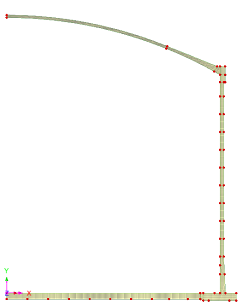
Storage Tank Modelling – 2D and 3D concrete models

For LUSAS version:	21.1
For software product(s):	LUSAS Tank
With product option(s):	Nonlinear, Dynamic, Thermal/Field, RC Slab design

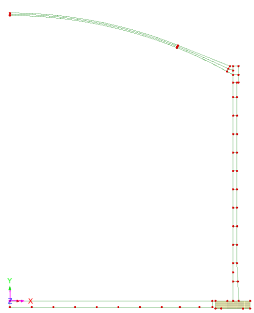
Description

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.

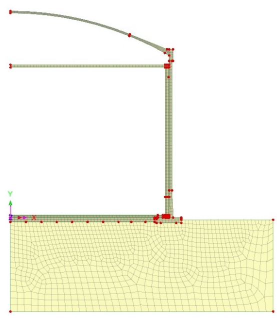
The range of concrete models include:



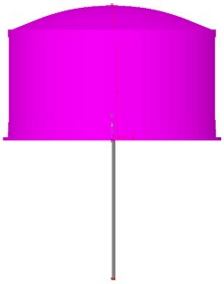
2D axisymmetric structural



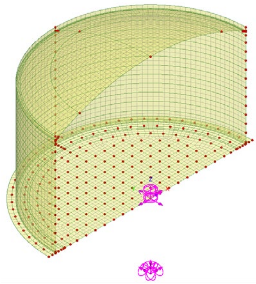
2D axisymmetric staged construction



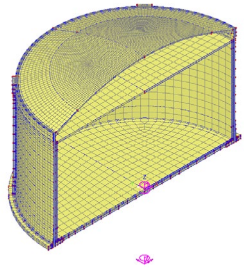
2D axisymmetric coupled



2D seismic
beam-stick



3D shell
(full/half)



3D solid coupled
(full/half/quarter)

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.

File
New…

Creating a New Model

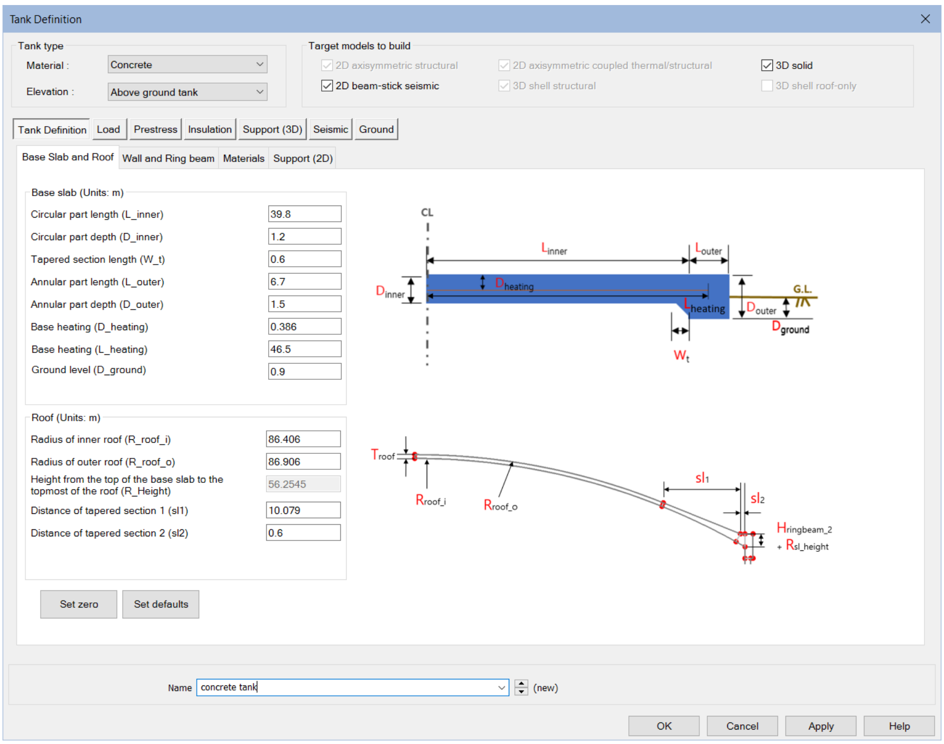
- Enter a file name of **tank**
- Use the default **User-defined** working folder.
- Ensure an Analysis type of **3D** is set. This is somewhat irrelevant since the analysis type will be automatically set when each tank model is created by LUSAS Tank.
- Click the **OK** button.

Defining tank 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.



- 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:

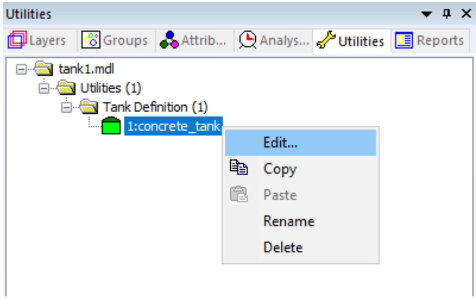
- Select the **2D axisymmetric coupled/thermal structural** checkbox. This adds an 'Insulation' button to the dialog.
- Select the 2**D beam-stick seismic** check box. This adds a 'Seismic' and 'Ground' button to the dialog.
- Select the **3D shell structural** check box. This add a '3D support' button 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 a 3D shell roof only model can be created. These types of tanks are not covered by this example, but model creation is 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 especially that a re-selection of the menu item **Tank > Tank Definition** will generate a new tank definition each time.

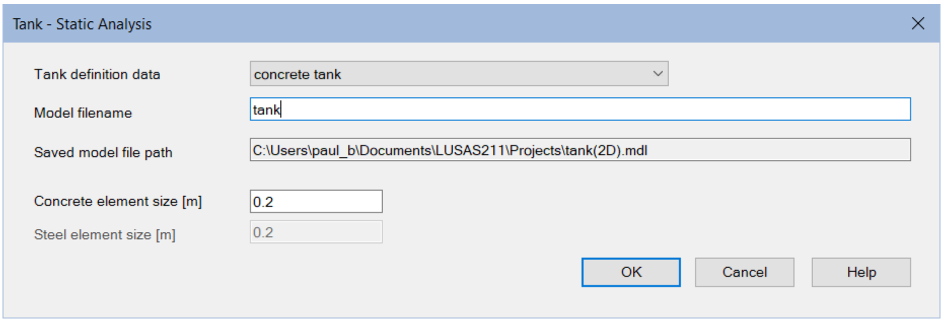
Concrete tank model examples available

- 2D axisymmetric structural
- 2D axisymmetric staged construction
- 2D axisymmetric coupled thermal/structural
- 2D beam-stick seismic
- 3D shell structural
- 3D solid coupled thermal/structural

2D axisymmetric structural tank model

A 2D axisymmetric structural analysis tank model will be created first.

Tank
Create 2D Model
Structural

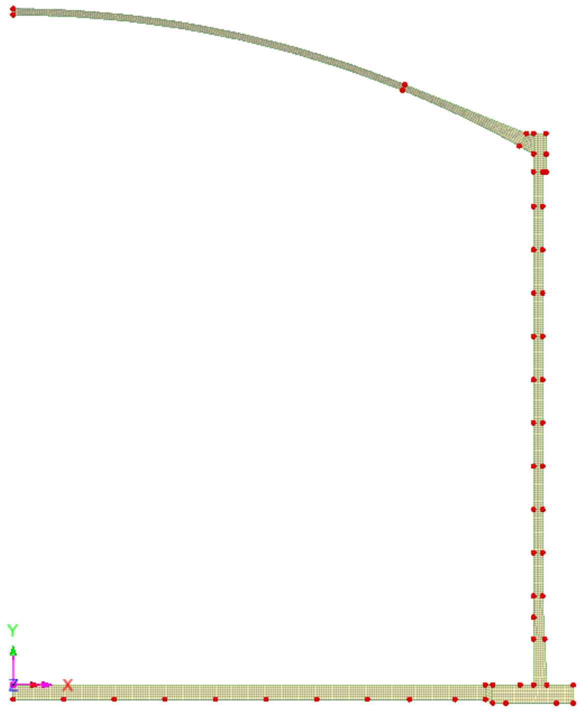


- If necessary, press the **Help** button for more information about this dialog and the type of model that is created.
- Accepting the default values on the dialog, enter a model filename of **tank**

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

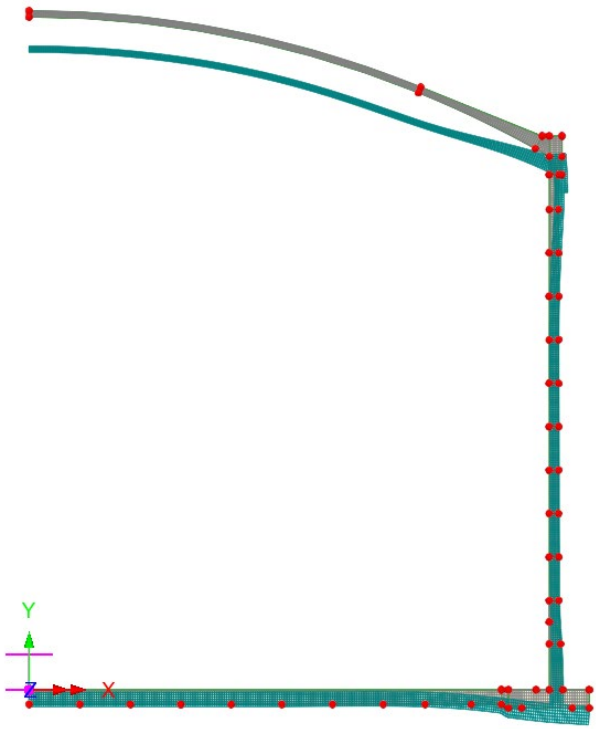
With the model loaded:

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.



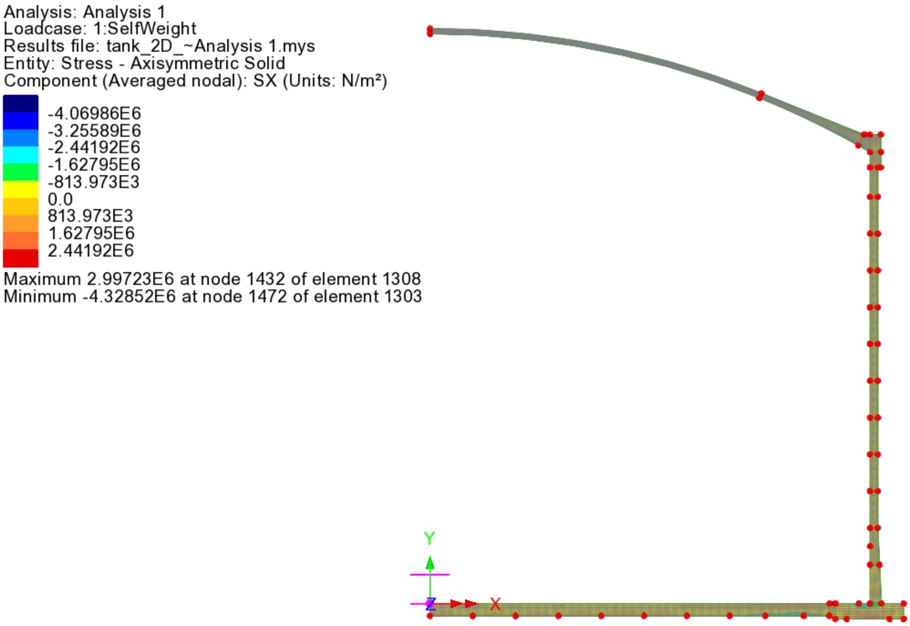
Viewing the results

- In the Layers Treeview, turn off the display of the **Deformed mesh** layer.
- In the Analyses Treeview, set active the loadcase **Self weight**

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.

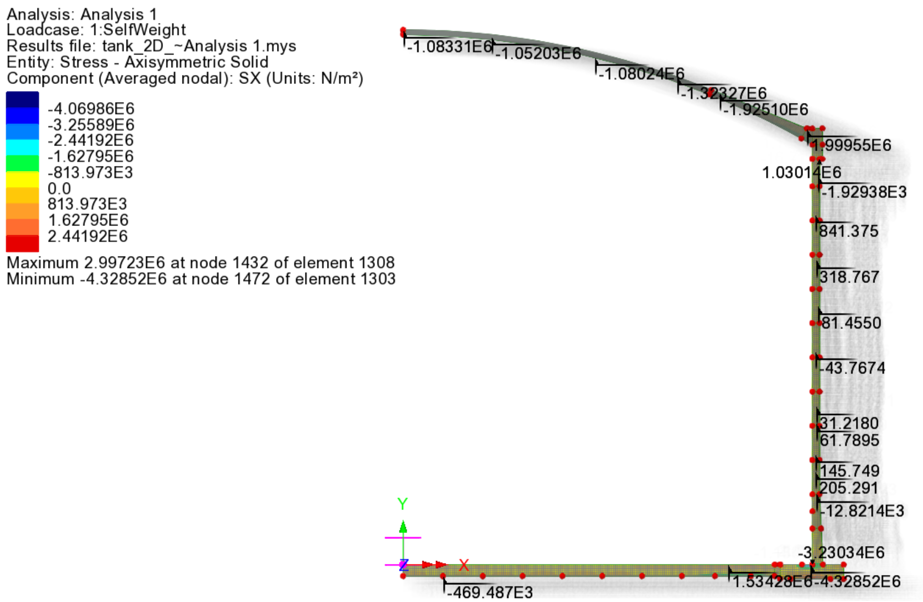
- Select entity **Stress** and component **SX** and click **OK.**



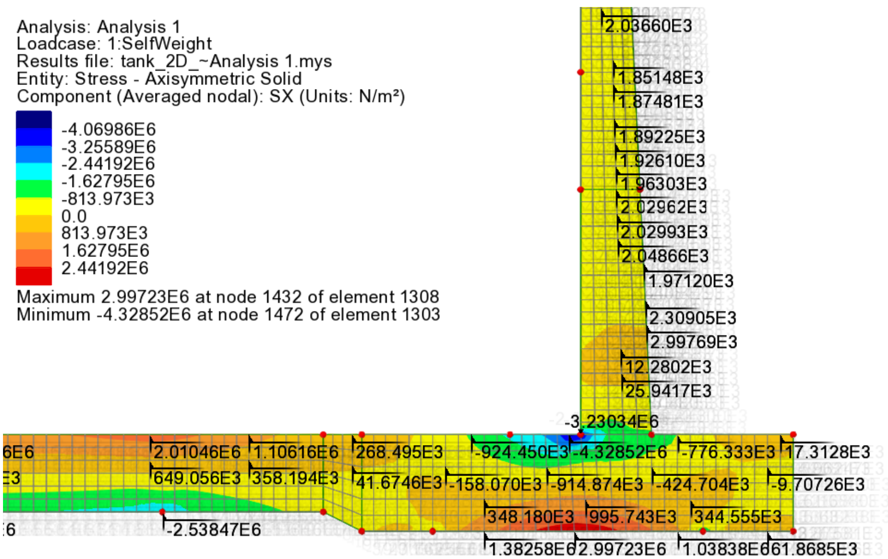
Values

- 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** and component **SX** and click **OK**.



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



 Resize the model to fit the view window.

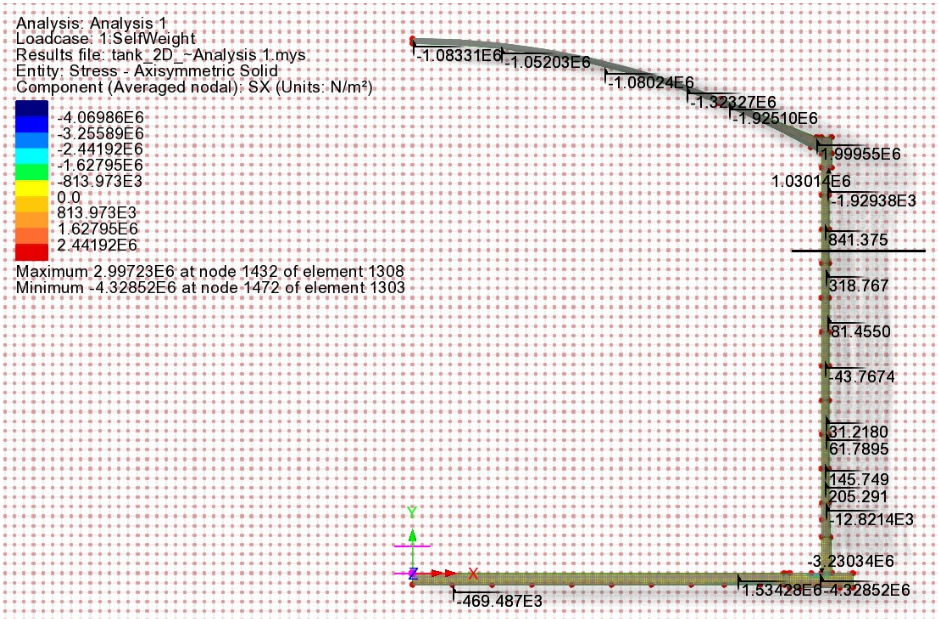
Graphs of stress and resultant forces through wall

The change of stress through the wall can be plotted.

With the whole model in view:

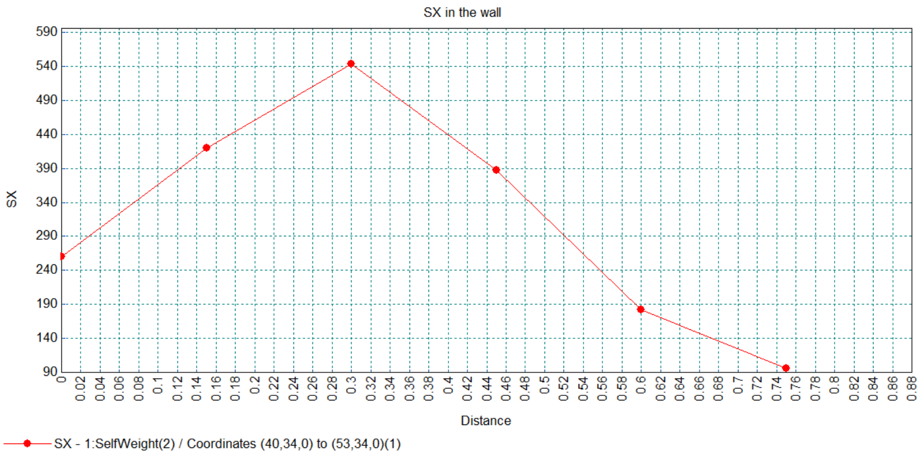
Utilities
Graph through 2D

- With **By cursor** selected, click **OK** to accept the defaults and draw a line through a chosen height of wall.



- On the Loadcases and Extent dialog ensure **Self weight** is selected and click **Next**.
- On the Slice data dialog select entity **Stress** and component **SX** 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**.

A graph showing the variation of SX with wall thickness is generated. As the model units are N,m, the stress unit is N/m^2. The X axis in the graph is the distance from the start point of the selected slicing line.



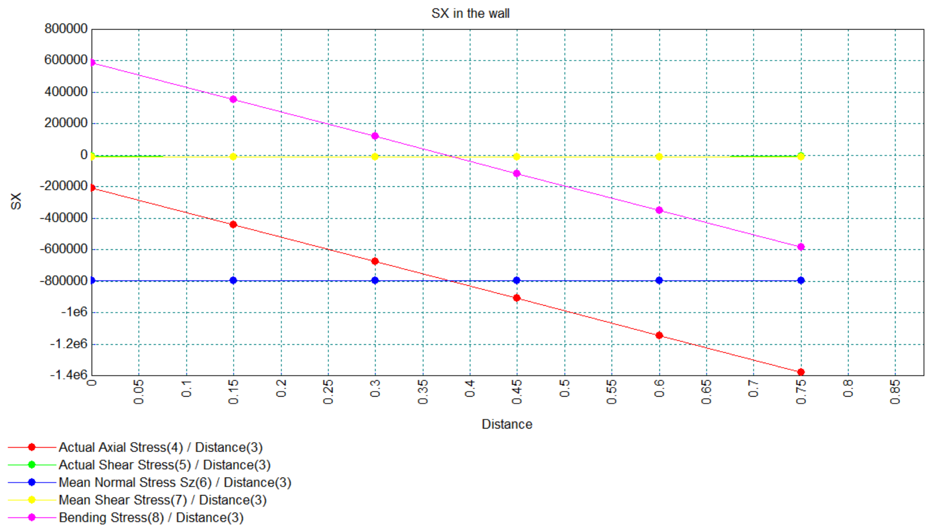
- Reselect the View window.

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

Utilities
Graph through 2D

- Select At location of existing graph and click OK.
- On the Loadcases and Extent dialog ensure **Self weight** 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.

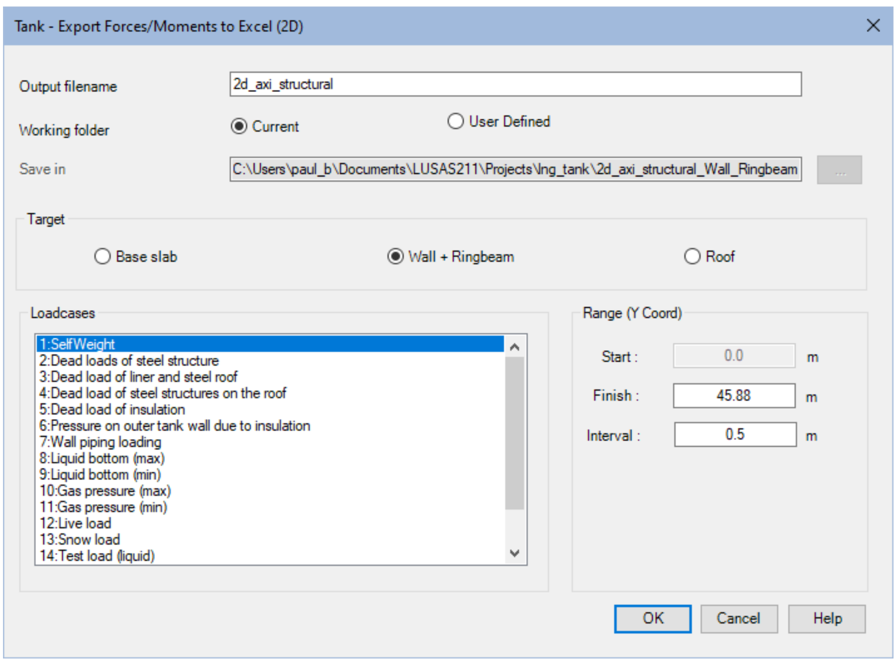


- Reselect the View window.

Export Forces to Excel (2D)

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

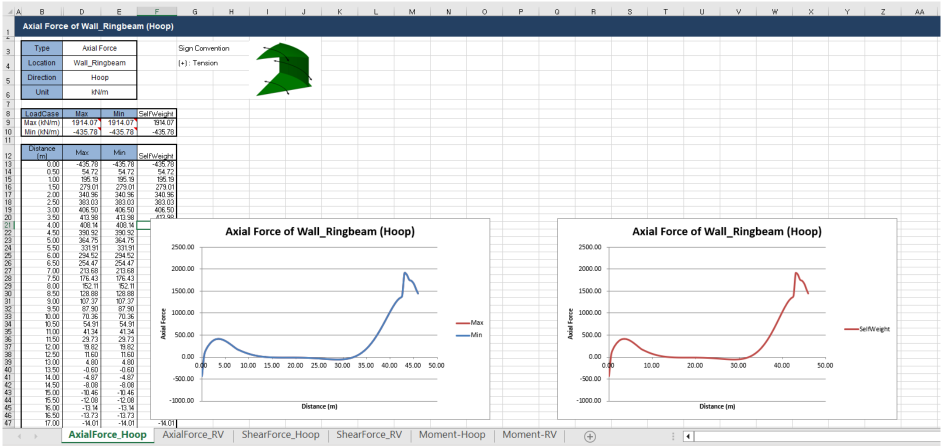
Tank
Excel Tools
Export Forces/Design Results



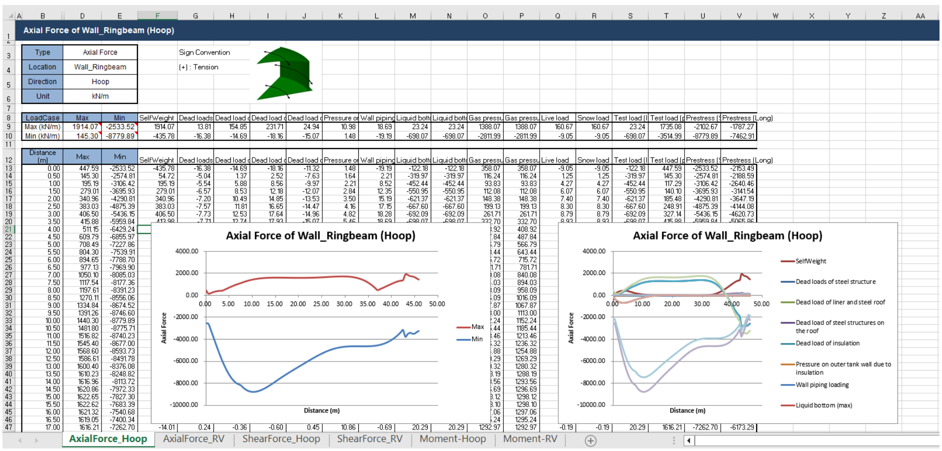
- Enter an output filename of **2d_axi_structural** (noting that the target name is appended to the name entered).
- For a target, choose **Wall + Ringbeam**
- In the Loacases panel, ensure **Self weight** is selected and press **OK**

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.



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.



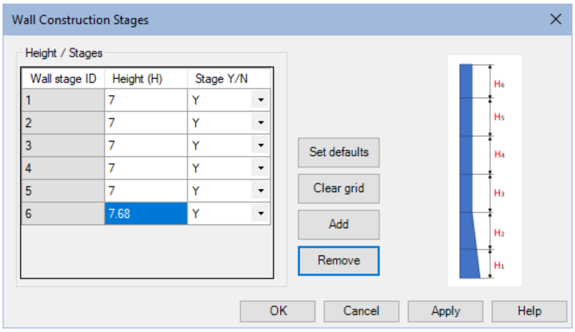
This completes this tank example.

2D staged construction tank model

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.

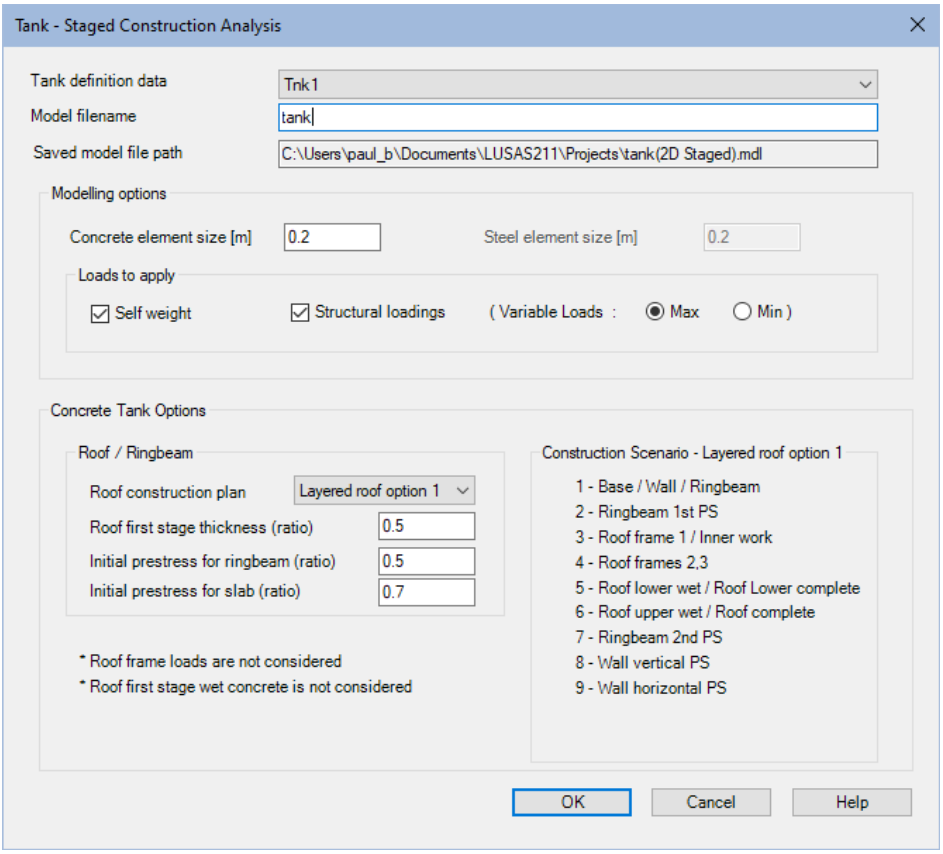
The total number of construction stages to be created will be as stated on the ‘Staged construction analysis’ dialog according to the chosen ‘Roof construction plan’, the ‘Roof 1st stage thickness (ratio)’, the ‘Initial Prestress for Ringbeam (ratio) and the ‘Initial Prestress for slab (ratio)’ plus 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, but 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. This would result in this dialog being displayed (that shows example inputs).



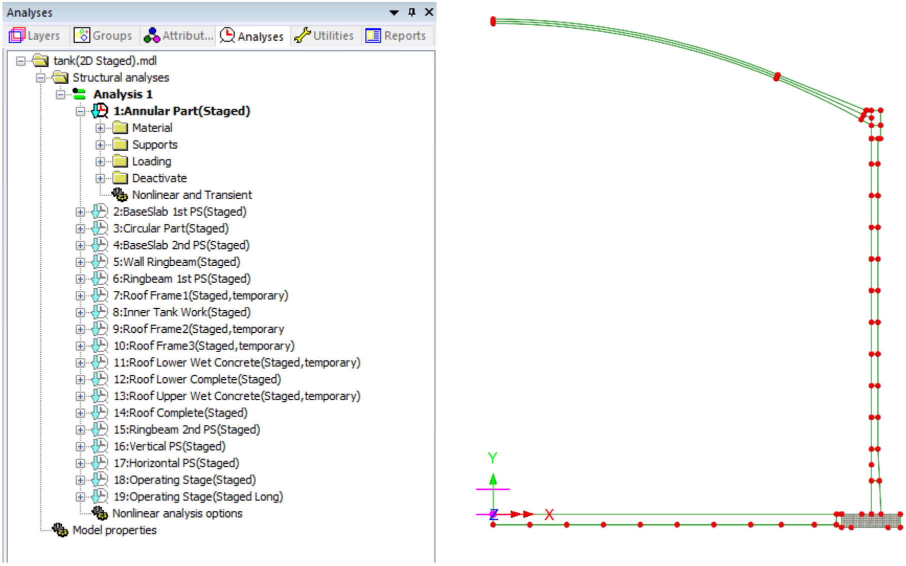
Tank
Create 2D Model
Staged Construction

To create the staged construction tank model:



- If necessary, 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 '(2DStaged)' 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.

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.

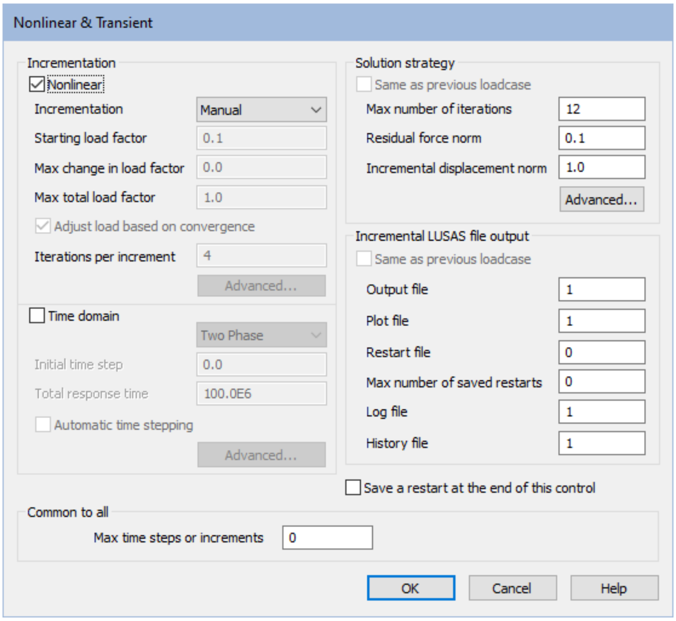
Nonlinear Analysis Controls

The geometry of the structure changes at each loadcase, so a Nonlinear Control is used (appearing as Nonlinear and Transient with the Analyses Treeview). When Nonlinear Control is set for the 1st loadcase 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 entry to see the default settings used



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:

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 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)

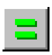
Stage 19 models the operating (in-service) stage for long-term.

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

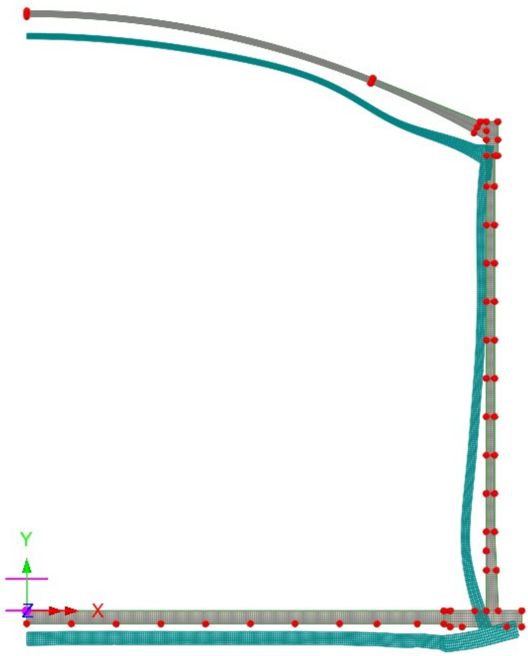
With the model loaded:

 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 (x86)\scripts\Tank" folder.

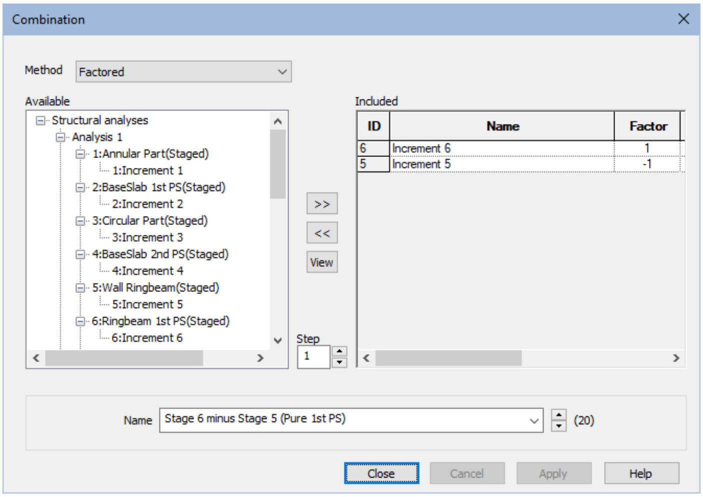
	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W	X
1	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	23	24
2	Description			Permanent											Variable									
3				Outer tank			Others	Creep and Shrinkage		Prestress					Roof frame/ concrete					Test		LNG pressure		Ga pres
4	no.	Code	Details	Outer tank WO roof	Outer tank WO uper 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	Hydros tatic	Pneum atic	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 Stage 6, hence the pure effect of the 1st Prestress can be obtained by defining a load combination for loadcase ‘Stage 6’ minus loadcase ‘Stage 5’. This would be done as follows:

Analyses
Basic Combination

Include **Stage 6** with a load factor of **1**, and **Stage 5** with a load factor of **-1** as shown, with a name of ‘**Stage 6 minus Stage 5 (Pure 1st PS)**’



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 Forces/Moments**.

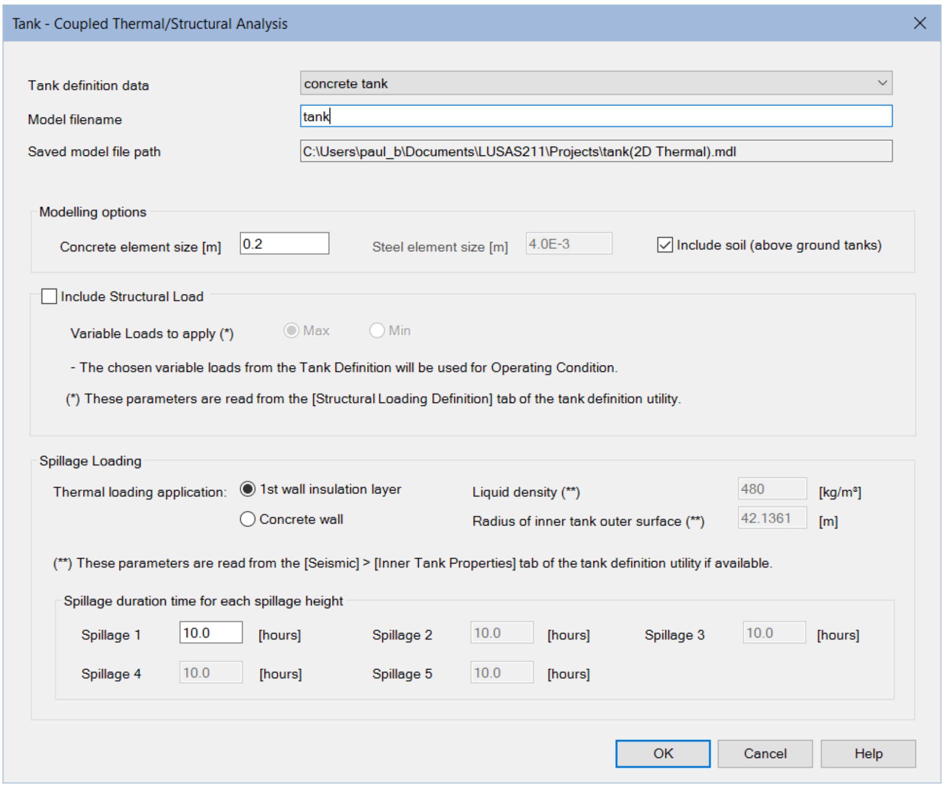
This facility requires the prior creation of input spreadsheets with the loadcase results (obtained from Export Forces/Design results) and combine them in Excel based on the factors read from the Tank Template Combination template.

2D coupled thermal/structural tank model

A 2D coupled thermal/structural analysis tank model is used to investigate temperature effects.

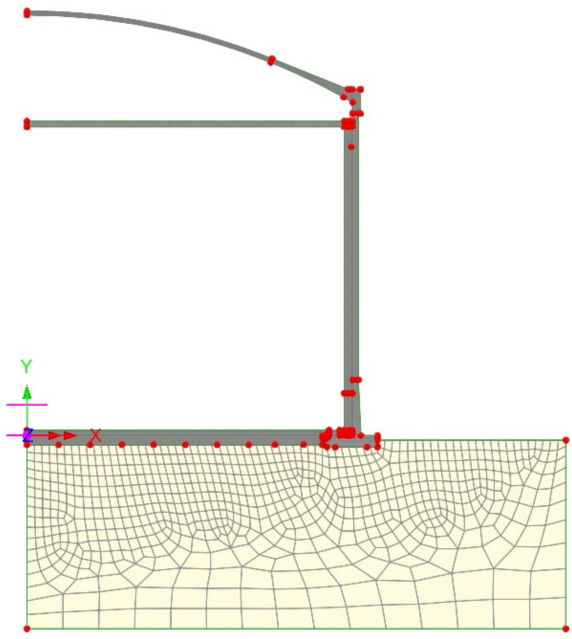
Thermal data from this 2D analysis can be exported for use by a 3D shell model because the evaluation of heat transfer through thickness cannot be performed with shell elements.

Tank
Create 2D Model
Coupled Thermal/Structural



- If necessary, 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.

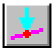
After a short while the following model will be created.



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.

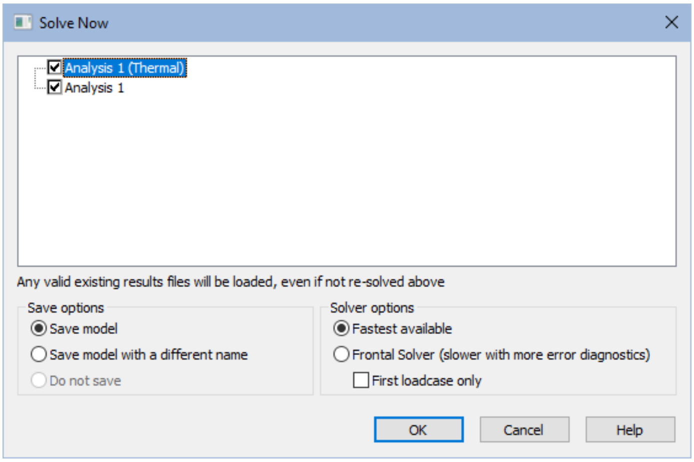
Viewing assigned loadings

 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.

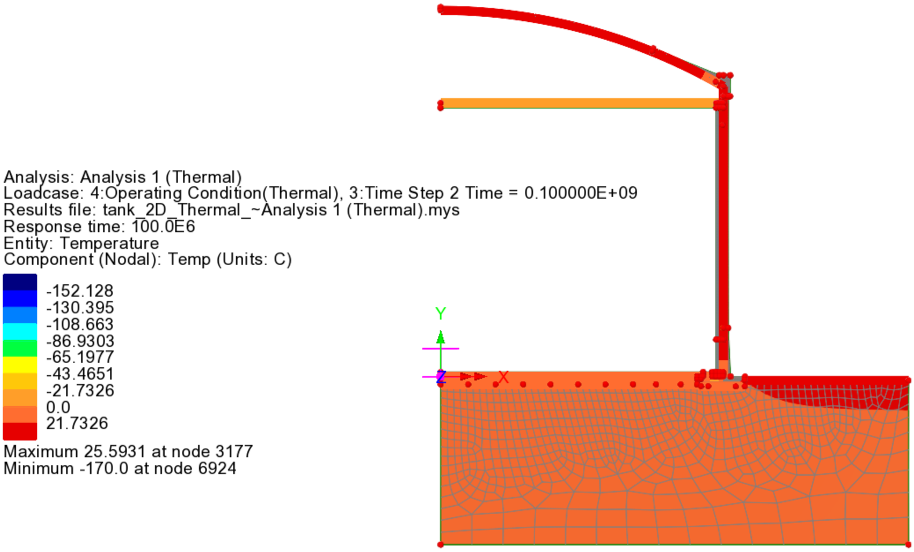
If the analysis is successful...

Analysis loadcase results are added to the Treeview.

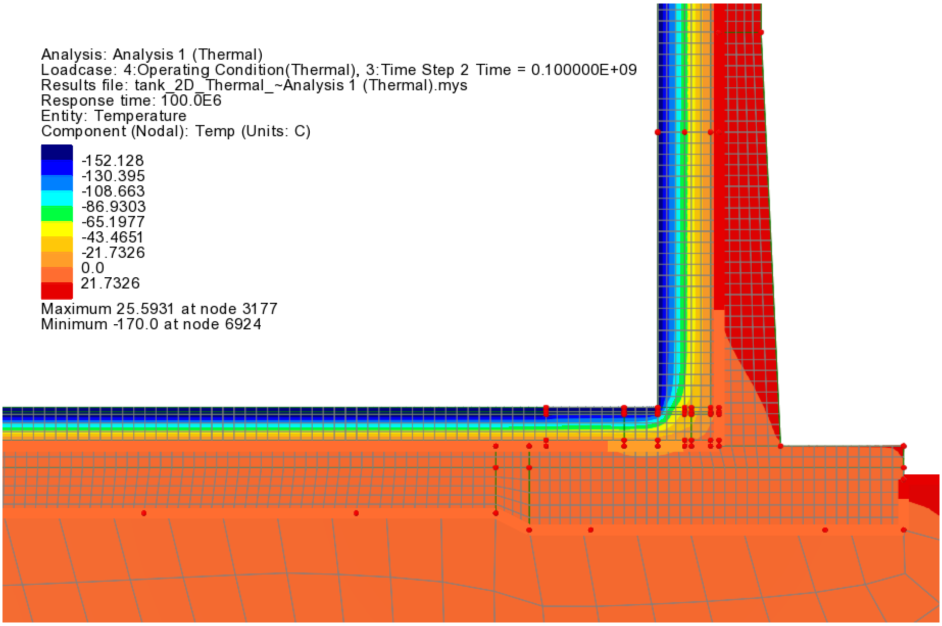
Viewing the results

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

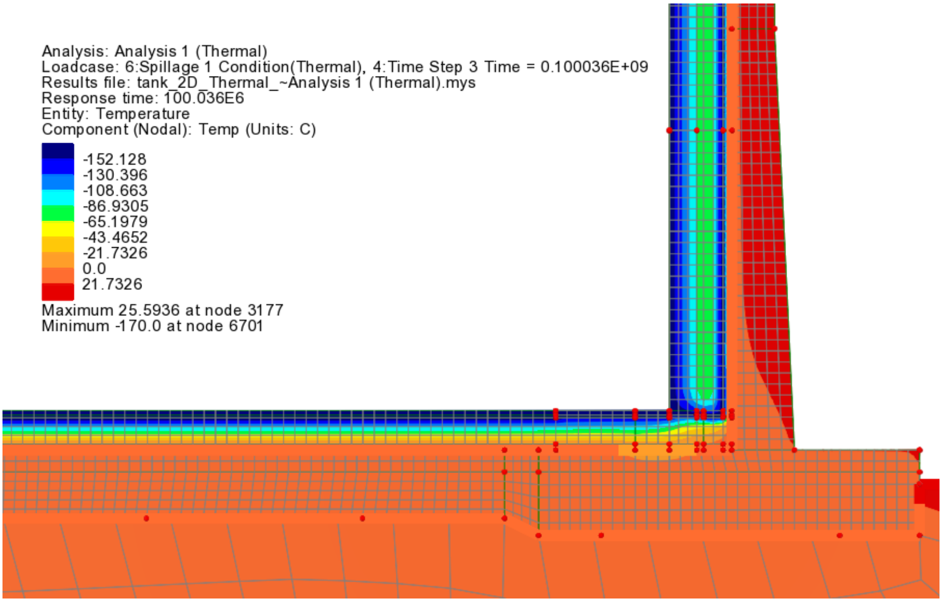
- In the Treeview, set active the loadcase **Operating Condition (Thermal)**
- With no features selected, click the right-hand mouse button in a blank part of the view window and select the **Contours** option to add the contours layer to the Treeview.
- Select entity **Temperature** and component **Temp**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



- Enlarging the view will show more detail.



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



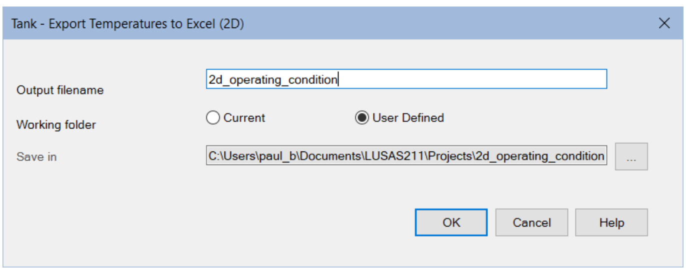
Exporting 2D thermal results

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:

Tank
Excel Tools
Export Temperature

- Enter an output filename of **2d_operating_condition**



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	23.70	25.33	0.50	24.49	-3.58	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	25.32	0.50	24.47	-3.71	0.01
2.00	23.61	25.33	0.50	24.47	-3.70	0.02
2.50	23.61	25.33	0.50	24.47	-3.71	0.04
3.00	23.61	25.32	0.50	24.47	-3.71	0.05
3.50	23.61	25.32	0.50	24.47	-3.71	0.07
4.00	23.61	25.32	0.50	24.47	-3.71	0.09
4.50	23.60	25.32	0.50	24.46	-3.72	0.12
5.00	23.61	25.32	0.50	24.47	-3.70	0.14
5.50	23.60	25.32	0.50	24.46	-3.73	0.17
6.00	23.62	25.32	0.50	24.47	-3.70	0.21
6.51	23.60	25.32	0.50	24.46	-3.73	0.24
7.01	23.61	25.32	0.50	24.46	-3.71	0.28
7.51	23.60	25.32	0.50	24.46	-3.73	0.33
8.01	23.60	25.32	0.50	24.46	-3.73	0.37

Roof | Wall_RingBeam | BaseSlab

This completes this example.

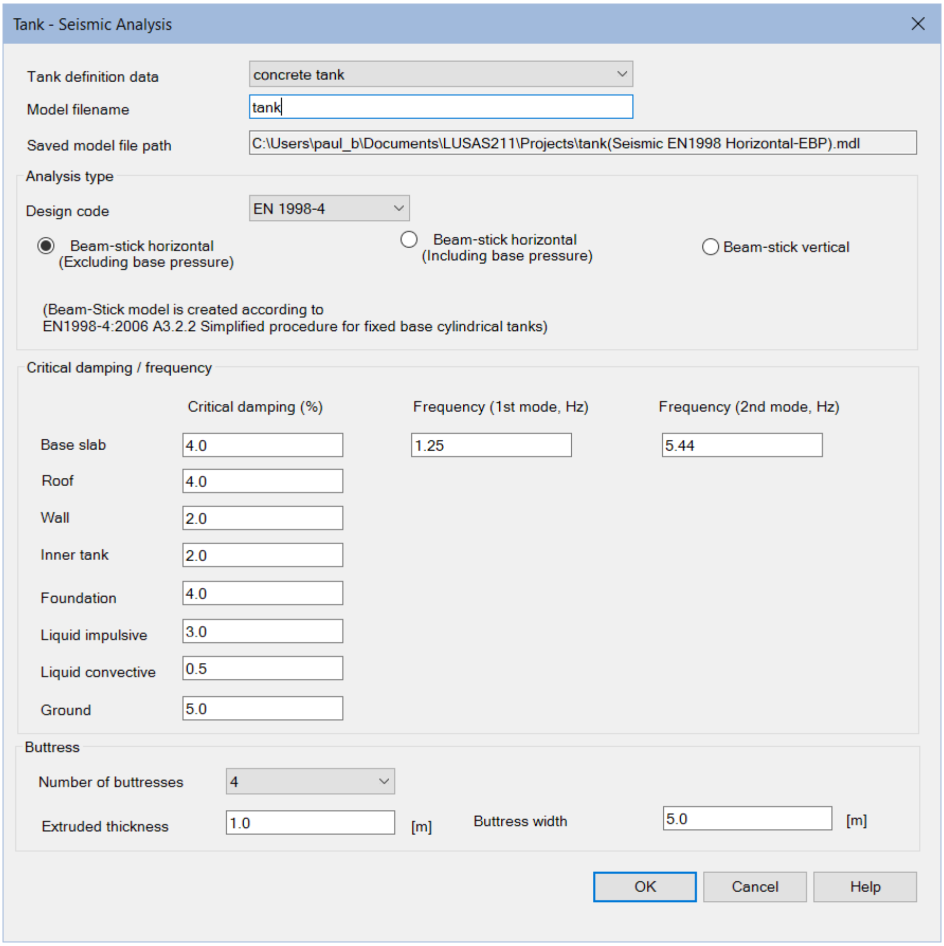
Additional notes

- 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.

Tank
Create 2D Model
Seismic…



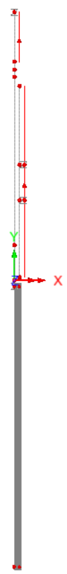
- If necessary, press the **Help** button for more information about this dialog and the type of model that is created.

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.

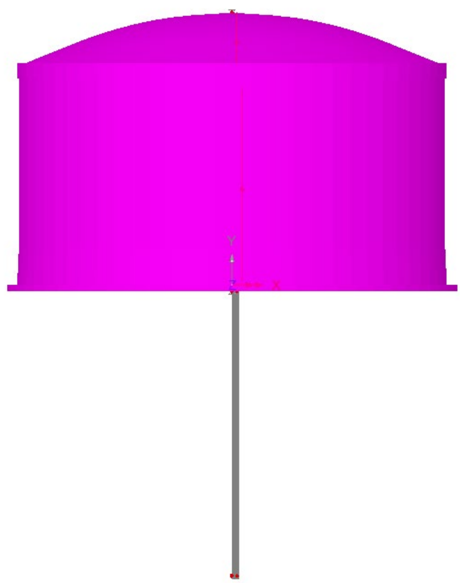
- Accepting the remaining default values on the dialog (to model horizontal actions excluding base pressure for the EN 1998-4 design code), enter a model filename of **tank** and click **OK**. The filename entered will be appended with '(Seismic EN1998 Horizontal-EBP)' to identify this model within the 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

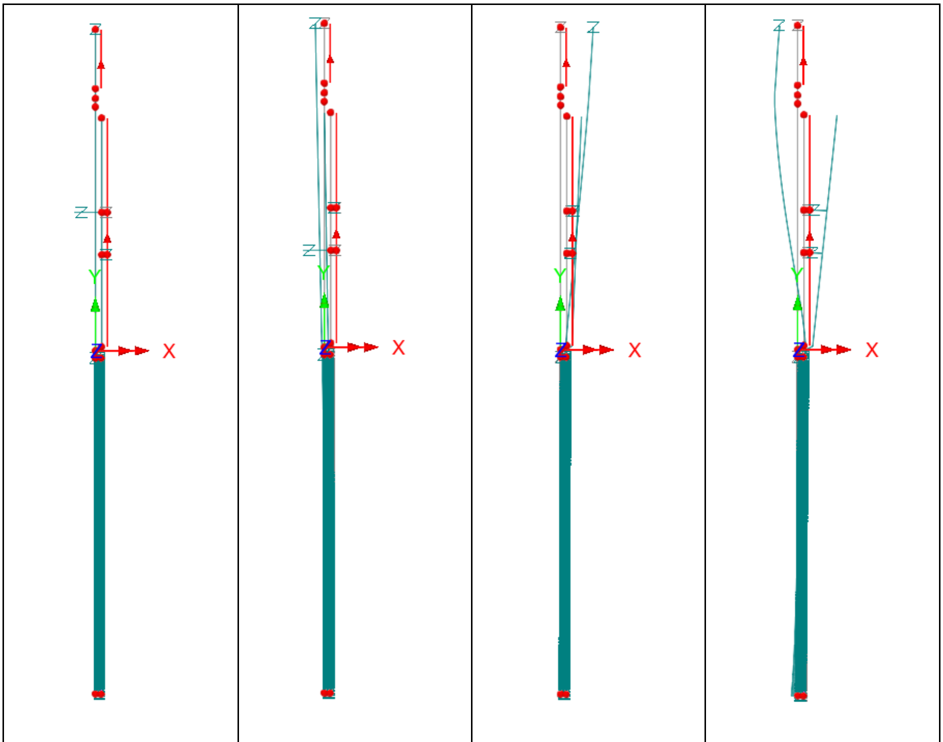
With the model loaded:

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

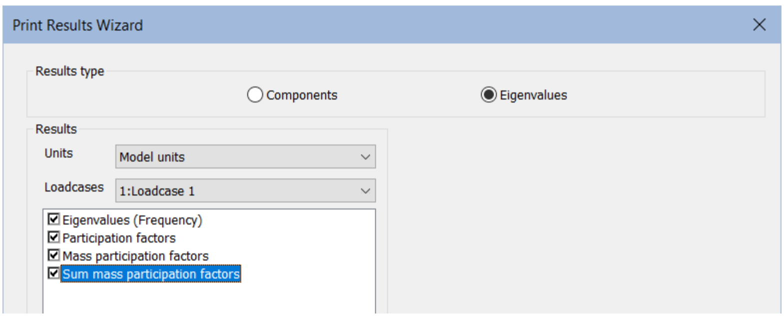
- Mode shapes may be viewed by setting active each loadcase in turn within the Analyses Treeview of the generated model. The first four modes shapes are shown.



Mode 1	Mode 2	Mode 3	Mode 4

Tabulating eigenvalue results

Utilities
Print results wizard



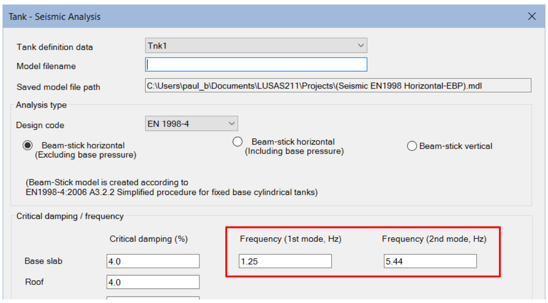
- Select result type **Eigenvalues**
- Select all 4 results check box options and click **OK**.

	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	29.2697E-18	0.0	0.0	0.0	0.675368	10.9011	0.0917338
5	5	0.170597	1.31484E-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	0.194276E-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	1.82416E-15	0.0	0.0	0.0	0.103791E-3	61.1417	0.0163555

- 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.

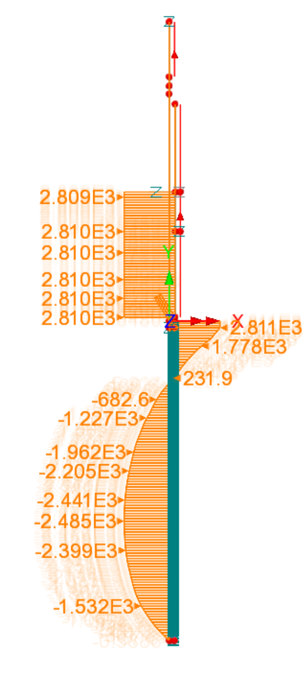
From this it would be reasonable to use the 2nd and 3rd frequencies obtained as the frequency range for computing damping constants for the 1st and 2nd mode frequencies, previously provided in the Tank - Seismic Analysis model creation dialog. This example does not do this however.



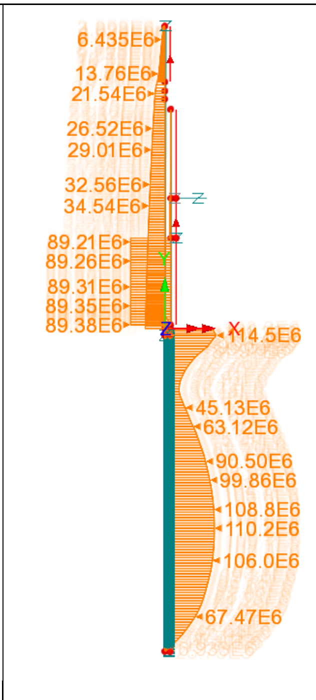
Diagrams

- 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** 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**

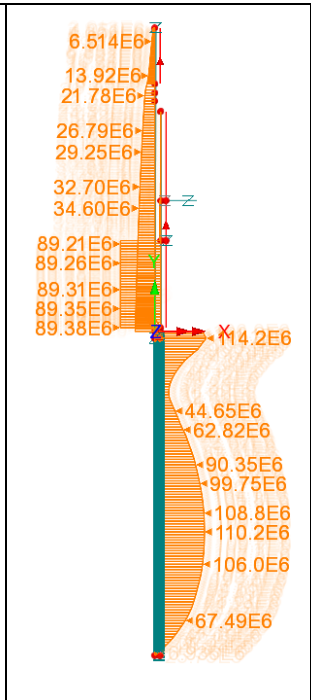
The diagrams below show the Fy results for each of these loadcase selections.



Mode 1



Response spectrum CQC



Response spectrum SRSS

Show the damping applied to each mode

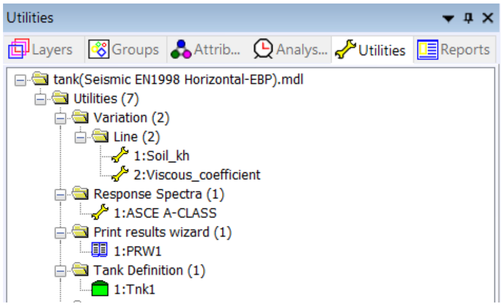
Because the option to **Include modal damping** is checked in the **Eigenvalue** control dialog, the modal damping factors computed for each mode are printed in the Solver output file, which has a file extension of '*.out'. This can be found at this location: **<Current working folder> \LUSASFiles32\ Example_EN1998_HorizontalBeamStick(IBP)**

M O D A L   D A M P I N G   F A C T O R S

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	0.239496E-01
4	4691.38	10.9011	0.434131E-01
5	7504.93	13.7877	0.560321E-01
6	15937.8	20.0925	0.879954E-01
7	19872.9	22.4363	0.878306E-01
8	30093.9	27.6096	0.111759
9	34985.1	29.7688	0.106512
10	52420.0	36.4392	0.159976
11	67807.0	41.4436	0.204024

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.



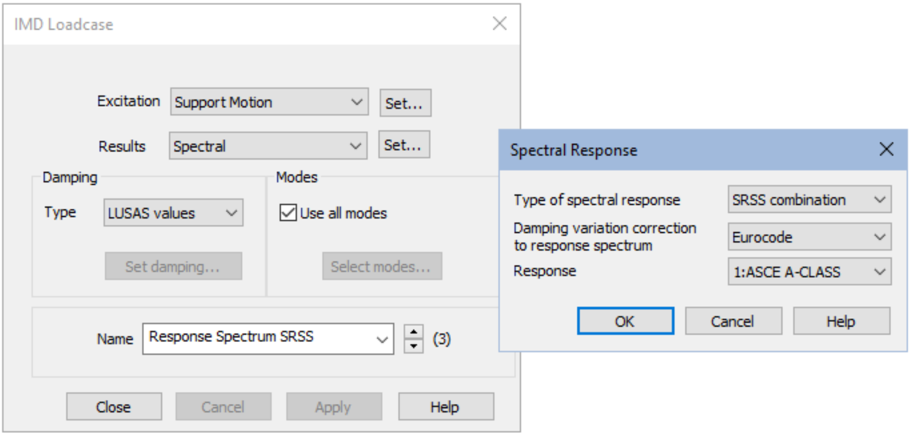
Note. Other design codes may be selected from the dialog that is displayed when double-clicking this entry.

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.

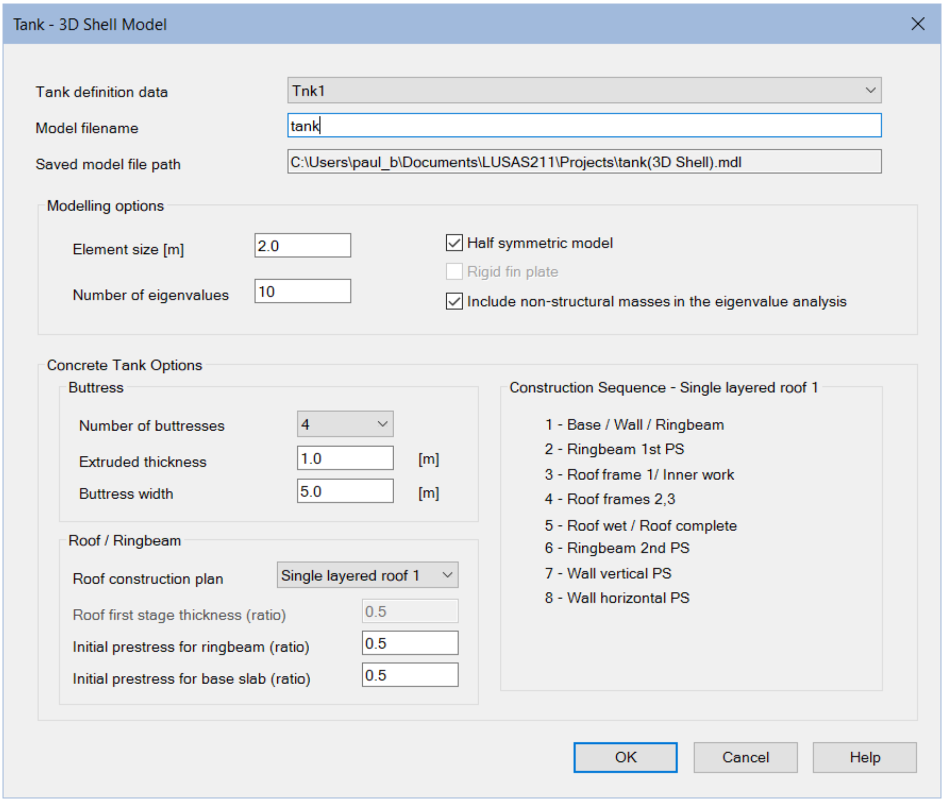
Additional notes

- ❑ 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 tank model

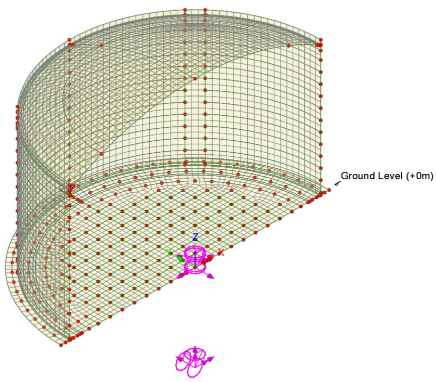
To create a 3D shell half-symmetric model:

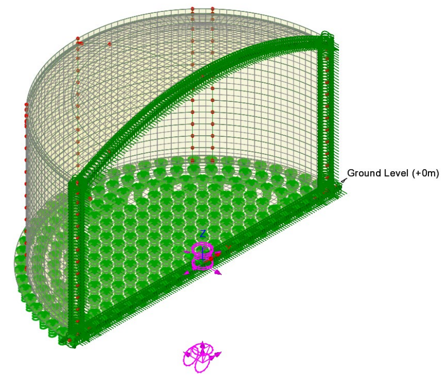
Tank
Create 3D Shell Model

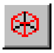


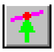
- If necessary, press the **Help** button for more information about this dialog and the type of model that is created.
- Accepting the default values, enter a model filename of **tank** and click **OK**. The filename entered will be appended with '(3D Shell)' to identify this model within the set of 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.

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.

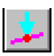
Note. By default, a 3D shell tank is supported on spring supports. Prior to creating a model, and from within the Tank Definition 3D Support tab a detailed foundation may be chosen which, depending on the tank elevation setting in use, allows for isolator, pedestal, raft and pile type settings and dimensions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

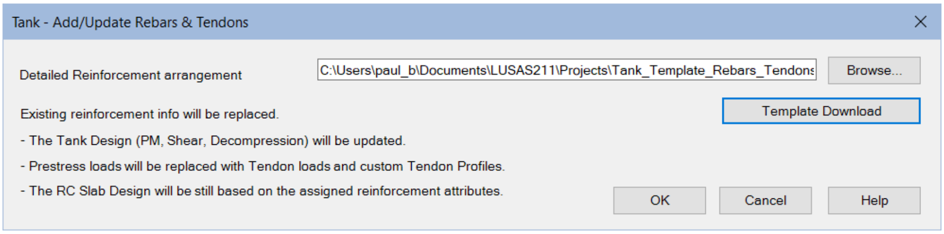
Viewing assigned loadings

 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.

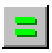
Tank
Design Checks
Add/Update Rebars and Tendons



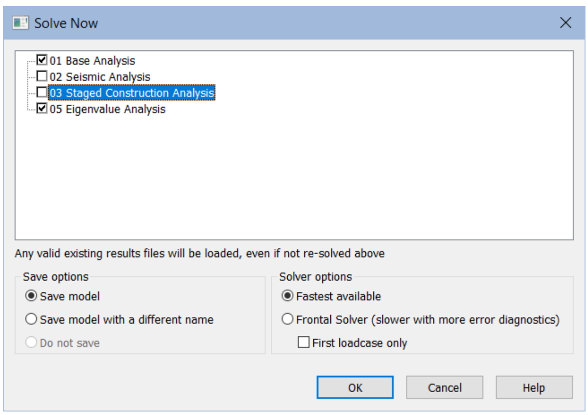
- On the dialog, press the **Template Download** button. A supplied template containing suitable rebar and tendon data will be saved to the working folder.
- Click **OK** to add rebar and replace all existing prestress loading with custom tendon profiles.

Running the analysis

With the model loaded:

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

For simplicity not all analyses 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

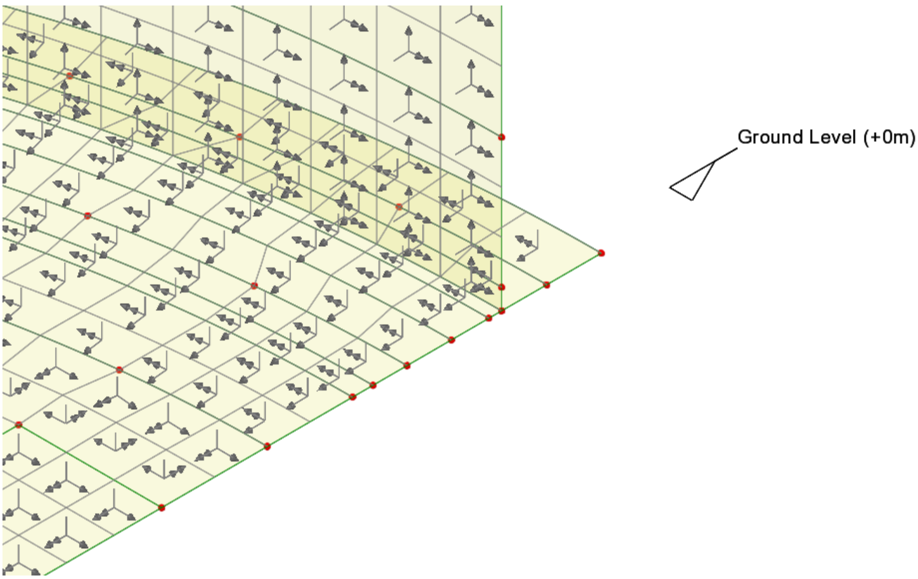
- Turn off the display of the **Deformed mesh** layer.
- In the Treeview, set active the loadcase **Self weight**

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 to enlarge the model will show the element axes more clearly.

The x-axis of the elements in the group Tank Wall 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.

So, 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:

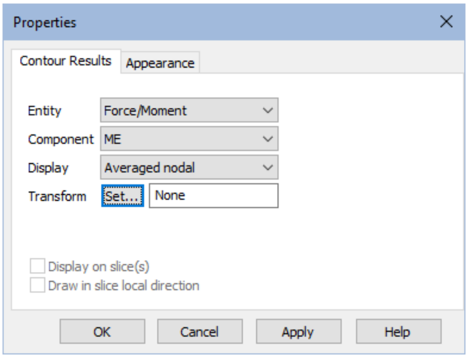
Results transformation dataset	Shell plane for	Local coordinate set
ResultTransformation_Roof	theta/phi	'Spherical for force Extraction'
ResultTransformation_Wall	theta/z	'Local Coord'
ResultTransformation_BaseSlab	r/theta	'Local Coord'

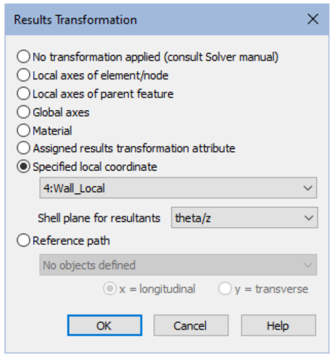
Plotting Contours

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 Mx 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.

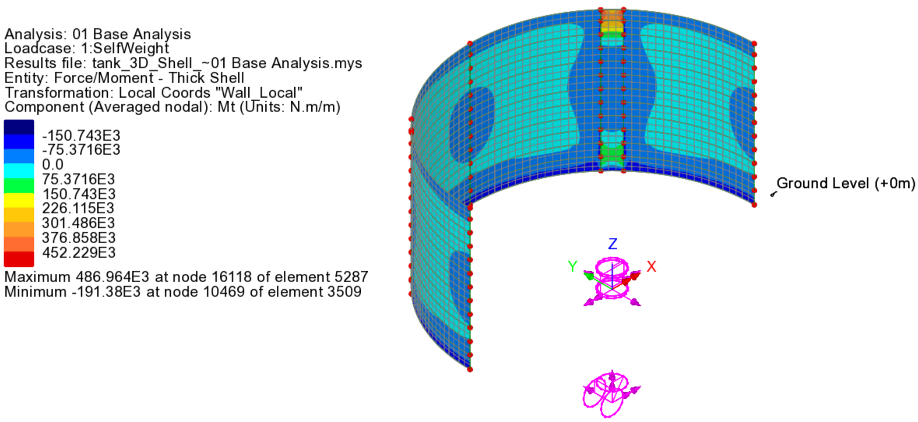
- On the contour properties dialog, select entity **Force/Moment**, press the transform **Set** button and for the **Specified local coordinate** select **Wall_Local** from the droplist provided and **theta/z** from the 'Shell plane for results' droplist, because the wall surface element axes have a theta and z direction. Click **OK**.





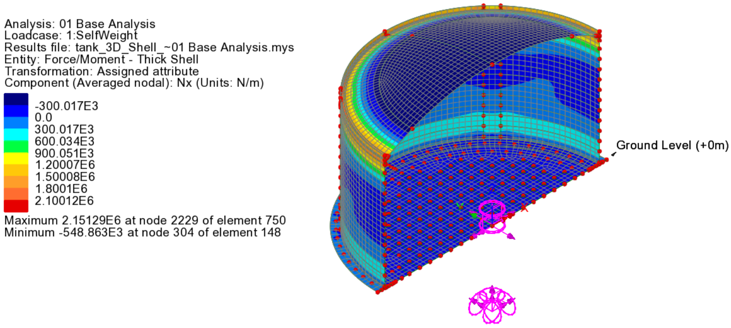
This now makes available a results component ‘Mt’, where ‘t’ represents tangent direction in the cylindrical local coordinate system.

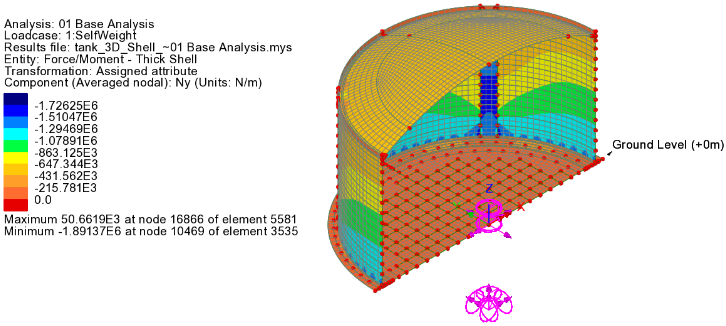
- Now, back on the contour properties dialog, select the result component ‘**Mt**’.
- To see only the forces/moments on the wall, in the Groups  Treeview, right-click on the **Tank_Wall** entry and select **Set As Only Visible.**



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.

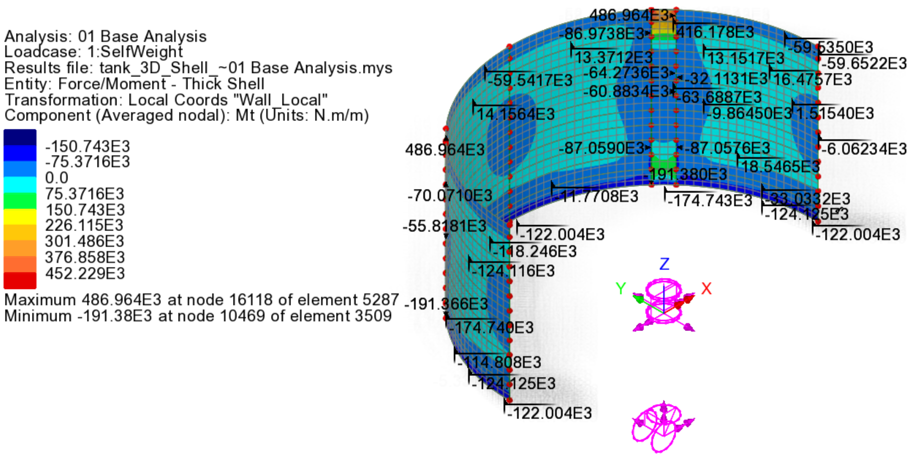
When this is done, any components with ‘x’ represent the results in the hoop direction (wall/roof) or radial direction (base slab), and those with ‘y’ represent results in the radial (roof) or vertical (wall) direction or the hoop (base slab) direction. If plotted on the whole model the following contour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





Values

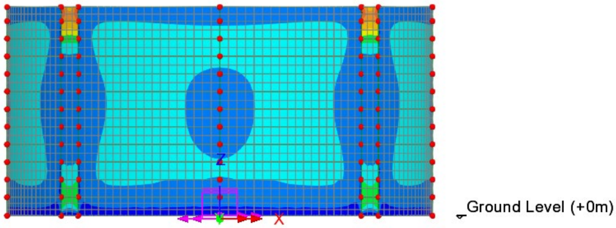
If necessary, values can be displayed for chosen nodes by adding the Values layer to the Layers treeview. Zooming in will provide more detail.



Graphing of results through the wall

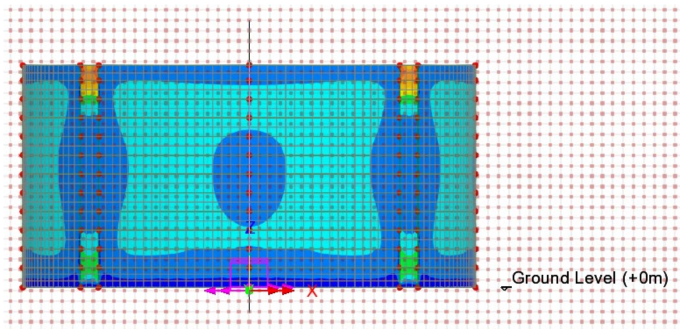
Arbitrary line sections may be taken through any surface of a two dimensional model or on a slice cut through a three dimensional solid model. The process of cutting a slice will generate two graph datasets in the Utilities Treeview; the first containing the distance along the line section and the second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

- Rotate the model to be viewed as shown in the following image.



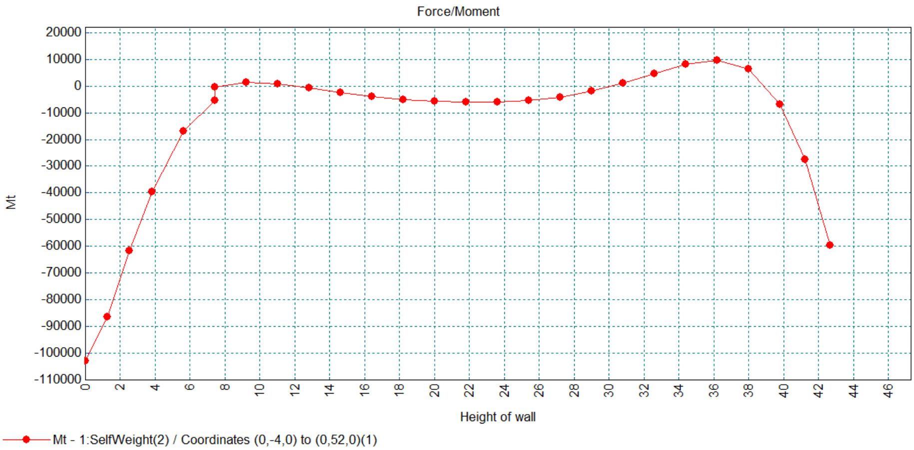
Utilities
Graph Through 2D

- 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 to above the top of the wall as shown:



- Then, on the 'Loadcases and Extent' dialog select **Self weight** and click **Next**, select results component **Mt** and click **Next**.

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



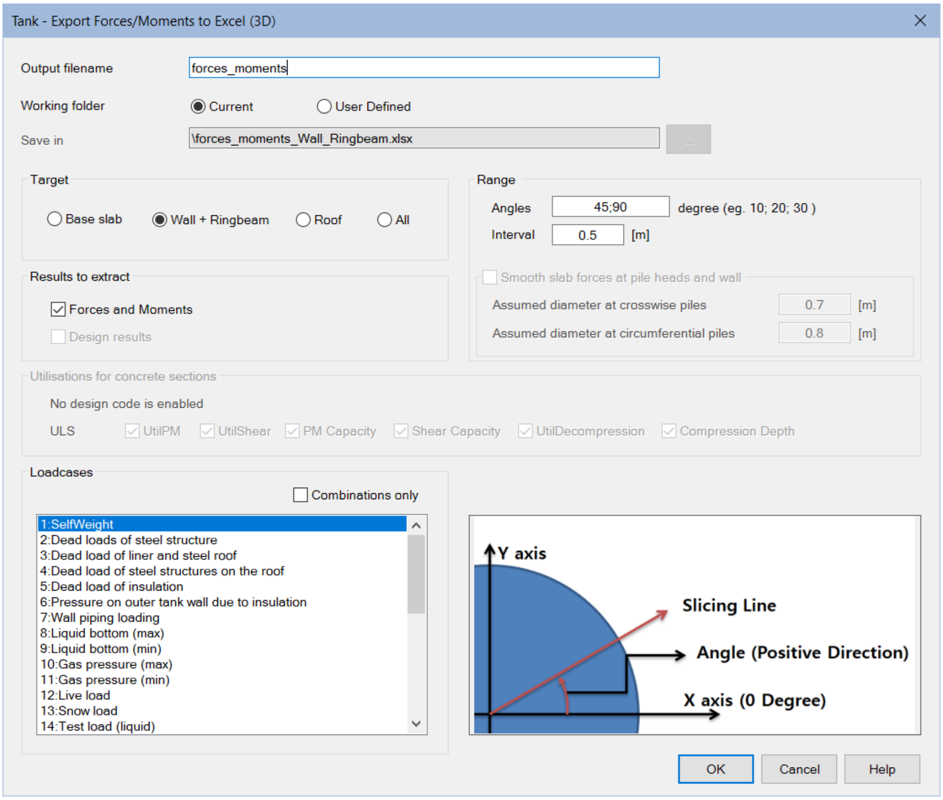
In this way, other ‘arbitrary slices’ can be taken through the wall manually to investigate other regions.

 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.

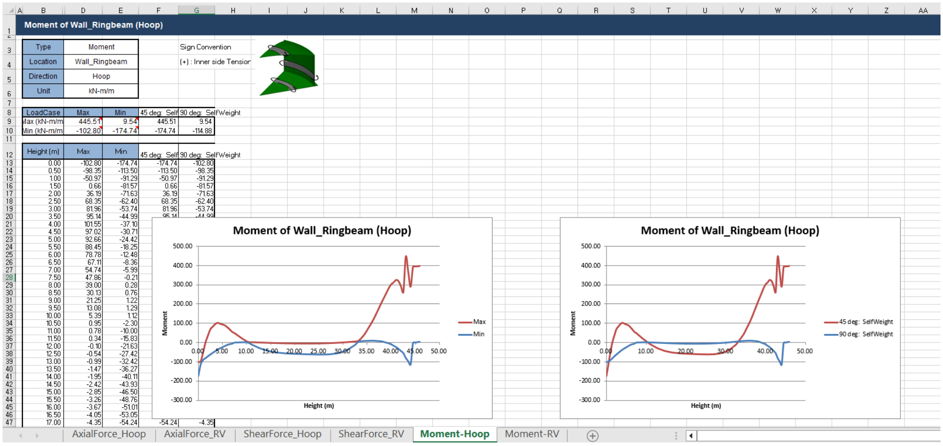
Tank
Excel Tools
Export Forces/Design Results



- 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 **Self Weight** is selected and click **OK**

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

- Outside of LUSAS Modeller, open this spreadsheet.



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:

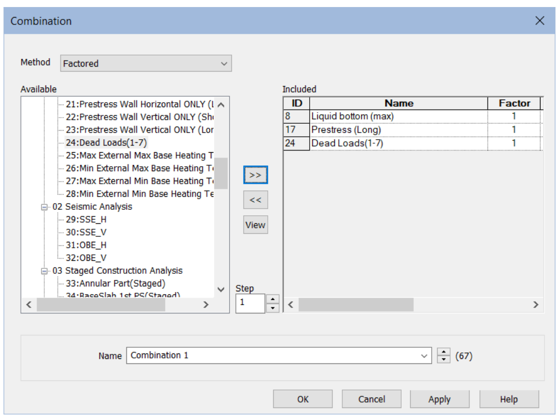
- ❑ Axial Force: (+) for Tension, (-) for Compression.
- ❑ Moment: (+) for Inner side tension, (-) for outer side tension.

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

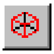
Straightforward combinations may be defined within LUSAS.

Analyses
Basic Combination…

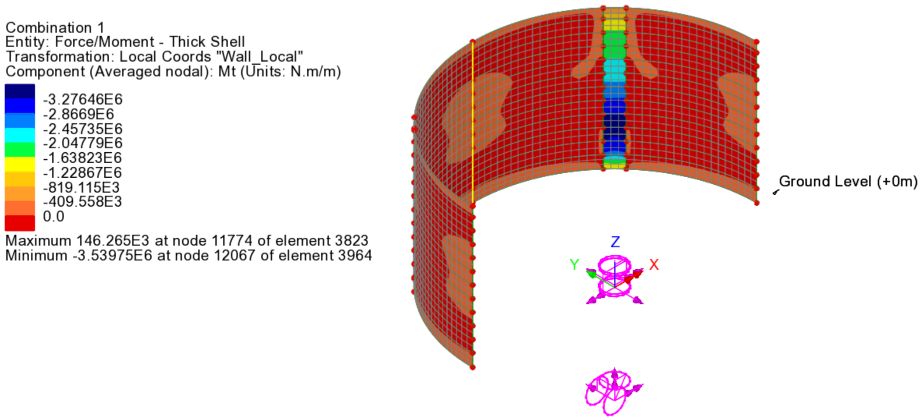


- Include **Liquid bottom (max)**, **Prestress (Long)** and **Dead Loads(1-7)** in the combination, accept the default loadcase name of **Combination 1** and click **OK**.

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

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

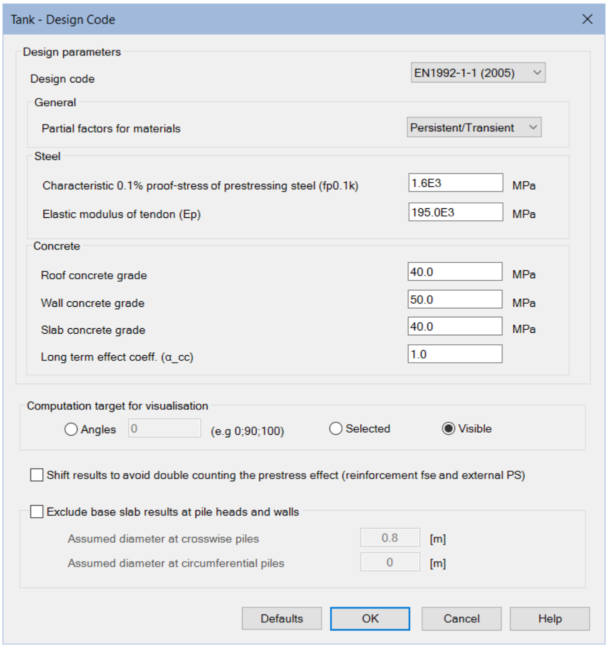
- In the Analyses  Treeview, set active **Combination 1**



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…

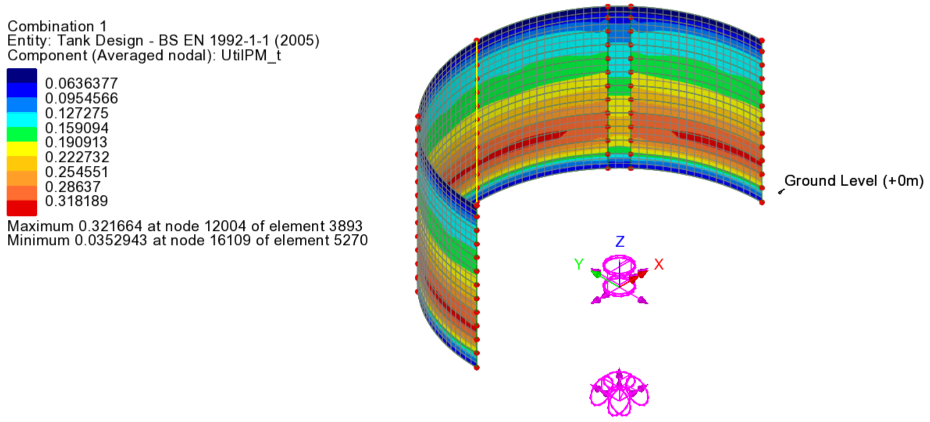


- Ensure design code **EN1992-1-1 (2005)** is selected.
- Ensure Computation target is set to **Visible** (to plot contours on the whole active part of the model) and click **OK**.

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.**

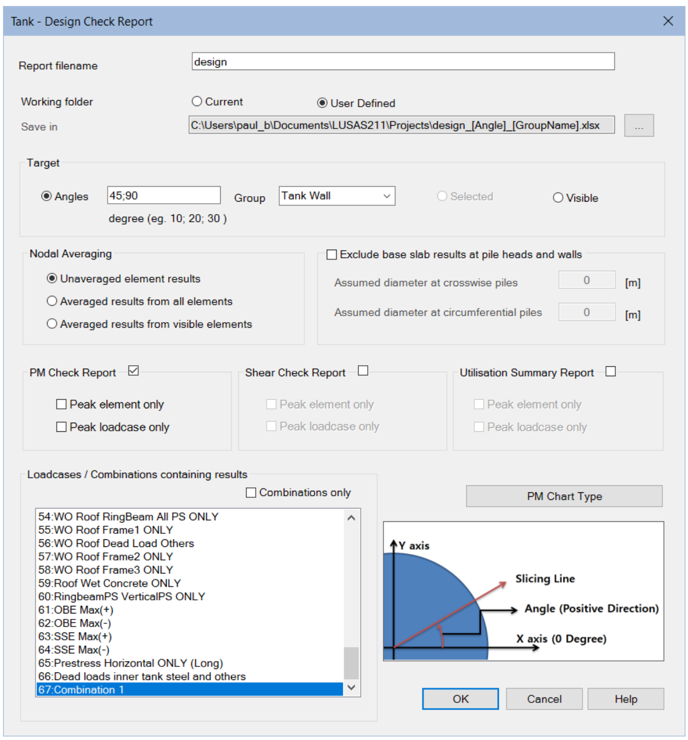
Contours of utilisation will be drawn on the model.



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.

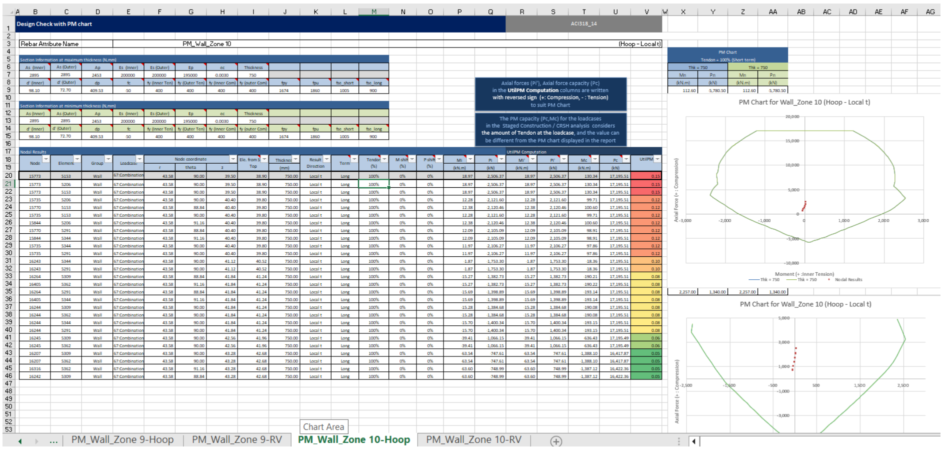
Tank
Design checks
Design Check Report…



- 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**
- 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_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.

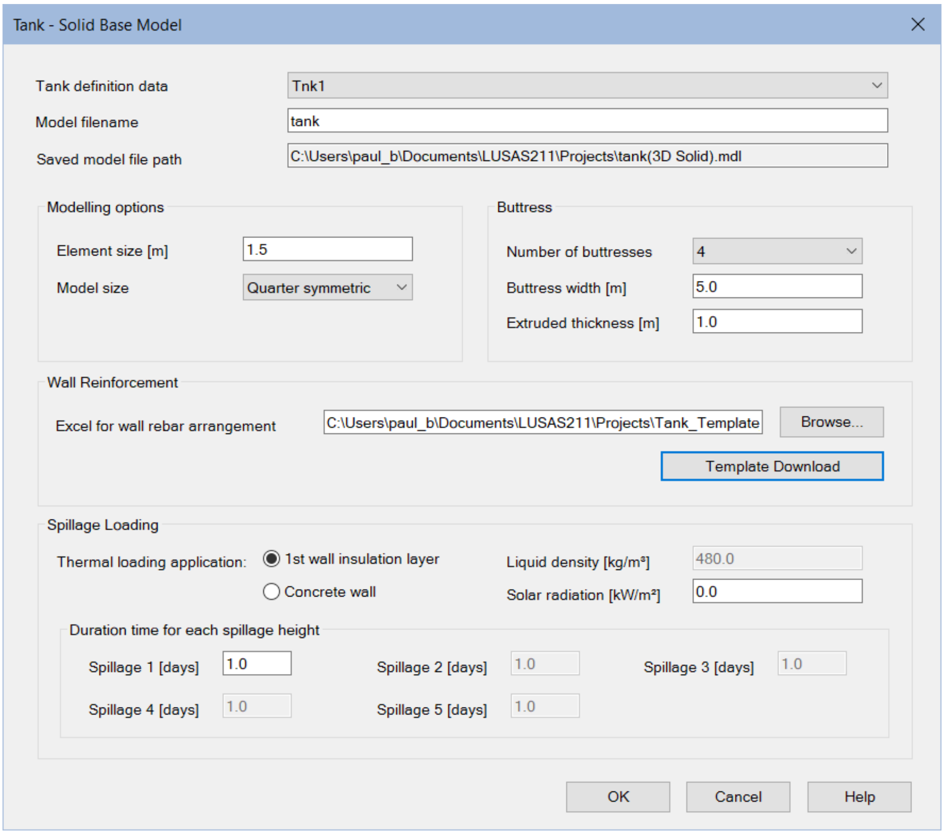
Additional analyses may be added to those present as required:

- **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

To create a 3D solid quarter-symmetric model:

Tank
Create 3D Solid Model



- If necessary, press the **Help** button for more information about this dialog and the type of model that is created.

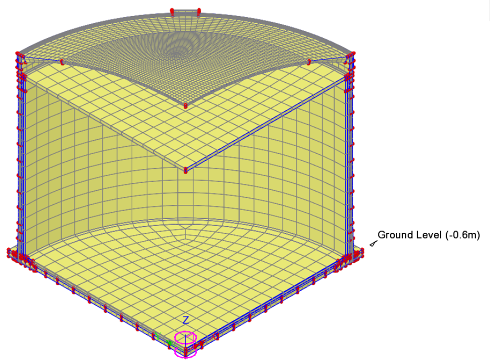
- Enter a model filename of **tank** noting that the filename entered will be appended with ‘(3D Solid)’ to identify this model within the set of tank models that are being created.

- 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.

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.

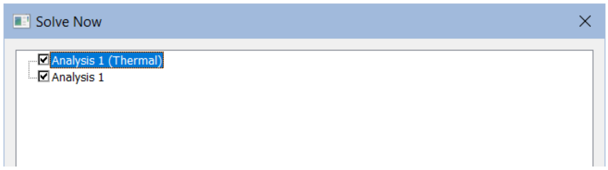
Viewing assigned loadings

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**.



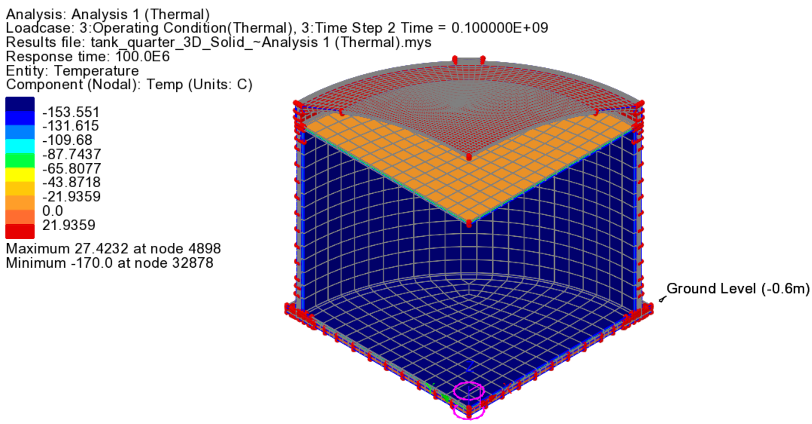
Note. Solving of multiple analyses for a solid model may take an appreciable time.

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.

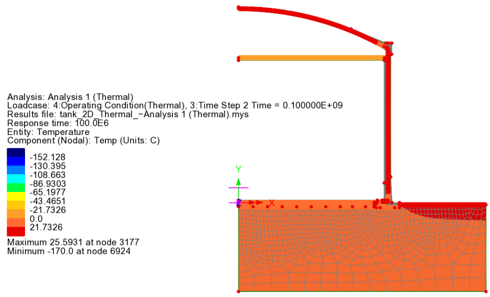
Viewing the results

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

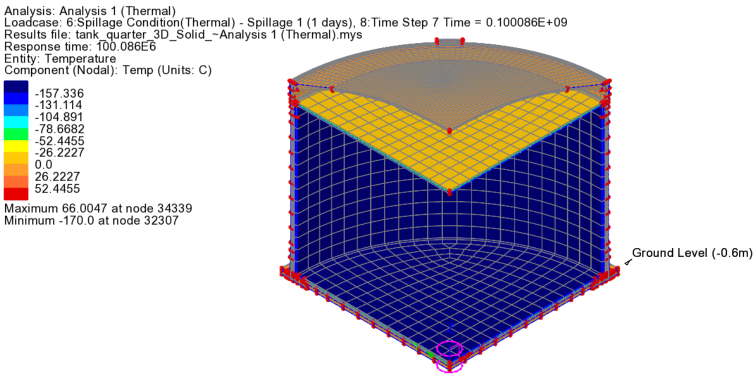
- In the Treeview, set active the loadcase **Operating Condition (Thermal)**
- With no features selected, click the right-hand mouse button in a blank part of the view window and select the **Contours** option to add the contours layer to the Treeview.
- Select entity **Temperature** and component **Temp**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



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).



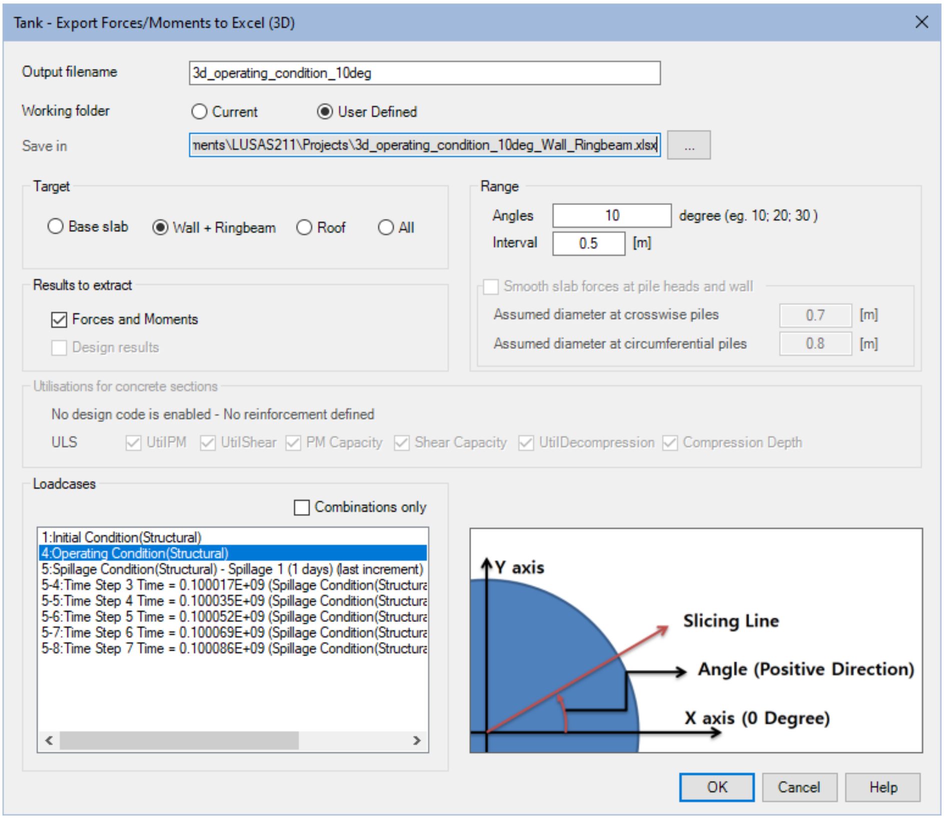
- In the Treeview, set active the loadcase **Spillage Condition (Thermal)**



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

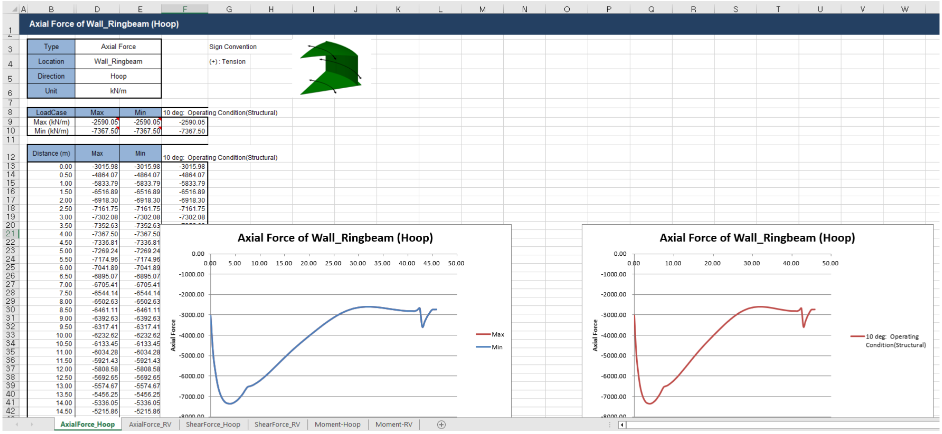


- Enter an output filename of **3d_operating_condition_10deg** noting that a target group name will also be added to the name entered.
- Select a target of **Wall + Ringbeam**
- Enter a slice angle of **10** degrees
- Ensure **Forces and Moments** is selected.
- In the loadcases panel ensure that **Operating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

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.