LUSAS

LNG Tank System User Manual

Part 1 - Tank Modelling

LNG Tank System User Manual (Online): Part 1 – Tank Modelling

LUSAS Version 19.1 : Issue 1

LUSAS

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

Tel: +44 (0)20 8541 1999

Fax +44 (0)20 8549 9399

Email: info@lusas.com

http://www.lusas.com

Distributors Worldwide

Copyright ©1982-2020 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	10
3D Shell Static Structural Analysis	
3D Shell Eigenvalue Analysis	25
2D Beam-Stick FSSI Seismic Analysis	
Exporting Forces from the 2D Axisymmetric Model	
Exporting Forces from the 3D Shell Model	42
Examples – User Inputs	45
Tank Definition	45
2D Axisymmetric Static Structural Analysis	66
2D Axisymmetric Staged Construction Analysis	
2D Axisymmetric Thermal Analysis	95
3D Shell Static Analysis	101
3D Shell Eigenvalue Analysis	129
2D Beam-Stick FSSI Seismic Analysis for Horizontal Actions	134
2D Beam-Stick FSSI Seismic Analysis for Vertical Actions	157

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 require 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 titled 'LNG Tank System: Part 2 – Design Checks' 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

2D Axisymmetric Static Structural Analysis

Elements

Due to the axisymmetric nature of circular tanks, a 2D axisymmetric model is commonly used.

Groups / Materials

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

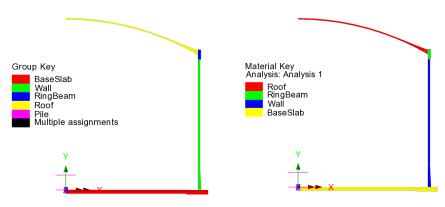


Fig 1 Group and Material Assignments for a 2D Axisymmetric Static Model

Support Condition for 2D Axisymmetric Model

Three support types are available for selection.

Tank Definition							
🖂 Include pile data		🖂 Include insulation		🖂 Includ	e seismic data		
Structure Definition Support type	Material Properties	Boundary Conditions	Loading	Prestress Load	Pile Arrangement (3D)	Seismic input 1	Seismic input 2
Pile Support Fixed Support Pile Support Regular Support	~		1			Dila	Current .

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.

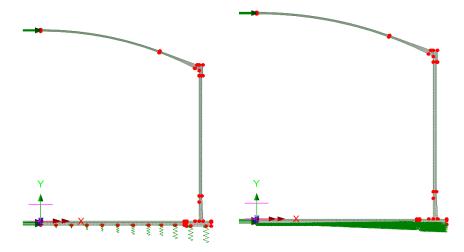


Fig 3 Support Types for a 2D Axisymmetric Static Model (Pile Support / Regular Support)

Loadings

Only the outer concrete 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.

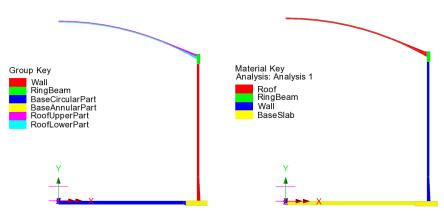


Fig 4 Group and Material Assignments in a 2D Axisymmetric Staged Construction Model

Support Condition

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

Construction Stages

Thirteen 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 if 'Roof ratio for 1st built' is not set to be 1. The materials are assumed to be linear elastic.

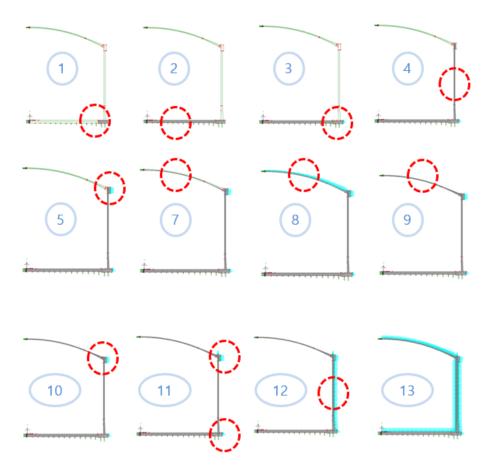


Fig 5 Activation and Deactivation in a Staged Construction Analysis Model

Stage	Description	Note
No. 1	Annular part	
No. 2	1) + Circular part	
No. 3	2) + Base PS	
No. 4	3) + Wall & Ringbeam	
No. 5	4) + Ringbeam 1 st PS	
No. 6	5) + Roof Lower Wet Concrete	

Stage	Description	Note
No. 7	6) + Roof Lower Complete	
No. 8	7) + Roof Upper Wet Concrete	
No. 9	8) + Roof Complete	
No. 10	9) + Ringbeam 2^{nd} PS	
No. 11	10) + Vertical PS	
No. 12	11) + Horizontal PS	
No. 13	12) + Other Loadings	

Table 1 Sequence of Construction Stages

If 'Roof ratio for 1st built' is set to be '1', then 11 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

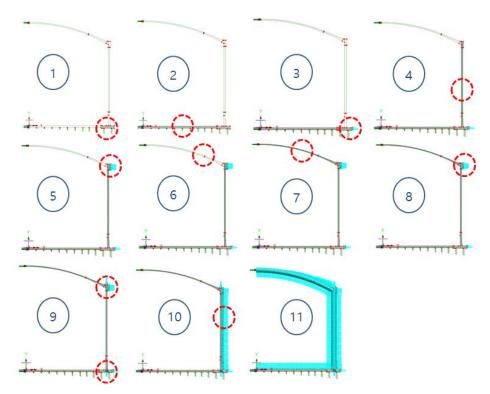


Fig 6 Birth and Death Sta	aged Construction Ana	lysis Model (Roof ratio	o for 1 st built =1)
ing o bir tir und b cutil bu	agea comparaction mina	1,515 1,10401 (11001 1441	JIOLI Dunie I)

Stage	Description	Note
No. 1	Annular part	
No. 2	1) + Circular part	
No. 3	2) + Base PS	
No. 4	3) + Wall & Ringbeam	
No. 5	4) + Ringbeam 1 st PS	
No. 6	5) + Roof Wet Concrete	
No. 7	6) + Roof Complete	
No. 8	7) + Ringbeam 2^{nd} PS	
No. 9	8) + Vertical PS	

Stage	Description	Note
No. 10	9) + Horizontal PS	
No. 11	10) + Other Loadings	

Table 2 Sequence of Construction Stages (Roof Ratio for 1st Built =1)

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.

New Model	
File name	LNG Tank
Working folder	
ORecent	C:\Users\overlaphonycoments\Usas190\Projects
() User-defined	C:\Users\overlapses
Model properties	
Analysis type	Coupled thermal/structural \checkmark Model units N,m,kg,s,C \checkmark
Analysis category	2D Axisymmetric \checkmark Timescale units Seconds \checkmark
Optional	
Startup template	None 🗸 Layout grid None 🗸
Title	
Job number	
	OK Cancel Help

Fig 7 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.

) Axisymmetric tructural Thermal		2D Axisymmetric Structural Thermal	
Element description Element type Axisymmetric solid Clement shape Quadrilateral Interpolation order Linear Clement name QAX4M	Regular mesh Altow transition pattern Allow irregular mesh Automatic Local x divisions Local y divisions O Irregular mesh Element size Element size 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 transition pattern Allow transition mesh Local x divisions Local y divisions Inregular mesh Element size I.0
Name AxisymmetricSolid	 ✓ (42) 	Name AxisymmetricSolid	 ✓ ▲ (42)

Fig 8 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.

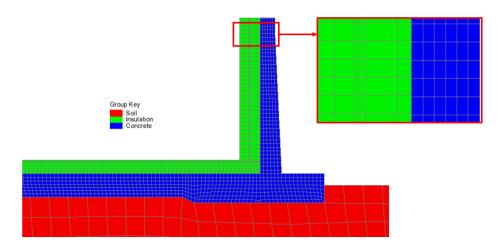


Fig 9 Insulation Elements Separated from Structure Elements

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 can be specified for soil depth if the 'Include soil' option is checked.

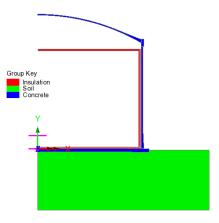


Fig 10 Mesh for 2D Axisymmetric Thermal Analysis

Groups / Materials

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

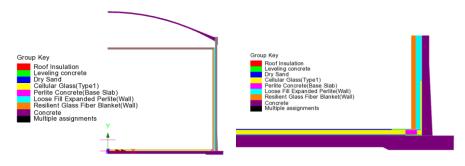


Fig 11 Group and Material Assignments in 2D Axisymmetric Thermal Analysis Model

Supports and Loading for Thermal Analysis

The 1st Loadcase

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

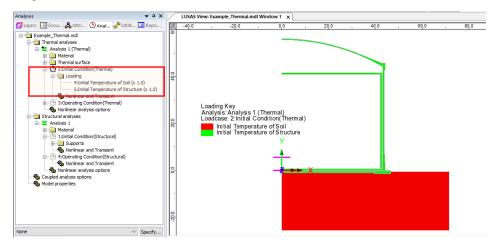


Fig 12 Thermal Analysis -1st Loadcase

The 2nd Loadcase

Liquid temperature is assigned to inner side of the insulation.

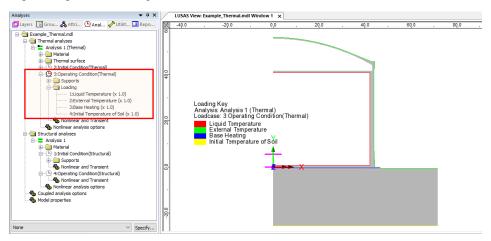


Fig 13 Thermal Analysis – 2nd Loadcase

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

Prescribed			×
 () Total		tal	
 	Free Fixed	Temperature	
 Temperature	0 •	5.0*Base Heating	
Name Base Heating	Close	Cancel Apply	Help

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 15]. The range of the loading is defined using a LUSAS field variation and can be modified by redefining the values of 'Base Heating'.

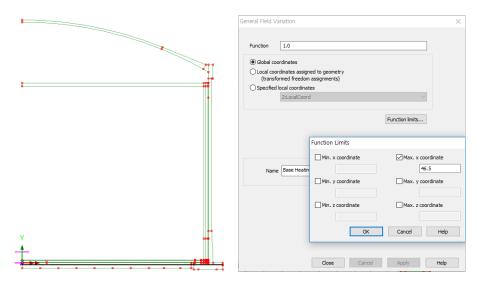


Fig 15 Base Heating Temperature Variation in a 2D Axisymmetric Thermal Analysis Model

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.

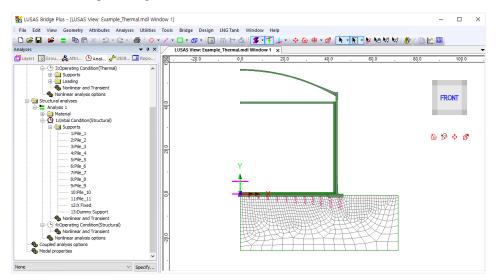


Fig 16 Pile Support for Structural Analysis following Thermal Analysis

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.

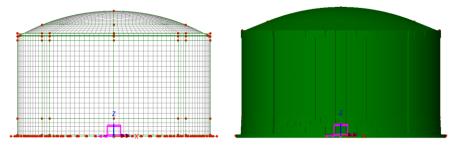


Fig 17 3D Shell Model for Static Analysis

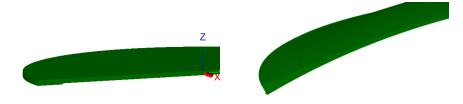


Fig 18 3D Shell Model Thickness Variation at Roof and Slab

Buttresses

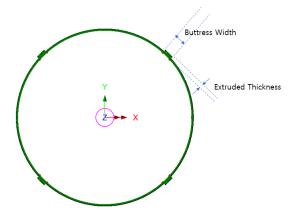
Buttresses can be included in the model with separate surfaces accepting separate geometric and material properties. The number of buttresses that can be defined is 0, 2, 3, 4 or 6.

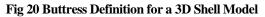
Buttress width should be stated for the straight length, not the curved length.

LNG Tank Modelling

INC Table Chills Analysis			
LNG Tank - Static Analysis			×
Tank definition data	Tnk1		\checkmark
Model filename	Example		Half only model
Saved model file path	C:₩Users₩ohsso₩Docu	ments₩Lusas19	0₩Projects¥
Element size (m) Analysis type	2,5		
○ 2D Axisymmetric solid		۵ کا) Shell
Wind load (EN1991, 1, 4, 2005)			Buttress
Basic wind velocity	37,5	(m/s)	Number of buttress 4
Roughness length	3,0E-3	(m)	Extruded thickness 1.0 (m)
Minimum height	1.0	(m)	
Orography factor	1.0		Buttress width 5,0 (m)
Terrain factor	0, 156		
Turbulence factor	1.0		Eigenvalue
Air density	1,25	(kg/m^3)	Number of eigenvalues 10
Soil height above slab bottom	0,9	(m)	☐ Include non-structural masses
			OK Cancel Help

Fig 19 User Input for the Number of Buttresses in a 3D Shell Model





Groups and Materials

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

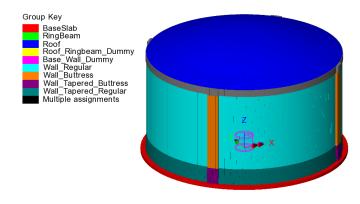


Fig 21 Groups in a 3D Shell Model

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

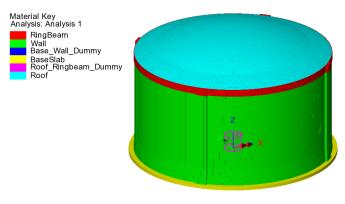


Fig 22 Material Assignments in a 3D Shell Model

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, the stiffness of each pile should be defined further from the user input dialog as shown in [Fig 23]. The spring support will be assigned to each of pile locations.

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

clude	pile data			Include insulati	on		🗸 Include	seismic	data	_			
ture D	efinition M:	aterial Prope	rties Bo	undary Conditio	ns Loadi	ng Prest	ress Load	Pile Arra	ngement (3D)	Seismic input i	1 Seismi	c input 2	
osswi	ise pile X co	ordinates (Ur	nits: m)										
P1	P2	P3	P4	P5	P6	P7	P8	P9			^		
), 0	4,2	8,4	12,6	16,8	21,0	25,2	29,4	33,6				Crosswise pile	
), ()	4,2	8,4	12,6	16,8	21.0	25,2	29,4	33,6					747
), ()	4,2	8,4	12,6	16.8	21.0	25,2	29,4						Circumferential piles
), O	4,2	8,4	12,6	16,8	21,0	25,2	29,4						A K
0,0	4,2	8,4	12,6	16,8	21,0	25,2	29,4					E Contraction	×
0.0	4,2	8.4	12.6	16,8	21.0	25,2					J.	La contra	
rossw	ise piles Y c	oordinates (I	Jnits: m)										
P1	P2	P3	P4	P5	P6	P7	P8	P9			^	Add pile in X	Define pile locations
0,0	0,0	0,0	0,0	0,0	0,0	0,0	0,0	0,0				Add pile in Y Set	zero Set defaults
-4,2	-4,2	-4,2	-4,2	-4,2	-4,2	-4.2	-4,2	-4,2				Crosswise pile stiffness	
-8,4	-8,4	-8,4	-8,4	-8,4	-8,4	-8,4	-8,4					Vertical stiffness (kN/m)	
-12,6	-12,6	-12,6	-12,6	-12,6	-12,6	-12,6	-12.6						523,018E3
-16,8	-16,8	-16,8	-16,8	-16,8	-16,8	-16,8	-16,8					Horizontal stiffness (kN/m)	42,297E3
-21.0	-21,0	-21,0	-21.0	-21,0	-21,0	-21.0					~	Type	CrossPiles
rcumf	erential piles	05.0	05.0	05.0	05.0		1				*		
ID	B (m)	Initia	l Theta	Number of piles	Vertical	Stiffness I/m)	Horizo Stiffness	ntal	Type				
	36.7	(de	gree)	piles 56	(KN 523.018E3		42.297E3	(KN/M)	Cirpiles	_		[Set defaults
	30,7	0.0		50 60	523,010E3		42,297E3		Cirpiles			i i	Set zero
	44.9	0.0		68	523,018E3		42,297E3		Cirpiles				
	1.1.9	0.0		**	000,0106.0		10,0016.0		aubuog				Add pile row
													Remove pile row
					Name	Tnk1					~ 🗘 (1)	

Fig 23 Input for Pile Locations and Stiffnesses

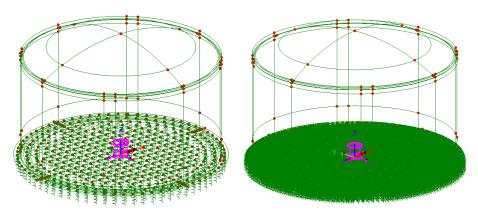


Fig 24 Support Condition for a 3D Shell Model (Pile Support / Regular Support)

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.

LNG Tank - Static Analysis				×
Tank definition data Model filename Saved model file path Element size (m) Analysis type	Tnk1 Example C:\#Users\obsco\#I 2.5	Documents₩LUS	AS190\Projects\Exampl	☐ Half only model
◯ 2D Axisymmetric solid			● 3D Shell	
Wind load (EN1991, 1, 4, 2005)			Buttress	
Basic wind velocity Roughness length Minimum height Orography factor Terrain factor Turbulence factor	37.5 3.0E-3 1.0 0.156 1.0	(m/s) (m) (m)	Number of buttress Extruded thickness Buttress width Eigenvalue	4 ~ (m) 5.0 (m)
Air density Soil height above slab bottom	0.9	(kg/m^3) (m)	Number of eigenvalue	
				OK Cancel Help

Fig 25 User Input for Wind Load for a 3D Shell Model

Other Options

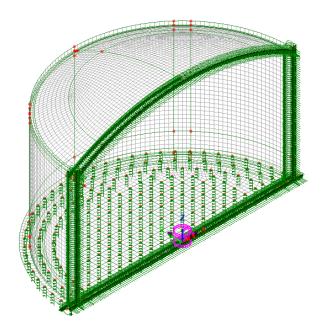
Half Only Model

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

LNG Tank Modelling

				×
Tank definition data Model filename Saved model file path Element size (m) Analysis type	Tnk1 Example C:\Users\obsco\D 2.5	ocuments₩LUS	AS190\Projects\Exampl	Half only model
🔾 2D Axisymmetric solid			③ 3D Shell	
Wind load (EN1991, 1, 4, 2005)			Buttress	
Basic wind velocity Roughness length Minimum height Orography factor Terrain factor Turbulence factor Air density Soil height above slab bottom	37.5 3.0E-3 1.0 0.156 1.0 1.25 0.9	(m/s) (m) (m) (kg/m^3) (m)	Number of buttress Extruded thickness Buttress width Eigenvalue Number of eigenvalues Include non-structural	4 v 1.0 (m) 5.0 (m) 10 masses Cancel Help

Fig 26 User Input for Wind Load on a 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, ringbeam) 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.'

LNG Tank - Static Analysis				×
Tank definition data Model filename Saved model file path Element size (m) Analysis type	Tnk1 Example C:\Users\ohsso\U 2,5	Documents\US	Ƴ SAS190₩Projects₩Exampl	Half only model
🔿 2D Axisymmetric solid			● 3D Shell	
Wind load (EN1991, 1, 4, 2005)			Buttress	
Basic wind velocity Roughness length Minimum height Orgaphy factor	37.5 3.0E-3 1.0 1.0	(m/s) (m) (m)	Number of buttress Extruded thickness Buttress width	4 ~ (m) 5.0 (m)
Terrain factor Turbulence factor Air density Soil height above slab bottom	0,156 1.0 1.25 0.9	(kg/m^3) (m)	Eigenvalue Number of eigenvalue Include non-struc	
				OK Cancel Help

Fig 28 User Input for Eigenvalue Analysis Model including Non-Structural Masses

ummary of Mass	Calculation				
DIMENSION					
	Dimension(m)				
Inner Tank Radius	42.1				
Tank Height	40.06				
LNG Height	38.92				
SUMMARY FOR	MASS				
	Volume	Unit mass	Structural mass	Total mass	
	m³	kg/m ^a	kg	kg	kg/m ³
Roof	3,950	2,500	9,875,937	11,985,937	3,0
Ringbeam(upper)	524	2,500	1,310,993	1,310,993	2,5
Ringbeam(lower)	463	2,500	1,156,758	1,156,758	2,5
Wall & Buttress	9,976	2,500	24,940,428	25,764,428	2,5
BaseSlab	8,719	2,500	21,797,085	24,925,085	2,8
LNG	216,714	480	104,022,703	104,022,703	4
Inner Tank	316	7,850	2,479,105	2,799,105	8,8
MASS DETAILS					
Component	Descriptions				Mass (kg)
Roof		/- Roof volume *	unit concrete mas	e)	9,875,9
(Marar)		eel roof structure		47	1,400.0
			the suspended ceili	ina	135.0
	Roof nozzles		and an appendice con		42.0
	Roof platform				400.0
	Roof pump &	crane			30.0
	Roof piping an				103.0
	Others	a support			100,0
	Total				11,985,93
Ring Beam		Ream (- Ding Rea	m volume * unit c	(area ateracia	2,467,7
King beam	wall barrier plat		in volume unit o	uncrete mass)	2,407,7
	wall piping and				
	Others	apport			
	Total				2,467,75
Outer Concrete Wa		= Wall volume * i	unit concrete mass		24,940,4
cate concrete tra	corner protecti		The concrete man		242.0
	wall barrier plat				494.0
	wall piping and				454,0
	Others	addour			dd,U
	Total				25,764,42
Base Slab		- Base dab unlive	e * unit concrete i	(marc)	21,797,0
uax: aldu	Others	- uase sidu volun	ann concrete i	maaaaj	3.128.0
	Total				24,925,08
Inner Steel Tank		teel tank volume	(mem leate t		24,925,08
inner ateer rank	shell stiffener	ices tank voidme	secci massj		2,4/9,1 45.0
	shell stiffener shell insulation	509/1			40,0
		3076)			
	top girder				
	Others				275,0
	To Ach.				
LNG	Total	olume * unit LNG			2,799,10 104,022,7

Fig 29 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.

LNG Tank - Static Analysis				×
Tank definition data Model filename Saved model file path Element size (m) - Analysis type	Tnk1 C:₩Users₩ohsso₩ 2.5	Documents\US	∨ SAS190₩Projects₩(3D),m(☑ Half only model
◯ 2D Axisymmetric solid			● 3D Shell	
Wind load (EN1991, 1, 4, 2005)			Buttress	
Basic wind velocity Roughness length Minimum height Orography factor Terrain factor	37.5 3.0E-3 1.0 1.0 0.156] (m/s)] (m)] (m)	Number of buttress Extruded thickness Buttress width	4 ~ (m) 5.0 (m)
Turbulence factor	1,0]	Eigenvalue Number of eigenvalue	es 10
Soil height above slab bottom	0,9	(kg/m^3) (m)	Include non-struct	
				OK Cancel Help

Fig 30 User Input for a 3D Shell Model for Eigenvalue Analysis

Analyses	▼ # × /	LUSAS View: e	example(3D) Windo	w1 x		
🔁 Layers 🔣 Groups 歳 Attrib 🕒 Analys 🦨 Utilities 💷	Reports	-60.0	-40.0	-20.0	0,0	20.0
example(30) Analysis Analysis 1 An	Include m	es required Min	Set damping mum v	Number of eigenvalues Shift to be applied		X Value 10 0.0
⊕-⊕ 15:Test load (Pneumatic) ⊕-⊕ 16:Prestress.load ⊕-⊕ 16:Prestress.load ⊕-⊕ 1:Privind load	OUnity	normalisation Mass issigned loading t) Stiffness	Type of eigensolver		Advanced
- A Model properties				ОК	Cancel	Help

Fig 31 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 concrete 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 32].

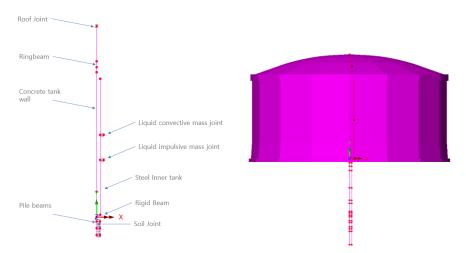


Fig 32 Beam-Stick Modelling Concept for Horizontal Actions

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

Geometric Properties are computed based user inputs and assigned as illustrated in [Fig 33].

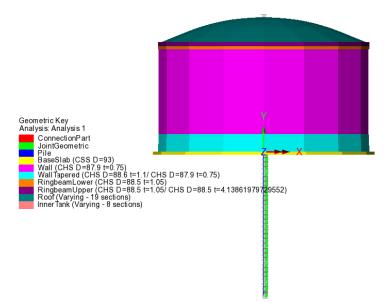


Fig 33 Geometric Properties in a Beam-Stick Horizontal Model

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

Material Properties

Material Properties are assigned as illustrated in [Fig 34].

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 computation is summarized as a spreadsheet and saved in the working folder with filename of '<model name>_<code name>_HorizontalBeamStick.xlsx'. (See [Fig 35] and [Fig 36])

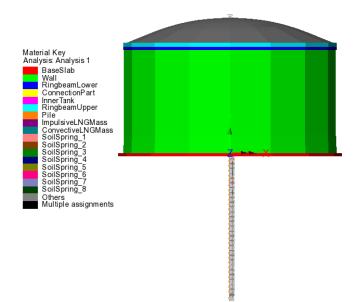


Fig 34 Material Properties in a Beam-Stick Horizontal Model

						MASS DETAILS		
Summary of Bean	n-Stick Model				998-4	Component	Descriptions	Mass (kg)
						Roof	Concrete Roof (= Roof volume * unit concrete mass)	9,875,937
DIMENSION							Roof liner + steel roof structure	1,400,000
Component	Dimension(m)						Suspended deck + insulation of the suspended ceiling	135.000
							Roof nozzles	42.000
Inner Tank Radius Tank Height	42.1						Roof platform	400.000
Tank Height LNG Height	40.06						Roof pump & crane	30.000
LNG Height	38.92							
							Roof piping and support	103,000
							Others	-
SUMMARY FOR N	IASS						Total	11,985,937
	Volume				Equivalent unit	Ring Beam	Concrete Ring Beam (= Ring Beam volume * unit concrete mass)	2,467,751
Component	m ³	ka/m ³	kg	ka	mass ko/m ³		wall barrier plate	
Roof	3.950	kg/m* 2.500	9.875.937	kg 11.985.937	Not Used		wall piping and support	
Ringbeam(upper)	5,950	2,500	1,310,993	1.310.993	2.500		Others	
Ringbeam(lower)	463	2,500	1,156,758	1,156,758	2,500		Total	2.467.751
Wall & Buttress	9.976	2,500	24.940.428	25.764.428	2,583	Outer Concrete Wall	Concrete Wall (= Wall volume * unit concrete mass)	24.940.428
BaseSlab	8,719	2,500	21,797,085	24.925.085	2.859	Outer Concrete Wall		
LNG	216,714	490	104,022,703	104,022,703	490		corner protection	242,000
Inner Tank	316	7,850	2,479,105	2,799,105	8,863		wall barrier plate	494,000
							wall piping and support	88,000
							Others	
SUMMARY FOR C	ALCULATED P	ROPERTIES					Total	25,764,428
1) Horizontal Model						Base Slab	Concrete base (= Base slab volume * unit concrete mass)	21,797,085
Component		Lever arm height	stiffness		rence		Others	3,128,000
	mc, Kg	hc, m	kc, N/m				Total	24.925.085
LNG Convective	50,527,854	23.53	19,974,995 11,325,839,357		998-4 998-4	Inner Steel Tank	Steel tank (= Steel tank volume * steel mass)	2.479.105
LNG Impulsive	53,494,849	16.13	11,325,839,357	EN 1	998-4	Inner steer rank		
2) Vertical Model							shell stiffener	45,000
.,	mass	stiffness					shell insulation(50%)	-
Component	mass m. Kg	k N/m					top girder	-
LNG Flexible	89,566,808	20 504 603 004		EN 1998-4			Others	275,000
LNG Rigid	104022702.7	2.05046E+16		EN 1998-4			Total	2,799,105
Roof	11.985.937			EN 1998-4		LNG	LNG (= LNG volume * unit LNG mass)	104.022.703
Pile(K) NoRoofTank	55.956.370	225.923.300.000		EN 1998-4			Total	104.022.703

Fig 35 Mass Summary for the Beam-Stick Model

						LNG impulsive sti			
erification for Be	am-Stick Mode				350.3	Component	Value	Unit	Remark
						tw	29.7905	mm	average wall thickness (inner tank)
						Es	2.00E+05	MPa	modulus of elasticity of inner tank
DIMENSION						ρ	7.8500	kN.s ² /m ⁴	mass density of inner tank
						C _w	0.1586		coefficients for determining the fundamentalfrequer
						C _I	0.0422		coefficients for determining the fundamentalfrequer
Inner Tank Radius	42.1					ωί	5.473	rad/s	circular frequency of the impulsive mode of vibrati
Tank Height LNG Height	40.06 38.92					ті	1.148	s	fundamental period of oscillation of the tank (plus impulsive component of the contents)
						ki	1,586,485,989	N/m	
SUMMARY FOR N	ASS					CALCULATED PRO			
	Volume	Unit mass	Structural mass	Total mass	Equivalent unit	1) Roof Mass & Stiff			
			structural mass		mass	Component	Value	Unit	Remark
					kg/m ³		11,985,937		mass of roof
Roof	3,950	2,500	9,875,937	11,985,937	Not Used	m_roof	11,900,957	kg Hz	fundamental frequency of oscillation of the roof
Ringbeam(upper)	524	2,500	1,310,993	1,310,993	2,500	T	#DIV/01	5 FIZ	fundamental period of oscillation of the roof
Ringbeam(lower)	463	2,500	1,156,758	1,156,758	2,500	k _{-root}	#DIV/0!	N/m	
Wall & Buttress	9,976	2,500	24,940,428	25,764,428	2,583	loor	#DIV/0:	N/III	
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 & Stiff			
Inner Tank	316	7,850	2,479,105	2,799,105	8,863	Component	Value	Unit	Remark
						m_ _{LNG}	104,022,703	kg	mass of LNG
						tw	29.7905	mm	average wall thickness (inner tank)
CALCULATED PRO	OPERTIES FOR H	ORIZONTAL	MODEL			Es	2.00E+05	MPa	modulus of elasticity of inner tank
1) LNG Mass & Hei	ght					ρι	480,0000	ka/m ³	mass density of LNG
		Lever arm	Lever arm			g	9.8070	m/sec ²	gravitational acceleration
		height (IBP)	height (EBP)			YL	4,7074	kN/m ³	specific weight of contained liquid
			hc(hi), m			T.	0.4504	s	fundamental period of oscillation of the LNG
LNG Convective	48,423,453	31.83	23.10						Indicamental period of oscillation of the civit
LNG Impulsive	52,963,803	33.36	14.60			k_ing	20,247,300,685	N/m	
2) LNG convective s	tiffness					3) Mass for Outer&			
Component	Value	Unit		Remark		Component	Value	Unit	Remark
g	9.8070	m/sec ²	gravitational accele	eration		m_OuterinnerTank	55,956,370	kg	mass at top of pile = total mass - LNG - roof
λ	5.8106	m ^{1/2} /s	coefficient as defin	ed in 9.3.4					
ως	0.6332	rad/s	circular frequency o	of oscillation of the	first(convective)	4) Mass & Stiffness			
Tc	9.9223	s	natural period of t	he first (convective)	mode of sloshing	Component	Value	Unit	Remark
kc	19.417.270	N/m				k_pile	225.923.300.000	N/m	

Fig 36 Computation Summary of Liquid Masses for the Beam-Stick Model

The material properties for the connection beam between concrete 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 37].

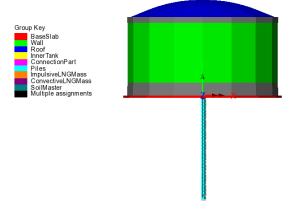


Fig 37 Groups in a Beam-Stick Horizontal Model

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 / fre	equency		
	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		
Pile	4.0		
LNG impulsive	3,0		
LNG convective	0,5		
Ground	5,0		

Fig 38 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.

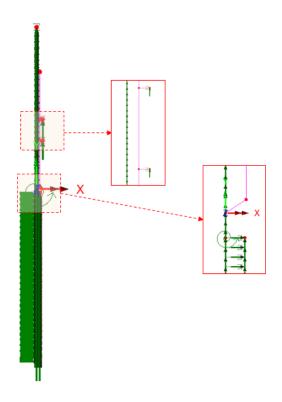


Fig 39 Support Conditions in a Beam-Stick Horizontal Model

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.

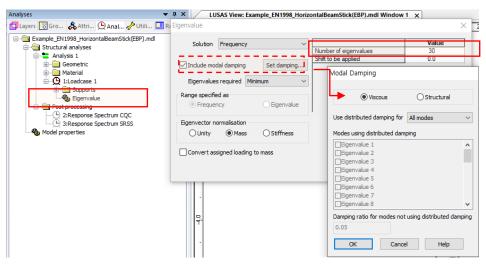


Fig 40 Eigenvalue Control for a Beam-Stick Horizontal Model

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 reponse 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.

Utilities 🗸 🕈 🗙	Response Spectrum - Design Code					×
🗗 La 🖾 Gr Å Att 🖍 Uti 🔍 An 💶 Re						
Example_EN1998_HorizontalBeamStick(4)	Design code ASCE-7-1	(2010)		~	Show graph	
⊇ a LNG Tank (1)	Incremental period	0.02	s	Maximum period	12	s
a Reference Path(2) → 1:Roof 2:InnerTank	Spectra definition Code defined	User defined	Response Spectrum Graph	L	1.0	
	Parameters		0.14			
	Site class	A	0.13		4.0	s
	Mapped spectral acceleration at short periods (Ss)	0.25	0.11	1 at	0.1	
	Spectral data		5 0.08- 5 0.07-			
	Site coefficient (Fa)	0.8			0.8	
	Short period response acceleration parameter (Sds)	0.133333	0.03 0.02	•	0.0533333	
	Period (T0)	0.08	0.01		0.4	S
			-0.02 -2.4 -0.4 16 36 56	7.6 9.6 11.6 13.6		
				Period (T)		
	Name ASCE	A-CLASS		~	(1)	
			Clo	Cancel	Apply	Help

Fig 41 Default design Response Spectrum for a Beam-Stick Horizontal Model

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 42].

Analyses	→ ⋣ X	.0	-200.0	-180.0	-160.0	140.0 -1	20.0 -100.0
🗗 Layers 📓 Groups 🖧 Attribu 🕲 Analys 🗸	🕈 Utilities 🛄 Reports	8					
testACL_ACI350_HorizontalBeamStick(IBP).mdl G Structural analyses The structural analyses Geometric Geometric Material	IMD Loadcase	Support Motio	n v	Set	×		
Q 1:Loadcase 1 Gupports	Results	Spectral	~	Set	Spectral Response		×
Construction C	Damping Type LUSAS va		Modes		Type of spectral resp Damping variation co to response spectrum	m Euro	C combination
	Name Res	oponse Spectrur		modes Apply	Response OK (z) Help	Cancel	Help

Fig 42 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 40]), modal damping is computed during the eigenvalue analysis and used at post-processing by selecting Damping Type as 'LUSAS values'.

See *Examples – User Inputs : 2D Beam-Stick FSSI Seismic Analysis for Horizontal Actions* for more information

Model for vertical actions

Elements

The concept of using a beam-stick model for vertical actions is illustrated in [Fig 43].

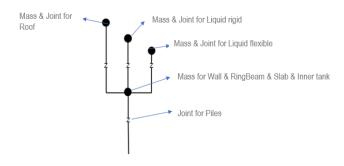


Fig 43 Beam-Stick Modelling Concept for Vertical Actions

The model is built using four joint elements as shown in [Fig 44]. Four joint elements share the node at the location of 'Mass for Wall & RingBeam & 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.

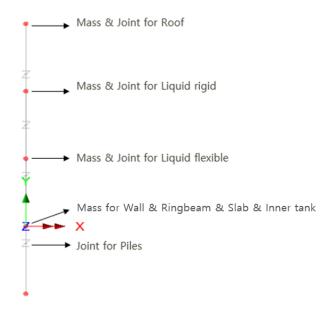


Fig 44 Beam-Stick Model for Vertical Actions

Geometric Properties

The following dataset is used.

Analysis cated	ory 2D In			
Analysis categ	20 11	Jane		
Use joint le	ength			
	nponent		Value	
Eccentricity (e:	-)		0.0	
	2)		0.0	
	2)			
	2)			
	2)			
Name	JointGeometric			
] 🔹 (1)

Fig 45 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 46].

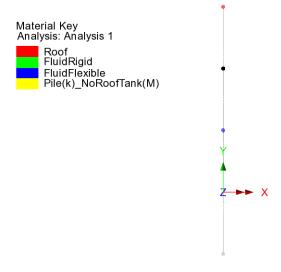


Fig 46 Material Properties in Beam-Stick Vertical Model

Details of how masses and stiffness are calculated are summarized in a spreadsheet form as shown in [Fig 35] and [Fig 36]. 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
H/R	0.924		LNG height divided by inner tank radius
ρι	480.0000	kg/m ³	mass density of LNG
Es	2.00E+11	N/m ²	modulus of elasticity of inner tank material
ν	0.2		poisson ratio of steel
s(ζ)	0.0361	m	wall thickness for $\zeta = 1/3 \ (\zeta = z/H_L)$
f(y)	1.0565		0.8<=γ<4 : 1.078+0.274 ln (γ) , γ<0.8 : 1 (A.41a, A41b
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_ _{LNG_r} (1)	52,900,941	kg	mass of LNG (rigidly moving) = $sqrt(m_{LNG_total}^2 - m_{LNG_t})$
m_ _{LNG_r} (2)	14,455,895	kg	mass of LNG (rigidly moving) = $m_{LNG,total} - m_{LNG,f}$
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), ref. A.17.
γ	0.9245		=H _i /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 oscillation of the liquid
T _{vd}	0.4153	s	fundamental period of oscillation of the liquid
k_lng_f	20,504,603,004	N/m	
k_lng_r	20,504,603,003,538,400	N/m	

Fig 47 Mass and Stiffness for Liquid for Beam-Stick Vertical Model

For the pile joint, the mass s 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 48]. This mass is assumed to move rigidly vertically.

3) Mass for Outer&In	ner Tank					
Component	Component Value		Remark			
m_OuterInnerTank	53,662,366	kg	mass at top of pile = total mass - LNG - roof			
4) Mass & Stiffness f	or Pile					
Component	Value	Unit	Remark			
k_pile	225,923,300,000	N/m				

Fig 48 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 48].

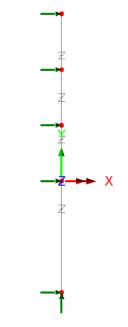


Fig 49 Supports in a Beam-Stick Vertical Model

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 : 2D Beam-Stick FSSI Seismic Analysis for Vertical Actions* 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 50]. For example, SY through the wall section can be used for computing vertical axial forces and bending moment.

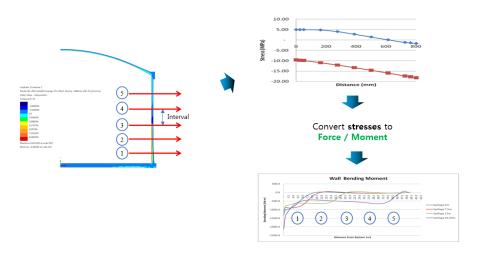
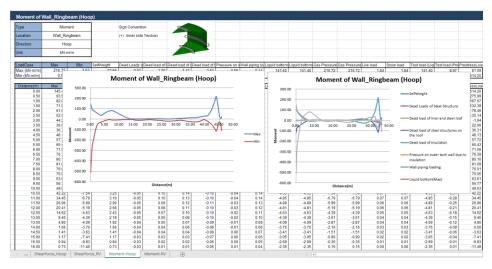
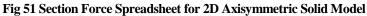


Fig 50 Converting Stress to Forces

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.





Roof - Exporting Forces

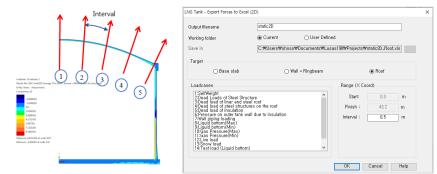


Fig 52 Exporting Forces for a 2D Axisymmetric Solid Model (Roof)

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

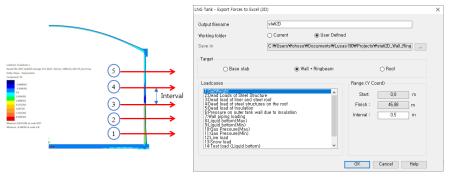


Fig 53 Exporting Forces for a 2D Axisymmetric Solid Model (Wall)

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 54 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 to Excel (2D)				×
Output filename Working folder Save in	static2D O Current C:\Users\Users\Users	⊚ User Definec so₩Documents₩Lusa		∀static2D_BaseSla	
Target	0	Wall + Ringbeam		⊖ Roof	
Loadcases I-ScittVelah 2:Dead Loads of Steel Struct 3:Dead load of steel structur 5:Pead load of steel structur 5:Pead load of steel structur 6:Pead load of steel structur 7:Wall piping loading 8:Liquid bottom(Min) 10:Gas Pressure(Max) 11:Gas Pressure(Ma) 11:Gas Pressure(Ma) 11:Gas Pressure(Ma) 11:Gas Pressure(Ma) 11:Gas Pressure(Ma) 11:Gas Pressure(Ma) 11:Gas Pressure(Ma) 11:Gas Pressure(Ma) 11:Gas Pressure(Ma) 11:Gas Pressure(Ma) 12:Show dd 14:Test load (Liquid bottom)	es on the roof	Ŷ	Range (X Ci Start: Finish : Interval :	oord) 0.0 m 46.5 m 0.5 m	
			OK	Cancel Help	

Fig 55 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 Forces to Exc	cel (3D)	>	×
Output filename	Example		
Working folder Save in	● Current ○ User Defined C:₩Users₩ohsso₩Documents₩Lusa:	s190₩Projects₩Example	
Target	⊖ Wall + Ringbeam ◯ Roof	Range Angles : [0:45 Interval : [0,5] m degree (eg. 10: 20; 30)	
Loadcases 1:SelfWeight 2:Dead Loads of Steel St 3:Dead load of steel str 5:Dead load of steel str 5:Dead load of Insulation 6:Pressure on outer tanf 7:Wall piping loading 8:Liquid bottom(Max) 9:Liquid bottom(Max) 9:Liquid bottom(Max) 10:Gas Pressure(Max) 11:Gas Pressure(Max) 11:Gas Pressure(Max) 12:Elve load 13:Snow load 14:Test load (Liquid bott		V axis Slicing Line Angle (Positive Direction) X axis (0 Degree) OK Cancel Help	

Fig 56 Exporting Forces for a 3D Shell Model

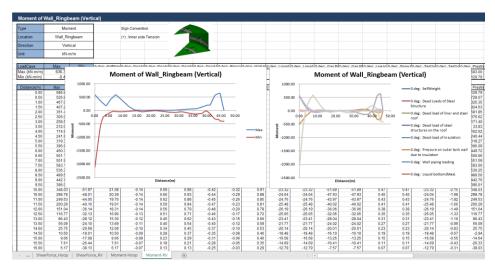


Fig 57 Section Forces Exported from a 3D Shell Model

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.

Definition						
Include pile data	Include insulation	🖂 Inclu	ıde seismic data			
cture Definition Material Proper	ties Boundary Conditions	Loading Prestress Load	d Pile Arrangement (3D)	Seismic input 1	Seismic input 2	
ncrete tank Insulations		·			·	
Base slab and Roof Wall and F	Ring beam					
Base slab (Units: m)						
Circular part length (L_inner)		39,8	L		Linner	Louter
Circular part depth (D_inner)		1,2	•			
Tapered section length (W_t)		0,6	Dinner -	D		
Annular part length (L_outer)		6.7	-	Pheating		Douter
Annular part depth (D_outer)		1,5				••• <u> </u>
Base heating (D_heating)		0, 386				Wt
5. (0.5.)						
Roof (Units: m)						
Radius of inner roof (R_roof_i)		86, 406	Troof			
Radius of outer roof (R_roof_o) Height from the top of the base		86,906 56,254				slı -
the roof (R_Height)	stab to the tophiost of	50, 254	Rroo	Li Rroof_o		sl2
Distance of tapered section 1 (s		10,079				Hringbeam_2
Distance of tapered section 2 (s	312)	0.6				+ Rsl_height
Set zero Set defaul	t					
		Name Tnk1			~ (1)	

Fig 58 Tank Definition Dialog

- □ Include Pile Data This option should be checked to define each pile location and its properties in a 3D shell model. If checked (ticked) the Pile Arrangement (3D) tab will appear
- □ **Include insulation** This option should be checked for Spillage analysis and Burnout analysis for both of which insulation should be modelled . If checked (ticked), extra tabs for insulation properties will appear.

□ Include Seismic Data This option should be checked for Seismic Analysis. If checked (ticked) the Seismic input1 and Seismic input2 tabs for seismic data will appear.

Structural Definition

Concrete Tank

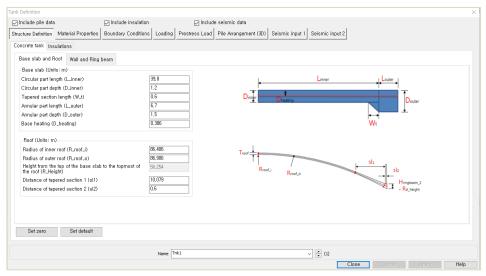


Fig 59 Tank Definition Dialog (Structure Definition/ Concrete Tank/ Base Slab and Roof)

Base Slab

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

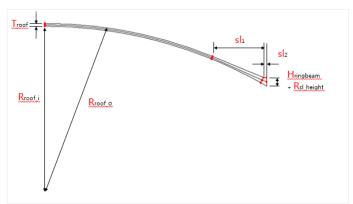


- □ Circular Part Length (L_inner): Defines the length of the circular part of the base slab where the piles are arranged orthogonally.
- □ Circular Pat Depth (D_inner): Defines the depth of the circular part of the base slab.
- □ **Tapered Section length (W_t):** Defines the length of the tapered section if it is considered in the model.

- □ Annular Part Length (L_outer): Defines the length of the annular part of the base slab where the piles are arranged in an annulus.
- □ Annular Part Depth (D_outer): Defines the depth of the annular part of the base slab.
- □ Base Heating (D_heating): Defines 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.

Roof

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



- **Radius of Inner Roof (R_roof_i):** Defines the inner radius of Roof.
- **Radius of Outer Roof (R_roof_o):** Defines the outer radius of Roof.
- □ Height from the top of the base slab to the topmost of the roof (R_Height): Defines the height between the top of the base slab and the top of the roof.
- □ Distance of tapered Section 1 (sl1): Defines the lateral distance of the tapered section 1.
- □ **Distance of Tapered Section2 (sl2):** Defines the lateral distance of the tapered section 2.

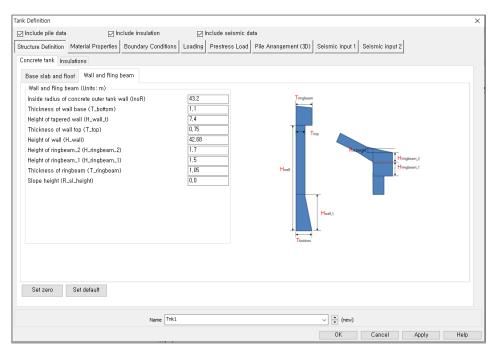
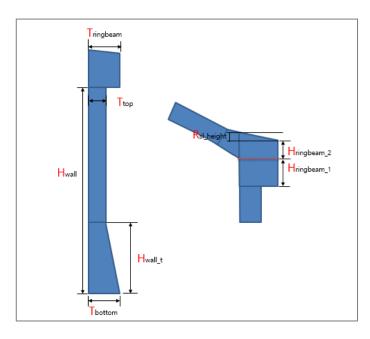


Fig 60 Tank Definition Dialog (Structure Definition/ Concrete Tank/ Wall and Ring Beam)

Wall and Ring Beam

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



- □ Inside radius of Concrete outer tank wall (InsR): Defines the inner radius of the concrete tank wall.
- □ Thickness of Wall Base (T_bottom): Defines the thickness of the bottom of the wall which is connected to the base slab.
- □ Height of Tapered wall (H_wall_t): Defines the height of tapered wall from the top surface of the base slab if the wall has a tapered section.
- □ Thickness of Wall Top (T_top): Defines the thickness of the top of wall which is connected to the Ringbeam.
- □ Height of wall (H_wall): Defines the height of wall from the top surface of the base slab.
- □ Height of Ringbeam_2 (H_ringbeam_2): Defines the height of the 2nd part of Ringbeam measured from the point where inner Roof is connected to Ringbeam to the top of the Ringbeam.
- □ Height of Ringbeam_1 (H_ringbeam_1): Defines the height of the 1st part of the Ringbeam measured from the bottom of the Ringbeam to the point where the inner Roof is connected to the Ringbeam.
- □ Thickness of Ringbeam (T_ringbeam): Defines the thickness of Ringbeam
- □ Slope height (**R_sl_height**): Defines the height difference between the left and right side of the Ringbeam.

Insulation

nclude pile data circe Denition Material Properties Boundary Conditions Loading Prestress Load Pile Arrangement (30) Seismic Input 1 Seismic input 2 ncrete tank Insulations Base incultations (Units: m) Layer 1 Layer 2 Layer 3 Layer 4 Layer 5 Layer 6 To Longth Thickness Material ID 2 41,5 0,105 2 Wall insulations (Units: m) Layer 1 Layer 2 Layer 3 Layer 4 To Longth Thickness Material ID 2 4255 0,1155 3 1 2 42,1 0,105 5 Bool insulations (Units: m) Layer 1 Layer 2 Press Material ID 2 42,1 0,105 5 2 42,1 0,105 5 Layer 1 Layer 2 To Longth Thickness Material ID 2 42,25 0,1155 3 2 42,1 0,105 5 Layer 1 Layer 2 To Longth Thickness Material ID 2 42,25 0,1155 3 2 42,1 0,105 7 2 42,1 0,105 7 2 42,1 0,105 7 Layer 1 Layer 2 To Longth Thickness Material ID 2 42,2 0,105 7 2 42,1 0,105 7	Definition									
ncrete tark [Insulations] Base insulations (Units: m) Laver 1 Layer 2 Layer 3 Layer 4 Layer 5 [1 1,7 0,105 2 4,15 0,105 2 Wall insulations (Units: m) Layer 1 Layer 2 Layer 3 Layer 4 1 0 Length Thickness Material ID 2 4,255 0,155 3 2 3,0495 0,155 5 Charles Material ID 2 4,255 1,155 3 2 3,0495 0,155 5 Charles Material ID 2 4,255 5 Charles Material ID 2 4,210 0,745 7 Charles Material ID										
Base insulations (Units: m) Laver 1 Layer 2 Layer 3 Layer 4 Layer 5 Layer 6 D Largh Thickness Material 1D 2 415 0 105 2 Wall insulations (Units: m) Layer 1 Layer 2 Layer 3 Layer 4 D Largh Thickness Material 1D 2 45255 0 1155 3 2 35,0495 1155 5 C Handle Remove Ford Insulations (Units: m) Layer 1 Layer 2 Layer 3 Layer 4 D Largh Thickness Material 1D 2 4,255 0 1155 5 2 35,0495 1155 5 C Handle Remove Ford Insulations (Units: m) Layer 1 Layer 2 D Largh Thickness Material 1D 2 4,255 0 1155 5 C Handle Remove Est defaults Est de	icture Definiti	on Materi	al Properties	Boundary Conditions	Loading Prestress Load	Pile Arrangement (3D)	Seismic input 1	Seismic input 2		
Layer 1 Layer 3 Layer 4 Layer 5 Layer 6 0 1.7 1.05 2 Set defaults Set defaults 2 41.5 0.105 2 Add Periode Set defaults Set defaults Set defaults 2 38.0405 0.155 3 2 2 38.0405 0.155 5 Set defaults 2 38.0405 0.155 5 Set defaults 2 38.0405 0.155 5 Set defaults 1 4:825 0.155 5 Set defaults 1 1 1:5 3 Set defaults 1 0 0:43:5 5 Set defaults 1 0 0:43:5 7 Set defaults 2 42:1 0:745 7 Set defaults 2 42:1 0:745 7 Set defaults 2 42:1 0:745 7 Set defaults 2 4:3 0:745 7 Set defaults 2 4:3 0:	ncrete tank	Insulations								
Set defaults 1 1 1 1 1 1 1 <td>Base insula</td> <td>tions (Units</td> <td>:: m)</td> <td></td> <td></td> <td>(</td> <td>Upto 6 segments ca</td> <td>an be defined for each Layer)</td> <td></td> <td></td>	Base insula	tions (Units	:: m)			(Upto 6 segments ca	an be defined for each Layer)		
D Langth Thickness Material ID 1.7. 0.105 1 2 4.15 0.105 2 Wall insulations (Units: m) 1 1 Layer 1 Layer 2 0.105 3 2 8,0485 0.105 5 Add Bornove Add Roor Insulations (Units: m) Add Intervention of the one of the set of the s	Layer 1 La	aver 2 Lav	er 3 Laver 4	Laver 5 Laver 6			Base and	d Wall Insulation		
Image: Set zero Add Bernove Add Wall insulations (Units: m) Set defaults Layer 1 Layer 3 2 36,0495 0.155 3 2 36,0495 0.1655 3 2 0.1655 3 3 2 0.1655 3 0.1655 3 0.1655 2 0.1655 3 0.1655 4 Bernove Renove Set defaults 5 Set defaults 6 dd Bernove Set defaults 6 dd 1 0.4823 0.745 2 0.745 5 2 0.745 5 2 0.745 5 2 0.745 5 2 0.745 5 2 0.745 7 Add Bernove Set defaults 5 5 2 2 6 Add						Set defaults		N		
2 41,5 0,105 2 Wall insulations (Units: m)						Sot zoro				
Add Wall insulations (Unlis: m) Layer 1 Layer 3 Layer 3 Layer 4 ID Logit 7 Add Bernove Set defaults Set zero Add Bernove Set defaults Set defaults Layer 1 Layer 3 Layer 2 Add D Legger 1 Layer 3 Layer 3 Layer 4 Set defaults Set defaults Set defaults Layer 7 Add Remove Set defaults Set defaults Set defaults Set defaults Set defaults Remove Set defau				2				I. I. in		
Wall Insulations (Units: m) Set defaults Layer 1 Layer 3 Layer 4 ID Layer 5 1 2 36,0455 0.155 5 Add Remove Set defaults Set defaults Set defaults Set defaults 1 Layer 2 Set defaults Set defaults 10 Layer 3 Layer 3 Set defaults 2 36,0455 0.155 5 Set defaults Set defaults Set defaults Set defaults Set defaults 10 Layer 2 Set defaults Set defaults Set defaults 10 Layer 3 Naterial ID Set defaults Set defaults 2 42.1 0.745 5 Set defaults Set defaults Name TH1 Thickness Material ID Set defaults Set defaults 1 Add Remove Set defaults Set defaults Set defaults 10 Layer 3 Naterial ID Set defaults Set defaults Set defaults 10 Layer 3 Naterial ID <td></td> <td></td> <td></td> <td></td> <td></td> <td>Add</td> <td></td> <td></td> <td></td> <td></td>						Add				
Layer 1 Layer 3 Layer 4 ID Langth Thickness Material ID 2 38,04/55 0.155 5 Add Bernove Set defaults Root Insulations (Unlis: m) Set defaults Set defaults 10 Layer 2 Set defaults Set defaults 10 Layer 3 Constructions Root Insulation 10 Layer 7 Add Remove Set defaults 10 Layer 7 Material ID Set defaults Set defaults 10 Layer 7 Add Remove Set defaults Set defaults 10 Layer 7 Material ID Set defaults Set defaults Set defaults 10 Layer 7 Material ID Set defaults Set defaults Set defaults Set defaults 10 Layer 7 Material ID Set defaults						Remove				
Layer 1 Layer 3 Layer 4 ID Length Thickness Material ID 2 36,0405 0.155 5 Add Bernove Set defaults Root Insulations (Units: m) Set defaults Set defaults 1 Layer 2 Set defaults Set defaults 10 Length Thickness Material ID 2 4423 0.745 5 2 421 0.745 7 Add Bernove Set defaults Name Thill V< to							1			
Layer Layer Layer Set defaults 1 4.8255 0.155 3 2 36.0455 0.155 5 Add Remove Set defaults Remove Set defaults Set defaults 1 1 1 1 1 1 1 1 1 1 1 1 1 2 36.0455 0.155 5 Set defaults Remove Set defaults Set defaults Set defaults 1 0.4823 0.745 5 2 42.1 0.745 7 Add Remove Add Remove Add Remove							Start Position		→LNG Tank center	
ID Length Thickness Material ID 2 36,0455 0.155 5 Add Remove	Layer 1 La	ayer 2 Lay	er 3 Layer 4			Out defaults	of Wall Insulat	en illi	Base Layer 6	
2 36,0465 0,155 5 Add Remove Roof Insulation Layer 1 Layer 2 TO Length Thickness Material ID 1 0,462 7,75 5 2 42,1 0,745 7 Name Trk1 Name Trk1 V 1 0 0 0 Contemportation Set defaults Set defaults Set defaults Set defaults V 1 0 0 0 V 1 0 0 V 1 0 0 0 V 1									Base Layer 4	
2 a, U455 b 2 a, U455 b Add Remove 0 - 050 - 050 <td></td> <td></td> <td></td> <td></td> <td></td> <td>Set zero</td> <td>1</td> <td>1</td> <td></td> <td></td>						Set zero	1	1		
Root insulations (Units: m) Set defaults Layer 1 Layer 2 10 Length 11 Length 12 Length 13 Length 14 Length	2 36,	0485	0,155	5		bbá				
Set defaults Set defaults Layer 1 Layer 2 ID Length Read Length Read Length ID Length Read Length Read Length ID Length Read Length Read Length ID Length ID Length Read Length ID ID Length Read Length ID										
Layer 1 Layer 2 10 Length Thickness Material ID 1 0.4623 0.745 5 2 42.1 0.745 7 Name Trk1 V 1 (0)						Remove				
Layer 1 Layer 2 10 Length Thickness Material ID 10 0.4829 0.745 5 2 42.1 0.745 7 Name Trk1 V 10 (0) Name Trk1 V 10 (0)	D								-	
10 Longth Thickness Material ID Set defaults 1 0.4623 0.745 5 Set zero Add 2 42.1 0.745 7 Add Remove For the first set defaults			: m)				Concrete	K	Roof Insulation	
1 0,4823 0,745 5 2 42,1 0,745 7 Add Remove 1						Set defaults				
2 42.1 0.745 7 Add Remove 1						0.1		1		
Add Fiernove						261 2610		111	Roof Layer 1	
Name Trik1 V 🕆 (1)	2 42,	1	0,745	7		Add				
Name Trik1 V 🕆 (1)						Pomouo				
						THEITHOVE				
					Name Tnk1		~	· (1)		
									Concol Apply Hal	

Fig 61 Tank Definition Dialog (Structure Definition/ Concrete Tank/ 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.

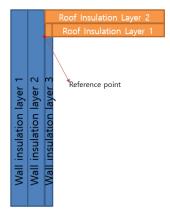
Wall Insulation

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

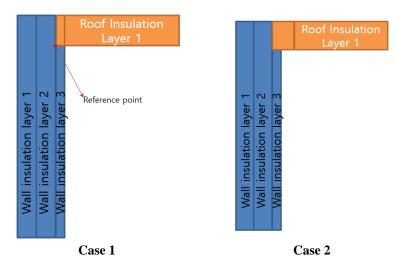
Roof insulation

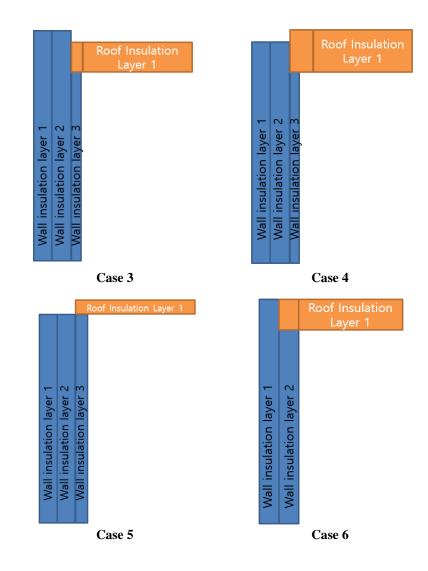
Roof insulation layers are assumed to sit on top of the innermost layer of wall insulation.

- 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 layer 3 \leq the length of the $1^{\rm st}$ segment of roof insulation layer 1

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

Include pile data		🖂 Include insula	ation	🖂 Inclu	ude seismic data	1				
ructure Definition Ma		Boundary Condit			d Pile Arranger		mic input 1 Seismic in	nut 2		
		Doundary Condi	Loading	Fiesdess Load	File Milaliger	THEIR (JD) JEIS	Seisinic in	putz		
Fank materials Insulat									 	
Material ID	Elastic Modulus (E, N/m ²)	Poissons ratio (v)	Mass Density (kg/m^3)	alpha	Thermal Conductivity (J/m.s.C)	Heat Capacity (J/m^3/C)	Description			
Concrete (Base)	35.0E9	0,2	2,5E3	10,0E-6	2,0	2,257E6	BaseSlab			
oncrete (Wall)			2,5E3	10,0E-6	2,0	2,257E6	Wall			
Concrete (Ringbeam)			2,5E3		2,0	2,257E6	RingBeam			
Concrete (Roof)	35.0E9	0.2	2,5E3	10.0E-6	2.0	2,257E6	Roof			
Set Zero	Set defaults									
Set Zero	Set defaults		Name [1	nk1			v) (t)			

Fig 62 Tank Definition Dialog (Material Properties – Tank Materials)

The *Tank Materials* tab contains the material properties for the base, wall, ringbeam and roof concrete required for the modelling the structure. Thermal Conductivity and Heat capacity should be entered only when thermal analysis is carried out.

Include pile dat	а	🛛 Include insul	ation	🖂 Inclu	de seismic dat	а				
ucture Definition	Material Properties	Boundary Cond	itions Loading	Prestress Load	Pile Arrange	ment (3D) Seis	mic input 1	Seismic input 2		
ink materials Ir	nsulation materials									
1aterial ID	Elastic Modulus (E, N/m^2)	Poissons ratio (v)	Density (rho, kg/m^3)	alpha	Thermal Conductivity (J/m,s,C)	Heat Capacity (J/m^3/C)	Descriptio	n		
Soil	1.0	0.2	0,0		2,0	1,97E6	Soil			
1	1,0	0,2	0,0	10,0E-6	2,0	2,257E6	Levelling c	oncrete		
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 Gl	ass(Type1)		
4	1.0		0.0		0.27	753,163E3		crete(Base Slab)		
5	1,0		0,0		0,0411	51,2E3		Expanded Perlite(Wall)		
6	1,0		0,0		0,032	9,6E3		lass Fibre Blanket(W,		
7	1.0	0,2	0,0	10.0E-6	0,0372	18,0627E3	Glass Fibre	e Blanket(Roof)		
Set zero	Set defaults	Add	Remove							

Fig 63 Tank Definition Dialog (Material Properties – Insulation 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 *Structure Definition* tab

Boundary Conditions

Tank Definition							×
🖂 Include pile data		🖂 Include insulation		🖂 Includ	e seismic data		
Structure Definition	Material Properties	Boundary Conditions	Loading	Prestress Load	Pile Arrangement (3D)	Seismic input 1	Seismic input 2
Support type							
Regular Support	~ Verti	cal stiffness		(MN/m/m^2)	Horizontal	stiffness	(MN/m/m^2)
							gular Spring Supports
Set zero	Set Defaults					L	
			Name T	nk1			 ✓ ★ (1)
							OK Cancel Apply Help

Fig 64 Tank Definition Dialog (Boundary Condition- Regular Support)

Support Type

This tab defines the support type for the bottom of the base slab. Options are: 'Fixed Support', 'Pile Support' or 'Regular 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 'Regular Support' is selected, one vertical and one horizontal stiffness should be defined. The stiffnesses should be a positive value in MN/m/m².

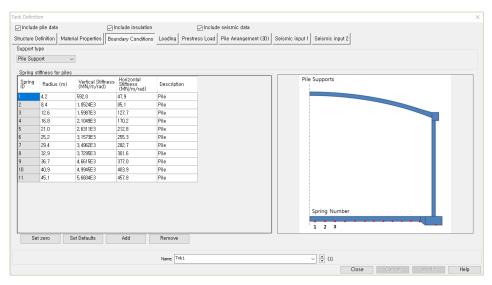


Fig 65 Tank Definition Dialog (Boundary Condition- Pile Support)

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

] Include pile data 🖓	Include insulat	tion	🖂 Include se	ismic data	
	· .		<u> </u>		
tructure Definition Material Propertie	es Boundary (Conditions Load	ing Prestres	s Load Pile Arrange	ment (3D) Seismic input 1 Seismic input 2
Structural Dead Loading Structural V	ariable Loading	Thermal Loadir	ng		
Initial liquid height 38,92	(m. For burn	out and seismic a	nalysis)		
	1				
Loading Type	Checking Load Type	Dimension(m)	Value (kN/m^2, kN/m)	Desciption	- Dead load of steel structure
Dead Loads of Steel Structure	q_4	0,0	0,0	Steel Structure	
Dead Loads of Steel Structure	P	42,1	72,9	Steel Structure	g.
Dead Loads of Steel Structure	q_r	42,1	0,098	Steel Structure	q_r
Dead load of liner and steel roof	q_1	43,2	0,404	Liner_base1	
Dead load of liner and steel roof	q_2	0,0	0,0	Liner_base2	
Dead load of liner and steel roof	q_3	0,0	0,0	Liner_base3	
Dead load of liner and steel roof	q_4	0,0	0,0	Liner_base4	
Dead load of liner and steel roof	q_r_liner	43,2	1.095	Liner_Roof	
Dead load of liner and steel roof	q_L	0,0	0,404	Liner_Wall	
Dead load of steel structures on t,	q_r_st	10,0	5, 75	SteelStructureF	
Dead load of Insulation	q_1	41,5	4,902	Insul_q1	
Dead load of Insulation	q_2	0,6	10,314	Insul_q2	p
Dead load of Insulation	q_3	0,6	30,059	Insul_q3	
Dead load of Insulation	q_4	0,5	24,645	Insul_q4	q_3
Dead load of Insulation	q_r	42,1	0,177	Insul_qr	q_1 q_2 q_4
Pressure on outer tank wall due t,	p_i	0,0	0,245	Insul_Pressure	
Wall piping loading	q_wp	0,0	1,2	Wall piping loa	
				~	Set zero Set defaults
<				>	0012010 0010010
	Na	ame Tnk1			 (new)
					OK Cancel Apply Help

Fig 66 Tank Definition Dialog (Loading – Structural Dead Loading)

Structural Dead Loading

This defines the structural dead loading to consider in analysis.

- □ **Initial Liquid Height** This defines the initial height of the contained liquid, which is used to compute the total LNG mass. It is measured from the top of the base slab.
- □ 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.
- □ Loaded Length [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.

nk Definition						
🛛 Include pile data	🖂 Include i	nsulation	🖂 Inclu	de seismic data		
tructure Definition Material Pro	operties Boundary C	onditions Load	ing Prestress Load	Pile Arrangement	(3D) Seismic input	1 Seismic input 2
Structural Dead Loading Struct	tural Variable Loading	Thermal Loadin				
addition of the Education		Thomas Loudin	-5			
						- Liquid Bottom
Loading Type	Checking Load Type	Dimension(m)	Max Value (kN/m^2, kN/m)	Min Value (kN/m^2, kN/m)	Desciptio	
Liquid bottom	q_1_liquid	42,1	183,662	183,662	Liquid_q1	
Liquid bottom	q_2_liquid	0.0	0.0	0.0	Liquid_q2	
Liquid bottom	q_3_liquid	0,0	0,0	0,0	Liquid_q3	
Liquid bottom	q_4_liquid	0,0	0,0	0,0	Liquid_q4	
Liquid wall (Max)	q_liquid_wall	0,0	0,0	0,0	Liquid_wa	
Liquid wall (Min)	q_liquid_wall	0,0	0.0	0.0	Liquid_wa	
Gas Pressure	P_g	0.0	29.0	29.0	GasPress	
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_hydrostatic	42,1	183,662	0,0	Hydrostati	
Test load (Liquid wall)	P_hydrostati	0.0	0.0	0.0	Hydrostati	
Test load (Pneumatic)	P_pneumatic	0,0	1,2	0.0	Pneumatic	
						بەلىندايدىل.
						q3
						q1 q2 q4
					0.	at zero Set defaults
<					> Se	a zero Ser detaurs
			Tnk1			
		Name				 ✓ ▲ (1)
						OK Cancel Apply Help

Fig 67 Tank Definition Dialog (Loading – 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.
- □ **Loaded Length:** 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.

⊲ Include pile data	🖂 Include in	sulation	🖂 Include s	seismic data			
tructure Definition Material P					ement (3D) Seismic input 1	Seismic input 2	
	·					Selamic input 2	
Structural Dead Loading Stru	stural Variable Lo	ading Inermal					
Loading Type	Spillage Height (m)	Temperature (C)	Convective Coefficient (J/m^2,s,C)	Type of boundary	Description		
Liquid Temperature	0,0	-170,0	0,0	Convection \sim	Liquid Temperature		
External Temperature	0,0	25,6	25,0	Convection ~	External Temperature		
Base Heating	0,0	5,0	0,0	Prescribed ~	Base Heating		
nitial Temperature (Soil)	0,0	15,1	0,0	Prescribed ~	Initial Temperature of Soil		
Initial Temperature (Structure)	0,0	15,1	0,0	Prescribed ~	Initial Temperature of Structure		
Set zero Set defaults	+ If ter Othe	mperature for bas	se heating is defin ng will not be con:	ed as other than zero. sidered in the analysis	it will be considered in the ar	valysis.	
Set zero Set defaults	• If ter Othe	mperature for bas rwise base heati Name Trrk1	se heating is defining in the cons	ed as other than zero sidered in the analysis	it will be considered in the ar	valysis,	

Fig 68 Tank Definition Dialog (Loading – Thermal Loading)

Thermal Loading

- □ Loading Type: Defines the type of temperature loading including LNG Temperature, External Temperature, Base Heating, Initial 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.
- □ **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.

Prestress Load

nk Definition										
⊲ Include pile dat	ta	🖂 includ	e insulation		🖂 Include	seismic data				
tructure Definition	Material Propertie	s Boundary	Conditions	Loading	Prestress Load	Pile Arrangeme	ent (3D) Seismic	input 1 Seismic input	2	
Vertical prestress										
	ce (Short Term, kN)	75	4,056E3				Total tendon for	ce (Long Term, kN)	754, 056E 3	
2D Conversion ((kN/m^2) :	3.1	67219E3		2.49375E3		2D Conversion ((kN/m^2) :	3.67219E3	2.49375E3
3D Shell Conver	sion (kN/m) ;	2.	75414E3		L .		3D Shell Conver	sion (kN/m) ;	2,75414E3	
Horizontal prestr	000									
Section ID	Start Position (m)	End Positio	n (m) Load (m)	ied Length	Prestress Load Short Term (kN/m^2)	Prestress Load Long Term (kN/m^2)	Description	Prestress load		Vertical Prestress
BaseSlab	0,0	0,0	1,2		370,275	370,275	BaseSlab			
1	0,0	3,8	3,8		319,291	319,291	1IOT			
2	3,8	7,4	3,6		205, 796	205, 796	210 T			2
3	7.4	11.0	3.6		206,208	206,208	SIOT			and a second sec
4	11.0	14,6	3.6		180,432	180,432	410 T			
5	14,6	18,2	3,6		154,656	154,656	5IOT			8
6	18,2	21,8	3,6		154,656	154,656	6IOT			道
7	21,8	25,4	3,6		128,88	128,88	7IOT		Horizontal Pre	stress
8	25.4	29.0	3,6		103,104	103, 104	8IOT			
9	29,0	32,6	3,6		103,104	103, 104	9IOT			
10	32,6	36,2	3,6		103,104	103, 104	10IOT			
11	36,2	39,8	3,6		103,104	103, 104	11IOT			
12	39,8	42,68	2,88		96,66	96,66	12IOT			Base Prestress
RingBeam	42,68	45,88	3.2		112,033	112,033	RingBeam			Base Prestress
<							>		1	/ertical Prestress
Set zero	Set defaults	Add		Remove						
				Name Tr	k1			~ 🗘 (1)		
									OK Cancel	Apply Help

Fig 69 Tank Definition Dialog (Prestress Load)

Vertical Prestress

□ Total Tendon Force (Long term/ Short term): Defines the total tendon force for vertical prestress. The vertical prestress load is calculated by dividing the Total tendon force by the loaded area. It is applied to both the top surface of the ringbeam and the bottom surface of the base slab over an area equivalent to the width of the bottom surface of the wall.

Horizontal Prestress

- □ Section ID: A unique positive integer ID should be defined, with the exception of the first and the last row.
- □ Start Position: Defines the start location of the prestress load. It should be defined from the top of the base slab, which is at a location of 0m.
- □ End Position: Defines the finishing location of the prestress load. It should be defined from the top of the base slab which is at a location of 0m.
- □ Loaded Length: Defines the loaded length in metres. Negative loaded lengths are not permitted in the modelling and may give an error message. A zero loaded length means that the loading is not considered in the analysis. A loaded length for the base prestress load will be automatically defined as the depth of

inner base slab and this value will be able to be changed after the analysis model is created by editing the attribute.

□ Prestress load (Short term/ Long term): Defines the magnitude of the structural loading in units of kN per square metre. A positive value should be entered regardless of the loading direction. The hoop forces in the tendon are applied as radial pressures by considering the radius of the tendon.

Pile Arrangement (3D)

Include p	pile data] Include insulati	n		🖂 Include	seismic	data				
cture De	efinition M	laterial Prop	erties B	oundary Conditio	ns Loadin	g Pres	tress Load	Pile Arra	ingement (3D)	Seismic input 1	Seism	nic input 2	
Crosswis	se pile X co	ordinates (l	Inits: m)									_	_
P1	P2	P3	P4	P5	P6	P7	P8	P9			^		
0.0	4,2	8,4	12,6	16,8	21.0	25,2	29,4	33,6				Crosswise piles	
0,0	4,2	8,4	12,6	16,8	21.0	25,2	29,4	33,6					
0,0	4,2	8,4	12,6	16,8	21,0	25,2	29,4					Circumferential piles	
0,0	4.2	8.4	12,6	16,8		25.2	29,4					and the second sec	
0.0	4,2	8,4	12,6	16,8	21,0	25,2	29,4						
0,0	4,2	8,4	12,6	16,8	21,0	25,2					~	First Contraction	
Crosswi	se piles Y i	coordinates	(Units: m)		01.0								-
P1	P2	P3	P4	P5	P6	P7	P8	P9			^	Add pile in X Define pile locations	
0,0	0,0	0,0	0,0	0,0	0,0	0,0	0,0	0,0				Add pile in Y Set zero Set defaults	
-4.2	-4,2	-4.2	-4.2	-4.2	-4.2	-4.2	-4.2	-4.2				Crosswise pile stiffness	
-8,4	-8,4	-8,4	-8,4	-8.4	-8,4	-8,4	-8,4					Vertical stiffness (kN/m) 523.018E3	-
-12,6	-12,6	-12,6	-12,6		-12,6	-12,6	-12,6						-
-16,8	-16,8	-16,8	-16,8		-16,8	-16,8	-16,8					Horizontal stiffness (kN/m) 42,297E3	_
-21.0	-21.0	-21.0	-21.0	-21.0	-21.0	-21.0					~	Type CrossPiles	
Circumfe	erential pile:	s											
ID	R (m) Init	al Theta legree)	Number of piles	Vertical S (kN/	(tiffness	Horiz Stiffness	ontal (kN/m)	Type				
	36,7	0,0		56	523,018E3		42,297E3		Cirpiles			Set defaults	
	40,8	0,0		60	523,018E3		42,297E3		Cirpiles			Set zero	
	44,9	0,0		68	523,018E3		42,297E3		Cirpiles			Add pile row	
												Remove pile row	
						Tok 1						(2)	
					Name	Tnk1					~ •	(1)	

Fig 70 Tank Definition Dialog (Pile Arrangement)

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 ${\sf X}$	13	Row in Y	Number of piles in X	
Add row in Y				
Remove row				
Start offset of piles in X (m)	0,0			
Start offset of piles in Y (m)	0.0			
General spacing of piles in X (m)	3,0			
General spacing of piles in Y (m)	3,0			
		OK	Cancel Apply	y Help

Fig 71 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.

Seismic Input 1

Tank Definition											\times
🖂 Include pile data		🖂 Include insulat	n	🖂 Include se	ismic data						
Structure Definition	Material Properties	Boundary Conditi	ns Loading Pr	estress Load Pil	le Arrangement	(3D) Seismic in	put 1 Seismic in	put 2			
Inner Tank Propertie	8 Non-Structural N	Masses									
Liquid mass											
Liquid unit mass	480,0	(kg/n	^3) Inr	ier tank inside rad	ius	42.1	(m)		Thickness 6	Height 6	
									Thickness 5	Height 5	
Inner tank geomet		2 3	4	5	6	7	8	_	Thickness 4	Height 4	
Thickness(m) Height(m)		0.0361 0.0 27,0 3,8	2 0.01	0.0	0,0	0.0	0.0		Thickness 3	Height 3	
(Tergin(TT)	3,00 2	27,0 3,0	0,12	0.0	0,0	0,0	0,0		Thickness 2	Height 2	
									Thickness 1	Height 1	
Inner tank materia	a proportion							>			
	Elastic Mod (E, N/m^2)	ulus Poissons r (v)	tio Mass Density (kg/m^3)	Coefficient of thermal expansion(/C)	Thermal Conductivity (1/m s C)	Heat Capacity (J/m^3,C)	Description				
Inner Tank	200,0E9	0,2	7,85E3		2,0	1,968E6	Inner Tank				
		I				1					
Set zero	Set defaults										
			Name Tnk1				• (a)				
			Name				 ✓ ▲ (1) 	OK	Cancel	Apply	Help
					<u> </u>			UK	Calicer	obbiA	nolp

Fig 72 Tank Definition Dialog (Seismic Input1 – Inner Tank Properties)

- □ Liquid Unit Mass This defines the LNG Unit Mass for convective and impulsive mass 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.

nclude pile data	Include insulation		seismic data				
ture Definition Material Pro	perties Boundary Conditions Load	ing Prestress Load	Pile Arrangement (3D) Seis	smic input 1 Seis	smic input 2		
er Tank Properties Non-Stru	ictural Masses						
Roof Ring Beam Wall	Base Slab Inner Steel Tank						
Descriptions		Mass(kg)					
Roof Liner + steel Roof Struct	ture	1,4E6					
Suspended deck & insulation	of the suspended ceiling	135,0E3					
Roof nozzles		42,0E3					
Roof platform		400,0E3					
Roof pump & crane		30,0E3					
Roof piping and support		103.0E3					
tool bibling and subbolic		100,020					
Others		0,0					
Others		0,0					
Others	faults	0,0					
Others Total	faults	0,0					
Others Total	faults	0,0				 	
Others Total	faults	0,0				 	
Others Total	faults	0,0				 	
Others Total	faults	0,0				 	
Others Total	faults	0,0					
Others Total		0,0		~			

Fig 73 Tank Definition Dialog (Seismic Input1 – 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.

Seismic Input 2

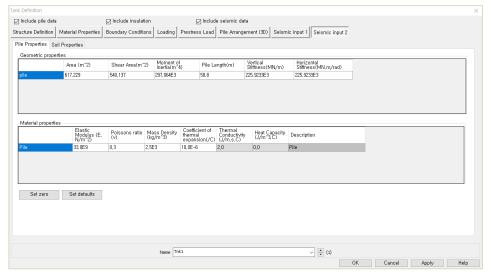


Fig 74 Tank Definition Dialog (Seismic Input2 – Pile Properties)

Pile Properties

- 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.
- □ **Material Properties:** Define material properties for piles which will be used to build a model for seismic analysis.

	e pile data		🖂 Include insulat		1					
ructure D	efinition Mate	ial Properties	Boundary Conditi	ons Loading Prestress Load Pile An	angement (3D) Seismic inpu	ut 1 Seis	mic input 2			
ile Prope	erties Soil Prop	erties								
Layer No,	Bottom EL(m)	Thickness of Layer(m)	kh(MN/m/m)	Description(Soil Profile)		^		•		
	0,0	0,0	0,0	Pile Head				PI	Layer Thickness	
1	-2,0	2,0	14,92E3	Backfill			Layer 1		Layer Thickness	
2	-4.0	2,0	25, 769E 3	Backfill			Layer 2			
3	-6.0	2.0	22,39E3	Backfill						
4	-8,0	2,0	21,549E3	Backfill			Layer 3			
5	-10,0	2,0	23,622E3	Backfill						
6	-12,0	2,0	32, 373E3	Silty Sand1			Layer 4			
7	-14.0	2.0	27,003E3	Silty Sand1						
3	-16,0	2,0	24,337E3	Silty Sand1			Layer 5			
9	-18,0	2,0	32,45E3	Silty Sand2						
10	-20,0	2,0	27,263E3	Silty Sand2			Layer 6			
11	-22,0	2.0	32,509E3	Silty Sand2						
12	-24.0	2,0	27,78E3	Silty Sand2						
13	-26,0	2,0	30, 789E3	Silty Sand2						
14	-28,0	2,0	35, 822E 3	Silty Sand3						
15	-30,0	2.0	36, 329E 3	SSilty Sand3			Add			
16	-32.0	2,0	36.336E3	Silty Sand3			Remove			
17	-34,0	2,0	35,847E3	Silty Sand3			Set defaults			
18	-36,0	2,0	36, 348E3	Silty Sand3			ser detaults			
19	-38,0	2,0	36, 355E 3	Silty Sand3		~	Set zero			

Fig 75 Tank Definition Dialog (Seismic Input2 – Soil Properties)

Soil Properties

- **Bottom EL:** 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.
- **Wh:** Defines the horizontal soil spring stiffness per unit length.

2D Axisymmetric Static Structural Analysis

User Inputs

The required user inputs for this model are as shown in [Fig 76].

ank Definition							
🖂 Include pile data	🖂 Include insulation		🖂 Include seism	ic data			
Structure Definition Material Properties	Boundary Conditions	Loading Prestre	ss Load Pile Ar	rangement (3D)	Seismic input 1	Seismic inp	ut 2
Concrete tank Insulations							
Base slab and Roof Wall and Ring I	beam						
Base slab (Units: m)							
Circular part length (L_inner)	3	39,8		L.		Linner	Louter
Circular part depth (D_inner)	1	1.2		-			
Tapered section length (W_t)	Ī	3,6		Dinner	Dheating		Douter
Annular part length (L_outer)	E	6,7		-	- Theorem		Couter
Annular part depth (D_outer)	1	1,5					←→ <u> </u>
Base heating (D_heating)	C	386					Wt
Roof (Units: m)							
Radius of inner roof (R_roof_i)	8	36, 406		Troof			
Radius of outer roof (R_roof_o)	8	36,906			1		, sh
Height from the top of the base slab the roof (R_Height)	to the topmost of	56,254		Rroo	U Rroof o		sl2
Distance of tapered section 1 (sl1)	1	0,079					Hringbeam 2
Distance of tapered section 2 (sl2)	0	1.6					RsLbeight
Set zero Set default	1						
Set Zero Set delaut							
		Name Tnk1				✓ ⁺ (1)	
							OK Cancel Apply Help

Fig 76 User Inputs for 2D Axisymmetric Static Analysis

The user dialog is available from LNG Tank>Static Analysis Wizard as shown in [Fig 77].

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

ank definition data	Tnk1 v
lodel filename	
aved model file path	C:\Users\operatorname{C:WUsers\operatorname{Users}C:WUsers
lement size (m) Analysis type	0.2
 2D Axisymmetric solid 	◯ 3D Shell

Fig 77 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.

Element Size

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

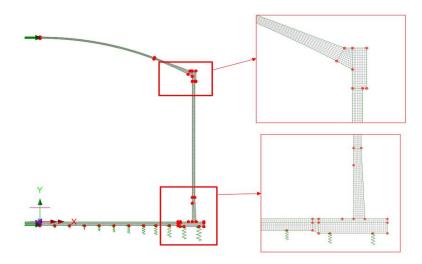


Fig 78 Mesh division for a 2D Axisymmetric Model

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

Geometric Properties

No geometric properties are required for 2D axisymmetric model.

Material Properties

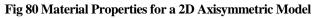
User defined material properties are assigned to the relevant surfaces.

Tank Definition								
☑ Include pile data		🖂 Include insul	ation	🖂 İnclu	ide seismic data			
Structure Definition Material Properties		Boundary Cond	itions Loadin	g Prestress Load	d Pile Arrangei	ment (3D) Seis	smic input 1 Seismic input 2	
Tank materials Insulation materials								
Material ID Elastic Modulus (E, N/m^2)		Poissons ratio (v) Mass Density (kg/m^3)		^y alpha	Thermal Conductivity (J/m.s.C)	Heat Capacity (J/m^3/C)	Description	
Concrete (Base)	35,0E9	0,2 2,5E3		10,0E-6	2,0	2,257E6	BaseSlab	
Concrete (Wall) 35,0E9		0,2	2,5E3	10,0E-6	2,0	2,257E6	Wall	
Concrete (Ringbeam) 35,0E9		0,2	2,5E3	10,0E-6	2,0	2,257E6	RingBeam	
Concrete (Roof)	35,0E9	0,2	2,5E3	10,0E-6	2,0	2,257E6	Roof	

Fig 79 User Inputs for Tank Materials

This can be found from LUSAS Modeller as shown in [Fig 80].

Attributes 🗸 🗸	$^{\pm}$ ×	\sim	20.0	10.0	0.0	10.0 20.0	30.0	40.0	50.0 60.0
🗊 Lay 📓 Gr 🖧 Att 😟 An 🥓 Util 💷 R	te		Isotropic						×
□ ■ karniple(2D).mdl □ ■ Attributes (81) □ ■ Mesh (31) □ ■ Line (30) □ ■ Surface (1) □ ■ 31:AdsymmetricSolid	^	50.0 60.0	Plastic	Creep	Damage	Shrinkage	Viscous	Two phase	Ko Initialisation
🖨 🔄 Material (4)			Dynami	c properties				Value	
😪 1:BaseSlab			Therma	al expansion		Young's modulus		35.0E9	
- & Z:Wall - & 3:RingBeam		40.0				Poisson's ratio Mass density		0.2 2.5E3	
- & 4:Roof						Coefficient of therm	nal expansion	10.0E-6	
⊡ Supports (12)									
2:Pile_2		30.0							
		8	1						
- & 6:Pile_6									
- & 7:Pile_7 - & 8:Pile 8		20.0							
- & 10:Pile_10 - & 11:Pile 11		1							
a 12:X Fixed		00							
🖨 🔄 Loading (33)		Ē							
- 🖧 1:SelfWeight		1				1			
- 🖧 4:Steel Structure_qr		8							
- & 5:Liner_base1 - & 6:Liner Roof				Name Base	Slab			~ 📮 (1))
- & 6:Liner_Koor									
8:SteelStructureRoof	~	10.0							
Text Output		1.5				CI	ose Cane	cel Apply	Help
Text Output									



Support Conditions

	pile data		nclude insulation			e seismic data		1	
		ial Properties Bou	undary Conditions	Loading Prestre	ess Load	Pile Arrangement (3D)	Seismic input 1	Seismic input 2	
upport t	/pe			-					
ile Sup	port ~								
	tiffness for piles								
Spring s ID	Radius (m)	Vertical Stiffness (MN/m/rad)	Horizontal Stiffness (MN/m/rad)	Description			Pile	e Supports	
	4,2	592,0	47,9	Pile					
	0,4	1,0524E3	05,1	Plie	-				
	12,6	1,5987E3	127,7	Pile					
	16.8	2,1049E3	170.2	Pile					
	21.0	2,6311E3	212,8	Pile					
	25,2	3, 1573E3	255, 3	Pile					
	29,4	3,4962E3	282,7	Pile					
	32,9	3,7295E3	301.6	Pile					
	36,7	4,6615E3	377.0	Pile					
D	40,9	4,9945E3	403,9	Pile					
1	45, 1	5,6604E3	457,8	Pile					
								Spring Number	
								Spring Number	
Se	t zero S	et Defaults	Add	Remove					_
				Name Tnk1				 ✓ (1) 	

Fig 81 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.

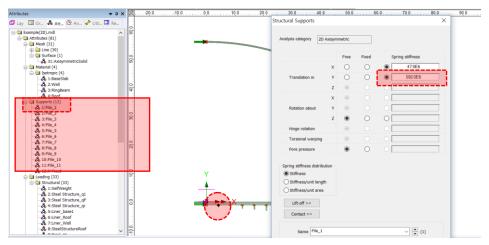


Fig 82 Pile Support for a 2D Axisymmetric Model

TEST CASE

If support type 'Regular Support' is chosen as shown in [Fig 83], the support definition will be as shown in [Fig 84].

Tank Definition								
🖂 Include pile data		☑ Include insulation		🖂 Include	e seismic data			
Structure Definition	Material Properties	Boundary Conditions	Loading	Prestress Load	Pile Arrangement (3D)	Seismic input 1	Seismic input 2	
Support type					ī.			
Regular Support	✓ Vertic	cal stiffness	1000	(MN/m/m^2)	Horizontal	stiffness	2000	(MN/m/m^2)

Fig 83 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'.)

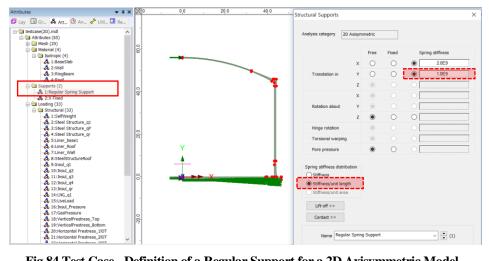


Fig 84 Test Case - Definition of a Regular Support for a 2D Axisymmetric Model

Loadings

A total of 16 loadcases is defined in the model.

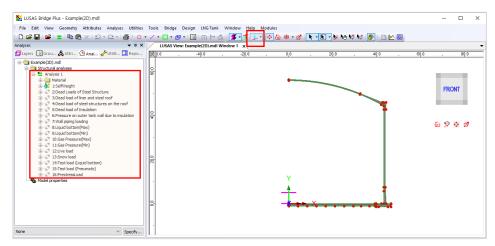


Fig 85 Loadcases available in a 2D Axisymmetric Static Analysis Model

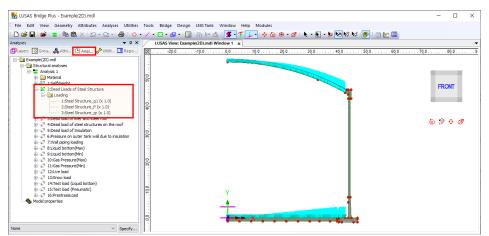
Self Weight

File Edit View Geometry Attributes Analyses Utilities Tools Bidge Design ING Tank Window Help Modules			
Image:	LUSAS Bridge Plus - Example(2D).mdl		– 🗆 ×
Instruct ● 3 × X Instruct ● 0 × X	File Edit View Geometry Attributes Analyses Utilities	Tools Bridge Design LNG Tank Window Help Modules	
Instruct ● 3 × X Instruct ● 0 × X	: 🗅 🚅 🔚 🖆 🚍 🖻 🔞 🗙 1 오 - 오 - 1 🚭 i 🧿	- / - 🗖 - 🞯 - 🔞 m + 🛆 💋 - 🕇 🔔 - 🕹 🏠 🕸 - 💋 λ - 🕅 - 😥 🏍 🕸 🖗 🧰 🐻 💷	
Image: Second Column Image: Second Column <td></td> <td></td> <td>-</td>			-
Image: Second Column Image: Second Column <td>Layers 🐻 Grou & Attri. 🕒 Anal 🖉 Utilit 🛄 Repo</td> <td>-20.0 -10.0 0.0 10.0 20.0 30.0 40.0 50.0 60.0 7</td> <td>0.0 80.0 9</td>	Layers 🐻 Grou & Attri. 🕒 Anal 🖉 Utilit 🛄 Repo	-20.0 -10.0 0.0 10.0 20.0 30.0 40.0 50.0 60.0 7	0.0 80.0 9
Structure analyses Image: Structure analyses Image			
Image: Second and the second secon	😑 🔄 Structural analyses		
Image: Second			
Supports Supports Image: Supports Gravity Image: Supports Gravity </td <td></td> <td>8</td> <td></td>		8	
Image: Construction Image: Construction Ima	Gupports		FRONT
Image: Provided Loss of Process Structure Image: Process Structure Image: Pr			
Image: Provide Stand Sta			
Image: Start load of imadeon Image: Start load of image: Start load of imadeon Image: Start load of image: Start load			
□ □ </td <td></td> <td></td> <td>6 2 0 0</td>			6 2 0 0
			an 5- • m
⊕ "1" Statute bottom(Ne) ⊕ "1" Statute			
⊕ "1" 10:63e Pessare(9ws) ⊕ "1" 12:0xe load ⊕ "1" 15:0reatreastod ⊕ "1" 15:0reatreastod			
⊕ "1 11GaPressent@ini) 0 ⊕ "1 1350m bad 0 ⊕ "1 1350m bad 0 ⊕ "1 157m bad patotion) 0 ⊕ Model properties 0			
In the last of the last			
⊕ 1 14 Ters tood (pusk bottom) ⊕ 1 15 Ters tood (pusk bottom) ⊕ 1 15 Ters tood (pusk bottom) ⊕ 1 15 Ters tood (pusk bottom) ⊕ Model properties		- X-	
the ¹			
	10 Model properties		
None Specify			
	None V Specify		

Fig 86 Self Weight in a 2D Axisymmetric Static Analysis Model

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.





Dead load of liner and steel roof

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

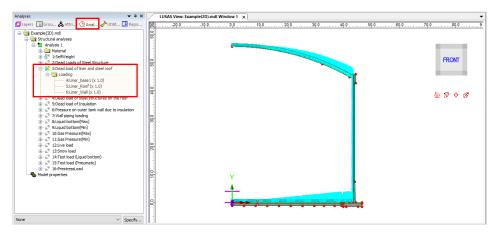


Fig 88 Dead Load of Liner and Steel Roof in a 2D Axisymmetric Static Analysis Model

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.

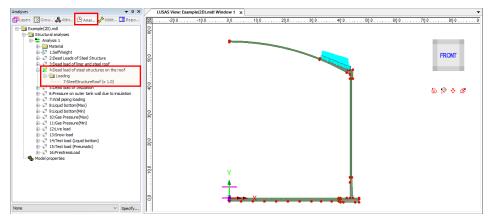


Fig 89 Dead Load of Steel Structures on the Roof in a 2D Axisymmetric Static Analysis Model

Dead load of Insulation

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

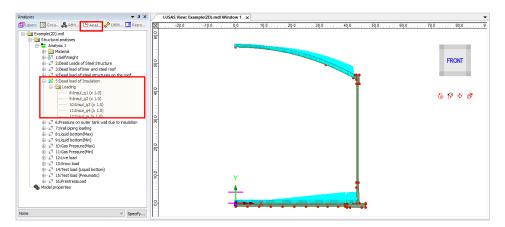


Fig 90 Dead Load of Insulation in a 2D Axisymmetric Static Analysis Model

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.

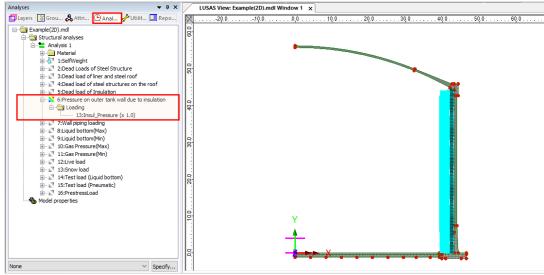


Fig 91 Insulation Pressure Load in a 2D Axisymmetric Static Analysis Model

Wall piping loading

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

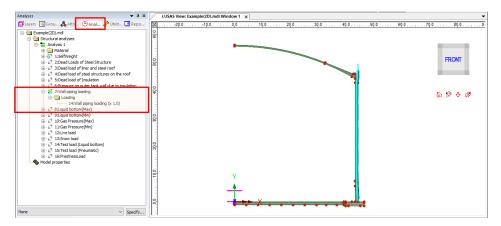


Fig 92 Wall Piping Loading in a 2D Axisymmetric Static Analysis Model

Liquid bottom (Max, Min)

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

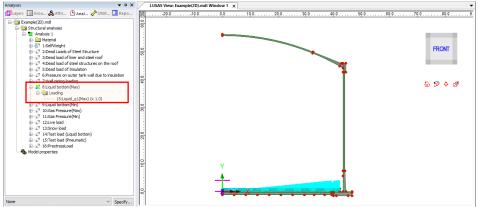


Fig 93 Liquid Bottom Loading in a 2D Axisymmetric Static Analysis Model

Gas pressure (Max, Min)

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

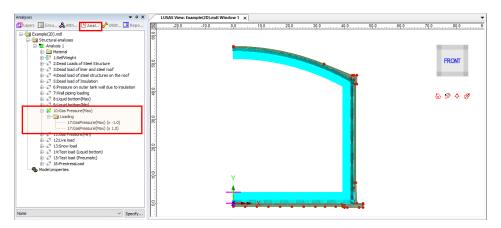


Fig 94 Gas Pressure Loading in a 2D Axisymmetric Static Analysis Model

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.

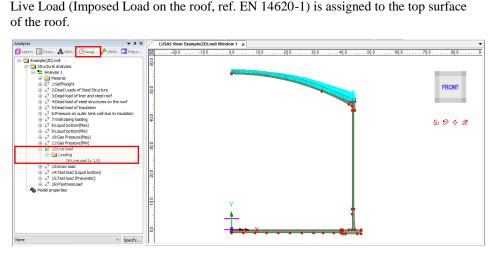


Fig 95 Live Load in a 2D Axisymmetric Static Analysis Model

Snow load

Snow load acts on the top surface of roof.

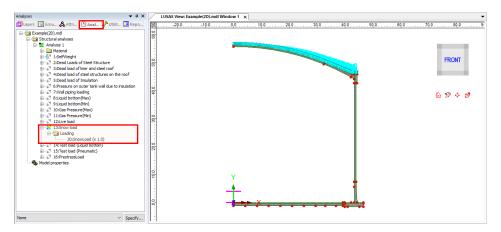


Fig 96 Snow Load in a 2D Axisymmetric Static Analysis Model

Test load (Liquid bottom)

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

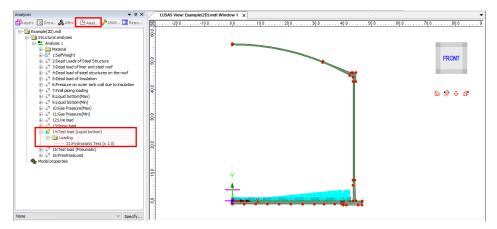


Fig 97 Test Load (Liquid Bottom) in 2D Axisymmetric Static Analysis Model

Test load (Pneumatic)

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

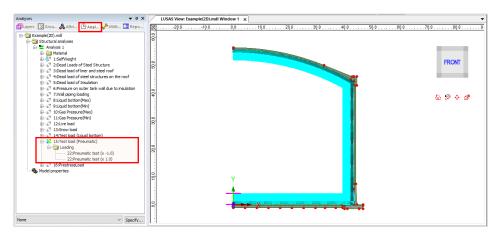


Fig 98 Test Load (Pneumatic) in a 2D Axisymmetric Static Analysis Model

Prestress Load

The effect of prestressing steel shall be converted to an equivalent external load and used as input in the Wizard.

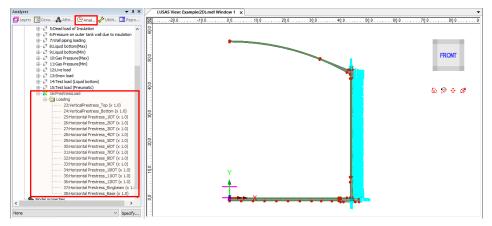


Fig 99 Prestress Load in a 2D Axisymmetric Static Analysis Model

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 👻 🕈 🗙	LUS	AS View: Example(2D).mdl Window 1 🗙
🖾 Layers 🚳 Grou 歳 Attri 🕑 Anal 🥓 Utilit 🛄 Repo	X	20.0 . 0,0 . 20.0 . 40.0 . 60.0 . 80.0 .
ConvectCo) and ConvectCo) and ConvectCo) and ConvectCo) and ConvectCo) and ConvectCo) Convec() ConvectCo) ConvectCo) ConvectCo) Convec()	. 200 . 40.0 . 60.0	Properties × Contour Results Contour Display Contour Range Seed Colours FRONT Entity Stress - Axisymmetric Solis × Component SX Display Averaged nodal × Transform Set Display on slice(s) Draw in slice local direction
Deformations No deformations drawn Window summery Details Wew axes Details Wew axis cube Details		OK Cancel Apply Help

Fig 100 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.

Analyses 👻 🖡 🗙	LUSAS View: Example(2D).mdl Window 1 X	-
🔟 Layers 🔯 Grou 歳 Attri 🕒 Anal 🥓 Utilit 💶 Repo	× 40.0	
Image: Second (20) and Image: Second (20) and Image: Second (20) and (20)	Analysis Analysis 1 Loadcase: 1 SelfWeight Entity Stress - Axalysis 1 mys Entity Stress - Axalysis 1 mys Component 5X (Unit IS M/n) 	RONT
None V Specify	11	

Fig 101 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.

	w: Example(2D).mdl Window 1 🗙
🖾 Layers 🔞 Grou 🚴 Attri 🕑 Anal 🥜 Utilit 💶 Repo 🕅 40.0	
Geometry Control Contr	ysis Analysis 1 case: 1:SelfWeipht Its Fie Example 20 - Analysis 1.mvs
Attributes Properties	× Solid Properties ×
Value Results Value Duplas Arronsison Weine propert Arronsison Weine propert Component RX Component RX Component RX Transform Setus Inone	Value Results Values Display
	18 1863 Significant figures Decimal places 6
Display on slice(s) OK Cancel	✓ Jeema paces ✓ ✓ Show trailing zeros Choose font Pen Symbol Font angle Pen ≠ 19 € Choose pen
Deformations No deformations drawn	
Window summary Details ✓ Wew axes Details ✓ Wew axis cube	OK Cancel Apply Help

Fig 102 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.

Layers 🗸 🗘 🗙	LUSAS View: Example(2D).mdl Window 1 x	•
🖾 Layers 🐼 Grou 歳 Attri 🕒 Anal 🥓 Utilit 💶 Repo	X 40.0	
Burget(C)) ad Europe(C)) ad Window 1 Utilities Wech Geonetry Cathrobuse Deformed neth Constructions SIX (Stees - Assymmetric Solid) Cathrobuse Wen properties Wen properties	Analysis Analysis 1 Loadcase: 1:SetIWeight Results If the Example 2DAnalysis 1 mys Entry, Stess-Analymmetric Solid Component SX (Units Nmr)	FRONT
	2 v	
Deformations No deformations drawn		
Window summary Details		
View axes Details		441878E6 3.00790E6
View axis cube		3.0079020

Fig 103 Values Displayed for Selected Nodes in a 3D Shell Model

Graph through 2D

Define a line from Geometry>Line>By Coords.

Enter Coo	ordinates		×
Grid sty			
	X Y 30 30 60 30	2 0 þ	•
Local co	ordinate		
Global	coordinates		\sim
[Set as active loc	al coordinate	
OK	Cance	l Help	

Fig 104 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	
Graph Through 2D × By cursor Grid offset Grid offset	Loadcases and Extent × Loadcases ©[:SelfWeight] Active All Specified Select. Create new window for each loadcase Extent Visible model
OK Cancel Help	< 뒤로(i) 다음(i) > 취소 도움말

Fig 105 Graph Through 2D in a 2D Axisymmetric Solid Model (1)

Entity Component Transform	Set None	0.0	Assultant effects from 2D model Resultant effects from slice Wean normal stress Sz Winomal stress Sz Winoma istress Sz Winoma istress Sz Winoma istress per radian Wachtual shear stress per radian Wachtual shear stress per radian	Display Graph Display Title SX in the Wall X Distance Y SX Show grid Show symbols Corner labels Auto-update Include existing graphs	X scale (a) Automatic (b) Manual min (b, c) max (1, c) (c) Use logarithmic scale Scale factor (1, c) Y scale (c) Manual min (b, c) max (1, c) (c) max
	Arc centre	(0.0 (0.0 7 0.0		v	Use logarithmic scale Scale factor 1.0
Width for (corridor averaging	0.0	×	Name Graph for SX	Save in treeview Display now

Fig 106 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.

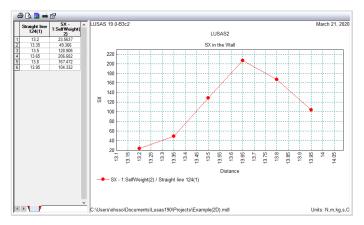


Fig 107 SX Graph for Sliced Line in a 2D Axisymmetric Solid Model

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.

Results component Entity Stress - Axisymmetric Solid Component SX Transform Set None One Calculate distance as angle Arc centre Arc centre Y 0.0 Z 0.0 Z	Resultant effects from 2D model Resultant effects from slice Mean normal stress 5z Mean lorses Mean stress Ser Mean stress Ser Mean stress Ser Mean stress Ser Mean shear stress per radian Mean stress per radian Actual shear stress per radian	Display Title Resultant Effects Titlckness V Results Show grid Corner labels Corner labels Graph 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 n	

Fig 108 Graph Through 2D in a 2D Axisymmetric Solid Model (3)

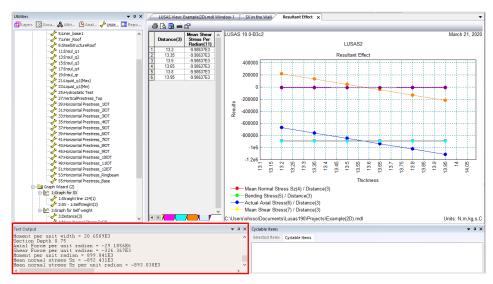


Fig 109 SX Graph for Sliced Line in a 2D Axisymmetric Solid Model

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.

Dutput filename	stati2D					
Working folder	O Current	 User Define 	d			
Save in	C:\Users\of	nsso₩Documents₩Lusa	is190₩Projects₩	'stati2D_Wall_R	ing	
Target						
⊖ Base slab	(🖲 Wall + Ringbeam		⊖ Roof		
Loadcases			Range (Y Co	ord)		
1:SelfWeight 2:Dead Loads of Steel Struc 3:Dead load of liner and ste	ture	^	Start:	0,0	m	
3:Dead load of liner and ster 4:Dead load of steel structur 5:Dead load of Insulation	el roof es on the roof		Finish :	45,88	m	
6:Pressure on outer tank wa 7:Wall piping loading 8:Liquid bottom(Max) 9:Liguid bottom(Min) 10:Gas Pressure(Max) 11:Gas Pressure(Min) 12:Live load	II due to insulation		Interval :	0,5	m	
13:Snow load 14:Test load (Liquid bottom)	1	~				

Fig 110 Export Forces for a 2D Axisymmetric Solid Model (1)

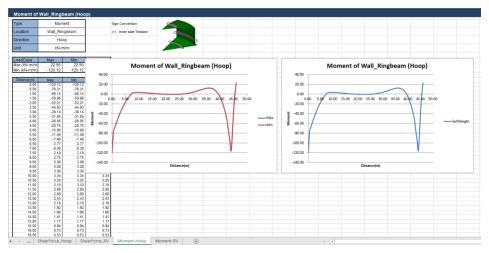


Fig 111 Section Force Spreadsheet for Self Weight

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

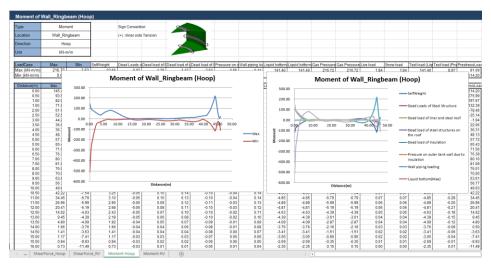


Fig 112 Section Force Spreadsheet for All Loadcases

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> Staged Construction Analysis as shown in [Fig 113].

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

ank definition data	Tnk1	\sim
lodel filename	Example	
aved model file path	C:\Users\obsco\Documents\USAS190\Projects\Exam	pl
lement size (m)	0,2	
ptions		
Roof ratio for 1st built	0,5	
🖂 Include self weight	☑ Include structural loadings	

Fig 113 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 114 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 115]. 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:

- □ the subsequent loadcases inherit the stress and strains from the previous loadcases
- □ the subsequent loadcases inherit the support conditions from the previous loadcases
- □ loading is not inherited

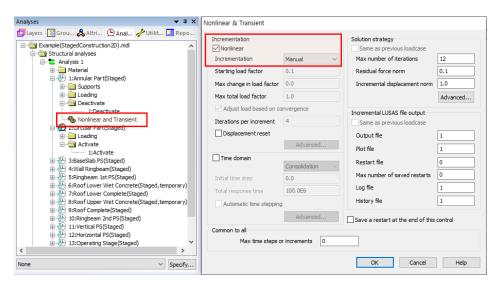


Fig 115 Nonlinear Control for a Staged Construction Analysis

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.

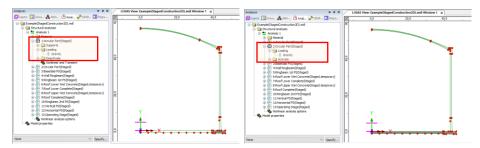


Fig 116 Loadings for Stage 1~2

Stage 3 : BaseSlab PS ~ Stage 4 : Wall RingBeam

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

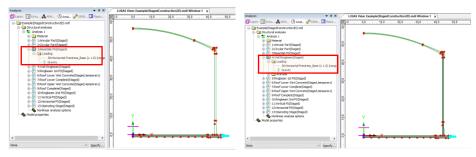


Fig 117 Loadings for Stage 3~4

Stage 5 : RingBeam PS ~ Stage 6 : Roof Lower Wet Concrete

Horizontal Prestress for the RingBeam is added in Stage 5, but with load factor of 0.5 which means only 50% of the defined RingBeam prestress is applied at this stage. By default, this ratio is set to the **'Roof Ratio for 1st built'** from the dialog input.

Stage 6 is same with Stage 5 as the Wizard does not define the loading for air raising pressure and the loading for wet concrete. If this loading should be considered, users can define this loading separately and assign to Stage 6.

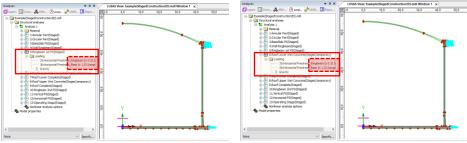


Fig 118 Loadings for Stage 5~6

Stage 7 : Roof lower complete ~ Stage 8 : Roof upper Wet Concrete

At Stage 7, the lower part of the Roof is added to the structure assuming that it is built in 2 stages, with 50% initially and the other 50% later. If the Roof is built in one go, there is no need to modify this model and this stage can be ignored when post-processing.

Stage 8 is for considering the upper 50% of the Roof as a loading to check the stability of lower half of the Roof while the upper part is being built. The self weight of the upper part of Roof is added at Stage 8.

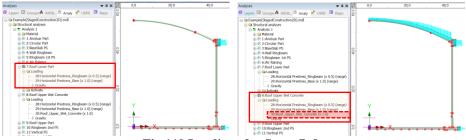


Fig 119 Loadings for Stage 7~8

The weight of the upper part of the roof is computed by the Wizard from the geometry as marked in [Fig 120]. The total weight is computed as 48.09E6 N, and the area of top surface of the Roof Lower Part is computed as $6218.422m^2$. From this the loading of 7.73334E3 N/m² is defined. This can be verified by assigning self weight to the upper part of the Roof and checking the reaction.

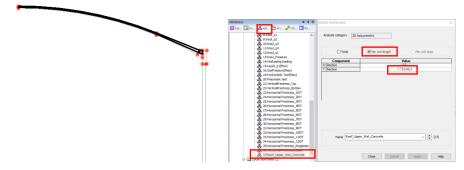
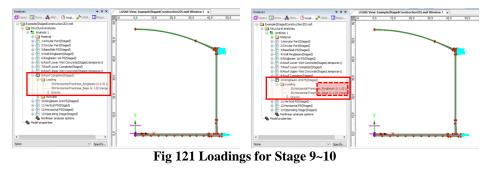


Fig 120 Load Definition for the Wet Concrete of Upper Part of the Roof

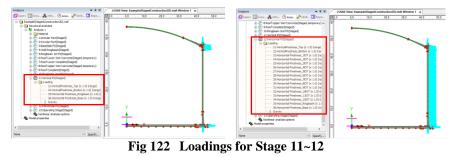
Stage 9 : Roof Complete ~ Stage 10 : RingBeam 2nd PS

At Stage 9, the upper part of the roof is added, and the loading for Wet Concrete is removed. The remained RingBeam prestress is added at Stage 10. (Load factor is changed from 0.5 to 1.0)



Stage 11 : Vertical PS ~ Stage 12 : Horizontal PS

The structure is fully built at Stage 10, and the additional loading of the Vertical Prestress is added to Stage 11, and Horizontal Prestress is added to Stage 12.



Stage 13 : Operating Stage

Stage 13 models the operating (in-service) stage.

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

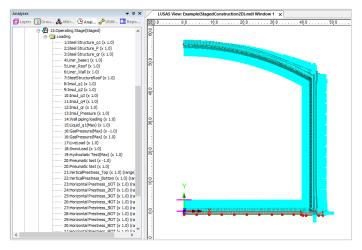


Fig 123 Loadings for Stage 13 of the Operating Stage

The prestress loadings are defined with the values obtained from User Input dialog and <u>only the short-term prestress is applied</u>. If long-term prestress is desired to be used at this stage, users can either update the load factor used at this stage or create separate prestress loading and substitute as appropriate.

Load Combination

Looking at U-C1-1 from the sample design load combination at [Fig 124], it might be necessary to extract the pure prestress (PS) effect from the staged construction analysis, due to the different load factors for self weight and the prestress loading respectively.

1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19
				Permanent														
	De	escription		Outer tank		Others		p and nkage		P	restress				Ro	of frame/	concrete	
no.	Code	Details	Outer tank WO roof	Outer tank WO uper roof	Outer tank Full	Others	Early	Late	Rb 1st	Rb All	Rb + Vertical	All PS Early	All PS Late	Roof Frame1	Roof Frame 2	Roof Frame 3	1st layer concrete	2nd layer concrete
1	U-C1-1		1.35						1.30					_				
2	U-C1-2	Tank WO roof + RB 1st	1.35						1.00									
3	U-C1-3	PS	1.00						1.30									
4	U-C1-4]	1.00						1.00									
5	U-C2-1		1.35						1.30					1.50				
6	U-C2-2	Tank WO roof + RB 1st	1.35						1.00					1.50				
7	U-C2-3	PS + Roof frame 1	1.00						1.30					1.50				
8	U-C2-4	1	1.00						1.00					1.50				

Fig 124 Example of a Design Load Combination

The 1st PS is introduced at Stage 5, hence the pure effect of 1st PS can be obtained by defining a load combination for 'Stage 5 -Stage 4'.

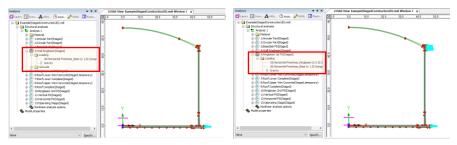


Fig 125 Loadings for Stage 4 and Stage 5

This can be achieved by specifying a load factor of -1 for Stage 4, and 1 for Stage 5 as illustrated in [Fig 126]. The load combination of 'Pure 1st PC' will be defined, and can be used for defining the design load combination U-C1-1 ~ U-C1-2 of the sample design load combination table.

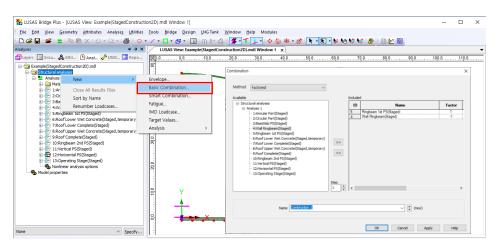


Fig 126 Definition of Load Combination for Pure PS effect

Adding Extra Stages

If additional stages are required, the ability to Copy and Paste loadcases will be useful, as illustrated at [Fig 127]. Other attributes such as 'Activate' and 'Loading' are also copied.

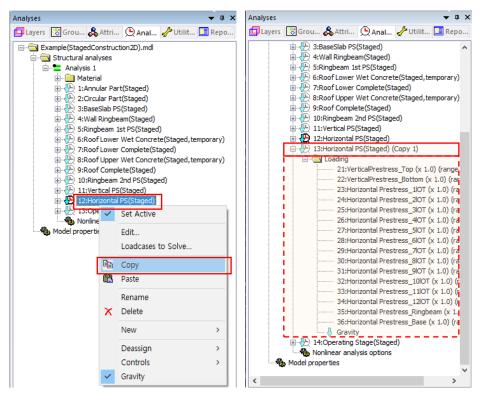


Fig 127 Adding Stages in the 2D Axisymmetric Staged Construction Analysis Model

2D Axisymmetric Thermal Analysis

User Inputs

The required user inputs for this model are as shown in [Fig 128].

nk Definition							
⊇ Include pile data	Include insulation	🖂 Includ	le seismic data				
tructure Definition Material Proper	ties Boundary Conditions Load	ng Prestress Load	Pile Arrangement (3D)	Seismic input 1	Seismic input 2		
Concrete tank Insulations							
Base slab and Roof Wall and							
	Hing beam						
Base slab (Units: m)	00.0						
Circular part length (L_inner)	39,8		-		Linner	Louter	
Circular part depth (D_inner)	1,2		- +	*			
Tapered section length (W_t)	0,6		Dinner	Dheating		Douter	r
Annular part length (L_outer)	6.7						
Annular part depth (D_outer)	1.5					•••	
Base heating (D_heating)	0, 386					Wt	
Roof (Units: m)							
Radius of inner roof (R_roof_i)	86,406		Troof				
Radius of outer roof (R_roof_o)	86,906		ŢŢ			slı	
Height from the top of the base the roof (R_Height)	slab to the topmost of 56,254		Rroo	Rroof_o		sl ₂	
Distance of tapered section 1 (sl1) 10,079					Hringbeam	2
Distance of tapered section 2 (sl2) 0.6					+ Rsl_heigh	
Set zero Set defau	It						
	Nam	Tnk1			~ 🕂 (1)		
					Cle	ose Cancel	Apply Help

Fig 128 User Inputs for 2D Axisymmetric Thermal Analysis

The user dialog is available from LNG Tank>Thermal Analysis Wizard as shown in [Fig 129].

• Enter a model filename and set the element size to **0.2**, the soil height above the soffit of the thickened slab to **1**, and press **OK** to build the model.

LNG Tank - Thermal Analysis		×
Tank definition data	Tnk1 ~	
Model filename	Example	
Saved model file path	C:#Users#ohsso#Documents#LUSAS190#Projects'	
Element size (m)	0.2	
🖂 Include soil		
Soil height above slab bottom (m)	0.9	
Soil depth (m)	25,0	
	OK Cancel Help	

Fig 129 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 1.0 m above the soffit of the thickened base slab..

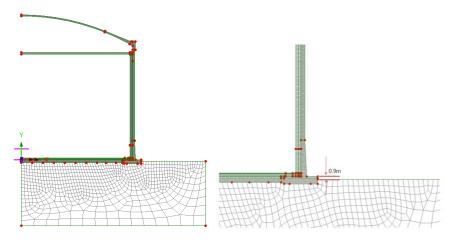


Fig 130 Elements for 2D Axisymmetric Thermal Analysis Model

Geometric Properties

No geometric properties are required for 2D axisymmetric model.

Material Properties

User defined material properties are assigned to the relevant surfaces.

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

Isotropic		< Isotropic	×
Elastic Thermal Elastic Creep Elastic Thermal Elastic Thermal	Shrinkage Viscous Two phase Ko Initialisation	Elastic Internal Plastic Creep Damage Shrinkage Viscous Elastic Thermal	Two phase Ko Initialisation
Poisso Mass	10 modules 33669 en ordin 02 annaly 28/3 ann of Thermal leganison 10.06.6	Plane change state 1000 Thermal conductively Specific two capacity Density Content to the day by Concerts for all hybridizes Convert type 1000	Value 2.0 2.257.6 1.0
Name BaseSlab	~) 🐑 (1)	Name BoseSlab	~ (1)
	Close Cancel Apply Help	Close Cano	tel Apply Help

Fig 131 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 132].

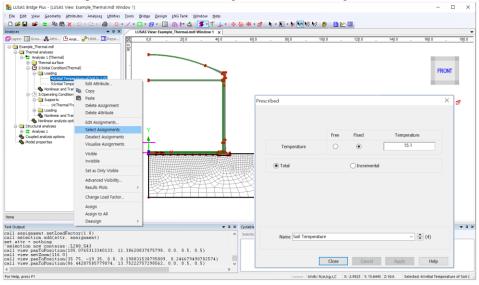


Fig 132 Initial Soil Temperature in a 2D Axisymmetric Thermal Analysis Model

Initial temperature of structure is defined and assigned as shown in [Fig 133].

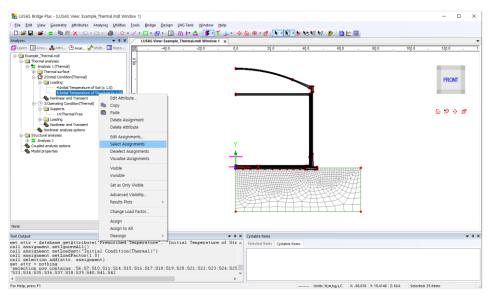


Fig 133 Initial Structure Temperature in a 2D Axisymmetric Thermal Analysis Model

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.

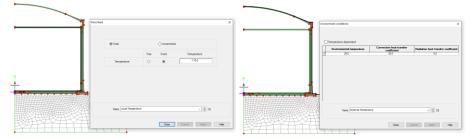


Fig 134 Operating Temperatures in a 2D Axisymmetric Thermal Analysis Model

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

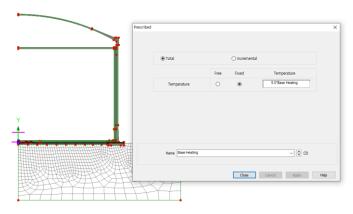


Fig 135 Base Heating Temperature in 2D Axisymmetric Thermal Analysis Model

The Base heating temperature is assigned to selected lines as shown in [Fig 136]. The range of the loading is defined using a LUSAS field variation and can be modified by editing the values of 'Base Heating'.

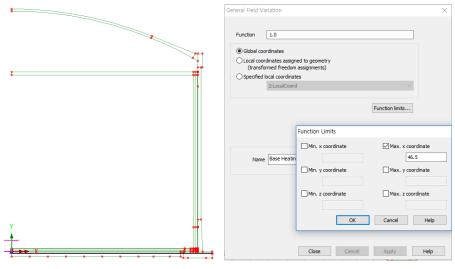


Fig 136 Base Heating Temperature Variation in 2D Axisymmetric Thermal Analysis Model

Refer to the section entitled *Examples – User Inputs* :

2D Axisymmetric Thermal Analysis for more information.

3D Shell Static 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 137].

Tank Definition										×
🖂 Include pile data		🖂 Include insulation		🗸 Includ	e seismic data					
Structure Definition	Material Properties	Boundary Conditions	Loading	Prestress Load	Pile Arrangement (3D)	Seismic input 1	Seismic input 2			
Concrete tank Ins	ulations									
Base slab and R	oof Wall and Ring	beam								
Base slab (Unit	ts:m)									
Circular part ler	ngth (L_inner)	1	39,8		L.		Linner	Louter		
Circular part de	pth (D_inner)	ĺ	1,2		-	•		-	Ŧ	
Tapered section	n length (W_t)	L	0,6		Dinner	Dheating			Douter	
Annular part len	igth (L_outer)	[6, 7		-	,			Cotter	
Annular part dep	pth (D_outer)		1.5						L	
Base heating (E)_heating)	[0, 386					Wt		
-Roof (Units: m)									
Radius of inner	r roof (R_roof_i)	1	36, 406		Troof					
	roof (R_roof_o)		36, 906			1		, slı		
Height from the the roof (R_Hei	e top of the base slat ght)	o to the topmost of	56, 254		Rroot	J Rroof o		sl2		
Distance of tap	ered section 1 (sl1)	[10,079					H H	ringbeam_2	
Distance of tap	ered section 2 (sl2)	[0,6					<u>~</u> ₹.	Rsl_height	
		-								
Set zero	Set default									
			Name Tr	42						
			Name Tr	N2			✓ ★ (new)			
							0	K Cancel	Apply	Help

Fig 137 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 **1.5**, and the other values as shown in [Fig 138].
- Ensure '3D shell' is chosen for Analysis Type.
- Enter **10** for Number of Eigenvalues.
- Leave the default values for **Wind Load** (EN1991) parameters. Soil Height above slab bottom is for the height from the soffit of the thickened base slab to the ground surface, so enter 0.9 in this example. The height (z) starting from ground surface is used for computing wind load and the value could be either positive (+) or negative (-).
- Select 4 for Number of buttresses, input 1.0 (m) for Extruded thickness and 5.0(m) for Buttress width.

Tank - Static Analysis				
Tank definition data Tnkl Model filename Example Saved model file path C:\Users\Userd\Users\Users\Users\Users\User\Users\Users\Users\Users\U			~	
			6	Half only model
		Documents\U	SAS190\Projects\Exampl	
Element size (m) Analysis type	1.5			
◯ 2D Axisymmetric solid			 3D Shell 	
Wind load (EN1991, 1, 4, 2005)			Buttress	
Basic wind velocity	37.5	(m/s)	Number of buttress	4 ~
Roughness length	3.0E-3	(m)	Extruded thickness	1.0 (m)
Minimum height	1.0	(m)	Buttress width	
Orography factor	1.0		Duiress widdi	5.0 (m)
Terrain factor	0,156			
Turbulence factor	1,0		Eigenvalue	
Air density	1,25	(kg/m^3)	Number of eigenvalues	10
Soil height above slab bottom	0,9	(m)	🗌 include non-structur	al masses

Fig 138 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 1.5m as per user input. Quadratic shell elements (QTS8) are used.

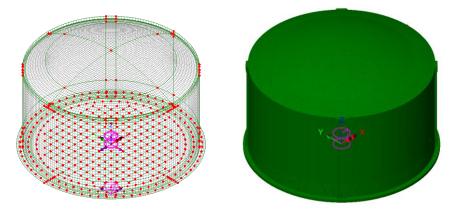


Fig 139 Mesh Arrangement and Geometric Properties for a 3D Shell Model

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.

Layers	+ a × 0.0	-70.0	-60.0	-50.0	-40.0	-30.0	-20.0	-10.0	0.0	10
Layers © Layers © Layers © Groups ▲ Antub @ Analys ✓ © Bample(30).mdl Window 1 © Unders © Wash © Octometry © Attributes © Octometry © Deformed mesh • View properties	Utilities a Reports 000000000000000000000000000000000000	18 Maximum sl Dotted Outline only Three Outlinetons only if se	Choose pen hade 60.0 shold 25.0	-50.0 ×	-40.0					
	% of elements remaining Colour by		Set		4 4 ++	*		++	÷.	
	Close	Cancel Ag	Hel	P	4 ++ ++	4 - >>-	*	4		
	R.				4 4 90 90	4 ++	Å +++	*		
Deformations No deformations drawn	10.0			L	1			4		

Fig 140 Element Local Axis in a 3D Shell Model

Geometric Properties

Geometric properties are defined as per user inputs. [Fig 141] illustrates how properties were defined for varying sections at the edge of the roof. The **variation dataset** can be reviewed from the Utilities $\sqrt[4]{}$ treeview.

Attributes			τņχ	0.0	. 32.0	34.0	36.0	. 38.0	40.0	42.0	44.0
🗗 Layers Å Attrib 🕒 Analy	💶 Reports 🔝	Groups 🥓	Utilities	Ω.			and a second				
Example(3D).mdl			^				Surface	e Grid Variation			×
	Geometric	Surface		48.0 50.0				K order Constant f order Quadratic X1 1.18342 0.673061 0.5	 ✓ 1 ✓ 3 		
- & 5:Roof, RingBeam, D - & 6:BaseSlab, Jnner & 7:BaseSlab, Japer & 8:BaseSlab, Outer & 9:Wall_Dummy, Boy & 10:Wall_Qummy, Buy & 11:Wall_Tapered, Bu & 13:Wall_Regular	ul tt	Thickness Eccentricity			Value. Thickness Roof2 Eccentricity Roof.			Name Thickness Roof	2	v ♥ (t)	
- & 14:Wall_Buttress - & 15:Ringbeam_Reguli - & 16:Ringbeam_Buttre - Material (6) - Gistropic (5) - & 1:BaseSlab		Name Roof	2			~ 🗘 (;	2)				
& 2:Wall -& 3:RingBeam -& 4:Roof		L	Close		ncel A	pply	Help				

Fig 141 Geometric Properties for Roof in 3D Shell Model

TEST CASE

If either the 'Extruded Thickness' or the 'Buttress Width' is set to 0 (zero), the mesh is defined as shown below.

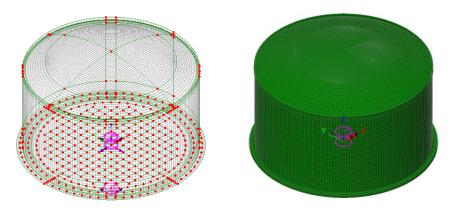


Fig 142 Mesh Arrangement and Geometric Properties for a 3D Shell Model with no Buttresses

Material Properties

Structural members

Material properties are defined and assigned as shown in [Fig 143].

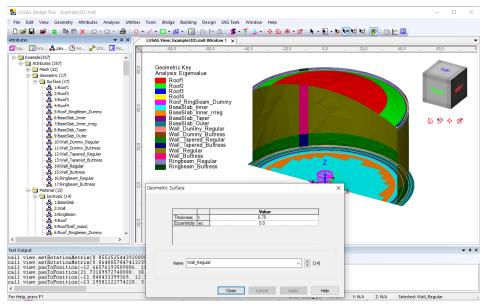


Fig 143 Material Properties in a 3D Shell Model

Support Conditions

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

Include pile	e data		🖂 Inc	lude insulatio	n		🖂 include	seismic data									
		Material Propertie:		dary Condition								أمسيني					
			_	Jary Condition		g Pres	aress Loau	Pile Arrangement (3	0) 50	eismic inp	ut i Seisi	ne mput z					
		coordinates (Units															
P1	P2	P3	P4	P5	P6	P7	P8	P9			^		111	::::			
0,0	4,2	8,4	12,6	16,8	21.0	25,2	29,4	33,6					Cros	sswise pile	* / / /		
0,0	4.2		12,6 12,6	16,8	21,0 21,0	25,2 25,2	29,4 29,4	33,6						::::	Circumferer		
0.0	4.2	8,4	12,6	16,8	21.0	25.2	29,4							::*		itiai piies	
0.0	4.2	8.4	12.6	16,8	21.0	25.2	29,4						111	1	<u></u>		
0.0	4.2		12,6	16,8	21.0	25,2	20,1							1			
	, in				a. a						~		1.1				
P1		coordinates (Unit	P4	P5	P6	P7	P8	P9			^	Add n	le in X		Define nil	e locations	
0.0	P2 0.0	0,0	0,0	0,0	0,0	0,0	0,0	0,0			Â						
-4.2	-4,2	-4,2	-4,2	-4.2	-4.2	-4.2	-4.2	-4.2				Add pi			t zero	Set def	aults
-8.4	-8,4	-8,4	-8,4	-8.4	-8.4	-8,4	-8,4	7.6					se pile stiffn				
-12,6	-12,6		-12,6	-12,6	-12,6	-12,6	-12,6					Vertical :	stiffness (kN	l/m)		523,018E3	
-16,8	-16,8		-16,8	-16,8	-16,8	-16,8	-16,8					Horizont	al stiffness ((kN/m)		42,297E3	
-21.0	-21.0		-21.0	-21.0	-21.0	-21.0						Туре				CrossPiles	3
on o	ntial nil		05.0	05.0	05.0	1			_		~					-	
rcumferen			heta	Number of	Vertical	Stiffness	Horizz	antal		-							
ID	R ()	(uegie		piles	Vertical (kN	/m)			00					1	50	t defaults	
	6,7	0,0	56		523,018E3		42,297E3	Cirpiles									
	0,8	0,0	60		523,018E3		42,297E3	Cirpiles							S	Get zero	
44	4,9	0,0	68		523,018E3	_	42,297E3	Cirpiles	_						Ado	d pile row	
														i i	Bernr	ove pile row	
		Attributes			Name		× Structura	al Supports			· •		Close	Carro	×	Apply	Hel
				Attr 🕑 An	📌 Util	~ ‡					-		Close	Cano	×	Apply	Hel
			- &	8:Base_Wall_D ts (4)	📌 Util	~ ‡	-	al Supports ysis category 3D			-		Close	Cane		Apoly	Hel
				8:Base_Wall_D ts (4) CrossPiles	🦨 Util. ummy	~ ‡	-				-	-		Cane		Apply	Help
			- & 110 - & 110 - & 210 - & 310	8:Base_Wall_D ts (4) CrossPiles Crplies(R = 36.7 Crplies(R = 40.8	🦨 Util ummy)	~ ‡	-			Free	Fixed	-	ng stiffness			Apply	Hel
		Lay	Sunnor Sunnor 	8:Base_Wall_D ts (4) CrossPiles Croles(R = 36.7 Croles(R = 40.8 Croles(R = 44.9	🦨 Util ummy)	~ ‡	-		x	Free	-	-				Apply	Hel
		Lay	- & Support - & Support - & 2:0 - & 3:0 - & 3:0 - & 4:0 - & Dis	8:Base_Wall_D ts (4) prossPiles prpiles(R = 36.7 prpiles(R = 40.8 prpiles(R = 44.9 p (273) crete (2)	🦨 Util ummy)	~ ‡	-		x	0	Fixed	Spri	ng stiffness	6		Apply	Hel
		Lay	- & 1:0 - & 2:0 - & 3:0 - & 3:0 - & 4:0 - & Dis - → Dis	8:Base_Wall_D ts (4) CrossPiles Cropies(R = 36.7 Cropies(R = 40.8 Cropies(R = 44.9 g (273) crete (2) (Patch (2)	🖋 Util. ummy)	~ ‡	-	ysis category 3D	Y	0	Fixed O	Spri	ng stiffness 42.297Er 42.297Er	6		Apply	He
		Lay	Support - & 110 - & 210 - & 310 - & 31	8:Base_Wall_D ts (4) CrossPiles Orpies(R = 36.7 Orpies(R = 44.9 (273) Crote (2) Patch (2) Patch (2)	🖋 Util. ummy))	~ ‡	-	ysis category 3D	Y Z	0 0 0	Fixed O O	Spri	ng stiffness 42.297E4	6		Apoly	Hel
		Lay	- & 1:0 - & 1:0 - & 2:0 - & 3:0 - & 3:0 - & 4:0 - & 4:0 - & 1 - &	8:Base_Wall_D ts (4) inossPiles inples(R = 36.7 Orples(R = 40.8 Orples(R = 44.9 Orples(R = 44.9)Orples(R = 44.9 Orples(R = 44.9)Orples(R = 44.	🖋 Util. ummy))	~ ‡	-	ysis category 3D	Y	0	Fixed O	Spri	ng stiffness 42.297Er 42.297Er	6		Apply	Hel
		Lay	Sunor Sunor 20 20 20 20 20 20 20 20 20 20	8:Base_Wall_D ts (4) rossPiles riples(R = 36.7 riples(R = 41.9 (273) rotete (2) Patch (2) 21:LiveLoa 23:SnowLo 23:SnowLo Patch dvisi Patch dvisi Patc	d ad	~ ‡	-	ysis category 3D	Y Z	0 0 0	Fixed O O	Spri	ng stiffness 42.297Er 42.297Er	6		epply	Hel
		Lay		8:Base_Wal_D ts (4) rossPiles imples(R = 36.7 imples(R = 41.9 (273) crete (2) Patch (2) & 21:LiveLoa & 23:SnowLo & Patch divisi uctural (271) 1:Steel Structu 2:Steel Structu	d d ad ons re_Q1	~ ‡	-	ysis category 3D	Y Z X	0 0 0	Fixed O O O O	Spri	ng stiffness 42.297Er 42.297Er	6		epply	Hel
		Lay	2 Santa 2 20 2 20 2 20 2 20 2 20 2 20 2 20 2 2	8:Base_Wal_D ts (4) 2rossPiles 2rples(R = 36.7 2rples(R = 36.7 2rples(R = 41.9 9 (273) crete (2) Patch (2) 21:LiveLoa 22:StowLo 23:SnowLo 23:SnowLo 23:SnowLo 23:SnowLo 23:StowL	d d ad ons re_Q1	~ ‡	-	ysis category 30 Translation in Rotation about	Y Z X Y	0 0 0 0	Fixed O O O O O	Spri	ng stiffness 42.297Er 42.297Er	6		Apply	Hel
		Lay	5 State	8:Base_Wall_D ts.(4) rossPiles imples(R = 46.9 imples(R = 44.9 (273) orete (2) Patch (2) & 21:LiveLoa & 23:SnowLo & Patch divisi Patch (2) i.1:Steel Structu 2:Steel Structu 2:Steel Structu 2:Steel Structu	d d d ad ons re_q1 re_p re_qr	~ ‡	-	ysis category 3D	Y Z X Y	0 0 0	Fixed O O O O	Spri	ng stiffness 42.297Er 42.297Er	6		Agoly	Hel
		Lay	2000 2000 2000 2000 2000 2000 2000 200	8:Base_Wall_D ts.(4) rossPiles imples(R = 46.9 imples(R = 44.9 (273) orete (2) Patch (2) & 21:LiveLoa & 23:SnowLo & Patch divisi Patch (2) i.1:Steel Structu 2:Steel Structu 2:Steel Structu 2:Steel Structu	d d d ad ons re_q1 re_p re_qr	~ ‡	Anal	ysis category 30 Translation in Rotation about	Y Z X Y	0 0 0 0	Fixed O O O O O	Spri	ng stiffness 42.297Er 42.297Er	6		Apply	Hel
		Lay		8:Base_Wal_D ts (4) crossNess srples(t = 30.7 prossNess srples(t = 40.8 project(t = 40.8 pr	d d d ad ons re_q1 re_p re_qr	~ ‡	Anal	ysis category 30 Translation in Rotation about Hinge rotation Torsional warping	Y Z X Y	 ○ ○<	Fixed 0 0 0 0 0 0 0 0 0 0 0 0 0	Spri	ng stiffness 42.297Er 42.297Er	6		Apply	Hel
		Lay	- 43 11 12 13 4 4 10 10 10 10 10 10 10 10 10 10 10 10 10	8:Base, Wall_D ts (4) prossNess prplicat(x = 36.7 prplicat(x = 36.7 prplicat(x = 40.8 prplicat(x = 40.8 prplicat(x = 44.9 or (273) or (273	d d d ad ons re_q1 re_p re_qr	~ ‡	Anal	ysis category 30 Translation in Rotation about Hinge rotation	Y Z X Y	 • •<	Fixed O O O O O O O O O O O O O O	Spri	ng stiffness 42.297Er 42.297Er	6		Apply	Hel
		Lay	- 42 11 12 13 4 4 15 18 17 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	8:88ac, Wal _D ts (4) rossPies rossPies rossPies - 30.7 rossPies - 40.8 rossPies - 40.8 ro	d d d ad eRaof	~ ‡	Anat	vis category 30 Translation in Rotation about Hinge rotation Torsional warping Pore pressure	Y Z X Y Z	 • •<	Fixed 0 0 0 0 0 0 0 0 0 0 0 0 0	Spri	ng stiffness 42.297Er 42.297Er	6		Apply	Hel
		Lay	- 42 11 12 13 4 4 15 18 17 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	8:88ac, Wal _D ts (4) rossPies rossPies rossPies - 30.7 rossPies - 40.8 rossPies - 40.8 ro	d d d ad eRaof	~ ‡	Anal	visis category 30 Translation in Rotation about Hinge rotation Torsional warping Pore pressure ing stiffness distributi	Y Z X Y Z	 • •<	Fixed 0 0 0 0 0 0 0 0 0 0 0 0 0	Spri	ng stiffness 42.297Er 42.297Er	6		Aabiy	Hel
		Lay	. 48 11 12 13 14 18 18 17 18 14 18 18 18 18 18 18 18 18 18 18 18 18 18	8:89ae_Wal_D ts.(4) prosetiles protective - 30 protective - 30 protective - 30 protective - 30 protective - 30 protective - 30 2723) crete (2) Patch (2) P		~ ‡	Anal	ysis category 30 Translation in Rotation about Hinge rotation Torsional warping Pore pressure ing stiffness distributis Stiffness	Y Z X Y Z	 • •<	Fixed 0 0 0 0 0 0 0 0 0 0 0 0 0	Spri	ng stiffness 42.297Er 42.297Er	6		acoly	Hel
		Lay	. 48 11 12 13 14 18 18 17 18 14 18 18 18 18 18 18 18 18 18 18 18 18 18	8:89ae_Wal_D ts.(4) prosetiles protective - 30 protective - 30 protective - 30 protective - 30 protective - 30 protective - 30 2723) crete (2) Patch (2) P		~ ‡	Analy Spp	visis category 30 Translation in Rotation about Hinge rotation Torsional warping Pore pressure ting stiffness distributi Stiffness distributi	Y Z X Y Z	 • •<	Fixed 0 0 0 0 0 0 0 0 0 0 0 0 0	Spri	ng stiffness 42.297Er 42.297Er	6		Aabiy	Hel
		Lay	. 48 11 12 13 14 18 18 17 18 14 18 18 18 18 18 18 18 18 18 18 18 18 18	8:89ae_Wal_D ts.(4) prosetiles protective - 30 protective - 30 protective - 30 protective - 30 protective - 30 protective - 30 2723) crete (2) Patch (2) P		~ ‡	Analy Spp	ysis category 30 Translation in Rotation about Hinge rotation Torsional warping Pore pressure ing stiffness distributis Stiffness	Y Z X Y Z	 • •<	Fixed 0 0 0 0 0 0 0 0 0 0 0 0 0	Spri	ng stiffness 42.297Er 42.297Er	6		Aabiy	Hel
		Lay	. 48 11 12 13 14 18 18 17 18 14 18 18 18 18 18 18 18 18 18 18 18 18 18	8:88ac, Wal _D ts (4) rossPies rossPies rossPies - 30.7 rossPies - 40.8 rossPies - 40.8 ro		~ ‡	Analy Spp	visis category 30 Translation in Rotation about Hinge rotation Torsional warping Pore pressure ting stiffness distributi Stiffness distributi	Y Z X Y Z	 • •<	Fixed 0 0 0 0 0 0 0 0 0 0 0 0 0	Spri	ng stiffness 42.297Er 42.297Er	6		Apply	Hel
		<		8:8:ex;Wal_D http://withub. http://w	d ad an an array of the second	- D Re	Anal	visis category 30 Translation in Rotation about Hinge rotation Torsional warping Pore pressure ing stiffness/unit area Stiffness/unit area	Y Z X Y Z	 • •<	Fixed 0 0 0 0 0 0 0 0 0 0 0 0 0	Spri	ng stiffness 42.297Er 42.297Er	6		Aabiy	Hel
		<		8:8:ex;Wal_D http://withub. http://w	d ad an an array of the second	- D Re	Anal	visis category 30 Translation in Rotation about Hinge rotation Torsional warping Pore pressure Stiffness distributi Stiffness/unit length Stiffness/unit length	Y Z X Y Z	 • •<	Fixed 0 0 0 0 0 0 0 0 0 0 0 0 0	Spri	ng stiffness 42.297Er 42.297Er	6		Aabiy	Hel
		<		8:89ae_Wal_D ts.(4) prosestless protection = 30.7 protection = 30.7 protection = 30.7 protection = 30.7 protection = 4.9 27237 crete (2) Patch (2) & 212.5 crete (2) Patch (2) & 23.5 molection = 4.9 Patch dava & 23.5 molection = 30.7 Street Structur Science Roof 6:Uner_Noof 6:Uner_Wal 7:SteetStructur 8:Insul_q1 9:Insul_q2 112.1 molection = 4.9 9:Insul_q2 112.1 molection = 4.9 9:Insul_q2 112.1 molection = 4.9 9:Insul_q2 112.1 molection = 4.9 9:Insul_q2 112.1 molection = 4.9 113.1 113.1 113.1 113.1 114	d ad an an array of the second	- D Re	Anal	visis category 30 Translation in Rotation about Hinge rotation Torsional warping Pore pressure ing stiffness/unit area Stiffness/unit area	Y Z X Y Z	 • •<	Fixed 0 0 0 0 0 0 0 0 0 0 0 0 0	Spri	ng stiffness 42.297Er 42.297Er	6 6 6 6 6 6 6 6 6 6 6 6 6 6 6 6 6 6 6 6		Aabiy	Hel

Fig 144 Support Definition in a 3D Shell Model

Close Cancel Apply Help

TEST CASE

For Help, press F1

By ticking 'Half only model', a symmetric half model is built.

LNG Tank - Static Analysis			×
Tank definition data Model tilename	Tnk1 ~	Half only model	
Saved model file path	C:\Users\overlaphatso\Documents\Lusas190\Projects\Example	- ·	
Element size (m) Analysis type	1.5		
O 2D Axisymmetric solid	● 3D Shell		

Fig 145 Option for Half Model

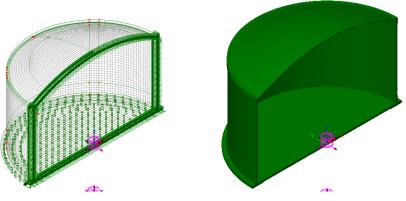


Fig 146 Half Symmetric Model

Loadings

Seventeen loadcases are defined in the model.

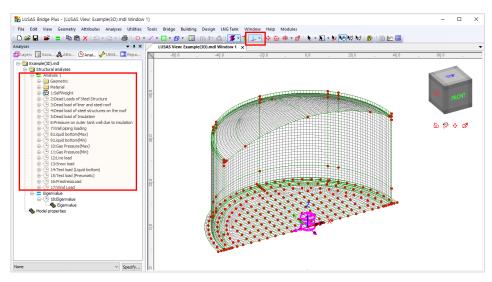
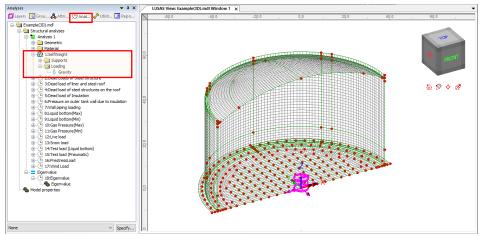


Fig 147 Loadcases Available in a 3D Shell Static Analysis Model







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.

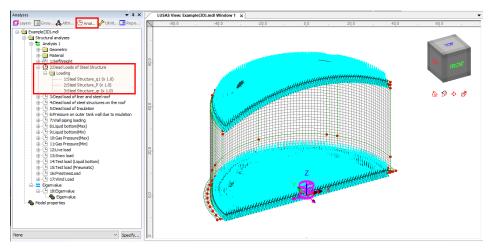


Fig 149 Dead Loads for Steel Structure in a 3D Shell Static Analysis Model

Dead load of liner and steel roof

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

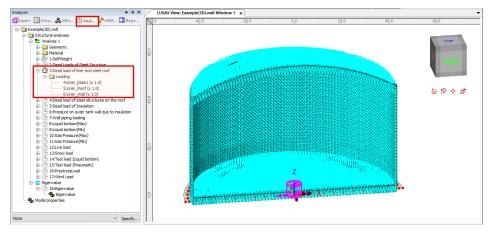


Fig 150 Dead Load of Liner and Steel Roof in a 3D Shell Static Analysis Model

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.

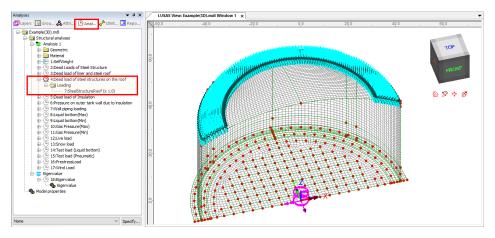


Fig 151 Dead Load of Steel Structures on the Roof in a 3D Shell Static Analysis Model

Dead load of Insulation

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

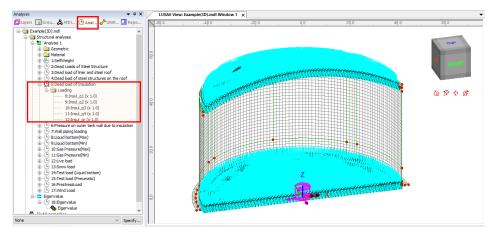


Fig 152 Dead Load of Insulation in a 3D Shell Static Analysis Model

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.

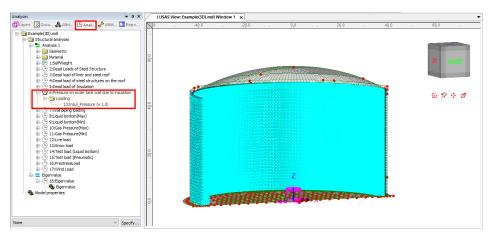


Fig 153 Insulation Pressure Load in a 3D Shell Static Analysis Model

Wall Piping Loading

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

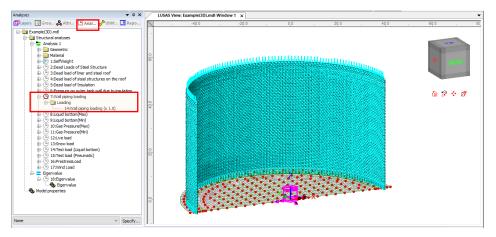


Fig 154 Wall piping loading in a 3D Shell Static Analysis Model

Liquid bottom (Max/Min)

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

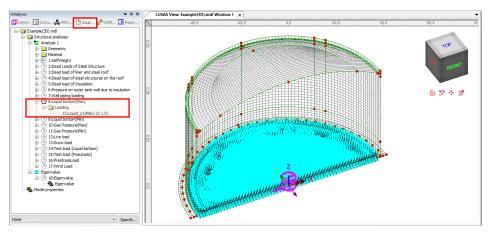


Fig 155 Liquid Bottom Loading in a 3D Shell Static Analysis Model

Gas Pressure(Max/Min)

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

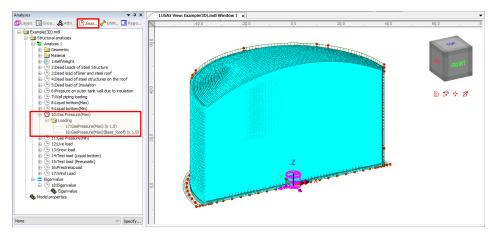


Fig 156 Gas Pressure Loading in a 3D Shell Static Analysis Model

Live load (Imposed Load on the roof)

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

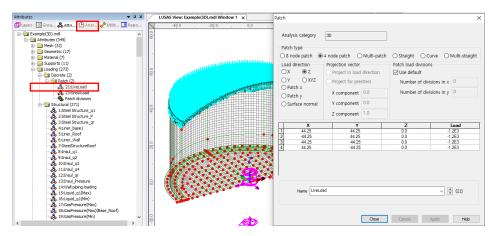


Fig 157 Live Load in a 3D Shell Static Analysis Model (Roof)

Snow load

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

Analysis cetegory 30 Patch type Conserved. Closed and Project in load direction Closed sold of Steel Structures on out ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel structures on the ratik wild as to inside on Closed sold of steel	Image: Second Control of Marketing Analysis a Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category Image: Second Control of Marketing Analysis Category <	Analyses 👻 🕈 🗙	LUSAS View: Example(3D).mdl Window 1 x	Patch	×
(0 · ① 15*tert tod (0 Peumatic) (0 Peumatic	Model properties	Complete Control of the set		Analysis category 30 Patch type 0 0 mode patch 0 4 node patch 0 mode patch 0 - 100 epiction vector 0 X 0 Z 0 Patch type Project for prestress 0 Patch y 0 Z 0 Surface normal X component 1 Component 1 0 2 Component 1 0 2 44.25 44.25 44.25 - 44.25	Straight Curve Multi-straight Patch load divisions Image: Constraint of the straight Image: Constraint of the straight of the s

Fig 158 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.

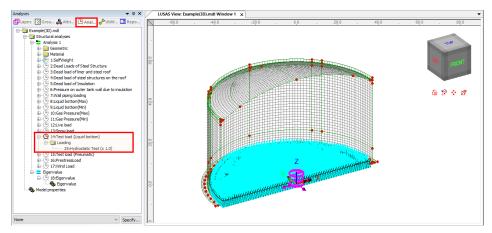


Fig 159 Test Load (Liquid Bottom) in a 3D Shell Static Analysis Model

Test load (Pneumatic)

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

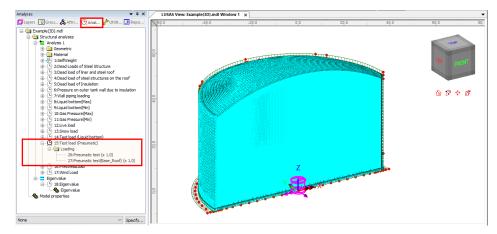


Fig 160 Test Load (Pneumatic) in a 3D Shell Static Analysis Model

Prestress Load

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

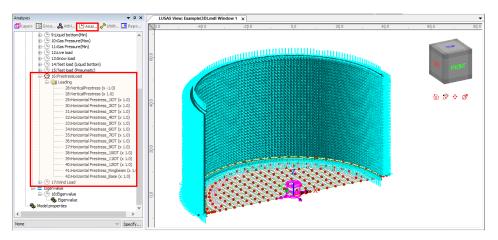


Fig 161 Prestress Load in a 3D Shell Static Analysis Model

Wind Load

Wind loading for the wall and roof is computed based on EN 1991-2. For the wall, separate loading datasets are defined for approximately each 1.0 m rise in height.

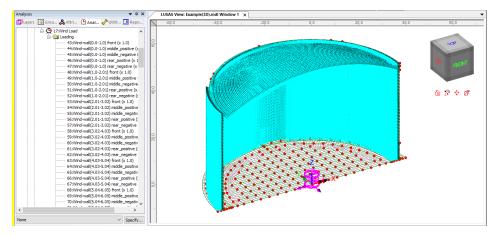


Fig 162 Wind Load in a 3D Shell Static Analysis Model

Viewing Results

Contours

The Layers is 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.

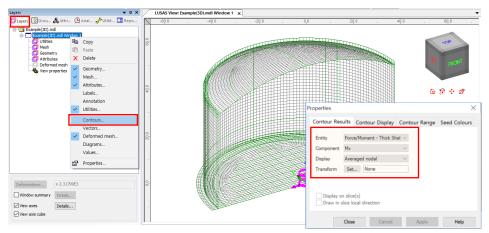


Fig 163 Selection for Contour Display in 3D Shell Model

Layers Groups A Attrib		-10.0 0.0
Layers Chooles Autob are Example(30).mdl Cam	Properties	
Deformations	Store element zwei intratations only if selected if selected <liif< td=""><td></td></liif<>	

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.

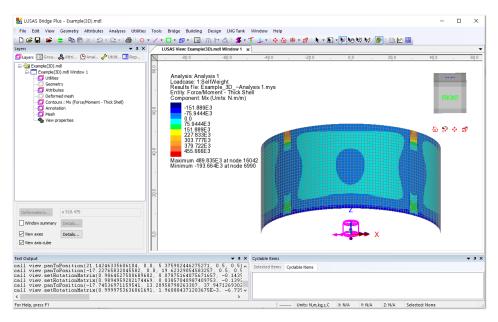


Fig 165 Mx Contour for Self Weight in a 3D Shell Model

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.

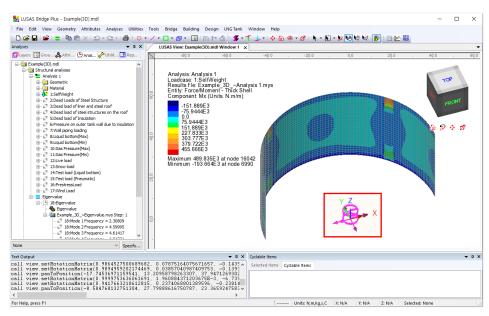
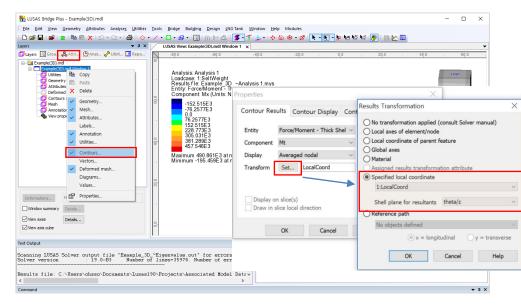
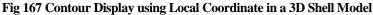


Fig 166 Local Coordinate in a 3D Shell Model

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.





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.

LUSAS Bridge Plus - Example(3D) mdl -	×
File Edit View Geometry Attributes Analyses Utilities Tools Bridge Building Design LNGTank Window Help	
: 🗅 🖆 🖬 📽 🗶 ' 요구요구'를 : ㅇㅋ/ㅋロㅋ@ㅋ' 📓 ' îî) 🕆 소!: 第ㅋギ 나ㅋ ☆ ûi � ㅋ♂ 🔭 + 第一級 🕸 (🏹 - 第三級 🕮	
Layers v 4 X LUSAS View: Example(3D).mdl Window 1 x	-
🗗 Layers 🔯 Grou. 🖧 Attri 🕀 Anat 🥓 Utilit 🖪 Rep	. 60.0
Analysis Analysis 1 Locations: 1:Sender 1:Sender 1: Control : Transformation: Locat Coords' Local Coord' Control : Transformation: Local Coord': Local Coord' Control : Transformation: Local Coord': Local Coord' Component: MI (Units N.m/m) Mean M	TOP FRONT
Properties ×	
< Contour Results Contour Display Contour Range Seed Colours	
Entity ForceMoment Component Mt Display Averaged nodel Transform Set LocaCoord	
Text • 4 × Cyclable Items	→ # ×
Cal 7, 37, 3471265302,4 Selected litems Cydable items Display on slice(s) 106756-76,753 Selected litems Cydable items Cal Draw in slice local direction 13956-62,92348 Feeder Cydable items Cal Traw in slice local direction 13956-8247868 Cydable items Cydable items	
cal Close Cancel Apply Heb	Transformed Mt

Fig 168 Mt Contour in a 3D Shell Model

107

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

Results Transformation 2	×
O Local axes of element	
O Local coordinate of parent feature	
O Global axes	
Specified local coordinate	
1:Spherical for ForceExtraction \checkmark	
Shell plane for resultants theta/phi ~	
O Reference path	
No objects defined $$	
$ \textcircled{\tiny{o}} x = \text{longitudinal} \qquad \bigcirc y = \text{transverse} $	
Name ResultTransformation_Roof \checkmark (1)	
Close Cancel Apply Help	

- TVP

	- Apl
Results Transformation	×
 Local axes of element Local coordinate of parent feature Global axes Specified local coordinate 	2
3:LocalCoord V	
Shell plane for resultants $% = -\frac{1}{2} \left(\frac{1}{2} - \frac{1}{2} \right) \left(\frac{1}{2} -$	
○ Reference path	
No objects defined \checkmark	
\bigcirc x = longitudinal \bigcirc y = transverse	
Name ResultTransformation_Wall v (2)	
Close Cancel Apply Help	

Results Transformation X	
O Local axes of element	2
O Local coordinate of parent feature	
○ Global axes	
Specified local coordinate	
3:LocalCoord \checkmark	
Shell plane for resultants r/theta ~	
O Reference path	
No objects defined $\qquad \qquad \lor$	
(i) x = longitudinal \bigcirc y = transverse	
Name ResultTransformation_BaseSlab v (3)	
OK Cancel Apply Help	

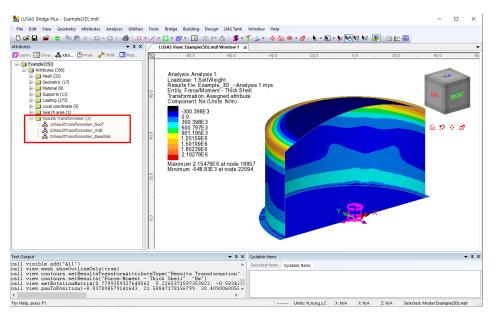


Fig 169 Results Transformation in a 3D Shell Model

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

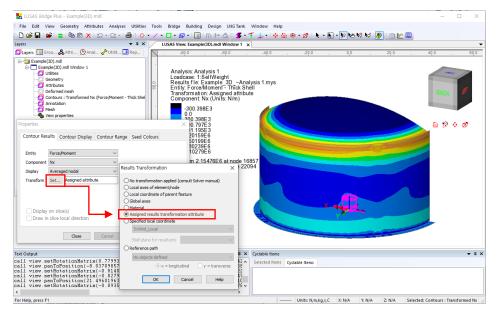


Fig 170 Contour Display using Results Transformation in a 3D Shell Model

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.

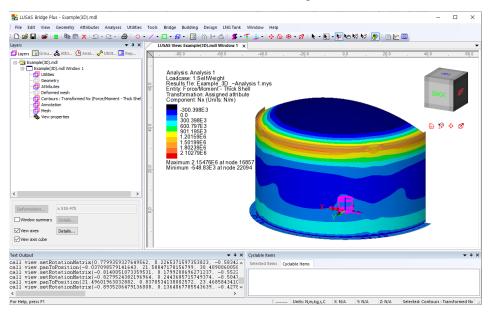


Fig 171 Nx Contours in a 3D Shell Model

Examples – User Inputs

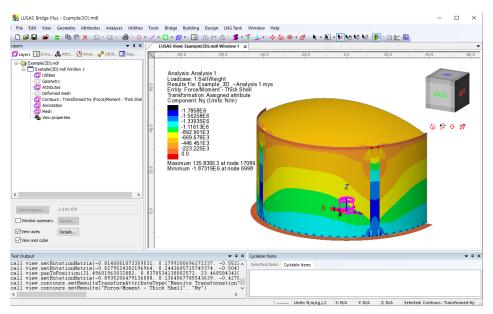
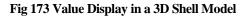


Fig 172 Ny Contours in a 3D Shell Model

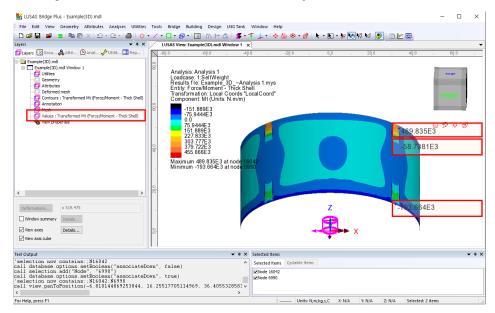
Values

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

🔀 LUSAS Bridge Plus - Example(3D).mdl – 🗆 🗙	<
: File Edit View Geometry Attributes Analyses Utilities Tools Bridge Building Design LNGTank Window Help	
□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□	
	•
📴 Layers 🕄 Grou. 歳 Attri 🕲 Anal 🌽 Utilit	-
Image: State in the series of the series	
Location Averaged nodel V Properties X	
Transform Set [LocalCoord	
Value Results Values Display Smbols Mana Sindo Mana 10.0 % Fabre detais Z	
Display on slice(s) Significant figures 6	
Observed places Image: Concell places Observed places Image: Concell places Image: Concell places Image: Concell places	
Test Output Pen # 19 🛟 Choose pen 0.0 °	×
call view panToPosition (-13.545551088000) call view values aetResultSTensformIocal Come Cancel Apply Heb call view values aetResultSTensformIocal Come Cancel Apply Heb call view values showOnlyIfSelected(true) v call view values showOnlyIfSelected(true) v	
C C 2 Units: N.m.kg.s,C X: N/A Y: N/A 2: N/A Selected: Values: Transformed Mt (FC	



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





Graph through 2D

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

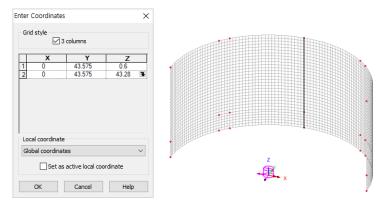


Fig 175 Line for Slicing Results in a 3D Shell Model

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

Graph Through 2D X	Loadcases and Extent	×
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 1449 ✓ Project line Normal to screen ✓ By selected surface ✓ At location of existing graph ✓	Loadcases © 1.5elfWeight Active All Specified Select Create new window for each loadcase Extent Visible model	
OK Cancel Help	< 뒤로(B) 다음(N) > 취소 도움말	

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

Results	component	Resultant effects from 2D model Resultant effects from slice	Display	X scale
Entity Component	Force/Moment v		Title Graph for Mt X Wall height Y Mt	Automatic O Manual min 0.0 max 1.0 Use logarithmic scale
	te distance as angle X 0.0		Show grid Show symbols Corner labels Include existing graphs	Scale factor 1.0 Y scale (a) Automatic Manual min 0.0 max 1.0 Use logarithmic scale
Width for a	z 0.0	Y	Name Graph for Mt for Self Weight	Scale factor 1.0

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.

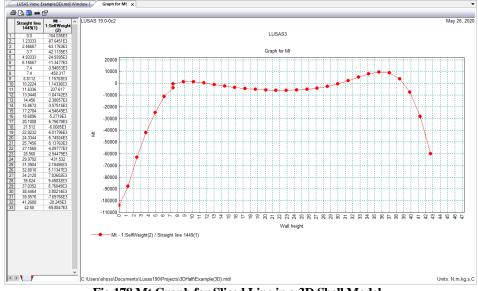


Fig 178 Mt Graph for Sliced Line in a 3D Shell Model

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 > Export Forces to Excel (3D).

- **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.

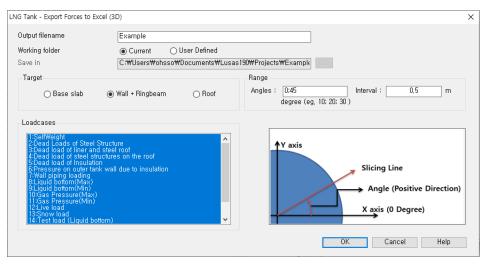
LNG Tank - Export Forces to Ex	(cel (3D)			Х
Output filename Working folder Save in	0	Jser Defined Documents₩Lus	as 190\Projects\Example	
Target O Base slab	● Wall + Ringbeam	⊖ Roof	Range Angles : 20 degree (eg. 10; 20; 30)	
Loadcases 1:SelfWeight 2:Dead Loads of Steel 1 3:Dead load of liner ann 4:Dead load of liner ann 5:Dead load of liner ann 6:Liquid bottom(Max) 9:Liquid bottom(Max) 10:Gas Pressure(Max) 10:Gas Pressure(Max) 10:Gas Pressure(Max) 12:Gae Pressure(Max) 14:Test load (Liquid botta)	d steel roof uctures on the roof n nk wall due to insulation		Slicing Line Angle (Positive Direction) X axis (0 Degree) OK Cancel Help	

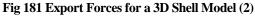
Fig 179 Export Forces for 3D Shell Model (1)

ype	Momen	t	Sign Convention							
ocation	Wall_Ringt	eam	(+): Inner side Tension	n						
lirection	Ноор									
init	kN-m/r	n								
oadCase fax (kN-m/m) fin (kN-m/m)	Max 20. -106.	40.00	Moment of Wal	ll_Ringbeam (Hoop)	_	40.00	Mome	nt of Wall_Ri	ngbeam (Hoop)	
Distance(m) 0.00 0.50 1.00	Max -106 -97 -88	20.00				20.00				
1.50	-97 -88 -77 -67 -56 -47 -38 -30 -30 -22	-20.00	5.00 10.00 15.00 20.00 2	25.00 30.00 35.00 40.00 45.00 5	0.00	0.00	10.00	20.00 \$0.00	40,00 50.00	
2.50	-56	-20.00				-20.00	1			
3.50	-38	-40.00			Max	-40.00	1			20 deg: SelfWeight
4.50	-22				Min	- W	1			20 deg: Serweight
5.00	-15	-60.00				-60.00	1			
6.00	-9	-80.00		V		-80.00			V	
6.50	0			v					V	
7.00	4	-100.00				-100.00				
8.00	9					-				
8.50		-120.00	Dist	tance(m)		-120.00		Distance(m)		
9.00	13							ustance(m)		
10.00	14.52	14.52	14.52							
10.50	15.00	15.00	15.00							
11.00	15.29	15.29	15.29							
11.50	15.57	15.57	15.57							
12.50	15.00	15.08	15.72							
13.00	15.77	15.77	15.77							
13.50	15.70	15.70	15.70							
14.00	15.62	15.62	15.62 15.53							
14.50	15.53									

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.





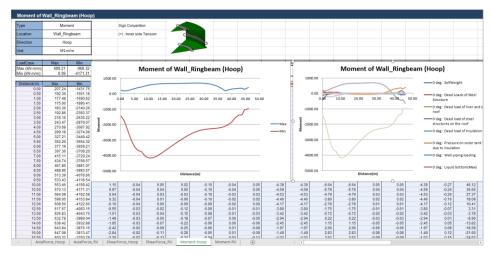


Fig 182 Section Force Spreadsheet for All Loadcases

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

3D Shell Eigenvalue Analysis

An Eigenvalue Analysis is created as a part of a creating a **3D Shell Static Analysis Model**.

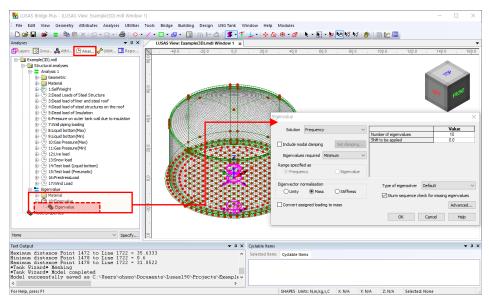


Fig 183 Eigenvalue Analysis in a 3D Shell Model

If the option to '**Include non-structural masses**' is checked, the equivalent mass is computed to include the non-structural masses, and the mass computation summary is provided in the working folder with the filename of **<model** name>_EigenvalueAnalysis.xlsx.

Examples – User Inputs

LNG Tank - Static Analysis				×
Tank definition data Model filename Saved model file path Element size (m) Analysis type	Tnk1 Example C:\Users\ohsso\U 2,5	Documents₩LUS	× AS190WProjectsWExampl	Half only model
◯ 2D Axisymmetric solid			● 3D Shell	
Wind load (EN1991, 1, 4, 2005)			Buttress	
Basic wind velocity Roughness length Minimum height Orography factor Terrain factor Turbulence factor Air density Soil height above slab bottom	37.5 3.0E-3 1.0 1.0 0.156 1.0 1.25 0.9	(m/s) (m) (m) (kg/m^3) (m)	Number of buttress Extruded thickness Buttress width Eigenvalue Number of eigenvalu	

Fig 184 Dialog for a 3D Shell Eigenvalue Analysis

ummary of Mas	s Calculation				
DIMENSION					
Component	Dimension(m)				
Inner Tank Radius	42.1				
Tank Height	40.06				
LNG Height	38.92				
SUMMARY FOR I	VIASS Volume	Unit mass	Structural mass	Total mass	Equivalent u mass
	m³	kg/m³	kg	kg	kg/m ³
Roof	3,950	2,500	9,875,937	11,985,937	3,03
Ringbeam(upper)	524	2,500	1,310,993	1,310,993	2,50
Ringbeam(lower)	463	2,500	1,156,758	1,156,758	2,50
Wall & Buttress	9,976	2,500	24,940,428	25,764,428	2,5
BaseSlab	8,719	2,500	21,797,085	24,925,085	2,8
LNG	216,714	480	104,022,703	104,022,703	4
Inner Tank	316	7,850	2,479,105	2,799,105	8,8

Fig 185 Mass Summary for an Eigenvalue Analysis

The computed equivalent unit mass (the density) for each component are defined separately and used for eigenvalue analysis.

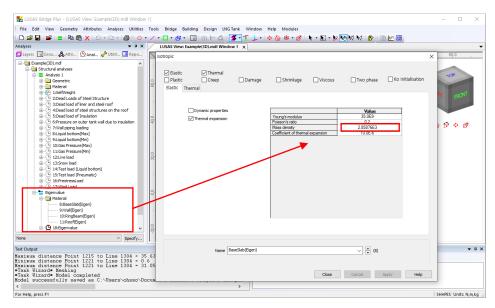


Fig 186 Mass for Eigenvalue Analysis

Viewing Results

Mode Shapes

The Layers the treeview in the LUSAS Modeller user interface controls what is displayed in the View window. Mode shapes can be observed by adding the **Deformed mesh** layer to the Layers TreeView.

Layers	▲ # >	0.0	-80.0	-60.0	-40.0	-20.0	0.0	20.0	40.0	60
	Analy Reports Groups & Utilities	0	80.0	-60.0	40.0	200	0,0	20.0	40.0	. 60
Window summary	Values de Properties 177.157E3 Details Details	40.0 -20.0					•		•	

Fig 187 Mode Shape from an Eigenvalue Analysis

Natural Frequencies

By selecting the menu item **Utilities>Print Results Wizard...** the Natural Frequencies and Participation Factors will be listed.

		 Components 	 Eigenvalues
Results			
Units	Model units	~	
Loadcases	18:Eigenvalue	~	
Particip Mass p	alues (Frequency) vation factors articipation factors ass participation factors		Predion
Display		Defaults	Show trailing zeros
☑ Display ☑ Save in Loadcases		Defaults	O Decimal places
Save in Loadcases Available	treeview		Show trailing zeros
Save in Loadcases Available		Defaults	Show trailing zeros Threshold value N/A

LU	SAS View: Exam	nple(3D).mdl Wind	ow 1 Sum m	ass participation fa	ctors x				
8	i 🖪 🛃 🖷	QQ							
	Mode 🔺	Sum Mass X	Sum Mass Y	Sum Mass Z	Sum Mass THX	Sum Mass THY	Sum Mass THZ	Frequency	Period
1	1	0.915901	96.8147E-18	0.169431E-12	2.34659E-15	4.19417E-3	0.039255	2.30809	0.433258
2	2	0.919452	4.13314E-15	1.72549E-12	42.0454E-15	5.35235E-3	0.039372	4.59995	0.217393
3	3	0.919452	0.385728E-3	1.76484E-12	45.1659E-9	5.35235E-3	0.039372	4.61417	0.216724
4	4	0.993637	0.385728E-3	1.53175E-9	45.2205E-9	0.062865	0.0398599	4.84321	0.206474
5	5	0.993637	8.74012E-3	0.458632	0.0172653	0.062865	0.0398599	4.90317	0.20395
6	6	0.994547	8.74012E-3	0.458632	0.0172653	0.0642204	0.0400356	5.06408	0.197469
7	7	0.994547	0.0155991	0.540607	0.0203377	0.0642204	0.0400356	5.2046	0.192138
8	8	0.994547	0.0158892	0.545147	0.0205245	0.0642204	0.0400356	5.77654	0.173114
9	9	0.994556	0.0158892	0.545147	0.0205245	0.064351	0.0440971	6.23003	0.160513
10	10	0.996628	0.0158892	0.545147	0.0205245	0.0662597	0.0441652	6.77019	0.147706

Fig 188 Natural Frequencies from an Eigenvalue Analysis

2D Beam-Stick FSSI Seismic Analysis for Horizontal

Actions

This example is based on the user inputs discussed in the section titled *Examples – User Inputs : 2D Beam-Stick FSSI Seismic Analysis for Horizontal Actions*

User Inputs

The required user inputs for this model are shown below. In addition to Structural Definition and Material Properties, **Seismic Input** should be defined.

Insulation Data

The thicknesses of the wall insulation and of the base insulation are used to create the 'Connection Part' – a beam element which connects the concrete tank and the inner tank.

Inner Tank Properties

Thickness variation, material properties, unit mass and inner side radius of inner tank must be defined.

nk Definition										
] Include pile data	🖂 Include i	insulation	🖂 Include	a seismic data						
ructure Definition Material Pr	roperties Boundary C	Conditions Loadin	ng Prestress Load	Pile Arrangement	(3D) Seismic in	put 1 Seismic inpu	12			
nner Tank Properties Non-St	ructural Masses									
Liquid mass										
Liquid unit mass	480,0	(kg/m^3)	Inner tank inside	radius	42, 1	(m)		Thickness 6	Height 6	
								Thickness 5	Height 5	
Inner tank geometric propert	ies 2	3	4 5	6	7	8		Thickness 4	Height 4	
Thickness(m) 0.0361 Height(m) 3.08	0.0361 27.0	0,012	0.01 0.0 6.12 0.0	0,0	0.0	0.0		Thickness 3	Height 3	
Theightenity 3,00	21,0	3,00	0,12 0,0	0,0	0,0	0,0		Thickness 2	Height 2	
								Thickness 1	Height 1	
 Inner tank material properties 	5						>			
Inner tank material properties		sons ratio Mass (kg/m	Density Coefficient thermal expansion	of Thermal Conductivity /C) (J/m.s.C)	Heat Capacity (J/m^3,C)	Description	>			
Inner tank material properties	stic Modulus Pois: N/m^2) (v)	sons ratio Mass (kg/m 7,85E3	Density Coefficient *3) expansion(10,0E-6	Conductivity	Heat Capacity (J/m [*] 3,C) 1,968E6	Description Inner Tank	>			
Inner tank material properties Ela (E,	stic Modulus Pois: N/m^2) (v)		 3) thermal expansion(of Thermal Conductivity /C) (J/m.s.C) 2,0			>			
Inner tank material properties Ela (E,	stic Modulus Pois: N/m*2) (v) DE9 0,2		 3) thermal expansion(of Thermal Conductivity (C) (J/m.s.C) 2.0			<u>,</u>			
Inner tank material properties Ela (E. Inner Tank 200,	stic Modulus Pois: N/m*2) (v) DE9 0,2		 3) thermal expansion(of Thermal Conductivity (U/m.s.C) 2,0			>			
Inner tank material properties Ela (E. Inner Tank 200,	stic Modulus Pois: N/m*2) (v) DE9 0,2		 3) thermal expansion(of Thermal Conductivity (U/m.s.C) 2.0			>			
Inner tank material properties	stic Modulus Pois: N/m*2) (v) DE9 0,2		10,0E-6	of Thermal Conductivity (J/C) 2.0			>			

Fig 189 User Inputs 1 for Seismic Analysis

Non-Structural Masses

Loadings other than self weight can be considered as additional masses in the seismic analysis.

k Definition				
] Include pile data	🛛 Include insulation		🖂 Includ	e seismic data
ructure Definition Material Properties	Boundary Conditions	Loading	Prestress Load	Pile Arrangement (3D) Seismic input
nner Tank Properties Non-Structural Ma	sses			
Roof Ring Beam Wall Base Sla	b Inner Steel Tank			
Descriptions			Mass(kg)	
Roof Liner + steel Roof Structure			1,4E6	
Suspended deck & insulation of the su	spended ceiling		135,0E3	
Roof nozzles			42,0E3	
Roof platform			400,0E3	
Roof pump & crane			30,0E3	
Roof piping and support			103,0E3	
Others			0,0	
Total			2,11E6	

Fig 190 User Inputs 2 for Seismic Analysis

Pile Properties

The piles are modelled using a series of beam elements, and the geometric and material properties of the pile group as a whole needs to be defined.

k Definition											
] Include pile data		⊘ Inclu	de insulation	. ⊡ In	clude seismic d	lata				_	
tructure Definition	Material Prop	erties B	oundary Conditions	s Loading	Prestress Load	Pile Arrangeme	ent (3D)	Seismic i	input 1 Seismic inpu	it 2	
Pile Properties 🔓	oil Properties										
Geometric prope	rties									_	
	Area (m^2)	Shear Area(m^2)	Moment of inertia(m^4) Pile L	ength(m) Ve Sti	ertical iffness(M	N/m)	Rotational Stiffness (MN,m/rad)		
pile	617,229		540, 137	297,064E3	58,8	225	5,9233E3		225,9233E3	1	
Material propertie				0 11 1	Thermal						
	Elastic	Poissons	Mass	Coefficient	Conductivity	Heat Capacity	Deen	detien.			
	Modulus (E, N/m^2)	Poissons ratio (v)	Density (kg/m^3)	of thermal expansion(/	Conductivity (J/m,s,C)	Heat Capacity (J/m^3,C)		ription			
Pile	Modulus (E, N/m^2)	Poissons ratio (v) 0,3	Density (kg/m^3)	of thermal	Conductivity	Heat Capacity (J/m^3,C) 0,0	Desci Pile	ription			
Pile Set zero	Modulus (E, N/m^2)	ratio (v)	Density (kg/m^3)	of thermal expansion(/	Conductivity (J/m,s,C)			ription			
	Modulus (E, N/m^2) 33,0E9	ratio (v)	Density (kg/m^3)	of thermal expansion(// 10.0E-6	Conductivity (J/m,s,C)				(new)		

Fig 191 User Inputs 3 for Seismic Analysis

Soil Properties

The soil properties for horizontal stiffness are used as the boundary condition for pile.

≥ Include	e pile data		🖂 Include insula	ion 🖂 Include seismic data					
tructure (Definition Mater	ial Properties	Boundary Conditi	ons Loading Prestress Load Pile Arrangement (3D)	Seismic input 1 🚦	Seismic input 2			
Pile Prop	enties Soil Prop	erties							
Layer No,	Bottom EL(m)	Thickness of Layer(m)	kh(MN/m/m)	Description(Soil Profile)	^		•		
Pile He	0,0	0.0	0.0	Pile Head				• Î	
1	-2,0	2,0	14,92E3	Backfill		Layer 1		Layer Thickness	
2	-4.0	2,0	25, 769E3	Backfill		Layer 2			
3	-6.0	2,0	22, 39E3	Backfill					
4	-8.0	2,0	21,549E3	Backfill		Layer 3			
5	-10,0	2,0	23,622E3	Backfill					
6	-12,0	2,0	32, 373E3	Silty Sand1		Layer 4			
7	-14.0	2.0	27.003E3	Silty Sand1					
8	-16,0	2,0	24,337E3	Silty Sand1		Layer 5			
9	-18,0	2,0	32,45E3	Silty Sand2					
10	-20,0	2,0	27,263E3	Silty Sand2		Layer 6			
11	-22.0	2.0	32,509E3	Silty Sand2					
12	-24.0	2,0	27.78E3	Silty Sand2					
13	-26,0	2,0	30, 789E3	Silty Sand2					
14	-28,0	2,0	35,822E3	Silty Sand3		Add			
15	-30,0	2,0	36, 329E 3	SSilty Sand3		Add			
16	-32,0	2,0	36, 336E 3	Silty Sand3		Remove			
17	-34,0	2,0	35,847E3	Silty Sand3		Set defaults			
18	-36,0	2,0	36, 348E 3	Silty Sand3					
19	-38,0	2,0	36, 355E 3	Silty Sand3	~	Set zero			

Fig 192 User Inputs 4 for Seismic Analysis

Seismic Analysis Wizard

The user dialog is available from LNG Tank>Seismic Analysis Wizard as shown in [Fig 193].

- Enter the model file name and select Design Code to EN1998-4, model type of Beam-Stick Horizontal.
- Set the number of buttress to 0 (zero).
- The required damping ratio for the design code can be defined for each of members.
- The 1st and 2nd mode frequency of the tank can be obtained from a separate eigenvalue analysis. This is used together with the damping ratio for computing damping coefficients for material properties of each member.

LNG Tank - Seismic Analysi	S	×
Tank definition data Model filename Saved model file path Analysis type	Tnk1 Example C:\Users\obsco\Documents\USAS190\Projects'	
Design code O Beam-stick hori: (Excluding base pr (Beam-Stick Horizont		
Critical damping / frequ	ency	
	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	
Pile	4.0	
LNG impulsive	3.0	
LNG convective	0,5	
Ground	5.0	
Buttress		
Number of buttress	4 ~	
Extruded thickness	1.0 (m) Buttress width 5.0 (m)	
	OK Cancel Help	

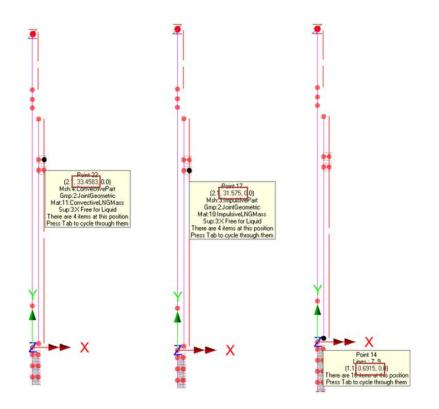
Fig 193 User Dialog for Seismic Analysis Wizard

Mesh

For modelling details see the section titled 2D Beam-Stick FSSI Seismic Analysis.

The properties used for the beam-stick model are summarized in the spreadsheet **Example_Seismic_Report(HorizontalIBP).xlsx** located in the current working folder.

The locations of convective and impulsive masses are defined as shown below. A computation summary is presented in the saved spreadsheet. The height of **convective mass** is at **32.77m** above the inner tank bottom (Y coordinate = 0.6915), and the **impulsive mass** is at **30.88m.** (Hence the Y coordinate in the model is 33.4583 and 31.575 respectively)



CALCULATED PRO	PERTIES FOR HO	RIZONT		IODEL						
1) LNG Mass & Heig	ht									
IBP (Including Base Pressure)										
Component	H/R	m _(c.i) /m h' _(c.i) /H			mass	Lever arm height				
					mc(mi), Kg	hc(hi), m				
LNG Convective	0.924	(0.49 0.84		50,527,854	32.77				
LNG Impulsive	0.924	0.51		0.79	53,494,849	30.88				
	EBI	P (Exluding	Base	Pressure)						
Component	H/R	m _(c,i) /n	n	h _(c,i) /H	mass	Lever arm height				
					mc(mi), Kg	hc(hi), m				
LNG Convective	0.924	(0.49	0.60	50,527,854	23.5				
LNG Impulsive 0.924		0.51		0.41	53,494,849	16.13				
Summary	Verifications-ACI3	50.3	Verifi	cations-EN1998-4	4 (+)					

Fig 194 Location of liquid masses in a Beam-Stick Model

Tip

A reference path was used to define the varying sections. These can be hidden as illustrated below.

Utilities		▼ ‡ × 📻	-
🔁 Layers 🛛 🛞 Groups 🖧 Att	trib 🕒 Analy 🦨 Utiliti 🛄	Repo	
Example EN1998 Hori: Example EN1998 Hori: Delibites (4) Comparison Specific Address Specific Address	ctra (1) CLASS (1)		
	Rename Delete		
	Edit Create Geometry		
	Select Deselect		4
~	Visualise Visualise at Points	X A	2→→ ×

Fig 195 Hide reference path in Beam-Stick Model

Geometric Properties

Roof

The elements defined for the roof represent the region of the tank as shown below.

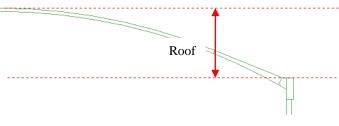
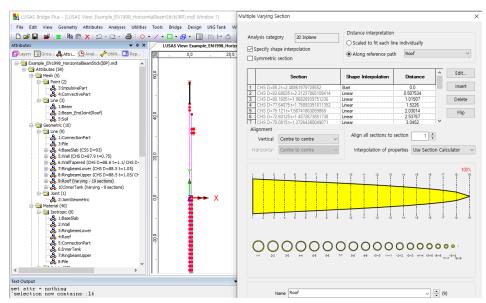


Fig 196 Roof in Beam-Stick Model

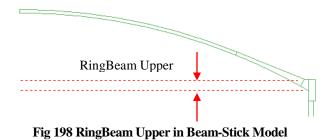


Varying Section properties are defined as shown below.

Fig 197 Geometric Properties for the Roof in a Beam-Stick Model

RingBeam Upper

The elements for RingBeam Upper represent the region of the tank as shown below.



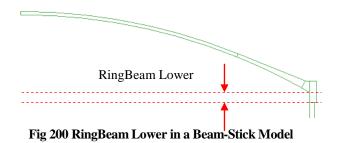
Varying Section properties are defined as shown below.

- & 3:ImpulsivePart	Geometric Line		×
	Analysis category 2D Inplane Definition Inform Library Rotation about centroid Mirrored about axis Define Properties	Properties for and 1 of line User Sections V Local V OHS D=88.5 t= 1.05 V I V V V V V Z	Properties for end 2 of line User sections Cocal CPIS D=88.5 t=4.13861979729552
-& 6:InnerTank -& 7:RingbeamUpper		Value	Value
- & 8:Pile	Cross sectional area (A)	288.469	1.09685E3
m #a Jointe (21)	Second moment of area about z axis (Izz)	275.798E3	978.116E3
< > >	Effective shear area in y direction (Asy)	144.234	550.625
ext Output	Eccentricity in y direction (ey)	0.0	0.0

Fig 199 Geometric Properties for RingBeam Upper in a Beam-Stick Model

RingBeam Lower

The elements for RingBeam Lower represent the region shown below.



Section properties are defined as shown below, with an outer diameter of (43.2 + 1.05) * 2 = 88.5m, and a thickness of 1.05m.

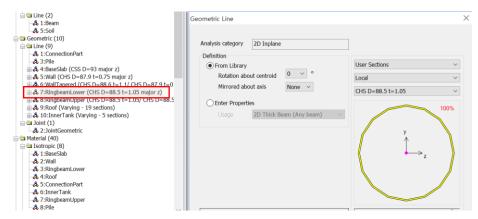


Fig 201 Geometric Properties for RingBeam Lower in a Beam-Stick Model

Wall

The elements for the Wall represent the extent of the wall with a constant thickness.

Section properties are defined as below, with outer diameter of (43.2 + 0.75) * 2 = **87.9m** and wall thickness of **0.75m**.

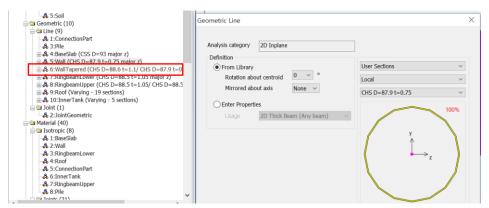


Fig 202 Geometric Properties for Wall in a Beam-Stick Model

Wall Tapered

The elements for Wall Tapered represent the extent of the wall having a varying thickness. Varying section properties are defined as shown below.

-& 5:Soil	p - 1		
= Geometric (10)	Geometric Line		×
ia Geometric (10)			
-& 1:ConnectionPart			
-& 3:Pile	Analysis category 2D Inplane		
	zo inplane		
4:BaseSlab (CSS D=93 major z)	Definition	Properties for end 1 of line	Properties for end 2 of line
& 5:Wall (CHS D=87.9 t=0.75 major z)	From Library	User Sections ~	User Sections ~
6:WallTapered (CHS D=88.6 t=1.1/ CHS D=87.9 t=0.	Rotation about centroid 0 ~ °		
& 7:RingbeamLower (CHS D=88.5 t=1.05 major z)	Kotauon about centroid	Local ~	Local ~
8:RingbeamUpper (CHS D=88.5 t=1.05/ CHS D=88.5	Mirrored about axis None V	CHS D=88.6 t=1.1	CHS D=87.9 t=0.75
& 9:Roof (Varying - 19 sections) & 4 10:InnerTank (Varying - 5 sections)		CHS D=88.0 t=1.1	CHS D=87.9 t=0.75
	Enter Properties		
B→ Joint (1) & 2:JointGeometric		100%	100%
	Usage 2D Thick Beam (Any beam) ~		
😑 🖻 Material (40)			
😑 🖴 Isotropic (8)	Alignment	y y	у
-& 1:BaseSlab	Alian end 2 to end 1	↑	^ ∧
-& 2:Wall			
-& 3:RingbeamLower	Align end 1 to end 2	\rightarrow_z	\rightarrow_{1}
-& 4:Roof	Vertical Centre to centre v 0.0		
-& 5:ConnectionPart			
-& 6:InnerTank	Horizontal Centre to centre V 0.0		
-& 7:RingbeamUpper			
-& 8:Pile			
in the Inlinte (21)	Interpolation of properties Enhanced ~		

Fig 203 Geometric Properties for Wall Tapered in Beam-Stick Model

Buttresses

Buttresses can be added as structural members by specifying the number of buttresses and their dimensions in the dialog.

Buttress					
Number of buttress	0	\sim			
Extruded Thickness	0	(m)	Buttress Width	0	(m)

As the inclusion of buttresses makes the model non-axisymmetric, this is considered in the model by increasing the thickness of wall and ringbeam to the equivalent thickness.

BaseSlab

The length of the line modelling the slab is equal to the slab thickness. If the annual part of slab has different thickness, an average thickness is computed and used.

From calculations the slab is modelled with vertical beam elements, having a circular section with diameter of 93m (section area of **6,792.91** m^2), and total length of **1.2835** m.

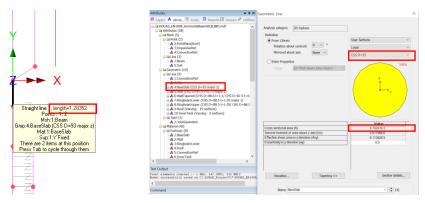


Fig 204 Section Properties for Base Slab in a Beam-Stick Model

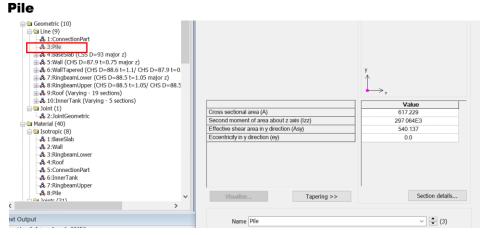


Fig 205 Section Properties for Pile in a Beam-Stick Model

Inner Tank

The varying thickness of the inner tank is defined as follows:

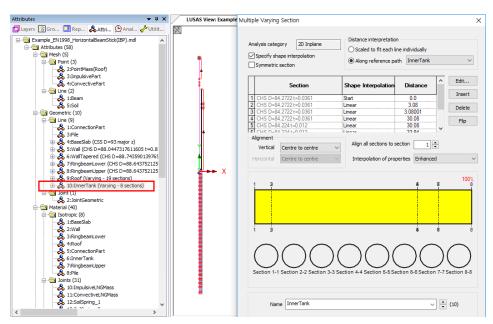


Fig 206 Section Properties for 'Inner Tank' in a Beam-Stick Model

The inner tank is modelled at X=1.0639 (not in the centre, but at a distance equal to the thickness of the wall insulation), hence an eccentricity in the y direction (ey) of -1.0639 was used in the geometric property definition. The properties for the bottom of the inner tank are as shown below.

Multiple Varying Section		22
Analysis category 20 Inplane	Istance interp Scaled to fit Along referer Rotation about centroid	User Sections
1 CHS D=84.27221=0.0361 3 2 CHS D=84.27221=0.0361 B 3 CHS D=84.27221=0.0361 Lot 4 CHS D=84.27221=0.0361 Lot 5 CHS D=84.27221=0.0361 Lot 6 CHS D=84.27221=0.0361 Lot 6 CHS D=84.27221=0.0361 Lot 7 CHS D=84.27221=0.0361 Lot 8 CHS D=84.27241=0.012 Lot	ar	Error: Not found *
Section 1-1 Section 2-2 Section 3-3 Section	Cost sectoral area (a) Cost sectoral area (b) Cost aectoral area (b) Cost aectoral dress about Effective these area in y direct Eccentricity in y direction (ey) 4.4 Section 5	
Name InnerTank	Visualse	Section details OK Cancel Help

Fig 207 Section Properties for Inner Tank in a Beam-Stick Model

Material Properties

Roof

The Input data was used for elastic modulus, Poisson's ratio, and mass density.

Rayleigh damping constants are computed and assigned as below.

Attributes 🗸 🖛 🛪	X		00.0 -8	0.0	-60.0	-40.0	-20.0	0.0	20.0	40.0	60.0 8
🗗 La 🝳 An 🖪 Re 🔄 Gr 🥓 Ut Å At	•	Isotropic									×
	0 60.0	Plastic	Cre	eep	Damage	Sł	rinkage	Viscous	Two ph	ase 🗌 Ko	o Initialisation
- & 1:BaseSlab - & 2:Wall - & 3:Rinobeami.ower - & 4:Roof - & 5:ConnectionWart - & 6:InnerTank - & 7:RingbeamUpper - & 8:Pile	20.0 40.0	✓ Dyn □ The	amic propertie rmal expansior		[ratio sity leigh damp	ing constant	(Value 35.0E9 0.2 2.5E3 0.51092 9032E-3	
	-20.0 0.0										

Fig 208 Material Properties for Roof in Beam-Stick Model

The wizard adds a joint element to the end of the line modelling the top of the roof, as shown below. The amount of additional mass is as per user input. (see [Fig 190].)

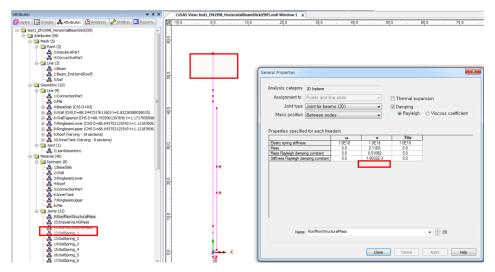


Fig 209 Non-structural mass on the Roof in a Beam-Stick Model

RingBeam Upper / RingBeam Lower

The Input data was used for elastic modulus, Poisson's ratio.

Attributes 👻 🕈 🕇	< 🖓	120.0	-100.0	-80.0	-60.0	-40.0	-20.0	0.0	20.0	40.0	60.0	
🕼 La 🖄 An 🖪 Re 🗟 Gr 🗲 Ut Å At		Isotropic										\times
& 10:InnerTank (Varying - 5 sections) Joint (1) & 2:JointGeometric Material (40) Sotropic (8)	09	. Dela		Creep	Damage	S	hrinkage	Viscous	Two pł	ase 🗌 K	o Initialisation	3
-& 1:BaseSlab	40.0	Elasti										
& 2:Wall & 3:RingbeamLower			Dynamic	properties						Value		
-& 4:Roof				mal expansion		Young's	modulus			35.0E9		
-& 5:ConnectionPart	20.0		merina			Poisson's	ratio			0.2		
- 🖧 6:Inner Lank	N	1				Mass der	nsity			2.5E3		
- 🖧 7:RingbeamUpper						Mass Ray	yleigh dam	ping constant		0.25546		
-& 8:Pile						Stiffness	Rayleigh d	amping constant	0.9	51599E-3		
⇒ Joints (31)	0.0											
-& 14:SoilSpring_3 -& 15:SoilSpring_4 -& 16:SoilSpring_5 -& 17:SoilSpring_6	-20.0											

Fig 210 Material Properties for Ringbeam in a Beam-Stick Model

Wall

The Input data was used for elastic modulus and Poisson's ratio. The Rayleigh damping constants are computed as below.

Attributes 🗸 🖛 🛪	Y	120.0	-100.0	-80.0	-60.0	-40.0	-20.0	0.0	20.0	40.0	60.0	_
🕼 La 😟 An 🖪 Re 🖾 Gr 🥓 Ut Å At		- Isotropic										×
⊕ ♣ 10:InnerTank (Varying - 5 sections) ∧ ⊕ ➡ Joint (1) └_♣ 2:JointGeometric ➡ Material (40) ⊖ ➡ Isotropic (8)	60.0	. 🗌 Pla		Creep	Damage	Sł	nrinkage	Viscous	Two pha	se 🗌 Ki	o Initialisation	
-& 1:BaseSlab	40.0	Elastic										
-& 2:Wall - & 3:Bingheaml over	4		Dynamic p	roperties						Value		
-& 4:Roof			Thermal e	xpansion	n Young's modulus Poisson's ratio				35.0E9 0.2			
-& 5:ConnectionPart -& 6:InnerTank	20.0					Mass den				0.2 9033E3		
& 7:RingbeamUpper						Mass Rayleigh damping constant Stiffness Rayleigh damping constant			0.25546 0.951599E-3			
	-20.0 0.0											

Fig 211 Material Properties for 'Wall' in a Beam-Stick Model

Base slab

The Input data was used for elastic modulus and Poisson's ratio. Rayleigh damping constants need to be calculated.

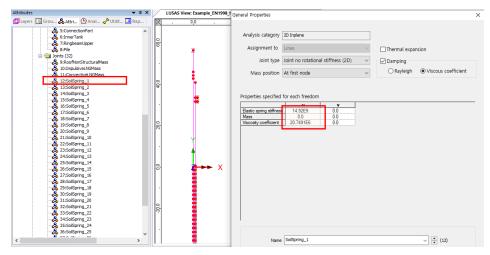


Fig 212 Material Properties for Soil Boundary in a Beam-Stick Model

Impulsive liquid mass & Stiffness

Following the code-based computation as summarized in the spreadsheet, the impulsive liquid mass and stiffness are applied to the model.

IBP (Including Base Pressure)									
Component	H/R	m _(c,i) /m	h' _(c,i) /H	mass	Lever arm height				
				mc(mi), Kg	hc(hi), m				
LNG Convective	0.924	0.49	0.84	50,527,854	32.77				
LNG Impulsive	0.924	0.51	0.79	53,494,849	30.88				

3) Stiffness for Impuls	ive Mass							
Component	Value	Unit	Remark					
H/R	0.92447		LNG height divided by inner tank radius					
ρι	480.0000	kg/m ³	mass density of LNG					
Es	2.00E+11	N/m ²	modulus of elasticity of inner tank material					
S	0.0348	m	equivalent uniform thickness of inner tank wall					
Ci	6.51359		coefficients for determining the fundamental frequency					
Timp	0.43182	S	fundamental period of oscillation of the tank (plus the i					
ki	11,325,839,357	N/m						

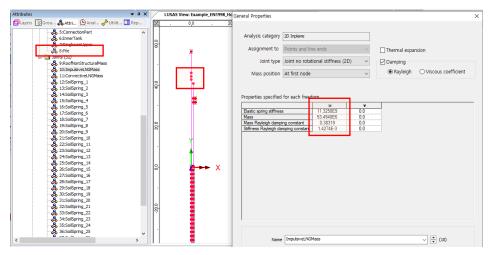


Fig 213 Material Properties for Impulsive liquid mass in Beam-Stick Model

Convective liquid mass & Stiffness

Following the code-based computation as summarized in the spreadsheet, the impulsive liquid mass and stiffness are applied to the model.

IBP (Including Base Pressure)									
Component	H/R	m _(c,i) /m	h' _(c,i) /H	mass	Lever arm height				
			_	mc(mi), Kg	hc(hi), m				
LNG Convective	0.924	0.49	0.84	50,527,854	32.77				
LNG Impulsive	0.924	0.51	0.79	53,494,849	30.88				

Value Remark H/R 0.924 Liquid height divided by inner tank radius s/m^(1/2) Cc 1.54 coefficients for determining the fundamentalfrequency T_{conv} 9.993 natural period of the first (convective) mode of sloshing s

N/m

19,974,995

2) Convective stiffness for Liquid

kc

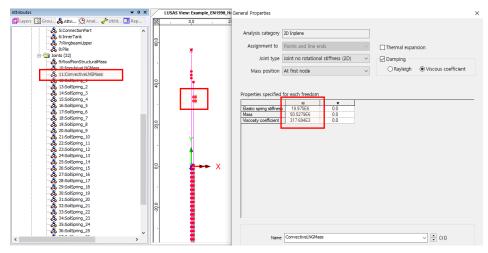


Fig 214 Material Properties for Convective Liquid Mass in a Beam-Stick Model

Inner Tank

The Input data is used for elastic modulus and Poisson's ratio. The Rayleigh damping constants are computed as follows:

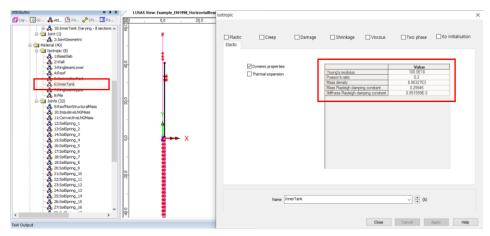


Fig 215 Material Properties for Inner Tank in a Beam-Stick Model

Viewing Results

Mode Shapes

The Layers panel in the LUSAS Modeller user interface controls what is displayed in the View window. Mode shapes can be observed by adding the **Deformed mesh** layer to the Layers treeview.

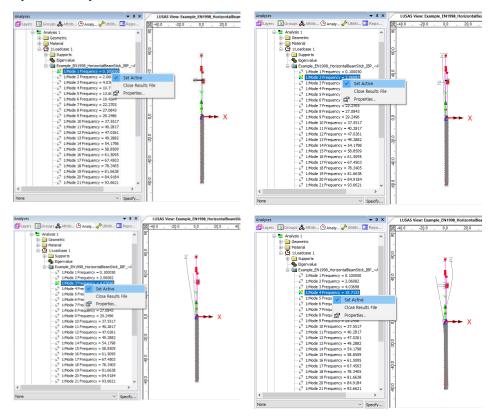


Fig 216 Mode Shapes from Eigenvalue Analysis with a Beam-Stick Model

Natural Frequencies

By selecting the menu item **Utilities>Print Results Wizard...**Natural Frequencies and Participation Factors can be displayed.

Looking at the mode shape and the mass participation factor, the 1^{st} mode is for convective liquid mass, and the subsequent modes are mixed modes. From this it would be reasonable to use the 2^{nd} and 3^{rd} frequencies as the frequency range for

computing damping constants. (e.g. f1=1.25, f2=5.44 for the 1st and 2nd frequencies in the Seismic Analysis Wizard dialog.)

Nach type Component Component <t< th=""><th></th><th></th><th>a 4</th><th>i 🖪 🖬 🧰</th><th>QQ</th><th></th><th></th><th></th><th></th><th></th><th></th><th></th></t<>			a 4	i 🖪 🖬 🧰	QQ							
Bandh 1 0 0.003 0		 Dasmaker 		Mode 🔺	Mass PF X	Mass PF Y	Mass PF Z	Mass PF THX	Mass PF THY	Mass PF THZ	Frequency	Perior
10.00xm 12.00xm/14 1	Components	Cognitations	1	1	0.194388	51,2376E-15	0.0	0.0	0.0	60.796E-9	0.10005	9.995
4 4	lts		2	2	0.292772	14.6319E-9	0.0	0.0	0.0	0.0180698	2.06082	0.485
# 4 7 33/67/8 9 44/84 6 0 0 0	dcates 12-Finenvalue	~	3	3	0.0675379	0.102479E-6	0.0	0.0	0.0	0.149097	4.03698	0.24
Pinchagen factor			4	4	7.33478E-3	0.484461E-6	0.0	0.0	0.0	0.701425	10.7133	0.093
Bit is participation factors	Eigenvalues (Frequency)		5	5	0.171007	8.17843E-9	0.0	0.0	0.0	0.0380017	13,6089	0.07
Piconiana participatori fadora ¹ / ₂ ² / ₂			6	6	0.0871761	24.4694E-9	0.0	0.0	0.0	0.0514653	19.4564	0.05
Image: Section of the secti			7	7	0.0328832	0.202533E-6	0.0	0.0	0.0	2.02404E-3	22,2705	0.044
10 16 9.8020746 8.510.06 0.8 <t< td=""><td>Sum mass participation ractors</td><td></td><td>8</td><td>8</td><td>0.0205394</td><td>0.43855E-6</td><td>0.0</td><td>0.0</td><td>0.0</td><td>6.47815E-3</td><td>27.0843</td><td>0.03</td></t<>	Sum mass participation ractors		8	8	0.0205394	0.43855E-6	0.0	0.0	0.0	6.47815E-3	27.0843	0.03
Import non Import			9	9	0.124017	0.269754E-6	0.0	0.0	0.0	0.0123545	29.2496	0.03
Image: Norm			10	10	0.582979E-3	0.131313E-9	0.0	0.0	0.0	0.262202E-6	37,5517	0.0
132 133 71 0004-0 24007-3 0 0.0 0.0 256071-3 642082 144 6236627-3 124082-3 0.0			11	11	0.495523E-3	70.3364E-9	0.0	0.0	0.0	2.57587E-3	40.2817	0.02
Image: Product Image:			12	12	0.881074E-3	59.1224E-9	0.0	0.0	0.0	2.59258E-3	47.0261	0.02
Implication			13	13	79.6084E-6	24.6835E-9	0.0	0.0	0.0	2.56587E-3	49,2882	0.02
Image: Constraint of parts Image: Constraint of parts <th< td=""><td></td><td></td><td>14</td><td>14</td><td>0.246427E-3</td><td>12.4995E-9</td><td>0.0</td><td>0.0</td><td>0.0</td><td>0.552797E-3</td><td>54.1798</td><td>0.01</td></th<>			14	14	0.246427E-3	12.4995E-9	0.0	0.0	0.0	0.552797E-3	54.1798	0.01
Produit 17 17 0.3572/14 86.1074/6 0.00 0.06.422-8 0.47550 Implifying row Implifying row <td></td> <td></td> <td>15</td> <td>15</td> <td>48.209E-6</td> <td>0.399051E-6</td> <td>0.0</td> <td>0.0</td> <td>0.0</td> <td>0.235912E-8</td> <td>58,8509</td> <td>0.01</td>			15	15	48.209E-6	0.399051E-6	0.0	0.0	0.0	0.235912E-8	58,8509	0.01
Image: Norm			16	16	2.38107E-6	2.34774E-9	0.0	0.0	0.0	0.9739838-6	61.5095	0.01
Instant plane Image Image <thimage< th=""> Image</thimage<>			17	17	0.385721E-6	8.41218E-9	0.0	0.0	0.0	40.6422E-8	67,4503	0.01
Dipuly now Born tubular proto 19 19 0.00<			18	18	0.611878E-6	0.461729E-9	0.0	0.0	0.0	3.89695E-6	78.3405	0.01
Open in treeview Default Threshold value N/A Name Threshold value N/A 1 0.1107/14.0 0.1017/14.0 0.021 0.000 0.05564.4 0.0401 Name Threshold value - 2 0.1107/14.0 0.000/14.8 0.00 0.05564.4 0.0401 222 23 0.1007/14.0 0.000/14.8 0.00 0.00 0.05564.4 0.0101 232 23 0.1007/14.0 0.000/14.8 0.00 0.00 0.05564.4 0.0101 242 24 7348444 0.000/14.8 0.00 0.00 0.05564.4 0.0101 242 24 7348444 0.000/14.8 0.00 0.00 0.00164.4 0.001 242 24 7348444 0.000/14.8 0.000 0.00 0.00164.4 0.001 0.001 242 24 7348444 0.000/14.8 0.00 0.00 0.00 0.00 0.001/14.4 0.001 0.001/14.4 0.001/14.4 0.00 <t< td=""><td></td><td>O becania passa</td><td>19</td><td>19</td><td>0.34447E-6</td><td>0.112016E-6</td><td>0.0</td><td>0.0</td><td>0.0</td><td>15.5751E-6</td><td>81.6638</td><td>0.01</td></t<>		O becania passa	19	19	0.34447E-6	0.112016E-6	0.0	0.0	0.0	15.5751E-6	81.6638	0.01
22 25 25 26 56 0.5 0.5 423324 96 2294 Name 10071 •			20	20	25.3367E-6	1.7521E-9	0.0	0.0	0.0	0.476741E-3	84,9184	0.0
22 22 9 10077-9 9 0006-9 0 0 0 4 5236-8 9 52296 22 22 0 10077-9 9 00004-4 40176-3 0 0 0 4 54296-4 105297-6 9 10027-8 9 10027-8 9 10027-8 9 10027-8 9 10027-8 9 10027-8 9 10027-8 9 10027-8 9 10027-8 9 10027-8 9 10027-8 10 1027-8 9 10027-8 10 1027-8 10 1027-8 9 10027-8 10 1027	Save in treeview Defaults	Threshold value N/A	21	21	0.176578E-9	0.152617E-6	0.0	0.0	0.0	30.5256E-6	93.6621	0.01
Name M0731 - 2 24 7.465564 0.565574 0.8 0.0 7.865564 0.00 7.865564 0.00 7.865564 0.00 7.865564 0.00 7.865564 0.00 7.865564 0.00 7.865564 0.00 7.865564 1.105564 1.105564 1.105564 1.105564 1.1055764 1.1055764 1.1055764 1.1055764 1.1055764 1.1055764 1.1055764 1.1055764 1.1055764 1.1055764 1.1055764 1.1055764 1.1055764 1.1055764 1.1055764 1.1055764 1.1055764 1.1055764 1.105577764 1.105577764 1.10557777777777777777777777777777777777			22	22	9.81037E-9	9.63686E-9	0.0	0.0	0.0	42.533E-8	95,2258	0.01
25 25 41 00086.0 99 17066.0 0.0 0.0 76 62756.6 111 125 78 28 3197266.4 5644071.0 0.0 0.0 1680066.4 111 125 27 29117266.4 55440726.6 0.0 0.0 1680066.4 111 125 27 2911726.4 55440726.6 0.0 0.0 0.0 111115.1 111947			23	23	0.7138838-6	4.65175E-9	0.0	0.0	0.0	5.45465E-6	102.132	9.79
36 36 31.97264-9 5.005446-12 0.0 0.0 0.0 1.665686-4 116.967 27 27 27.00107764-6 5.47077-6 0.0 0.0 0.021116-5 119.967	Name PRW1	~ 🗘 (new)		24	7.66836E-9	0.105082E-9	0.0	0.0	0.0	7.85055E-6	105.636	9.466
27 27 0.981878E-6 5.47407E-9 0.0 0.0 0.0 0.0 0.102111E-3 119.947			25		41.0096E-9	99.1735E-9	0.0	0.0	0.0	78.6275E-6	111.325	8.982
			26	26	31.9726E-9	5.00045E-12	0.0	0.0	0.0	1.685688-6	116.963	8.549
28 28 152020E 9 22.950E 9 0.0 0.0 13.38827E 6 131.541			27	27	0.981878E-6	5.47407E-9	0.0	0.0	0.0	0.102111E-3	119.947	8.33
			28	28	1.526268-9	72.959E-9	0.0	0.0	0.0	13.3887E-6	131.541	7.602

Fig 217 Natural Frequencies from Eigenvalue Analysis

Diagram

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

• Add the **Diagrams** layer and choose **Force/Moment – Thick 2D Beam** for **Entity**, **Fy** for **Component**, then the Shear Force Diagram is displayed.

Layers 🔻 🕈 🗙		LUSAS View: Example_EN1998_HorizontalBeamStick((BP).mdl Window 1 x	
🔁 Layers 🛛 Groups 歳 Attrib 🕒 Analy 🥓 Utiliti 💶 Repo	X	∛ -20.0 , 0,0 , 20.0 , 40.0 , 60.0 , 80.0 , 100.0 , 120.0 , 140.0 , 160.0 , 180.0 ,	
Example_EN1998_HorizontalBeamStick(IBP).mdl Example_EN1998_HorizontalBeamStick(IBP).mdl Window 1 Utilities Mesh	. 0.99		
Geometry			<
- D Attributes - D Deformed mesh - D Diagrams : Fy (Force/Moment - Thick 2D Beam)	. 40.0		
	00	Entity Force/Moment	
	8	Component Fy v Location Internal points v	
	1	. Location Internal points V	
	8.		
	20.0		
	-40.0	Cose Cancel Apply Help	
Deformations x 109.688	0.03-		
View axes Details	0.08-		

Fig 218 Shear Force Diagram from a Beam-Stick Model

The results combined with the given response spectrum is displayed by 'Setting Active' the Post Processing loadcases as shown below.

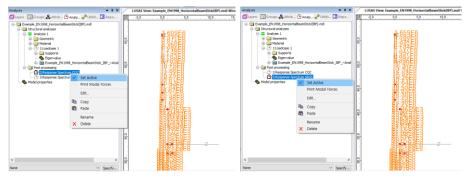


Fig 219 Shear Force Diagram for CQC & SRSS from a Beam-Stick Model

Damping applied to each mode

Because **Include modal damping** is checked in the **Eigenvalue control**, the modal damping factors computed for each mode are printed in the output file.

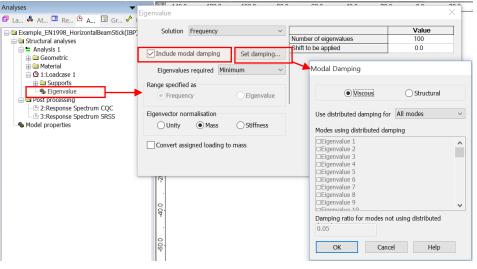


Fig 220 Eigenvalue Control for a Beam-Stick Model

The output file will have a file extension of '*.out' and can be found in this location: <Current working folder> \ Associated Modal Data \ Example_EN1998_HorizontalBeamStick(IBP)

	MODAL DAMPING FACTORS
이름	MODE FIGENVALUE FREQUENCY VISCOUS DAMPING
Backups Sessions	1 0.395176 0.100050 0.499743E-02 2 167.665 2.06082 0.241436E-01
Kample_EN1998_HorizontalBeamStick_IBP_~Analysis 1.dat Example EN1998 HorizontalBeamStick IBP ~Analysis 1.log	3 643.387 4.03698 0.239720E-01 4 4531.10 10.7133 0.437928E-01 5 7311.47 13.6089 0.555867E-01
Example_EN1998_HorizontalBeamStick_IBP_~Analysis 1_mys	6 14944.6 19.4564 0.865010E-01 7 19580.3 22.2705 0.866604E-01 8 28959.7 27.0843 0.107467
Example_EN1998_HorizontalBeamStick_IBP_~Analysis 1.out Solution Shortcut to Example_EN1998_HorizontalBeamStick(IBP)	9 33775.4 29.2496 0.102656 10 55669.6 37.5517 0.147496

Fig 221 Modal damping factors from Beam-Stick Model

Design Response Spectrum

By default, the Wizard uses the response spectrum based on ASCE7-10 (2010).

Utilities 🗸 🕈 🛪	Response Spectrum - Design Code				×
	Design code ASCE-7-10 Curve definition Incremental period Spectra definition	0 (2010) 0.02 s	V Maximum period	Show graph 6.0	S
2:InnerTank		User defined	Scale factor	1.0	
	Mapped spectral acceleration at short periods (Ss)	A ~ 0.25	Long transition period (TL) Mapped spectral acceleration at one second period (S1)	4.0 0.1	S
	Spectral data Site coefficient (Fa) Short period response acceleration parameter (Sds) Period (10)	0.8 0.133333	Site coefficient (Fv) One second period response acceleration parameter (Sd1) Period (TS)	0.8	s
< >>	Name ASCE	A-CLASS	Cose Cancel	(1) Apply He	sip

Fig 222 Design Response Spectrum used in Beam-Stick Model by default

Design spectrums can be defined by selecting the menu item **Utilities> Response Spectrum...**

Response Spectra ×	Response Spectrum - User defined
€ Response Spectrum - User defined ⊖ Response Spectrum - Design Code	Chipdexment Special carve Objective c
	$\frac{\boxed{\frac{1}{2} \underbrace{\frac{1}{2} $
	Response spectrum Name (marketering) (c) (over)
< Back Next > Finish Cancel Apply Help	< Back Next > Finish Cancel Apply Help

Fig 223 User-defined Response Spectrum

This can be used for post-processing by changing the **IMD loadcase** attribute as illustrated below.

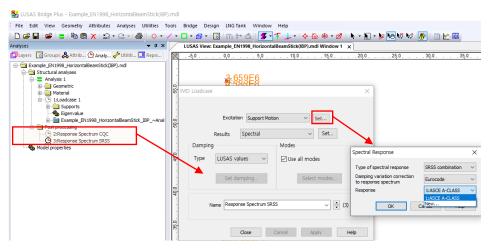


Fig 224 Change of Response Spectrum for Post-Processing

2D Beam-Stick FSSI Seismic Analysis for Vertical Actions

User Inputs

The required user inputs are the same with those for Horizontal Actions.

Seismic Analysis Wizard

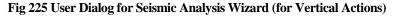
The user dialog is obtained by selecting the menu item LNG Tank> Seismic Analysis...

• Enter the model filename and select a Design Code of EN1998-4, and a model type of Beam-Stick Vertical.

The required damping ratio for the design code can be defined for each of members.

The **1**st **mode frequency** for the roof can be obtained from a separate eigenvalue analysis. This is used for computing the stiffness of roof joint element.

LNG Tank - Seismic Analys	is	×
Tank definition data Model filename Saved model file path Analysis type	Tnk1 ~ Example C:#UsersWohssoWDocuments#LUSAS190#Projects'	
Design code O Beam-stick hori (Excluding base pr (Beam-Stick Horizon)		
Critical damping / frequ	Jency	
Roof Pile LNG flexible LNG Rigid	Critical damping (%) Frequency (1st mode, Hz) 5.0 5.0 5.0 5.0 5.0 5.0	
Buttress Number of buttress Extruded thickness	4 v 1.0 (m) Buttress width 5.0 (m)	
	OK Cancel	Help



Mesh

Modelling details are discussed at 2D Beam-Stick FSSI Seismic Analysis.

The properties used for this beam-stick model are summarized in the spreadsheet named **Example_Seismic_Report(Vertical).xlsx** which can be found in the current working folder.

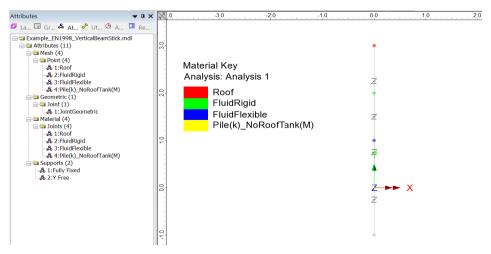


Fig 226 Mesh for a Beam-Stick Model for Vertical Actions

Material Properties

The details for computing properties are summarized in the spreadsheet.

Roof

1) Mass & Stiffness for Roof			
Component	Value	Unit	Remark
m_ _{roof}	11,985,937	kg	mass of roof
f	5.0000	Hz	fundamental frequency of oscillation of the roof
Т	0.2000	s	fundamental period of oscillation of the roof
k_roof	11,829,645,817	N/m	

	General Properties	×
Lyper: SGroups & Attrib () Analy & Utilit [] Repo Example_INI998 VerticaBeamStick.md Analy & Utilit [] Repo Analy & Point (4) Analy & ShudFloyd Astrid (4) Geometric (1) Joint (1) Astrid (4) Joint (7) Astrid (4) Astrid (4) Joint (7) Astrid (4) Astrid (4) Joint (7) Astrid (4) Astrid (5) Joint (7) Astrid (6) Astrid (7) Astr	Analysis category 2D Irplane Assignment to Points and line ends Joint type Joint no rotational stiffness (2D) Mass position At first node Properties specified for each freedom Basic sping stiffness 11 2959E5 Viscosity coefficient 37 6549E6	☐ Thermal expansion ☑ Damping ○ Rayleigh
	Name Roof	 Cancel Apply Help

Fig 228 Roof Properties in a Beam-Stick Model for Vertical Actions

Fluid-Flexible

m_ _{LNG_f}	89,566,808	kg	mass of LNG (radial breathing), ref. A.40.
· · · · · · · · · · · · · · · · · · ·			
k_lng_f	20,504,603,004	N/m	

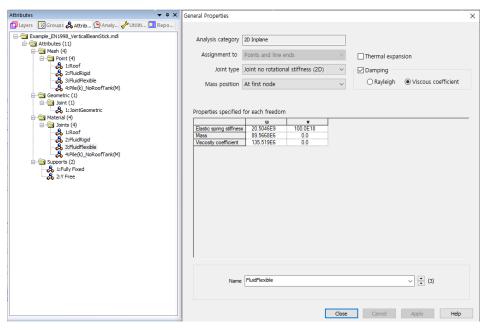


Fig 229 Material Properties for Fluid-Flexible in a Beam-Stick Model for Vertical Actions

Fluid-Rigid

m_ _{LNG_r} (1)	52,900,941	kg	mass of LNG (rigidly moving) = $sqrt(m_{LNG,total}^{2} - m_{UNG,f}^{2})$
m_ _{LNG_r} (2)	14,455,895	kg	mass of LNG (rigidly moving) = $m_{LNG,total} - m_{LNG,f}$
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), ref. A.17.
	•		
k_LNG_r	20,504,603,003,538,400	N/m	

	General Properties	×
Layer: S Groups & Attrib Analy Utilits Repo Groups & Attrib Analy Utilits Repo Gonderse Stock.md Gonderse Stock.md	Analysis category D Inplane Assignment to Points and line ends Joint type Joint no rotational stiffness (2D) Mass position At first node Properties specified for each freedom Basic spring stiffness 20 Min Viscous coefficient Mass 104 023E6 Viscous/coefficient 146 046E9	
	Name FluidRigid (2)	

Fig 230 Fluid-Rigid Properties in a Beam-Stick Model for Vertical Actions

Pile(k)_NoRoofTank(M)

3) Mass for Outer&Inner Tank					
Component	Value	Unit	Remark		
m_OuterInnerTank	53,662,366	kg	mass at top of pile = total mass - LNG - roof		
4) Mass & Stiffness f	or Pile				
Component	Value	Unit	Remark		
k_pile	225,923,300,000	N/m			

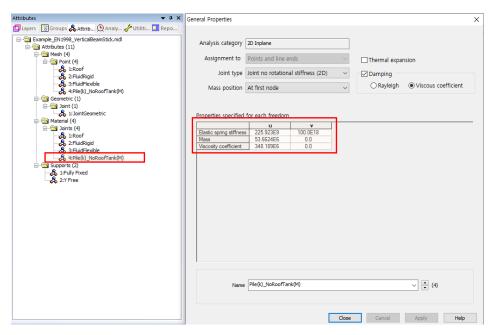


Fig 231 Material Properties for Pile(k)_NoRoofTank(M) in Beam-Stick Model for Vertical Actions

Viewing Results

Value

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

• Select the Values layer and choose Force/Moment – Thick 2D Beam for Entity, Fx for Component to display the Axial Forces.

Layers 👻 A 🗙		LUSAS View: Example_EN1998_VerticalBeamStick.mdl Window 1 x	
🗐 Layers 🐻 Groups 歳 Attrib 🕒 Analy 🥓 Utiliti 🛄 Repo	1	1,0	.6,0
Example_EH1999_VertcaBeam5tkLmd Example_EH1999_VertcaBeam5tkLmd Contempt_EH1999_VertcaBeam5tkLmd Window 1 Genety Genet	3.0	*31.6698E6	×
Eth. Janes 11 V. Antelionieri - 35 Your (Au 21)	2,0	Transform Set None	
	1.0	T132.201E6	
	00	2097,1¥E6	
		率132.201E6	
	1.0	104.471E6	
Deformations x 44.5939		31.6698E6	
Window summary Details Wew axes Details	20	[★] 209.711E6	

Fig 232 Axial Forces from Beam-Stick Model for Vertical Actions

• Set the Values Display to show all values (i.e. set 100% for both maximum and minimum), and set the **Response Spectrum CQC** active.

The axial forces for each joint element are displayed as shown below. The axial forces effectively represent the forces of each member (roof, liquid) acting on the slab.

Properties ×	Analyses	★ ♯ X	LUSAS View: Example_EN1998_VerticalBeamStick.md
Properties	🗗 Layers 🔣 Groups 歳 Attrib 🕒 Analy.	🥕 Utiliti 🛄 Repo	1.0 0,0
Value Results Values Display Entity Force,Moment Component Fx Location Averaged nodal Transform Set None	Comple EN1998 VerkaBeanStick.me Generation analyses Generatic Generat	icalBeamStick~Analysis 1.m;	\$ ≭ 31.6698E6
Display on slice(s)	I:Mode 1 Frequent I:Mode 2 Frequent I:Mode 3 Frequent I:Mode 3 Frequent I:Mode 4 Frequent I:Mode 4 Frequent I:Mode 4 Frequent I:Mode 4 Frequent I:I:Mode 4 Frequent I:I:I:I:I:I:I:I:I:I:I:I:I:I:I:I:I:I	y = 4.75650 y = 6.66528 y = 3830.39	₩ 104.471E6
	🕒 3:Response Spectrum SF	 Set Active 	
Close Cancel Apply Help	Model properties	Print Modal Forces	₹132.201E6
Properties ×		Edit	
Value Results Values Display	E	🗈 Сору	1
	0	Paste	20971¥E6
Show values of selection Deform		Rename	~209.7 ITE6
Symbols Maxima 100.0 % Failure details	:	X Delete	₹132.201E6
Significant figures Decimal places			<u>न</u> <u>न</u> 104.471E6
✓ Show traiing zeros Choose font			31.6698E6
Pen Symbol Font angle			
	<	>	₩209.711E6
Close Cancel Apply Help	None	Specify	

Fig 233 Axial Forces for CQC combination from Beam-Stick Model for Vertical Actions

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