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:





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 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 Definition					×
Tank type Concrete V Blevation : Above ground tank V Tank Definition Load Prestress Insulation Support Base Sieb and Roof Wall and Ring heam Materials V	Target models to build D oxisymmetric structural D oxisymmetric structural D oxisymmetric seismic ant (30) Seismic Ground Sumont (20)	2D axisymmetric coupled 3D shell structural	thermal/structural	✓ 3D solid 3D shell roof-only	
Base leating (L-heating) Base heating (L-heating) Circular part depth (D_inner) Tapered section length (V_1) Annular part depth (D_outer) Base heating (D_heating) Base heating (L_heating) Ground level (D_ground)	39.8 12 0.6 6.7 1.5 0.386 46.5 0.9	Linner Dheating	Louter theating Wt	G.L. Douter \$ 77 Dground	
Roof (Units: m) Radius of inner roof (R_roof_o) Radius of outer roof (R_roof_o) Height from the top of the base slab to the top most of the roof (R_Height) Distance of tapered section 1 (sl1) Distance of tapered section 2 (sl2) Set zero Set defaults	86.406 86.906 56.2545 10.079 0.6 Troof I Rroof J Rroof J	Rvoot,o	sh sh	SI2 Hingbeam_2 ♣ Rat_height	
Name concrete tank		✓ (new)			
			OK Canc	el Apply He	lp

• 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 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 $\sqrt[4]{2}$ Treeview by double-clicking on the created utility or using the 'Edit' option on the context menu provided.

Utilities			▼ ‡ ×
🔁 Layers 🛛 🗟 Groups 🕹 Attrib.	. 🕒	Analys 🦨 Utilitie	s 🛄 Reports
□- 🔄 tank1.mdl □- 🔄 Utilities (1) □- 🔄 Tank Definition (1)			
imentation 1:concrete_tank		Edit	
	e,	Сору	
	Ē	Paste	
		Rename	
		Delete	



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 <u>new</u> 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**
- □ <u>3D solid coupled thermal/structural</u>

2D axisymmetric structural tank model

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

Tank definition data	concrete tank \checkmark
Model filename	tank
Saved model file path	C:\Users\paul_b\Documents\LUSAS211\Projects\tank(2D).mdl
Concrete element size [m]	0.2
Steel element size [m]	0.2

- 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 3, Attributes \clubsuit and Analyses 3 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.



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.

2D axisymmetric structural tank model



• Reselect the View window.

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 **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.

Utilities Graph through 2D



• Reselect the View window.

Export Forces to Excel (2D)

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

Output filename	2d_axi_structural			
Working folder	Ourrent	O User Defined		
Save in	C:\Users\paul_b\Do	ocuments\LUSAS211\Projects\In	g_tank\2d_axi_struc	tural_Wall_Ringbeam
Target				
◯ Base slab		Wall + Ringbeam		⊖ Roof
Loadcases			Range (Y Coor	d)
1:SelfWeight 2:Dead loads of steel structu	re	^	Start :	0.0
3:Dead load of liner and stee 4:Dead load of steel structure 5:Dead load of insulation	l roof es on the roof		Finish :	45.88
6:Pressure on outer tank wal 7:Wall piping loading	l due to insulation		Interval :	0.5
8:Liquid bottom (max) 9:Liquid bottom (min)				
11:Gas pressure (max) 12:Live load				

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).

all Constructio	n Stages				
Height / Stages					-
Wall stage ID	Height (H)	Stage Y/N			He
1	7	Y	•		+
2	7	Y	-		Hs
3	7	Y	-	Set defaults	
4	7	Y	•	Cordonadito	
5	7	Y	•	Clear grid	Ha
6	7.68	Y	•		+
				Add	Hz
				Remove	+
					I

Tank - Staged Construction Analysi	5		\times
Tank definition data Model filename	Tnk1 tank		
Saved model file path Modelling options Concrete element size [m] Loads to apply Self weight	C:\Users\paul_b\Documents\LUSA	s211\Projects\tank(2D Staged).mdl element size [m] 0.2 iable Loads : Max O Min)	
Concrete Tank Options Roof / Ringbeam Roof construction plan Roof first stage thickness (Initial prestress for ringbea Initial prestress for slab (ra * Roof frame loads are not c * Roof first stage wet concre	Layered roof option 1 ∨ (ratio) 0.5 sm (ratio) 0.5 atio) 0.7 onsidered	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	
	Tank - Staged Construction Analysis Tank definition data Model filename Saved model file path Modelling options Concrete element size [m] Loads to apply ☑ Self weight Concrete Tank Options Roof / Ringbeam Roof forst stage thickness in Initial prestress for ringbea Initial prestress for slab (rational) * Roof first stage wet concreted	Tank - Staged Construction Analysis Tank definition data Tnk 1 Model filename tank Saved model file path C:\Users\paul_b\Documents\LUSA Modelling options Concrete element size [m] 0.2 Loads to apply Structural loadings (Var Concrete Tank Options Structural loadings (Var Concrete Tank Options Structural loadings (Var Concrete Tank Options O.5 Initial prestress for ringbeam Roof forst stage thickness (ratio) 0.5 Initial prestress for slab (ratio) 0.7 * Roof first stage wet concrete is not considered * Roof first stage wet concrete is not considered	Tank - Staged Construction Analysis Tank definition data Tnk1 Model filename tankl Saved model file path C:\Users\paul_b\Documents\LUSAS211\Projects\tank(2D Staged).mdl Modelling options Concrete element size [m] 0.2 Loads to apply Self weight O Structural loadings (Variable Loads :

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 8, Attributes \clubsuit and Analyses 9 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

ncrementation		Solution strategy	
Nonlinear		Same as previous loadcase	
Incrementation	Manual 🗸	Max number of iterations	12
Starting load factor	0.1	Residual force norm	0.1
Max change in load factor	0.0	Incremental displacement norm	1.0
Max total load factor	1.0		Advanced
Adjust load based on c	onvergence	Incremental LUSAS file output	
Iterations per increment	4	Same as previous loadcase	
	Advanced	Output file	1
Time domain		Plot file	1
	Two Phase \sim	Doctort flo	
Initial time step	0.0	Restart file	
Total response time	100.0E6	Max number of saved restarts	0
Automatic time steppin	g	Log file	1
	Advanced	History file	1
		Save a restart at the end of this	control
Common to all			
Max time steps	or increments 0		

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 \bigcirc 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 <u>only the long-term prestress is applied</u>.

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 <u>2D</u> <u>Axisymmetric structural example</u>.

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.

1	Α	В	с	D	E	F	G	н	1.	J	к	L	м	N	0	Р	Q	R	S	т	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									Perma	nent												Var	iable	
3		De	scription		Outer tank	¢	Others	Cree Shrii	p and nkage		P	restress				Roo	f frame/	concrete		Т	est	LN pres	iG sure	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	Roof Frame 2	Roof Frame	1st layer concrete	2nd layer concrete	Hydros tatic	Pneum atic	LNG Max	LNG Min	Gas Max
5	1	U-C1-1		1.35						1.30					-									
6	2	U-C1-2	WO_roof +	1.35						1.00														
7	3	U-C1-3	RB_1st_PS	1.00						1.30														
8	4	U-C1-4		1.00						1.00														
9	5	U-C2-1		1.35						1.30					1.50									
10	6	U-C2-2	WO _root +	1.35						1.00					1.50									
11	7	U-C2-3	Roof frame 1	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:

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)'

ural analyses nalysis 1 - 1:Annular Part(Staged) 	^		Included ID 6 Incre	N ment 6	ame	Factor
ural analyses nalysis 1]- 1:Annular Part(Staged) i	^		ID 6 Incre	N ment 6	ame	Facto
halysis 1 - 1:Annular Part(Staged) - 1:Increment 1 - 2:BaseSlab 1st PS(Staged)			6 Incre	ment 6		
 1:Annular Part(Staged) 1:Increment 1 2:BaseSlab 1st PS(Staged) 			b incre	mont h		
- 1:Increment 1 - 2:BaseSlab 1st PS(Staged)			6 Inorro	mont 5		1
2:BaseSlab_1st PS(Staged)			J Incre	anen 5		
, mensee and a congres,						
2:Increment 2		>>				
3:Circular Part(Staged)						
		<<				
4:BaseSlab 2nd PS(Staged)		View				
- 4:Increment 4						
- 5:Wall Ringbeam(Staged)						
- 5:Increment 5						
6:Ringbeam 1st PS(Staged)						
- 6:Increment 6	~	Step				
	>	1 📮	<			
	3:Circular Part(Staged) 3:Circular Part(Staged) 4:BaseSlab 2nd PS(Staged) 4:BaseRised Staged) 4:Increment 4 5:Wall Ringbeam(Staged) 5:Increment 5 6:Ringbeam ist PS(Staged) 6:Increment 6	- 3:Graular Part(Staged) - 3:Increment 3 - 4:BaseSlab 2nd PS(Staged) - 4:Increment 4 - 5:Ivall Ringbeam(Staged) - 5:Silorcement 5 - 6:Ringbeam Ist PS(Staged) - 6:Increment 6	3:Circular Part(Staged) 3:Circular Part(Staged) 4:BaseSlab 2nd P(S(staged)) 4:BaseSlab 2nd P(S(staged)) 4:BaseM(Staged) 5:Sival Ringbeam(Staged) 5:Cincrement 5 6:Ringbeam 1st PS(Staged) 5:Lep 1	3:Circular Part(Staged) <	3:Cfrular Part(Staged) → 3:Increment 3 4:BaseSlab 2nd P(S(staged)) +4:Increment 4 +5:Wall Ringbeam(Staged) → 5:Wall Ringbeam(Staged) → 5:Increment 5 -6:Increment 6 > Step 1 → <	3:Circular Part(Staged) -3:Increment 3 4:BaseSlab 2nd P(S(staged) -4:Increment 4 -5:Wall Ringbeam(Staged) -5:Increment 5 -6:Increment 6 > Step 1 - 4: - 4: - 4: - 5:Wall Ringbeam(Staged) - 5:Increment 5 - 6:Increment 6 - 5: - 6: - 7: - 7:

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.

Analyses Basic Combination

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	Tank - Coupled Thermal/Structural Analys	is			×
Create 2D Model Coupled Thermal/Structural	Tank definition data Model filename Saved model file path	concrete tank tank C:\Users\paul_b\Document	s\LUSAS211\Projects\tank(2[D Thermal).mdl	×
	Modelling options Concrete element size [m] 0.2	Steel element s	ize [m] 4.0E-3	✓ Include soil (above)	ground tanks)
	Include Structural Load Variable Loads to apply (*) The chosen variable loads from (*) These parameters are read from Spillage Loading	Max Min the Tank Definition will be used n the [Structural Loading Definit	for Operating Condition. tion] tab of the tank definition	utility.	
	Thermal loading application: Co	wall insulation layer	Liquid density (**) Radius of inner tank outer se	480 42.1361	[kg/m³] [m]
	(**) These parameters are read from Spillage duration time for each spill Spillage 1 10.0 [ho Spillage 4 10.0 [ho	the [Seismic] > [Inner Tank Pro lage height urs] Spillage 2 urs] Spillage 5	10.0 [hours]	tion utility if available. Spillage 3 10.0	[hours]
				OK Cancel	Help

- 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 3, Attributes \clubsuit and Analyses 2 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.

Analysis 1 (mermal)	
ny valid existing results files will be loaded, e	even if not re-solved above
ny valid existing results files will be loaded, e Save options	even if not re-solved above
ny valid existing results files will be loaded, e Save options ③ Save model	even if not re-solved above Solver options
ny valid existing results files will be loaded, e Save options ③ Save model ○ Save model with a different name	even if not re-solved above Solver options

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 Dreeview, 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:
- Enter an output filename of 2d_operating_condition

	2d operating con	dition	
Output filename			
Working folder	O Current	User Defined	
Save in	C:\Users\paul_b\D	ocuments\LUSAS211\Projects\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.

Tank
Excel Tools
Export
Temperature

A	В	С	D	E	F	G	Н	
1	Temperature	of Roof						
4								
3	Туре	Tempe	rature					
4	Location	Ro	of					
5	Unit	Cels	sius					
6								
7	No of Slices	Gap (m)	Outer Radius (m)	Rad. Origin Y (m)	Min. Thickness (m)			
8	88.00	0.50	86.91	-30.65	0.50			
9								
10	Distance (m)	Inner Temp (°C)	Outer Temp (°C)	Thickness (m)	Average Temp (°C)	Linear Gradient (°C/m)	Vertical Distance (m)	
11	0.00	23.70	25.33	0.50	24.49	-3.58	0.00	
12	0.50	23.61	25.33	0.50	24.47	-3.71	0.00	
13	1.00	23.62	25.33	0.50	24.47	-3.69	0.01	
14	1.50	23.61	25.32	0.50	24.47	-3.71	0.01	
15	2.00	23.61	25.33	0.50	24.47	-3.70	0.02	
16	2.50	23.61	25.33	0.50	24.47	-3.71	0.04	
17	3.00	23.61	25.32	0.50	24.47	-3.71	0.05	
18	3.50	23.61	25.32	0.50	24.47	-3.71	0.07	
19	4.00	23.61	25.32	0.50	24.47	-3.71	0.09	
20	4.50	23.60	25.32	0.50	24.46	-3.72	0.12	
21	5.00	23.61	25.32	0.50	24.47	-3.70	0.14	
22	5.50	23.60	25.32	0.50	24.46	-3.73	0.17	
23	6.00	23.62	25.32	0.50	24.47	-3.70	0.21	
24	6.51	23.60	25.32	0.50	24.46	-3.73	0.24	
25	7.01	23.61	25.32	0.50	24.46	-3.71	0.28	
26	7.51	23.60	25.32	0.50	24.46	-3.73	0.33	
	Roof	Wall_RingBeam	BaseSlab	(+)	24.46	373	0.37	

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.

ank definition data		concrete tank			\sim		
lodel filename		tank					
aved model file path	n	C:\Users\paul_	b\Docu	ments\LUSAS211\Projects\tar	nk(Seismic EN	1998 Horizontal-EBP).mdl
nalysis type							
esign code		EN 1998-4	N	/			
Beam-stick hor (Excluding base)	rizontal pressure)			 Beam-stick horizontal (Including base pressure) 		O Beam-stick vert	ical
(Beam-Stick model EN1998-4:2006 A3	l is created 3.2.2 Simpl	according to ified procedure fo	or fixed	base cylindrical tanks)			
ritical damping / free	quency						
	Critical	damping (%)		Frequency (1st mode, Hz)	Fred	uency (2nd mode, H	z)
Base slab	4.0			1.25	5.4	4	
Roof	4.0						
Wall	2.0						
Inner tank	2.0						
Foundation	4.0						
Liquid impulsive	3.0						
Liquid convective	0.5						
Convert	5.0						
Ground	0.0						
uttress							
Number of buttress	es 4		~				,
Extruded thickness	1	.0		[m] Buttress width	5.0		[m]

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.



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 3, Attributes \clubsuit and Analyses 2 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 W	/izard	<
	Results type	○ Components	
	Results Units Loadcases I Eigenvali I Participa I Mass par I Sum ma	Model units 1:Loadcase 1 lues (Frequency) ation factors articipation factors ass 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 ×								
8	9 🖪 🖪 (d 🔍 🔍							
	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 16	0.0	0.0	0.0	0 103701⊏ 3	61 1/17	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.

Tank - Seismic Analysis		×
Tank definition data Model filename Saved model file path Analysis type	Trik1 C'Usarsiyaul_b/Documents/LUSAS211Projects/(Seismic EN1988 Horizontal-EBP) mdl	
Design code Beam-stick horizontal (Excluding base pressure) (Beam-Stick model is created EN1998-4:2006 A3.2.2 Simpli	EN 1998-4 Beam-stick horizontal (Including base pressure) Beam-stick vertical according to ed procedure for fixed base cylindrical tanks)	
Critical damping / frequency		
Critical c Base slab 4.0 Roof 4.0	amping (%) Frequency (1st mode, Hz) Frequency (2nd mode, Hz) 1.25 5.44	

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 D 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: <<u>Current working folder</u>> <u>LUSASFiles32</u> 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	0.239496E-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	11 1136	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 \checkmark Treeview.

Utilities					▼ ₽ ×
Layers	Groups	🛃 Attrib	Analys	💤 Utilities	Reports
	(Seismic EN19 Jtilities (7) Variation (2 Juine (2) Juine	98 Horizontal) I_kh cous_coefficie	-EBP).mdl		
	Response S	pectra (1) A-CLASS wizard (1)			
	Tank Definit	ion (1)			



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 C 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.

IMD Loadcase Excitati	on Support Motio	n 🗸 Set	×	
Results	Spectral	~ Set	Spectral Response	×
Damping Type LUSAS	Svalues V	Modes Use all modes Select modes	Type of spectral response SRSS combination Damping variation correction to response spectrum Eurocode Response 1:ASCE A-CLASS	>
Name (Response Spectru Cancel	m SRSS v (3)	OK Cancel Help	

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	

Tank definition data Tnk1	~
Model filename tank Saved model file path C:\Users\paul_b\Documents\	LUSAS211\Projects\tank(3D Shell).mdl
Modelling options Element size [m] 2.0 Rig Number of eigenvalues 10 V Inc	alf symmetric model gid fin plate clude non-structural masses in the eigenvalue analysis
Concrete Tank Options Buttress Number of buttresses Extruded thickness Buttress width 5.0 [m] Decif (Dic la	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
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	6 - Ringbeam 2nd PS 7 - Wall vertical PS 8 - Wall horizontal PS

- 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 3, Attributes \clubsuit and Analyses 2 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	Tank - Add/Update Rebars & Tendons				×
Add/Update Rebars and	Detailed Reinforcement arrangement	s\paul_b\Documents\LUSAS211\Projects	\Tank_Template_F	Rebars_Tendons	Browse
lendons	Existing reinforcement info will be replaced.			Template D)ownload
	- The Tank Design (PM, Shear, Decompression) will be up	dated.			
	- Prestress loads will be replaced with Tendon loads and c	ustom Tendon Profiles.			
	- The RC Slab Design will be still based on the assigned re	inforcement attributes.	ОК	Cancel	Help

- 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:



For simplicity not all analyses will be run for this example.

Solve Now	×
✓ 01 Base Analysis ✓ 02 Seismic Analysis ── 02 Seismic Analysis ── 03 Staged Construction Analysis ✓ 05 Eigenvalue Analysis	
Any valid existing results files will be loaded,	even if not re-solved above
Save options	Solver options
 Save model 	 Fastest available
 Save model with a different name 	 Frontal Solver (slower with more error diagnostics)
O Do not save	First loadcase only
	OK Cancel Help

• On the Solve Now dialog <u>deselect</u> the Seismic Analysis and Staged Construction Analysis check boxes, then click OK.

Analysis loadcase results are added to the \bigcirc 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 D 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 streeview. 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**.

Properties	×	Results Transformation	>
Contour Results Appearance Entity Force/Moment Component ME Display Averaged nodal Transform Set None		No transformation applied (consult Solve Local axes of element/node Local axes of parent feature Global axes Material Assigned results transformation attribute Specified local coordinate 4:Wall_Local	r manual) e
Display on slice(s) Draw in slice local direction		Shell plane for resultants theta/z Reference path No objects defined x = longitudinal	y = transverse
OK Cancel Apply	Help	OK Cancel	Help

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, rightclick 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 in 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	Tank - Export Forces/Mo	ments to Excel (3D)		\times
Excel Tools	Output filonemo	farcas momente		
Forces/Design Results	Working folder	Current Ouser Defined		
	Save in	\forces_moments_Wall_Ringbeam.xlsx		
	Target O Base slab	Wall + Ringbeam O Roof O All	Range 45:90 degree (eg. 10; 20; 30) Interval 0.5 [m]	
	Results to extract	ments	Smooth slab forces at pile heads and wall Assumed diameter at crosswise piles 0.7 Assumed diameter at circumferential piles 0.8	
	Utilisations for concret No design code is ULS Uti	e sections enabled IPM ↓ UtilShear ♥ PM Capacity ♥ Shea	r Capacity 🔽 UtilDecompression 📝 Compression Depth	
	Loadcases 2:Dead loads of steel 3:Dead load of liner: 4:Dead load of steel 5:Dead lo	Combinations only	Y axis Slicing Line Angle (Positive Direction) X axis (0 Degree)	

- 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.

oment of	Wall_Ringbeam (Hoc	p)														
Type	Noment	Size Convection														
ocation	Well Rinnheam	(a) : Incer side Tensio														
irection	Hoop	(-) . We side rensid														
Unit	kN-m/m															
			_													
adCase (kN-m/m	Max Min 45 d 445.51 9.54	eg: Self 90 deg: SelfWeight 445.51 9.54														
(kN-m/m	-102.80 -174.74	-174.74 -114.88														
ight (m)	Max Min 45 d	er Self 90 den Selfu/einht														
0.00	-102.80 -174.74 .98.75 .113.50	-174.74 -102.80 113.50 -98.35														
100	-50.97 -91.29 0.65 -81.57	-50.97 -91.29														
2.00	36.19 -71.63	36.19 -71.63														
3.00	81.95 -53.74 95.14 -44.99	81.96 -53.74 96.14														
4.00	101.55 -37.10 97.02 -30.71	M	oment of Wall	Ringbe	am (Ho	(qc				м	loment	of Wal	I Ringb	eam (Ho	(qc	
5.00 5.50	92.66 -24.42 88.45 -18.25	500.00	-	-	•		-		500.00							
6.00	78.78 -12.48 67.11 -8.36	400.00				1			400.00							
7.00	54.74 -5.99 47.86 -0.21	4.0.00				.N			100.00					_N		
8.00	39.00 0.29 30.13 0.76	300.00			- 1	٧.			300.00					/\'		
9.00	2125 122 13.08 129	200.00			_/				200.00							
10.00	5.39 1.12 0.95 -2.30	E 100.00			/		Max	umen	100.00	\sim			/		45 deg:	SelfWeight
11.50	0.76 -10.00	2 000				~	Min	2	0.00					~ ~	90 deg:	SelfWeight
12.50	-0.10 -2163 -0.54 -27.42	0.00 500 10	00 15-00 20:00 25:0	30.00	35.00 40.0	45.00 50.00			0.00	10.0	0 20	00 3	0.00	40.00 50	00	
13.50	-0.35 -32.42 -147 -36.27 195 - 40.11	-100.00				v			- 100.00					V		
14.50	-2.42 -43.93	-200.00							-200.00							
15.50	-3.26 -48.76	-300.00	Height	(m)					-300.00			Height (m)				
16.50	-4.05 -53.05	54.741 .4.351										2				
- H.00 L	-4.54 -54.24	-Ver 1 - 1.30														





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	Combination	×
Basic Combination	Method Factored ~	
	Available Included	
	- 21:Prestress Wall Horizontal ONLY (I A ID Name	Factor
	22:Prestress Wall Vertical ONLY (Shi 8 Liquid bottom (max)	1
	- 23:Prestress Wall Vertical ONLY (Lor 17 Prestress (Long)	1
	24:Dead Loads(1-7) 24 Dead Loads(1-7)	1
	2.5 Max Exernal Mins Base Heading T 2.5 Max Exernal Mins Base Heading T 2.7 Max External Mins Base Heading T 2.8 Sistem Analysis 2.2 Seismic Analysis 2.2 Seismic Analysis 2.2 Seismic Analysis 3.5 SE, V 3.0 SSE, V 3.0 SS	
	$\langle \rangle$ γ $1 + \langle \rangle$	>
	Name Combination 1 V (6)	') Help

• 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**

lesign parameters	
Design code	EN1992-1-1 (2005) ~
General	
Partial factors for materials	Persistent/Transient \vee
Steel	
Characteristic 0.1% proof-stress of prestressing steel (fp0.1k)	1.6E3 MPa
Electic modulus of tendon (En)	195.0E3 MPa
Easile modulus of tendori (Ep)	NPa MPa
Concrete	
Roof concrete grade	40.0 MPa
Wall concrete grade	50.0 MPa
Slab concrete grade	40.0 MPa
Long term effect coeff. (a_cc)	1.0
	lastad 🔍 Visibla
(a.g 0;90;100)	lected Visible
Shift results to avoid double counting the prestress effect (rein	nforcement 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]

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.

Tank



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.

ank	Tank - Design Check Report		×
Design checks	Report filename	design	
Report	Working folder	O Current User Defined	
	Save in	C:\Users\paul_b\Documents\LUSAS211\Projects\d	esign_[Angle]_[GroupName].xlsx
	Target	Group Tank Wall ~	Selected O Visible
	Nodal Averaging	Exclude base slab r	esults at pile heads and walls
	Unaveraged element res Averaged results from all	Assumed diameter a	t crosswise piles 0 [m]
	O Averaged results from vi	sible elements	t circumferential piles 0 [m]
	PM Check Report	Shear Check Report	Utilisation Summary Report
	Peak element only	Peak element only	Peak element only
	Peak loadcase only	Peak loadcase only	Peak loadcase only
	Loadcases / Combinations cont	aining results	
		Combinations only	PM Chart Type
	54.WO Roof Fingleem AII's 55.WO Roof Fremat ONLY 56.WO Roof Fremat ONLY 58.WO Roof Fremat ONLY 58.Roof Roof Fremat ONLY 60.RingbeemPS VerticeIPS C 61.0BE Max(+) 62.OBE Max(+) 63.SSE Max(+) 64.SSE Max(+) 65.Prestress Horizontal ONL' 66.Dead loads inner tank stee 67.Combination 1	Y (Long) el and others	Slicing Line Angle (Positive Direction) X axis (0 Degree) 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
- 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

Tank definition data	Tnk1		
Model filename	tank		
Saved model file path	C:\Users\paul_b\Doc	uments\LUSAS211\Projects\tank(3D	Solid).mdl
Modelling options		Buttress	
Element size [m]	1.5	Number of buttresses	4 ~
Model size	Quarter symmetric V	Buttress width [m]	5.0
		Extruded thickness [m]	1.0
Wall Reinforcement	rrangement C:\Users\paul_t	o\Documents\LUSAS211\Projects\Tar	nk_Template Browse Template Download
Wall Reinforcement Excel for wall rebar a Spillage Loading	rrangement C:\Users\paul_t	b\Documents\LUSAS211\Projects\Ta	nk_Template Browse Template Download
Wall Reinforcement Excel for wall rebar a Spillage Loading Thermal loading app	rrangement C:\Users\paul_t	o\Documents\LUSAS211\Projects\Tar	Template Browse Template Download
Wall Reinforcement Excel for wall rebar a Spillage Loading Thermal loading app	rrangement <u>C:\Users\paul_t</u> ication: 1st wall insulation laye Concrete wall 	o\Documents\LUSAS211\Projects\Tar Pr Liquid density [kg/m³] Solar radiation [kW/m²]	Image: mk_Template Browse Template Download 480.0 0.0
Wall Reinforcement Excel for wall rebar a Spillage Loading Thermal loading app Duration time for ea	rrangement C:\Users\paul_t ication: The spillage height	o\Documents\LUSAS211\Projects\Tar P P Liquid density [kg/m³] Solar radiation [kW/m²]	Image: mk_Template Browse Template Download 480.0 0.0
Wall Reinforcement Excel for wall rebar a Spillage Loading Thermal loading app Duration time for ea Spillage 1 [days]	rrangement <u>C:\Users\paul_t</u> ication: 1st wall insulation laye Concrete wall ach spillage height 1.0 Spillage	er Liquid density [kg/m³] Solar radiation [kW/m²]	Image 3 [days] Image 3 [days]

• 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.



Dr

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 8, Attributes \clubsuit and Analyses 9 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.

Solve Now	\times
전 <mark>(Analysis 1 (Thermal))</mark> 전 Analysis 1	



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

On completion, analysis loadcase results are added to the \bigcirc 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 Dreeview, 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						

Tank - Export Forces/Mom	ents to Excel (3D)		×
Output filename	3d_operating_condition_10deg		
Working folder	O Current		
Save in	nents\LUSAS211\Projects\3d_operating_co	ndition_10deg_Wall_Ringbeam_xlsx	
Target O Base slab	all + Ringbeam 🔿 Roof 🔿 All	Range Angles 10 degree (eg. 10; 20; 30) Interval 0.5 [m]	
Results to extract Forces and Momen Design results Utilisations for concrete see No design code is ena	ts tions bled - No reinforcement defined	Smooth slab forces at pile heads and wall Assumed diameter at crosswise piles Assumed diameter at circumferential piles 0.8 [m]	
Loadcases 1-Initial Condition(Structur 4-Operating Condition(Structur 5-Spilage Condition(Structur 5-A:Time Step 2 Time = 0 5-7:Time Step 4 Time = 0 5-8:Time Step 7 Time = 0	Combinations only intral intrel intral intrel intrel intrel intrel intre	Slicing Line Angle (Positive Direction X axis (0 Degree) OK Cancel Help	n)

- 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.

-4 A	В	D	E	F	G	н	1	J	K	L	м	N	0	P	Q	B	S	т	U	V.	w
1	Axial Forc	e of Wall_Rin	igbeam (H	loop)																	
3	Type	Axial F	force	1	Sign Convention																
4	Location	Wal_Rin	igbeam		(+):Tension			~													
5	Direction	Hor	Ноор				_														
6	Unit	kN/m						-													
8	LoadCase	Max	Min	10 deg: Opera	ing Condition)	Structural)															
10	Min (kN/m)	-7367.50	-7367.50	-7367.50																	
	Distance (m)	Mar	Min		_																
12				10 deg: Operal	ing Condition	Structural)															
13	0.0	-3015.98	-3015.98	-3015.98																	
14	0.0	-4804.07	-4864.07	-4864.07																	
16	1.0	0033.79	-0033.71	-0033.79																	
17	2.0	-0010.03	-6010.01	-0010.09																	
10	2.0	716176	7101.7	710176																	
10	2.0	7202.09	-7302.05	-7302.02																	
20	3.5	7352.63	-7352.63																		
211	4.0	-7367.50	-7367.56				e										· · · ·		<i></i>		
22	4.5	.7335.81	.7335.8		A	kiai Ford	e ot wa	iii_king	beam (F	100p)					A	xiai Forc	e or wa	II_KINGD6	eam (Ho	op)	
23	5.0	-7269.24	-7269.24																		
24	5.5	7174.96	-7174.95	0.00										0.00	I						
25	6.0	-7041.89	-7041.8		1.00 5.00	10.00 15.0	0 20.00 :	25.00 30.0	0 35.00 40	45.00	00.00			0	.00 3	0.00 20	100 30	40.0	0 50.00		
26	6.5	-6895.07	-6895.07	-1000.00	+									-1000.00							
27	7.0	-6705.41	-6705.41																		
28	7.5	0 -6544.14	-6644.14	-2000.00										-2000.00							
29	8.0	-6502.63	-6502.63																		
30	8.5	-6461.11	-6461.11	1000.00						-10				1000.00					10		
31	9.0	-6392.63	-6392.63	8 3000.00				/		V				8					V		
32	9.5	-6317.41	-6317.41	ă										No.			/				
33	10.0	-6232.62	-6232.63	-4000.00			/				Ma:	٠ ــــــــــــــــــــــــــــــــــــ		₩ -4000.00	1		/				: Operating
34	10.5	-6133.45	-6133.45	3	1		/				Min			3						Condi	:ion(Structural)
30	11.0	-6034.28	-0034.28	-5000.00	1		<i>,</i>							-5000.00	1						
30	11.5	5921.43	-5921.43		1										1						
20	12.0	-6808.58	-5808.58	-6000.00	1									-5000.00	1	/					
30	12.5	-3692.65	-5692.65	3000.00	1	/						-		5000.00	1	/					
30	13.0	-0574.67	-0574.67		11 /	-									11 1						
41	13.0	6000.20	-0400.20 6008.08	-7000.00										-7000.00							
40	14.0	-3330.00	-335.05												-						
42	14.0	-0210.80	-0210.80	.8000.00	1						_			.8000.00							
	> Ax	ialForce_Hoop	AxialForce	E_RV ShearF	orce_Hoop	ShearForce	RV Mo	ment-Hoop	Moment-I	RV 🕀				1							

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.