

LNG Tank System User Manual

Concrete Tank - Part 2 - Design Checks

LNG Tank System

User Manual (Concrete Tank) Part 2 - Design Checks

LUSAS Version 20.0 : Issue 1

LUSAS

Forge House, 66 High Street, Kingston upon Thames, Surrey, KT1 1HN, United Kingdom

Tel: +44 (0)20 8541 1999

Fax +44 (0)20 8549 9399

Email: info@lusas.com

http://www.lusas.com

Distributors Worldwide

Copyright ©1982-2023 LUSAS All Rights Reserved.

Table of Contents

LN	G Tank Design Checks	1
Ba	se Model	3
	Preparations	3
	Build Base Model	7
	Update Reinforcement and Prestress	. 28
Th	ermal Analysis	29
	Thermal Analysis for Max Environmental Temperature	. 29
	Thermal Analysis for Min Environmental Temperature	. 31
	Update Base Model	. 34
Sei	ismic Analysis	35
	Preparation	. 35
	Update Base Model	. 35
	User Updates	64
	User Updates	. 65
De	sign Load Combinations	69
	Template for Design Load Combinations	69
	Update Base Model	. 70
De	sign Check	73
	COP Parameters	. 73
	Design Checks for Tank	. 77
	PM Chart Report	. 82
	Design Checks for RC Slab	. 84

LNG Tank Design Checks

The LNG Tank System carries out design checks to supported design codes for specified load combinations.

Its use requires the **MicroSoft Excel** spreadsheet application to be installed in advance for full functionality as certain applications may use it during the design or reporting process. For example, the Wizard imports the spreadsheet template for design load combinations.

Due to the low temperature of LNG, a thermal analysis requiring solid elements has to be performed and this often causes difficulties for load combinations that are based on structural results when the structural analysis is performed using shell elements. The LNG Tank System imports the temperatures and temperature gradients that are obtained from thermal analysis using solid elements into shell elements and allows all required load combinations to be assembled in a single model, to enable design check results to be obtained in an efficient way.

This manual focuses on the procedures involved in performing design checks using the LNG Tank System. A separate manual titled 'LNG Tank System: Part 1 - Tank Wizard' provides details of modelling concepts used to build the range of models supported.

User inputs shown in the manual 'LNG Tank System: Part 1 – Tank Wizard' are used in this manual unless otherwise stated.

Base Model

Preparations

User Inputs

Select the menu item LNG Tank >Tank Definition...

The required user inputs for this model are as marked in [Fig1]

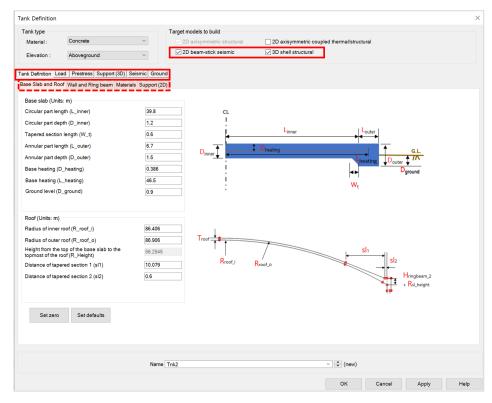


Fig 1 User Input for a Base model for design check

Base Model

ank Definition			
Tank type Material : Concrete ~ Elevation : Aboveground ~	Target models to build	2D axisymmetric coupled thermal/struc	tural
ank Definition Load Prestress Support (3D) Seismic Groun Inner Tank Properties Non-Structural Masses Lumped Foundat			
Roof Ring Beam Wall Base Slab Inner Steel Tank Descriptions	Mas	- [(]	
Roof Liner + steel Roof Structure	1.4E		
Suspended deck & insulation of the suspended ceiling	135.0	E3	
Roof nozzles	42.08	3	
Roof platform	400.0		
Roof pump & crane	30.00		
Roof piping and support Others	103.0	E3	
Total	2.116	-	
Set zero Set defaults			
Name	Tnk2	 ✓ * (new) 	
		ОК	Cancel Apply Help

Fig 2 User Inputs for a Base model for design check including non-structural masses to Eigenvalue Analysis

To include non-structural masses to eigenvalue analysis, 2D beam-stick seismic option should be ticked and non-structural masses should be defined.

k Definition				
ink type	Target models to build			
Material : Concrete ~	2D axisymmetric structura	2D axisymmetric coupled t	hermal/structural	
Elevation : Aboveground ~	✓ 2D beam-stick seismic	✓ 3D shell structural		
k Definition k Definition Salba and Roof Wall and Ring beam Materials Support Vall and ring beam (Units: m) mide radius of concrete outer tank wall (InsR) Thickness of wall base (T_bottom) teight of tapered wall (H_wall_t) Thickness of tapered wall (H_wall_t) teight of tapered wall (H_wall_t) teight of ringbeam_2 (H_ringbeam_2) teight of ringbeam (T_ringbeam) Hore height (R_st_height)	rt (2D) 43.2 1.1 7.4 0.75		Set defaults Clear grid Add Remove	Ha
		_		
			OK Cancel	Apply Help
Set zero Set defaults Wall	stages Openings			
Vali	openniga			
N	ame Tnk2	~	(new)	

Fig 3 User Inputs for a Base model for design check including wall stages to Staged construction/ CRSH Analysis

To include wall stages to Staged construction/ CRSH analysis, Wall construction stages should be defined.

Following user inputs should be defined.

- □ Height(H) [m] : inputs the height of each wall lot. This value should be a positive.
- □ Stage Y/N : input 'Y' if a separate stage should be created for the wall lot in the model. Otherwise input 'N'. However, if the input value is 'N' for wall ID "1", it is assumed that the wall lot 1 is activated together with the base annular part as shown in the Fig 4 The stage of activating wall lot 1 when 'N' for 'Staged Y/N'.



Fig 4 The stage of activating wall lot 1 when 'N' for 'Staged Y/N'

type		I	arget models to b	build				
erial :	Concrete	~	🔽 2D axisymm	etric structural	2D axisymmetric coup	led thermal/structural	I	
vation :	Aboveground	~	☑ 2D beam-stic	ck seismic	☑ 3D shell structural			
	.oad Prestress Support (3D)							
	oof Wall and Ring beam Mate	rials Support (2D)						
					Tringbeam			
	Openings							×
kness o	Openings							
ht of tap	Description	Opening 1	Opening 2	^				
kness o	Openings width (Wo)	0	0			Wgap		
ht of wa	PS free length (Wgap)	0	0	Op	ening 1		Wg	ap
ht of rin	r o nee lengur (wgap)	Ū	0		Theta	Opening 2		н
ht of rin kness o	Opening elevation (H1)	0	0		Vo	Wo		H2
kness o e height	Opening height (H2)	0	0					
e neigni								H1 +
	PS free height (H)	0	0					
	• • • • • • •			~				
	Set defaults Clear g	rid		1.Th	ata' is the angle between open	ing contex and the ad	liccont huttroop	
					ata is the angle between open	ing center and the ad	gacent buttless	center
					ок	Cancel	Apply	Help
				_				
Set zero	o Set defaults	Wall stages	Openings					
		Name Tr				 (new) (new) 		

Fig 5 User Inputs for a Base model for design check including openings to Staged construction/ CRSH Analysis

To include openings to Staged construction/ CRSH analysis, 'Openings' should be defined.

Complete all the inputs and click **OK** to save the data with the name 'Tnk1'.

A Reinforcement and Prestress tendon arrangement is not required to build the corresponding model.

Build Base Model

The Base Model for a design check can be built by selecting the menu item LNG Tank> Create 3D Shell Model...

Tank definition data Tnk1		×
Model filename Example		
Saved model file path C:\Users\ohsso	o\Documents\LUS	AS200\Projects\Example.mdl
Modeling options		
Element size (m)	Half	symmetric model
	Inclu	ide temporary opening
Number of eigenvalue 10	🗸 Inclu	ide non-structural masses in the eigenvalue analysis
Number of buttress 4 Extruded thickness 1.0 Buttress width 5.0	(m) (m)	1 - Base / Wall / Ringbeam 2 - Ringbeam 1st PS 3 - Roof frame 1/ Inner work 4 - Roof frames 2,3
Roof / Ringbeam		5 - Roof wet / Roof complete 6 - Ringbeam 2nd PS
Roof construction plan Single layered r	roof 1 🗸 🗸	7 - Wall vertical PS
Roof first stage thickness (ratio) 0.5	5	8 - Wall horizontal PS
Initial prestress for ringbeam (ratio)	5	
Initial prestress for base slab (ratio)	5	



Roof construction plan

4 options are available which are 'Single layered roof 1', 'Single layered roof 2', 'Layered roof option 1' and 'Layered roof option 2', based on the construction plan for the roof. The description under 'Construction scenario' group box will be updated upon the selection change.

Roof first stage thickness (ratio)

The roof thickness for 1st build will vary according to this value in the staged construction analysis model.

Initial prestress for ringbeam (ratio)

The % of ringbeam PS that will be applied when the construction scenario is 'Ringbeam 1^{st} PS'.

Initial prestress for base slab (ratio)

The % of base slab PS that will be applied when the construction scenario is 'Base slab 1^{st} PS'.

- Enter the model file name, and set the element size to **2.0**. The other values as shown in [Fig 6].
- Enter **10** for **Number of Eigenvalues**. Tick the 'Include non-structural masses' checkbox to include non-structural masses to eigenvalue analysis.
- Select 4 for Number of buttresses, 1 for Extrude thickness and 5 for Buttress width.
- Roof construction plan is set to 'Single layered roof 1'. Initial prestress for ringbeam is set to 0.5 and Initial prestress for base slab is set to 0.5. Initial Prestress amount for Ringbeam and Base slab will also vary at the 'Ringbeam 1st PS (staged)' and 'Baseslab 1st PS (staged)' in staged construction analysis model.

Mesh

The elements as shown below, with a maximum element size less than 2.0m as per user input. Quadratic shell elements (QTS8) are used.

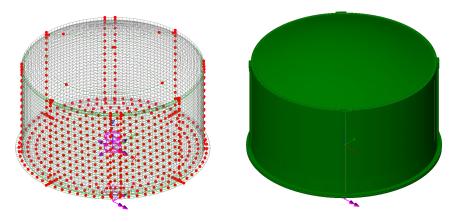


Fig 7 Mesh Arrangement and Geometric Properties for a Base Model for Design Check

The element local axis can be displayed as shown below. The wizard produces elements having a local x axis in the horizontal direction for the Wall and Roof. The element shape in the Slab cannot be regular due to the variable pile arrangement hence the local axis of the elements for the Slab is not consistent.

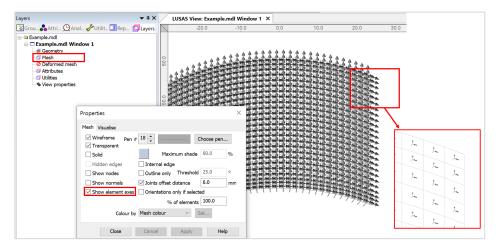


Fig 8 Element Local Axis in a Base model for design check

Geometric Properties

Geometric properties are defined as per user inputs illustrates how properties were defined for varying sections at the edge of the roof. The **variation dataset** can be reviewed from the Utilities **d** treeview.

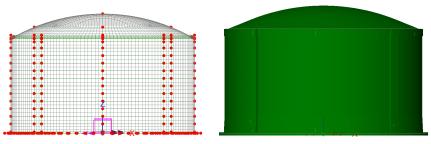


Fig 9 3D Shell Model for Base model for design check

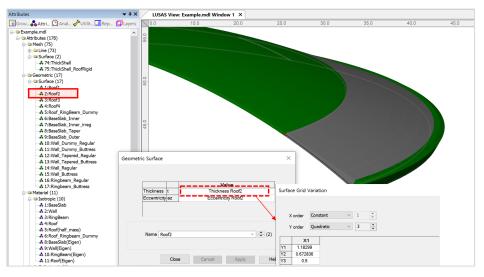


Fig 10 Geometric Properties for Roof in Base model for design check

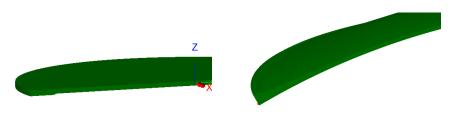


Fig 11 3D Shell Model Thickness Variation at Roof and Slab

Buttresses

Buttresses can be included in the model with separate surfaces accepting separate geometric and material properties. The number of buttresses that can be defined is 0, 2, 3, 4 or 6.

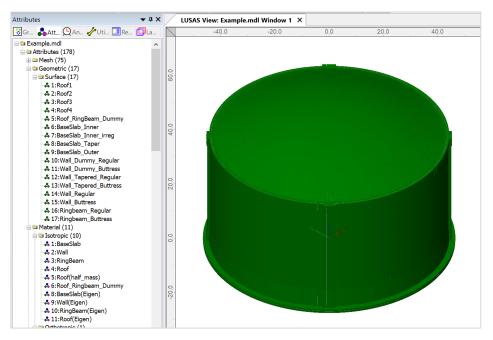


Fig 12 Base Model for Design Check including buttresses

Base Model

ank definition data	Tnk1	~
Model filename	Example	
Saved model file path	C:\Users\ohsso\Docum	ents\LUSAS200\Projects\Example.mdl
Modeling options		
Element size (m)	.0	Half symmetric model
. ,		Include temporary opening
Number of eigenvalue	0	Include non-structural masses in the eigenvalue analysis
Extruded thickness Buttress width	1.0 (m 5.0 (m	3 - Roof frame 1/ Inner work 4 - Roof frames 2,3
Roof / Ringbeam		5 - Roof wet / Roof complete 6 - Ringbeam 2nd PS
Roof construction plan	Single layered roof 1	7 - Wall vertical PS
Roof first stage thickness (rat	tio) 0.5	8 - Wall horizontal PS
Initial prestress for ringbeam	(ratio) 0.5	
Initial prestress for base slab	(ratio) 0.5	

Fig 13 User Input for the Number of Buttresses in a Base model for design check

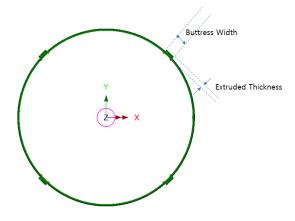


Fig 14 Buttress Definition for a 3D Shell Model

Groups and Materials

The main groups created are named Roof, Wall, and BaseSlab. Two sets of dummy elements, which work as rigid links between the Roof and Ringbeam, and Wall and BaseSlab., are grouped separately, to aid with results-processing.

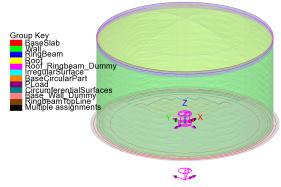


Fig 15 Groups in a 3D Shell Model

After user input, material properties are assigned to relevant members.

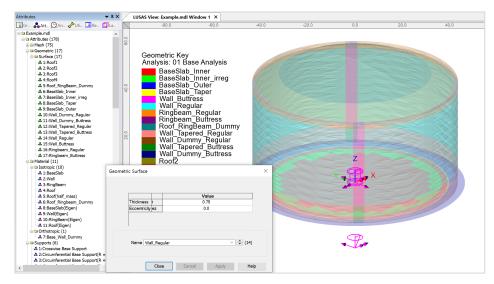


Fig 16 Material Assignments in a 3D Shell Model

Support Conditions

Three different types of support conditions can be defined from Support(3D) tab in Tank Definition.

Use Support (2D) conditions

This option is only available when either 'Fixed Support' or 'Regular Support' is set for Support type in Supports(2D) > Tank Definition. The same support condition defined in Support (2D) will be defined and assigned to Base model for design check.

Regular Support

The regular stiffness (stiffness per unit area) must be stated. A spring support will be assigned to all the bottom line of slab.

If '3D shell structural' option is chosen in Tank Definition, following two options are available.

Simplified foundation

The stiffness of each pile should be defined as shown in [Fig]. The spring support will be assigned to each of pile locations. The Wizard accepts two sets of support stiffness (horizontal and vertical); one for crosswise piles and the other for circumferential piles. If the pile stiffness is different for each pile location due to the ground condition, it can be modified from the Modeller interface by defining different support conditions. If the crosswise piles are zero, then the model does not include crosswise piles and only includes circumferential piles.

The spring stiffnesses are converted into N/m unit in LUSAS Modeller.

Detailed foundation

The stiffness of each pile should be defined as shown in [Fig 21]. The spring support will be assigned to each of pile locations. The Wizard accepts two sets of support stiffness (horizontal and vertical); one for crosswise piles and the other for circumferential piles. If the pile stiffness is different for each pile location due to the ground condition, it can be modified from the Modeller interface by defining different support conditions. If the crosswise pile coordinates are zero, then the model does not include crosswise piles and only includes circumferential piles.

The spring stiffnesses are converted into N/m unit in LUSAS Modeller.

k type –						arget mod	tale to b	wild								
aterial :	Co	oncrete		~	10	-		etric structural		20 ~~	isymmetric (coupled the	ermel/strue	tural		
								ck seismic			iell structura		annavaduc	ai		
evation :	: At	boveground		~		01200	-ann-aut	UN UNISHING		e op si	ion or ucidia					
Definition	Load F	Prestress	Support (3D	Seismic	Ground											
Suppor	t															
ipport typ	pe		(Circumferer	tial Suppor	t									• • • • • • • • •	1
and for all f	oundation		~						Ve	rtical	Horizont ^			Cross	wise piles	
uhimed to	oundation			ID	R [m]		l theta gree]	Number of piles	stiff	ness I/m]	stiffnes: [kN/m]	A	dd		///	
				1	36.7	0.0		56	523.01		2.297E3		Del		Circumferential piles	
o. cir : o. cross	184 a: 213			2	40.8	0.0		60	523.01	18E3 4	2.297E3	Set	t zero			
	156	6.1965E3				0.0			010.0			-		For	N. C.	
X4 Cir 1					14 9	0.0		68	523.01	18E3 4	2 297F3 ×	Set d	lefaults			
		7157E3		3	*** 71											
X ² Cir : X ² Cros	s: 63.			3 (_	>					
X² Cros			Г	3 < Vertical sti			523.018				ss [kN/m]	42.297	E3]		
X² Cros Isswise Gr	s : 63. support sti rid wizard	iffness	E	<			523.018		Horizor	ntal stiffne	>	42.297	E3			
X² Cros sswise Gr	s : 63. support sti	iffness	P4	<			523.018	E3	Horizor	ntal stiffne	>	42.2971 P5	E3	P7 /	Add column	
X² Cros sswise Gr Ccoordii P1	s : 63. support sti rid wizard nates (Unit	iffness ts: m)	P4 12.6	< Vertical sti	ffness [kN/r	n] [E3 Y coordinat	Horizor es (Units	ntal stiffne : m)	> ss [kN/m]			P7 0.0	Add column Add row	
X ² Cros: sswise Gr (coordii P1 0.0	s : 63. support sti rid wizard nates (Unit P2	iffness ts: m) P3		Vertical sti	ffness [kN/r P6	n] [E3 Y coordinat	Horizor es (Units P2	ntal stiffne : m) P3	> ss [kN/m] P4	P5	P6		<u> </u>	
X ² Cros: sswise Gr (coordii P1 0.0 0.0	s : 63. support sti rid wizard nates (Unit P2 4.2	ts: m) P3 8.4	12.6	Vertical sti P5 16.8	ffness [kN/r P6 21.0	n] [P7 25.2		E3 Y coordinat P1 0.0	Horizor es (Units P2 0.0	ntal stiffne : m) P3 0.0	> ss [kN/m] P4 0.0	P5 0.0	P6 0.0	0.0	Add row	
X² Cros: sswise Gr (coordii	s : 63. support sti rid wizard nates (Unit P2 4.2 4.2	ffness ts: m) P3 8.4 8.4	12.6 12.6	Vertical sti P5 16.8 16.8	Ffness [kN/r P6 21.0 21.0	n] [P7 25.2 25.2		E3 Y coordinat P1 0.0 -4.2	Horizor es (Units P2 0.0 -4.2	ntal stiffne : m) P3 0.0 -4.2	> ss [kN/m] P4 0.0 -4.2	P5 0.0 -4.2	P6 0.0 -4.2	0.0 -4.2	Add row Del column	
X ² Cros: osswise Gr C coordin P1 0.0 0.0 0.0 0.0 0.0	s : 63. support sti rid wizard nates (Unit P2 4.2 4.2 4.2 4.2	ffness ts: m) P3 8.4 8.4 8.4 8.4	12.6 12.6 12.6	Vertical sti P5 16.8 16.8 16.8	Freess [kN/r P6 21.0 21.0 21.0	n] P7 25.2 25.2 25.2 25.2 25.2		E3 Y coordinat P1 0.0 -4.2 -8.4 -12.6 +6.0	Horizor es (Units P2 0.0 -4.2 -8.4	ntal stiffne :: m) P3 0.0 -4.2 -8.4	SS [kN/m] P4 0.0 -4.2 -8.4	P5 0.0 -4.2 -8.4	P6 0.0 -4.2 -8.4	0.0 -4.2 -8.4	Add row Del column Del row	
X ² Cros: sswise Gr (coordii P1 0.0 0.0 0.0 0.0	s : 63. support sti rid wizard nates (Unit P2 4.2 4.2 4.2 4.2 4.2	ffness ts: m) P3 8.4 8.4 8.4 8.4	12.6 12.6 12.6 12.6	Vertical sti P5 16.8 16.8 16.8 16.8	Freess [kN/r P6 21.0 21.0 21.0 21.0 21.0	n] P7 25.2 25.2 25.2 25.2 25.2		E3 Y coordinat P1 0.0 -4.2 -8.4 -12.6	Horizor P2 0.0 -4.2 -8.4 -12.6	ntal stiffne : m) P3 0.0 -4.2 -8.4 -12.6	> SSS [KN/m] P4 0.0 -4.2 -8.4 -12.6	P5 0.0 -4.2 -8.4 -12.6	P6 0.0 -4.2 -8.4 -12.6	0.0 -4.2 -8.4 -12.6	Add row Del column Del row Set zero	
X ² Cross sswise Gr Ccoordii P1 0.0 0.0 0.0	s : 63. support sti rid wizard nates (Unit P2 4.2 4.2 4.2 4.2 4.2	ffness ts: m) P3 8.4 8.4 8.4 8.4	12.6 12.6 12.6 12.6	Vertical sti P5 16.8 16.8 16.8 16.8	Freess [kN/r P6 21.0 21.0 21.0 21.0 21.0	n] P7 25.2 25.2 25.2 25.2 25.2		E3 Y coordinat P1 0.0 -4.2 -8.4 -12.6 +6.0	Horizor P2 0.0 -4.2 -8.4 -12.6	ntal stiffne : m) P3 0.0 -4.2 -8.4 -12.6	> SSS [KN/m] P4 0.0 -4.2 -8.4 -12.6	P5 0.0 -4.2 -8.4 -12.6	P6 0.0 -4.2 -8.4 -12.6	0.0 -4.2 -8.4 -12.6	Add row Del column Del row Set zero	
X ² Cross asswise Gr Coordii P1 0.0 0.0 0.0 0.0	s : 63. support sti rid wizard nates (Unit P2 4.2 4.2 4.2 4.2 4.2	ffness ts: m) P3 8.4 8.4 8.4 8.4	12.6 12.6 12.6 12.6	Vertical sti P5 16.8 16.8 16.8 16.8	Freess [kN/r P6 21.0 21.0 21.0 21.0 21.0	n] P7 25.2 25.2 25.2 25.2 25.2		E3 Y coordinat P1 0.0 -4.2 -8.4 -12.6 +6.0	Horizor P2 0.0 -4.2 -8.4 -12.6	ntal stiffne : m) P3 0.0 -4.2 -8.4 -12.6	> SSS [KN/m] P4 0.0 -4.2 -8.4 -12.6	P5 0.0 -4.2 -8.4 -12.6	P6 0.0 -4.2 -8.4 -12.6	0.0 -4.2 -8.4 -12.6	Add row Del column Del row Set zero	
X ² Cros: sswise Gr (coordii P1 0.0 0.0 0.0 0.0	s : 63. support sti rid wizard nates (Unit P2 4.2 4.2 4.2 4.2 4.2	ffness ts: m) P3 8.4 8.4 8.4 8.4	12.6 12.6 12.6 12.6	Vertical sti P5 16.8 16.8 16.8 16.8	Freess [kN/r P6 21.0 21.0 21.0 21.0 21.0	n] P7 25.2 25.2 25.2 25.2 25.2		E3 Y coordinat P1 0.0 -4.2 -8.4 -12.6 +6.0	Horizor P2 0.0 -4.2 -8.4 -12.6	ntal stiffne : m) P3 0.0 -4.2 -8.4 -12.6	> SSS [KN/m] P4 0.0 -4.2 -8.4 -12.6	P5 0.0 -4.2 -8.4 -12.6	P6 0.0 -4.2 -8.4 -12.6	0.0 -4.2 -8.4 -12.6	Add row Del column Del row Set zero	

L

ttributes • • • × Structural Sup	ports					
🖥 Gro 💑 Attr 🕒 Ana 🥓 Util 🛄 Rep 🛑 Lay						
∃⊜ Example.mdl						
Analysis cate Attributes (178)	gory 3D					
🕀 🖴 Mesh (75)						
🕀 🖴 Geometric (17)			_			
🕀 🖴 Material (11)			Free	Fixed	Sp	ring stiffness
Supports (6)		х	0	0		42.297E6
			0		- L	
Sicircumferential Base Support(R = 30.7) Translat	ion in	Y	\bigcirc	\bigcirc		42.297E6
4: Circumferential Base Support(R = 44.9)		z	0	0		523.018E6
-\$ 5:Z Fixed		2	0	0		523.018E0
- & 6:Free		х	۲	\bigcirc	$\circ \Gamma$	
🖃 🖴 Loading (57)			~	0		
Discrete (2)	n about	Y	۲	\bigcirc		
⊟⊜ Patch (2)		z	۲	0	0	
A 21:LiveLoad(Roof)		-	0	0		
A 23:SnowLoad(Roof) Hinge n	otation		۲	\bigcirc	0	
Patch divisions			~	0		
	al warping		۲	\bigcirc		
- & 1:Steel Structure_q1 - & 2:Steel Structure P Pore pro	essure		۲	0		
	200 an e		0	0		
-\$ 4:Liner base1						
Spring stiffr	ness distribu	ution				
& 6:Liner Wall	5					
- 7:SteelStructureRoof						
- 💑 8:Insul_q1	s/unit lengt	n				
\$9:Insul_q2 Stiffnes	s/unit area					
-& 10:Insul_q3						
-& 11:Insul_q4 Lift-off	>>					
-& 12:Insul_qr						
-& 13:Insul_Pressure Contac	t >>					
- & 14:Wall piping loading						
- 🎝 15:Liquid_q1(Max)			_	_		•
	ne Crossw	ise Base	Support			~ 🗘 (1)
- 4 17:GasPressure(Max)						
- \$ 18:GasPressure(Max)(Base_Roof) - \$ 19:GasPressure(Min)						
		Close		Cancel	Ap	ply Help
- & 20:GasPressure(Min)(Base_Roof)		0.000			, ib	noip

Fig 17 Support Definition in a Base model for design check

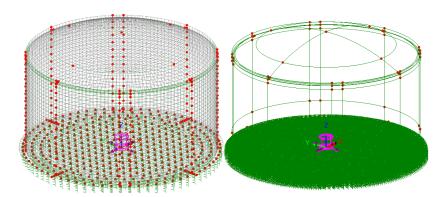


Fig 18 Support Condition for a Base model for design check (Pile Support / Regular Support)

Loadings

28 loadcases are created in a Base model for design check. Wind load can be added after a base model is created. (LNG Tank> Add Loading> Wind...)

LNG Tank - Add wind loading			×
Design code		EN1991-1-4 (2005)	~
Design code parameters			
Basic wind velocity		37.5	[m/s]
Roughness length		3.0E-3	[m]
Minimum height		1.0	[m]
Orography factor		1.0	
Terrain factor		0.156	
Turbulence factor		1.0	
Air density		1.25	[kg/m^3]
	Defaults	OK Cancel	Help

Fig 19 User Input for Wind Load for a Base model for design check

Other Options

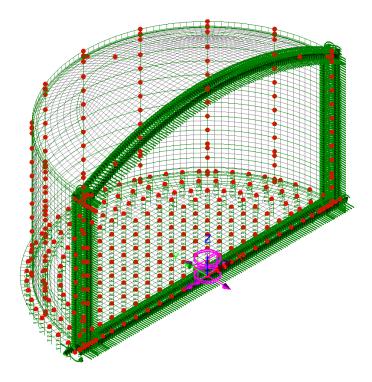
Half Only Model

A half model is produced with symmetrical support conditions when the 'Half only model' option is selected.

Base Model

LNG Tank - Base Model for	Design Check		\times
Tank definition data Model filename Saved model file path	Tnk1 C:\Users\ohsso\D	Vocuments\LUSAS200\Projects\.mdl	
Modeling options			
Element size (m)	2.0	Half symmetric model	
Number of eigenvalue	e 10	✓ Include non-structural masses in the eigenvalue analysis	

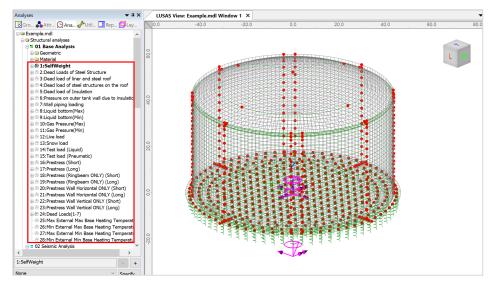
Fig 20 Option for Half Model

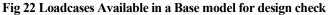


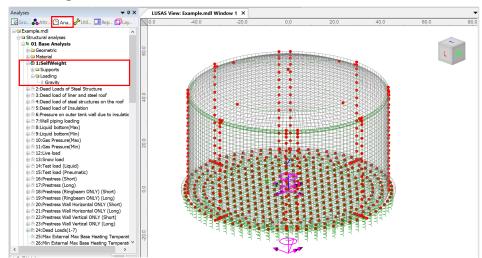


Loadings

28 loadcases are defined for the base model for design check.







Self Weight

Fig 23 Self Weight in a Base model for design check

Dead Loads of Steel Structure

The dead load of the steel inner tank is defined including wall plate, secondary bottom, bottom plate, annular plate and suspended deck.

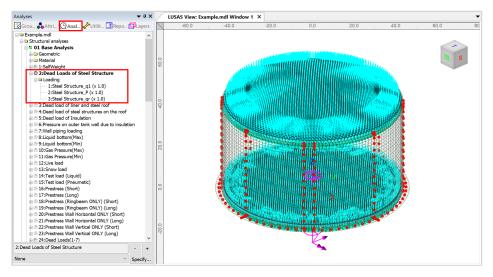
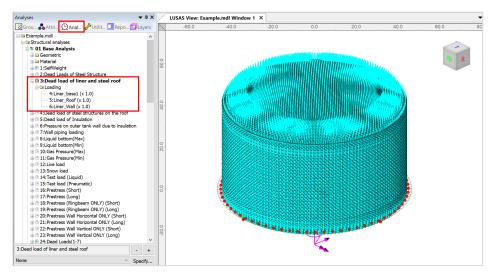


Fig 24 Dead Loads for Steel Structure in a Base model for design check

Dead load of liner and steel roof

The total weight of the roof plate and frame need to be specified to design the roof.





Dead load of steel structures on the roof

For the design of the outer tank, the loadings due to the steel structure on the roof as well as the pipe work on the roof should be considered as distributed load on the roof.

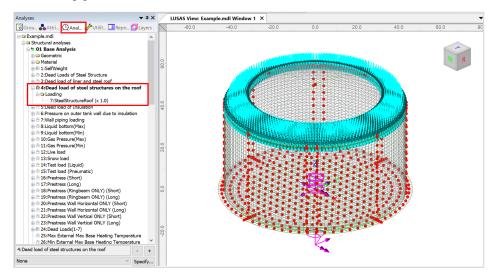
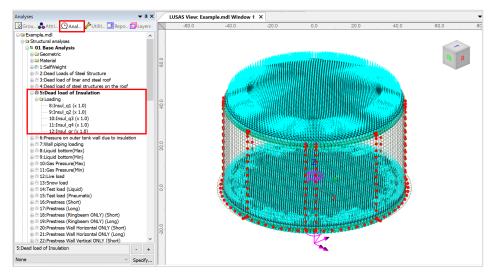
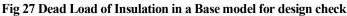


Fig 26 Dead Load of Steel Structures on the Roof in a Base model for design check

Dead load of Insulation

All insulation to the base, wall and suspended deck are defined.





Pressure on outer tank wall due to insulation

The insulation (e.g. loose fill perlite) in the region between the inner tank and outer tank is assumed to exert a horizontal loading on the outer tank.

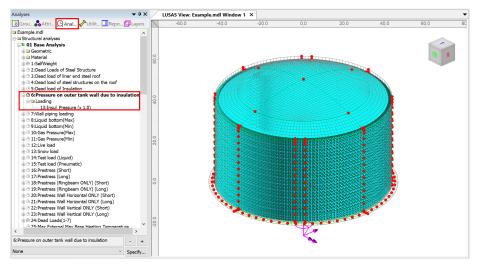


Fig 28 Insulation Pressure Load in a Base model for design check

Wall Piping Loading

Wall piping loading acts on the outer surface of the ringbeam and wall.

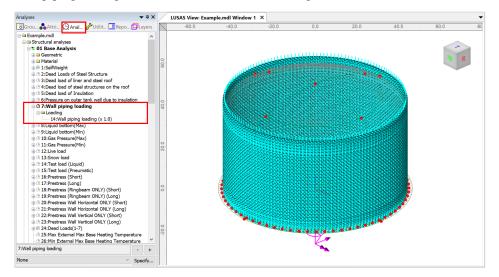
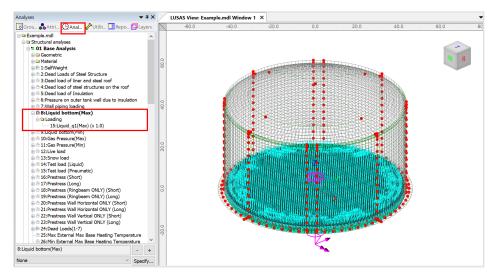
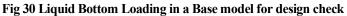


Fig 29 Wall piping loading in a Base model for design check

Liquid bottom (Max/Min)

The Liquid weight acts on the top surface of the base slab.





Gas Pressure(Max/Min)

Design gas pressure acts on the inner surface of the concrete tank.

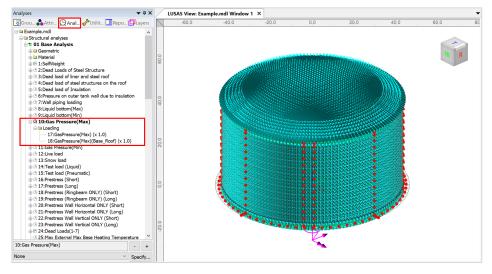


Fig 31 Gas Pressure Loading in a Base model for design check

Live load (Imposed Load on the roof)

Live Load (Imposed Load on the roof, ref. EN 14620-1) is assigned on the top surface of the roof.

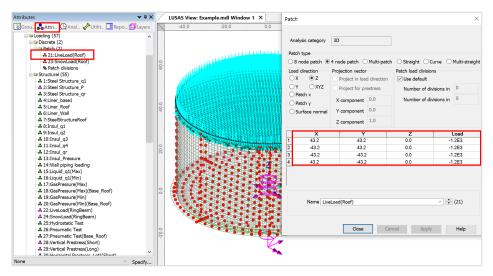


Fig 32 Live Load in a Base model for design check (Roof)

Snow load

The snow load is assigned on the top surface of the roof.

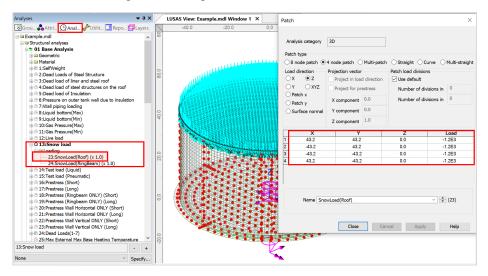
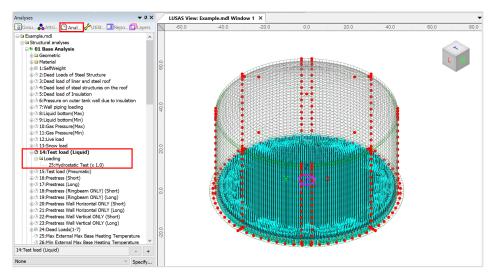
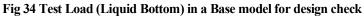


Fig 33 Snow Load in a Base model for design check (Roof)

Test load (Liquid bottom)

The Test load (Liquid bottom) acts on the top surface of the base slab.





Test load (Pneumatic)

Test load (Pneumatic) acts on the inner surfaces of the concrete tank.

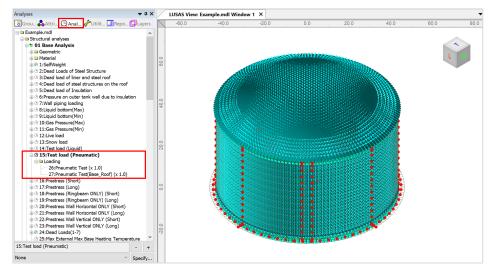


Fig 35 Test Load (Pneumatic) in a Base model for design check

Prestress Load

The effect of the prestressing steel tendons needs to be converted to equivalent external load and used for the input in the Wizard.

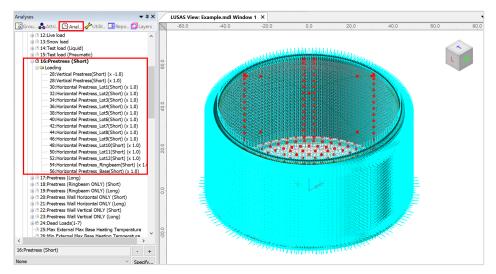


Fig 36 Prestress Load in a Base model for design check

Analyses

Five analyses having a total of 66 loadcases are set-up in the model.

- □ Base Analysis A static analysis. A total of 28 loadcases are provided including 4 temperature loading which can be added from a separate Thermal Analysis. Loadcase for wind load is not created when Base model is built. However, it can be added using 'Add loading> Wind...'
- □ Seismic Analysis Equivalent peak seismic acceleration and hydrodynamic loading are to be added.
- □ Staged Construction Analysis 29 construction stages are defined with self weight only if 'Roof split 1' is set for Roof construction plan.
- □ CRSH Analysis CRSH analysis is not created when Base model is built. However, it could be added using 'Add loading> Creep and Shrinkage' menu. The same number of loadcases for Staged construction analysis will be created. As for Staged Construction Analysis, combinations have been added to consider the pure effect from Creep and Shrinkage alone. This is obtained by subtracting the results in Analysis 5 (including CRSH) and the results in Analysis 4 (not considering CRSH).
- **Eigenvalue Analysis** To obtain Eigenvalue results.

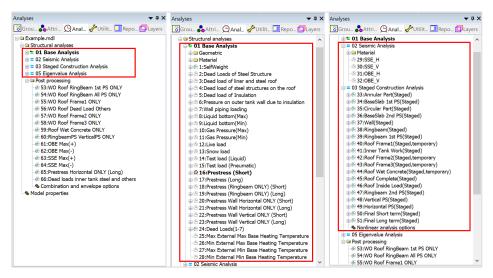


Fig 37 Analyses Available in Base Model

Roof element layers

The roof is built in two layers. Hence, two sets of overlapping surfaces have been created to consider the staged construction. As only the final layer is required for the other analyses, one set of surfaces is deactivated for the 1st loadcase of Base Analysis, Thermal Analysis, Seismic Analysis, and Eigenvalue analysis.

Loadings

Self weight is assigned in all analyses.

- In the Base Analysis, 28 loadcases are provided, including 4 temperature loading.
- In the other analyses (with the exception of the self weight) loading is not yet assigned.

Base Load Combinations

As seen in [Fig 63], 17 load combinations are pre-defined. These combinations are used to obtain the isolated effect of adding loading from a Staged Construction Analysis, so that they can be used as a single loadcase in the design load combination. A staged construction analysis is nonlinear by definition and hence the principle of superposition is not generally applicable. However, this simplified approach provides a systematic and efficient way of verifying hundreds of load combinations in the context of linear design according to a code of practice.

For example, the load combination of '**WO Roof Ringbeam 1st PS ONLY**' is for the subtraction of 'Ringbeam[Staged]' from 'Ringbeam 1st PS[Staged]'.

- 1) Ringbeam[Staged] : Wall & Ringbeam is built. (i.e. no roof)
- 2) RingBeam 1st PS[Staged] : Wall & Ringbeam is built. (i.e. no roof). 1st PS is added.

Both loadcases are included in the Staged Construction Analysis, and the combination of 2)-1) is used to obtain the effect of the 1st PS loading during the construction stage.

Analyses 🔷 🕈 🗙	Combination	×
Strutt Q Anal. Vulit. ■ Repo Q Layers Structural analyses Structural analyses Structural analyses 0.1 Base Analysis Structural analyses 0.1 Base Analyses Structural analyses 0.1 Base Analys	Method Factored Veriable 52:Mode 10 Frequency = 4.59906 Post processing 10 95:W0 Roof Fanal ONLY 95:W0 Roof Fanal ONLY 95:W0 Roof Franal ONLY	•
	Close Cancel Apply Help	

Fig 38 Pre-defined Load Combinations - WO Roof 1st PS only

Template for load combinations

A template for use in defining design load combinations is saved in the current working folder with name of the form: **<Model name>_ComboTemplate.xlsx**.

Update Reinforcement and Prestress

If the reinforcement or prestress tendon arrangement needs to be updated after the model is built, it can be updated by selecting the menu item LNG Tank > Design Checks> Add/Update Reinforcement...

This only updates the existing properties. If the attribute is to be de-assigned or deleted, it should be done manually.

If one-side only rebar is required, use a negligible value for Es. (e.g. a very small value

Thermal Analysis

Thermal analysis that requires the evaluation of heat transfer through thickness cannot be performed with shell elements, so a thermal analysis using 2D axisymmetric solid elements is required instead.

Thermal Analysis for Max Environmental Temperature

Select the LNG Tank > Create 2D model> Thermal Analysis... menu item, and assuming the default inputs are for maximum temperature, perform a 2D Thermal Analysis with the input data of 'Tnk1 – Max Temperature'

nk definition data	Tnk1				~
	Examp	ole-Max			
odel filename	C:\Use	rs\ohsso\Documents\LUSAS2(00\Projects\Example-	-Max_Thermal.md	
aved model file path					
Modeling options					
Concrete element size [m]		Steel element size [m]	0.0	Include	e soil for aboveground tanks
Include Structural Load					
Variable Loads to apply(*)				⊖ Min	
valiable Loads to apply()	/				
- The chosen variable load	ds from the Tank D	efinition will be used for Opera	atina Condition.		
(*) These parameters are r		efinition will be used for Ooera tural Loading Definition] tab c		attribute.	
(*) These parameters are r Spillage Loading	read from the [Struc	tural Loading Definition] tab c	of the tank definition	attribute.	∩Wali
(*) These parameters are r	read from the [Struc	etural Loading Definition] tab c	of the tank definition	attribute.) Wali
(*) These parameters are r Spillage Loading	read from the [Struc	tural Loading Definition] tab c	of the tank definition		○ Wali [480 [kg/m³]
(*) These parameters are r Spillage Loading Application target above Cor Radius of inner tank outer su	read from the [Struc mer Protection urface(*)	tural Loading Definition] tab c	of the tank definition ion layer id density(*)	480 [kg/m³]
(*) These parameters are r Spillage Loading Application target above Cor Radius of inner tank outer su	read from the [Struc ner Protection urface(*) ad from the [Seismi	etural Loading Definition] tab c	of the tank definition ion layer id density(*)	480 [kg/m³]
(*) These parameters are r Spillage Loading Application target above Con Radius of inner tank outer su (*) These parameters are rea	read from the [Struc ner Protection urface(*) ad from the [Seismi	etural Loading Definition] tab c	of the tank definition ion layer id density(*)	480 [kg/m³]
(*) These parameters are r Spillage Loading Application target above Cor Radius of inner tank outer su (*) These parameters are rea Spillage uurauon ume for et	read from the [Struc ner Protection urface(*) ad from the [Seismi acm spinage neight	tural Loading Definition] tab c	of the tank definition on layer .id density(* ab of the tank definiti [hour]) ion attribute if ava	[480 [kg/m ^a] ilable.

• Enter **Example - Max** for the model file name, and click **OK**.

Fig 39 Dialog for Thermal Analysis (Max)

The current shell model will be closed and a new 2D axisymmetric solid model suitable for thermal analysis will be built.

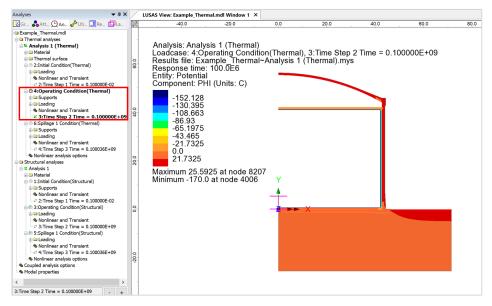


Fig 40 Thermal Analysis for Max Environmental Temperature

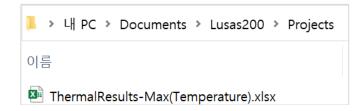
• Solve the model, and in the Analyses 🕒 treeview, set the loadcase Operating Condition (Thermal) active.

The Equivalent Uniform Temperature and Linear Temperature Gradient through thickness can be computed and saved in a spreadsheet by selecting the menu item LNG Tank > Excel Tools> Export Temperatures...

LNG Tank - Export Tempera	tures to Excel (2D)	\times				
	ThermalResults-Max					
Output filename						
Working folder	Current User Defined					
Save in	C:\Users\ohsso\Documents\Lusas200\Projects\ThermalResults-Max(Tempe					
	OK Cancel Help					

Fig 41 Extraction Thermal Max Results

In the current working folder, the file **ThermalResult-Max.xlsx** containing the extracted results for Roof/Slab/Wall is created.



Temperature of	Temperature of Roof									
Туре	Temperature									
Location	Roof									
Unit	Celsius									
No of Slices	Gap (m)	Outer Radius (m)	Rad. Origin Y (m)	Min. Thickness (m)						
91.00	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				
0.51	23.61	25.33	0.50	24.47	-3.71	0.00				
1.01	23.62	25.33	0.50	24.47	-3.70	0.01				
1.52	23.61	25.32	0.50	24.47	-3.71	0.01				
2.03	23.61	25.32	0.50	24.47	-3.71	0.02				
2.54	23.61	25.33	0.50	24.47	-3.71	0.04				
3.04	23.61	25.32	0.50	24.47	-3.71	0.05				
3.55	23.61	25.32	0.50	24.47	-3.71	0.07				

Fig 42 Extracted Thermal Results in a Spreadsheet

Thermal Analysis for Min Environmental Temperature

To perform additional thermal analyses for a minimum environment temperature, reselect the menu item LNG Tank >Tank Definition...

Thermal Analysis

ank Definition							
Tank type		Target models to	o build				
Material : Concrete	~	2D axisymmetric structural		🗸 2D axisymmet	2D axisymmetric coupled thermal/structural		
Elevation : Aboveground	~	i 2D beam∹	stick seismic	✓ 3D shell struct	ural		
ank Definition Load Prestress Insulation							
Structural Dead Loading Structural Varial	ble Loading Thermal I	Loading					
Load type	Length [m]	Temperature [C]	Convective coefficient [J/m ² .s.C]	Type of boundary		Description	
Initial Temperature (Structure)	0.0	10	0.0	Prescribed	•	Initial Temperature of Structure	
Initial Temperature (Soil)	0.0	10	0.0	Prescribed	·	Initial Temperature of Soil	
Soil Bottom Depth & Temperature	25.0	15.1	0.0	Prescribed	•	Soil Bottom where Temperature is constant	
External Temperature	0.0	10	25.0	Convection	•	External Temperature	
Liquid Temperature	0.0	-170.0	166.47	Prescribed	•	Liquid Temperature	
Base Heating	0.0	5.0	0.0	Prescribed	•	Base Heating	
Spillage 1	38.263	-170.0	166.47	Prescribed	•	Spillage 1	
Spillage 2	0.0	-170.0	166.47	Prescribed	•	Spillage 2	
Spillage 3	0.0	-170.0	166.47	Prescribed	•	Spillage 3	
Spillage 4	0.0	-170.0	166.47	Prescribed	•	Spillage 4	
Spillage 5	0.0	-170.0	166.47	Prescribed	•	Spillage 5	
Set zero Set defaults	* The temperature	for base heating wil	Il only be considere	ed if a value other than ze	ero is	defined.	_
	Name	e Tnk3				✓ + (new)	
						OK Cancel Apply	Help

Fig 43 User Input for Min Environmental Temperature

- Enter 10 (Celsius) for both the External Temperature and the Initial Temperature (Soil). The Initial Temperature (Structure) is the temperature at the time when concrete is poured, hence there is no need to change at the moment.
- Save these inputs with name 'Tnk2-Min Temperature'.
- Select the menu item LNG Tank > Create 2D Model> Coupled Thermal /Structural... and enter Example Min for the model file name, then click OK.

nk definition data	Tnk2-	Min Temperature			~	
	Exam	ple-Min				
del filename	C:\Lle	ars/obsso/Documents/L	JSAS200\Projects\Example	Min Thermal md	1	
ved model file path	0.100				•	
Modeling options						
Concrete element size [m]		Steel element si	0.0	🗹 Includ	e soil for aboveground tanks	
✓ Include Structural Load						
				◯ Min		
Variable Loads to apply(*) - The chosen variable loads f			r Operating Condition.	~		
Variable Loads to apply(*) - The chosen variable loads f (*) These parameters are reac Spillage Loading	d from the [Stru		r Operating Condition.	~		
Variable Loads to apply(*) - The chosen variable loads f	d from the [Stru	ctural Loading Definitio	r Operating Condition.	~) Wali	
Variable Loads to apply(*) - The chosen variable loads f (*) These parameters are reac Spillage Loading	d from the [Stru Protection	ctural Loading Definitio	r Operating Condition. n] tab of the tank definition	attribute.	Wall 480 [kg/m²]	
Variable Loads to apply(*) - The chosen variable loads f (*) These parameters are reac Spillage Loading Application target above Corner	d from the [Stru Protection ce(*)	etural Loading Definitio • 1st Wall 42.1361 [m]	r Oceratino Condition. n] tab of the tank definition Insulation layer id density(*	attribute.	480 [kg/m³]	
Variable Loads to apply(*) - The chosen variable loads f (*) These parameters are reac Spillage Loading Application target above Corner Radius of inner tank outer surface	d from the [Stru Protection ce(*) from the [Seism	etural Loading Definitio	r Oceratino Condition. n] tab of the tank definition Insulation layer id density(*	attribute.	480 [kg/m³]	
Variable Loads to apply(*) - The chosen variable loads f (*) These parameters are reac Spillage Loading Application target above Corner Radius of inner tank outer surfac (*) These parameters are read f	d from the [Stru Protection ce(*) from the [Seism	etural Loading Definitio	r Oceratino Condition. n] tab of the tank definition Insulation layer id density(*	attribute.	480 [kg/m³]	

Fig 44 Dialog for Thermal Analysis (Min)

- Solve the model, and in the Analyses 🕀 treeview, set the loadcase **Operating Condition** (**Thermal**) active.
- Export results into a spreadsheet by selecting the menu item LNG Tank > Excel Tools> Export Temperatures...

LNG Tank - Export Tempe	eratures to Excel (2D)
	ThermalResults-Min
Output filename	
Working folder	◯ Current
Save in	C:\Users\ohsso\Documents\Lusas200\Projects\ThermalResults-Min(Temper
	OK Cancel Help

Fig 45 Extraction Thermal Min Results

Update Base Model

The thermal analysis results can be converted to 3D loading for load combinations.

- Close the thermal model, and open the shell model of 'Example(CodeChecking).mdl'.
- Select the menu item LNG Tank > Add loading> Thermal ... then select the spreadsheets for thermal analysis results and click OK.

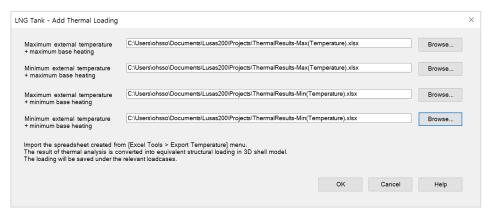
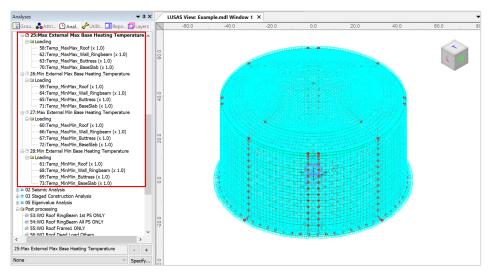


Fig 46 Dialog for Adding Thermal Loading

The thermal analyses results are converted into equivalent structural temperature loadings in the 3D shell model.





Seismic Analysis

Preparation

A seismic analysis considering fluid-soil-structure interaction under seismic action should be carried out prior to the 3D shell model investigation. The inertial and hydrodynamic peak effects obtained from a seismic analysis can be transformed to equivalent static loading for a 3D model in the form of accelerations that will act on the structural masses and other structural loadings.

Update Base Model

Both OBE and SSE loadings can be defined by selecting the menu item LNG Tank > Add Loading> Seismic...

prizontal direction			
	OBE	SSE	
Roof acceleration	4.0	8.0	m/s²
Nall acceleration	2.5	5.0	m/s²
Base acceleration	2.0	4.0	m/s²
LNG force	250.0E3	500.0E3	kN
Moment from inner tank base (IBP)	500.0E3	960.0E3	kN-m
Moment from inner tank base (EBP)	50.0E3	60.0E3	kN-m
	OBE	SSE	
Roof acceleration	3.0	6.0	m/s²
Wall acceleration	1.5	3.0	m/s²
Base acceleration	1.5	3.0	m/s²
nner tank acceleration	1.5	3.0	m/s²
LNG force	200.0E3	400.0E3	kN

Fig 48 Dialog for Adding Seismic Loadings

Horizontal Loadings

Based on the given inputs, the loadings are defined as shown below. The acceleration loadings are directly used for defining Body Force loading in Modeller, and the other loadings are converted to equivalent structural loadings.

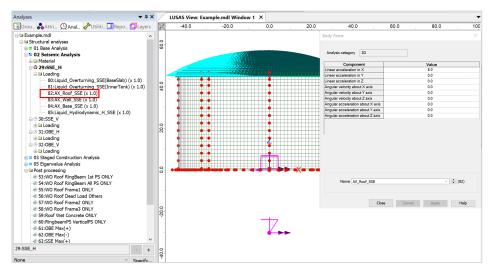


Fig 49 Horizontal Seismic Loading for Roof (SSE)

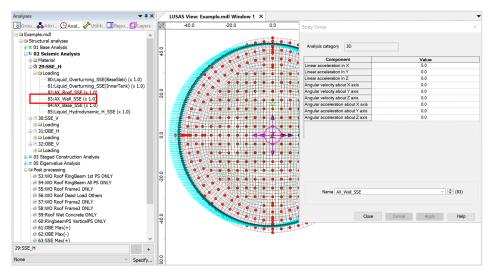


Fig 50 Horizontal Seismic Loading for Wall (SSE)

Seismic Analysis

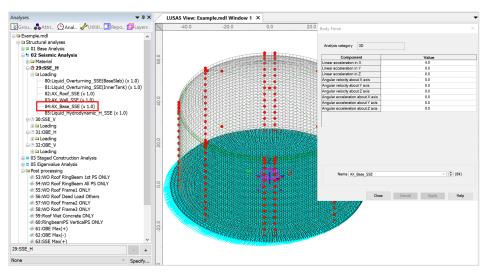


Fig 51 Horizontal Seismic Loading for Base Slab (SSE)

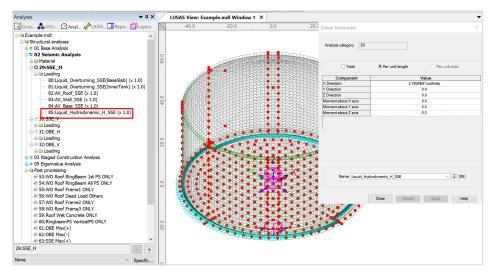


Fig 52 Horizontal Seismic Loading of Liquid Force (SSE)

- □ The force of the liquid is transferred to the Base Slab through the inner tank, so the loading is applied at the location of the inner tank wall.
- □ The total force defined from user input is 500E3 kN in the global X direction, however the pressure of the liquid acts perpendicular to the inner tank wall surface with an intensity following a cosine variation.
- □ A cylindrical local coordinate system is applied to the lines to ensure loading is in a radial direction.

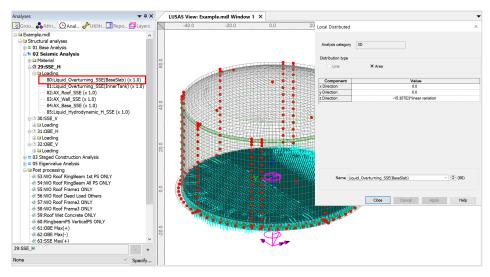


Fig 53 Horizontal Seismic Loading of Overturning Moment from Base Slab (SSE)

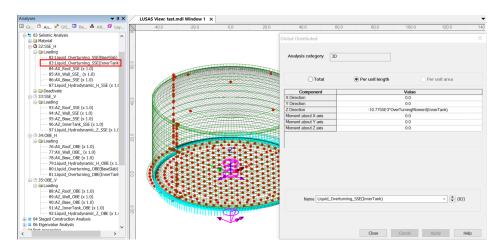


Fig 54 Horizontal Seismic Loading of Overturning Moment from Inner Tank (SSE)

Vertical Loadings

The acceleration loading is directly used to define Body Force loading in Modeller except for the acceleration for the Inner Tank. As the Inner Tank is not included in the meshed model, the loading is converted to equivalent structural loading.

Seismic Analysis

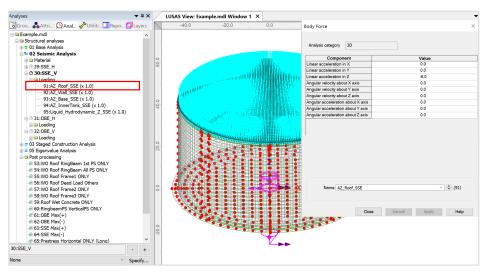
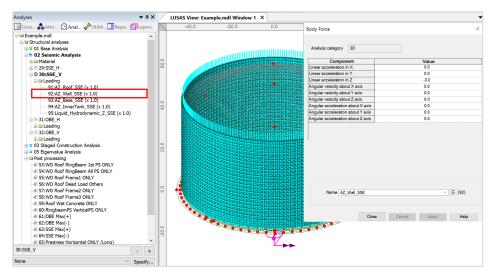


Fig 55 Vertical Seismic Loading for Roof (SSE)





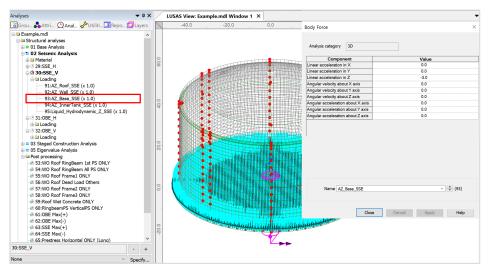


Fig 57 Vertical Seismic Loading for Base Slab (SSE)

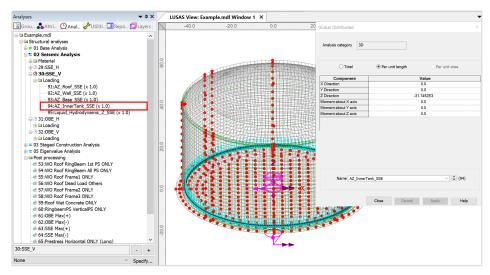


Fig 58 Vertical Seismic Loading for Inner Tank (SSE)

□ The inner tank is not included as structural elements, so the vertical loading from the inner tank should be converted for structural loading.

Seismic Analysis

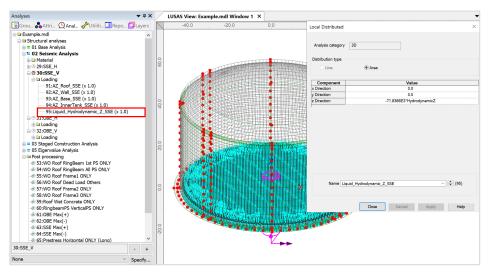


Fig 59 Vertical Seismic Loading for Liquid (Hydrostatic, SSE)

□ The given liquid pressure is 400E3 kN is converted as a distributed load for 3D model.

Attributes 🗸 🕈 🗙	Global Distributed	×
Image: Second	Analysis category 3D	
- ♣ 92:AZ_Wall_SSE - ♣ 93:AZ_Base_SSE - ♣ 94:AZ_InnerTank_SSE - ♣ 95:Liquid Hydrodynamic Z SSE	◯ Total	Per unit length Per unit area
-& 96:Insul_q1_constr	Component	Value
- 🎝 97:Insul_q2_constr	X Direction	60.4851E3
-& 98:Insul_q3_constr	Y Direction	0.0
-& 99:Insul_q4_constr	Z Direction	-34.921E3
100:Roof Frame1(Staged,temporary)	Moment about X axis	0.0
\$ 101:Roof Frame2(Staged,temporary	Moment about Y axis	0.0
-& 102:Roof Frame3(Staged,temporary)	Moment about Z axis	0.0
- \$ 103:Roof Wet Concrete(Staged,temporary)		
a Local coordinate (4) b C l'istriction (2) b C l		
⊖ Splindrical (3)		
3:Wall Local		
-& 4:Wind Local		
⇒ Spherical (1)		
Spherical for ForceExtraction		
Generation and a second se		
-\$ 1:Activate		
🖃 🖴 Deactivate elements (3)		
- 🖧 1:Deactivate		
2:Deactivate(Roof ring)	Name Roof Frame	e1(Staged,temporary) V + (100)
- 🖧 3:Deactivate(Roof center)		
🖃 🔤 Search area (1)		
- 🕹 1:SearchArea(Roof)		
🖃 🕾 Results Transformation (3)		Close Cancel Apply Help

Fig 60 Roof Frame1 Loading

As these loadings are not permanent loading, they are not inherited by the subsequent stages and are marked as '**temporary**' in the loadcase name.

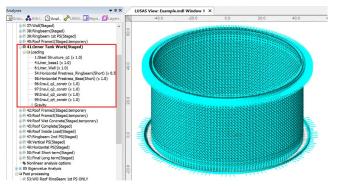


Fig 61 Stage 9 of Staged Construction Analysis

Stage 9 assumes that the inner tank has been built. All insulation loading except for 'Roof Liner' will be defined and assigned at this stage.

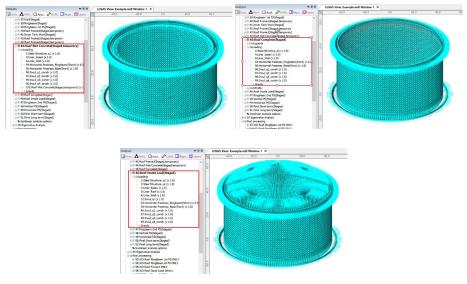


Fig 62 Stage 12-14 of Staged Construction Analysis

Stage 12 assumes that the roof is being built and the poured concrete is acting as a loading on the ringbeam.

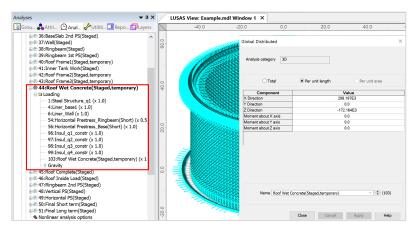


Fig 63 Stage 12 of Staged Construction Analysis

Stage 13 assumes that the roof is completed (Roof ratio for 1st built is set to 1.0. As shown below. At this stage the wet concrete loading assigned at **Stage 12** is removed and replaced with the body force of the Roof.



Fig 64 Roof Shape at Stage 13

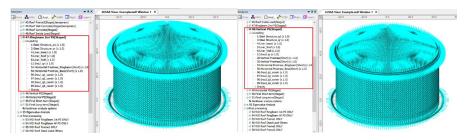


Fig 65 Stage 15 ~ 16 of Staged Construction Analysis

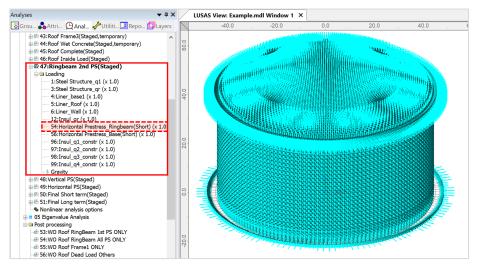


Fig 66 Stage 15 of Staged Construction Analysis

At Stage 15, 50% of additional RingBeam Prestress is added. (The Load Factor is updated from 0.5 to 1.0 for Horizontal Prestress_Ringbeam load.)

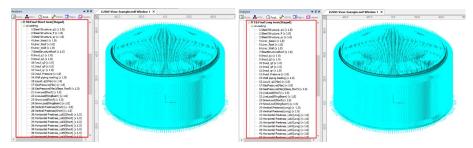


Fig 67 Stage 18 ~ 19 of Staged Construction Analysis

At Stage 16 the vertical prestress is added.

At Stage 17 all the horizontal prestress for the Wall is added.

Stage 18 is the final stage. The structures are complete as built, and all loadings for the operating condition are added.

Stage 19 models long-term effects. The prestress values are updated to those for long-term PS.

• Single layered roof 2

Stage Description Note	Stage	Description	Note
------------------------	-------	-------------	------

Stage	Description	Note
No. 1	Annular part	
No. 2	1) + Base 1^{st} PS	
No. 3	2) + Circular part	
No. 4	3) + Base 2 nd PS	
No. 5	4) + Wall is added up in stages	
No. 6	5) + Ringbeam 1 st	
No. 7	6) + Wall End 1 st PS	
No. 8	7) + Ringbeam 1 st PS	
No. 9	8) + Roof Frame 1	
No. 10	9) + Inner Tank Work	
No. 11	10) + Roof Frame 2	
No. 12	11) + Roof Frame 3	
No. 13	12) + Ringbeam	
No. 14	13) + Roof Wet Concrete	
No. 15	14) + Roof Complete	
No. 16	15) + Roof Inside Load	
No. 17	16) + Vertical PS	
No. 18	17) + Horizontal PS	
No. 29	18) + Final Short term	
No. 30	19) + Final Long term (Long term PS applied)	

Table 1 Sequence of Construction Stages

• Layered roof option 1

Stage	Description	Note
No. 1	Annular part	
No. 2	1) + Base 1^{st} PS	

Stage	Description	Note
No. 3	2) + Circular part	
No. 4	3) + Base 2 nd PS	
No. 5~6	4) + Wall & Ringbeam is added up in stages	
No. 7	6) + Ringbeam 1 st PS	
No. 8	7) + Roof Frame 1	
No. 9	8) + Inner Tank Work	
No. 10	9) + Roof Frame 2	
No. 11	10) + Roof Frame 3	
No. 12	11) + Roof Lower Wet Concrete	
No. 13	12) + Roof Lower Complete	
No. 14	13) + Roof Lower Inside Load	
No. 15	14) + Roof Upper Wet Concrete	
No. 16	15) + Roof Complete	
No. 17	16) + Ringbeam 2 nd PS	
No. 18	17) + Vertical PS	
No. 19	18) + Horizontal PS	
No. 20	19) + Final Short term	
No. 21	20) + Final Long term (Long term PS applied)	

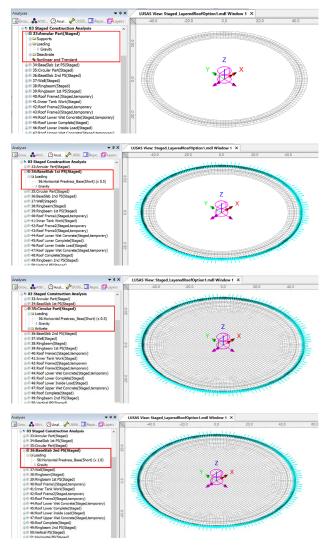


Fig 68 Stage 1 ~ 4 of Staged Construction Analysis

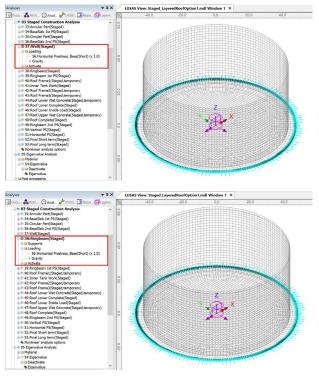
Stage 1 builds the annular part of the slab.

Stage 2 adds the 1st PS for the Base Slab

Stage 3 the central part of slab.

Stage 4 adds the 2nd PS for the Base Slab.

Stage 5 models the construction of the first lift of the Wall.



Note that self weight is always assigned when a new part of the structure is added. Loading defined and assigned at a stage is inherited by the subsequent stages.

Fig 69 Stage 5 ~ 6 of Staged Construction Analysis

At Stage 6, the Wall and RingBeam construction is complete.

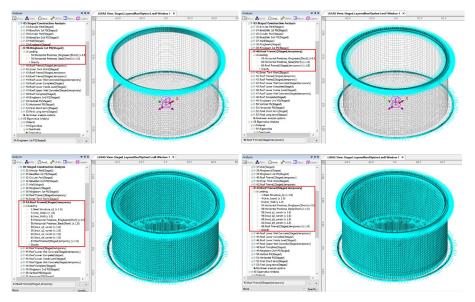


Fig 70 Stage 7 ~ 11 of Staged Construction Analysis

At **Stage 7**, 50% of RingBeam prestress is assumed to be applied if the **'Initial prestress for Ringbeam (ratio)'** is set to 0.5. (A Load Factor of 0.5 was used for the Horizontal Prestress_Ringbeam load.) If **'Initial prestress for Ringbeam (ratio)'** is set to a different value then the Ringbeam prestress at **Stage 7** will have a different load accordingly.

Stages 8, 10 and **11** assume that there could be temporary loads on the line where the roof and RingBeam are connected. The loadings for Roof Frame 1, Roof Frame 2 and Roof Frame3 are defined using user inputs. The user needs to input total loading and Modeller will automatically convert this to the equivalent load per unit length according to the length of line the loading is assigned to.

tributes 🗸 🕈	×	Global Distributed		>	
Grou. Attri QAnal. //Utiliti Repo Layer - 55:Horizontal Prestress_Ringbeam(Long) - 56:Horizontal Prestress_Base(Short) - 57:Horizontal Prestress_Base(Long) - 55:Horizontal Prestress_Base(Long)	^	Analysis category 3D			
- so:insul_q1_constr - so:insul_q2_constr - so:insul_q3_constr - so:insul_q4_constr		◯ Total	• Per unit length	O Per unit area	
- 62:Roof Upper Wet Concrete1		Component		Value	
& 63:Roof Upper Wet Concrete2		X Direction	6	0.4851E3	
-& 64:Roof_Upper_Wet_Concrete3 -& 55:Roof_Upper_Wet_Concrete4 -& 66:Roof Frame1(Staged,temporary) -& 67:Roof Frame2(Staged,temporary		Y Direction		0.0	
		Z Direction	-	34.921E3	
		Moment about X axis		0.0	
		Moment about Y axis		0.0	
-& 68:Roof Frame3(Staged,temporary)		Moment about Z axis		0.0	
69:Roof Lower Wet Concrete(Staged,temporary)				0.0	
a Local coordinate (4)					
⊖ tylindrical (3)					
-♣ 1:LocalCoord -♣ 3:Wall Local					
a Spherical (1)					
Spherical (1) Spherical for ForceExtraction					
Activate elements (1)					
□					
\$ 1:Deactivate					
2:Deactivate(Roof ring)		Name Roof Fram	e1(Staged,temporary)	~ 🗘 (66)	
3:Deactivate(Roof center)					
⊟⊜ Search area (1)					
- 🖧 1:SearchArea(Roof)					
🖃 🖴 Results Transformation (3)			Close Cancel	Apply Help	

Fig 71 Roof Frame1 Loading

As these loadings are not permanent loading, they are not inherited by the subsequent stages and are marked as '**temporary**' in the loadcase name.

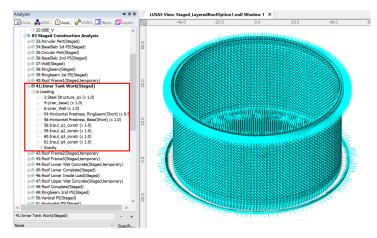


Fig 72 Stage 9 of Staged Construction Analysis

Stage 9 assumes that the inner tank has been built. All insulation loading except for 'Roof Liner' will be defined and assigned at this stage.

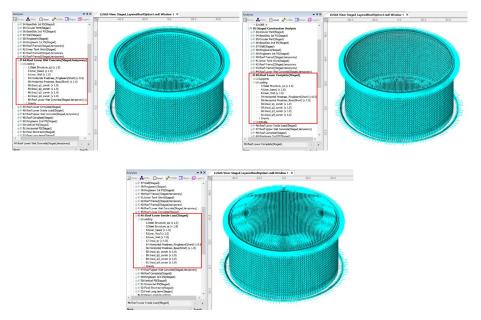


Fig 73 Stage 12-14 of Staged Construction Analysis

Stage 12 assumes that the lower half of the roof is being built and the poured concrete is acting as a loading on the ringbeam.

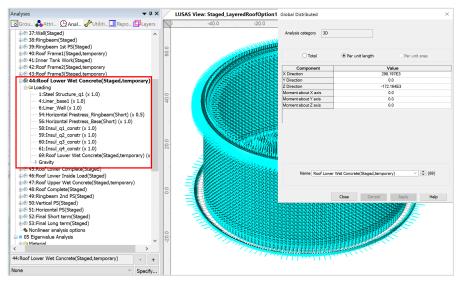


Fig 74 Stage 12 of Staged Construction Analysis

Stage 13 assumes that the lower half of the roof is built (if Roof ratio for 1^{st} built = 0.5 from **Tank Definition**), and the lower part of the roof is newly activated. As shown below, the geometric properties used represent those for only half of the Roof at stage 13, only becoming geometric properties for the whole roof at stage 16.

Stage 13 assumes that the lower part of roof is completed. At this stage the wet concrete loading assigned at Stage 12 is removed and replaced with the body force of the lower part of Roof.

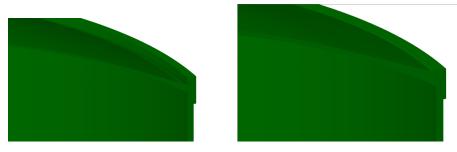


Fig 75 Roof Shape at Stage 13, 16.

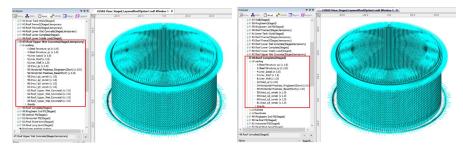


Fig 76 Stage 15 ~ 16 of Staged Construction Analysis

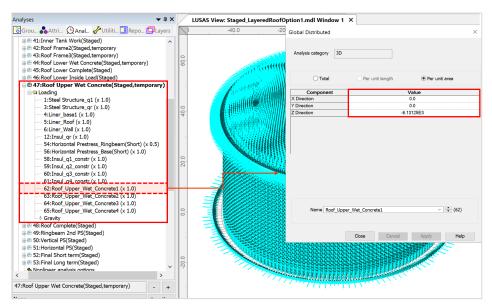


Fig 77 Stage 15 of Staged Construction Analysis

Stage 15 models the upper half of the Roof being built with the poured concrete acting as a load on the already cast lower half of the Roof.

Stage 16 assumes that the upper part of the Roof is now built. The wet concrete loading assigned at Stage 15 is removed and replaced with the body force of the upper part of the Roof.

At Stage 17, 50% of additional RingBeam Prestress is added. (The Load Factor is updated from 0.5 to 1.0 for Horizontal Prestress_Ringbeam load.)

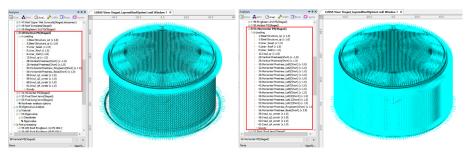


Fig 78 Stage 18 ~ 19 of Staged Construction Analysis

At Stage 18, the vertical prestress is added.

At Stage 19, all the horizontal prestress for the Wall is added.

Stage 20 is the final stage. The structures are complete as built, and all loadings for the operating condition are added.

Stage 21 models long-term effects. The prestress values are updated to those for long-term PS.

If 'Roof ratio for 1st built' is set to 1, the following sequence for the staged construction analysis will be applied.

Stage	Description	Note
No. 1	Annular part	
No. 2	2) + Base 1st PS	
No. 3	3) + Circular part	
No. 4	4) + Base 2^{nd} PS	
No. 5~6	5) + Wall & Ringbeam is added up in stages	
No. 7	6) + Wall End 1 st PS	
No. 8	7) + Ringbeam 1 st PS	
No. 9	8) + Roof Frame 1	
No. 10	9) + Inner Tank Work	
No. 11	10) + Roof Frame 2	
No. 12	11) + Roof Frame 3	
No. 13	12) + Roof Lower Wet Concrete	
No. 14	13) + Roof Lower Complete	
No. 15	14) + Roof Lower Inside Load	
N0. 16	15) + Wall End 2 nd PS	
No. 17	16) + Ringbeam 2 nd PS	
No. 18	17) + Roof Upper Wet Concrete	
No. 19	18) + Roof Complete	
No. 20	19) + Vertical PS	

• Layered roof option 2

Seismic Analysis

Stage	Description	Note
No. 21	20) + Horizontal PS	
No. 22	21) + Final Short term	
No. 23	22) + Final Long term (Long term PS applied)	

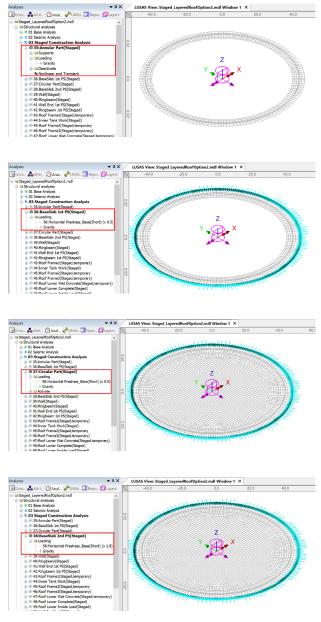


Fig 79 Stage 1 ~ 4 of Staged Construction Analysis

Stage 1 builds the annular part of the slab.

Stage 2 adds the 1st PS for the Base Slab

Stage 3 the central part of slab.

Stage 4 adds the 2nd PS for the Base Slab.

Stage 5 models the construction of the first lift of the Wall.

Note that self weight is always assigned when a new part of the structure is added. Loading defined and assigned at a stage is inherited by the subsequent stages.

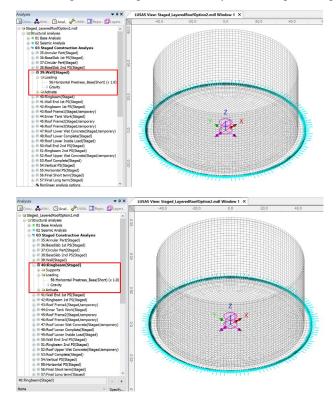


Fig 80 Stage 5 ~ 6 of Staged Construction Analysis

At Stage 6, the Wall and RingBeam construction is complete.

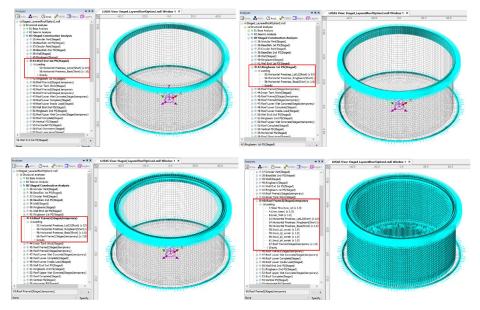


Fig 81 Stage 7 ~ 11 of Staged Construction Analysis

At **Stage 7**, 50% of Wall end 1st prestress is assumed to be applied (A Load factor of 0.5 was used for the Horizontal Prestress Wall End 1st load).

At **Stage 8**, 50% of RingBeam prestress is assumed to be applied if the "**Initial prestress for Ringbeam (ratio)**' is set to 0.5. (A Load Factor of 0.5 was used for the Horizontal Prestress_Ringbeam load.) If "**Initial prestress for Ringbeam (ratio)**' is set to a different value then the Ringbeam prestress at **Stage 8** will have a different load accordingly.

Stages 9, 11 and **12** assume that there could be temporary loads on the line where the roof and RingBeam are connected. The loadings for Roof Frame 1, Roof Frame 2 and Roof Frame3 are defined using user inputs. The user needs to input total loading and Modeller will automatically convert this to the equivalent load per unit length according to the length of line the loading is assigned to.

·			
Attributes 🔷 🕈 Attributes	Global Distributed		×
Groun Antrin Q Anal. A Utiliti. Repo. Layers 52:Horizontal Prestress_Lot12(Short) 4 52:Horizontal Prestress_Lot12(Long) 54:Horizontal Prestress_Ringbeam(Short) 55:Horizontal Prestress_Ringbeam(Long)	Analysis category 3D		
- ♣ 56:Horizontal Prestress_Base(Short) - ♣ 57:Horizontal Prestress_Base(Long) - ♣ 58:Insul q1 constr	⊖ Total	Per unit length Per unit are	ea.
\$ 59:Insul q2_constr	Component	Value	
& 60:Insul q3 constr	X Direction	60.4851E3	
& 61:Insul_q4_constr	Y Direction	0.0	
	Z Direction	-34.921E3	
	Moment about X axis	0.0	_
	Moment about Y axis	0.0	
	Moment about Z axis	0.0	(
& 66:Roof Frame1(Staged,temporary)			
-& 67:Roof Frame2(Staged,temporary -& 68:Roof Frame3(Staged,temporary)			
69:Roof Lower Wet Concrete(Staged,temporary)			
□ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □			
☐			
-\$ 1:LocalCoord			
- A 3:Wall Local			
🖃 🖳 Spherical (1)			
2:Spherical for ForceExtraction			
😑 🔤 Activate elements (1)	Name Dec Com		(66)
🖧 1:Activate	Name Roof Fran	ne1(Staged,temporary) ~	(66)
😑 🔤 Deactivate elements (3)			
- S 1:Deactivate			
- & 2:Deactivate(Roof ring)		Close Cancel Apply	Help
3:Deactivate(Roof center)	_	Close Cancel Apply	help

Fig 82 Roof Frame1 Loading

As these loadings are not permanent loading, they are not inherited by the subsequent stages and are marked as '**temporary**' in the loadcase name.

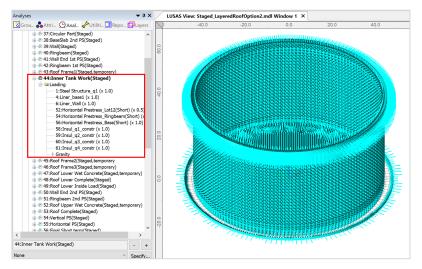


Fig 83 Stage 10 of Staged Construction Analysis

Stage 10 assumes that the inner tank has been built. All insulation loading except for 'Roof Liner' will be defined and assigned at this stage.

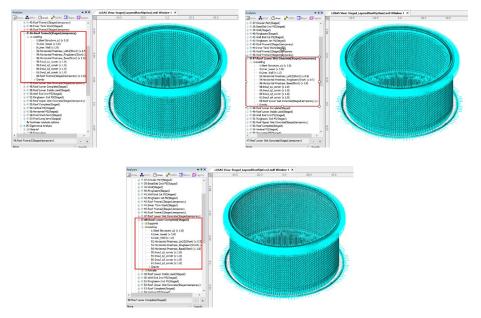


Fig 84 Stage 12-14 of Staged Construction Analysis

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

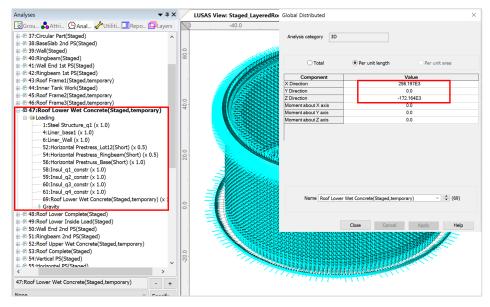


Fig 85 Stage 13 of Staged Construction Analysis

Stage 14 assumes that the lower half of the roof is built (if Roof ratio for 1^{st} built = 0.5 from **Tank Definition**), and the lower part of the roof is newly activated. As shown below, the geometric properties used represent those for only half of the Roof at **stage 14**, only becoming geometric properties for the whole roof at **stage 19**.

Stage 14 assumes that the lower part of roof is completed. At this stage the wet concrete loading assigned at **Stage 13** is removed and replaced with the body force of the lower part of Roof.



Fig 86 Roof Shape at Stage 14 and Stage 19.

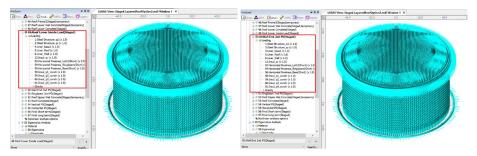


Fig 87 Stage 15 ~ 16 of Staged Construction Analysis

At Stage 16, 50% of additional Prestress for Wall End lot is added. (The Load Factor is updated from 0.5 to 1.0 for Horizontal Prestress_12LOT(Short) load.)

At Stage 17, 50% of additional RingBeam Prestress is added. (The Load Factor is updated from 0.5 to 1.0 for Horizontal Prestress_Ringbeam load.)

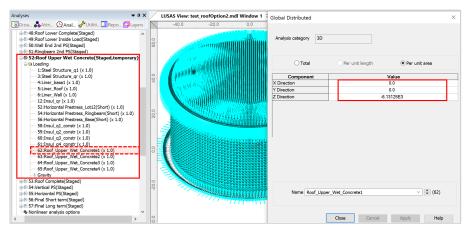


Fig 88 Stage 18 of Staged Construction Analysis

Stage 18 models the upper half of the Roof being built with the poured concrete acting as a load on the already cast lower half of the Roof.

Stage 19 assumes that the upper part of the Roof is now built. The wet concrete loading assigned at **Stage 18** is removed and replaced with the body force of the upper part of the Roof.

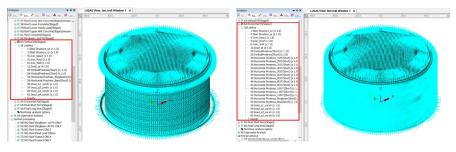


Fig 89 Stage 18 ~ 19 of Staged Construction Analysis

At Stage 20, the vertical prestress is added.

At Stage 21, all the horizontal prestress for the Wall is added.

Stage 22 is the final stage. The structures are complete as built, and all loadings for the operating condition are added.

Stage 23 models long-term effects. The prestress values are updated to those for long-term PS.

If 'Roof ratio for 1st built' is set to 1, the following sequence for the staged construction analysis will be applied.

Stage	Description	Note
No. 1	Annular part	
No. 2	1) + Base 1 st PS	
No. 3	2) + Circular part	
No. 4	3) + Base 2 nd PS	
No. 5~16	4) + Wall & Ringbeam is added up in stages	
No. 17	16) + Ringbeam 1 st PS	
No. 18	17) + Roof Frame 1	
No. 19	18) + Inner Tank Work	
No. 20	19) + Roof Frame 2	
No. 21	20) + Roof Frame 3	
No. 22	21) + Roof Wet Concrete	
No. 23	22) + Roof Complete	
No. 24	23) + Roof Lower Inside Load	
No. 25	24) + Ringbeam 2 nd PS	
No. 26	25) + Vertical PS	
No. 27	26) + Horizontal PS	
No. 28	27) + Final Short term	
No. 29	28) + Final Long term (Long term PS applied)	_ 1)

Table 3 Sequence of construction stages (Roof ratio for 1st built = 1)

User Updates

Construction Sequence

If required, additional loadings or stages can be added.

To duplicate loading types within the Analyses \bigcirc treeview, the Copy and Paste options can be used as shown below. This will create additional stages (loadcases) and include all loadings previously assigned to the copied loadcase.

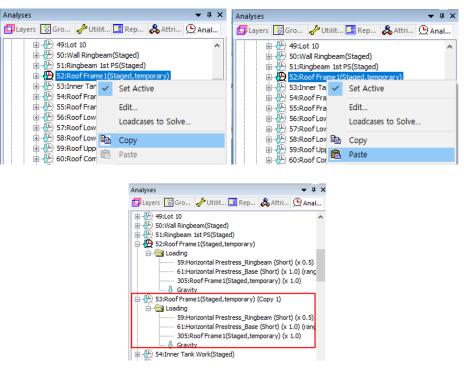


Fig 90 Copy and Paste of Stages (Loadcases)

The required activation/deactivation/loadings/support can now be assigned or removed for this stage. The loadings at other loadcases can be also copied and pasted in the same way if required.

User Updates

Loadings

As discussed, at in the section titled *Staged Construction Analysis*, some loadings may need updated.

Construction Schedule

The duration (length of time) of each stage is set to 10 days by default for all stages. This should be updated to follow the actual construction schedule.

Each stage uses a Nonlinear Control, and both the time and Total Response Time should be updated together.

For example, if the time gap between Stage 6 and Stage 7 needs to be changed to 15 days, Nonlinear Control for Stage 6 should be updated.

Default settings for Stage 6 are shown below.

Analyses 🗸 🗘 🗸	Nonlinear & Transient			
🔞 Grou 💑 Attri 🕒 Anal 🦨 Utiliti 💷 Repo 🗇 Layers	Noniniear oc mansiene			
Activate	Incrementation Solution strategy			
Nonlinear and Transient	✓ Nonlinear Same as previous loadcase			
⊟ To:BaseSlab 2nd PS(CRSH)	Incrementation	Manual ~	Max number of iterations	12
🗄 😑 Loading				
Nonlinear and Transient	Starting load	0.1	Residual force norm	0.1
⊟	Max change in load factor	0.0	Incremental displacement	0.0
🗄 🖨 Loading	Max change in load factor		Incremental displacement	
Activate Nonlinear and Transient	Max total load factor	1.0		Advanced
→ Nonlinear and Transient	Adjust load based on o	pyorgopco		
Supports		unvergence	Incremental LUSAS file output	
	Iterations per increment	4	Same as previous loadcase	
	Displacement reset			-
S Nonlinear and Transient	Displacement reset		Output file	1
🖃 🕾 73: Ringbeam 1st PS(CRSH)		Advanced	Plot file	1000
🕀 😑 Loading	Time domain			
Nonlinear and Transient		Viscous ~	Restart file	0
⊟ 74:Roof Frame1(CRSH,temporary)			Max number of saved	0
🕀 🚍 Loading	Initial time step	10.0	Hax humber of saved	
Nonlinear and Transient	Total response time	60.0	Log file	1
area 75:Inner Tank Work(CRSH)			Libbar Cla	1
Nonlinear and Transient	Automatic time steppin	g	History file	L
		Advanced		
Loading		Auvanceu	Save a restart at the end of this	s control
Nonlinear and Transient	Common to all			
⇒ 77:Roof Frame3(CRSH,temporary)	Frame3(CRSH,temporary) Max time steps or increments 100000			
🕀 😑 Loading	Hax une steps t			
Solution Nonlinear and Transient				
Roof Wet Concrete(CRSH,temporary)			OK Cancel	Help
< >				

Fig 91 Nonlinear Control for Stage of Ringbeam (Default)

- **Initial Time Step :** Analysis is performed at every 10 days.
- **Total Response Time :** This stage lasts up to 60 days from the start of 1st Stage.

The number of days that the current stage lasts for is the <u>Total Response Time of</u> current stage minus <u>Total Response Time of the previous stage</u>.

By modifying **Total Response Time** to **65**, this stage lasts for 5 more days. (e.g. a total of 15 days)

Note that the Total Response Time for the subsequent stages should be also updated. Otherwise, the 7th Stage will last only 5 days.

Modifying **Initial Time Step** is optional and depends on the accuracy required. With a smaller time step, the creep and shrinkage material properties are updated frequently (e.g. using smaller time gaps) hence the accuracy would increase, however the solution time will increase accordingly.

Tip

The unit of time is set on the Model Properties dialog by selecting the **File > Model Properties** menu item.

Properties	×			
General Backups Geometry Meshing Attributes Options C · · Title Analysis category 3D · Precision shown in dialogs Only show compatible options O Decimal places				
Model units N,m,kg,s,C Output in feet and inches Timescale units Days	Vertical axis X axis Y axis Y axis Gravity m/s ² -9.81			
Decimal marker As Windows V				
Close Cance	I Apply Help			

Fig 92 Model Properties

Age

The Wizard built model assumes that each member is activated at the same concrete Age. The Age property represents the concrete age at the time of activation. (i.e. the time gap between pouring the concrete and removal of formwork.) If a different age is required for some members, another Age attribute should be defined and assigned manually for those.

Design Load Combinations

Template for Design Load Combinations

The template for Design Load Combination is saved in the current working folder where the Base Model was built, with the name [Model name]_ComboTemplate.xlsx, as shown below. The template can also be downloaded from the Design Load Combination dialog.

Load Factors Worksheet

1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	Loa	dc	ase	e in	dex	(7	28	29	30	31	32
							Permanent Variable																								
	De	scription		Outer tank		Others		ep and inkage		P	restress				Ro	of frame/	concrete			Test		NG Isure		as isure	Live Load	Snow Load	Temp	earture	Wind	0	8E
no.	Code	Details	Outer tank WO roof	Outer tank WO uper roof	Outer tank Full	Others	s Early	Late	Rb 1st	Rb All	Rb + Vertical	AI PS Early	AI PS		Roof Frame	Roof Frame 3	1st laye	r 2nd lay	er Hydro e atic	st Pneum tic	LNG Max	LNG Min	Gas Max	Gas Min	Live Load (roof)	Snow Load (roof)	Temp Max	Temp Min	Wind	OBE Hor.	OBE Vert.
1	U-C1-1		1.35			-	<u>+</u> -	-	1.30				- 1	1.77	H-1		1 1-7	414	+ -				-							-	
2	U-C1-2	WO_roof +	1.35						1.00																						
3	U-C1-3	RB_1st_PS	1.00				L		1.30																						
4	U-C1-4								1.00												÷ -	-	-	_							
5	U-C2-1		i Co	de a	nd I	Deta	ails		1.30					1.50						l Loa	d f	act	orc								
6	U-C2-2	WO_roof + RB_1st_PS +	O_roof +						1.00					1.50							u i	act	013	1							
7	U-C2-3	Roof_frame_1	1.00						1.30					1.50									T -								
8	U-C2-4		1.00						1.00					1.50																	

Fig 93 Template for Design Load Combinations, Load Factors

- □ Loadcase Index The numbers in this row are used to match a Modeller loadcase number with a loadcase and its associated details in this template. The row should contain a series of numbers with no duplication.
- **Code** and **Details** Code and Details are used for naming the combination data in Modeller.
- **Load Factors** Load factors for each loadcase is defined here.
- **Others** All other data are for users reference, and not used for processing.
- □ Loadcase to consider can be added. (more columns as necessary can be added.)
- □ Load combinations can be added. (more rows as necessary can be added)

Loadcases Worksheer

A	В	C	D	E	F	G	H	1.1	J	K		L	M	
LC No.	Loadcase Name	Column	Load Category											
1	SelfWeight	6	Outer tank Full	L	User Guide for 'Loadcases' sheet 1. Loadcase Name (Column B)									
2	Dead Loads of Steel Structure	7	Others	1										
3	Dead load of liner and steel roof	7	Others		- Should be identical with loadcase name in the model									
4	Dead load of steel structures on the roof	7	Others	2	LF col	umn (Colum	n C, Load Fa	ctor Column)						
5	Dead load of Insulation	7	Others		Refer	to the numb	er at 1st rov	of 'LoadFact	ors' sheet					
6	Pressure on outer tank wall due to insulation	7	Others		- Put 0	if the loadca	ise is not use	d in the com	bination.					
7	Wall piping loading	7	Others	3	. Note									
8	Dead load of Insulation Constr	0			- Loadc	ases not to	be used in th	e combination	ns can be rer	noved fron	n 'Load	icases' sh	neet.	
9	Liquid bottom(Max)	22	LNG Max		- Loado	ases can be	added at any	row, as man	y as required	l.				
10	Liquid bottom(Min)	23	LNG Min											
11	Liquid wall (Max)	0		L	User Guide for 'LoadFactors' sheet 1. LF column index									
12	Liquid wall (Min)	0		1										
13	Gas Pressure(Max)	24	Gas Max		- The top line should be maintained with unique number 2. Code Name / Details									
14	Gas Pressure(Min)	25	Gas Min	2										
15	Live load	26	Live Load (roof)		- This columns are used for defining the name of combination. 3. Note									
16	Snow load	27	Snow Load (roof)	3										
17	Test load (Liquid)	20	Hydrostatic		 Row 2-4 are for user's reference only, and free to update. Combination data should start from the 5th row. Loadcases to be factored can be added as many as requiried. 									
18	Test load (Pneumatic)	21	Pneumatic											
19	Prestress (Short)	13	All PS Early											
20	Prestress (Long)	14	All PS Late		- Comb	inations can	be added as	many as req	uired.					

Fig 94 Template for Design Load Combinations, Loadcases

- □ Loadcase Name The loadcase names defined in Modeller. The loadcase number may change during the process of updating the model, so the loadcase name is used in the definition of the Load Combination. Note that the loadcase names used must be the same as the loadcase names defined in Modeller.
- □ (LF) Column This column is used to match a Modeller loadcase to a corresponding loadcase on the LoadFactors worksheet by entering Loadcase Index on the LoadFactors worksheet. For example, the <u>Self Weight</u> loadcase in Modeller is used in the <u>'Outer Tank Full'</u> combination defined in LoadFactors sheet. By entering 0 (zero), the loadcase is ignored and will not be used in the combination.
- **Others** All other data are for users reference only, and not used for processing.

Update Base Model

The template is imported into Modeller by selecting the LNG Tank > Design checks> Design Load Combination...

LNG Tank - Design Load Combinations	×
	Template Download
Combination data	Browse,
	OK Cancel Help

Fig 95 Dialog for Design Load Combination

- **Template Download** Downloads the template to be used for creating load combinations to the current working folder.
- **Combination data** Select the load combination template.

On re-loading this template, the **Code** and **Details** columns in the **LoadFactors** worksheet will be compared with combination names in Modeller and the load factors will be updated. If any new combinations are present, they will be added. However, any existing combinations will not be deleted.

• # × Combination × 🖾 Layers 🔯 Grou... 歳 Attri... 🕒 Anal... 🥓 Utilit... 🔝 Repo Comparing the second seco Method Factored Included - Post processing - 96:WO Roof Base PS ONLY ^ ID Na Factor WO Roof Base PS ONLY WO Roof Tank ONLY WO Roof RingBeam 1st PS ONLY 96:W0 Roof Base PS ONLY 97:W0 Roof Trank ONT 98:W0 Roof RingBean 1st PS ONLY 99:W0 Roof RingBean 1 JF SONLY 101:W0 Roof Frame 2 ONLY 101:W0 Roof Frame 2 ONLY 101:W0 Roof Frame 2 ONLY 102:W0 Roof Frame 2 ONLY 102:W0 Roof Frame 2 ONLY 96 97 1.3 1.35 1.3 PROVINC ROOF TAINE ONLY
 PRIVIC Roof RingBeam IST PS ONLY
 P9:WO Roof RingBeam All PS ONLY
 100:WO Roof Frame1 ONLY
 101:WO Roof Dead Load Others
 102:WO Roof Frame2 ONLY >> 103:WO Roof Frame3 ONLY 104:WO Roof Wall and Lower Roof 105:Roof Lower Wet Concrete ONLY
 105:Roof Lower Wet Concrete ONLY
 105:Roof Upper Wet Concrete ONLY
 107:RingbeamPS Vert caPS ONLY
 108:CRSH Early ONLY << 105:Roof Lower Wet Concrete ONLY 106:Roof Upper Wet Concrete ONLY Æ 107:RingbeamPS VerticalPS ONLY 108:CRSH Early ONLY 109:CRSH Late ONLY of + RB 1st PS 112:U-C1-1_Tank WO roof + RB 1st PS 113:U-C1-2_Tank WO roof + RB 1st PS 114:U-C1-3_Tank WO roof + RB 1st PS 114:U-C1-3_Tank WO roof + RB 1st PS 115:U-C1-4_Tank WO roof + RB 1st PS Step 1 114:U-C1-3_Tank WO roof + RB 1st PS ÷ < 115:U-C2-1_Tank WO roof + RB 1st PS + Roof?frame 116:U-C2-1_Tank WO roof + RB 1st PS + Roof?frame 117:U-C2-2_Tank WO roof + RB 1st PS + Roof?frame 118:U-C2-3_Tank WO roof + RB 1st PS + Roof?frame 119:U-C2-4_Tank WO roof + RB 1st PS + Roof?frame Name U-C1-1_Tank WO roof + RB 1st PS ~ 🗘 (112) 110:Envelope (Max) 111:Envelope (Min) Close Cancel Apply Help

The design load combinations are created as shown below.

Fig 96 Load Combinations Created in Modeller

Design Check

COP Parameters

Design code parameters can be defined by the selecting the menu item LNG Tank > Design Checks > Enable...

esign parameters			
Design code		EN1992-1-1 (2005) ~
General			
Partial factors for materials		Persistent/Transier	nt ~
Steel			
Yield stress of tendon (fyp)		400.0	MPa
Elastic modulus of tendon (Ep)		200.0E3	MPa
Concrete			
Roof concrete grade		40.0	MPa
Wall concrete grade		50.0	MPa
Slab concrete grade		40.0	MPa
Long term effect coeff. (a_cc)		1.0	
Computation target for visualization			
Angle (e.g 0;90;100)	⊖ Selected	◯ Visible	
✓ fse is used and PS is being applied as an external lo	ading (Shift will b	e applied to results)	
Exclude base slab results at pile heads and walls			
Assumed diameter at crosswise piles	0	[m]	
Assumed diameter at circumferential piles	0	[m]	

Fig 97 Dialog for Design Parameters – EN1992-1-1 (2005)

- Partial Factors for Materials: The partial factors for materials are defined based on Table 2.1N on EN1992. For Persistent & Transient, 1.5, 1.15, 1.15 are given to γ_c for concrete, γ_s for reinforcing steel, γ_s for prestressing steel respectively. For Accidental, 1.2, 1.0, 1.0 are given.
- **Vield Stress of Reinforcement (fy)**: Yield strength of reinforcing steel in MPa.

- **Vield Stress of Tendon (fyp)** : Yield strength of prestressing steel in MPa.
- **Elastic Modulus of reinforcement (Es) :** Elastic Modulus of reinforcing steel in MPa.
- □ Elastic Modulus of Tendon (Ep) : Elastic Modulus of prestressing steel in MPa
- **Roof concrete grade :** Concrete strength in MPa.
- □ Wall concrete grade : Concrete strength in MPa.
- **Base concrete grade :** Concrete strength in MPa.
- **Long term effect coeff.** (α_{cc}) : Long term effect coefficient (EN1991 only)
- □ Max Concrete Compressive Strain : Ultimate strain at concrete failure (ACI 318-14 only)
- **Vield Stress of Reinforcement (fy)** : Yield strength of reinforcing steel in MPa.
- **Yield Stress of Tendon (fyp)** : Yield strength of prestressing steel in MPa.
- □ Tensile strength of tendon (fpu) : Tensile strength of prestressing steel in MPa. (ACI 318-14 only)
- □ Computation Target : The design check computations will be performed for the targets of
 - **Default**: The node at Y=0 and X>=0.
 - Selected: The nodes that user selected before opening this dialog
 - Visible: All visible nodes in Modeller.
- □ fse is used and PS is being applied as an external loading
- When 'fse' is specified in the reinforcement template and the PS being applied as external loading, the code-checking that creates the PM chart will <u>double count</u> the PS effect.
- Ticking this option shifts the results to avoid double counting the PS effect.
- When a loadcase (or load combination) does not have PS as external loading, this option should be disabled (unticked).
- It makes no difference if 'fse' is not specified in the reinforcement template.

LNG Tank - Design Code				×
Design parameters Design code			ACI318_14	~
Steel Yield stress of tendon (fyp) Tensile strength of tendon (fpu) Elastic modulus of tendon (Ep)			400.0 1.86E3 200.0E3	MPa MPa MPa
Concrete Roof concrete grade Wall concrete grade Slab concrete grade Max concrete compressive strain (e_cu)			40.0 50.0 40.0 3.0E-3	MPa MPa MPa
Computation target for visualization Angle 0 (e.g 0;90)	;100)	Selected	◯ Visible	
☑ fse is used and PS is being applied as an	n external loadi	ng (Shift will be	applied to results)	
Exclude base slab results at pile heads a Assumed diameter at crosswise pile: Assumed diameter at circumferential	s	0	[m] [m]	
	Defaults	ОК	Cancel	Help

Fig 98 Dialog for Design Parameters – ACI318_14

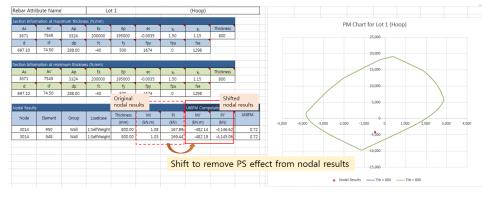
- **Vield Stress of Tendon (fyp)** : Yield strength of prestressing steel in MPa.
- □ Tensile strength of tendon (fpu) : Tensile strength of prestressing steel in MPa. (ACI 318-14 only)
- □ Max Concrete Compressive Strain : Ultimate strain at concrete failure (ACI 318-14 only)

LNG Tank - Design Code	×
Design parameters	
Design code	GB50010-2010 ~
General	
Loading conditions	Persistent/Transient ~
Significance coefficient (γ_0)	1.0
Uncertainty coefficient (γ_Rd)	1.0
Stability coefficient (ϕ)	1.0
Steel	
Prestressed reinforcement type	Strand1860 ×
Area of shear reinforcement per surface area [mm²/m²]	0.0
Concrete	C40 ~
Roof concrete grade	C50 ~
Wall concrete grade	
Slab concrete grade	C40 ~
Advanced	
Axial stress tolerance (% of fc)	1.0
Computation target for visualization	
Angle 0 (e.g 0;90;100) Oselected	◯ Visible
✓ fse is used and PS is being applied as an external loading (Shift will be Exclude base slab results at pile heads and walls Assumed diameter at crosswise piles Assumed diameter at circumferential piles	[m]
Assumed diameter at circumierential piles 0	[m]
Defaults OK	Cancel Help

Fig 99 Dialog for Design Parameters - GB5001-2010

- □ Loading conditions : This is used to identify if the design values or the characteristic values of the materials should be used, in accordance with clause 3.3.4;
- □ Significance coefficient(γ_0) : In line with clause 3.3.2, the design loads are multiplied by the significance coefficient of the structure;
- □ Uncertainty coefficient (γ _Rd)t : In line with clause 3.3.2, the design resistance should be divided by the uncertainty coefficient. Note that when "Seismic" is selected in the loading conditions droplist, this is renamed to "Seismic adjustment coefficient (γ _RE)

- **Stability coefficient**(φ) : This is defined in clause 6.2.15 and is used for the calculation of the pure compression capacity in accordance with that same section;
- □ Conventional reinforcement type: Based on the reinforcement type selected, the elastic modulus and yield strengths of the conventional reinforcement are calculated
- □ **Prestressed reinforcement type** : Based on the prestressed reinforcement type selected, the elastic modulus and yield strengths of the tendons are calculated
- □ Area of shear reinforcement per surface area(mm²/m²):
- □ Concrete grade for roof/wall/slab: This was added, since [C1] defines the design concrete strength (Table 4.1.4-1) based on the concrete grade
- □ Axial stress tolerance (% of fc):



- Shifted force, Pi' = Pi fse * Apd
- Shifted moment, Mi' = Mi fse * Apd * ecc_ps

Design Checks for Tank

Once design code parameters have been defined and the OK button is pressed a **Tank Design** Entity will then be available for selection in relevant result processing dialogs. The associated available components are shown below.

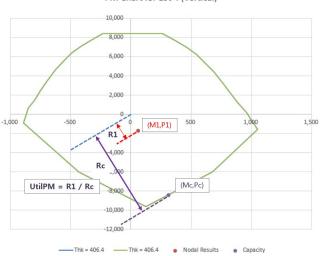
Layers	Properties Contour Results Appearance	×
Example.mdl Window 1 Geometry Deformed mesh Attributes G Utilities G Mesh O' Annotation O' Contours : None	Entity Tank Design - BS EN 195 ~ Component UtilPM_t Display UtilPM_rz Transform Pcapacity_t Pcapacity_rz Mcapacity_t	
- % View properties	Mcapacity_rz UtilDC_tz UtilDC_rz UtilShear_t Draw in s UtilShear_rz ShearResist_tz ComDepth_t ComDepth_t	Help

Fig 100 Components for Design Checks

Whenever a results component is chosen, or a loadcase is set active, the design check for a selected component will take place. Design checks are carried out on a node-bynode basis using analysis results and relevant design code formulae.

UtilPM (PM Utilization)

UtilPM can be checked with reference to a PM chart, as illustrated below. A value of less than 1 means it satisfies the design code.



PM Chart for Lot 4 (Vertical)

Fig 101 Definition of UtilPM

- □ UtilPM_t is the force and moment utilization in the hoop direction, and UtilPM_rz is the utilization in the radial direction for the Roof and Base Slab and for the vertical direction for the Wall.
- Pcapacity_t and Pcapacity_rz is for the computed Pc at the given P1/M1 slope, Mcapacity_t, Mcapacity_rz is for the Mc at the given P1/M1 slope.

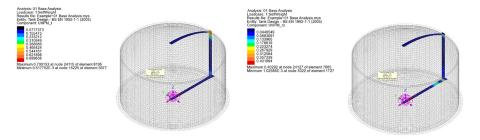


Fig 102 Contour for UtilPM_t, UtilPM_rz

UtilDC (Decompression Utilization)

UtilDC aims to check if 25mm of concrete around a tendon is in compression. <u>UtilDC</u> is only available if **EN1992** is chosen, and a value of less than 1 means it satisfies the design code.

The UtilDC calculation assumes the stress distribution through the thickness is linear.

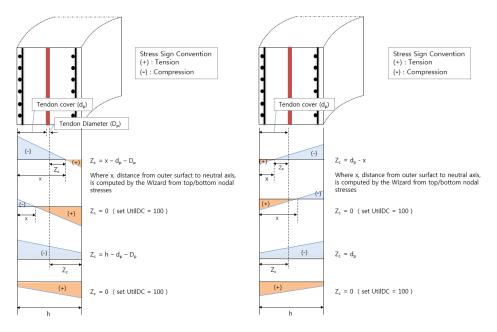


Fig 103 Zc for Decompression Check

The tendon cover (d_p) is the distance from outer surface to the tendon surface.

When the section is fully in compression the maximum value is set to 100. (ie. if Zc<=0.25mm, UtilDC becomes 100.)

UtilDC = Zdec /
$$Zc = 25$$
 / Zc .

□ UtilDC_t is the tendon and stress utilization in the hoop direction, and UtilDC_rz is the utilization in the radial direction for the Roof and Base Slab, and in the vertical direction for the Wall.

UtilDC is only available for sections that contain prestress tendons. In the hoop direction, UtilDC_t is available for Wall and Slab. In the radial and vertical direction, UtilDC_rz is available only for Wall.

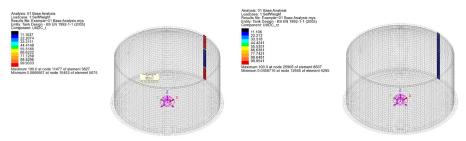


Fig 104 Contours for UtilDC_t, UtilDC_rz

If concrete at the prestress tendon location is in tension UtilDC is set to 100.

ShearResist (Shear Resistances)

Shear capacity for the tank components is based on concrete shear resistance as per COP specifications.

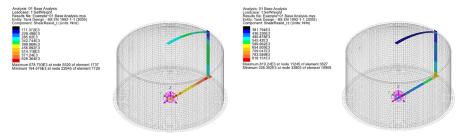


Fig 105 Contour for ShearResist_t, ShearResist_rv

UtilShear (Shear Utilization)

Shear utilisation factors are given as the absolute ratio between the shear forces (Sp, St, Sz, Sr) and the relevant shear resistances (ShearResist_t and ShearResist_rz). A value for UtilShear > 1 denotes failure in shear.

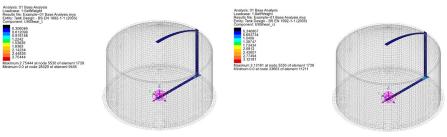


Fig 106 Contour for UtilShear_t, UtilShear_rv

PM Chart Report

A spreadsheet report that includes a PM chart can be produced by selecting the menu item LNG Tank > Design Checks> Design Check Report ...

	DesignCheckReport			
eport filename	Designeneekkepon			
/orking folder	Current	O User Defined		
Vorking lolder		0.000		
ave in	C:\Users\ohsso\Docume	ents\Lusas200\Projects\De	signCheckReport_[Angle].xlsx	
Target				
Angles 20	Group All	~	O Selected O Visible	
degree (eg.	10; 20; 30)			
Nodal Averaging		Exclude base sla	b results at pile heads and walls	
Unaveraged element	results	Assumed diameter	at crosswise piles 0.7	[m]
Averaged results from	all elements	/ tobulied diameter		tend.
-		Assumed diameter	at circumferential piles 0.8	[m]
 Averaged results from 	n visible elements			
PM Check Report	Shear Chec	k Report	Utilization Summary Report	
Peak element onl	y Pea	k element only	Peak element only	
Peak loadcase or	nly Pea	k loadcase only	Peak loadcase only	
Loadcases / Combinations o	•	tions only		
1:SelfWeight		tions only	PM Chart Type	
2:Dead Loads of Steel Strue	cture	<u>^</u>		
3:Dead load of liner and ste		↑ Y axis	5	
4:Dead load of steel structu 5:Dead load of Insulation	res on the roof			
6:Pressure on outer tank w	all due to insulation		🦼 Slicing Line	
7:Wall piping loading				
8:Liquid bottom(Max) 9:Liquid bottom(Min)			Angle (Positive Dir	ection
			X axis (0 Degree)	
10:Gas Pressure(Max)				
10:Gas Pressure(Max) 11:Gas Pressure(Min)				

Fig 107 Dialog for Design Check Report with PM Chart

□ **Report Target** The design check computations will be performed for the targets of

- **Default**: The node at Y=0 and X>=0.
- Selected : Any nodes that were selected before opening this dialog
- Visible: All visible nodes in Modeller.

If 'Default' is selected for Report Target, pre-defined target nodes will be used for each of the 15 types of different reinforcement arrangements in the current model, and the

 Design Clock with PM duart
 01192

 Design Clock with PM duart
 01192

 Design Clock with PM duart
 0011

 Design Clock with PM duart
 0011

report will contain 30 worksheets for producing PM charts for two directions (hoop/vertical or hoop/radial) for all 15 rebar arrangements.

Fig 108 Design Check Report with PM Chart for Lot 1

- □ Section Information at maximum thickness (1) The maximum thickness from the selected nodes is printed, for which the PM chart is displayed.
- □ Section Information at minimum thickness (2) The minimum thickness from the selected nodes is printed, for which the PM chart is displayed.
- □ Node Node number for UtilPM computation
- **□** Element Element numbers sharing the node
- **Group** Group name where the node is included in.
- **Thickness** Section thickness at the node location
- □ Mi Bending moment at the node
- **Pi** Axial force at the node
- □ Mi', Pi' The shifted Mi, Pi when 'fse is used and PS is being applied as an external loading' option is ticked from the Design Code dialog.
- **UtilPM** PM Utilization
- □ PM Chart Two PM Charts are presented; one for the maximum thickness section, the other for minimum thickness section of the selected nodes. The value for Pcapacity in Modeller has different sign from the value of Pcapacity stated in the PM report

Design Checks for RC Slab

Design checks for RC Slab is for concrete crack checks. By selecting the **Design>RC Slab Design** menu item, followed by an appropriate design code, a number of other design checks are available. For more information, please refer to the LUSAS user manual.

-									>
RC Slab/Wall I	Design								
Country	UK								\sim
Design code	BS E	N 1992-1	-1:2004	/NA:	2005				\sim
Partial factor	s for materi	als							
Concrete (yo	•)		1.5						
Reinforcing	steel (γs)					1.15			
Coefficient fo	or lona term	effects f	or conc	rete					
αcc	j					0.85			
ucc									
Defaults		C	Ж		Car	icel		Help	
		С	Ж		Car	icel		Help	×
Properties			ЭК		Car	icel		Help	×
Properties	ts Appeara		ж		Car	icel		Help	×
Properties Contour Resul		ance		~	Car	Icel		Help	×
Properties Contour Resul Entity	RC Slab/W	ance		~	Car	icel		Help	×
Properties Contour Resul	RC Slab/W Util(Max)	ance		~	Car	icel		Help	×
Properties Contour Resul Entity	RC Slab/W	ance		× ×	Car	icel		Help	×
Properties Contour Resul Entity Component	RC Slab/W Util(Max) Util(Max) Util(ULS) Util(Asmin	ance Vall - BS		~	Car	icel		Help	×
Properties Contour Resul Entity Component Display	RC Slab/W Util(Max) Util(Max) Util(ULS) Util(Asmin Util(SLS)	ance Vall - BS		~	Car	icel		Help	×
Properties Contour Resul Entity Component Display	RC Slab/W Util(Max) Util(Max) Util(ULS) Util(Asmin Util(SLS) Util(CC)	ance Vall - BS ,Asmax)		~	Car	icel		Help	×
Properties Contour Resul Entity Component Display Transform	RC Slab/W Util(Max) Util(ULS) Util(Asmin, Util(SLS) Util(CC) Util(SLS.S) Util(SLS.S)	ance Vall - BS ,Asmax)		~	Car	icel		Help	×
Properties Contour Resul Entity Component Display Transform	RC Slab/W Util(Max) Util(Max) Util(ULS) Util(Asmin, Util(SLS) Util(CC) Util(SLS.S) Util(AsCC)	ance /all - BS ,Asmax))		~	Car	Incel		Help	*
Properties Contour Resul Entity Component Display Transform	RC Slab/W Util(Max) Util(ULS) Util(Asmin Util(SLS) Util(SLS.S) Util(AsCC) Assh Util(Shear)	ance Vall - BS ,Asmax))		~	Car	Incel		Help	×
Properties Contour Resul Entity Component Display Transform	RC Slab/W Util(Max) Util(Max) Util(ULS) Util(Asmin, Util(SLS) Util(CC) Util(SLS.S) Util(AsCC)	ance Vall - BS ,Asmax))		~	Car			Help	*

Fig 109 Dialog for RC Slab Design and Result Components (1)

Properties				×
Contour Resul	ts Appearance			
Entity	RC Slab/Wall - BS EN 19	\sim		
Component	Util(Max)	\sim		
Display	Util(Max)(T) Util(Asx(T))	^		
Transform	Util(Asy(T))			
Display o	Ssx(T) - ULS Ssy(T) - ULS Fc(T) - ULS Sc(T) - ULS Util(Sc(T)) - ULS Util(TCx(T)) Util(TCx(T))			
	Util(MinAsx(T)) Util(MinAsv(T))		Apply	Help

Fig 110 Dialog for RC Slab Design and Result Components (2)

LUSAS

TITI

LUSAS, Forge House, 66 High Street, Kingston upon Thames, Surrey, KT1 1HN, UK Tel: +44 (0)20 8541 1999 | Fax: +44 (0)20 8549 9399 | info@lusas.com | www.lusas.com