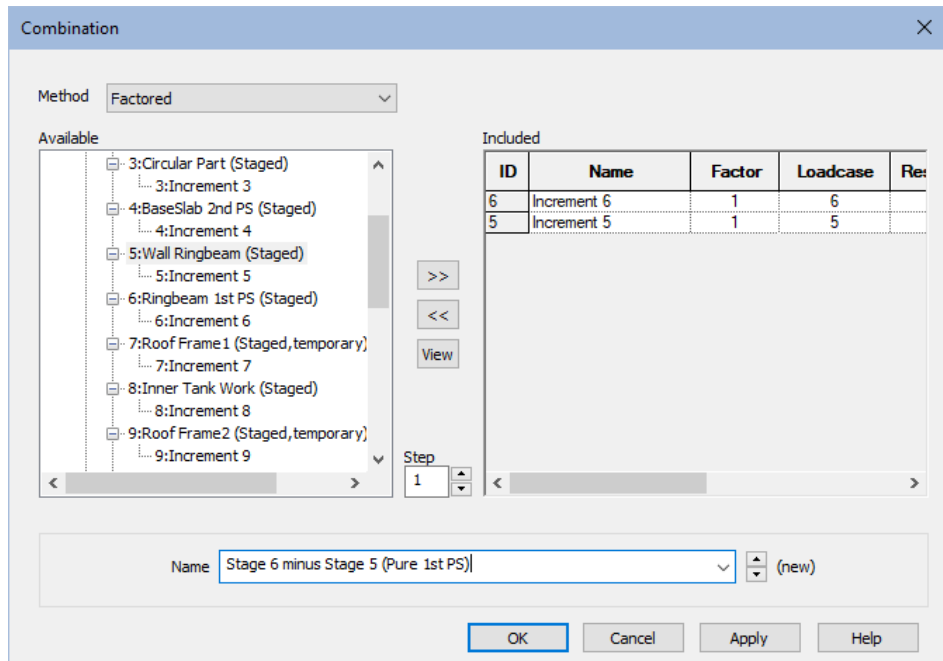
This sample Tank_Template_Combinations.xlsx spreadsheet is provided in the “<LUSAS installation folder>LUSASxxx\Programs\scripts\Tank” folder.

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X
1	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	23	24
2	Permanent														Variable									
3	Description			Outer tank		Others	Creep and Shrinkage		Prestress						Roof frame/ concrete				Test		LNG pressure	Gas pressure		
Outer tank WO upper roof				Outer tank Full	Others	Early	Late	Rb 1st	Rb All	Rb + Vertical	All PS Early	All PS Late	Roof Frame 1	Roof Frame 2	Roof Frame 3	1st layer concrete	2nd layer concrete	Hydrostatic	Pneumatic	LNG Max	LNG Min	Gas Max		
4	no.	Code	Details	Outer tank WO roof	Outer tank WO upper roof	Outer tank Full	Others	Early	Late	Rb 1st	Rb All	Rb + Vertical	All PS Early	All PS Late	Roof Frame 1	Roof Frame 2	Roof Frame 3	1st layer concrete	2nd layer concrete	Hydrostatic	Pneumatic	LNG Max	LNG Min	Gas Max
5	1	U-C1-1	WO_roof + RB_1st_PS	1.35						1.30														
6	2	U-C1-2		1.35							1.00													
7	3	U-C1-3		1.00							1.30													
8	4	U-C1-4		1.00							1.00													
9	5	U-C2-1	WO_roof + RB_1st_PS + Roof_frame_1	1.35						1.30						1.50								
10	6	U-C2-2		1.35							1.00						1.50							
11	7	U-C2-3		1.00							1.30						1.50							
12	8	U-C2-4		1.00							1.00						1.50							

Looking at combination ‘U-C1-1’ from this spreadsheet, it might be necessary to extract the pure prestress effect from the staged construction analysis, due to the different load factors for self-weight and the prestress loading respectively.

The Ringbeam 1st Prestress is introduced at loadcase 6 (i.e Ringbeam 1st PS (staged) hence the pure effect of the that loadcase can be obtained by defining a load combination for loadcase 6 minus loadcase 5. This would be done by selecting the **Analyses > Basic Combination** menu item and on its dialog include **Increment 6** with a load factor of 1, and **Increment 5** with a load factor of -1 as shown, with a name of ‘**Stage 6 minus Stage 5 (Pure 1st PS)**’

2D Axisymmetric Staged Construction Tank Model



The resulting load combination of ‘Stage 6 minus Stage 5 (Pure 1st PS)’ can be used for defining the design load combinations U-C1-1 and U-C1-2 of the sample design load combination table.

Design combinations can be defined by selecting the menu item **Tank > Excel Tools > Design Combinations for Force/Moments**.

This facility requires the prior creation of input spreadsheets with the loadcase results (obtained from Export Forces/Design results) and combining them in Excel based on the factors read from the Tank Template Combination template.

2D Axisymmetric Coupled Thermal / Structural Tank Model

A 2D coupled thermal/structural analysis tank model is used to investigate temperature effects. Thermal data from this 2D analysis can be exported for use by a 3D shell model because the evaluation of heat transfer through thickness cannot be performed with shell elements.

Creating the model

To create a 2D coupled thermal/structural analysis tank model from previously defined tank definition data:

Tank
Create 2D Model
Coupled Thermal/Structural

2D Axisymmetric Coupled Thermal / Structural Tank Model

Tank - Coupled Thermal/Structural Analysis

Tank definition data: concrete tank

Model filename: tank

Saved model file path: C:\Users\paul_b\Documents\Lusas220\Projects\tank(2D Thermal).mdl

Modelling options

Concrete element size [m]: 0.2 Steel element size [m]: 4.0E-3 ☒ Include soil (above ground tanks)

☒ Include Structural Load

Variable Loads to apply (*) ☒ Max ☐ Min

- The chosen variable loads from the Tank Definition will be used for Operating Condition.

(*) These parameters are read from the [Structural Loading Definition] tab of the tank definition utility.

Spillage Loading

Thermal loading application: ☒ 1st wall insulation layer ☐ Outer tank wall

Liquid density [kg/m³]: 480.0

Solar radiation [kW/m²]: 0

Spillage duration time for each spillage height

Spillage 1 [days]: 1 Spillage 2 [days]: 1 Spillage 3 [days]: 1

Spillage 4 [days]: 1 Spillage 5 [days]: 1

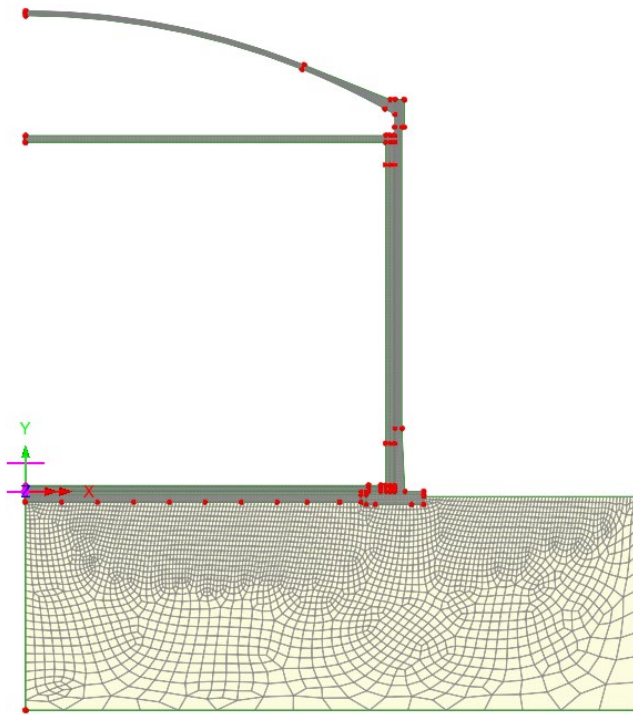
OK Cancel Help



Tip. Press the **Help** button for more information about this dialog and the type of model that is created.

- Accepting the default values on the dialog, enter a model filename of **tank** and click **OK**. The filename entered will be appended with '(2DThermal)' to identify this model within the set of tank models that are being created.
- Click **OK** to save changes to the previously defined model.

After a short while the following model will be created.



Note. Browsing the Groups , Attributes  and Analyses  treeviews will show the groups of features added, the attributes created and assigned to the model, and the analyses defined.



Note. Context menus for the items present can be used to locate features assigned to groups and to visualise attribute assignments.

Viewing assigned loadings



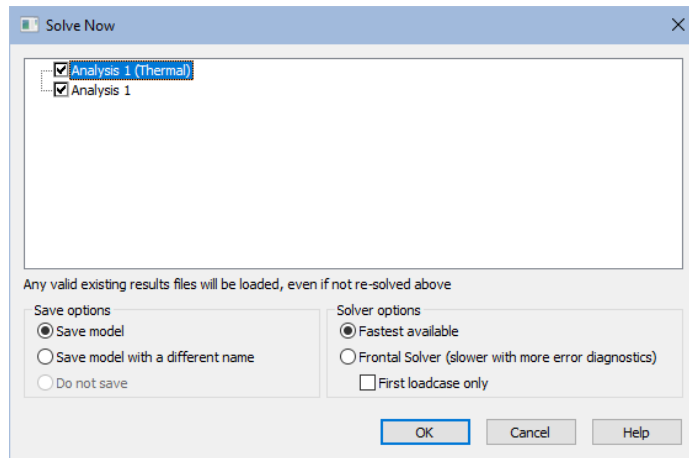
By ensuring the loading arrows are being drawn, and setting active each loadcase in turn, the loadings assigned to the model can be viewed and checked.

Running the analysis

With the model loaded:



Select the **Solve Now** button from the toolbar and click **OK** to run the analyses listed.



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...

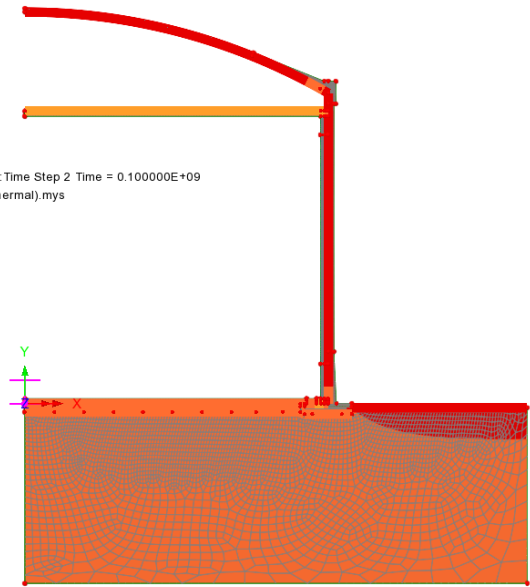
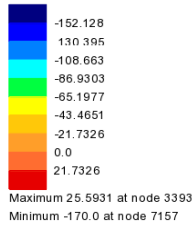
Analysis loadcase results are added to the  Treeview.

Viewing the results

Thermal and structural analysis loadcase results are present in the  Treeview.

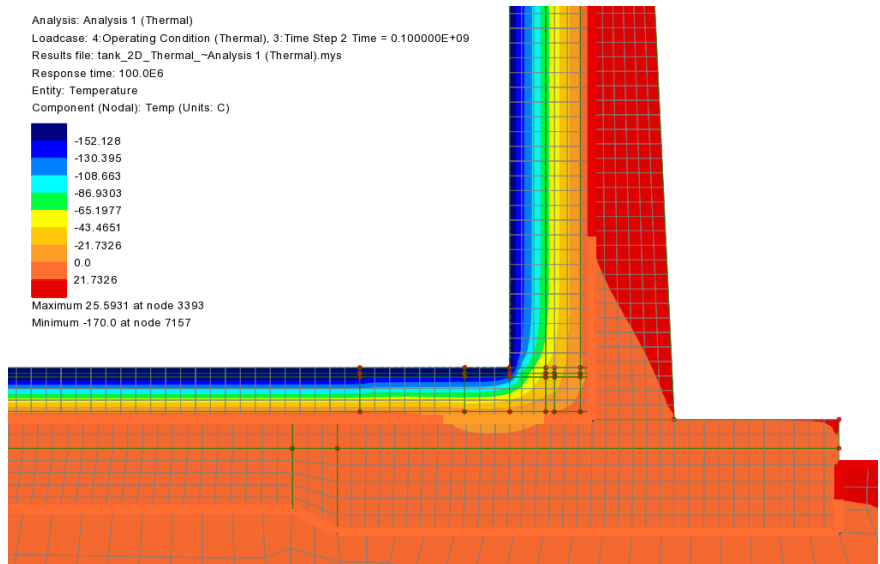
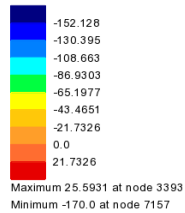
- In the  Treeview, set active the loadcase **Operating Condition (Thermal)**
- With no features selected, click the right-hand mouse button in a blank part of the view window and select the **Contours** option to add the contours layer to the  Treeview.
- Select entity **Temperature** and component **Temp** and click **OK**.

Analysis: Analysis 1 (Thermal)
 Loadcase: 4: Operating Condition (Thermal), 3: Time Step 2 Time = 0.100000E+09
 Results file: tank_2D_Thermal_~Analysis 1 (Thermal).mys
 Response time: 100.0E6
 Entity: Temperature
 Component (Nodal): Temp (Units: C)

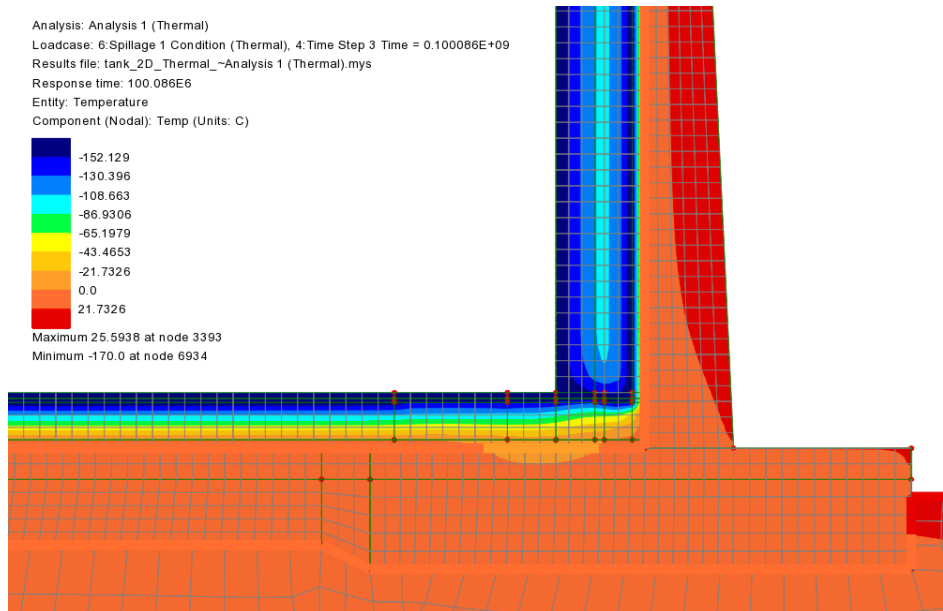


- Enlarging the view will show more detail.

Analysis: Analysis 1 (Thermal)
 Loadcase: 4: Operating Condition (Thermal), 3: Time Step 2 Time = 0.100000E+09
 Results file: tank_2D_Thermal_~Analysis 1 (Thermal).mys
 Response time: 100.0E6
 Entity: Temperature
 Component (Nodal): Temp (Units: C)



- In the  Treeview, set active the loadcase **Spillage 1 Condition (Thermal)**



Exporting 2D thermal results

Thermal analysis results for a loadcase in a 2D thermal/structural model can be exported to a spreadsheet to enable thermal loading to be added as equivalent structural loading to a 3D shell model.

- With the loadcase **Operating Condition (Thermal)** active:
- Enter an output filename of **2d_operating_condition** and press **OK**.

Tank
Excel Tools
Export
Temperature

Tank - Export Temperatures to Excel (2D)

Output filename:

Working folder: ☐ Current ☒ User Defined

Save in: ...

Interval: m

The spreadsheet created in the specified folder comprises worksheets containing temperature information for named regions of the tank, e.g. Roof, Wall_RingBeam and BaseSlab.

Temperature of Roof						
Type	Temperature					
Location	Roof					
Unit	Celsius					
No of Slices	Gap (m)	Outer Radius (m)	Rad. Origin Y (m)	Min. Thickness (m)		
88.00	0.50	86.91	-30.65	0.50		
Distance (m)	Inner Temp (°C)	Outer Temp (°C)	Thickness (m)	Average Temp (°C)	Linear Gradient (°C/m)	Vertical Distance (m)
0.00	23.71	25.33	0.50	24.49	-3.57	0.00
0.50	23.61	25.33	0.50	24.47	-3.71	0.00
1.00	23.62	25.33	0.50	24.47	-3.69	0.01
1.50	23.61	25.33	0.50	24.47	-3.71	0.01
2.00	23.62	25.33	0.50	24.47	-3.70	0.02
2.50	23.62	25.33	0.50	24.47	-3.70	0.04
3.00	23.62	25.33	0.50	24.47	-3.70	0.05
3.50	23.62	25.33	0.50	24.47	-3.70	0.07
4.00	23.62	25.33	0.50	24.47	-3.70	0.09
4.50	23.61	25.33	0.50	24.47	-3.71	0.12
5.00	23.62	25.33	0.50	24.47	-3.70	0.14
5.50	23.60	25.33	0.50	24.47	-3.72	0.17
6.00	23.62	25.33	0.50	24.47	-3.69	0.21
6.51	23.60	25.33	0.50	24.47	-3.73	0.24
7.01	23.61	25.33	0.50	24.47	-3.71	0.28
7.51	23.60	25.33	0.50	24.47	-3.72	0.33

This completes this example.

Additional notes

- ❑ Thermal analysis results from solving a 2D thermal / structural model can be applied to a 3D shell model as equivalent structural loading using the **Add Loading > Thermal** menu item. To do so, up to four maximum and minimum thermal loading conditions need to be defined and solved in separate 2D thermal / structural models, and the loadcase 'Operating Condition Thermal' set active in each model prior to exporting the temperatures to unique Excel spreadsheet names.
- ❑ Resultant forces from integration of stresses through thickness can also be obtained at regular intervals and for selected loadcases.

2D Beam-stick Seismic Tank Model

If seismicity is to be evaluated, a 2D beam-stick model can be created to provide seismic results that can also be applied to a 3D shell model. Horizontal actions excluding and including base pressure, and vertical actions may be modelled.

Create model

To create a 2D beam-stick model from previously defined tank definition data:

Tank
Create 2D Model
Seismic...

2D Beam-stick Seismic Tank Model

Tank - Seismic Analysis

Tank definition data: concrete tank

Model filename: tank

Saved model file path: C:\Users\paul_b\Documents\LUSAS211\Projects\tank(Seismic EN1998 Horizontal-EBP).mdl

Analysis type

Design code: EN 1998-4

☒ Beam-stick horizontal (Excluding base pressure) ☐ Beam-stick horizontal (Including base pressure) ☐ Beam-stick vertical

(Beam-Stick model is created according to EN1998-4:2006 A3.2.2 Simplified procedure for fixed base cylindrical tanks)

Critical damping / frequency

	Critical damping (%)	Frequency (1st mode, Hz)	Frequency (2nd mode, Hz)
Base slab	4.0	1.25	5.44
Roof	4.0		
Wall	2.0		
Inner tank	2.0		
Foundation	4.0		
Liquid impulsive	3.0		
Liquid convective	0.5		
Ground	5.0		

Buttress

Number of buttresses: 4

Extruded thickness: 1.0 [m] Buttress width: 5.0 [m]

OK Cancel Help



Tip. Press the **Help** button for more information about this dialog and the type of model that is created.



Note. Damping coefficients are computed based on the user inputs for a desired damping ratio (%) and the frequency range of the structure that would be obtained from a separate eigenvalue analysis. For structural members and impulsive liquid mass, Rayleigh Damping Coefficients are computed and used in the material definition. For Soil springs and convective mass, a Viscous Coefficient is used for horizontal movement considering the moving mass above the ground.

- Accepting the remaining default values on the dialog (to model horizontal actions excluding base pressure for the EN 1998-4 design code), enter a model filename of **tank** and click **OK**. The filename entered will be appended with '(Seismic

EN1998 Horizontal-EBP)’ to identify this model within the set of tank models that are being created.



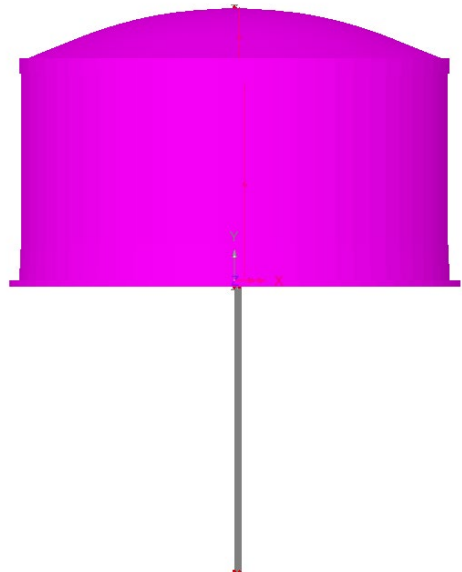
Note. Valid buttress options are 0, 2, 3, 4 and 6. As the inclusion of buttresses makes the model non-axisymmetric, values other than zero are considered in the model by increasing the thickness of wall and ringbeam to an appropriate equivalent thickness.



Press the Fleshing on/off button to see the region of the model that represents the tank.



Fleshing ‘Off’



Fleshing ‘On’

The adopted arrangement of components allows for capturing the complex seismic behaviour of the liquid tank system in a simplified but accurate model.

If necessary, refer to the online help for a 2D seismic analysis tank model for more details of the model that is built.



Note. Browsing the Groups , Attributes  and Analyses  treeviews will show the groups of features added, the attributes created and assigned to the model, and the analyses defined.



Note. Context menus for the items present can be used to locate features assigned to groups and to visualise attribute assignments.

Running the analysis

With the model loaded:

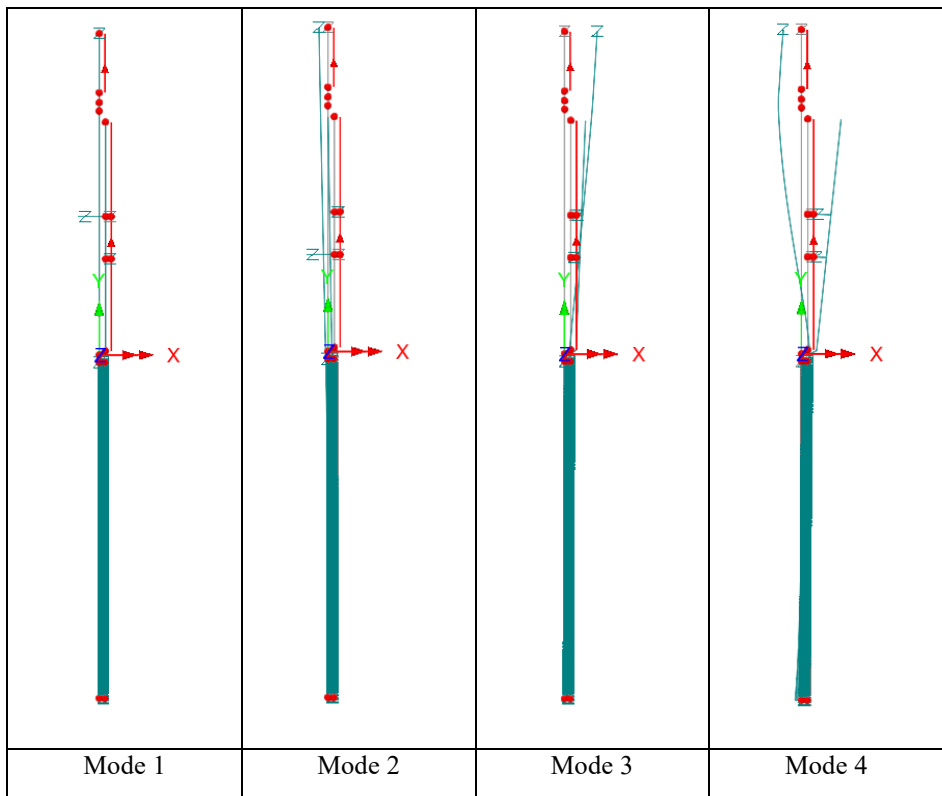


Select the **Solve Now** button from the toolbar and click **OK** to run the analyses listed.

Analysis loadcase results are added to the  Treeview and a deformed mesh layer is added to the view window.

Viewing the results

- Mode shapes may be viewed by setting active each loadcase in turn within the Analyses  Treeview of the generated model. The first four modes shapes are shown.



Tabulating eigenvalue results

Utilities

Print results wizard

Print Results Wizard

Results type

☐ Components ☒ Eigenvalues

Results

Units: Model units

Loadcases: 1: Loadcase 1

☒ Eigenvalues (Frequency)
☒ Participation factors
☒ Mass participation factors
☒ Sum mass participation factors

- Select result type **Eigenvalues**
- Select all 4 results check box options and click **OK**.

LUSAS View: tank[Seismic EN1998 Horizontal-EBP.mdl Window 1] Mass participation factors

	Mode	Mass PF X	Mass PF Y	Mass PF Z	Mass PF THX	Mass PF THY	Mass PF THZ	Frequency	Period
1	1	0.19259	15.1517E-21	0.0	0.0	0.0	36.8976E-9	0.100056	9.99442
2	2	0.280807	3.44612E-15	0.0	0.0	0.0	0.0108583	2.1688	0.461085
3	3	0.0872404	1.32489E-15	0.0	0.0	0.0	0.164939	3.94813	0.253284
4	4	6.13121E-3	29.2686E-18	0.0	0.0	0.0	0.675368	10.9011	0.0917338
5	5	0.170597	1.31404E-15	0.0	0.0	0.0	0.0443415	13.7877	0.0725282
6	6	0.100097	7.19531E-15	0.0	0.0	0.0	0.0565405	20.0925	0.0497698
7	7	0.0305648	0.291223E-15	0.0	0.0	0.0	1.86515E-3	22.4363	0.0445707
8	8	0.0212402	80.2383E-15	0.0	0.0	0.0	8.90615E-3	27.6096	0.0362193
9	9	0.108737	0.395673E-15	0.0	0.0	0.0	0.0160443	29.7688	0.0335922
10	10	0.504893E-3	7.35611E-15	0.0	0.0	0.0	32.2804E-6	36.4392	0.027443
11	11	0.425841E-3	44.4875E-15	0.0	0.0	0.0	2.6161E-3	41.4436	0.0241292
12	12	89.9823E-6	35.7218E-15	0.0	0.0	0.0	28.5691E-6	45.1776	0.0221349
13	13	0.671022E-3	0.318606E-12	0.0	0.0	0.0	4.54182E-3	48.9366	0.0204346
14	14	0.194278E-3	0.350701E-12	0.0	0.0	0.0	0.961124E-3	55.8099	0.017918
15	15	12.6119E-6	0.29471E-12	0.0	0.0	0.0	0.103411E-3	59.4636	0.016817
16	16	74.1501E-6	1.82418E-15	0.0	0.0	0.0	0.103791E-3	61.1417	0.0163555
17	17	0.130055E-6	0.88289E-12	0.0	0.0	0.0	12.1222E-6	71.0539	0.0140738
18	18	33.198E-9	0.508516E-12	0.0	0.0	0.0	0.114254E-3	74.6845	0.0133897
19	19	0.231096E-6	0.245715E-12	0.0	0.0	0.0	67.1555E-6	82.8546	0.0120693
20	20	2.71135E-9	96.511E-15	0.0	0.0	0.0	15.0234E-6	84.4793	0.0118372

Model info | 1: Loadcase 1 (Eigenvalues) | 1: Loadcase 1 (Participation factors) | 1: Loadcase 1 (Mass participation factors) | 1: Loadcase 1 (Sum mass participation factors)

- On the spreadsheet displayed select the **Mass participation factors** tab.



Note. Looking at the mode shape and the mass participation factor, the 1st mode is for convective liquid mass, and the subsequent modes are mixed modes.

2D Beam-stick Seismic Tank Model

From this it would be reasonable to use the 2nd and 3rd frequencies obtained as the frequency range for computing damping constants for the 1st and 2nd mode frequencies, previously provided in the Tank - Seismic Analysis model creation dialog. This example does not do this however.

Tank - Seismic Analysis

Tank definition data: Trk1

Model filename:

Saved model file path: C:\Users\paul_b\Documents\LUSAS211\Projects\Seismic EN1998 Horizontal-EBP.mdl

Analysis type: EN 1998-4

☒ Beam-stick horizontal (Excluding base pressure) ☐ Beam-stick horizontal (Including base pressure) ☐ Beam-stick vertical

(Beam-Stick model is created according to EN1998-4:2006 A3 2.2 Simplified procedure for fixed base cylindrical tanks)

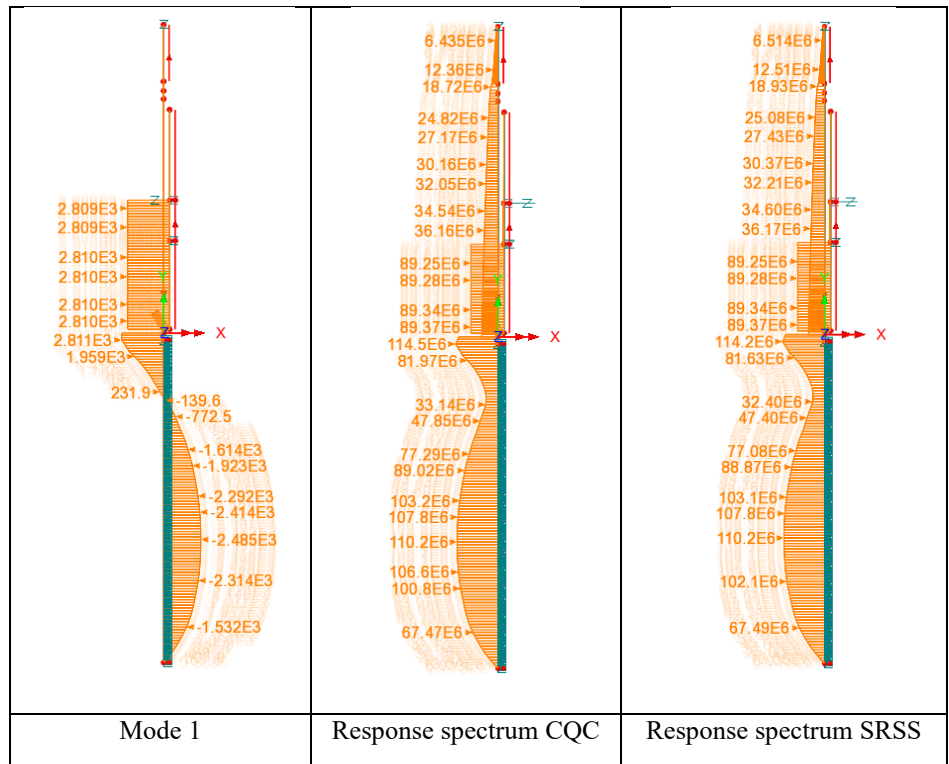
Critical damping / frequency

	Critical damping (%)	Frequency (1st mode, Hz)	Frequency (2nd mode, Hz)
Base slab	4.0	1.25	5.44
Roof	4.0		

Diagrams

- In the Treeview, set active the loadcase **Mode 1**
- With no features selected, click the right-hand mouse button in a blank part of the view window and select the **Diagrams** option to add the diagrams layer to the Treeview.
- Select entity **Force/Moment – Thick 2D Beam** and component **Fy** and click **OK**. The results plot for this selection is shown on the following set of images.
- In the Treeview, set active the loadcase **Response spectrum CQC**
- In the Treeview, set active the loadcase **Response spectrum SRSS**

The diagrams below show the Fy results for each of these loadcase selections.



Show the damping applied to each mode

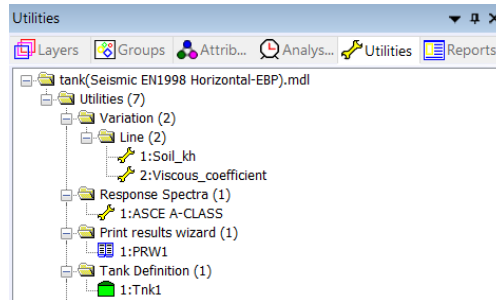
Because the option to **Include modal damping** is checked in the **Eigenvalue** control dialog, the modal damping factors computed for each mode are printed in the Solver output file, which has a file extension of '*.out'. This can be found at this location:

<Current working folder> \LUSASFiles\Example_EN1998_HorizontalBeamStick(IBP)

MODAL DAMPING FACTORS				
MODE	EIGENVALUE	FREQUENCY	VISCOUS DAMPING	
1	0.395226	0.100056	0.499826E-02	
2	185.694	2.16880	0.237803E-01	
3	615.380	3.94813	0.239494E-01	
4	4691.38	10.9011	0.434129E-01	
5	7504.93	13.7877	0.560310E-01	
6	15937.8	20.0925	0.879946E-01	
7	19872.9	22.4363	0.878300E-01	
8	30093.9	27.6096	0.111758	
9	34985.1	29.7688	0.106509	
10	52420.0	36.4392	0.159976	

Design Response Spectrum

By default, the Tank system uses the response spectrum based on ASCE7-10 (2010). This entry is present in the Utilities  Treeview.



Note. Other design codes may be selected from the dialog that is displayed when double-clicking this entry.

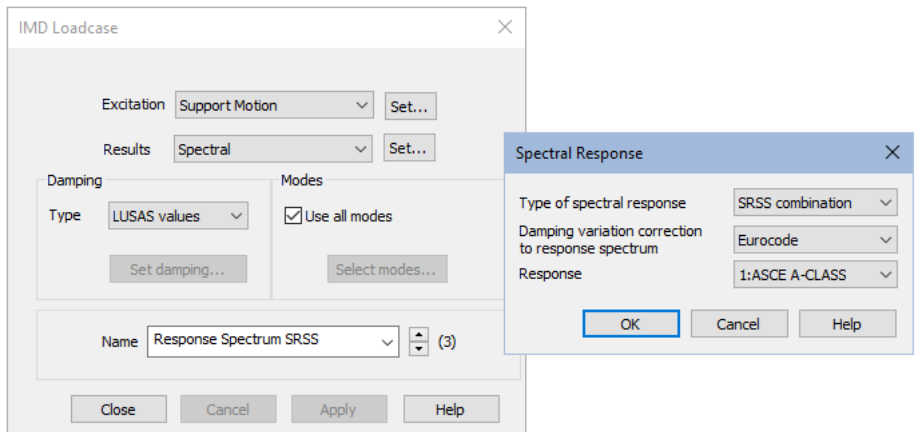


Note. User-defined spectrums and other design code spectrums can be defined by selecting the menu item **Utilities > Response Spectrum...**

Changing a design response spectrum

The response spectrum to be used for post-processing can be changed by editing an existing **IMD loadcase** attribute in the  Treeview.

- In the  Treeview, double click (for example) the **Response Spectrum SRSS** entry.
- On the IMD loadcase dialog press the **Set** button next to the Results droplist and use the Response droplist to choose a previously defined or new response spectrum.



This completes this example.

Additional notes

- ❑ Seismic analysis results from solving a 2D seismic beam-stick model can be used to provide seismic loading for a 3D shell model. This will require manual conversion of peak results into equivalent accelerations and forces to apply to the 3D shell model using the **Add Loading > Seismic** facility.

3D Shell Structural Tank Model

Creating the model

To create a 3D shell half-symmetric structural model from previously defined tank definition data:

Tank
Create 3D Shell Model

3D Shell Structural Tank Model

Tank - 3D Shell Model

Tank definition data: Tank1

Model filename: tank

Saved model file path: C:\Users\paul_b\Documents\Lusas220\Projects\IMDPlus CRH Analysis\tank(3D Shell)

Modelling options

Element size [m]: 2.0

Number of eigenvalues: 10

☒ Half symmetric model ☐ Include temporary opening

☐ Rigid fin plate ☒ Include non-structural masses in the eigenvalue analysis

Concrete Tank Options

Buttress

Number of buttresses: 4

Extruded thickness: 1.0 [m]

Buttress width: 5.0 [m]

Roof / Ringbeam

Roof construction plan: Single layered roof 1

Roof first stage thickness (ratio): 0.5

Initial prestress for ringbeam (ratio): 0.5

Initial prestress for base slab (ratio): 0.5

Prestress

Vertical prestress type: Tendon Loads

Construction Sequence - Single layered roof 1

- 1 - Base / Wall / Ringbeam
- 2 - Ringbeam 1st PS
- 3 - Roof frame 1/ Inner work
- 4 - Roof frames 2,3
- 5 - Roof wet / Roof complete
- 6 - Ringbeam 2nd PS
- 7 - Wall vertical PS
- 8 - Wall horizontal PS

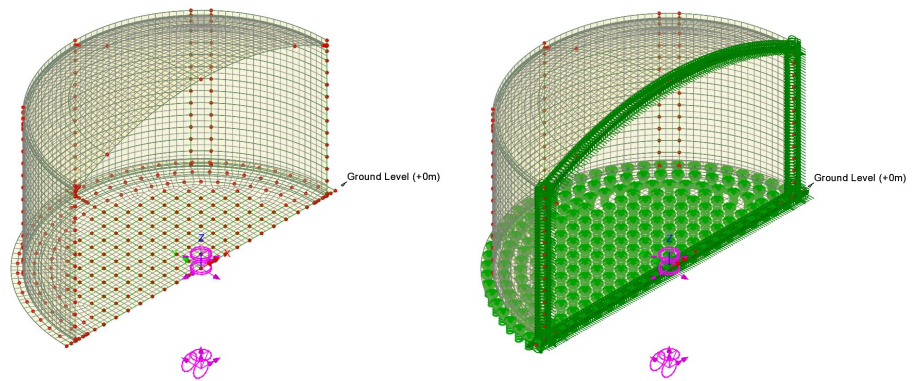
OK Cancel Help



Tip. Press the **Help** button for more information about this dialog and the type of model that is created.

- Accept all the default values and settings, noting that no temporary openings are defined by default in the tank definition data (in the Wall and Ring beam Tank - Openings dialog), so none will be modelled in this example) and enter a model filename of **tank** and click **OK**. The filename entered will be appended with '(3D Shell)' to identify this model within the set of tank models that are being created.
- Click **OK** to save any changes to the previously defined model.
- Click **Yes** to accept that the wall will be constructed all at once and not in stages.
- Click **Yes** to proceed with adding construction stage loads, then click **OK** to accept the default values presented.

From the settings made, the following model will be created.



Press the Isometric button to obtain this view of the model.



Press the Support on/off button to see the assigned supports.



Note. By default, a 3D shell tank is supported on spring supports. Prior to creating a model, and from within the Tank Definition 3D Support tab, a detailed foundation may be chosen which, depending on the tank elevation setting in use, allows for isolator, pedestal, raft and pile type settings and dimensions to be specified. A different foundation may also be added to a model after it has been built. Help topics available provide more details.



Note. Tendon, thermal, seismic, staged construction, creep and shrinkage, spillage, and wind loading (from 2D tank model analysis and from other sources) may all be added to a 3D shell model prior to any design checks being made. However, for this short explanatory example, these will not be considered.



Note. Browsing the Groups , Attributes  and Analyses  treeviews will show the groups of features added, the attributes created and assigned to the model, and the analyses defined.



Note. Context menus for the items present can be used to locate features assigned to groups and to visualise attribute assignments.

Viewing assigned loadings



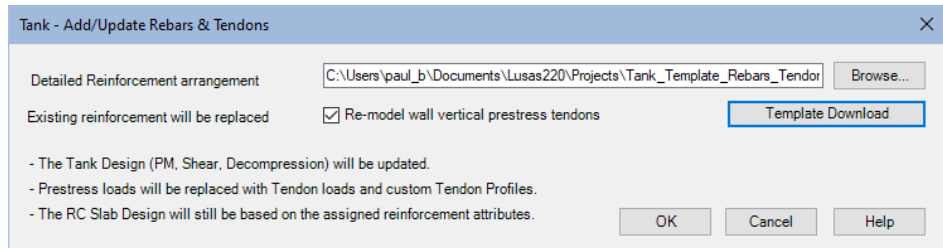
By ensuring the loading arrows are being drawn, and setting active each loadcase in turn, the loadings assigned to the model can be viewed and checked.

Defining reinforcement and tendons

For 3D shell models of concrete tanks, prestress loading is applied to the model initially by using dummy tendons (located at mid-thickness throughout the height of the tank

wall) and at a spacing of approximately 1m centre-to-centre. The exact tendon profile, location and spacing arrangement required can defined and imported from a spreadsheet.

Tank
Design Checks
Add/Update
Rebars and
Tendons



- On the dialog, press the **Template Download** button. A supplied template containing suitable rebar and tendon data will be saved to the working folder.
- Click **OK** to add rebar and replace all existing prestress loading with custom tendon profiles.a design cj

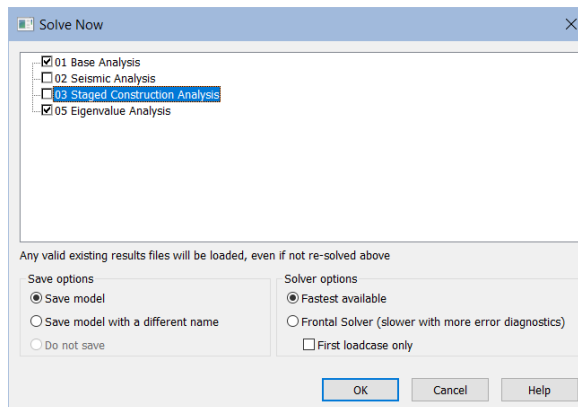
Running the analysis

With the model loaded:



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

For simplicity only the Base Analysis and the Eigenvalue Analysis will be run for this example.



- On the Solve Now dialog deselect the **Seismic Analysis** and **Staged Construction Analysis** check boxes, then click **OK**.

Analysis loadcase results are added to the  Treeview and a deformed mesh layer is added to the view window.

Viewing the results

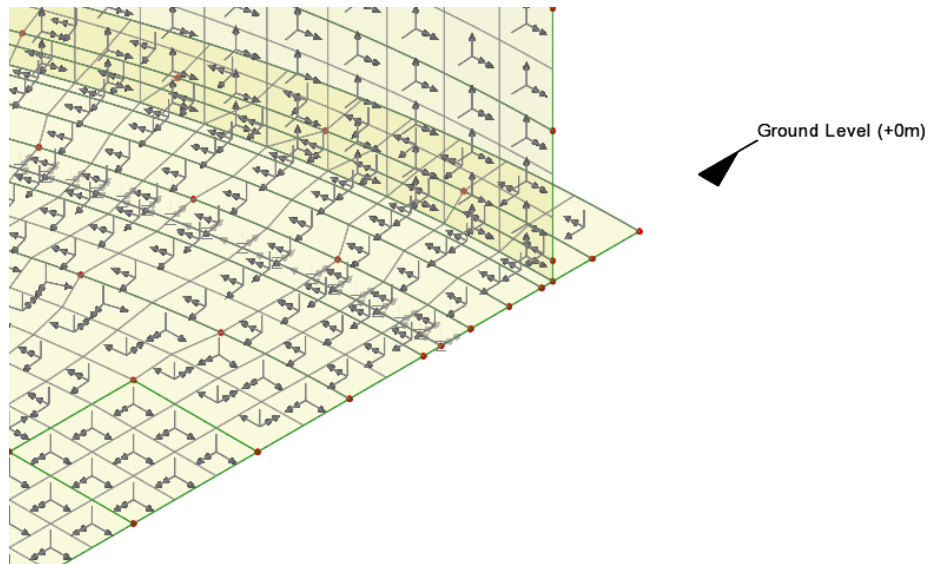
- Turn off the display of the **Deformed mesh** layer.
- In the  Treeview, set active the loadcase **SelfWeight**

Understanding element axes

In a 3D shell model the element local axes are not consistent in the structure as a whole. To see the element axes:

- In the  Treeview, double-click the **Mesh** entry and select **Show element axes**.
- Zooming in on the wall and base slab connection will show the element axes in those regions more clearly.

The x-axes of the elements in the group ‘Tank Wall’ (as identified by double headed arrows) are seen to be aligned horizontally within the wall. The elements in the group ‘Tank Base Annular Part’ similarly are aligned consistently, but the elements in the ‘Tank BaseCircularPart’ are inconsistently aligned – as shown in this enlarged view of the model



Note. The named groups in the Groups  Treeview, can be **Set As Only Visible** in turn to see element axes for each set of features in isolation.

To plot results for a consistent direction, local coordinate sets and results transformation datasets can be used.

- In the  Treeview, double-click the **Mesh** entry and turn off the selection of **Show element axes**.

Local coordinate sets

In the model created, cylindrical and spherical local coordinates are present in the Attributes  Treeview. A cylindrical local coordinate system is used to obtain forces in the BaseSlab and Wall, and a Spherical local coordinate system is used to obtain forces in the Roof.

Results transformation datasets

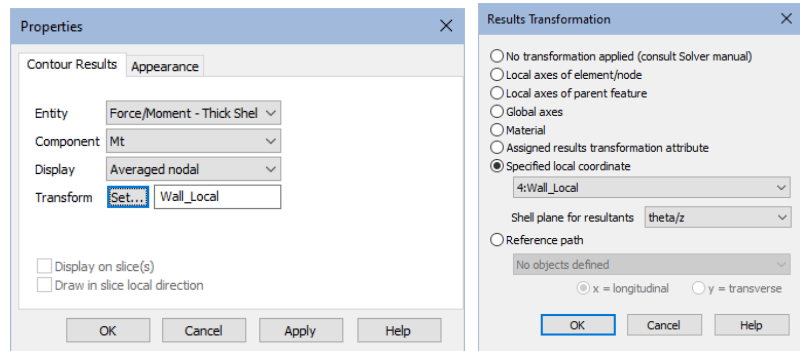
In the model created, results transformation datasets are also present and assigned to the roof, wall, and base slab respectively. These are:

Results transformation dataset	Shell plane for	Local coordinate set
ResultTransformation_Roof	theta/phi	‘Spherical for force Extraction’
ResultTransformation_Wall	theta/z	‘Local Coord’
ResultTransformation_BaseSlab	r/theta	‘Local Coord’

Plotting Contours

With regard to the bending moment in the wall, as the element local x-axis is in the horizontal direction in the model, the horizontal directional moment of M_x could be displayed for a selected loadcase. However, because the element local axes are not consistent in the structure as a whole, a local coordinate system is to be used for viewing results.

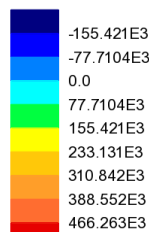
- With no features selected, click the right-hand mouse button in a blank part of the view window and select the **Contours** option to add the contours layer to the  Treeview.
- On the contour properties dialog, select entity **Force/Moment – Thick Shell**, then press the transform **Set** button and for the **Specified local coordinate** select **Wall_Local** from the droplist provided and **theta/z** from the ‘Shell plane for results’ droplist, because the wall surface element axes have a theta and z direction. Click **OK**.



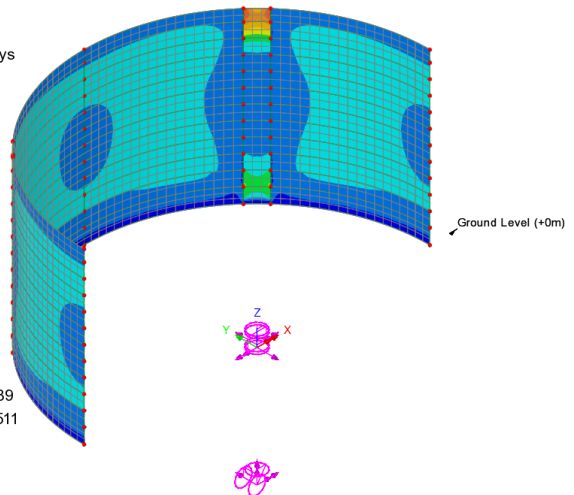
This now makes available a moment results component 'Mt', where 't' represents the tangent direction in the cylindrical local coordinate system.

- Now, back on the contour properties dialog, select the result component **Mt** and click **OK**. The contours will be drawn on the whole model.
- To see only the forces/moments on the wall, in the Groups  Treeview, right-click on the **Tank Wall** entry and select **Set as Only Visible**.

Analysis: 01 Base Analysis
 Loadcase: 1:SelfWeight
 Results file: tank_3D_Shell_~01 Base Analysis.mys
 Entity: Force/Moment - Thick Shell
 Transformation: Local Coords "Wall_Local"
 Component (Averaged nodal): Mt (Units: N.m/m)



Maximum 488.58E3 at node 16281 of element 5289
 Minimum -210.814E3 at node 10632 of element 3511

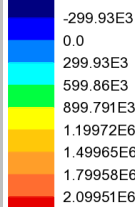


Note. If, on the Results Transformation dialog shown previously the option **Assigned results transformation attribute** option was chosen, results components of **Nx** and **Ny** could then be selected instead of **Mt**.

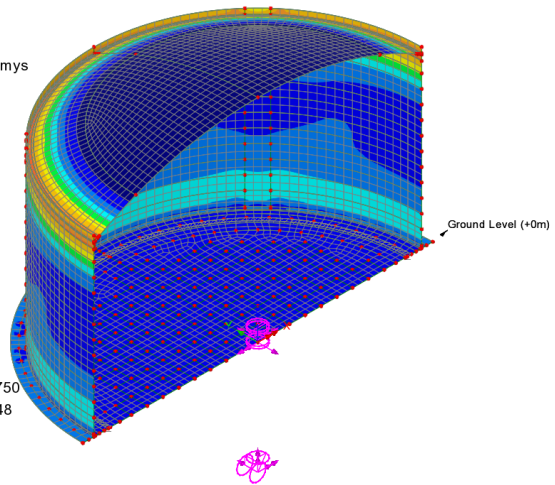
When this is done, any components with 'x' represent the results in the hoop direction (wall/roof) or radial direction (base slab), and those with 'y' represent results in the radial (roof) or vertical (wall) direction or the hoop (base slab) direction. If plotted on the whole model the following contour plots would be obtained.

3D Shell Structural Tank Model

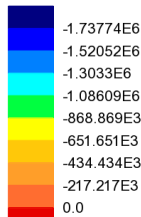
Analysis: 01 Base Analysis
Loadcase: 1:SelfWeight
Results file: tank_3D_Shell_~01 Base Analysis.mys
Entity: Force/Moment - Thick Shell
Transformation: Assigned attribute
Component (Averaged nodal): Nx (Units: N/m)



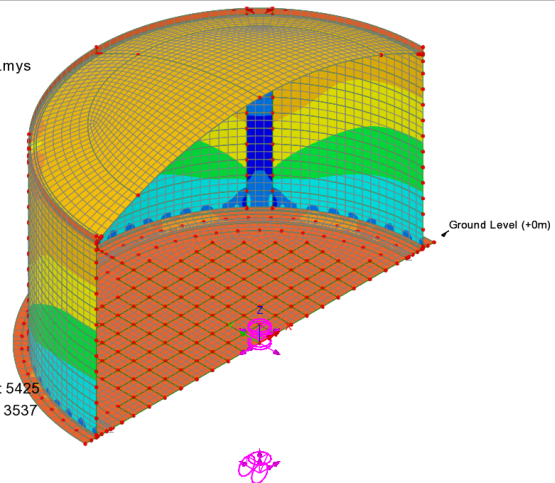
Maximum 2.15043E6 at node 2229 of element 750
Minimum -548.943E3 at node 304 of element 148



Analysis: 01 Base Analysis
Loadcase: 1:SelfWeight
Results file: tank_3D_Shell_~01 Base Analysis.mys
Entity: Force/Moment - Thick Shell
Transformation: Assigned attribute
Component (Averaged nodal): Ny (Units: N/m)



Maximum 56.7779E3 at node 16768 of element 5425
Minimum -1.89818E6 at node 10632 of element 3537



Values

- With the Tank1 Wall still set as only visible, values can be displayed for chosen nodes by adding the **Values** layer to the Layers  treeview. Zooming in will provide more detail.

Analysis: 01 Base Analysis

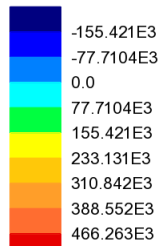
Loadcase: 1:SelfWeight

Results file: tank_3D_Shell_~01 Base Analysis.mys

Entity: Force/Moment - Thick Shell

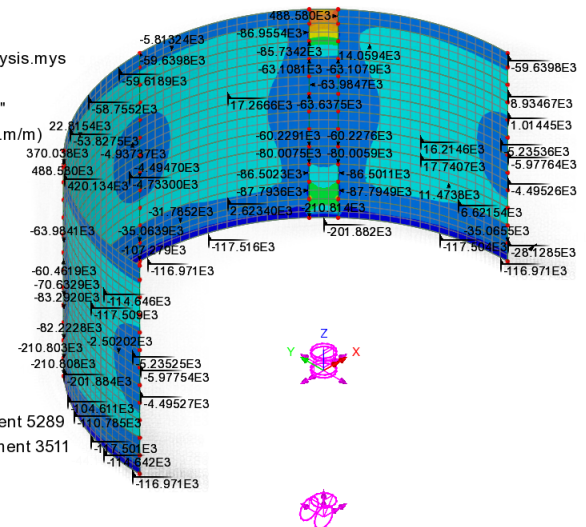
Transformation: Local Coords "Wind_Local"

Component (Averaged nodal): Mt (Units: N.m/m)



Maximum 488.58E3 at node 16281 of element 5289

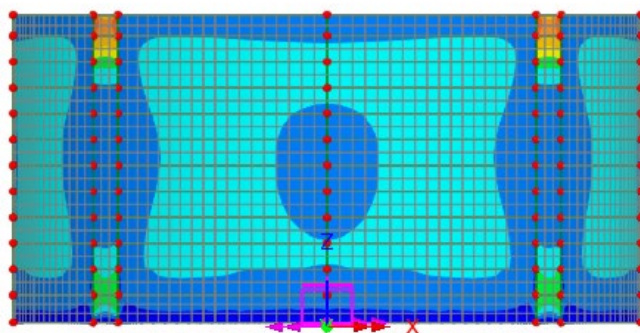
Minimum -210.814E3 at node 10632 of element 3511



Graphing of results through the wall

Arbitrary line sections may be taken through any surface of a two dimensional model or on a slice cut through a three dimensional solid model. The process of cutting a slice will generate two graph datasets in the Utilities Treeview; the first containing the distance along the line section and the second containing the specified results along the line.

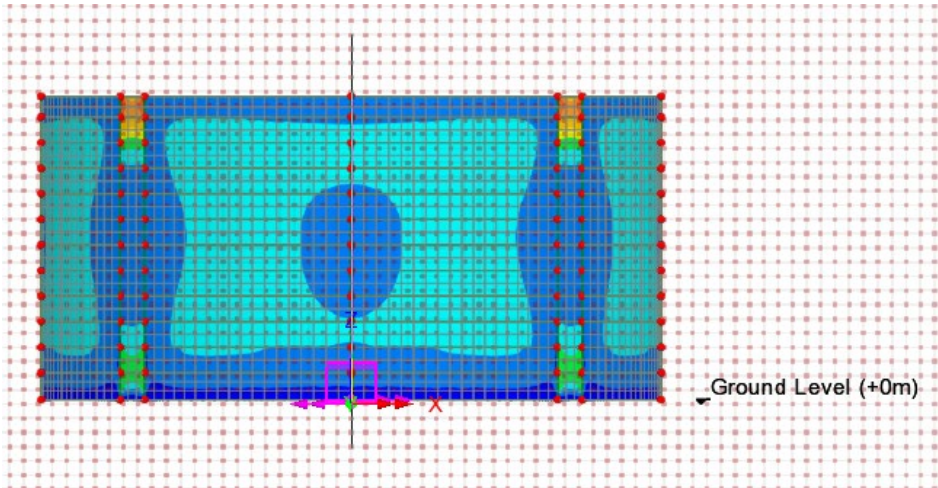
- Turn off the display of the Values layer
- Press the Y-axis button to view the model as shown in the following image.



Ground Level (+0m)

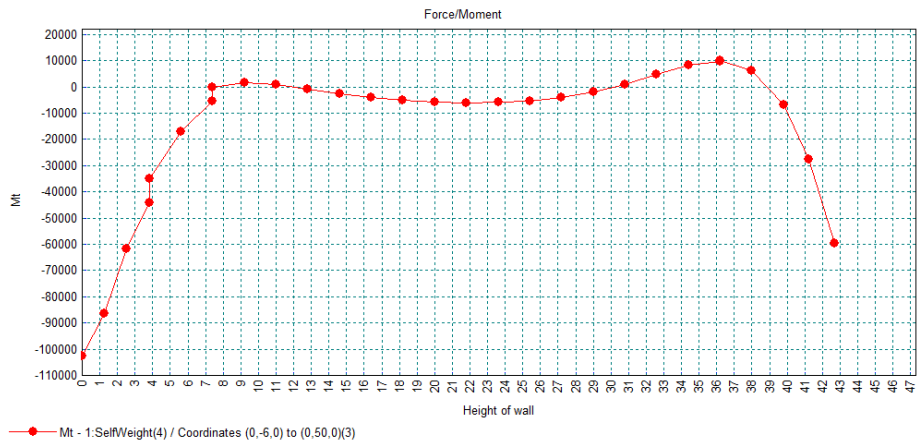
3D Shell Structural Tank Model

- On the dialog, ensure that **Snap to grid** is selected and enter a grid size of **2** and click **OK**.
- In the view window click and drag the cursor from below the bottom of the wall moving upwards to above the top of the wall as shown:



- Then, on the 'Loadcases and Extent' dialog, select **SelfWeight** and click **Next**, select entity **Force/Moment – Thick Shell** and results component **Mt** and click **Next**.
- Enter a title **Force/Moment**, an x-axis name of **Height of wall**, a y-axis name of **Mt** and click **Finish**.

A graph showing the variation of M_t with wall height is generated. As the units of the model are N,m, the unit for moment force is N-m.



In this way, other 'arbitrary slices' could be taken through the wall manually to investigate other regions.



Re-select the normal cursor.

Export forces/design results to Excel (3D)

As an alternative to producing graphs manually and individually from slices through a model, forces and design results can be calculated for any slice through the model and exported to a spreadsheet, where graphs of data are also produced.

Tank

Excel Tools

Export
Forces/Design
Results

3D Shell Structural Tank Model

Tank - Export Forces/Moments to Excel (3D)

Output filename:

Working folder: ☒ Current ☐ User Defined

Save in:

Target

☐ Base slab ☒ Wall + Ringbeam ☐ Roof ☐ All

Results to extract

☒ Forces and Moments
☐ Design results

Range

Angles: degree (eg. 10; 20; 30)
Interval: [m]

☐ Smooth slab forces at pile heads and wall

Assumed diameter at crosswise piles: [m]
Assumed diameter at circumferential piles: [m]

Utilisations for concrete sections

No design code is enabled

ULS ☒ UtilPM ☒ UtilShear ☒ PM Capacity ☒ Shear Capacity ☒ UtilDecompression ☒ Compression Depth

Loadcases

☐ Combinations only

1. SelfWeight
2. Dead loads of steel structure
3. Dead load of liner and steel roof
4. Dead load of steel structures on the roof
5. Dead load of insulation
6. Pressure on outer tank wall due to insulation
7. Wall piping loading
8. Liquid bottom (max)
9. Liquid bottom (min)
10. Gas pressure (max)
11. Gas pressure (min)
12. Live load
13. Snow load
14. Test load (liquid)

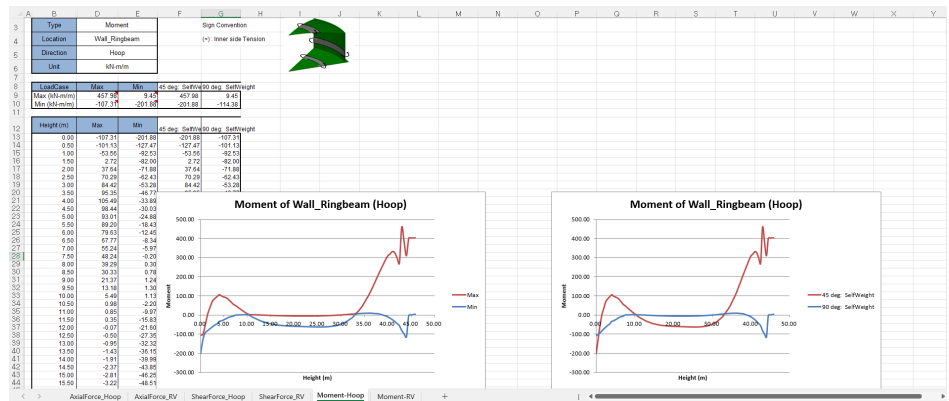
Diagram illustrating the Slicing Line and Angle (Positive Direction) for the tank model. The X axis is 0 Degree.

OK Cancel Help

- Enter an output filename of **forces_moments** (noting that the name of the target will be appended to the name entered).
- Select a target of **Wall + Ringbeam**
- For the range, enter angles of **45;90**
- For 'Results to extract' ensure that only **Forces and Moments** is selected. (Design results will be looked at later in this example)
- Ensure that only the loadcase **SelfWeight** is selected and click **OK**

A spreadsheet named **forces_moments_Wall_Ringbeam.xlsx** will be created within your Projects folder.

- Outside of LUSAS Modeller, open this spreadsheet and select the **Moment-Hoop** tab.



Note: A cylindrical local coordinate system is used to obtain forces in the BaseSlab and Wall, and a Spherical local coordinate system is used to obtain forces in the Roof.

Sign convention:

- ☐ Axial Force: (+) for Tension, (-) for Compression.
- ☐ Moment: (+) for Inner side tension, (-) for outer side tension.

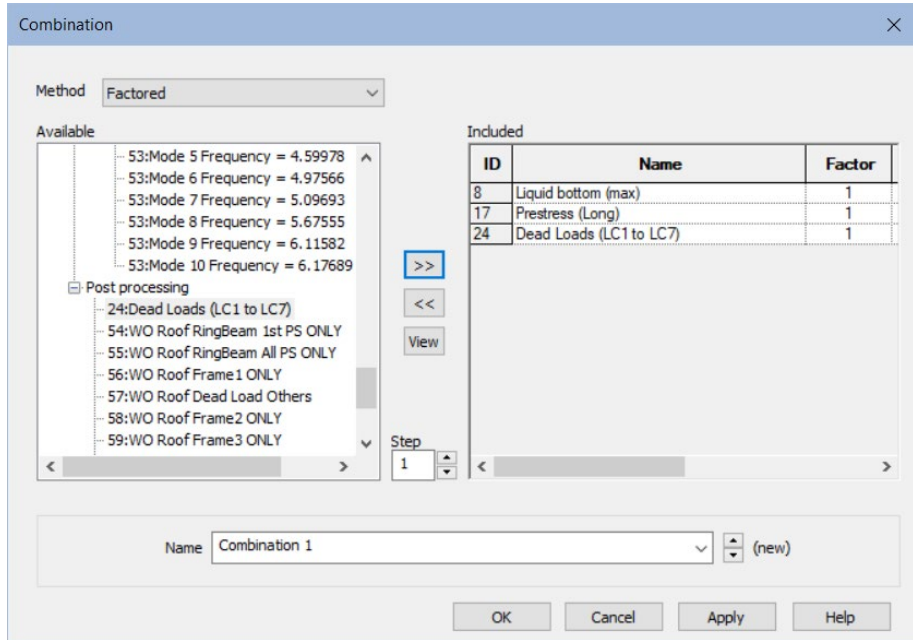


Note: If all loadcases were selected from the list box (and not just the self-weight as in this example), the forces for each loadcase for each defined angle would be exported and saved in the spreadsheet.

Defining a basic combination

Straightforward combinations may be defined within LUSAS.

Analyses
Basic
Combination...



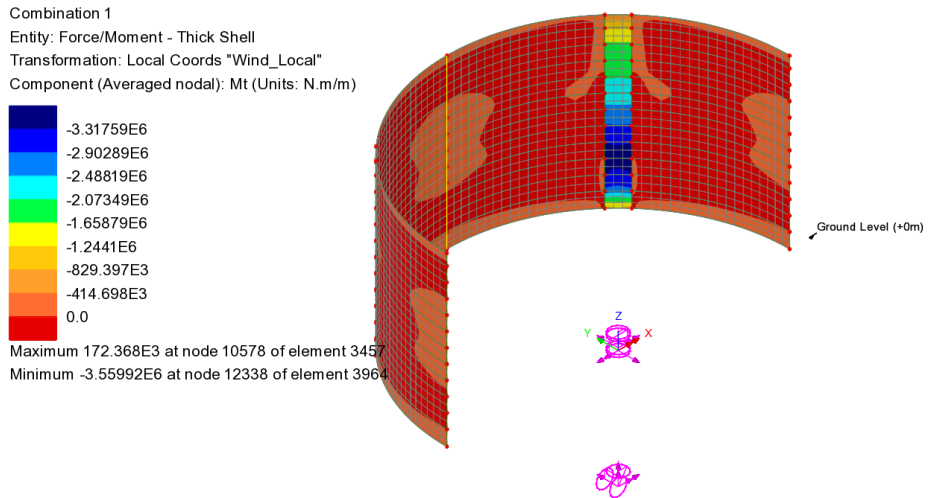
- Include **Liquid bottom (max)** and **Prestress (Long)** from the Base Analysis section and **Dead Loads(1-7)** from the Post-processing section in the combination, accept the default loadcase name of **Combination 1** and click **OK**.

Note that the loadcase 'Dead loads(1-7)' includes the loadcases for self-weight, dead loads on the structure, wall pressure and piping loading for ease of inclusion.



Press the Isometric button to return to this view of the model and reselect the normal cursor.

- In the Analyses  Treeview, set active **Combination 1**



Note. Whilst load combinations can be defined using general combination facilities (as done in this example) they can be defined more quickly for LUSAS Tank created models by specifying data within a Design Load Combinations template. This can be accessed after a design code has been defined by selecting the menu item **Tank > Design Checks > Design Load Combination**

Enabling a design code check

Tank
Design Checks
Enable...

3D Shell Structural Tank Model

Tank - Design Code

Design parameters

Design code: EN1992-1-1 (2005)

General

Partial factors for materials: Persistent/Transient

Steel

Characteristic 0.1% proof-stress of prestressing steel ($f_{p0.1k}$): 1.6E3 MPa

Elastic modulus of tendon (E_p): 195.0E3 MPa

Concrete

Roof concrete grade: 40.0 MPa

Wall concrete grade: 50.0 MPa

Slab concrete grade: 40.0 MPa

Long term effect coeff. (α_{cc}): 1.0

Computation target for visualisation

☐ Angles 0 (e.g 0,90,100) ☐ Selected ☒ Visible

☐ Shift results to avoid double counting the prestress effect (reinforcement fse and external PS)

☐ Exclude base slab results at pile heads and walls

Assumed diameter at crosswise piles: 0.8 [m]

Assumed diameter at circumferential piles: 0 [m]

Defaults OK Cancel Help

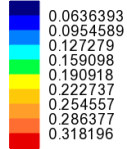
- Ensure design code **EN1992-1-1 (2005)** is selected.
- Ensure Computation target is set to **Visible** (to plot contours on the whole active part of the model) and click **OK**.

Once defined, a Tank design results entity for **EN1992-1-1 (2005)** (and its associated results components) can then be found within the contours and values properties droplists.

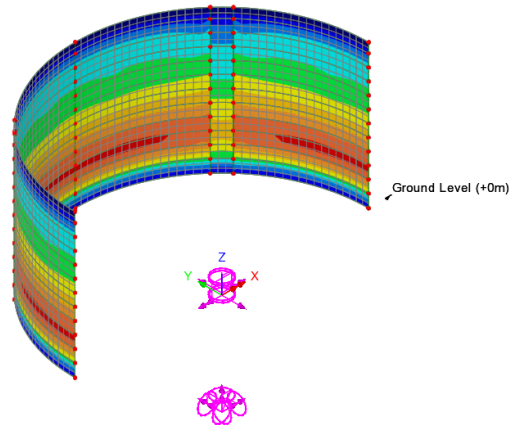
- In the Layers  Treeview, double-click the **Contours** layer and select the entity **Tank design BS EN1992-1-1 (2005)** and results component **UtilPM_t** and click **OK**.

Contours of utilisation will be drawn on the model.

Combination 1
Entity: Tank Design - BS EN 1992-1-1 (2005)
Component (Averaged nodal): UtilPM_t



Maximum 0.321701 at node 12376 of element 3976
Minimum 0.0353241 at node 16378 of element 5296



Export forces/design results to Excel (3D)

Detailed force/moments and design results for selected regions of a tank and loadcases of interest can be exported to a spreadsheet.

Tank

Design checks

Design Check

Report...

Tank - Design Check Report

Report filename:

Working folder: ☐ Current ☒ User Defined

Save in: ...

Target

☒ Angles: Group: ☐ Selected ☐ Visible

degree (eg. 10; 20; 30)

Nodal Averaging

☒ Unaveraged element results
☐ Averaged results from all elements
☐ Averaged results from visible elements

☐ Exclude base slab results at pile heads and walls

Assumed diameter at crosswise piles: [m]
Assumed diameter at circumferential piles: [m]

PM Check Report ☒ ☐ Peak element only ☐ Peak loadcase only

Shear Check Report ☐ ☐ Peak element only ☐ Peak loadcase only

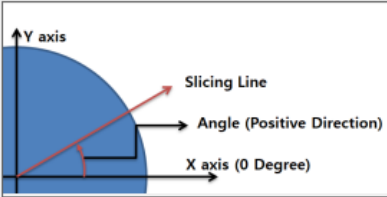
Utilisation Summary Report ☐ ☐ Peak element only ☐ Peak loadcase only

Loadcases / Combinations containing results

☐ Combinations only

19:Prestress (Ringbeam ONLY) (Long)
20:Prestress Wall Horizontal ONLY (Short)
21:Prestress Wall Horizontal ONLY (Long)
22:Prestress Wall Vertical ONLY (Short)
23:Prestress Wall Vertical ONLY (Long)
25:Max External Max Base Heating Temperature
26:Min External Max Base Heating Temperature
27:Max External Min Base Heating Temperature
28:Min External Min Base Heating Temperature
24:Dead Loads (LC1 to LC7)
61:RingbeamPS VerticalPS ONLY
66:Prestress Horizontal ONLY (Long)
67:Dead loads inner tank steel and others
68:Combination 1

PM Chart Type



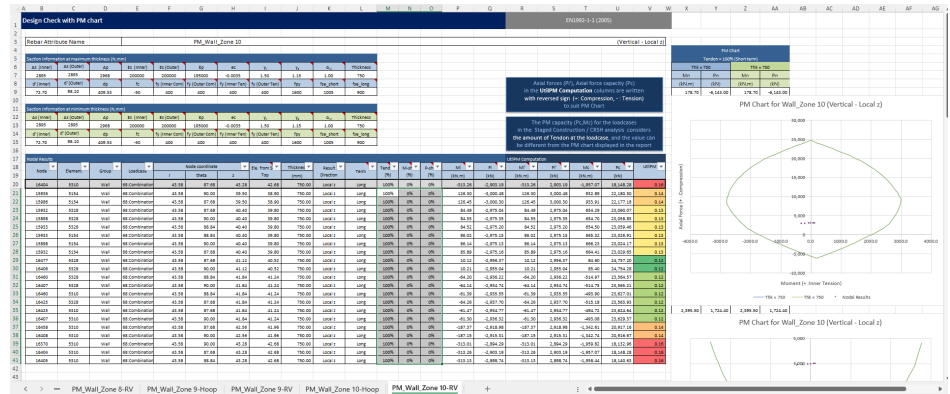
OK Cancel Help

- Enter a report name of **design** (noting that the angle and group name will be appended to the name of each spreadsheet created).
- Enter target angles of **45;90**
- Select a target group of **Tank Wall_Ringbeam**
- Ensure that only **PM Check Report** is selected.
- Ensure that only loadcase **Combination 1** is selected and click **OK**



Note: A separate spreadsheet of results is created within your Projects folder for each angle specified.

- Open the spreadsheet **design_90_Tank_Wall_Ringbeam_PM.xlsx** from within your Projects folder.
- Browse the tabs to see the types of results that are output.



This completes this example.

Additional notes

After a 3D shell 'base' model has been created, additional loading may be considered by using **Tank > Add loading** menu items:

- ❑ **Thermal** - If a thermal analysis is required, note that the evaluation of heat transfer through thickness cannot be performed with shell elements, so thermal analysis using unique 2D axisymmetric thermal/structural models should be carried out first to investigate maximum and minimum loading conditions. Temperatures obtained from these 2D analysis may be exported to unique Excel spreadsheets before using the menu item **Tank > Add loading > Temperature** to add the respective maximum and minimum thermal loading conditions to the 3D shell model as equivalent structural loading.
- ❑ **Seismic** - Seismic loading obtained from a separate dynamic model/analysis (e.g. a 2D beam-stick model) can be added to this model.
- ❑ **Staged construction** - All structural dead loads and either maximum or minimum variable loads as defined on the Tank definition 'Load' tab file will be added to the model. Variable loads may have a load factor applied.

- ❑ **Creep and Shrinkage** - Once a staged construction analysis has been set up, a new creep and shrinkage analysis can be created which will consider the same stages but in a viscous analysis in the time domain.
- ❑ **Spillage** - Spillage equivalent temperatures can be imported from a spreadsheet in the same way as described for other thermal loads.
- ❑ **Wind loading** - Wall and roof loading is computed according to a selected design code.

Additional analyses may be added to those present as required:

- ❑ **Eigenvalue Buckling analysis** - Eigenvalue buckling analyses based on existing loadcases.
- ❑ **GNL Buckling analysis** - A geometrically nonlinear buckling analysis based on an existing loadcase.

3D Solid Coupled Thermal / Structural Tank Model

Creating the model

To create a 3D solid quarter-symmetric model from previously defined tank definition data:

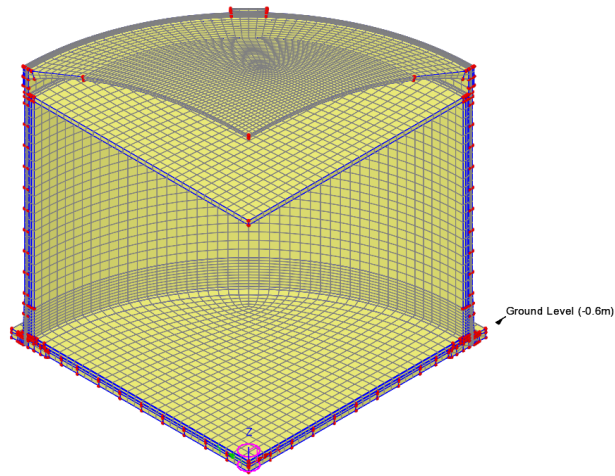
Tank
Create 3D Solid Model



Tip. Press the **Help** button for more information about this dialog and the type of model that is created.

- Enter a model filename of **tank** noting that the filename entered will be appended with '(3D Solid)' to identify this model within the set of tank models that are being created.
- For 'Model size' select **Quarter symmetric** from the droplist. Click **OK** to save any changes to the previously defined model.
- Press the **Template Download** button. A supplied template containing suitable rebar and tendon data will be saved to the working folder.
- Click **OK** to add rebar and replace all existing prestress loading with custom tendon profiles.

From the settings made, the following model will be created.



Note. A 3D solid tank model is supported only on simple spring supports. A quarter model is used because of symmetry and to keep solution time to a minimum for this example.



Note. Browsing the Groups , Attributes  and Analyses  treeviews will show the groups of features added, the attributes created and assigned to the model, and the analyses defined.



Note. Context menus for the items present can be used to locate features assigned to groups and to visualise attribute assignments.

Viewing assigned loadings



By ensuring the loading arrows are being drawn, and setting active each loadcase in turn, the loadings assigned to the model can be viewed and checked.

Running the analysis

With the model loaded:

-  Select the **Solve Now** button from the toolbar, then click **OK**.





Note. Solving of multiple analyses for a solid model may take an appreciable time.

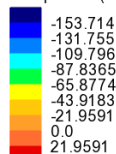
On completion, analysis loadcase results are added to the  Treeview and a deformed mesh layer is added to the view window. The deformed mesh layer is only of relevance for a structural loadcase.

Viewing the results

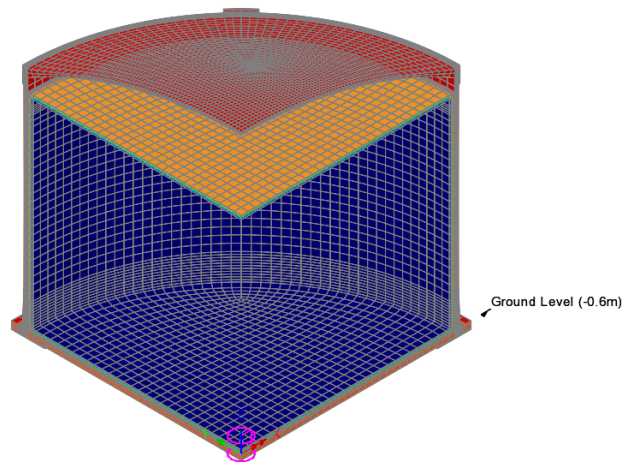
Thermal and structural analysis loadcase results are present in the  Treeview.

- In the  Treeview, set active the loadcase **Operating Condition (Thermal)**
- With no features selected, click the right-hand mouse button in a blank part of the view window and select the **Contours** option to add the contours layer to the  Treeview.
- Select entity **Temperature** and component **Temp** and click **OK**.

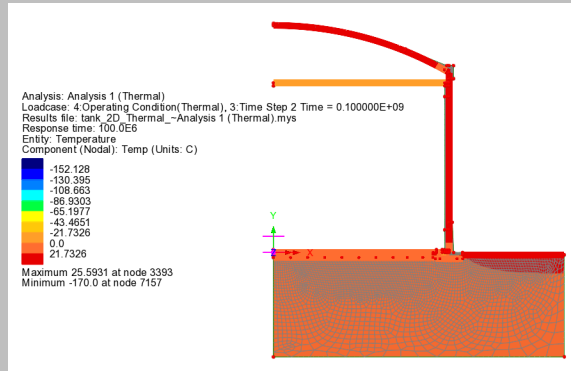
Analysis: Analysis 1 (Thermal)
 Loadcase: 3:Operating Condition (Thermal), 3:Time Step 2 Time = 0.100000E+09
 Results file: tank_apo_3D_Solid_~Analysis 1 (Thermal).mvs
 Response time: 100.0E6
 Entity: Temperature
 Component (Nodal): Temp (Units: C)



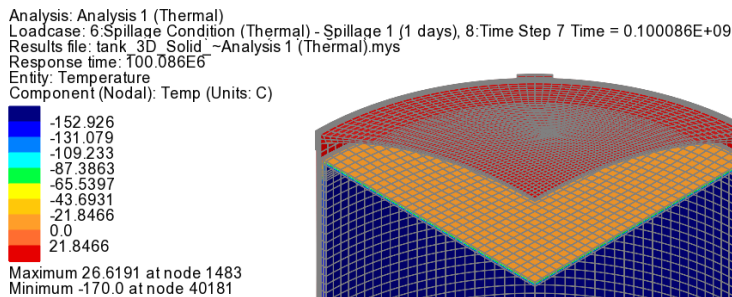
Maximum 27.6322 at node 1482
 Minimum -170.0 at node 40181



The values obtained from the 3D solid model can be seen to be similar to those obtained from the 2D coupled thermal/structural analysis (shown right).



- In the  Treeview, set active the loadcase **Spillage Condition (Thermal) – Spillage 1**



Exporting forces/design results

Results for chosen loadcases can be exported to a spreadsheet. Axial force, shear force and bending moments in two orthogonal directions (according to the results transformations) are provided.

Tank

Excel Tools

Export
Forces/Design
Results

Tank - Export Forces/Moments to Excel (3D)

Output filename:

Working folder: ☐ Current ☒ User Defined

Save in: ...

Target:

☐ Base slab ☒ Wall + Ringbeam ☐ Roof ☐ All

Results to extract:

☒ Forces and Moments
☐ Design results

Utilisations for concrete sections:

No design code is enabled - No reinforcement defined

ULS ☒ UtilPM ☒ UtilShear ☒ PM Capacity ☒ Shear Capacity ☒ UtilDecompression ☒ Compression Depth

Loadcases:

☐ Combinations only

1:Initial Condition (Structural)
4:Operating Condition (Structural)
5:Spillage Condition (Structural) - Spillage 1 (1 days) (last increment)
5-4:Time Step 3 Time = 0.100017E+09 (Spillage Condition (Structural)
5-5:Time Step 4 Time = 0.100035E+09 (Spillage Condition (Structural)
5-6:Time Step 5 Time = 0.100052E+09 (Spillage Condition (Structural)
5-7:Time Step 6 Time = 0.100069E+09 (Spillage Condition (Structural)
5-8:Time Step 7 Time = 0.100086E+09 (Spillage Condition (Structural)

Range:

Angles: degree (eg. 10; 20; 30)
Interval: [m]

☐ Smooth slab forces at pile heads and wall

Assumed diameter at crosswise piles: [m]
Assumed diameter at circumferential piles: [m]

Diagram illustrating the slicing line and angle for force extraction:

OK Cancel Help

- Enter an output filename of **3d_operating_condition_10deg** noting that a target group name will also be added to the name entered.
- Select a target of **Wall + Ringbeam**
- Enter a slice angle of **10** degrees
- Ensure **Forces and Moments** is selected.
- In the loadcases panel ensure that **Operating temperature** is selected and click **OK**.

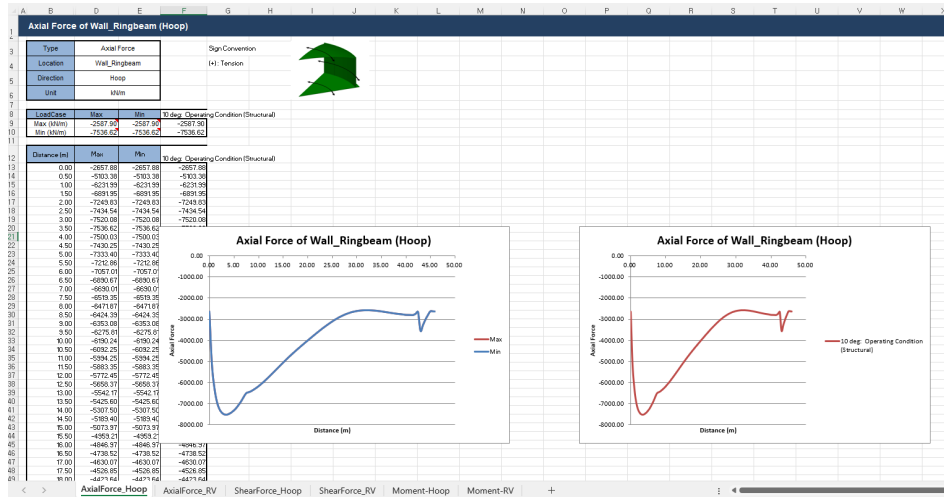


Note. Forces in the target selected are extracted by slicing the geometry at the defined angle, which calculates the forces from integration of stresses for those slices through the model.

The spreadsheets created in the specified folder comprise worksheets that contain force and moment data for the specified regions of the tank, e.g. Roof, Wall + RingBeam and BaseSlab.

3D Solid Coupled Thermal / Structural Tank Model

The spreadsheet for the target of Wall + RingBeam is shown below.



This completes this example.

Additional notes

- ❑ The exporting of forces for 3D solid models cannot exclude slab results at pile heads and will not include results from any buttress that may be present at each specified angle.
- ❑ For 3D solid models, the wall forces computed from integrating stresses through thickness do not include the shells contribution, which should be added manually.

