

LNG Tank System User Manual

Metallic Tank - Part 1 - Tank Modelling

LNG Tank System User Manual (Metallic Tank) Part 1 – Tank Modelling

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

LNG Tank Modelling	1
Overview	1
2D Axisymmetric Static Structural Analysis	3
2D Axisymmetric Construction Stage Analysis	5
2D Axisymmetric Thermal Analysis	8
3D Shell Static Structural Analysis	
3D Shell Eigenvalue Analysis	31
2D Beam-Stick FSSI Seismic Analysis	33
Exporting Forces from the 2D Axisymmetric Model	48
Exporting Forces from the 3D Shell Model	52
Examples – User Inputs	55
Tank Definition	55
2D Axisymmetric Static Structural Analysis	86
2D Axisymmetric Staged Construction Analysis	119
2D Axisymmetric Thermal Analysis	126
3D Shell Analysis	132

LNG Tank Modelling

Overview

LNG Tank Wizards produce a variety of base models of full containment circular tanks to allow optional subsequent design checks to be carried out. The modelling techniques used to build the models aim to satisfy engineering requirements however engineers should check and modify the models created to ensure that they are appropriate to meet their specific needs.

The use of the Tank Wizards requires the **MicroSoft Excel** spreadsheet application to be installed in advance for full functionality as certain applications of the Wizard may use it during the design or reporting process. For example, the Wizard for a Seismic Analysis produces a computation summary and the forces calculated can be exported to a spreadsheet.

For LNG tanks, thermal analysis will generally need to be undertaken in addition to structural analysis due to the very low liquid temperatures involved.

This manual focuses on the details of modelling concepts used to build the range of models supported. A separate manual covers the procedures involved in performing design checks using the LNG Tank System.

Capabilities

The Wizards perform automatic creation of models for the following analyses, and results output tasks:

- 2D Axisymmetric Static Structural Analysis
- 2D Axisymmetric Construction Stage Analysis
- 2D Axisymmetric Thermal Analysis
- 3D Shell Static Structural Analysis
- 3D Shell Eigenvalue Analysis
- 2D Beam-Stick FSSI Seismic Analysis
- Export Forces from the 2D Axisymmetric Model

- Export Forces from the 3D Shell Model
- Export Temperature

2D Axisymmetric Static Structural Analysis

Elements

Due to the axisymmetric nature of circular tanks, a 2D axisymmetric model is commonly used. As the roof of double steel tank is not axisymmetric due to the discrete polar beams and bracings, 2D model should not be adopted. However, 2D solid model for thermal analysis is required, so for the roof plate and the polar beams are modelled with 2D axisymmetric solid elements and for the lateral rings are modelled with 2D axisymmetric shell elements.

Groups / Materials

Model features are defined in individual groups for easier post-processing and updating of the model.

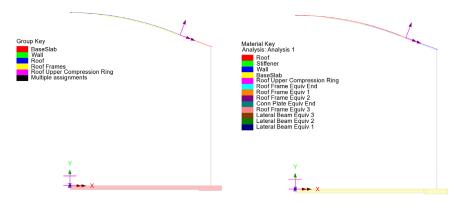


Fig 1 Group and Material Assignments for a 2D Axisymmetric Static Model

Support Condition for 2D Axisymmetric Model

Three support types are available for selection.

Tank type			Target models to build		
	Metallic	~			
Material :	Wetallic		2D axisymmetric structural	2D axisymmetric coupled thermal/structural	
Elevation :		~	2D beam-stick seismic	3D shell structural	
ank Definition Lo				☐ 3D shell structural	
ank Definition Lo	ad			3D shell structural	
ank Definition Lo	ad			☐ 3D shell structural	

Fig 2 Support Types Available

Fixed Support

Fully fixed supports are assigned to the base slab.

Pile Support

The stiffness (stiffness per unit radian) of a pile must be stated. A spring support will be assigned to the bottom of slab, at the given radial locations.

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.

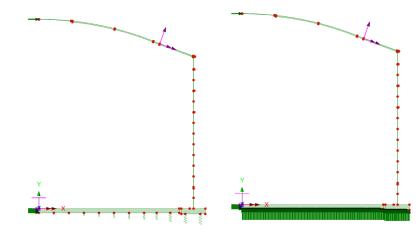


Fig 3 Support Types for a 2D Axisymmetric Static Model (Pile Support / Regular Support)

Loadings

Only the outer tank is built in the model. This will be investigated using 16 static loadcases.

See *Examples – User Inputs : 2D Axisymmetric Static Structural Analysis* for more information.

2D Axisymmetric Construction Stage Analysis

Elements

The staged construction model is built using 2D axisymmetric solid elements.

Groups / Materials

In addition to the groups defined in the 2D static model, extra groups are defined to simplify activation and deactivation of features when modelling the construction stages.

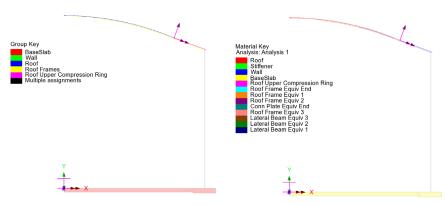


Fig 4 Group and Material Assignments in a 2D Axisymmetric Staged Construction Model

Support Condition

Support types available are the same as those for the 2D Axisymmetric Static Analysis model.

Construction Stages

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

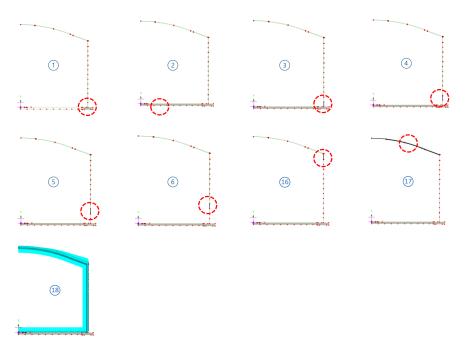


Fig 5 Activation and Deactivation in a Staged Construction Analysis Model

Stage	Description	Note
No. 1	Annular part	
No. 2	1) + Circular part	
No. 3 ~ No.16	2) + Wall 1 ~ Wall 14	
No. 17	16) + Roof	
No. 18	17) + Other Loadings	

Table 1 Sequence of Construction Stages

Loadings

The loadings are the same as those described for the 2D Axisymmetric Static Structural Analysis model. However, for this use loading is to be assigned in a step-by-step manner to each of construction stages.

See the *Examples – User Inputs : 2D Axisymmetric Staged Construction Analysis* for more information.

2D Axisymmetric Thermal Analysis

If the temperature of liquid in the tank is very low, a thermal analysis will have to be performed. The purpose of a thermal analysis is to obtain the temperature variation through the thickness of the structure thickness and to obtain the thermal stress and strains induced by the temperature gradient.

The thermal analysis should be followed by a structural analysis that uses the results of the thermal analysis (e.g. temperature distribution) as the input loading. This type of analysis is called as Thermo-Mechanical Coupled Analysis.

In LUSAS, both thermal analysis and structural analysis can be performed within a single model by setting the analysis type to be 'Coupled thermal/structural' when a model is first created.

lew Model			
File name	LNG Tank		
Working folder			
ORecent	C:\Users\ohsso\Downloads		
• User-defined	C:\Users\ohsso\Documents\LUSAS20	00\Projects	Set
Model properties			
Analysis type	Coupled thermal/structural \sim	Model units	N,m,kg,s,C 🛛 🖂
Analysis category	2D Axisymmetric \checkmark	Timescale units	Seconds ~
Optional			
Startup template	None ~	Layout grid	None 🖂
Title			
Job number			
	OK	Cancel	Help

Fig 6 New Model Dialog Setting Thermal/Structural Coupled Analysis

Elements

In a coupled analysis, 2D axisymmetric solid elements require element details to be specified for both the structural and thermal analyses.

D Axisymmetric Structural Thermal		2D Axisymmetric Structural Thermal	
Element description Element type Axisymmetric solid Axisymmetric solid Element shape Quadritateral Therpolation order Linear Element name QAX4M	Regular mesh Allow transition pattern Allow transition pattern Altonation Local x divisions Local y divisions Inregular mesh Element size I.0	Element description Thermal element type Axisymmetric solid field Element shape Quadrilateral Interpolation order Linear Element name QXF4	Regular mesh Allow transition pattern Allow tragular mesh Autow tragular mesh Local x divisions Local y divisions Dregular mesh Element size
Name AxisymmetricSolid	✓ (42)	Name AxisymmetricSolid	✓ ▲ (42)

Fig 7 Element Definition for 2D Axisymmetric Thermal Analysis

Insulation

Tank insulation is included in the model explicitly for thermal analysis. Both thermal and structural elements are assigned, but as the stiffness of insulation is low the Wizard does not consider the insulation to be structural. As a result, the insulation and structure do not share nodes, and elements are completely separate.

The 'thermal gap' properties are to be applied between insulation and structure to model the temperature transferred in thermal analysis.

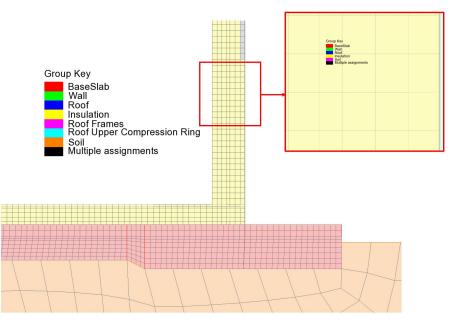


Fig 8 Insulation Elements Separated from Structure Elements

Ground (Soil)

As the ground temperature affects the structure's temperature distribution, the ground can be included in the model, extending 25m beyond the base slab. A user-defined value for soil depth at which a constant temperature is maintained defined in Tank Definition is used if the 'Include soil' option is checked. It is assumed that at the defined soil depth, a user-defined temperature is maintained.

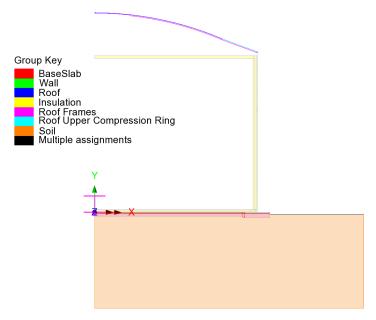
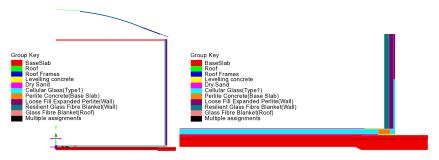


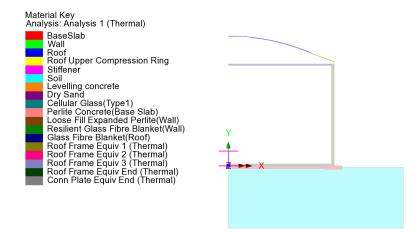
Fig 9 Mesh for 2D Axisymmetric Thermal Analysis

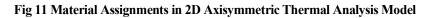
Groups / Materials

In addition to the groups defined in the 2D Axisymmetric Static Structural Analysis model, some groups are defined for insulation.









Supports and Loading for Thermal Analysis

The 1st Loadcase

The initial temperature of the concrete structure and the ground are defined and assigned.

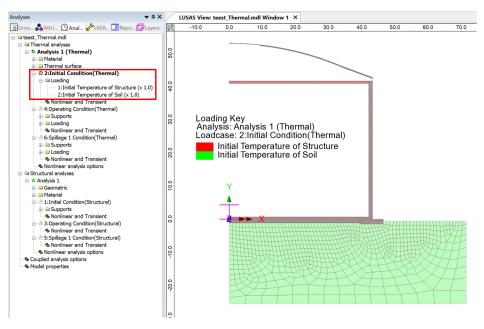


Fig 12 Thermal Analysis -1st Loadcase

The 2nd Loadcase

Liquid temperature is assigned to inner side of the insulation.

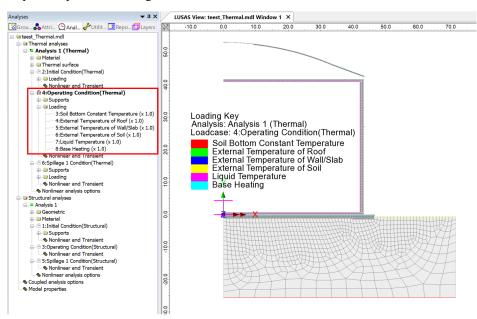
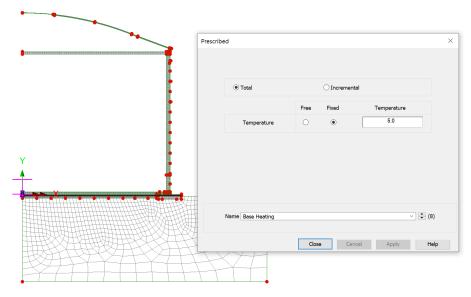


Fig 13 Thermal Analysis – 2nd Loadcase



If Base Heating temperature is specified from the Input Dialog, a Prescribed Temperature loading is defined and assigned to the base heating line.

Fig 14 Base Heating Temperature in a 2D Axisymmetric Thermal Analysis Model

Base heating temperature is assigned to the selected line as shown in [Fig 20]. The range of the loading is defined using a LUSAS field variation and can be modified by redefining the values of 'Base Heating'.

	General Field Variation	×
	Function 1.0	
	Global coordinates Function Limits Local coordinates assi	
	Uccal coordinates ass (transformed freek ☐ Min. x coordinate ☑ Max. x coordinate ○ Specified local coordii	
	3:LocalCoo Min. y coordinate Max. y coordinate	
	Min. z coordinate	
Y	OK Cancel Help	
	Name Base Heating	
	OK Cancel Apply Help	

Fig 15 Base Heating Temperature Variation in a 2D Axisymmetric Thermal Analysis Model

Supports and Loadings for Structural Analysis

Structural supports are assigned to the bottom of the slab according to the support type chosen on the input dialog.

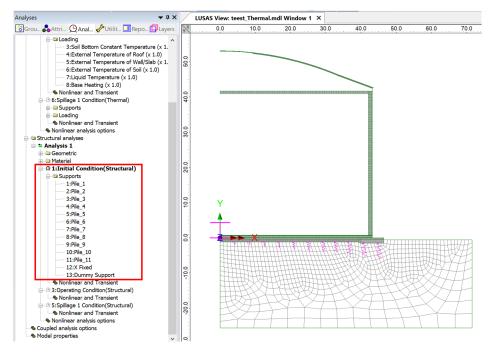


Fig 16 Pile Support for Structural Analysis following Thermal Analysis

The purpose of this analysis is to obtain thermal stress, so no additional structural loading is defined.

3D Shell Static Structural Analysis

3D shell models are used when tank loadings are not axisymmetric. The outer tank is modelled using shell elements. All loading defined for the 2D axisymmetric model is also used for this model, and wind loading is also applied.

Elements & Geometric Properties

Shell elements are positioned at and along the centre of sections. Any varying section thickness is applied using the LUSAS variations facility.

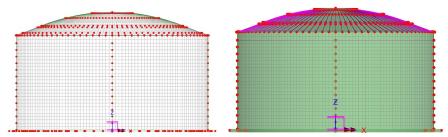


Fig 17 3D Shell Model for Static Analysis



Fig 18 3D Shell Model Thickness Variation at Roof and Slab

Groups and Materials

The main groups created are named Roof, Wall, BaseSlab, Roof Frames and Roof Upper Compression Ring. One set of dummy elements, which work as rigid links between Wall and BaseSlab., are grouped separately, to aid with results-processing.

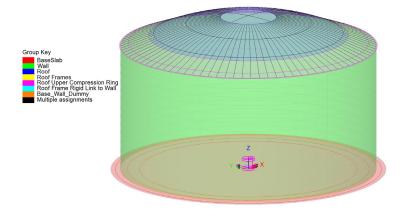


Fig 19 Groups in a 3D Shell Model

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

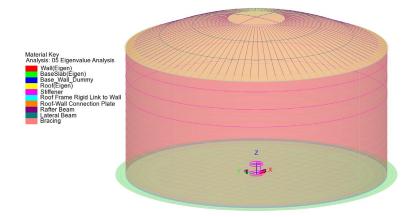


Fig 20 Material Assignments in a 3D Shell Model

Support Conditions

Three different types of support conditions can be defined.

Fixed Support

Fully fixed supports are assigned to the base slab.

Pile Support

If 'Pile Support' is chosen in Tank Definition as shown in [Fig 32 - 31], the stiffness of each pile should be defined further from the user input dialog as shown in [Fig 32].

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.

Material :	Metallic	~		axisymmetric structural	2D axisymmetric coupled thermal/structural
Elevation :	Aboveground	~	✓ 20	beam-stick seismic	☑ 3D shell structural
hk Definition	oad Insulations S	upport (3D) Seism	ic Ground		
	Roof Lateral Roof	Polar Beam Section	n Materials Supp	ort (2D)	
Support type Pile Support	~			Jpdate from Support(3D)	
Pile stiffnesse	s				
Spring ID	Radius [m]	Vertical stiffness [MN/m/rad]	Horizontal stiffness [MN/m/rad]	Description	Pile Supports
	4.2	592.0	47.9	Pile	
2	8.4	1.0524E3	85.1	Pile	
3	12.6	1.5987E3	127.7	Pile	
4	16.8	2.1049E3	170.2	Pile	
5	21.0	2.6311E3	212.8	Pile	
6	25.2	3.1573E3	255.3	Pile	
7	29.4	3.4962E3	282.7	Pile	
8	32.9	3.7295E3	301.6	Pile	Spring Number
9	36.7	4.6615E3	377.0	Pile	1 2 3
<				>	
Set zero	Set default	s Add	Remove	•	
			Name Tnk1		 (1)

Fig 21 Input for Pile Locations and Stiffnesses

If 'Simplified foundation' for Support Type is selected, the spring support will be assigned to each of pile locations.

					Та	arget models to	build								
aterial :	Me	atallic		~		☑ 2D axisymmetric structural ☑ 2D axisymmetric			tisymmetric c	oupled the	ermal/struc	tural			
evation	Ab	oveground		~		✓ 2D beam-st	ick seismic		☑ 3D sh	ell structural					
Definitior	Load	nsulations	Support (3	D) Seismi	Ground										
Suppor	t														
pport ty	pe		c	Circumferer	ntial Support	t									
nplified f	oundation		J	ID	R [m]	Initial theta [degree]	Number of piles	Vert stiffn [kN	ness	Horizont ^ stiffnes: [kN/m]	A	dd	c	rosswise	piles
lo. cir :	184			1	36.7	0.0	56	523.01		2.297E3	[Del			Circumferential piles
lo. cross				2	40.8	0.0	60	523.01	8E3 4	2.297E3	Set	zero			
X ² Cir : X ² Cros		i.1965E3 7157E3		3	44 9	0.0	68	523.01	8F3 4	2 297F3 ×	Set d	efaults	F		
osswise	support stif	fness													
Gr	support stif id wizard nates (Unit				ffness [kN/n	n] 523.011	8E3 Y coordinate			ss [kN/m]	42.297	Ξ3			
Gr	id wizard		P4		ffness [kN/n P6	n] 523.011				ss [kN/m] P4	42.2971 P5	E3 P6	P7	^	Add column
Gr K coordi	id wizard nates (Unit	s: m)		Vertical sti			Y coordinate	es (Units:	m)				P7 0.0	^	Add column Add row
Gr Coordi P1	id wizard nates (Unit P2	s: m) P3	P4	Vertical sti	P6	P7 ^	Y coordinate	es (Units: P2	m) P3	P4	P5	P6		^	
Gr (coordi P1 0.0	id wizard nates (Unit P2 4.2	s: m) P3 8.4	P4 12.6	Vertical sti P5 16.8	P6 21.0	P7 ^ 25.2	Y coordinate P1 0.0	P2 0.0	m) P3 0.0	P4 0.0	P5 0.0	P6 0.0	0.0	^	Add row
Gr X coordi P1 0.0 0.0	id wizard nates (Unit P2 4.2 4.2	s: m) P3 8.4 8.4	P4 12.6 12.6	Vertical sti P5 16.8 16.8	P6 21.0 21.0	P7 ^ 25.2 25.2	Y coordinate P1 0.0 -4.2	es (Units: P2 0.0 -4.2	m) P3 0.0 -4.2	P4 0.0 -4.2	P5 0.0 -4.2	P6 0.0 -4.2	0.0	^	Add row Del column
Gr C coordi P1 0.0 0.0 0.0 0.0	id wizard nates (Unit P2 4.2 4.2 4.2 4.2	s: m) P3 8.4 8.4 8.4 8.4	P4 12.6 12.6 12.6	Vertical sti P5 16.8 16.8 16.8	P6 21.0 21.0 21.0 21.0	P7 ^ 25.2 25.2 25.2	Y coordinate P1 0.0 -4.2 -8.4	es (Units: P2 0.0 -4.2 -8.4	m) P3 0.0 -4.2 -8.4	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
Gi P1 0.0 0.0 0.0	id wizard nates (Unit P2 4.2 4.2 4.2 4.2	s: m) P3 8.4 8.4 8.4 8.4	P4 12.6 12.6 12.6 12.6 12.6	Vertical sti P5 16.8 16.8 16.8 16.8 16.8	P6 21.0 21.0 21.0 21.0	P7 ^ 25.2 25.2 25.2 25.2 25.2 25.2	Y coordinate P1 0.0 -4.2 -8.4 -12.6	es (Units: P2 0.0 -4.2 -8.4 -12.6	m) P3 0.0 -4.2 -8.4 -12.6	P4 0.0 -4.2 -8.4 -12.6	P5 0.0 -4.2 -8.4 -12.6 16 0	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

Fig 22 Input for Pile Locations and Stiffnesses (Simplified foundation)

However, 'Detailed foundation' for 'Support Type' is selected, additional 'Foundation' tab is appeared and piles are modelled based on the user inputs. The pile length is as same as the total 'soil depth' defined in 'Ground' from Tank Definition. Either static stiffness or dynamic stiffness for 'Pile toe stiffness' is assigned to bottom of the piles and Static Soil Stiffness or Dynamic Soil Stiffness is assigned to the pile beams according to the selected 'Pile support options' in the Static Analysis Dialog [Fig 36]. Piles could be added to the existing liquid tank model without piles by modifying Tank Definition to include properties for piles and executing 'Add foundation' menu.

LNG Tank Modelling

					Та	arget models to I	build									
aterial :	Me	etallic		~		2D axisymm	etric structural	1	✓ 2D axis	ymmetric (coupled the	ermal/struct	tural			
evation	: Ab	oveground		~		✓ 2D beam-sti	ck seismic		✓ 3D she	ll structura	I					
Definition	n l beol in	nsulations	upport (30	D) Seismi	c Ground											
	t Foundati															
upport ty	ре		с	Circumferer	ntial Suppor	t								• • •		
etailed fo	undation		~			Initial theta	Number of					dd	Cross	swise		
				ID	R [m]	[degree]	base supports									
lo. cir :	184			1	36.7	0.0	56					Del			Circumferential	
lo. cross				2	40.8	0.0	60				Set	zero	H	S.		
ΣX ² Cir : ΣX ² Cros		5.1965E3 7157E3		3	44.9	0.0	68				Set d	lefaults	Fr	*		
osswise	support stir	ffness	Į													
G	rid wizard															
	rid wizard inates (Unit	s: m)					Y coordinat	tes (Units:	: m)						Add as how	
		s: m) P3	P4	P5	P6	P7 ^	Y coordinat	tes (Units: P2	: m) P3	P4	P5	P6	P7 -	^	Add column	
X coordi	inates (Unit		P4 12.6	P5 16.8	P6 21.0	P7 ^ 25.2				P4 0.0	P5 0.0	P6 0.0	P7 ·	^	Add column Add row	
X coordi P1	inates (Unit P2	P3					P1	P2	P3							
X coordi P1 0.0	P2 4.2	P3 8.4	12.6	16.8	21.0	25.2	P1 0.0	P2 0.0	P3 0.0	0.0	0.0	0.0	0.0		Add row	
X coordi P1 0.0 0.0	P2 4.2 4.2	P3 8.4 8.4	12.6 12.6	16.8 16.8	21.0 21.0	25.2 25.2	P1 0.0 -4.2	P2 0.0 -4.2	P3 0.0 -4.2	0.0	0.0	0.0	0.0		Add row Del column	
X coordi P1 0.0 0.0 0.0	P2 4.2 4.2 4.2 4.2	P3 8.4 8.4 8.4	12.6 12.6 12.6	16.8 16.8 16.8	21.0 21.0 21.0	25.2 25.2 25.2	P1 0.0 -4.2 -8.4	P2 0.0 -4.2 -8.4	P3 0.0 -4.2 -8.4	0.0 -4.2 -8.4	0.0 -4.2 -8.4	0.0 -4.2 -8.4	0.0 -4.2 -8.4	^	Add row Del column Del row	
× coordi P1 0.0 0.0 0.0 0.0	P2 4.2 4.2 4.2 4.2 4.2	P3 8.4 8.4 8.4 8.4 8.4	12.6 12.6 12.6 12.6	16.8 16.8 16.8 16.8	21.0 21.0 21.0 21.0	25.2 25.2 25.2 25.2 25.2	P1 0.0 -4.2 -8.4 -12.6	P2 0.0 -4.2 -8.4 -12.6	P3 0.0 -4.2 -8.4 -12.6	0.0 -4.2 -8.4 -12.6	0.0 -4.2 -8.4 -12.6	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 coordi P1 0.0 0.0 0.0 0.0	P2 4.2 4.2 4.2 4.2 4.2	P3 8.4 8.4 8.4 8.4 8.4	12.6 12.6 12.6 12.6	16.8 16.8 16.8 16.8	21.0 21.0 21.0 21.0	25.2 25.2 25.2 25.2 25.2	P1 0.0 -4.2 -8.4 -12.6	P2 0.0 -4.2 -8.4 -12.6	P3 0.0 -4.2 -8.4 -12.6	0.0 -4.2 -8.4 -12.6	0.0 -4.2 -8.4 -12.6	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 coordi P1 0.0 0.0 0.0 0.0	P2 4.2 4.2 4.2 4.2 4.2	P3 8.4 8.4 8.4 8.4 8.4	12.6 12.6 12.6 12.6	16.8 16.8 16.8 16.8	21.0 21.0 21.0 21.0	25.2 25.2 25.2 25.2 25.2	P1 0.0 -4.2 -8.4 -12.6	P2 0.0 -4.2 -8.4 -12.6	P3 0.0 -4.2 -8.4 -12.6	0.0 -4.2 -8.4 -12.6	0.0 -4.2 -8.4 -12.6 te e	0.0 -4.2 -8.4 -12.6 46 0	0.0 -4.2 -8.4 -12.6		Add row Del column Del row Set zero	
× coordi P1 0.0 0.0 0.0 0.0	P2 4.2 4.2 4.2 4.2 4.2	P3 8.4 8.4 8.4 8.4 8.4	12.6 12.6 12.6 12.6	16.8 16.8 16.8 16.8	21.0 21.0 21.0 21.0	25.2 25.2 25.2 25.2 25.2	P1 0.0 -4.2 -8.4 -12.6	P2 0.0 -4.2 -8.4 -12.6	P3 0.0 -4.2 -8.4 -12.6	0.0 -4.2 -8.4 -12.6	0.0 -4.2 -8.4 -12.6 te e	0.0 -4.2 -8.4 -12.6	0.0 -4.2 -8.4 -12.6	`	Add row Del column Del row Set zero	

Fig 23 Input for Pile Locations and Stiffnesses (Simplified foundation)

k Definition								
ank type			odels to build					
Material : Metallic		✓ 2D	axisymmetric structural	2D axisymmetric	c coupled thermal/structural			
Elevation : Aboveground		✓ 2D	beam-stick seismic	☑ 3D shell structur	al			
	_							
k Definition Load Insulations Su	upport (3D)	Seismic Ground						
se Support Foundation								
Foundation								
Туре	Include	Height/Thickness [m]	Section shape	D1 [m]				
Pile (Cir)		NA	Circular Hollow	. 0.8			D2	
			Circular Hallow	•		T	→ •–	
Pile (Cross)		NA	Circular Hollow	0.7		• (
							D1	
							-1	
						•		
c				>		•		
				>		•		
Subgrade stiffness	Dahimin	-7 100.0			100.0	•		
	iess [MN/m/n	n²] 100.0	Horiz	ontal stiffness [MN/m/m²]	100.0	•		
Subgrade stiffness			Horiz		100.0	•		
Subgrade stiffness Vertical stiffn	laced by sprir	ng supports)		ontal stiffness [MN/m/m²]		523.018E3		
Subgrade stiffness Vertical stiffn	laced by sprir Vertical	ng supports) I [kN/m] 523.0	18E3		Vertical [kN/m]			
Subgrade stiffness Vertical stiffn	laced by sprir Vertical	ng supports)	18E3	ontal stiffness [MN/m/m²]		523.018E3 42.297E3		
Subgrade stiffness Vertical stiffn Pile stiffnesses (when piles are rel Circumferential piles	laced by sprir Vertical Horizon	ng supports) I [kN/m] 523.0 ntal [kN/m] 42.29	18E3	ontal stiffness [MN/m/m²]	Vertical [kN/m]			
Subgrade stiffness Vertical stiffn Vertical stiffn Pile stiffnesses (when piles are rela Circumferential piles Material properties are defined it	laced by sprin Vertical Horizon in the Materia	ng supports) I [kN/m] 523.0 ntal [kN/m] 42.29 al tab	18E3 7E3	ontal stiffness [MN/m/m²]	Vertical [kN/m]			
Subgrade stiffness Vertical stiffn Vertical stiffn Pile stiffnesses (when piles are rela Circumferential piles Material properties are defined i	laced by sprin Vertical Horizon in the Materia	ng supports) I [kN/m] 523.0 ntal [kN/m] 42.29 al tab	18E3 7E3	ontal stiffness [MN/m/m²]	Vertical [kN/m]			
Subgrade stiffness Vertical stiffn Vertical stiffn Pile stiffnesses (when piles are reli Circumferential piles Material properties are defined i	laced by sprin Vertical Horizon in the Materia	ng supports) I [kN/m] 523.0 ntal [kN/m] 42.29 al tab	18E3 7E3	ontal stiffness [MN/m/m²]	Vertical [kN/m]			
Subgrade stiffness Vertical stiffn Vertical stiffn Pile stiffnesses (when piles are reli Circumferential piles Material properties are defined i	laced by sprin Vertical Horizon in the Materia	ng supports) I [kN/m] 523.0 ntal [kN/m] 42.29 al tab	18E3 7E3	ontal stiffness [MN/m/m²]	Vertical [kN/m]			
Subgrade stiffness Vertical stiffn Vile stiffnesses (when piles are reli Circumferential piles Material properties are defined i Pile heights and horizontal supp	laced by sprin Vertical Horizon in the Materia	ng supports) I [kN/m] 523.0 ntal [kN/m] 42.29 al tab	18E3 7E3	ontal stiffness [MN/m/m²]	Vertical [kN/m]			
Subgrade stiffness Vertical stiffn Ille stiffnesses (when piles are reli Circumferential piles Material properties are defined i Pile heights and horizontal supp	laced by sprin Vertical Horizon in the Materia	ng supports) I [kN/m] 523.0 ntal [kN/m] 42.29 al tab	18E3 7E3	ontal stiffness [MN/m/m²]	Vertical [kN/m]			
Subgrade stiffness Vertical stiffn Vile stiffnesses (when piles are reli Circumferential piles Material properties are defined i Pile heights and horizontal supp	laced by sprin Vertical Horizon in the Materia	ng supports) I [kN/m] 523.0 ntal [kN/m] 42.29 al tab	18E3 7E3	ontal stiffness [MN/m/m²]	Vertical [kN/m]			
Subgrade stiffness Vertical stiffn Pile stiffnesses (when piles are reli Circumferential piles Material properties are defined i Pile heights and horizontal supp	laced by sprin Vertical Horizon in the Materia	ng supports) I [kN/m] 523.0 ntal [kN/m] 42.29 al tab	18E3 7E3	ontal stiffness [MN/m/m²]	Vertical [kN/m]			
Subgrade stiffness Vertical stiffn Pile stiffnesses (when piles are reli Circumferential piles Material properties are defined i Pile heights and horizontal supp	laced by sprin Vertical Horizon in the Materia	ng supports) I [kN/m] 523.0 ntal [kN/m] 42.29 al tab	18E3 7E3	ontal stiffness [MN/m/m²]	Vertical [kN/m]			
Pile stiffnesses (when piles are rel Circumferential piles - Material properties are defined i - Pile heights and horizontal supp	laced by sprin Vertical Horizon in the Materia	ng supports) [kN/m] 523.0 ttal [kN/m] 42.29 al tab v the inputs in the Groun	18E3 7E3	ontal stiffness [MN/m/m²]	Vertical [kN/m] Horizontal [kN/m]			

Fig 24 Input for Pile Dimensions (Detailed foundation)

LNG Tank Modelling

Materia	e				Target models	s to build					
	t: P	Metallic		\sim	🔽 2D axis	ymmetric struct	tural [·	2D axisymmetric of	coupled thermal/stru	uctural	
Elevatio	on: A	Aboveground		~	🗹 2D bear	m-stick seismic	: 6	✓ 3D shell structural			
		-									
		Insulations Sup	oport (3D) Se	eismic Grour	nd						
oil Prope	rties										
Stiffness	distribution:	Constant value	•	~							
Layer No.	Soil depth [m]	Thickness of layer [m]	Static kh [MN/m/m]	Static kv [MN/m/m]	Dynamic kh [MN/m/m]	Dynamic kv [MN/m/m]	Lumped kh [MN/m/m]	Description(
D	0.0	0.0	0.0	0.0	0.0	0.0	0.0	Start of soil p			G.L
1	-2.0	2.0	19.0793	0.0382	38.1586	0.0763	14.92E3	Backfill	Layer 1		Piles
2	-4.0	2.0	32.9527	0.0659	65.9054	0.1318	25.769E3	Backfill	Layer 2		Files
3	-6.0	2.0	28.6317	0.0573	57.2634	0.1145	22.39E3	Backfill	Layer 3		Layer thickness
4	-8.0	2.0	27.5563	0.0551	55.1125	0.1102	21.549E3	Backfill	Layer 4		
5	-10.0	2.0	30.2072	0.0604	60.4143	0.1208	23.622E3	Backfill	Layer n		
3	-12.0	2.0	41.3977	0.0828	82.7954	0.1656	32.373E3	Silty Sand1			_
7	-14.0	2.0	34.5307	0.0691	69.0614	0.1381	27.003E3	Silty Sand1	Add	Set zero	
c								×	Remove	Set defaults	

Fig 25 Input for Pile Dimensions (Detailed foundation)

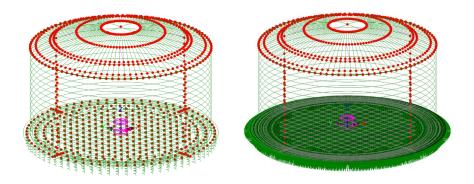


Fig 26 Support Condition for a 3D Shell Model (Pile Support(Simplified foundation) / Regular Support)

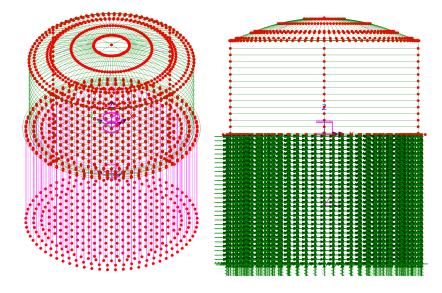


Fig 27 Support Condition for a 3D Shell Model (Pile Support(Detailed foundation))

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.

Loadings

16 loadcases, as defined for a 2D Axisymmetric Model, are all included in a 3D Shell Model. Wind load can be added using **LNG Tank> Add loading> Wind...** menu.

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 28 User Input for Wind Load for 3D Shell Model

Other Options

Half Only Model

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

LNG Tank - Base Model for Design	Check	×
Tank definition data Model filename Saved model file path Modeling options Element size (m) 2.0 Number of eigenvalue 10		✓ Lusas200\Projects\.mdl Half symmetric model Rigid for conn. plate Include temporary opening Include non-structural masses in the eigenvalue analysis
Concrete Tank Options Buttress Number of buttress Extruded thickness Buttress width Roof / Ringbeam Roof construction plan Roof first stage thickness (ratio Initial prestress for ringbeam (ra Initial prestress for base slab (r	atio) 1	Construction Scenario - Single layered roof 1 1 - Base / Wall / Ringbeam 2 - Ringbeam 1st PS 3 - Roof frames 1/ Inner work 4 - Roof frames 2,3 5 - Roof wet / Roof complete 6 - Ringbeam 2nd PS 7 - Wall vertical PS 8 - Wall horizontal PS
		OK Cancel Help

Fig 29 User Input for a 3D Shell Model (Half Model)

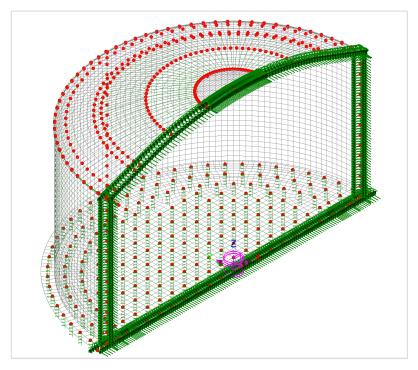


Fig 30 3D Shell Model (Half Model)

Include non-structural masses

Non-structural masses are converted into equivalent structural masses for an Eigenvalue Analysis. If this option is ticked, non-structural masses for each member (e.g. base slab, wall, roof) will be added to the mass of each member to compute the sum of total mass including non-structural masses. The equivalent structural masses will be computed by dividing total mass including non-structural masses by volume of each member. The calculation spreadsheet will be created with the same name as the model filename and stored in the user-defined working folder. The use of the 'Include non-structural masses' option is explained further in the section titled 'Examples of 3D Shell Analysis.'

Tank definition data						
del filename						
red model file path	C:\Use	C:\Users\ohsso\Documents\Lusas200\Projects\.mdl				
odeling options						
Element size (m)		\checkmark	Half symmetric model Rigid for conn. plate			
			Include temporary opening			
Number of eigenvalue	0		Include non-structural masses in the eigenvalue analysis			
oncrete Tank Options Buttress Number of buttress 0		~	Construction Scenario - Single layered roof 1 1 - Base / Wall / Ringbeam			
Extruded thickness	1.0	(m)	2 - Ringbeam 1st PS 3 - Roof frames 1/ Inner work			
Buttress width	5.0	(m)	4 - Roof frames 2,3			
Roof / Ringbeam			5 - Roof wet / Roof complete 6 - Ringbeam 2nd PS			
Roof construction plan	Single la	yered roof 1 🛛 🖂	7 - Wall vertical PS 8 - Wall horizontal PS			
Roof first stage thickness (rat	tio)	1.0	8 - Wall horizontal PS			
Initial prestress for ringbeam	(ratio)	1				
Initial prestress for base slab	(ratio)	0.5				

Fig 31 User Input for Eigenvalue Analysis Model including Non-Structural Masses

DIMENSION					
Component	Dimension(m)				
Inner Tank Radius	42.1				
Tank Height	40.06				
LNG Height	38.92				
SUMMARY FOR M	ASS				
		11-14-14-14-14-14	Characterization of the second	Total man	Equivalent ur
Component	Volume		Structural mass		
	m³	kg/m ³	kg	kg	kg/m ³
Roof plate	106	7,800	825,740	2,935,740	27,73
Lateral beam	3.93E-01	7,800	3.07E+03	3,066	7,80
Polar beam	9.11E+00	7,800	7.11E+04	71,086	7,80
Bracing	1.29E-01	7,800	1.00E+03	1,003	7,80
Wall	117	7,800	912,893	1,736,893	14,84
BaseSlab	8,719	2,500	21,797,085	24,925,085	2,85
LNG	216,714	480	104,022,703	104,022,703	48
Inner Tank	316	7,850	2,479,105	2,799,105	8,86
MASS DETAILS					
Component	Descriptions				Mass (kg)
Roof	Roof plate (= Roof plate volume * unit steel mass)				
	Roof liner + steel roof structure				
	Suspended deck + insulation of the suspended ceiling				
	Roof nozzles				135,00
	Roof platform				400,00
	Roof pump & cra	ne			30,00
	Roof piping and support				
	Others				
	Total				2.935.74
Ring Beam		g Room volumo * u	nit stool mass)		2,533,7-
King beam	Ring Beam (= Ring Beam volume * unit steel mass) wall barrier plate				
	wall piping and support				
	Others				
	Total				
Outer Wall		mo * unit stool mos	-)		912,89
outer wan	Wall (= Wall volume * unit steel mass) corner protection				
	wall barrier plate	242,00			
	wall piping and s				454,00
	Others	арроге			88,00
	Total				1,736,89
Base Slab	Base (= Base slab	21,797,08			
	Others	3.128.00			
	Total				24,925,08
Inner Steel Tank	Inner Steel tank (= Steel tank volume * steel mass)				
Inner Steel Tank	shell stiffener	2,479,10			
		09()			45,00
	shell insulation(5	0%)			
	top girder				
	Others				
LNG	Others Total	me * unit LNG mass	X		275,00 2,799,10 104,022,70

Fig 32 Summary of Mass Calculation for Eigenvalue Analysis including Non-structural Masses

See Examples – User Inputs : 3D Shell Static Analysis for more information.

3D Shell Eigenvalue Analysis

When a 3D Shell model is created, an eigenvalue analysis is added by defining the number of target eigenvalues to extract.

ank definition data Tnk1	Tnk1				
lodel filename					
aved model file path C:\Users\oh	C:\Users\ohsso\Documents\Lusas200\Projects\.mdl				
Modeling options					
Element size (m)	🗸 Halt	f symmetric model	Rigid for conn. plate		
	Incl	ude temporary opening			
Number of eigenvalue 10	✓ Incl	ude non-structural mass	es in the eigenvalue analysis		
Concrete Tank Options Buttress Number of buttress Extruded thickness	(m)	Construction Scenari 1 - Base / Wall / 2 - Ringbeam 19 3 - Roof frames	st PS		
Buttress width 5.0	(m)	4 - Roof frames 5 - Roof wet / Ro			
Roof / Ringbeam		6 - Ringbeam 2r			
Roof construction plan Single layered	d roof 1 🛛 🖂	7 - Wall vertical			
Roof first stage thickness (ratio)	1.0	8 - Wall horizont	tal PS		
Initial prestress for ringbeam (ratio)	1				
Initial prestress for base slab (ratio)	0.5				

Fig 33 User Input for a 3D Shell Model for Eigenvalue Analysis

LNG Tank Modelling

Analyses 🗸 🕈 🗸	LUSAS View: 3D Shelll Full.mdl Window 1 ×	
🐼 Grou 💑 Attri 🕒 Anal 🥓 Utiliti 💷 Repo 🗇 Layers	-80.0 -70.0 -60.0 -50.0 -40.0	-30.0
Structural analyses	Eigenvalue	×
= 02 Seismic Analysis = 03 Staged Construction Analysis	Solution Frequency V	Value 10
e) = 05 Eigenvalue Analysis ⊕ ≅ Material	Include modal damping Set damping Shift to be applied	0.0
	Eigenvalues required Minimum Range specified as Frequency Eigenvalue	
- 44:Mode 4 Frequency - 0.87585 - 44:Mode 5 Frequency - 0.875403 - 44:Mode 5 Frequency - 0.878403 - 44:Mode 6 Frequency - 0.883982	Eigenvector normalisation Type of eigensolver Default O Unity O Mass Stiffness	~ missing eigenvalue
	Convert assigned loading to mass OK Cancel	Advanced Help

Fig 34 Eigenvalue Analysis in a 3D Shell Model

2D Beam-Stick FSSI Seismic Analysis

A lumped mass beam-stick model is produced to perform a dynamic analysis under earthquake conditions. The concept of using generalized single degree of freedom systems to represent the impulsive and convective modes of vibration of tank-liquid system is extensively discussed in the works by authors such as (Haroun & Housner, 1981) and (Wang, Teng, & Chung, 2001).

The beam-stick model includes:

- 1) The outer steel tank.
- 2) The fluid-structure-interaction (FSI) effects of the inner tank together with the dynamic behaviour of the stored liquid.
- 3) The soil-structure-interaction (SSI).

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

Model for horizontal actions

Elements

The main elements used in the modelling are outlined in [Fig 46].

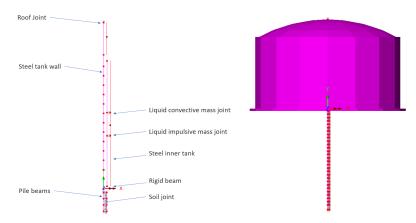


Fig 35 Beam-Stick Modelling Concept for Horizontal Actions

A joint element is used to add non-structural masses to the top of the roof.

Joint elements are used for impulsive and convective liquid masses attached to the inner tank. Joint elements are used for soil springs linked with piles.

Geometric Properties

Roof is idealized by a single series of beam element with Circular Hollow Section.

Geometric Properties are computed based user inputs and assigned as illustrated in [Fig 51].

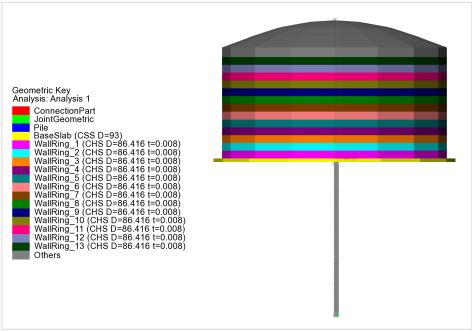


Fig 36 Geometric Properties in a Beam-Stick Horizontal Model

The Connection Part is regarded as rigid, and 1 x 1m section is used.

Material Properties

Material Properties are assigned as illustrated in [Fig 52].

The structural masses and non-structural masses are distributed in the relevant element by adjusting the unit mass of each member to include the non-structural masses. However, the non-structural masses for the roof are separately assigned to the top of roof as a lumped mass.

The masses and locations of liquid for convective and impulsive effect are computed based on either [ACI 350.3] or [EN1998-4], and the details of the computations are summarized as a spreadsheet and saved in the working folder with filename of '<model name>_<code name>_HorizontalBeamStick.xlsx'. (See [Fig 53] and [Fig 54])

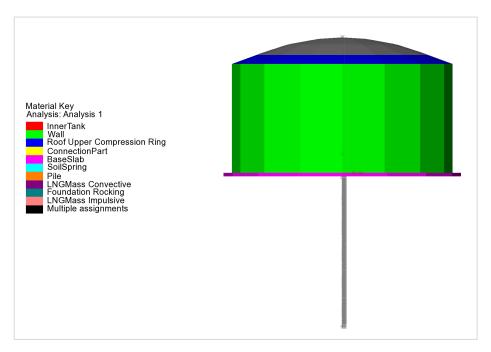


Fig 37 Material Properties in a Beam-Stick Horizontal Model

LNG Tank Modelling

					1	3) Stiffness for Impuls	ive Mass			
						Component	Value	Unit	Remark	
						H/R	0.92447		LNG height divided by inner tank radius	
						Pi	480.0000	kg/m ³	mass density of LNG	
						Es	2.00E+11	N/m ²	modulus of elasticity of inner tank material	
erification for Beam	-Stick Model				98-4	s	0.0348	m	equivalent uniform thickness of inner tank wall	
						C,	6.51359		coefficients for determining the fundamental frequency	
DIMENSION						Time	0.43182	5	fundamental period of oscillation of the tank (plus the	
						ki	11.325.839.357	N/m		
	Dimension(m)				-		///			
Inner Tank Radius (R)	42.1									
Tank Height (H)	40.06					CALCULATED PROP	ERTIES FOR VERTICAL M			
LNG Height (H _L)	38.92					1) Mass & Stiffness for		NODEL		
						Component	Value	Unit	Remark	
							3,015,865	kg	mass of roof	
SUMMARY FOR MASS						m_reat f	5,015,865	Kg Hz	fundamental frequency of oscillation of the roof	
	•				Equivalent unit	T	N/A	5	fundamental period of oscillation of the roof	
			Structural mass		mass	k_reat	N/A	N/m		
	m³	kg/m ³	kg	kg	kg/m ³	"roof	10/4		-	
Roof	103	7,800	825,740	3,015,865	8,812	2) Mass & Stiffness for	ING			
Lateral Beam	0	7,800	3,066	-	-	Component	Value	Unit	Remark	
Polar Beam	9	7,800	71,086	-	-	H/R	0.924	Unit	LNG height divided by inner tank radius	
Bracing	0	7,800	1,003	-	-		480.0000	3	mass density of LNG	
Connection plate	1	7,850	4,970	-	-	Pi		kg/m ³		
Wall & Buttress	117	7,800	912,893	1,736,893	14,840	Es	2.00E+11	N/m ²	modulus of elasticity of inner tank material	
BaseSlab	8,719	2,500	21,797,085	24,925,085	2,859	V	0.3		poisson ratio of steel	
LNG	216,714	480	104,022,703	104,022,703	480	s(ζ)	0.0361	m	wall thickness for $\zeta = 1/3$ ($\zeta = 2/H_L$)	
Inner Tank	316	7,850	2,479,105	2,799,105	8,863	f(y)	1.0565		0.8 <nγ<4:1.078+0.274 (a.41a,="" (γ),="" a41b<="" in="" td="" γ<0.8:1=""></nγ<4:1.078+0.274>	
						P _{vf}	16,085	kg/m ²	hydrodynamic pressure on the wall base, from A.40.	
						m_LNG_f	89,566,808	kg	mass of LNG (radial breathing), ref. A.40.	
CALCULATED PROPER	TIES FOR HORIZONT	AL MODEL				m(1)	52,900,941		mass of LNG (rigidly moving) = sqrt(m ² , ² , ²)	
1) LNG Mass & Height						m_ _{LNG_f} (2)	14,455,895		mass of UNG (rigidly moving) = m_UNS_post-m_UNS_f	
	IBP (I	ncluding Base P	ressure)			P _{vr}	18,681.6000	kg/m ²	hydrodynamic pressure on the wall, from A.17	
					Lever arm	m_UNG r (3)	104,022,703	kg	mass of LNG (rigidly moving), ref. A.17.	
					height	v	0.9245		=H_/R	
				mc(mi), Kg	hc(hi), m	Υ1	1.699140		=π / (2y)	
LNG Convective	0.924	0.49	0.84	50,527,854	32.77	lo(v1)	1,8629		bessel function order 0	
LNG Impulsive	0.924	0.51	0.79	53,494,849	30.88	I ₁ (Y ₁)	1.1953		bessel function order 1	
	EBP (E	xcluding Base P	ressure)			f _{vd}	2,4734	Hz	fundamental frequency of oscillation of the liquid	
	H/R		E 01	mass	Lever arm	T _{vd}	0.4043		fundamental period of oscillation of the liquid	
Component	n/K		h _(c,i) /H		height	k_LNG r	21.631.229.542	N/m	and a period of operation of the liquid	
				mc(mi), Kg	hc(hi), m		21,631,229,542,194,300	N/m		
LNG Convective	0.924	0.49	0.60	50,527,854	23.53	k_ing_r	21,001,229,542,194,300	iN/m		
.NG Impulsive	0.924	0.51	0.41	53,494,849	16.13					
1110						3) Mass for Outer&Inr		11-14	Dowel	
2) LNG convective stiffn	Value	Unit		Remark		Component	Value	Unit	Remark	
Component H/R	Value 0.924	Uhit	LNG height divided b			m_OuterInnerTank	29,461,083	kg	mass at top of pile = total mass - LNG - roof	
Cc	1.54	s/m ^(1/2)	coefficients for deter		otalfreeuencu		-			
	9,993	s/m ^{····}	natural period of the			4) Mass & Stiffness for				
T _{conv}			natarar period of the	. mar (convective) in	out or anothing	Component	Value	Unit	Remark	
kc	19,974,995	N/m				k_ole	225,923,300,000	N/m		

Fig 38 Mass Summary for the Beam-Stick Model

erification for Be	am-Stick Model				50.3	3) LNG impulsive stiff	ness		
						Component	Value	Unit	Remark
DIMENSION						tw	29.7905	mm	average wall thickness (inner tank)
						Es	2.00E+05	MPa	modulus of elasticity of inner tank
	Dimension(m)					ρ	7.8500	kN.s ² /m ⁴	mass density of inner tank
Inner Tank Radius	42.1					C _w	0.1586		coefficients for determining the fundamentalfrequenc
Tank Height	40.06					C	0.0422		coefficients for determining the fundamental frequenc
LNG Height	38.92					ωί	5.473	rad/s	circular frequency of the impulsive mode of vibration
						т	1.148	s	fundamental period of oscillation of the tank (plus th impulsive component of the contents)
SUMMARY FOR M	ASS					ki	1,586,485,989	N/m	
Component	Volume	Unit mass	Structural mass	Total mass	Equivalent unit mass				
	m ³		kg		kg/m ³	CALCULATED PROF	ERTIES FOR VERTI	CAL MODEL	
Roof	103	7,800	825,740	3.015.865	8.812	1) Roof Mass & Stiffn	ess		
Lateral beam	0	7,800	3,066			Component	Value		Remark
Rafter beam	9	7,800	71.086	-	-	m_roof	3,015,865	kg	mass of roof
Bracing	0	7,800	1,003			f	-	Hz	fundamental frequency of oscillation of the roof
Connection plate	1	7,850	4,970	-		T	N/A	s	fundamental period of oscillation of the roof
Wall & Buttress	117	7,800	912,893	1.736.893	14,840	k_roof	N/A	N/m	
BaseSlab	8,719	2,500	21,797,085	24,925,085	2,859				
LNG	216,714	480	104,022,703	104,022,703	480	2) LNG Mass & Stiffne	255		
Inner Tank	316	7.850	2,479,105	2,799,105	8,863	Component			Remark
						m_LNG	104,022,703	kg	mass of LNG
						tw	29.7905	mm	average wall thickness (inner tank)
CALCULATED PROP	PERTIES FOR HORIZ	ZONTAL MOD	EL			Es	2.00E+05	MPa	modulus of elasticity of inner tank
1) LNG Mass & Heigh	t					ρι	480.0000	kg/m ³	mass density of LNG
	mass		Lever arm			g	9.8070	m/sec ²	gravitational acceleration
	mass	height (IBP)	height (EBP)			YL	4,7074	kN/m ³	specific weight of contained liquid
	mc(mi), Kg		hc(hi), m			T _v	0.4504	s	fundamental period of oscillation of the LNG
LNG Convective	48,423,453	31.83	23.10			k_ing	20,247,300,685	N/m	
LNG Impulsive	52,963,803	33.36	14.60			►_LNG	20,247,300,003	14/10	
2) LNG convective st	iffoorr					3) Mass for Outer&In	ner Tank		
Component	Value	Unit		Remark		Component	Value	Unit	Remark
g	9,8070	m/sec ²	gravitational accele			m_OuterInnerTank	29,461,083	kg	mass at top of pile = total mass - LNG - roof
λ	5,8106	m ^{1/2} /s	coefficient as define						
ωc	0.6332	rad/s	circular frequency o		irst(convective)	4) Mass & Stiffness fo	r Pile		
Tc	9.9223	s	natural period of the			Component			
kc	19,417,270	N/m				k_pile	225,923,300,000	N/m	

Fig 39 Computation Summary of Liquid Masses for the Beam-Stick Model

The material properties for the connection beam between steel wall and inner tank are assumed to be the same as that defined for the base slab.

Groups

The groups defined in the model are summarized in [Fig 55].

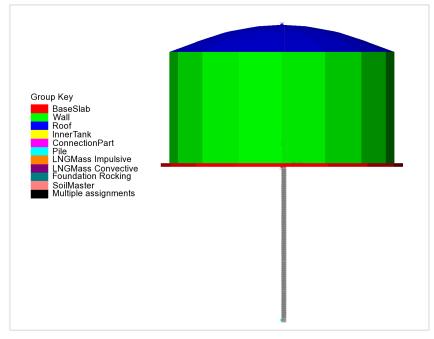


Fig 40 Groups in a Beam-Stick Horizontal Model

Damping Coefficients

Damping coefficients are computed based on the user inputs for desired damping ratio (%) and the frequency range of the structure obtained from a separate eigenvalue analysis.

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

Fig 41 User Inputs for Damping for Seismic Analysis

For structural members and impulsive liquid mass, Rayleigh Damping Coefficients are computed and used in the material definition.

For Soil springs and convective mass, a Viscous Coefficient (=Damping Ratio * $2*\sqrt{km}$) is used for horizontal movement considering the moving mass above the ground.

Support Conditions

Vertical supports are assigned to all members.

As the pile group is modelled by a series of beam elements in a single position, a rotational support representing the resistance to the overturning moment is added to the pile head.

Lumped stiffness for Soil properties defined in Tank Definition is used in beam stick horizontal model.

Fank ty	ре				Target model	s to build						
Mater	ial : N	Aetallic		~	2D axisymmetric structural 2D axisymmetric con			coupled thermal/structural				
Eleva	tion : A	boveground		~	☑ 2D beam-stick seismic		5	☑ 3D shell structural				
nk Defi	nition Load	Insulations Su	pport (3D) Se	eismic Grou	nd							
oil Prop	erties											
Stiffnes	s distribution:	Constant valu	e	~								
Layer No.	Soil depth [m]	Thickness of layer [m]	Static kh [MN/m/m]	Static kv [MN/m/m]	Dynamic kh [MN/m/m]	Dynamic kv [MN/m/m]	Lumped kh [MN/m/m]	Description(C	L		
	0.0	0.0	0.0	0.0	0.0	0.0	0.0	Start of soil p			G.L.	
I	-2.0	2.0	19.0793	0.0382	38.1586	0.0763	14.92E3	Backfill	Layer 1		Piles	
	-4.0	2.0	32.9527	0.0659	65.9054	0.1318	25.769E3	Backfill	Layer 2			
	-6.0	2.0	28.6317	0.0573	57.2634	0.1145	22.39E3	Backfill	Layer 3		Layer thickness	
ł	-8.0	2.0	27.5563	0.0551	55.1125	0.1102	21.549E3	Backfill	Layer 4			
	-10.0	2.0	30.2072	0.0604	60.4143	0.1208	23.622E3	Backfill	Layer n			
	-12.0	2.0	41.3977	0.0828	82.7954	0.1656	32.373E3	Silty Sand1			_	
	-14.0	2.0	34.5307	0.0691	69.0614	0.1381	27.003E3	Silty Sand1	Add	Set zero		
								····· ×	Remove	Set defaults		
Stati	stiffness c vertical stiffr		15.0E3 30.0E3			stiffness [kN/m	-					
Dyna	anno verudal s	tiffness [kN/m]		[ynamic nonzo	ental stiffness [k	awang					
				Name	Tnk1				× 🗘 (1)			

Fig 42 lumped stiffness for Soil Springs for Seismic Analysis

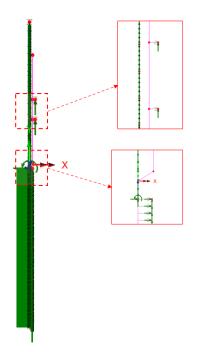


Fig 43 Support Conditions in a Beam-Stick Horizontal Model

Loadings

For the transient dynamic analysis, time history seismic acceleration/velocity/displacement would be used for loading. The Wizard is designed to prepare the model for a Response Spectrum Analysis, so no loading is required.

If required, the model can be easily transformed to a transient dynamic analysis model by adding time history loading data using 'Load Curve'. Refer to the LUSAS Modeller Reference Manual for more details.

Analysis Control

By default, the target number of modes is set to 30. This would be need to be increased if not found to be sufficient to capture sufficient response.

The Wizard sets the **'Include model damping'** option 'on'. This does not affect the result of natural frequencies and eigenvalues but ensures that damping is considered in the calculation of the results forces that are obtained.

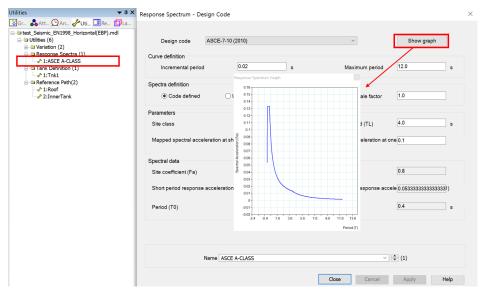
Analyses 👻 🕈 🗙	Eigenvalue	×
Croups & Attrib Q Analy & Utiliti Repo D Layers Status Status Stat	Solution Frequency V Include modal damping Set damping	Number of eigenvalues 30 Srift to Be applied 0.0 Modal Damping
	Eigenvalues required Minimum V Range specified as © Frequency Eigenvalue	Viscous Structural Use distributed damping for All modes
Post processing Or 2:Response Spectrum CQC O 3:Response Spectrum SRSS Model properties	Eigenvector normalisation O Unity	Modes using distributed damping Bigenvalue 1 Bigenvalue 2
	0 1	Bigerwalue 3 Bigerwalue 4 Bigerwalue 5 Bigerwalue 7 Bigerwalue 7

Fig 44 Eigenvalue Control for a Beam-Stick Horizontal Model

Response Spectrum

By default, a Response Spectrum corresponding to ASCE, one of the design response spectrums available in the LUSAS database, is defined by the Wizard.

A different response spectrum can be selected and used in the model, and a 'User Defined Response Spectrum' is available by selecting the **Utilities>Response Spectrum** menu item.





Options for Post-Processing

After solving the model, the results of eigenvalue analysis will be loaded on LUSAS Modeller. The results are combined in accordance with the options available in post-processing loadcase. The options can be defined manually by selecting the **Analyses>IMD loadcase** menu item and making selections as shown in [Fig 63].

Analyses	▲ 廿 ×		-200.0	-180.0	-160.0	-140.0	-120.0	-100.0
🖆 Layers 🔞 Groups 🖧 Attribu 😟 Analys 🛷	The second second second second	80.0			~			
Structural analyses Structural analyses Sharper	IMD Loadcase				×			
	Excitation	Support Motio	on ~	Set				
	Results	Spectral	~	Set	Spectral Respon	nse		×
Post processing O 2:Response Spectrum CQC	Damping		Modes		Type of spectral		CQC combine	ation
General Stress Spectrum SRSS	Type LUSAS vi	LUSAS values 4 Use				Damping variation correction Eurocode to response spectrum Response 1:ASCE A-CLAS:		(2
	Set da							LASS 3
					0	ОК	Cancel	Help
	Name Re:	sponse Spectru	m CQC	~	(2)			
		lose	Cancel	Apply	Help			

Fig 46 Post Processing Options for a Beam-Stick Horizontal Model

The method of combining the modes to obtain the maximum structural effects is chosen. Two post-processing loadcases are defined by default: one for CQC combination, the other for SRSS combination.

The formulae to be used for damping variation correction are set to 'Eurocode' by default, the available options are Eurocode, Kapra, Tolis & Faccioi, and Bommer & Elnashai.

The design response spectrum is chosen.

If the 'Include modal damping' option is checked from Eigenvalue analysis control dialog (see [Fig 61]), modal damping is computed during the eigenvalue analysis and used at post-processing by selecting Damping Type as 'LUSAS values'.

See *Examples – User Inputs* for more information.

Model for vertical actions

Elements

The concept of using a beam-stick model for vertical actions is illustrated in [Fig 64].

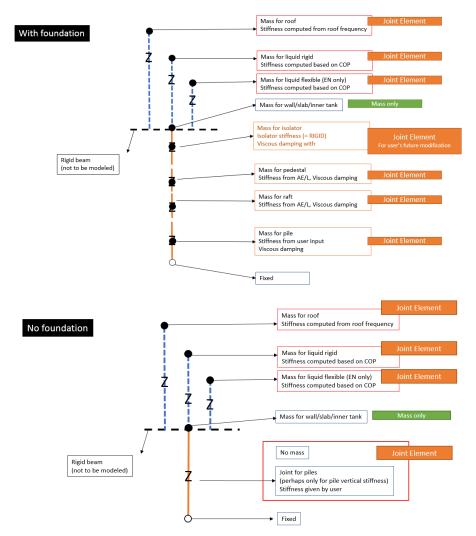


Fig 47 Beam-Stick Modelling Concept for Vertical Actions

The model is built using four joint elements as shown in [Fig 65]. Four joint elements share the node at the location of 'Mass for Wall & Slab & Inner tank'. The length of

joint elements does not affect the analysis result. Different joint lengths are shown here only for visualization purposes.

If design code ACI 350.3 is chosen for building the model, the 'Mass and Joint for Liquid Rigid' joint element is not included.

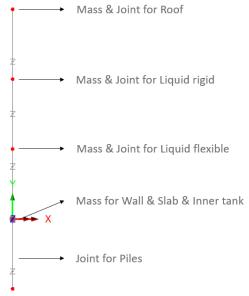


Fig 48 Beam-Stick Model for Vertical Actions

Geometric Properties

The following dataset is used.

And the second second		1						
Analysis categ	gory 2	D Inplane						
🗌 Use joint le	ength							
	nponent		Val	Je				
Eccentricity (e:	-)		0.0					
	2)							
	2)							
	2)	I						
Name					(1)			

Fig 49 Geometric Properties for Joint Elements for Beam-Stick Vertical model

Material Properties

Mass, stiffness, and damping coefficients are assigned for material properties for joint element as shown in [Fig 67].

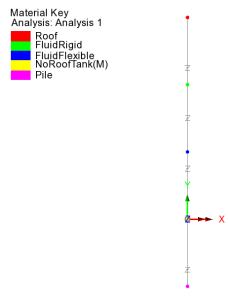


Fig 50 Material Properties in Beam-Stick Vertical Model

Details of how masses and stiffness are calculated are summarized in a spreadsheet form as shown in [Fig 53] and [Fig 54]. Values in red are written by the Wizard, and others are computed by the spreadsheet, hence the values in this spreadsheet can be used for verification by comparing with data from LUSAS Modeller.

Mass & Stiffness	for LNG						
Component	Value	Unit	Remark	< Contract of the second se			
H/R	0.924		LNG height divided by inner tar	nk radius			
ρι	480.0000	kg/m ³	mass density of LNG				
Es	2.00E+11	N/m ²	modulus of elasticity of inner t	ank material			
ν	0.2		poisson ratio of steel				
s(ζ)	0.0361	m	wall thickness for ζ = 1/3 (ζ = z/H_L)				
f(γ)	1.0565		0.8<=γ<4 : 1.078+0.274 ln (γ) ,	γ<0.8 : 1 (A.41a, A41			
P _{vf}	16,085	kg/m ²	hydrodynamic pressure on the	wall base, from A.40.			
m_ _{LNG_f}	89,566,808	kg	mass of LNG (radial breathing)	, ref. A.40.			
m(1)	52,900,941	kg	mass of LNG (rigidly moving) = sqrt(m_LNG, pail ~m, LNG,				
m_ _{LNG_r} (2)	14,455,895	kg	mass of LNG (rigidly moving) = r	m_LNG_total=m_LNG_f			
Pvr	18,681.6000	kg/m ²	hydrodynamic pressure on the	wall, from A.17			
m_ _{LNG_r} (3)	104,022,703	kg	mass of LNG (rigidly moving), re	ef. A.17.			
γ	0.9245		=H _L /R				
γ1	1.699140		=π / (2γ)				
I ₀ (γ ₁)	1.8629		bessel function order 0				
Ι ₁ (γ ₁)	1.1953		bessel function order 1				
f _{vd}	2.4081	Hz	fundamental frequency of osci	llation of the liquid			
T _{vd}	0.4153	s	fundamental period of oscillat	ion of the liquid			
k_lng_f	20,504,603,004	N/m					
k_lng r	20,504,603,003,538,400	N/m					

Fig 51 Mass and Stiffness for Liquid for Beam-Stick Vertical Model

For the pile joint, the mass is defined as the sum of the total mass excluding the roof. The stiffness is defined by user input. This is summarized in the spreadsheet as shown in [Fig 69]. This mass is assumed to move rigidly vertically.

3) Mass for Outer&Inne	er Tank				
Component	Value	Unit		Remark	
mouterInnerTank	29,461,083	kg	mass at top of pile = total mass - LNG - roof		
4) Mass & Stiffness for	Pile				
Component	Value	Unit	Remark		
kpile	225,923,300,000	N/m			

Fig 52 Mass and Stiffness for Pile Joint for Beam-Stick Vertical Model

Damping Coefficients

Viscous Coefficients (calculated as the Damping Ratio * $2^*\sqrt{km}$) are computed for each joint and applied.

Support Conditions

Only vertical movement is allowed for all members.

The end of the pile joint is fully fixed. The mass considered in the pile joints comprises the mass moving as a rigid body in the vertical direction, i.e. the sum of the mass for the outer tank (excluding the roof) and the inner tank. The stiffness is defined from user input. These values are summarised in the spreadsheet shown in [Fig 69].

If foundation tab is specified in Tank Definition, it is reflected in model by connecting joint elements in series.



Fig 53 Supports in a Beam-Stick Vertical Model

Loadings / Analysis Control / Response Spectrum / Options for post-processing

These values and settings are the same as those for the model for horizontal action.

See *Examples – User Inputs* : for more information.

Exporting Forces from the 2D Axisymmetric Model

Section forces for the 2D Axisymmetric Solid Model are exported and saved as a spreadsheet.

The stress distributions at the slicing lines can be converted into section forces as shown in [Fig 71]. For example, SZ through the wall section can be used for computing vertical axial forces and bending moment.

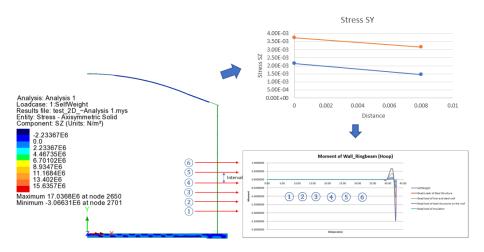


Fig 54 Converting Stress to Forces

The forces for the sliced section are automatically calculated by the Wizard from LNG Tank> Export Forces to Excel (2D).

- **Output file name** is for the name of result spreadsheet.
- **Target** is for selecting members from which the results will be exported.
- **Range** is for defining the range of results that will be exported.
- □ **Interval** is the distance between the slicing lines that are temporarily created at regular intervals for results calculation.

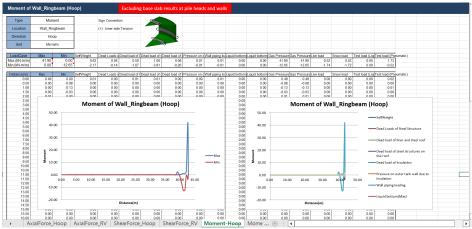


Fig 55 Section Force Spreadsheet for 2D Axisymmetric Solid Model

Roof - Exporting Forces

	LNG Tank - Export Forces/Morr	nents to Excel (2D)			×
Analysis: Analysis 1 Loadcase: 1.SelfWeight Results file: 2D Axisymmetric Solid 2D -Analysis 1.mys	Output filename Working folder Save in Target Base slab	Tees, Reol/D-d Current @ User Define CUbertichtasic/Document/Luse200/Project/Test, R Usertichtasic/Document/Luse200/Project/Test, R Uset + Ringbeam		Roof	
Component: SZ (Units: N/m²)	Loadcases		Range (X Coord	i)	
7.5676166 0.0 7.5676168 115256 32.702856 30.270456 37.83956 37.83956	1:SetWeight 2:Dead Loads of Steel Structure 3:Dead load of liner and steel ro 4:Dead load of steel structures o 8:Dead load of Insulation 6:Pressue on outer tank wall di 7:Well piping loading 8:Liquid bottom(Max) 9:Liquid bottom(Max)	on the roof	Start : Finish : Interval :	0.0 43.59 0.5	m m
45.4056E6 52.973326E6 at node 2647 of element 2174 Minimum -13.6576E6 at node 2665 of element 2182	10 Gas Pressure(Max) 11 Gas Pressure(Max) 12 Live load 13 Snow load 14 Test load (Liquid) 15 Test load (Pneumatic)		ОК	Cancel	Help

Fig 56 Exporting Forces for a 2D Axisymmetric Solid Model (Roof)

The 'Roof' group is used for extracting forces. The range is defined for x coordinates from centre of roof to the perimeter of the roof. The interval value is the arc length of the slicing locations.

Wall - Exporting Forces

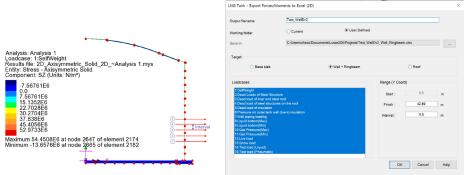


Fig 57 Exporting Forces for a 2D Axisymmetric Solid Model (Wall)

The 'Wall' & 'RingBeam' groups are used for extracting forces. Values of 'Start' and 'Finish' for the range are automatically defined for Y coordinates measured from the bottom to the top end of the wall and ring beam.

Base Slab - Exporting Forces

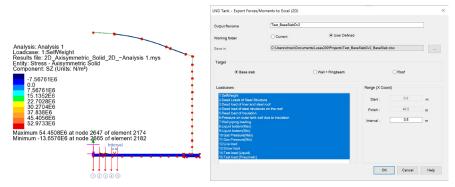


Fig 58 Exporting Forces for a 2D Axisymmetric Solid Model (Base Slab)

The 'BaseSlab' group is used for extracting forces. Values of 'Start' and 'Finish' for the range are defined for X coordinates from the centre to the perimeter of the base slab.

Exporting Forces of Specific Named Groups

This can be used not only for the Wizard built model but also for the user-built models, providing that the relevant groups are defined in the model with the name of **Wall_RingBeam**, **Roof**, **BaseSlab** and that the **Structural Definition** part in the **Tank Definition** is defined.

LNG Tank - Export Forces/Mome	ents to Excel (2D)					×	
Output filename Working folder Save in	2D Extracted Forces Current C:\UserS\ohsso\Documents\Lusas200\Projects\2D Extracted Forces_BaseS\ab.xlsx						
Target ● Base slab		◯ Wall + Ringbeam		○ Boof			
Loadcases 1:SelfWeight 2:Dead Loads of Steel Structure 3:Dead load of steel Structures 4:Dead load of steel structures or 5:Dead load of steel structures 6:Pressure on outer tank wall due 7:Wall piping loading 8:Liquid bottom(Max) 9:Liquid bottom(Max) 9:Liquid bottom(Max) 9:Liquid bottom(Max) 11:Gas Pressure(Max) 11:Gas Pressure(Max) 12:Live Ioad 13:Snow Ioad 14:Test Ioad (Preumatc) 15:Fest Ioad (Preumatc)	n the roof		Range (X Coord) Start : Finish : Interval :		m m m		
			ОК	Cancel	Help		

Fig 59 Exporting Forces for a 2D Axisymmetric Solid Mode

Exporting Forces from the 3D Shell Model

Section forces for the selected slicing angles in the 3D Shell Model are extracted by the wizard and exported to a spreadsheet. This is the same as would be done within Modeller by selecting the menu item **Utilities > Graph Through 2D** for selected loadcases and selecting slicing angles.

This can be used not only for the Wizard built model but also for user-built models, providing that the relevant groups are defined in the model with the name of **Wall_RingBeam**, **Roof**, and **BaseSlab**.

LNG Tank - Export Results to Excel (3D)	×
Output filename test Working folder • Current • User Defined Save in C:\Users\ohsso\Documents\Lusas200\Projects\test	_[group name].xlsx
Target ○ Base slab ○ Wall + Ringbeam ○ Roof ● All	Range Angles : 20 degree (eg. 10; 20; 30) Interval : 0.5
Results to extract Forces and Moments 🗹 Design results 🗹	Exclude forces on the base slab at pile heads and wall Crosswise piles : 0.7 [m] Circumferential piles : 0.8 [m]
Utilisations ULS UtilPM UtilShear PM Capacity Shear Capa UtilDecompression Compression Depth	^{scity} No design code is enabled
Loadcases Combinations only SelfWeight Coad of Steel Structure Dead Load of Iner and steel roof Dead Load of Insulation Pressure on outer tank wall due to insulation Case Pressure(Max) Dictade Dead (Hage) Dictade Dead (Liquid) Dictate (Resumptio) Dictate (Resumptio) Dictate (Resumptio) Dictate (Resumptio) Dictate Dead (Resumptio) Dictate Dea	Slicing Line Angle (Positive Direction) X axis (0 Degree)

Fig 60 Exporting Forces for a 3D Shell Model

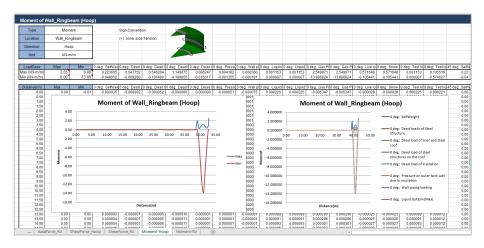


Fig 61 Section Forces Exported from a 3D Shell Model

Examples – User Inputs

This chapter explains how user inputs are used in Wizard-built models. The aim is to give users more understanding about the models created, so that they can be updated for performing other analysis tasks, or to trouble shoot any issues with their models.

Tank Definition

The examples in this manual are all based on data defined using this dialog.

k Definition								
			Target models to build					
Material : Metallic		\sim	2D axisymmetric structural	🗹 2D axisymmetric coup	oled thermal/struc	tural		
Elevation : Abovegr	ound	~	2D beam-stick seismic	✓ 3D shell structural				
· · · · · · · · · · · · · · · · · · ·								
k Definition Load Insulation	ons Support (3D)) Seismic Groun	d					
se Slab Wall Roof Lateral	Roof Polar Bea	m Section Materia	als Support (2D)					
ase slab								
Description	L[m]	D[m]	^					
nner part (Linner, Dinner)	39.8	1.2	CL					
			-	Linner		Louter		
Outer part (Louter, Douter)	6.7	1.5	-					
aper part (Wt)	0.6	0.0	D _{inner}	Dheating		1	G.L.	
			······································			heating	D _{outer}	
ase Heating (Lheating,	46.5	0.386				↔	Dground	
			- v			w		
c			> .					
			,					
			,					
			,					
·			,					
			·					
			<u>,</u>					
			<u>></u>					
			<u>, , , , , , , , , , , , , , , , , , , </u>					
			· · ·					
			<u>,</u>					
			· · · ·					
			<u>,</u>					
			<u>,</u>					
			· · · ·					
	fefaults		<u>,</u> ,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,					
	lefaults		<u>,</u>					
	lefaults		<u>,</u>					
	fefaults		<u>,</u>					
	lefaults		<u>,</u>					
	lefaults	Name			v 😌 (t)			
	Iefaults	Name			 ✓ ◆ (1) 			

Fig 62 Tank Definition Dialog

- □ **Material** One of tank material type should be selected between 'Concrete' and 'Double Steel'.
- □ Elevation One of elevation type should be selected between 'Aboveground' and 'Elevated/Isolated'.
- □ 3D shell Structural This option should be checked to define each pile location and its properties in a 3D shell model. If checked (ticked) the Support(3D) tab will appear
- □ 2D Axisymmetric Coupled Thermal-Mechanical This option should be checked for Thermal analysis where insulation should be modelled. If checked (ticked), extra tabs for insulation properties will appear.
- □ 2D Beam-Stick Seismic This option should be checked for Seismic Analysis. If checked (ticked) the Seismic and Ground tabs for seismic data will appear.

Structural Definition

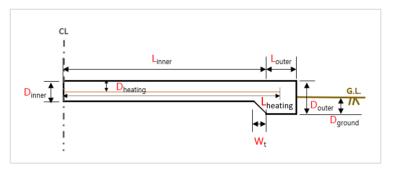
Metallic Tank

ink type		Target models to build		
Material : Metallic	~	2D axisymmetric structural	2D axisymmetric coupled thermal/structural	
Elevation : Abovegro	und ~	2D beam-stick seismic	SD shell structural	
k Definition Load Insulatio				
ase slab Description	L[m] D[m	^		
Inner part (Linner, Dinner)	39.8 1.2	CL		
Outer part (Louter, Douter)	6.7 1.5		Linner	
Taper part (Wt)	0.6 0.0	Dinner	Dheating GL	
Base Heating (Lheating,	46.5 0.38	3		
Set zero Set dr	əfaults			
Setzero Set de		Name Trikt	 ✓ (t) 	

Fig 63 Tank Definition Dialog (Tank Definition/ Base Slab)

Base Slab

Dimensions for the Base Slab should be entered. The input value must be a positive numerical value.



- □ Inner Part (L_inner): Defines the length of the circular part of the base slab where the piles are arranged orthogonally.
- □ Inner Part (D_inner): Defines the depth of the circular part of the base slab.
- □ Outer Part (L_outer): Defines the length of the annular part of the base slab where the piles are arranged in an annulus.
- **Outer Part (D_outer):** Defines the depth of the annular part of the base slab.
- □ Taper Part (W_t): Defines the length of the tapered section if it is considered in the model.
- □ Base Heating (L_heating, D_heating): Defines the length of heating line from the center of the tank and the depth from the top surface of the base slab to the heating line if base heating is considered in the analysis. Base heating is installed to maintain constant temperature in base slab.
- Ground Level (D_ground): Defines the height from the top surface of ground to the bottom of the base slab. This value is used if soil is included in a thermal analysis.

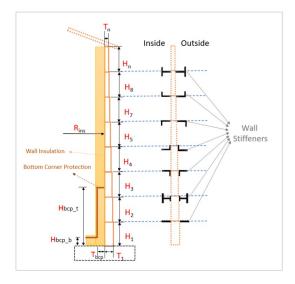
Examples – User Inputs

ank type					dels to build axisymmetric structural	✓ 2D axisymmetric coupled thermal/structural
Elevation		veground			peam-stick seismic	✓ 20 axisymmetric coupled memaisticcural ✓ 3D shell structural
nk Definiti ase Slab Radius	Wall Roof La			ction Materials Suppor	t (2D)	anna To
Inside	e radius (Rins)	4	3.2	[m]		Inside Outside
-leight / T	hickness		Stiffener			H _n
Wall ID	Height (H) [m]	Thickness (T) [mm]	section	Stiffener Stage A		H ₈
1	3.0	8.0	0	Out Y		H ₇
	3.0	8.0	0	Out Y		R _{ins} H _s Wall Stiffeners
	3.0	8.0	0	Out Y		Wall Insulation
•	3.0	8.0	0	Out Y	Add	
5	3.0	8.0	0	Out Y v	Remove	Hbcp_t H ₂
	rotection (Units rotection start (0.617		Hbcp_b \overline{T}
Corner pr	otection end (I	H_bcp_e)*		5.617		* Guidance for corner protection inputs based on the current insulation data
Corner pr	otection thickr	ess (T_bcp)*		0.155	-	- Corner protection start: 0.105 or 0.567 or 0.617 or 0.6915
Set 2	tero S	et defaults				- Corner protection end : 5.617 - Corner protection thickness: 0.155
				Name Tnk1		~ 🗘 (1)

Fig 64 Tank Definition Dialog (Tank Definition/ Wall)

Wall

Dimensions for the Wall should be entered into the boxes. The input value must be a positive numerical value.



- **Inside radius (Rins):** Defines the inner radius of the double steel tank wall.
- **Height (H):** Defines the height of each wall section.
- **Thickness(T):** Defines the thickness of each wall section.
- □ Stiffener Section ID: Defines stiffener section ID of each wall section. Zero should be defined if no stiffener is included in the wall section. Otherwise, you must enter a value that matches one of the Section IDs defined in *Beam Section* tab.
- □ Stage N/Y: Defines whether the stage should be separated at each wall section. 'Y' should be selected if the stage should be separated at the wall section. Otherwise 'N' should be selected.
- □ Corner Protection Start: Defines the height of the corner protection start based on the top of the base slab.
- **Corner Protection End:** Defines the height of the corner protection end based on the top of the base slab.
- **Corner Protection Thickness**: Defines the thickness of corner protection.

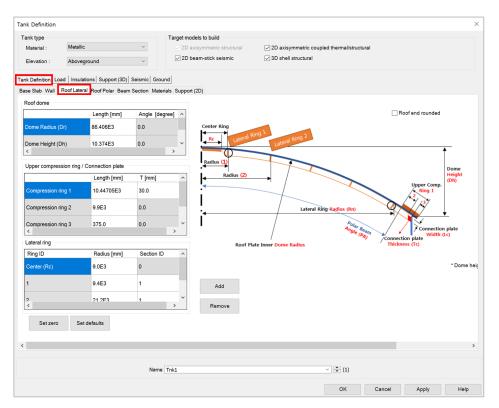
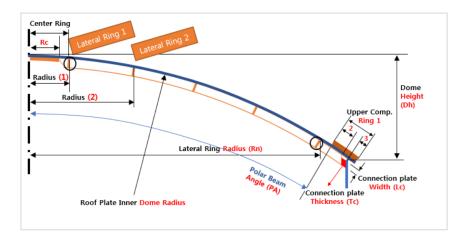


Fig 65 Tank Definition Dialog (Tank Definition/ Roof Lateral)

Roof Lateral

Dimensions for the Roof should be entered. The input value must be positive numerical value.



Roof dome

- **Dome Radius (Dr):** Defines the inner radius of roof.
- **Dome Height (Dh):** Defines the height from the top of Wall to the top most of the inner roof.
- **Polar beam angle (Pa):** Defines the angle from the center of the roof to the left side of the Roof End.

Upper compression ring/Connection plate

- Compression ring 1 Length(mm): Defines the length of upper compression ring 1.
- □ Compression ring 1 Thickness(mm): Defines the thickness of upper compression ring 1.
- □ Compression ring 2 Length(mm): Defines the length of upper compression ring 2.
- □ Compression ring 2 Thickness(mm): Defines the thickness of upper compression ring 2.
- Compression ring 3 Length(mm): Defines the length of upper compression ring 3.
- □ Compression ring 3 Thickness(mm): Defines the thickness of upper compression ring 3.
- **Connection plate Length (Lc, mm):** Defines the length of connection plate.
- □ Connection plate Thickness (Tc, mm): Defines the thickness of connection plate.

Lateral Ring

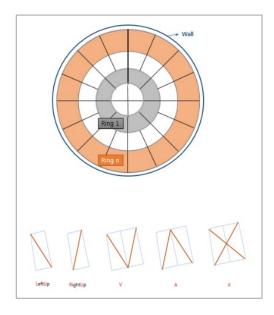
- **Radius(mm):** Defines the radius in hoop direction of each lateral ring.
- □ Section ID: Defines the section ID of each lateral ring. The defined section ID must match one of the Section IDs defined in *Beam Section* tab.

ank type				Tara	et models to	build
terial :	Metallic		~	-		netric structural
			~		2D beam-s	
Elevation :	Abovegrou	und	~			
k Definition	.oad Insulation	s Suppor	t (3D) Seismic	Ground		
			Beam Section		upport (2D)	
Plate / Polar t	eam / Bracing Plates	No of				
RingID	thickness	polar	Polar beam section-ID	No of bracings	Bracing type	Bracing section-ID
	[mm]	beams		-		
Center (Rc)	8.0	4	4	0	LU	0
1	8.0	92	3	0	LU ·	2
			-	-		
2	8.0	92	3	4	LU ·	2
3	8.0	92	3	4	RU ·	2
Set zer	n Set de	faults	Add	Be	move	
Set zer	o Set de	faults	Add	Re	move	
Set zer	o Set de	faults	Add	Re	move	
Set zer	o Set de	faults	Add	Re	move	
Set zer	o Set de	faults	Add	Re	move	
Set zer	o Set de	faults		Re Iame Tnk1	move	
Set zer	o Set de	faults			move	

Fig 66 Tank Definition Dialog (Tank Definition/ Roof Polar)

Roof Polar

Thickness of Roof plates, the number of stiffeners, type of stiffeners and section IDs should be entered. The input values must be positive numerical value.



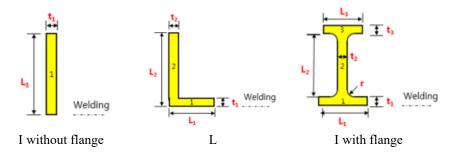
- **Plates Thickness (mm):** Defines the thickness of each roof plate.
- □ No. of Polar Beam: Defines the number of polar beams defined in each ring ID.
- □ Polar Beam Section-ID: Defines the section ID for polar beam. The defined section ID must match one of the Section IDs defined in *Beam Section* tab.
- □ No. of Bracing: Defines the number of bracings. If 'V', 'A', or 'X' type is selected for Bracing Type, the number of bracing set (two beams are in a set) should be defined.
- **Bracing Type:** Defines the bracing type among 'LU', 'RU', 'V', 'A' and 'X'.
- □ **Bracing Section-ID:** Defines the section ID for bracing. The defined section ID must match one of the Section IDs defined in *Beam Section* tab.

	pe	Target models to build							
Mater	tterial : Metallic		2D axisymmetric coupled thermal/structural						
Elevation : Above		Aboveground	Aboveground V 2D beam-stick seismic					mic	☑ 3D shell structural
e Sla		d insulations Roof Lateral Ro L1 [mm] 100.0 100.0 100.0 100.0 100.0				L3 [mm] 0.0 0.0 80.0 80.0) t3 [mm] 0.0 0.0 10.0 10.0	r [mm] 0.0 10.0 10.0 10.0	Roof Wall Count Clockwise Towards Center Clockwise Towards edge
									I (no flanges)
									I (no flanges)

Fig 67 Tank Definition Dialog (Tank Definition/ Beam Section)

Beam Section

Dimensions for the beam sections should be entered into the boxes. The input value must be a positive numerical value.



□ Shape: Defines the shape of section

- I without flange - L1, T1 are required
- L
 L1, T1, L2, T2 are required
- I with flange - L1, T1, L2, T2, L3, T3, r are required

Insulation

Tank Definition						
Tank type Material : Elevation :	Metallic Abovegroun	ıd	~		els to build symmetric structural am-stick seismic	☑ 2D axisymmetric coupled thermal/structural ☑ 3D shell structural
Tank Definition Lc		Support (3D)	Seismic Gro	und		
Dimensions Mate						
Base insulation		r4 Layer5 Laye	r 6 Louor 7	Lavor 9		(Upto 6 segments can be defined for each Layer)
ID	Length	Thickness	Material ID	Layer o	Set zero	Base and Wall Insulation
1	1.7	0.105	1		Set defaults	
2	41.5	0.105	2		Add	
Layer 1 Layer	2 Layer 3 Laye Length	r 4 Layer 5 Lay Thickness	er 6 Material ID		Set zero	Start Pontion d' Wall Insultion Base Layer 9 Base Layer 4
			Material		Set zero	of Wall Insulation
1	4.9255	0.155	3		Set defaults	Base Layer 3 Base Layer 2 Base Layer 2
2	36.0485	0.155	5		Add	Reference Position X Base Layer 1 O.= (0.0)
Roof insulation Layer 1 Layer	ns (Units: m) 2 Layer 3 Laye	er 4			Set zero	Concere Roof Insulation
ID	Length	Thickness	Material ID		Set defaults	
1	0.4829	0.745	5	_		Roof Layer 1
2	42.1	0.745	7		Add	
			Name	Tnk1		(1)
						OK Cancel Apply Hel

Fig 68 Tank Definition Dialog (Tank Definition/ Insulation)

- □ Length: Defines the length of each segment of insulation in each layer. Rows for additional segments can be added to each layer by clicking the 'Add' button on the right.
- □ **Thickness:** Defines the thickness of each segment of insulation in each layer. Rows for additional segments can be added to each layer by clicking the Add button on the right.

- □ **Material ID:** Defines the material properties that are assigned to each segment of insulation. The ID must match one of the material properties that is defined in the *Insulation Materials* tab in *Material Properties* tab.
- **Set Zero:** Sets all the input values to zero for the specific Insulation.
- **Add:** Add a row to define a new segment for each layer of Insulation.
- **Remove:** Removes the selected row.
- □ The sum of the height of the Wall Insulation and the total thickness of Base Insulation should not exceed the sum of the height of the Ringbeam_1 and the Wall Height.

Base Insulation

A maximum of 6 layers of base insulation can be defined.

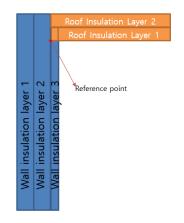
Wall Insulation

A maximum of 4 layers of wall insulation can be defined.

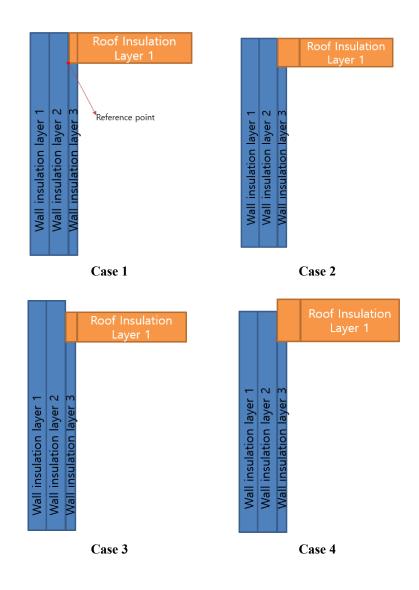
Roof insulation

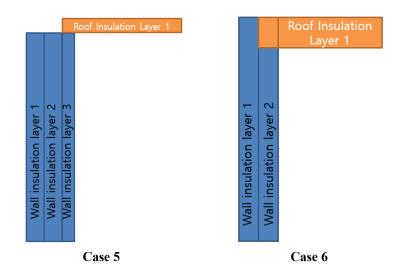
Roof insulation layers are assumed to sit on top of the innermost layer of wall insulation.

- The top-left point of the innermost layer of wall insulation is used as the 'reference point' for modelling the roof insulation.
- The sum of the total length of the roof insulation for a layer and the total thickness excluding the last layer for the wall insulation should be equal to inner diameter of concrete wall.



Several examples of defining wall and roof insulation follow:





Case 1

3 wall insulation layers and 1 roof insulation layer are defined.

- Length of wall insulation layer3 + Thickness of roof insulation layer1 = Length of wall insulation layer2
- Thickness of wall insulation layer3 = Length of 1st segment of roof insulation layer1

Case 2

3 wall insulation layers and 1 roof insulation layer are defined.

- Length of wall insulation layer3 + Thickness of roof insulation layer1 = Length of wall insulation layer2
- Thickness of wall insulation layer3 < the length of the 1st segment of roof insulation layer1

Case 3

3 wall insulation layers and 1 roof insulation layer are defined.

- Length of wall insulation layer3 + Thickness of roof insulation layer1 < Length of wall insulation layer2
- Thickness of wall insulation layer3 = Length of the 1st segment of roof insulation layer1

Case 4

3 wall insulation layers and 1 roof insulation layer are defined.

- Length of wall insulation layer3 + the thickness of roof insulation layer1 > the length of wall insulation layer2
- Thickness of wall insulation layer3 < the length of the 1st segment of roof insulation layer1

Case 5

3 wall insulation layers and 1 roof insulation layer are defined

- Length of wall insulation layer1/Layer2 and Layer3 are identical
- Thickness of wall insulation layer3 < the length of the 1st segment of roof insulation layer1

Case 6

2 wall insulation layers and 1 roof insulation layer defined.

- Length of wall insulation layer2 + the thickness of roof insulation layer1 = the length of wall insulation layer1
- Thickness of wall insulation layer2 = the length of the 1st segment of roof insulation layer1

Material Properties

Tank type			Targe	t models to build					
Material :	Metallic		× 🗸	2D axisymmetric :	structural	✓ 2D axisymmetr	ic coupled thermal/	structural	
Elevation :	Abovegroun	d	~	2D beam-stick se	ismic	☑ 3D shell structu	ural		
		Support (3D) Seis	mial Cround						
		of Polar Beam Sec		pport (2D)					
Material ID			Poisson's ratio (v)	Mass density [kg/m³]	CTE (alpha, [1/C])	Thermal conductivity [J/m.s.C]	Heat capacity [J/m³/C]	Description	
Concrete (Base)		35.0E9	0.2	2.5E3	10.0E-6	2.0	2.257E6	BaseSlab	
oncrete (Wall)		209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Wall	
oncrete (Roof)		209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Roof	
iteel (Upper comp	ression ring)	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Roof Upper Compression Ring	
olar Beam		209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Rafter Beam	
ateral Beam		209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Lateral Beam	
racing		209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Bracing	
Connection Plate		209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Bracing	
aft		35.0E9	0.2	2.5E3	10.0E-6	0.0	0.0	Raft	
'ile (Cir)		35.0E9	0.2	2.5E3	10.0E-6	0.0	0.0	Pile (Cir)	
* Isolator proper Set zero	ties can be de Set defa	fined for various ty	pes from modeler	and should be de	fined and assign	ed manually.			
			Name Tnk1				~ 🗘 (1)		

Fig 69 Tank Definition Dialog (Tank Definition–Materials)

The *Tank Definition / Materials* tab contains the material properties for the Concrete (Base), Steel (wall), Steel (Roof), Steel (Upper compression ring), Polar Beam, Lateral Beam, Bracing, Connection Plate, Pile(Cr) and Pile(Cross) required for the modelling the structure. Thermal Conductivity and Heat capacity should be entered only when thermal analysis is carried out.

ype erial: Me	tallic	~	Target models to	build netric structural	✓ 2D axisy	mmetric coupled th	ermal/structural
ation : Ab	oveground	~	✓ 2D beam-st	ick seismic	☑ 3D shell	structural	
finition Load	sulations Support (3D) Seismic Groun	d				
sion: Materials							
Material ID	Elastic modulus [E, N/m²]	Poisson's ratio (v)	Density (rho, [kg/m³])	CTE (alpha, [1/C])	Thermal conductivity [J/m.s.C]	Heat capacity [J/m³/C]	Description
Soil	1.0	0.2	0.0	10.0E-6	2.0	1.97E6	Soil
1	1.0	0.2	0.0	10.0E-6	2.0	2.257E6	Levelling concrete
2	1.0	0.2	0.0	10.0E-6	0.79	2.257E6	Dry Sand
3	1.0	0.2	0.0	10.0E-6	0.0446	90.4344E3	Cellular Glass(Type1)
4	1.0	0.2	0.0	10.0E-6	0.27	753.163E3	Perlite Concrete(Base Slab)
5	1.0	0.2	0.0	10.0E-6	0.0411	51.2E3	Loose Fill Expanded Perlite(Wall)
6	1.0	0.2	0.0	10.0E-6	0.032	9.6E3	Resilient Glass Fibre Blanket(Wall)
7	1.0	0.2	0.0	10.0E-6	0.0372	18.0627E3	Glass Fibre Blanket(Roof)
at zero Set	t defaults Add	d Remov	<i>(</i> 2)				
2010 30	Add Add	Kemov					

Fig 70 Tank Definition Dialog (Insulations / Materials)

The *Insulation Materials* tab should list all of the material properties of each type of insulation required for the modelling the structure. The unique ID numbers must include all of the material properties that have been assigned in the *Insulations* tab in *Tank Definition* tab

Boundary Conditions

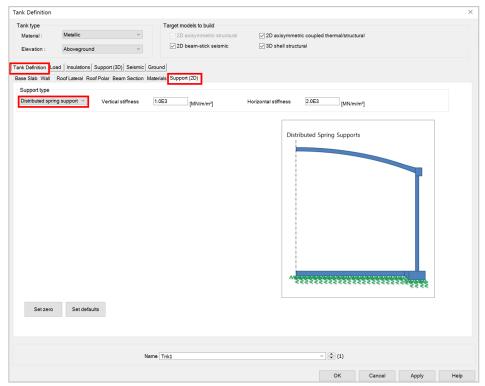


Fig 71 Tank Definition Dialog (Support(2D)- Distributed spring support)

Support Type

This tab defines the support type for the bottom of the base slab. Options are: 'Fixed Support', 'Pile Support' or 'Distributed spring support'. If 'Pile Support' is selected, the pile stiffness for spring supports should be defined and the unique ID numbers must include all of the pile stiffness. If 'Distributed spring support' is selected, one vertical and one horizontal stiffness should be defined. The stiffnesses should be a positive value in MN/m/m².

ank type Material :	Metallic	~		odels to build	2D axisymmetric coupled thermal/structural
Elevation :	Aboveground	· · · · · · · · · · · · · · · · · · ·	✓ 20	beam-stick seismic	☑ 3D shell structural
se Slab Wall		Polar Beam Sector		ort (2D)	
Support type		rolar beam beam	indicinals		
Pile Support	~		l.	Jpdate from Support(3D)	
Pile stiffnesses					
Spring ID	Radius [m]	Vertical stiffness [MN/m/rad]	Horizontal stiffness [MN/m/rad]	Description	Pile Supports
	4.2	592.0	47.9	Pile	
2	8.4	1.0524E3	85.1	Pile	
3	12.6	1.5987E3	127.7	Pile	
4	16.8	2.1049E3	170.2	Pile	
5	21.0	2.6311E3	212.8	Pile	
6	25.2	3.1573E3	255.3	Pile	
7	29.4	3.4962E3	282.7	Pile	
8	32.9	3.7295E3	301.6	Pile	Spring Number
9	36.7	4.6615E3	377.0	Pile	1 2 3
<				>	
Set zero	Set default	s Add	Remove		
			Name Tnk1		× (1)

Fig 72 Tank Definition Dialog (Support(2D)- Pile Support)

Spring Stiffness for Piles

This tab defines the vertical and horizontal stiffness for the piles. The stiffness should be entered as a positive value in MN/m/rad. The radius is the distance from the centre of the tank to where each equivalence spring support is located.

Loading

Tank type			Target mod	els to build	
Material : Metall	ic	~	🗹 2D ax	symmetric structural	☑ 2D axisymmetric coupled thermal/structural
Elevation : Above	ground	~	☑ 2D be	am-stick seismic	☑ 3D shell structural
ank Definition Load nsul Structural Dead Loading St		ort (3D) Seismic			
Load type	Notation	Dimension [m]	Value [kN/m ² , kN/m]	Description	- Dead load of steel structure
Dead Loads of Steel Str	q_1	42.1	1.343	Steel Structure_q1	
Dead Loads of Steel Str	q_2	0.0	0.0	Steel Structure_q2	
Dead Loads of Steel Str	q_3	0.0	0.0	Steel Structure_q3	d
Dead Loads of Steel Str	q_4	0.0	0.0	Steel Structure_q4	
Dead Loads of Steel Str	Ρ	42.1	72.9	Steel Structure_P	
Dead Loads of Steel Str	q_r	42.1	0.098	Steel Structure_qr	
Dead load of liner and st	q_1	43.2	0.404	Liner_base1	
Dead load of liner and st	q_2	0.0	0.0	Liner_base2	
Dead load of liner and st	q_3	0.0	0.0	Liner_base3	
Dead load of liner and st	q_4	0.0	0.0	Liner_base4	
Dead load of liner and st	q_r_liner	43.2	1.095	Liner_Roof	_
<					>
					Set zero Set defaults
		N	ame Tnk1		✓ 🗢 (1)
					• • • •

Fig 73 Tank Definition Dialog (Load-Structural Dead Loading)

Structural Dead Loading

This defines the structural dead loading to consider in analysis.

- □ Loading Type Defines the type of structural loading including dead load and hydrostatic load. Data tips and other details such as load direction and where to apply can be seen on the right.
- □ **Dimension(m)** Defines the loaded length in metres. Negative loaded lengths are not permitted and may give an error message. A zero loaded length means that the loading is not considered in the analysis.
- □ Value Defines the magnitude of the structural dead loading in units of kN per square metre or kN per metre length. A positive value should be entered regardless of the loading direction. The structural loading will be automatically defined by correctly matching the load direction shown the load assignment image.

Tank type Material Metalli	_	~	Target mod							
Material : Metalli	c	Ť		kisymmetric structu				ed thermal/structu	iral	
Elevation : Above	ground	~	✓ 2D be	eam-stick seismic		3D shell	structural			
nk Definition Load neul tructural Dead Loading Str Load type Liquid bottom Liquid bottom		Dimension [m] 42.1 0.0		Min value [kN/m², kN/m] 183.662 0.0 0.0	Description Liquid_q1 Liquid_q2 Liquid_q3		- Liquid Botto	m		
Liquid bottom	q_4_liquid	0.0	0.0	0.0	Liquid_q4					
Gas Pressure	P_g	0.0	29.0	29.0	GasPres					
Live load	q_r_live	0.0	1.2	0.0	LiveLoad					
Snow load	q_r_snow	0.0	1.2	0.0	SnowLoad					
Test load (Liquid bottom)	P_hydro	42.1	183.662	0.0	Hydrosta			L.		ر الدر الدر الدر الدر الدر الدر الدر الد
Test load (Liquid wall)	P_hydro	0.0	0.0	0.0	Hydrosta		1	<u>.</u>		q3
Test load (Pneumatic)	P_pneu	0.0	1.2	0.0	Pneumat			q1		q ₂ q4
							Set zero	Set defaults		
		N	ame Tnk1					× 🗘 (1)		

Fig 74 Tank Definition Dialog (Load – Structural Variable Loading)

Structural Variable Loading

Defines the structural variable loadings to consider in analysis.

- □ Loading Type: Defines the type of structural loading including dead load and hydrostatic load. Data tips and other details such as load direction and where to apply can be seen on the right.
- □ **Dimension[m]:** Defines the loaded length in metres. Negative loaded lengths are not and may give an error message. A zero loaded length means that the loading is not considered in the analysis.
- □ Max/ Min Value: Defines the magnitude of structural variable loading in units of kN per square meter or kN per meter length. A positive value should be entered regardless of the loading direction. The structural loading will be automatically defined by correctly matching the load direction shown the load assignment image.

Tank type		Target models t	o build			
Material : Metallic	~	🖂 2D axisyn	nmetric structural	2D axisymmetr	ric coupled thermal/structural	
Elevation : Aboveground	~	✓ 2D beam-	stick seismic	G 3D shell structu	ural	
	1					
	oort (3D) Seismic Gro					
Structural Dead Loading Structural Varia	ble Loading	Loading				_
Load type	Length [m]	Temperature [C]	Convective coefficient [J/m ² .s.C]	Type of boundary	Description	
Initial Temperature (Structure)	0.0	15.1	0.0	Prescribed	Initial Temperature of Structure	
Initial Temperature (Soil)	0.0	15.1	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	25.6	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	
	* The second second				n le define d	
Set zero Set defaults	i ne temperature	ioi pase neating wi	il omy be considere	ed if a value other than ze	io is delined.	
	Nam	e Tnk1			 (1) 	

Fig 75 Tank Definition Dialog (Load – Thermal Loading)

Thermal Loading

- □ Loading Type: Defines the type of temperature loading including Liquid Temperature, External Temperature, Base Heating, Initial Temperature (structural, soil), Soil bottom Depth & Temperature and Spillage Temperature.
- **LNG Temperature**: LNG Temperature which is applied to the inside of the inner tank.
- **External Temperature:** Ambient temperature applied to the outer tank.
- □ **Base Heating:** Temperature for the base heating system that is applied to the heating line if a base heating system is considered in an analysis. The heating line could be defined in the *Structural Definition* tab. If any value except zero is entered (which is the distance from the top of the base slab to the heating line) then the base heating temperature will be considered in the analysis.
- □ **Initial Temperature:** Initial temperature that is applied to whole model. Thermal stress is zero at this temperature.
- □ Soil bottom Depth & Temperature: Soil bottom temperature that is applied to bottom of the soil.

- □ Convective Coefficient: Defines the convective coefficient that is only required when Convection is entered for the Type of Boundary.
- □ **Type of Boundary:** Defines the type of boundary which should be selected. Options are: **'Prescribed'** or **'Convection'**. If **'Prescribed'** is selected, LUSAS Prescribed temperature is used to define temperature loading and the temperature where the loading is applied will be maintained at the defined value. If **Convection** is selected, **Convection Coefficient** should be entered and LUSAS Environmental temperature is used to define temperature loading. The temperature where the loading is applied will vary by the convection coefficient entered.

Support (3D)

type				Та	rget models to	build								
	letallic		~		-	metric structural		✓ 2D axi	symmetric c	oupled the	ermal/struc	tural		
evation : A	boveground		~		✓ 2D beam-s	tick seismic		✓ 3D she	ell structural					
			_											
	Insulations	Support (30	D) seismic	Ground										
Support														
pport type		С	Circumferent	ial Support									• • • • • •	
		- r)/0	tical	Horizont ^				rosswise piles	• • •
nplified foundation		~	ID	R [m]	Initial theta [degree]	Number of piles	stiff	ness I/m]	stiffnes: [kN/m]	A	dd			ţţ.
o.cir: 18	4		1 3	6.7	0.0	56	523.01	8E3 42	2.297E3	(Del		Circu	mferential piles
o. cross : 21	3		2 4	0.8	0.0	60	523.01	8E3 42	2.297E3	Set	zero			
	6.1965E3		3 4	49	0.0	68	523.01	853 41	297F3 ~	Set d	efaults			
X ² Cross : 63	.7157E3	[<						>					
sswise support s	tiffness													
Grid wizard			Vertical stiff	ness [kN/n	n] 523.01	8E3	Horizor	tal stiffnes	s [kN/m]	42.297	3			
< coordinates (Ur	its: m)					Y coordinate	es (Units	: m)						dd column
P1 P2	P3	P4	P5	P6	P7 ^	P1	P2	P3	P4	P5	P6	P7	^ A0	ad column
F1 F2				21.0	25.2	0.0	0.0	0.0	0.0	0.0	0.0	0.0		Add row
0.0 4.2	8.4	12.6	16.8	21.0	20.2					-	-	_	_	
	8.4	12.6	16.8	21.0	25.2	-4.2	-4.2	-4.2	-4.2	-4.2	-4.2	-4.2	D	el column
0.0 4.2					_	-4.2 -8.4	-4.2 -8.4	-4.2 -8.4	-4.2 -8.4	-4.2 -8.4	-4.2 -8.4	-4.2 -8.4	-	el column Del row
0.0 4.2 0.0 4.2	8.4	12.6	16.8	21.0	25.2									

Fig 76 Tank Definition Dialog (Support (3D))

Support Type

Options are 'Use support (2D) conditions', 'Simplified foundation', or 'Detailed foundation'.

Crosswise piles X Coordinates

Defines X coordinates for piles which are located in the fourth quadrant from the centre of the tank. The value should be a positive number. If all crosswise piles coordinates are zero, then the crosswise pile is not included, and only circumferential piles are included in the model.

Crosswise piles Y Coordinates

Defines Y coordinates for piles which are located in the fourth quadrant from the centre of the tank. The value should be a negative number. If all crosswise piles coordinates are zero, then the crosswise pile is not included, and only circumferential piles are included in the model.

LNG Tank - Define Pile Locations				×
Number of piles in X Add row in Y Remove row	13	Row in Y	Number of piles in X	
Start offset of piles in X (m) Start offset of piles in Y (m) General spacing of piles in X (m) General spacing of piles in Y (m)	0,0 0,0 3,0 3,0			
		OK	Cancel Appl	y Help

Fig 77 Tank Definition Dialog (Pile Arrangement – Define Pile Locations)

Define pile locations

- **Number of piles in X:** Defines the number of piles in the X direction
- □ Add Row in Y: Add a row in Y direction with a defined number of piles in X direction.
- **Remove Row:** Remove the last row in the Y direction.
- □ Start offset of piles in X(m): Defines the start offset of piles in X direction. If this value is zero, X coordinates for the piles in the first column are zero.
- □ Start offset of piles in Y(m): Defines the start offset of piles in Y direction. If this value is zero, Y coordinates for the piles in the first row are zero.
- **General Spacing of piles in X(m):** Defines the spacing of piles in the X direction.
- General Spacing of piles in Y(m): Defines the spacing of piles in the Y direction.
- □ Horizontal Stiffness: Defines the horizontal stiffness of the crosswise piles.

Crosswise piles stiffness

- □ Vertical Stiffness: Defines the vertical stiffness of the crosswise piles.
- **Horizontal Stiffness:** Defines the horizontal stiffness of the crosswise piles.
- **Type:** Defines the name of crosswise piles which is used as dataset name.

Circumferential piles

- **R**: Defines the radius of the ring of piles.
- □ **Initial Theta:** Defines the angle (theta) between the X axes and the location of first pile. If the first pile is placed on the X axis, then initial theta will be zero.
- □ **Number of piles:** Defines the number of piles positioned the same distance from the centre of the tank.
- **Vertical Stiffness:** Defines the vertical stiffness of the circumferential piles.
- **Horizontal Stiffness:** Defines the horizontal stiffness of the circumferential piles.
- **Type:** Defines the name of crosswise piles, which is used as dataset name.

Foundation

If 'Detailed foundation' for 'Support Type' is selected, piles are modelled with beam elements and 'Foundation' tab newly appears so that input values for foundation can be defined.

Examples – User Inputs

Tank Definition									×
Tank type			Target mo	dels to build					
Material :	Metallic			xisymmetric structural	✓ 2D axisymmetric coup	led thermal/structura			
			- 2D b	eam-stick seismic	✓ 3D shell structural				
Elevation :	Aboveground		~ 0200						
Tank Definition Load	d Insulations Support (3D) Se	aismic Ground						
Base Support Four									
Foundation									
Туре	Inclu	ude	Height/Thickness [m]	Section shape	D1 [m]				
Pile (Cir)	6	2	NA	Circular Hollow	0.8	2 2	÷.	D2	
Pile (Cross)	5	2	NA	Circular Hollow	0.7				
								\bigcirc	
								D1	
<					>		-		
Subgrade stiffne	855								
5									
	Vertical stiffness [MI	N/m/m ²	100.0	Horizo	ntal stiffness [MN/m/m²]	100.0			
Pile stiffnesses (when piles are relaced by	y sprind	g supports)						
		ertical [8E3	Crosswise piles V	ertical [kN/m]	523.018E3		
			al [kN/m] 42.297	E3		lorizontal [kN/m]	42.297E3		
		01120114	ar [dentif]			ionzontai (krivinj	42.20720		
- Material prope	rties are defined in the N	laterial	tab						
- Pile heights an	nd horizontal support will	follow	the inputs in the Ground	tab					
Set zero	Set defaults								
			News Told			× 🗘 (1)			
			Name Tnk1			× (1)			
						OK	Cancel	Apply	Help
								1460	

Fig 78 Tank Definition Dialog (Support (3D)>Foudation)

□ Foundation : Defines the section shape and the dimensions of piles (Crosswise, Circumferential)

Seismic

ank type					Target model	Is to be	uild									
Material :	Metallic			~	🗹 2D axis	ymme	tric structu	ural 💽	2D	axisymmet	ric coupl	ed therma	al/structural			
Elevation :	Aboveground	ł		~	🗹 2D bea	m-stic	k seismic	5	3D	shell struct	ural					
k Definition Loa er Tank Properti				eismic Groun												
iquid Liquid density	480.	0	[kg	g/m³]		Liqu	iid height	38.92	1	[m]						
inner tank dimer																
Inside radius	42.1		[m	ןי												
nner tank geom	etric properties 1	2		3	4	5		6	7		8	^	Thickness 6		Height 6	
Thickness(m)	0.0361	0.036	1	0.012	0.01	0.01		0.0	0.0		0.0		Thickness 5	Í	Height 5	
Height(m)	3.08	27.0		3.86	6.12	0.0		0.0	0.0		0.0	~	Thickness 4	Î	Height 4	
(>	Thickness 3	Î	Height 3	
nner tank mater	ial properties				Coefficie						_	_	Thickness 2	Î	Height 2	
	Elastic modu (E, [N/m ²])		oisson's atio (v)	Mass density [kg/m²]	of therma expansio [/C]	al	Thermal conductivi [J/m.s.C]			Description	'n		Thickness 1	Ì	Height 1	
nner Tank	200.0E9	0.3	3	7.85E3	10.0E-6	:	2.0	1.968E6		Inner Tank						
		_														
Set zero	Set defaults															

Fig 79 Tank Definition Dialog (Seismic – Inner Tank Properties)

- Liquid density This defines the LNG density for convective and impulsive mass in seismic analysis.
- **Liquid height** This defines the initial height of LNG in seismic analysis.
- □ Inner Tank Inside Radius Defines the inside radius of the inner tank which will be used to compute total LNG mass and Inner tank volume.
- □ Inner Tank Thickness and Height Defines the thickness and height of the inner tank, which will be used to compute total Inner tank volume
- □ Inner Tank Material Properties This defines the material properties of the inner tank, which will be used to create a seismic model.

Examples – User Inputs

Tank type Maerial: Meetallic Isvetion: Aboveground 2D axisymmetric structural 2D beam-stick seismic Set seismic 2D beam-stick seismic 2D beam-stick seismic Pactor 0 foot Inserve sees Bool Structure 1 4E6 Support 0 foot Inserve sees Bool Structure 1 4E6 0 foot Inserve sees Bool Structure 1 4E6 0 foot Inserve sees Bool Structure 1 4E6 1 4E6 1 4E6 1 4E6 1	
Aboveground 2 D beam-stick seitemic 2 D beam-sti	
k Vefinian Laad insulations Support (2015 Seismic Ground er Tank Properde Ton-Structural Masses Jaba Inner Steel Tank Descriptions Mass Stab Inner Steel Tank Descriptions Masse Staba Inner Steel Tank Roof Liner + steel Roof Structura Roof nozzles 42.0E3 Roof nozzles 42.0E3 Roof platform A00.0E3 Roof platform 03.0E3 Roof platform 03.0E3 Roof platform 03.0E3 Others 00.0 Total Constructuration Con	
er Tark Properier Non-Structural Masses Junior Steel Tark Roof Ring Beam Wall Bases Slab Inner Steel Tark Descriptions Masse Slab Inner Steel Tark Descriptions Masse Slab Inner Steel Tark Roof Liner + steel Roof Structure 14.6 Suspended deck & insulation of the suspended ceiling 15.0E3 Roof pozzles 0.0E3 Roof platform 0.0 Others 0.0 Total 0.0 Set zero Set defaults	
Descriptions Mass [kg] Roof Liner + steel Roof Structure 14E6 Suspended deck & insulation of the suspended ceiling 15.0E3 Roof nozzles 42.0E3 Roof platform 40.0E3 Roof platform 30.0E3 Roof piping and support 103.0E3 Others 0.0 Total 2.11E6	
Rod Liner + steel Roof Structure 1.4EB Suspended deck & insulation of the suspended ceiling 135.0E3 Roof nozzles 42.0E3 Roof platform 400.0E3 Roof pling and support 03.0E3 Others 0.0 Total 2.1E6	
Suspended deck & insulation of the suspended ceiling 135.0E3 Roof platform 400.0E3 Roof platform 30.0E3 Roof platform 30.0E3 Others 0.0 Total 10.0 Set zero Set defaults	
Roof nozzles 42.0E3 Roof platform 40.0.E3 Roof pliping and support 0.0. Total 0.0 Set zero Set defaults	
Roof platform 400.0E3 Roof plump & crane 03.0E3 Others 0.0 Total Collection Collectio	
Roof pump & crame 30.0E3 Roof pping and support 103.0E3 Differs 0.0 Total 2.11E6	
Roof piping and support 0.0 Others 0.0 Total 2.11E6	
Others 0.0 Total 2.11E6	
Total 2.11E6	
Set defaults	
Name Tnk1 V 🗘 (1)	

Fig 80 Tank Definition Dialog (Seismic - Non-Structural Masses)

Non-Structural Masses

This tab defines masses for the non-structural parts which will be used to compute additional mass for seismic analysis.

nk type		Т	arget models to build					
laterial :	Metallic	~	2D axisymmetric st	tructural	🗹 2D axisyn	nmetric coupled thermal/struc	tural	
levation :	Aboveground	~	2D beam-stick seis	smic	✓ 3D shell s	structural		
	Abbregiound							
Definition L	.oad Insulations Support (3D) Seis	mic Ground						
r Tank Prope	erties Non-Structural Masses Lumped	d Foundation						
							_	
Geometric pr	reportion						Use 3D support inputs	
Name	lopelues	Exist	Area [m ²]	Shear	area [m²]	Moment of inertia [m ⁴]	Length [m]	
Pile (Lumped	d)		617.23	540.14		297.064E3	NA	
Lumped isol	lator							
Total m	ass of lumped isolator [kg] = isolator r	mass x numb	er of base support =	15	8.8E3			
Total m Lumped pile	aass of lumped isolator [kg] = isolator i stiffnesses							
Total m Lumped pile	ass of lumped isolator [kg] = isolator r				18.8E3			
Total m Lumped pile [Vertica	aass of lumped isolator [kg] = isolator i stiffnesses	s of pile/soil [I	/N/m]	22				
Lumped pile [Vertica	ass of lumped isolator [kg] = isolator i stiffnesses I beam stick model] Vertical stiffnes	s of pile/soil [I	/N/m]	22	25.9233E3			

Fig 81 Tank Definition Dialog (Seismic – Limped Foundation)

- □ Geometric Properties: Define geometric properties for piles which will be used to build a model for seismic analysis. Piles are to be modelled with a series of elements in a single line. Values for area, inertia and stiffness for 'Pile' should be for the total of all piles acting as a group.
- □ **Total mass of lumped isolator:** Define the total mass of lumped isolator which will be used to build a model for seismic analysis.
- □ Vertical Stiffness of Pile/Soil: Define the vertical stiffness of pile/soil which will be used to build a vertical beam stick model for seismic analysis.
- □ **Rotational Stiffness of Pile head:** Define the rotational stiffness of pile head which will be used to build a horizontal beam stick model for seismic analysis.

ank typ	e				Target model	s to build						
Materi	al: M	letallic		~	🔽 2D axis	ymmetric struct	tural	2D axisymmetric	coupled thermal/stru	ictural		
Elevat	ion : A	boveground		~	✓ 2D bear	m-stick seismic	:	3D shell structur	al			
		-										
		Insulations Su	pport (3D) Se	eismic Grou	nd							
oil Prop	erties											
Stiffness	distribution:	Constant value	e	~								
.ayer No.	Soil depth [m]	Thickness of layer [m]	Static kh [MN/m/m]	Static kv [MN/m/m]	Dynamic kh [MN/m/m]	Dynamic kv [MN/m/m]	Lumped kh [MN/m/m]	Description(1		
	0.0	0.0	0.0	0.0	0.0	0.0	0.0	Start of soil p		1		
	-2.0	2.0	19.0793	0.0382	38,1586	0.0763	14.92E3	Backfill			G.L.	
	-2.0	2.0	18.0785	0.0302	30.1300	0.0705	14.8223	Dackin	Layer 1		Piles	
	-4.0	2.0	32.9527	0.0659	65.9054	0.1318	25.769E3	Backfill	Layer 2			
	-6.0	2.0	28.6317	0.0573	57.2634	0.1145	22.39E3	Backfill	Layer 3		Layer thickness	
	-8.0	2.0	27.5563	0.0551	55.1125	0.1102	21.549E3	Backfill	Layer 4			
	-10.0	2.0	30.2072	0.0604	60.4143	0.1208	23.622E3	Backfill	Layer n			
	-12.0	2.0	41.3977	0.0828	82.7954	0.1656	32.373E3	Silty Sand1				
	-14.0	2.0	34.5307	0.0691	69.0614	0.1381	27.003E3	Silty Sand1	Add	Set zero		
								· · · · · · · · · · · · · · · · · · ·	Remove	Set defaults	3	
_	stiffness											
	stiπness : vertical stiffn	ess [kN/m]	15.0E3	s	Static horizonta	l stiffness [kN/m	n] 15.	0E3				
			30.0E3			-	20	0E3				
Dyna	mic vertical st	tiffness [kN/m]	50.0E3		Dynamic horizo	ontal stiffness [k	(N/m] 30.					
				Name	Tnk1				~ 🗘 (1)			

Fig 82 Tank Definition Dialog (Ground – Soil Properties)

Soil Properties

- □ Stiffness distribution: Defines the way to define stiffness through each soil layer. Available options are 'Constant value' and 'Linear Interpolation'. If 'Constant value' is selected, stiffness through the soil layer is constant. Otherwise, stiffness will vary by linear interpolation order. The value should be negative.
- □ Soil Depth: Defines the level (elevation) of a soil layer with respect to the pile head which is at a location of zero. The value should be negative.
- □ Thickness of Layer: Defines the thickness of each layer. The value should be positive.
- **Static Kh:** Defines the static horizontal soil spring stiffness per unit length.
- **Static Kv:** Defines the static vertical soil spring stiffness per unit length.
- **Dynamic Kh:** Defines the dynamic horizontal soil spring stiffness per unit length.
- **Dynamic Kv:** Defines the dynamic vertical soil spring stiffness per unit length.
- **Lumped Kh:** Defines the lumped horizontal soil spring stiffness per unit length.

- □ Static vertical stiffness: Defines the static vertical stiffness which is applied to pile toe.
- □ Static horizontal stiffness: Defines the static horizontal stiffness which is applied to pile toe.
- **Dynamic vertical stiffness:** Defines the dynamic vertical stiffness which is applied to pile toe.
- **Dynamic horizontal stiffness:** Defines the dynamic horizontal stiffness which is applied to pile toe.

2D Axisymmetric Static Structural Analysis

User Inputs

The required user inputs for this model are as shown in [Fig 100].

ink type			Target models to build						
Material : Metallic		~	2D axisymmetric structural	✓ 2D axisymmetric co	upled thermal/struc	ctural			
levation : Abovegro	ound	~	✓ 2D beam-stick seismic	✓ 3D shell structural					
Definition Load Insulatio Slab Wall Roof Lateral ise slab	Roof Polar Bean	n Section Materi							
Description	L[m]	D[m]	^						
nner part (Linner, Dinner)	39.8	1.2		CL .					
Outer part (Louter, Douter)	6.7	1.5		Linner					
aper part (Wt)	0.6	0.0	Dinner	D _{heating}		heating	G.I	<u></u>	
ase Heating (Lheating,	46.5	0.386		I •		↔	Dground		
			>			Wt			
	tefaults		>	:		vv			
Set zero Set d	fefaults			:		ve			
	iefaults	Name		:	× R (1)	ve			

Fig 83 User Inputs for 2D Axisymmetric Static Analysis

The user dialog is available from LNG Tank>Create 2D Model> Structural as shown in [Fig 101]. Specify a model filename and set the element size to 0.2 m and press OK to build the model.

LNG Tank - Static Analysis		\times
Tank definition data	Tnk1 ~	
Model filename]
Saved model file path	C:\Users\ohsso\Documents\LUSAS200\Projects\(2D).mdl]
Concrete element size (m)	0.2	
Steel element size (m)	0.2	
	OK Cancel Help	

Fig 84 User Dialog for 2D Axisymmetric Static Analysis

Meshing

Element Type

LUSAS elements 'QAX4M', which are suitable for a 2D axisymmetric model, are defined and assigned to steel roof plate and roof frames. For lateral beams, 'BXS12' is used.

Element Size

The largest element size used in the model will be less than 0.2m as per user input.

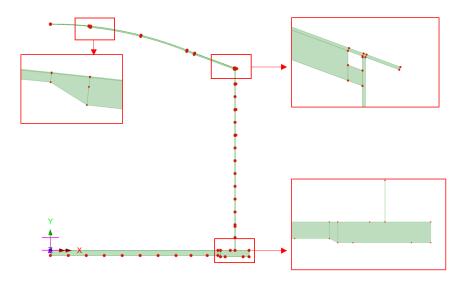


Fig 85 Mesh division for a 2D Axisymmetric Model

The numbers of mesh divisions are computed to obtain an element size smaller than 0.2m as per user input.

Geometric Properties

Roof Plate

Roof is modelled using 2D axisymmetric solid element. Note the Roof End is straight.

Polar Beams

Pola beams have bending stiffness in the roof. Although they are not axisymmetric, this should be included in the model as structural members.

Initially the polar beams could be idealized as 2D axisymmetric shell elements (see Appendix A for details), however this is not adopted because that we do not support the eccentricity in the axisymmetric shell element.

Both the roof plate and the polar beam will be modelled with 2D axisymmetric solid elements, and the lateral ring beam will be with 2D axisymmetric shell.

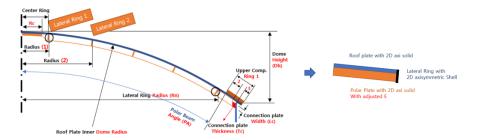


Fig 86 Polar beams for a 2D Axisymmetric Model

The thickness of the polar beam will be the same with the polar beam height.

However, considering polar beams are modelled with 2D axisymmetric solid elements and height of the polar beams are used as the thickness, the volume of the polar beam will be too much exaggerated. Hence, the equivalent elastic modulus(E) is used in order to get the section capacity closer to the reality.

Bracing

In 2D axisymmetric model, bracings are included as added masses to the polar beam members.

Wall Stiffener

Wall stiffeners are modelled as defined in Tank Definition.

Examples – User Inputs

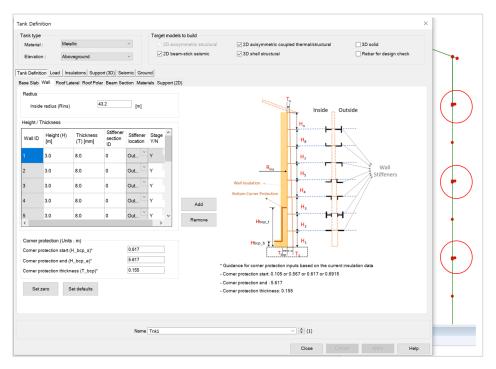
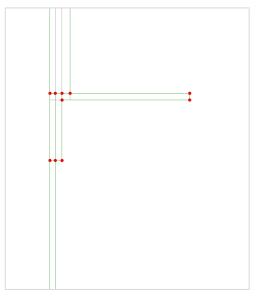


Fig 87 User input for Wall Stiffener for a 2D Axisymmetric Model





Material Properties

User defined material properties are assigned to the relevant surfaces.

Tank type			Targe	t models to build					
Material :	Metallic		× 🗸	2D axisymmetric :	structural	✓ 2D axisymmet	ric coupled thermal/	structural	
Elevation :	Abovegroun	d	~	2D beam-stick se	ismic	☑ 3D shell struct	ural		
		Support (3D) Seis							
Base Slab Wall	Roof Lateral Ro	of Polar Beam Sec	tion Materials Su	pport (2D)					
Material ID		Elastic modulus (E, [N/m²])	Poisson's ratio (v)	Mass density [kg/m³]	CTE (alpha, [1/C])	Thermal conductivity [J/m.s.C]	Heat capacity [J/m³/C]	Description	
Concrete (Base)		35.0E9	0.2	2.5E3	10.0E-6	2.0	2.257E6	BaseSlab	
Concrete (Wall)		209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Wall	
Concrete (Roof)		209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Roof	
Steel (Upper cor	npression ring)	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Roof Upper Compression Ring	
Polar Beam		209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Rafter Beam	
Lateral Beam		209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Lateral Beam	
Bracing		209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Bracing	
Connection Plate	9	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Bracing	
Raft		35.0E9	0.2	2.5E3	10.0E-6	0.0	0.0	Raft	
Pile (Cir)		35.0E9	0.2	2.5E3	10.0E-6	0.0	0.0	Pile (Cir)	
* Isolator prop Set zero		fined for various ty	pes from modeler	and should be de	fined and assign	ed manually.			
			Name Tnk1				~ 🗘 (1)		

Fig 89 User Inputs for Tank Materials

This can be found from LUSAS Modeller as shown in [Fig 112].

However, the mass density is recomputed to add the mass of bracings.

Total mass of polar beams per Ring

The total mass of polar beams per ring is computed from

1) Volume of polar beams in a ring

Vp = No. of polar beam * Area of polar beam section * Length of polar beam in the ring

Total mass of polar beam in a ring
 Mp = Vp * mass density of polar beam

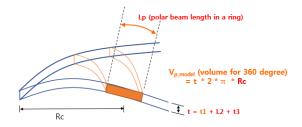
- 3) Volume of bracings in a ringVb = No. of bracing per ring * Area of bracing section * length of bracing
- 4) Total mass of bracings in a ringMb = Vb * mass density of bracing

Therefore, the total mass will be Mp + Mb.

Equivalent mass density of polar beams per Ring

The equivalent mass for the polar beam member incorporating bracings is computed as below;

 $mass_pb = (Mp + Mb) / (Vp, Total volume modelled)$



 $Vp_model = t * 2 * \pi * Rc$ (Rc = Radius (X coordinate) to the polar beam surface centroid)

Length of bracing beams

The length of each bracing is computed, which is required to compute the total mass of the bracings.

Attributes 🗸 🕈 Attributes	LUSAS View: 2D_Axisymmetric_Solid(2D).mdl Window 1 ×	
🐯 Gro 🚴 Attr 🕒 Ana 💷 Rep 🗇 Lay 🥓 Utili	Isotropic	×
	Plastic Creep Damage Shrinkage Viscous Elastic	Two phase Ko Initialisation
e e Geometric (3)		
😑 🖼 Line (3)	Dynamic properties	Value
- & 1:Lateral Beam Thickness 1	Thermal expansion Young's modulus	35.0E9
	Poisson's ratio	0.2
-& 3:Lateral Beam Thickness 3	Mass density	2.5E3
🖨 🖼 Material (15)	Coefficient of thermal expansion	10.0E-6
A : Isaesibo A : Isaesibo A : Roof Upper Compression Ring A : Roof Upper Compression Ring A : Roof Upper Compression Ring A : Staffare Beam A : Staffare Beam A : Staffarer A : Staffarer		
	Name <mark>l BaseSlab</mark>	~ (1)
	Close Can	cel Apply Help

Fig 90 Material Properties for a 2D Axisymmetric Model

Support Conditions

ank type				nodels to build	
Material :	Metallic	~		axisymmetric structural	2D axisymmetric coupled thermal/structural
Elevation :	Aboveground	~	20	beam-stick seismic	☑ 3D shell structural
nk Definition Lo	ad Insulations S	Support (3D) Seismi	ic Ground		
	Roof Lateral Root	Polar Beam Sectio	n Materials Supp	ort (2D)	
Support type					
Pile Support	~			Jpdate from Support(3D)	
Pile stiffnesses					
Spring ID	Radius [m]	Vertical stiffness [MN/m/rad]	Horizontal stiffness [MN/m/rad]	Description	Pile Supports
	4.2	592.0	47.9	Pile	
2	8.4	1.0524E3	85.1	Pile	
3	12.6	1.5987E3	127.7	Pile	
4	16.8	2.1049E3	170.2	Pile	
5	21.0	2.6311E3	212.8	Pile	
6	25.2	3.1573E3	255.3	Pile	
7	29.4	3.4962E3	282.7	Pile	
В	32.9	3.7295E3	301.6	Pile	Spring Number
9	36.7	4.6615E3	377.0	Pile	1 2 3
<				>	
Set zero	Set default	ts Add	Remove	•	
			Name Tnk1		(1)

Fig 91 User Inputs for Boundary Conditions

The user input of 592 MN/m/rad for vertical stiffness is converted to 592E6 N/m/rad in LUSAS Modeller.

tributes 🗸 🕈 🗸	LUS	SAS View: 2D_Axi	symmetric_Solie	d(2D).mdl Window 1	×					
Gro 💑 Attr 🕒 Ana 🛄 Rep 🗇 Lay 🦨 Utili	×	0.0	10.0	20.0		30.0		40.0	50.0	
- & 8:Stiffener ^ - & 13:Lateral Beam Equiv 1 - & 14:Lateral Beam Equiv 2	50.0	•		Structural Supports						;
 \$ 15:Lateral Beam Equiv 3 \$ Orthotropic (4) \$ 9:Roof Frame Equiv 1 	20			Analysis category	2D Axisyn	nmetric				
						Free	Fixed	Spr	ing stiffness	
⊖- Supports (12)	40.0				х	0	$^{\circ}$	۲	47.9E6]
→\$ 1:Pile_1 →\$ 2:Pile_Z	4			Translation in	Y	0	0	۲	592.0E6]
- & 3:Pile_3 - & 4:Pile_4					Z	۲	0	0		
-\$ 5:Pile_5 -\$ 6:Pile_6	1				х	۲				
- & 7:Pile_7 - & 8:Pile_8	30.0			Rotation about		۲	0			4
-& 9:Pile_9 -& 10:Pile_10	8				Z	۲	0	0		_
- \$ 11:Pile_11 - \$ 12:X Fixed	1			Hinge rotation		•				_
🖨 🔤 Loading (22)				Torsional warp	bing	•	0			_
B to Discrete (2) B to Discrete (2) B to Discrete (2)				Pore pressure		۲	0			
- & 19:LiveLoad - & 20:SnowLoad - Patch divisions	20.0			Spring stiffness dis	stribution					
General Structural (20) → \$1:Steel Structure_q1 → \$2:Steel Structure_P	-			O Stiffness/unit I	-					
-& 3:Steel Structure_qr -& 4:Liner_base1 -& 5:Liner Roof	10.0			Lift-off >>						
& 6:Liner_Wall & 7:SteelStructureRoof	-	Y		Contact >>						
-& 8:Insul_q1 -& 9:Insul_q2 -& 10:Insul_q3		<u> </u>		Name Pile	e_1				× 🔹 (1)	
-& 11:Insul_q4 -& 12:Insul_q4	0.0		X			_				
- & 13:Insul_Pressure - & 14:Wall piping loading			• •	1	Clos	e	Cancel	App	Help	

Fig 92 Pile Support for a 2D Axisymmetric Model

TEST CASE

If support type 'Regular Support' is chosen as shown in [Fig 115], the support definition will be as shown in [Fig 116].

Tank Definition				>
Tank type Material :	Metallic	Target models to build ZD axisymmetric structural	2D axisymmetric coupled thermal/structural	
Elevation :	Aboveground	✓ 2D beam-stick seismic	✓ 3D shell structural	
ank Definition	ad Insulations Support (3D) S	eismic Ground		
	Roof Lateral Roof Polar Beam S	ection Materials Support (2D)		

Fig 93 Test Case - Regular Support for a 2D Axisymmetric Model

A vertical stiffness of 1000 MN/m/m² is converted into 1E9 N/m in LUSAS Modeller and applied as 1E9 N/m/ m^2 by selecting the '*Stiffness/unit length*' option. (In a 2D axisymmetric model, 'stiffness/unit length' is converted to be 'stiffness/unit area'.)

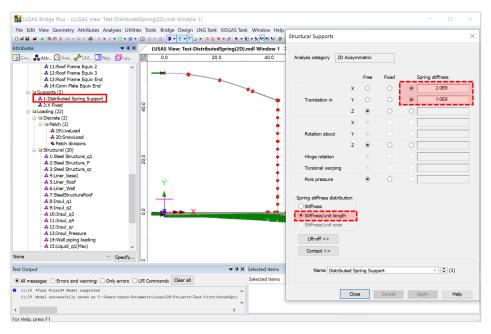
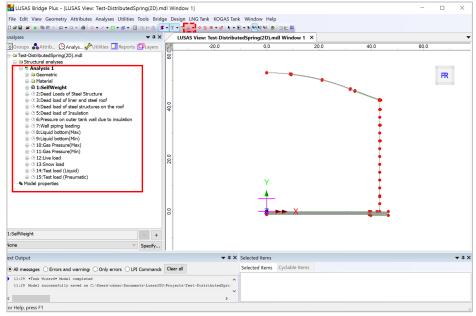


Fig 94 Test Case - Definition of a Regular Support for a 2D Axisymmetric Model

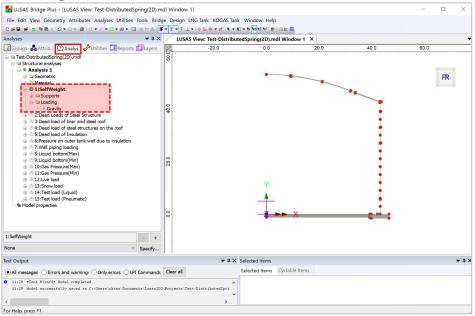
Loadings

A total of 15 loadcases is defined in the model.





Self Weight





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. In a construction situation, the dead load of suspended deck, 'qr' is evaluated as a structural load.

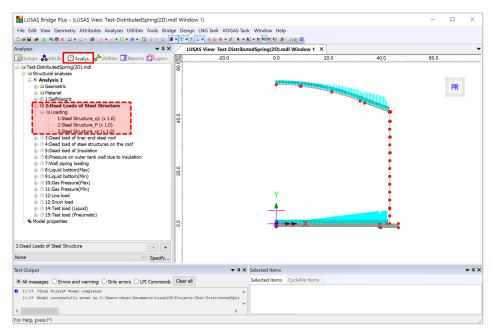


Fig 97 Dead Load for Steel Structure in a 2D Axisymmetric Static Analysis Model

Dead load of liner and steel roof

The total weight of the roof plate and frame are required to design the roof frame.

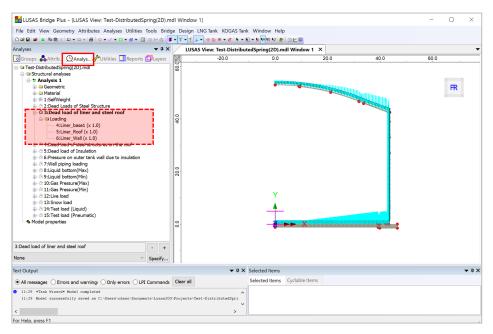


Fig 98 Dead Load of Liner and Steel Roof in a 2D Axisymmetric Static Analysis Model

Dead load of steel structures on the roof

For the design of the outer tank, the loading due to the steel structure on the roof as well as the pipework on the roof should be considered as a distributed load on the roof.

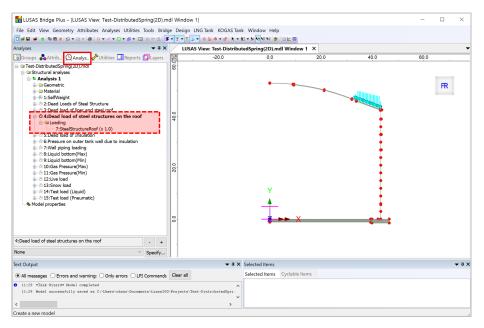


Fig 99 Dead Load of Steel Structures on the Roof in a 2D Axisymmetric Static Analysis Model

Dead load of Insulation

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

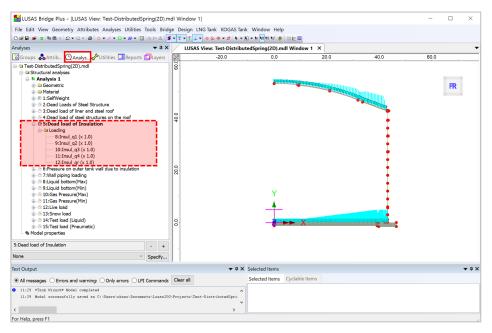


Fig 100 Dead Load of Insulation in a 2D Axisymmetric Static Analysis Model

Pressure on outer tank wall due to insulation

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

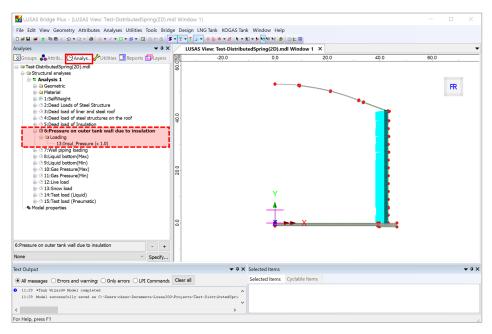


Fig 101 Insulation Pressure Load in a 2D Axisymmetric Static Analysis Model

Wall piping loading

The weight of the contained liquid acts on outer surface of wall.

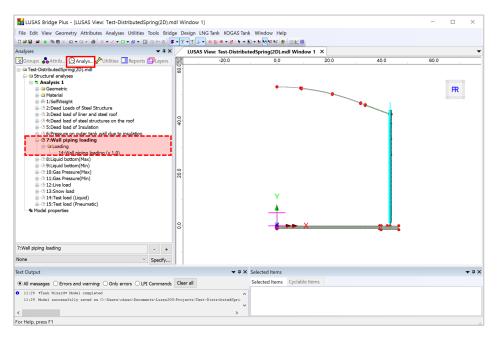


Fig 102 Wall Piping Loading in a 2D Axisymmetric Static Analysis Model

Liquid bottom (Max, Min)

The weight of the contained liquid acts on the base slab.

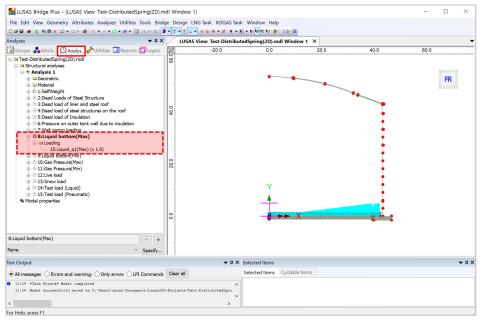


Fig 103 Liquid Bottom Loading in a 2D Axisymmetric Static Analysis Model

Gas pressure (Max, Min)

Gas pressure is assigned to the inner surface of concrete tank.

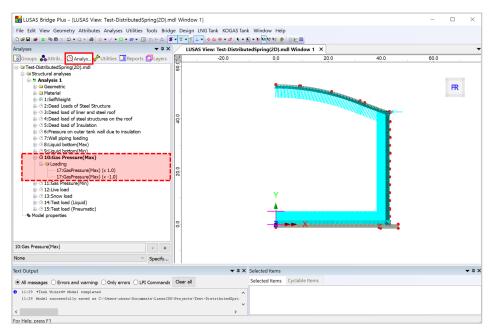


Fig 104 Gas Pressure Loading in a 2D Axisymmetric Static Analysis Model

Live load (Imposed Load on the roof)

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

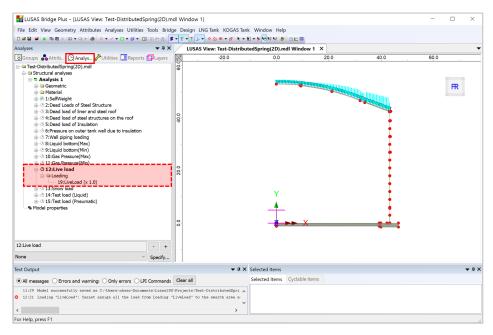


Fig 105 Live Load in a 2D Axisymmetric Static Analysis Model

Snow load

Snow load acts on the top surface of roof.

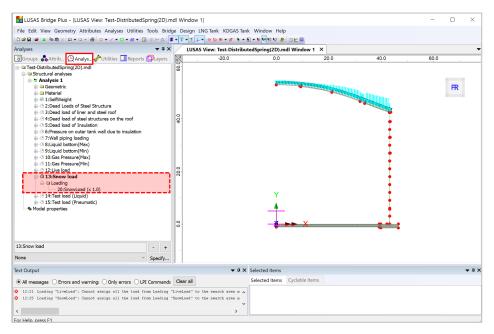


Fig 106 Snow Load in a 2D Axisymmetric Static Analysis Model

Test load (Liquid bottom)

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

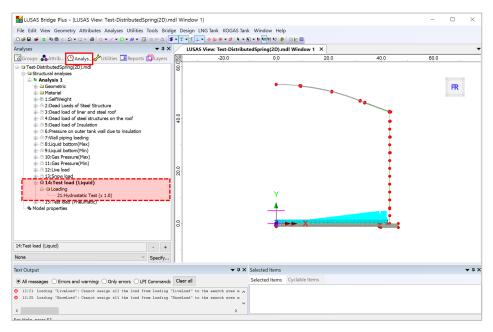


Fig 107 Test Load (Liquid Bottom) in 2D Axisymmetric Static Analysis Model

Test load (Pneumatic)

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

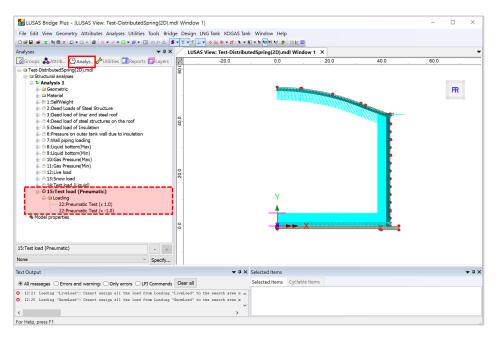


Fig 108 Test Load (Pneumatic) in a 2D Axisymmetric Static Analysis Model

Viewing Results

Contours

The Layers panel in the LUSAS Modeller user interface controls what is displayed in the View window.

Select to add **Contours** and choose **Axisymmetric-Solids** for **Entity**, **SX** for **Component**, and the contour plot for SX will be displayed. SX represents the stress in the global X direction. Positive values are for tensile stress.

Layers	▲ 廿 ×	LUSAS View: 2D_Axisymmetric_Solid(2D).mdl Window 1 ×
🖃 😂 2D_Axisymmetric_Soli		Properties
Deformed mesh	Copy Paste	Contour Results Appearance
Geometry Annotation Utilities	> Delete> Geometry	Entity Stress - Axisymmetric So 🗸
Contours : SX (Si		Component SX Display Averaged nodal
	Labels Annotation Utilities	Transform Set None
	 Contours Vectors Deformed mesh 	Display on slice(s) Draw in slice local direction
	Diagrams Values Properties	Close Cancel Apply Help

Fig 109 Selection for Contour Display in a 2D Axisymmetric Solid Model

If the 1st loadcase of Self Weight is set active, the horizontal stress of SX is displayed as shown below.

Layers 🗸 🕈 🛪	LUSAS View: test(2D).mdl Window 1 ×
🐻 Grou 💑 Attri 🚇 Analy 🥓 Utiliti 💷 Repo 🗊 Layers	50.0 -40.0 -30.0 -20.0 -10.0 0.0 10.0 20.0 30.0 40.0 5
E ← text(2D,mdl C = Ctext(2D),mdl Window 1 ← Mesh ← Deformed mesh ← Atributes ← Contour: SX (Stress - Axisymmetric Solid) ← Gannetry ← Annotation ← Utilities ▲ View properties	Analysis: Analysis 1 Loadcase: 1:SelfWeight Results file: test 2D - Analysis 1.mys Entity: Stress - Axisymmetric Solid Component: SX (Units: N/m ²) -6.20469E6 -5.17057E6 -4.13646E6 -3.10234E6 -2.06823E6 -1.03411E6 0.0 1.03411E6 0.0 1.03411E6 0.0 Maximum 2.90872E6 at node 2653 of element 2180 Minimum -6.39831E6 at node 2683 of element 2200
Deformations No deformations drawn	
Window summary Details View axes Details	
Defaults	

Fig 110 SX Contour for Self Weight in a 2D Axisymmetric Solid Model

Values

Values can be directly displayed for the chosen nodes by right-clicking on the Window entry in the Layers in the Values and adding the Values layer to the View window.

SGrou. ♣Attri (Analy ♥Utiliti IRepo (PLayers) ➡ test(2D).mdl ➡ test(2D).mdl ➡ Test(2D).mdl Wesh	Properties × Value Results Values Display R
⊨ □ test(2D).mdl Window 1	Value Results Values Display FR
Deformed mesh Arnibutes Contours : SX (Stress - Axisymmetric Solid) Generity Generity Willies View properties View properties O	Symbols Maxima 0.0 96 Values Minima 0.0 96 Deform Significant figures 6 Failure details Deform Significant figures 6 Failure details Deform Show trailing zeros Choose font Pen 19 Choose pen 0.0 OK Cancel Apply Help Maximum 6.398372E6 at node 2653 of element 2180 Minimum -6.3983126 at node 2683 of element 2200
Deformations No deformations drawn Window summary Details	, v terretaria de la constante de la constant
View axes Details	

Fig 111 Value Display in a 2D Axisymmetric Solid Model

If particular nodes are selected in the view window, the values are displayed for just those nodes.

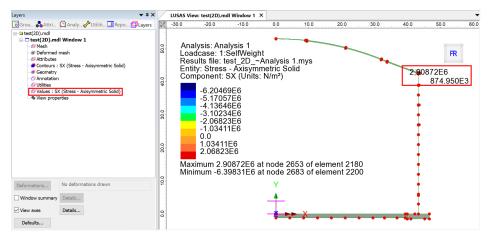


Fig 112 Values Displayed for Selected Nodes in a 3D Shell Model

Graph through 2D

Define a line from Geometry>Line>By Coords.

Ente	er Coordinat	es		\times
-0	Grid style	3 columns		
	X	Y	Z	
1		30 30	0 b	3
F			,	
Ē	ocal coordinat	e		
	Global coordin	ates		\sim
	Set a	as active local coor	dinate	
	OK	Cancel	Help	

Fig 113 Line for Slicing Results in a 2D Axisymmetric Solid Model

From Utilities > Graph Through 2D, select By selected line and SX for result component.

Graph Through 2D X	
By cursor Snap to grid Grid size 1.0 Grid offset (0.0, 0.0, 0.0) Generate new annotation line By selected line Straight line 184 V Project line In Z direction V By selected surface V At location of existing graph V	Loadcases and Extent X Loadcases (5:SelfWeight) Active All Specified Select Create new window for each loadcase Extent Visible model
OK Cancel Help	< 뒤로(B) 다음(N) > 취소 도움말

Fig 114 Graph Through 2D in a 2D Axisymmetric Solid Model (1)

ce Data	×	Display Graph	
Calculate distance as angle Arc centre Z 0.0 Calculate distance as angle Calculate dista	Resultant effects from 2D model Resultant effects from slice Mean normal stress Sz Mean stress Sz Mean shear stress Mean shear stress Mean shear stress Mean shear stress Mean shear stress per radian Media shear stress per radian Media shear stress per radian	x Ustance y SX Show grid ⊘ Show symbols ⊘ Corner labels ⊘ Auto-update Include existing graphs Use logarithmic scale % Automatic ∩ Ma min 0.0 ma ∪ Use logarithmic scale % Automatic Scale facto	anual x 1.0 x 1.0
Width for corridor averaging 0.0		Name Graph for SX Save in treeview	Display now
< 뒤로(B) 다음(N)	> 취소 도응말	< 뒤로(B) 마침 취소	도응말

Fig 115 Graph Through 2D in a 2D Axisymmetric Solid Model (2)

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

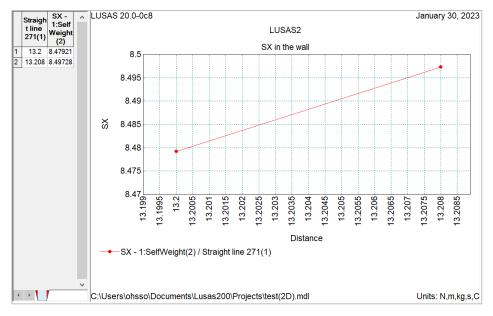


Fig 116 SX Graph for Sliced Line in a 2D Axisymmetric Solid Model

If '**Resultant effects from 2D model**' is selected from the dialog, the forces at the sliced section are computed and printed in the text window.

ce Data	×	Display Graph	
X Arc centre Y	Plearding stress is 32 Plearding stress is 32 PMean shear stress PMean shear stress PMean shear stress per radian PBending stress per radian PActual shear stress per radian	Resultant Effects X Thickness Y Results Show grid Show symbols Corner labels Auto-update Include existing graphs Øraph for SX	X scale Automatic Manual min 0.0 max 1.0 Use logarithmic scale Scale factor 1.0 Y scale Automatic Manual min 0.0 max 1.0 Use logarithmic scale Scale factor 1.0
Width for corridor	0.0	Name Graph for Self Weight	Save in treeview 🗹 Display no
L	· · · · · · · · · · · · · · · · · · ·	Name Graph for Self Weight	Save in treeview

Fig 117 Graph Through 2D in a 2D Axisymmetric Solid Model (3)

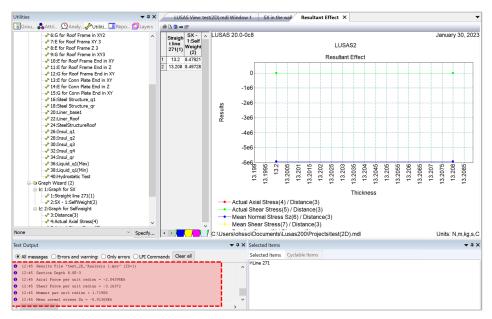


Fig 118 SX Graph for Sliced Line in a 2D Axisymmetric Solid Model

Export Forces to Excel (2D)

Forces calculated can be exported using LNG Tank > Export Forces to Excel (2D).

With the results file loaded and loadcase(s) selected in the list box, the inputs shown below will create a spreadsheet containing section forces including axial force, shear force, moment force for Wall & RingBeam.

LNG Tank - Export Forces/Mon	nents to Excel (2D)				2
Output filename	2D Extracted Force	es			
Working folder	⊖ Current	• User Defined			
Save in	C:\Users\ohsso\Do	ocuments\Lusas200\Projects\2D Extrac	ted Forces_Wall_Ring	jbeam.xlsx	
Target					
⊖ Base slab		● Wall + Ringbeam		◯ Roof	
Loadcases			Range (X Coord)	
1:SelfWeight 2:Dead Loads of Steel Structure 3:Dead load of liner and steel ro			Start :	0.0	m
4:Dead load of steel structures of 5:Dead load of Insulation	on the roof		Finish :	42.69	m
6:Pressure on outer tank wall de 7:Wall piping loading 8:Liquid bottom(Max)	ue to insulation		Interval :	0.5	m
9:Liquid bottom(Min) 10:Gas Pressure(Max)					
11:Gas Pressure(Min) 12:Live load 13:Snow load					
14:Test load (Liquid) 15:Test load (Pneumatic)					
			ОК	Cancel	Help

Fig 119 Export Forces for a 2D Axisymmetric Solid Model (1)

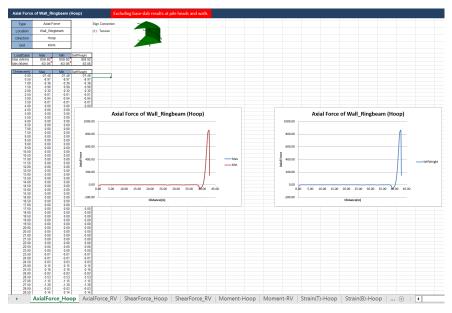


Fig 120 Section Force Spreadsheet for Self Weight

If all loadcases from the list box are selected, the forces for all loadcases are computed.

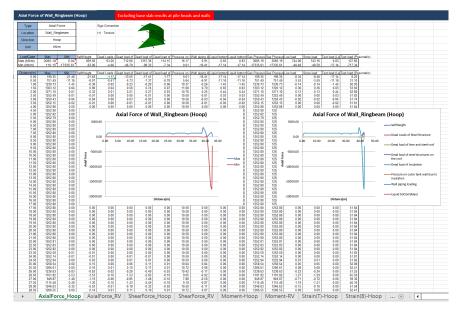


Fig 121 Section Force Spreadsheet for All Loadcases

Sign convention

Axial Force: (+) for Tension, (-) for Compression

Moment: (+) for Inner side tension, (-) for outer side tension

2D Axisymmetric Staged Construction Analysis

This example is based on the user inputs discussed in the chapter titled *Examples* – *User Inputs*.

Refer to the section titled **2D** Axisymmetric Construction Stage Analysis for more information.

User Inputs

The required user inputs for this model are the same as for 2D Axisymmetric Static Analysis.

The user dialog is available by selecting the menu item LNG Tank> Create 2D model> Staged Construction Analysis as shown in [Fig 162].

• Enter a model filename, set the element size to **0.2** m, and press **OK** to build the model.

	Tnk1	Tnk1 ~						
del filename	2D Staged	2D Staged Construction C:\Users\ohso\Documents\LUSAS200\Projects\2D Staged Construction(StagedConstruction2						
ved model file path	C:\Users\o							
Modeling options								
Concrete element size (m)	0.2		Steel element size (m) 0.2					
Loads to apply								
Self weight	Struct	ural loadings	● Max ○ Min) (Variable Loads:					
Roof construction plan Roof first stage thickness		roof option 1	1 - Base / Wall / Ringbeam 2 - Ringbeam 1st PS 3 - Roof frame 1 / Inner work					
Initial prestress for ringbe	am (ratio)	0.5	4 - Roof frames 2,3					
Initial prestress for slab (r	atio)		5 - Roof lower wet / Roof Lower complete 6 - Roof upper wet / Roof complete					
			7 - Ringbeam 2nd PS					
* Roof frame loads are not	considered		8 - Wall vertical PS					
* Roof first stage wet concr	ete is not cons	sidered	9 - Wall horizontal PS					

Fig 122 User Dialog for 2D Axisymmetric Staged Construction Analysis

Meshing / Geometric Properties / Material Properties / Support Conditions

These are the same as for the 2D Axisymmetric Static Analysis model.

Activation and Deactivation

Activation and deactivation of elements enables the modelling of a staged construction or demolition process. Activate and deactivate attributes are defined from the **Attributes> Activate and Deactivate** menu item and are assigned to features. As selected features are activated and/or deactivated the elements within those features are themselves activated and/or deactivated.

In the 1st loadcase, the 'Deactivate' attribute is assigned to all features except the annular part of Base Slab. In the 2nd loadcase, the 'Activate' attribute is assigned to the circular part of the Base Slab.

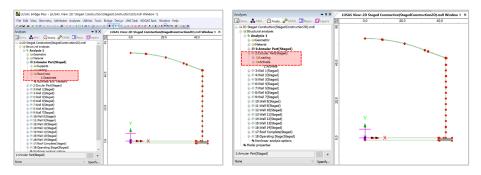


Fig 123 Activate and Deactivate Assignment in the Model

The full scenario is as illustrated at [Fig 5].

Control for Nonlinear Analysis

The geometry of the structure changes at each loadcase, so a Nonlinear Control should be defined as shown in [Fig 164]. If Nonlinear Control is set for the 1st loadcase, it is applied to all the other subsequent loadcases unless otherwise defined separately for them.

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

- $\hfill\square$ the subsequent loadcases inherit the stress and strains from the previous loadcases
- □ the subsequent loadcases inherit the support conditions from the previous loadcases
- $\hfill\square$ loading is not inherited.

2D_Staged Construction(StagedConstruction2D).mc	11				
Structural analyses	Incrementation		Solution strategy		
🗎 🛎 Analysis 1	✓ Nonlinear		Same as previous loadcase		
e 😑 Geometric	Incrementation	Manual ~	Max number of iterations	12	
Material					
= Supports	Starting load	0.1	Residual force norm	0.1	
Loading	Max change in load facto	0.0	Incremental displacement	1.0	
	Hax change in load facto		Incremental displacement		
Nonlinear and Transient	Max total load factor	1.0		Advanced.	
🖶 🗠 2:Circular Part(Staged)	Adjust load based on a	opvordopco			
⊕ ⊕ 3:Wall 1(Staged)	Adjust load based of t	onvergence	Incremental LUSAS file output		
⊕⊕ 4:Wall 2(Staged)	Iterations per increment	4	Same as previous loadcase		
🕀 5:Wall 3(Staged)	Displacement reset				
⊕ @ 6:Wall 4(Staged)	Displacement reset		Output file	1	
⊕ @ 7:Wall 5(Staged)		Advanced	Plot file	1	
🕀 🖗 8:Wall 6(Staged)	Time domain		Flot life	-	
9:Wall 7(Staged)	I Ime domain	Consolidation ~	Restart file	0	
⊕ № 10:Wall 8(Staged) ⊕ № 11:Wall 9(Staged)		Consolidation		0	
₩ (2 11:Wall 9(Staged)	Initial time step	0.0	Max number of saved	U	
₩ 12:Wall 10(Staged)		100.0E6	Log file	1	
⊕ 14:Wall 12(Staged)	Total response time	100.026	-		
Http://www.incompetition.org/actionality/a	Automatic time steppir	a	History file	1	
Hterefore (Construction of the second		-			
17:Roof Complete(Staged)		Advanced	Save a restart at the end of the	nis control	
18:Operating Stage(Staged)	Common to all				
Nonlinear analysis options					
Model properties	Max time steps	or increments 0			

Fig 124 Nonlinear Control for a Staged Construction Analysis

Loading

As the 'Manual' Nonlinear Control does not inherit the loading defined in the previous loadcases, all loading that apply to the current loadcase should be assigned separately.

Stage 1 : Annular Part ~ Stage 2 : Circular Part

Self weight is assigned by using 'Gravity' loading.

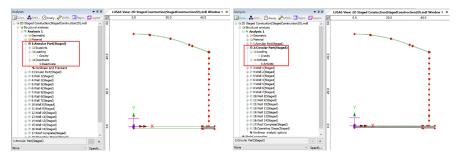


Fig 125 Loadings for Stage 1~2

Stage 3 : Wall 1 ~ Stage 16 : Wall 14

Wall segments are activated according to the user input in Tank Definition.

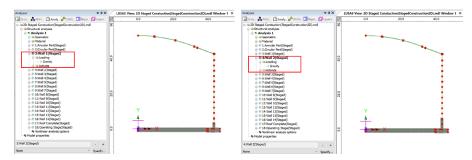


Fig 126 Loadings for Stage 3~4

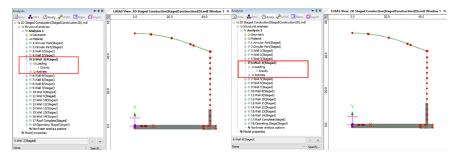


Fig 127 Loadings for Stage 5~6

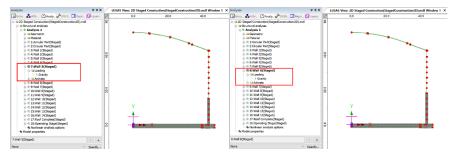


Fig 128 Loadings for Stage 7~8

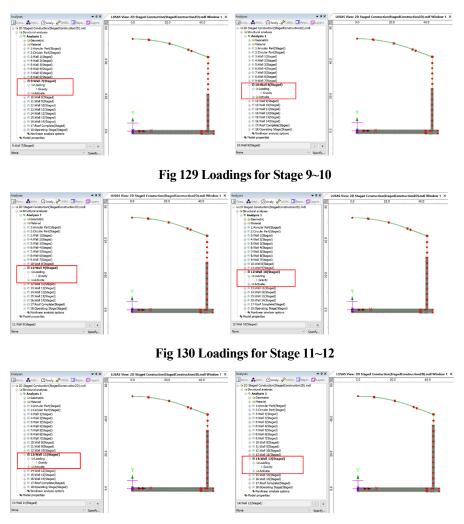


Fig 131 Loadings for Stage 13~14

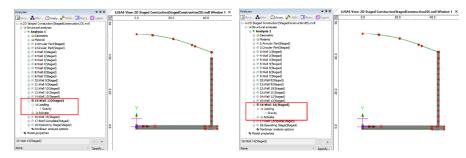


Fig 132 Loadings for Stage 15~16

Stage 17 : Roof Complete ~ Stage 18 : Operating Stage

At Stage 17, the roof is added. Stage 18 models the operating(in-service) Stage. All the loadings used in the 2D Axisymmetric Static Analysis Model are all included in this stage.

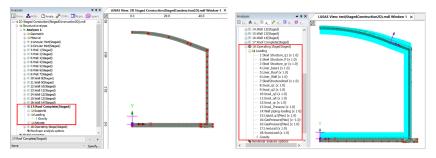


Fig 133 Loadings for Stage 17~18

Adding Extra Stages

If additional stages are required, the ability to Copy and Paste loadcases will be useful, as illustrated at [Fig 174]. Other attributes such as 'Activate' and 'Loading' are also copied.

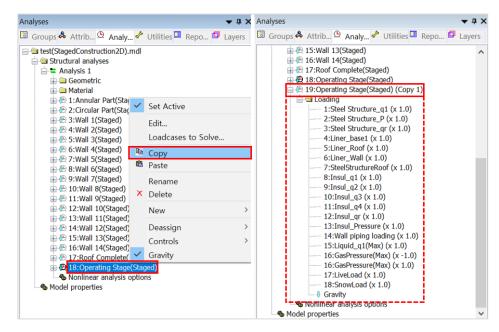


Fig 134 Adding Stages in the 2D Axisymmetric Staged Construction Analysis Model

2D Axisymmetric Thermal Analysis

User Inputs

The required user inputs for this model are as shown in [Fig 175].

ink type							>
Material : Elevation :	Metallic Abovegro	und	~	Target models to build 2D axisymmetric structural 2D beam-stick seismic	✓ 2D axisymmetric coupled then ✓ 3D shell structural	rmal/structural	
se Slab Wall Base slab Description Inner part (Lin Outer part (Lo Taper part (W Base Heating	Roof Lateral	Image: Support (3D) Root Polar Beau L[m] 39.8 6.7 0.6 46.5		rialis Support (2D)	Linner Dheating	Heating Doute \$ 71 Deces \$ 71 Dec	
< Set zero	o Set de	faults		,		•• t	

Fig 135 User Inputs for 2D Axisymmetric Thermal Analysis

The user dialog is available from LNG Tank>Create 2D Model> Coupled Thermal/ Structural...as shown in [Fig 177].

• Enter a model filename and set the Concrete element size to **0.2**, Steel element size to **0.04** and press **OK** to build the model.

LNG Tank - Coupled Th	ermal/Structural A	analysis					>
Tank definition data Model filename Saved model file path		Tnk1	nts\LUSAS200\Pr	⊃jects_Thermal.m	ıdl	~	
Modeling options Concrete element	size [m]	Steel eleme	ent size [m]	4.0E-3	✓ Include	e soil for aboveg	ground tanks
Include Structural	Load						
Variable Loads	o apply(*)				Min		
(*) These parame		a Tank Definition will be usi he [Structural Loading Def			ribute.		
Spillage Loading							
Application target a	bove Corner Protec	tion 1st 	Wall Insulation la	yer		○ Wall	
Radius of inner tan	k outer surface(*)	42.1361 [m]		id density(*)		480	[kg/m³]
(*) These paramete	rs are read from the	e [Seismic] > [Inner Tank P	Properties] tab of	the tank definitior	attribute if ava	ilable.	
Spillage duration	ume ior each spillag	je neigni					
Spillage 1	10.0 [ho	ur] Spillage 2	10.0	[hour]	Spillage 3	10.0	[hour]
Spillage 4	10.0 [ho	ur] Spillage 5	10.0	[hour]			
					ОК	Cancel	Help

Fig 136 User Dialog for 2D Axisymmetric Thermal Analysis

Meshing

Both structural elements and thermal elements are defined together. The element size will be a maximum of 0.2m as per user input. The ground is modelled up to a height of 'Ground Level' in Tank Definition above the soffit of the thickened base slab.

se slab Wall Roof L	ateral Roof Polar B	eam Section Material Su	rt (2D)	
Base Slab				
Description	L[m]	D[m]	CL	
Inner part	39.8	1.2	1	
Outer part	6.7	1.5	Linner Louter	
Taper part	0.6	0.0	D _{inner} D _{heating}	G.L.
Base Heating	46.5	0.386	₽ _{oute}	
Ground Level	0.0	0.9	4 ▶	ground



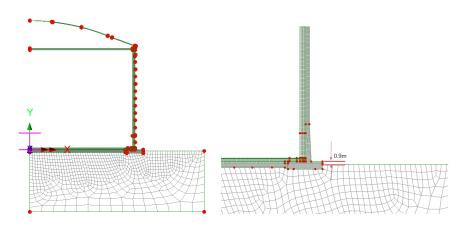


Fig 138 Elements for 2D Axisymmetric Thermal Analysis Model

Geometric Properties

No geometric properties are required for 2D axisymmetric model.

Material Properties

User defined material properties are assigned to the relevant surfaces.

otropic PElastic Thermal Plastic Creep Dam Elastic Thermal	nage Shrinikage Viscous	Two phase Ko Initialisation	Sotropic Clastic Thermal Ratic Crew Damage Strinkage Viscous Two phase Elastic Hormal	Ko Initialisation
☐Dynamic properties Ø Thermal expansion	Young's modulus Preserve initia Masa damata Coefficient of thermal expansion	Volus 3565 02 2583 10066	Phase change state None Thermal conductivity 2.2	Yaluo 000000000000000000000000000000000000
Name BaseSlab		· (1)	Name (BaseSlab ~)	(1)
	Close	incel Apply Help	Close Cancel App	ly Help

The mechanical and thermal properties for BaseSlab are as shown below.

Fig 139 Material Properties of BaseSlab for a 2D Axisymmetric Thermal Analysis Model

Support Conditions

Pile Support is used as per user input, as discussed in [2D Axisymmetric Static Structural Analysis].

Loadings

Thermal Analysis > Initial Conditions

Initial Soil Temperature is defined and assigned as shown in [Fig 184].

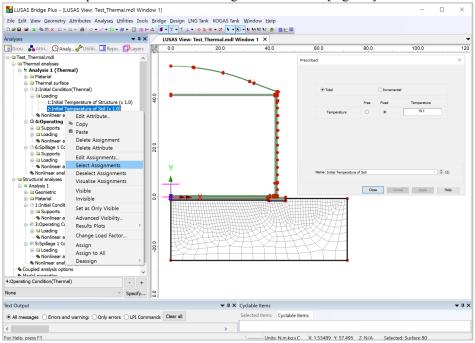
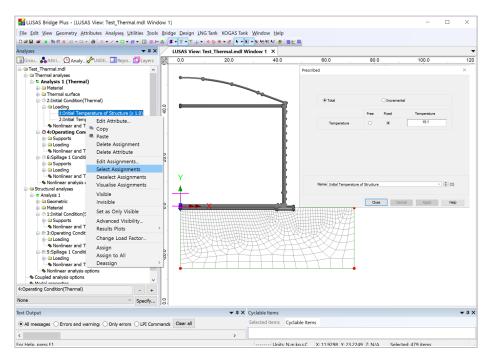


Fig 140 Initial Soil Temperature in a 2D Axisymmetric Thermal Analysis Model

Initial temperature of structure is defined and assigned as shown in [Fig 185].





Thermal Analysis > Operating Conditions

Liquid temperature is defined as an Environmental Temperature and assigned to the inner face of the tank. The air temperature is also defined as an Environmental Temperature and is assigned to the outer face of the tank.

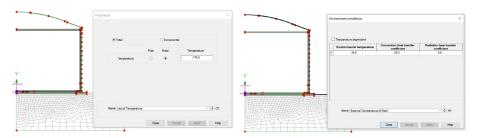


Fig 142 Operating Temperatures in a 2D Axisymmetric Thermal Analysis Model

Base heating is assumed from the User Input, which is assumed to be consistent all the time, hence it is defined as a Prescribed Temperature Loading, and assigned to the line inside base slab.

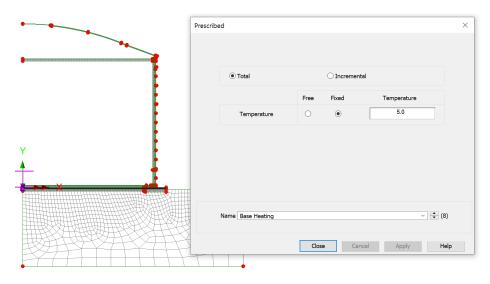


Fig 143 Base Heating Temperature in 2D Axisymmetric Thermal Analysis Model

The Base heating temperature is assigned to selected lines as shown in [Fig 188]. The range of the loading is defined using a LUSAS field variation and can be modified by editing the values of 'Base Heating'.

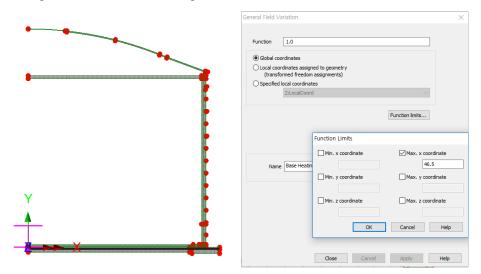


Fig 144 Base Heating Temperature Variation in 2D Axisymmetric Thermal Analysis Model

Refer to the section titled *Examples – User Inputs: 2D Axisymmetric Thermal Analysis* for more information.

3D Shell Analysis

This example is based on the user inputs described in the section titled *Examples* – *User Inputs*

Refer to the heading titled 3D Shell Static Structural Analysis for more information.

User Inputs

The required user inputs for this model are as marked in [Fig 188].

ink Definition								×
Tank type		Target	models to build					
Material: Metal	lic		D axisymmetric structural	2D axisymmetric coup	led thermal/struct	ural		
	eground		D beam-stick seismic	✓ 3D shell structural				
Elevation : Abov	eground	-				-		
ank Definition Load Insu	lations Support (3	D) Seismic Ground						
ase Slab Wall Roof Late	eral Roof Polar Be	eam Section Materials Supp	port (2D)					
Base slab								
Description	L[m]	D[m] ^						
Inner part (Linner, Dinne		1.2	CL					
inner part (Linner, Dinne	39.0	1.2	1					
Outer part (Louter, Dout	er) 6.7	1.5	4	Linner				
Taper part (Wt)	0.6	0.0	D _{inner}	Dheating		+	G.L.	
raper part (wit)	0.0	0.0	inner L			heating Do	10	
Base Heating (Lheating	, 46.5	0.386	!			•	Dground	
<		~				w		
Set zero S	et defaults							
		Name Tnk1			× 🗘 (1)			
					Close			Help

Fig 145 User Inputs for a 3D Shell Static Analysis

The user dialog is available from the LNG Tank>Static Analysis Wizard menu item.

- Enter the model file name, and set the element size to **2.0**, and the other values as shown in [Fig 189].
- Ensure '3D shell' is chosen for Analysis Type.
- Enter 10 for Number of Eigenvalues.

NG Tank - Base Model for Design Check.						
Tank definition data Tnk1	k1 ~					
Model filename Exampl	Example C:\Users\ohsso\Documents\LUSAS200\Projects\Example.mdl					
Saved model file path C:\User						
Modeling options						
Element size (m) 2.0	✓ Half symmetric model ☐ Rigid fin plate ✓ Include temporary opening					
Number of eigenvalue 10	\checkmark Include non-structural masses in the eigenvalue analysis					
Concrete Tank Options Buttress Number of buttress Extruded thickness Buttress width Buttress width Roof / Ringbeam Roof first stage thickness (ratio) Initial prestress for ringbeam (ratio) Initial prestress for base slab (ratio)	(m) Construction Scenario - Single layered roof 1 1 - Base / Wall / Ringbeam 2 - Ringbeam 1st PS 3 - Roof frame 1/ Inner work 4 - Roof frames 2,3 5 - Roof wet / Roof complete 6 - Ringbeam 2m PS 0.5 0.5 0.5 0.5					
	OK Cancel Help					

Fig 146 User Dialog for a 3D Shell Static Analysis Model

Mesh

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

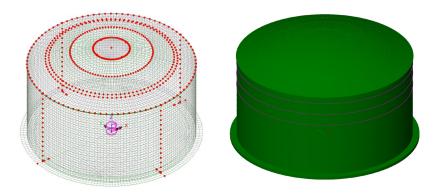


Fig 147 Mesh Arrangement and Geometric Properties for a 3D Shell Model

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.

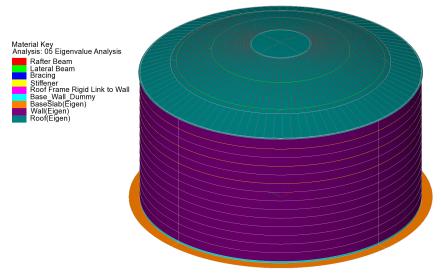
rs ayers Groups A Attrib Analys Example(3D).mdl	✓ Q × 0.0 ✓ Utilities ■ Reports	-70.0	-60.0	-50.0	-40.0	-30.0	-20.0	-10.0	0.0	
Example(30).mdl Window 1 Outros Mesh Geometry Attributes O Deformed mesh View properties	Properties Mesh Visualise			×	.44					
	✓ Wireframe Pen # Solid Hidden parts	Maximum sha	Choose pen ade 60.0 9 nold 25.0 4	b	Å					
	Show element axes % of elements remaining	Orientations only if sele				4 ++ ++	*	å ++	4	
	Close	Cancel App	Help			**	Å	Å 	4 ++	
	20.1				1	**	*	*	4 ++	
eformations No deformations drawn	10.0							٨	4	

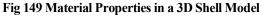
Fig 148 Element Local Axis in a 3D Shell Model

Material Properties

Structural members

Material properties are defined and assigned as shown in [Fig 193].





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.

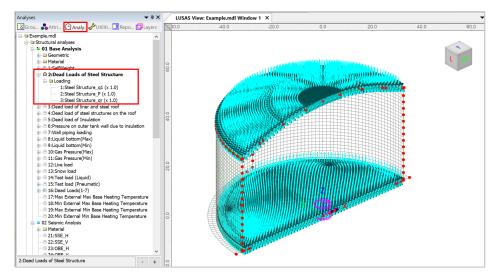


Fig 150 Dead Loads for Steel Structure in a 3D Shell Static Analysis Model

Dead load of liner and steel roof

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

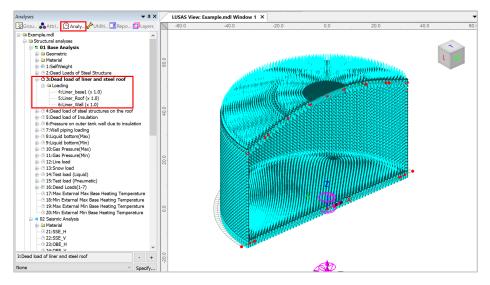
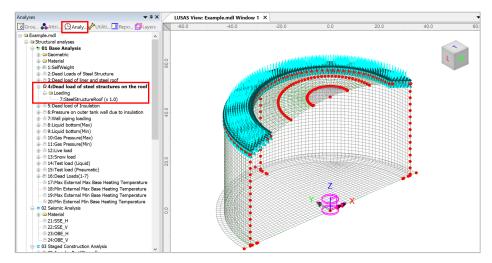
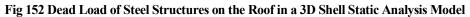


Fig 151 Dead Load of Liner and Steel Roof in a 3D Shell Static Analysis Model

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.





Dead load of Insulation

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

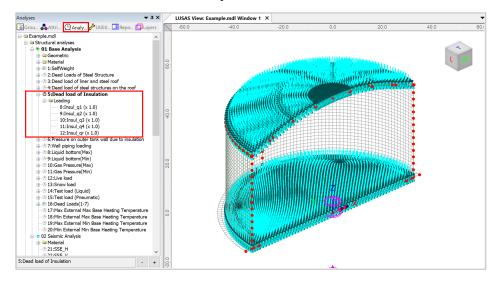
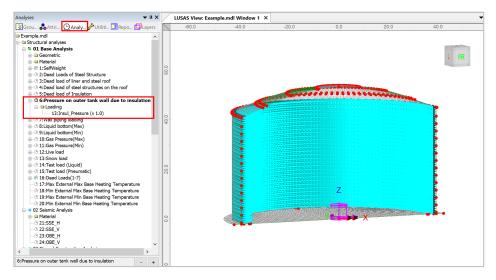
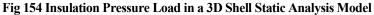


Fig 153 Dead Load of Insulation in a 3D Shell Static Analysis Model

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.





Wall Piping Loading

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

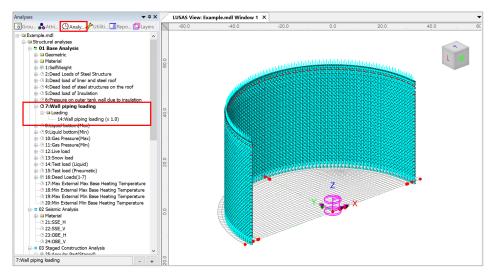


Fig 155 Wall piping loading in a 3D Shell Static Analysis Model

Liquid bottom (Max/Min)

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

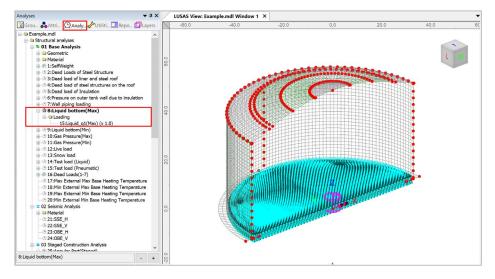
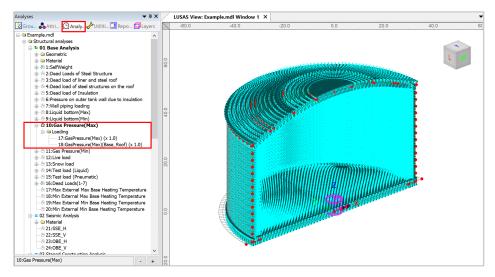
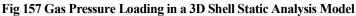


Fig 156 Liquid Bottom Loading in a 3D Shell Static Analysis Model

Gas Pressure(Max/Min)

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





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($R = 0 \sim R = 43.23$ m).

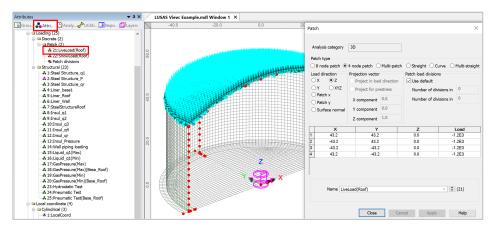


Fig 158 Live Load in a 3D Shell Static Analysis Model (Roof)

Snow load

The snow load is assigned on the top surface of the roof ($R = 0 \sim R=43.5555$)

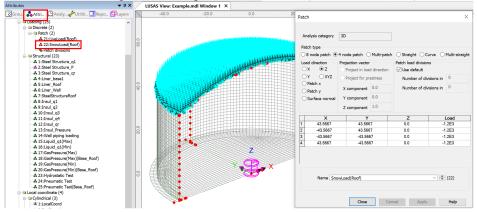
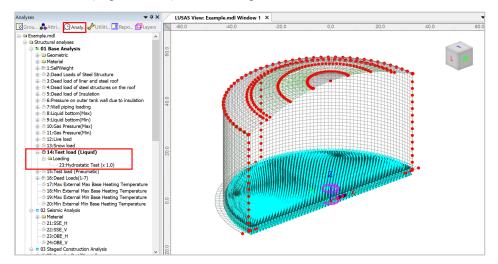
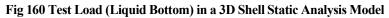


Fig 159 Snow Load in a 3D Shell Static Analysis Model (Roof)

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.

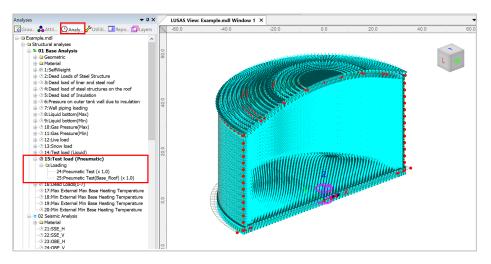


Fig 161 Test Load (Pneumatic) in a 3D Shell Static Analysis Model

Wind Load

Wind loading for the wall and roof can be added by using 'LNG Tank> Add Loading> Wind...' menu. Three types of design codes which are EN1991-1-4 (2005), GB50009 (2012), ASCE 7-16 are provided as follows.

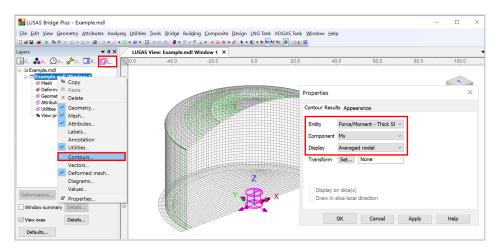
LNG Tank - Add wind lo	ading				×	
Design code Design code parameters			EN1991-1-4 (2 EN1991-1-4 (2 GB50009 (2012 ASCE 7-16	005)	~	
Basic wind velocity			37.5		[m/s]	
Roughness length			3.0E-3		[m]	
Minimum height			1.0		[m]	
Orography factor			1.0			
Terrain factor			0.156			
Turbulence factor			1.0			
Air density			1.25		[kg/m^3]	
		Defaults	ОК	Cancel	Help	
Analyses 🗸 🕈 🗙	LUSAS View:	Example.mdl Window 1	I X			•
Image: Construction of the second	40.0 0.09-	. 400 .	-200	0.0	20.0	40.0 60.0

Fig 162 Wind Load in a 3D Shell Static Analysis Model

Viewing Results

Contours

The Layers the treeview in the LUSAS Modeller user interface controls what is isplayed in the View window. Add **Contours** and choose **'Force/Moment-Thick Shell'** for Entity, **'Mx'** for Component, then the contour for Mx is displayed.





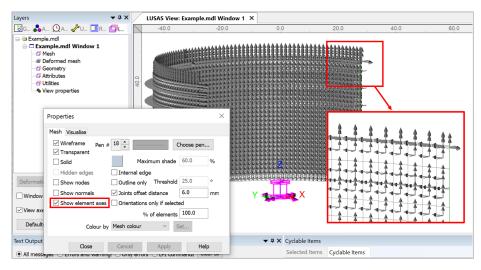


Fig 164 Element Local Axis in a 3D Shell Model

With regard to the moment in the wall, as the element local x-axis is for horizontal direction in the model, the horizontal directional moment is displayed for the selected loadcase as shown below.

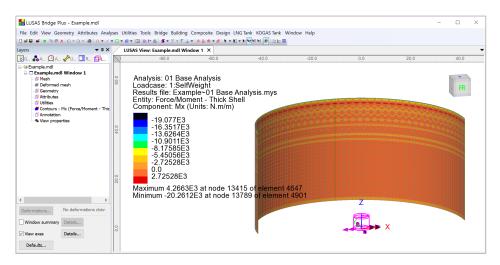


Fig 165 Mx Contour for Self Weight in a 3D Shell Model

The element local axes are not consistent in the structure as a whole, so it is recommended to use a local coordinate system for viewing results. In the Wizard-built model, a cylindrical local coordinate is already defined, with the name of **'LocalCoord'**, as shown below.

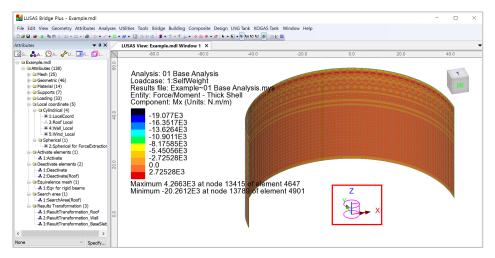


Fig 166 Local Coordinate in a 3D Shell Model

This local coordinate can be used for viewing results as shown below. Select **'LocalCoord'** for Specified local coordinate, and **'theta/z'** for Shell plane for resultants as the wall surface element axis have a theta and z direction.

LUSAS Bridge Plus - Example.mc							_	
Eile Edit View Geometry Attributes					ndow <u>H</u> elp			
00000000000000000000000000000000000000				• 🛛 • 📴 🕹 🖉 🖉 • 🖫				
		ISAS View: Example.mdl \						•
🔀 G 💑 A 🖉 A 🥓 U 🛄 R 🗯		-80.0	-60.0	-40.0	-20.0	0.0	20.0 40.0)
A View properties	Display Transparent Reset Copy Paste Delete	Analysis: 01 Ba Loadcase: 1:Se Results file: Exi Component: Mi -19.077E: -16.3517T -13.62641 -0.9011f -8.175851 -5.450561	elfWeight ample~01 Bass oment - Thick t < (Units: N.m/m 3 53 53 53 53 53 53 53	e Analysis.mys Shell	Properties Contour Results Access		Result Tantomation	T FR ×
	Move Up Move Down	-2.72528	Ξ3				No transformation applied (consult Solver mar Local axes of element/node	iual)
	Properties	0.0 2.72528E	3		Entity Force/Mo Component Mx	ment - Thick Sł 🗸	Local coordinate of parent feature Global axes	
				0.4.4	Direles Aurorad	nodal ~	O Material	
<	>	Minimum -20.20	612E3 at node 1	3415 of element 4 13789 of element	4! Transform Set	None	Assigned results transformation attribute Specified local coordinatx 1:LocalCoord	~
Deformations No deformations	drav				Display on slice(s)		Shell plane for r/theta	~
Window summary Details	0.				Draw in slice local o	tirection	No objects defined	~
View axes Details				Y	Close	Cancel Appl	⊙ x = longitudinal ⊖ y = tra	nsverse
Defaults					▼ # X Cyclable Items	s 	OK Cancel H	Help

Fig 167 Contour Display using Local Coordinate in a 3D Shell Model

If a local coordinate of 'LocalCoord' is chosen, the result component of 'Mt' can be displayed, where 't' represents tangent direction in the cylindrical local coordinate system.

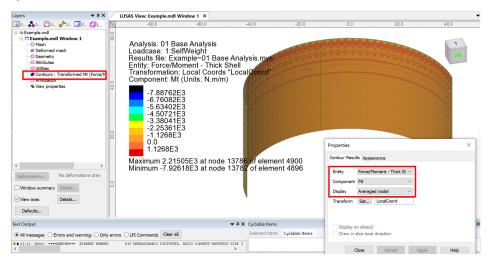
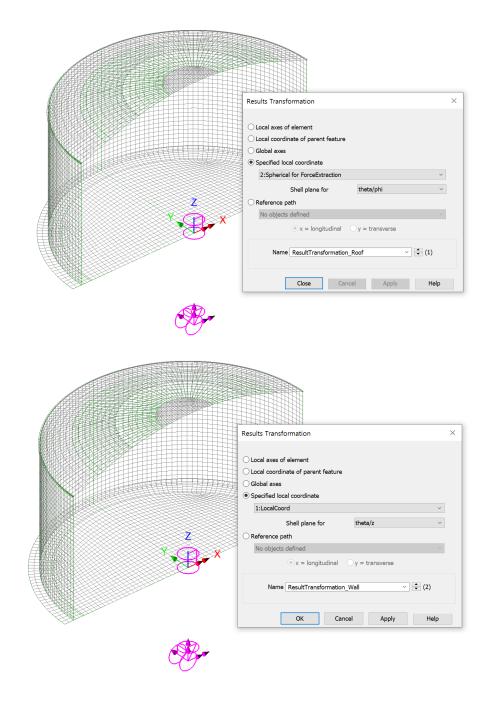


Fig 168 Mt Contour in a 3D Shell Model

In the Wizard-built model, a Results Transformation dataset is also defined and assigned to roof, wall and base slab respectively, as shown below.

Examples – User Inputs



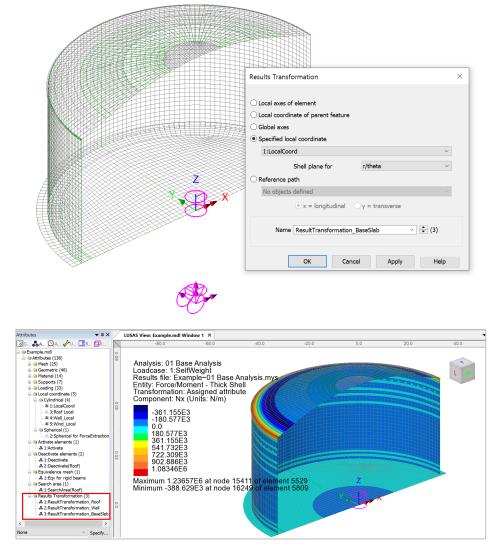


Fig 169 Results Transformation in a 3D Shell Model

This results transformation can be used for viewing results as shown below. Select **'Assigned results transformation attribute'**.

LUSAS Bridge Plus - Example.mdl						- 🗆 ×
File Edit View Geometry Attributes Analyses Ut	ilities Tools Bridge Building Composite Design LN	IG Tank KOGAS Tank	Window Help			
	9 • 🖾 m H 🛆 💈 • T • Ŧ 💺 • 🔶 💩 🗣 • Ø 🕨 • 🕅	• 🕫 ha hi hi hi 🖉 🔊 🗠 🖬				
	USAS View: Example.mdl Window 1 ×					-
🔞 G 🍰 A 🖉 A 🥓 U 💷 R 💷 L	-80.0 -60.0	-40.0	-20.0	0,0	20.0	40.0
Complement	Analysis: 01 Base Analysis Loadcase: 1:SelfWeight Results file: Example-01 Base A Entity: Force/Moment - Thick Sh Transformation: Assigned attribu Component: Nx (Units: N/m) -361.155E3	ell 📐 🤇				BKL
Properties Contour Results Appearance	Results Transformation			i i che contra da la da Gla na general da da E contra en en da da		
Congoont NN Congo	No transformation applied (consult Solver manual) Local asso of element/inde Local coordinate apress feature Gebel area Specified boal coordinate Swind Local Shell plane for Shell plane for	11 of elemen 49 of eleme	nt 5529 nt 5809	-		
Text Output	• x = longitudinal • y = transverse	× Cyclable Items				▲ 廿 X
All messages Errors and warning: Only error	OK Cancel Help	Selected Items	Cyclable Items			

Fig 170 Contour Display using Results Transformation in a 3D Shell Model

If the 'Assigned results transformation attribute' option is chosen, results components of 'Nx' and 'Ny' can be displayed. Any components with 'x' represent the results of hoop direction (wall/roof) or radial (base slab), and those with 'y' represent results of radial (roof) or vertical (wall) direction or hoop (base slab) direction.

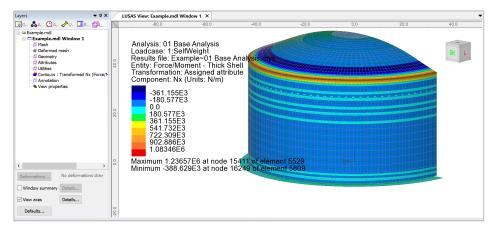


Fig 171 Nx Contours in a 3D Shell Model

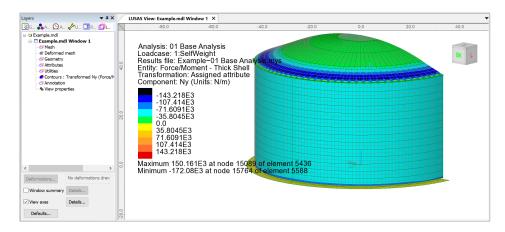


Fig 172 Ny Contours in a 3D Shell Model

Values

Values can be displayed for chosen nodes by adding the Values layer to the Layers interview.

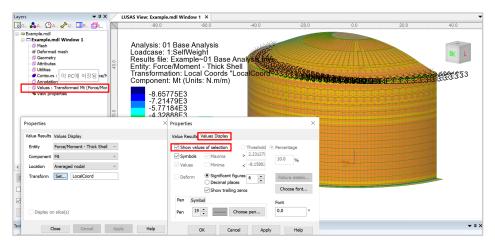


Fig 173 Value Display in a 3D Shell Model

Selecting nodes in the View window shows values for just those nodes.

yers 🔻 🕈 🛪		LUSAS View: Example.mdl Window	w 1 ×					
SG ♣A @A ≁U 💷R 💷L	20	.0 -80.0	-60.0	-40.0	-20.0	0,0	20.0	40.0
Decample and Decample.mdl Window 1 Deformed mesh Geometry Geom	20.0 40.0 60.0	Analysis: 01 Base A Loadcase: 1:SelfW Results file: Exampl Entity: Force/Mome Transformation: Loc Component: Mt (Un -8.65775E3 -7.21479E3 -5.77184E3 -5.77184E3 -2.88592E3 0.0 1.44296E3 2.88592E3	eighṫ le∼01 Base An nt - Thick Shel cal Coords "Lo	calCoord"	3.52994E3			R
Deformations No deformations drav		Maximum 3.52994E Minimum -9.45669E				Z -0.140621E-		
Window summary Details	0.0							Destruction of
View axes Details								
Defaults								

Fig 174 Values Displayed for Selected Nodes in a 3D Shell Model

Graph through 2D

Define a line from Geometry>Line>By Coords...

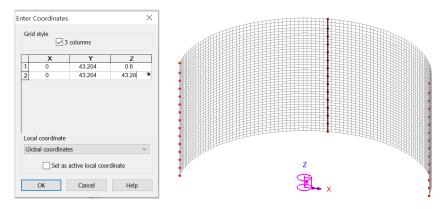


Fig 175 Line for Slicing Results in a 3D Shell Model

From Utilities > Graph Through 2D, select By selected line and Mt for result component.

Fig 176 Graph Through 2D in a 3D Shell Model (1)

Results c	omponent		 Resultant effects from 2D m Resultant effects from slice 	odel	Display		X scale			
Component Transform	Force/Moment	~	Mean normal stress Sz ∭Rending stress ⊠Normal stress Sz	^	Title Graph for Mt X Wall height Y Mt Show grid Show symbo		V scale			
	ate distance as angle X Arc centre Y Z	0.0		~	Corner labels	Auto-update	Automatic O Manual min 0.0 max 1.1 Use logarithmic scale Scale factor 1.1	0		
Width for o	orridor averaging	0.0	L		Name Graph for N	∕lt for Self Weight	Save in treeview 🛛 Disp	olay no		

Fig 177 Graph Through 2D in a 3D Shell Model (2)

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

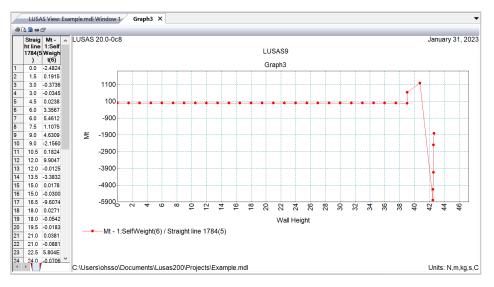


Fig 178 Mt Graph for Sliced Line in a 3D Shell Model

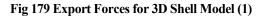
Export Forces to Excel (3D)

The forces calculated for the sliced section can be exported to a spreadsheet by selecting the menu item LNG Tank > Excel Tools > Export Forces...

- **Output file name** is for the name of the result spreadsheet.
- **Target** is for selecting members from which the results will be exported.
- □ Angles defines where slices should be taken in the model. Multiple angles can be defined by using a semi-colon (;) as a separator. (e.g. 10;20;30)
- **Interval** defines the distance between each value.

With **SelfWeight** selected from the list box for Loadcases, the inputs shown below will create a spreadsheet containing section forces including axial force, shear force and moment force for Wall + RingBeam at a slicing angle of 20 degrees.

utput filename	Test	
orking folder	Current Ourrent	
ave in	C:\Users\ohsso\Documents\Lusas200\Project	ts\Test_Wall_Ringbeam.xlsx
Target		Range
O Base slab	● Wall + Ringbeam ○ Roof ○ All	Angles : 20 degree (eg. 10; 20; 30) Interval : 0.5 [m]
oadcases		Exclude forces on the base slab at pile heads and wall Diameter of crosswise piles : 0.7 [m]
	Combinations only	Diameter of circumferential piles : 0.8 [m]
5:Dead load of Insula	and steel roof structures on the roof tition tank wall due to insulation } x)	Y axis Slicing Line Angle (Positive Direction
14:Test load (Liquid) 15:Test load (Pneum 16:Dead Loads(1-7)		X axis (0 Degree)



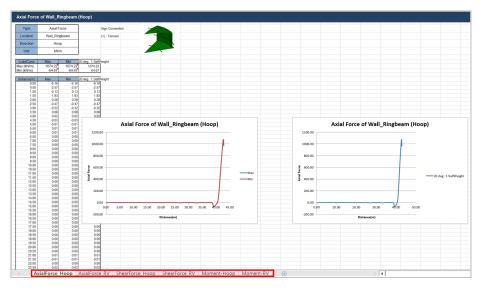


Fig 180 Section Force Spreadsheet for Self Weight

If **Angles** is defined as **0;45**, and all loadcases are selected from the list box for Loadcases, then the forces of all loadcases for the two different angles are exported and saved in the spreadsheet.

Dutput filename	Test All Loadcases		
Vorking folder	Current Ourrent		
ave in	C:\Users\ohsso\Documents\Lusas200\Projects\	Test All Loadcases_Wall_Ringbeam.	
Target		Range	
🔿 Base slab 🔘	Wall + Ringbeam O Roof O All	Angles : 0;45 degree	e (eg. 10; 20; 30)
00		Interval : 0.5 [m]	
Results to extract		Exclude forces on the base slab at pile	e heads and wall
Forces and Mom	ents	Diameter of crosswise piles :	0.7 [m]
Utilisations for concrete		Diameter of circumferential piles :	0.8 [m]
Utilisations for concrete	sections habled - No reinforcement defined M ☑ UtilShear ☑ PM Capacity ☑ Shear	Diameter of circumferential piles :	0.8 [m]
Utilisations for concrete No design code is e ULS UtiliF Loadcases	sections nabled - No reinforcement defined M UtilShear VPM Capacity V Shear Combinations only	Diameter of circumferential piles :	0.8 [m]
Design results Utilisations for concrete No design code is e ULS	sections habled - No reinforcement defined M UtilShear PPM Capacity Shear Combinations only	Diameter of circumferential piles :	0.8 [m]
Obsign results Utilisations for concrete No design code is e ULS Util Loadcases	sections habled - No reinforcement defined M UtilShear PM Capacity Shear Combinations only Structure structures on the roof	Diameter of circumferential piles :	0.8 [m]
Oesign results Utilisations for concrete No design code is et ULS UsilF Loadcases 1:SetWeight 2:Desd Loads of Steel 3:Desd load of liner an 4:Desd load of steel st 5:Desd load of steel st 7:Vall piping loading	sections nabled - No reinforcement defined M UtilShear PM Capacity Shear Combinations only Structure statee roof in	Diameter of circumferential piles :	0.8 [m]
Design results Utilisations for concrete No design code is et ULS Utili Loadcases 1:SetWeight 2:Desd Loads of Steel 1 3:Desd load of line an 4:Desd load of line an 4:Desd load of line lan 5:Desd load of linelait 6:Pressure on outer tai 9:Lagud bottom(Man) 9:Ligud bottom(Man)	sections nabled - No reinforcement defined M UtilShear PM Capacity Shear Combinations only Structure statee roof in	Diameter of circumferential piles : Capacity UtiDecompression Comp	ession Depth
Design results Utilisations for concrete No design code is et ULS UtiliF Loadcases SetWeight Coded of liner and desed load of liner and debed load of liner and debed load of liner and debed load of liner and SetWeight Se	sections nabled - No reinforcement defined M UtilShear PM Capacity Shear Combinations only Structure statee roof in	Diameter of circumferential piles : Capacity UtilDecompression Compr Y axis Slici	ession Depth

Fig 181 Export Forces for a 3D Shell Model (2)

Type	Axial F	orce		Sign Convent	ion	-																
Location	Wall_Rin	gbeam		+) : Tension																		
Direction	Hoc	ip.					- T															
Unit	kNi	m																				
LoadCase	Max	Min	0 deg: 1 SelV	0 deg: 2 De	20 deg: 3 De	20 deg: 4:Dei 2	deg 5 De 2	0 deg: 6.Pre2	0 deg: 7.Wa	0 deg: 8 Ligi	20 deg: 9 Lig	20 deg: 10 G	0 deg: 11:G	20 deg: 12 Lik	20 deg: 13:Sr	20 deg: 14 Té2	0 deg: 15:Te	20 deg: 16 Di2	20 deg: 17 M 20	deg: 18.M/20	deg: 19.Ma2	20 dec
sx (kN/m) n (kN/m)	3459.50 703.71	0.00	1074.22 -64.67	65.87	744.08	1456.31 -91.95	118.98 -6.86	14.17 3.12	1.38 -4.40	0.00	0.00	2761.89 -18104.68	2761.89 -18104.66	775.12	776.20	0.00	114.28 -749.16	3459.50 -209.77	0.00	0.00	0.00	
istance(m)	Мах			0 deg: 2.De	20 deg 3 De	20 deg 4 Dei 2	deg: 5.De 2	0 deg: 6 Pre20	0 deg: 7 Wa				0 deg: 11:G	20 deg 12 Lic	20 deg: 13.5c		0 deg: 15:Te	20 deg 16 Di2	20 deg: 17 M 20		deg: 19.M2	20 deg
0.00	948.87	-19.30 -8.04	-6.16 -2.57	-0.18	-3.60		-0.33	7.56	-4.40	0.00	0.00	948.87	948.87 1126.41	-2.25 -0.94	-2.25	0.00	39.26 46.61	-19.30	0.00	0.00	0.00	
1.00	1259.64	0.00	0.13	0.00			0.01	10.65	0.09	0.00	0.00	1259.64	1259.64	0.05	0.05	0.00	52.12	0.41	0.00	0.00	0.00	
1.50	1348.55	0.00	1.93	0.06	1.13	1.45	0.10	11.53	1.38	0.00	0.00	1348.55	1348.55	0.70	0.71	0.00	55.90	6.05	0.00	0.00	0.00	
2.00	1267.16	0.00	0.28	0.01	0.16	0.21	0.01	10.73	0.20	0.00	0.00		1267.16	0.10	0.10	0.00	52.43 50.90	0.88	0.00	0.00	0.00	
3.00	1230.10	-1.47	-0.47	-0.01			-0.02	10.36	-0.33	0.00	0.00		1280.79	-0.12	-0.12	0.00	52.17	-1.01	0.00	0.00	0.00	
3.50	1257.33	0.00	0.08	0.00	0.05	0.06	0.00	10.63	0.06	0.00	0.00	1257.33	1257.33	0.03	0.03	0.00	52.03	0.26	0.00	0.00	0.00	
4.00	1254.25	0.00	0.02	0.00	0.01	0.02	0.00	10.60	0.01	0.00	0.00		1254.25	0.01	0.01	0.00	51.90	0.06	0.00	0.00	0.00	
4.50	1251.54	-0.11			A	orce of W	- 11 0:	(I	1>		10	1251.54 1252.71	12		A!.		£ 14/-11	Dischar	am (Hoo	- 1		
5.50	1252.71	0.00			AXIAI FO	orce or w	an_kin	speam (i	поор)			1252.71	12		AXIA	ii Force c	or wall	_kingbea	am (Hoo	p)		
6.00	1253.91	0.00	5000	.00							G	1253.91	12	5000.00								
6.50	1253.17	0.00									0	1253.17	12							: 1:SelfWeigh	t	
7.00	1253.19	0.00							- N/	r	0	1253.19	12					V.				
7.50	1253.23	0.00		~ ~					~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~			1253.23	12	0.00		~	~	<u>×</u>		: 2:Dead Load	ls of Steel	
8.50	1253.24	0.00			500 10.00	15.00 20	10 25.00	30.00 35.	00 40.00	45.00	6	1253.24	12	0.00	0 10.00	20.00	10.00 40	50.00	Struct	ure		
9.00	1253.22	0.00		0.00							6	1253.22	12					1 30.00	20 des	r: 3:Dead load	of liner and	
9.50	1253.23	0.00							- 1		10	1253.23	12					1	steel #			
10.00	1253.23	0.00	g -5000	.00							2	1253.23	12 8	-5000.00					20 des	e: 4:Dead load	of steel	
10.50	1253.23	0.00	- Pe						- 1		-Max G	1253.23	12 5							ures on the roo		
11.50	1253.23	0.00	Atal Fo								-Min 10	1253.23	12 12					1		a: 5:Dead load	Indianation	
12.00	1253.23	0.00	4 -10000	.00						_	- Nein 30	1253.23	12 4	-10000.00				-	20.04	p biblead load	OF INSUMPOR	1
12.50	1253.23	0.00									0	1253.23	12									
13.00	1253.23 1253.23	0.00									2	1253.23	12							p: 6:Pressure o		6 I.
13.50	1253.23	0.00	-15000	.00						-	100	1253.23	12	-15000.00				-		ue to insulation		
14.50	1253.23	0.00									10	1253.23	12					1		g: 7:Wall pipin	g loading	
15.00	1253.23	0.00									10	1253.23	12									
15.50	1253.23 1253.23	0.00	-20000								2	1253.23	12						20 deg	g: 8:Liquid bot	tom(Max)	
16.00	1253.23	0.00	-20000			D	istance(m)				- G	1253.23	12	-20000.00		Distance	(m)					
17.00	1253.23	0.00	L									1253.23	12									
17.50	1253.23	0.00	0.00	0.00	0.00	0.00	0.00	10.59	0.00	0.00	0.00	1263.23	1253.23	0.00	0.00	0.00	51.86	0.00	0.00	0.00	0.00	
18.00	1253.23	0.00	0.00	0.00		0.00	0.00	10.59	0.00	0.00	0.00	1253.23	1253.23	0.00	0.00	0.00	51.96	0.00	0.00	0.00	0.00	
18.50	1253.24	0.00	0.00	0.00			0.00	10.59	0.00	0.00	0.00	1253.24	1253.24	0.00	0.00	0.00	51.96 51.96	0.00	0.00	0.00	0.00	
19.50	1253.23	0.00	0.00	0.00			0.00	10.59	0.00	0.00	0.00		1253.19	0.00	0.00	0.00	51.86	0.00	0.00	0.00	0.00	
20.00	1253.21	0.00	0.00	0.00	0.00	0.00	0.00	10.59	0.00	0.00	0.00	1253.21	1253.21	0.00	0.00	0.00	51.98	0.00	0.00	0.00	0.00	
20.50	1253.25	0.00	0.00	0.00			0.00	10.59	0.00	0.00	0.00		1253.25	0.00	0.00	0.00	51.88	0.00	0.00	0.00	0.00	
21.00	1253.68	0.00	0.01	0.00			0.00	10.59	0.00	0.00	0.00		1253.68 1253.00	0.00	0.00	0.00	51.88 51.85	0.02	0.00	0.00	0.00	
21.50	1253.00	-0.01	-0.01	0.00			0.00	10.50	0.00	0.00	0.00		1253.00	0.00	0.00	0.00	51.85	-0.01	0.00	0.00	0.00	
	1253.46	0.00	0.02	0.00			0.00	10.59	0.00	0.00	0.00		1253.46	0.00	0.00	0.00	51.87	0.02	0.00	0.00	0.00	

Fig 182 Section Force Spreadsheet for All Loadcases

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

Sign convention

Axial Force: (+) for Tension, (-) for Compression

Moment: (+) for Inner side tension, (-) for outer side tension.

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