

LNG Tank System User Manual

Concrete Tank - Part 1 - Tank Modelling

LNG Tank System

User Manual: (Concrete Tank) Part 1 – Tank Modelling

LUSAS Version 20.0 : Issue 1

LUSAS

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

Tel: +44 (0)20 8541 1999

Fax +44 (0)20 8549 9399

Email: info@lusas.com

http://www.lusas.com

Distributors Worldwide

Copyright ©1982-2023 LUSAS

All Rights Reserved.

Table of Contents

LNG Tank Modelling	1
Overview	1
2D Axisymmetric Static Structural Analysis	3
2D Axisymmetric Construction Stage Analysis	5
2D Axisymmetric Thermal Analysis	14
3D Shell Static Structural Analysis	22
3D Shell Eigenvalue Analysis	32
2D Beam-Stick FSSI Seismic Analysis	33
Exporting Forces from the 2D Axisymmetric Model	46
Exporting Forces from the 3D Shell Model	50
Examples – User Inputs	53
Tank Definition	53
2D Axisymmetric Static Structural Analysis	83
2D Axisymmetric Staged Construction Analysis	104
2D Axisymmetric Thermal Analysis	116
3D Shell Static Analysis	123
3D Shell Eigenvalue Analysis	159
2D Beam-Stick FSSI Seismic Analysis for Horizontal Actions	163
2D Beam-Stick FSSI Seismic Analysis for Vertical Actions	192

LNG Tank Modelling

Overview

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

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

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

This manual focuses on the details of modelling concepts used to build the range of models supported. A separate manual 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					×
Tank type		Та	rget models to build		
Material :	Concrete	~	2D axisymmetric structural	2D axisymmetric coupled thermal/structural	
Elevation :	Aboveground	~	☑ 2D beam-stick seismic	☑ 3D shell structural	
Tank Definition Load Prestress Insulations Support (3D) Seismic Ground					
Dase Siab and No	or wai and thing beam materi	ala ouppoir(LD)			
Support type					
Pile Support Fixed Support	~		Update from Support(3D)		
Pile Support					
Distributed sprin	ng support	Horizonts	le le		

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.

Distributed Spring 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 / Distributed Spring Support)

Loadings

Only the outer concrete tank is built in the model. This will be investigated using 17 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

Fourteen 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 first stage thickness' 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

- Layered roof option 1

Stage	Description	Note
No. 1	Annular part	
No. 2	1) + Base 1^{st} PS	
No. 3	2) + Circular part	
No. 4	3) + Base 2nd PS	
No. 5	(4) + Wall	
No. 6	5) + Ringbeam	
No. 7	6) + Ringbeam 1 st PS	
No. 8	7) + Roof Frame1	
No. 9	7) + Inner Tank Work	
No. 10	9) + Roof Frame 2	
No. 11	9) + Roof Frame 3	

Stage	Description	Note
No. 12	9) + Roof Lower Wet Concrete	
No. 13	7) + Roof Lower Complete	
No. 14	13) + Roof Upper Wet Concrete	
No. 15	14) + Roof Complete	
No. 16	15) + Ringbeam 2 nd PS	
No. 17	16) + Vertical PS	
No. 18	17) + Horizontal PS	
No. 19	18) + Operating Stage	Prestress short
No. 20	18) + Operating Stage (Long)	Prestress long

 Table 1 Sequence of Construction Stages – Layered roof option



Fig 6 Activation and Deactivation in a Staged Construction Analysis Model

Stage	Description	Note
No. 1	Annular part	
No. 2	1) + Base 1 st PS	
No. 3	2) + Circular part	
No. 4	3) + Base 2nd PS	
No. 5	4) + Wall	
No. 6	5) + Ringbeam	
No. 7	6) + Wall End 1 st PS	
No. 8	7) + Ringbeam 1 st PS	
No. 9	8) + Roof Frame1	
No. 10	8) + Inner Tank Work	
No. 11	10) + Roof Frame 2	
No. 12	10) + Roof Frame 3	
No. 13	10) + Roof Lower Wet Concrete	
No. 14	8) + Roof Lower Complete	
No. 15	14) + Roof Upper Wet Concrete	
No. 16	14) + Roof Complete	
No. 17	16) + Wall End 2 nd PS	
No. 18	17) + Ringbeam 2 nd PS	
No. 19	18) + Vertical PS	
No. 20	19) + Horizontal PS	
No. 21	20) + Operating Stage	Prestress short

- Layered roof option 2

Stage	Description	Note
No. 22	20) + Operating Stage (Long)	Prestress long

Table 2 Sequence of Construction Stages – Layered roof option 2



Fig 7 Activation and Deactivation in a Staged Construction Analysis Model

Stage	Description	Note
No. 1	Annular part	
No. 2	1) + Base 1^{st} PS	
No. 3	2) + Circular part	
No. 4	3) + Base 2nd PS	
No. 5	(4) + Wall	
No. 6	5) + Ringbeam	

Stage	Description	Note
No. 7	6) + Ringbeam 1 st PS	
No. 8	7) + Roof Frame1	
No. 9	7) + Inner Tank Work	
No. 10	9) + Roof Frame 2	
No. 11	9) + Roof Frame 3	
No. 12	7) + Roof Wet Concrete	
No. 13	7) + Roof Complete	
No. 14	13) + Ringbeam 2 nd PS	
No. 15	14) + Vertical PS	
No. 16	15) + Horizontal PS	
No. 17	16) + Operating Stage	Prestress short
No. 18	16) + Operating Stage (Long)	Prestress long

 Table 3 Sequence of Construction Stages – Single Layered roof 1





Stage	Description	Note
No. 1	Annular part	
No. 2	1) + Base 1^{st} PS	
No. 3	2) + Circular part	
No. 4	3) + Base 2nd PS	
No. 5	(4) + Wall	
No. 6	5) + Ringbeam 1 st	
No. 7	6) + Wall End 1 st PS	
No. 8	7) + Ringbeam 1 st PS	
No. 9	8) + Wall Ringbeam	
No. 10	9) + Roof Frame1	
No. 11	9) + Inner Tank Work	
No. 12	11) + Roof Frame 2	
No. 13	11) + Roof Frame 3	
No. 14	9) + Wall End 2 nd PS	
No. 15	14) + Ringbeam 2 nd PS	
No. 16	15) + Roof Wet Concrete	
No. 17	15) + Roof Complete	
No. 18	17) + Vertical PS	
No. 19	18) + Horizontal PS	
No. 20	19) + Operating Stage	Prestress short
No. 21	19) + Operating Stage (Long)	Prestress long

- Single Layered roof 2

Table 4 Sequence of Construction Stages – Single Layered roof 2

If the 'Roof first stage thickness' is set to be '1', then 17 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 9 Birth and Death Staged Construction Analysis Model (Roof ratio for 1st built =1)

Stage	Description	Note
No. 1	Annular part	
No. 2	1) + Base 1^{st} PS	
No. 3	2) + Circular part	
No. 4	3) + Base 2nd PS	
No. 5	4) + Wall Ringbeam	
No. 6	5) + Ringbeam 1 st PS	
No. 7	6) + Roof Frame1	
No. 8	6) + Inner Tank Work	

Stage	Description	Note
No. 9	8) + Roof Frame 2	
No. 10	8) + Roof Frame 3	
No. 11	8) + Roof Wet Concrete	
No. 12	6) + Roof Complete	
No. 13	12) + Ringbeam 2 nd PS	
No. 14	13) + Vertical PS	
No. 15	14) + Horizontal PS	
No. 16	15) + Operating Stage	Prestress short
No. 17	16) + Operating Stage (Long)	Prestress long

 Table 5 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.

If 'Spillage' loading for 'Thermal Loading' is defined in the Tank Definition, additional loadcases for a spillage condition are also created.

File name	LNG Tank		
Working folder			
Recent	C:\Users\ohsso\Downloads		
• User-defined	C:\Users\ohsso\Documents\LUSAS	5200\Projects	Set
Model properties			
Analysis type	Coupled thermal/structural	 Model units 	N,m,kg,s,C
Analysis category	<select></select>	 Timescale units 	Seconds ~
Optional			
Startup template	None ~	Layout grid	None ~
Title			

Fig 10 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.

tructural Thermal		Structural Thermal	
Element description Element type Axisymmetric solid Element shape Guadrilateral interpolation order Linear OElement name OAX4M	Regular mesh Allow transition pattern Allow transition pattern Automatic Local x divisions Local y divisions Inregular mesh Element size I.0	Element description Thermal element type Axisymmetric solid field Element shape Quadrilateral Interpolation order Linear Element name QXCF4	Regular mesh Allow transition pattern Allow tragular mesh Automatic Local x divisions Local y divisions Dregular mesh Element size Local x division
Name AxisymmetricSolid	Element size 1.0	VAP4	Element size 1.0

Fig 11 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 12 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.





Groups / Materials

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



Fig 14 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 15 Thermal Analysis -1st Loadcase

The 2nd Loadcase

Liquid temperature is assigned to inner side of the insulation.



Fig 16 Thermal Analysis – 2nd Loadcase

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

	Prescribed				×
· · · · · · · · · · · · · · · · · · ·	Total			al	
 		Free	Fixed	Temperature	
	Temperature	0	۲	5.0*Base Heating	
	Name Base Heatin	9		 ✓ ▲ (3) 	



Base heating temperature is assigned to the selected line as shown in [Fig 20]. This line will be split as the base heating length defined in Tank Definition if it is not the same as the total length off the base slab.



Fig 18 Base Heating Temperature in a 2D Axisymmetric Thermal Analysis Model



Fig 19 Base Heating Length for 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 20 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 21 3D Shell Model for Static Analysis



Fig 22 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 - Base Model for Design C	heck		\times
Tank definition data Model filename Saved model file path Modeling options Element size (m) 2.0 Number of eigenvalue 10	Tnk1 Example C:\Users\ohsso\Documents\LU	SAS200\Projects\Example.mdl f symmetric model lude temporary opening lude temporary opening	
Concrete Tank Options Buttress Number of buttress Extruded thickness Buttress width	4 ~ (m) 5.0 (m)	Construction Scenario - Single layered roof 1 1 - Base / Wall / Ringbeam 2 - Ringbeam 1st PS 3 - Roof frame 1 / Inner work 4 - Roof frames 2,3	
Roof / Ringbeam Roof construction plan Roof first stage thickness (ratio) Initial prestress for ringbeam (rat Initial prestress for base slab (rat	Single layered roof 1 ~ 0.5	5 - Roof wet / Roof complete 6 - Ringbeam 2nd PS 7 - Wall vertical PS 8 - Wall horizontal PS	
		OK Cancel Help	

Fig 23 User Input for the Number of Buttresses in a 3D Shell Model



Fig 24 Buttress Definition for a 3D Shell Model

Groups and Materials

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



Fig 25 Groups in a 3D Shell Model

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



Fig 26 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 38]. 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.

iterial :				Та	rget models	s to buil	d							
	Concrete			~	🔽 2D axisy	ymmetri	ic structural	\checkmark	2D axi	symmetric c	oupled the	ermal/struct	tural	
vation :	Above ground			~	✓ 2D bear	n-stick :	seismic	\checkmark	3D she	all structural				
efinition Lo	ad Prestress Ir	sulations	Support	(3D) Seismic	Ground									
Support														
pport type			Circumfer	ential Support										
nplified found	lation	~	ID	R [m]	Initial th [degre	ieta ee]	Number of piles	Vertic stiffnes [kN/m	al is 1	Horizont ^ stiffnes [kN/m]	A	Add	Crosswi	se piles
o, cir :	184		1	36.7	0.0	5	i6	523.018E	3 42	2.297E3	(Del		Circumferential piles
o. cross :	213		2	40.8	0.0	6	10	523.018E	3 42	2.297E3	Set	t zero		
X² Cir : X² Cross :	156.1965E3 63.7157E3		3	44 9	0.0	6	18	523 018F	3 43	2 297F3 ×	Set d	lefaults	Fr	
sswise supp	ort stiffness													
Grid wit	zard		Vertical	stiffness [kN/m	n] 523	3.018E3		Horizontal	stiffnes	is [kN/m]	42.297	E3		
coordinates	s (Units: m)					- 6	Y coordinate:	s (Units: m)					Add column
P1 F	P2 P3	P4	P5	P6	P7 ^	•	P1	P2	P3	P4	P5	P6	P7 ^	Add column
	8.4	12.6	16.8	21.0	25.2		0.0 0	0.0	0.0	0.0	0.0	0.0	0.0	Add row
0.0 4													-4.2	Del column
0.0 4 0.0 4	.2 8.4	12.6	16.8	21.0	25.2		-4.2 -	4.2	4.2	-4.2	-4.2	-4.2	1.6	Dercolumn
0.0 4 0.0 4 0.0 4	1.2 8.4 1.2 8.4	12.6 12.6	16.8 16.8	21.0 21.0	25.2 25.2		-4.2 - -8.4 -	4.2 8.4	-4.2 -8.4	-4.2 -8.4	-4.2	-4.2	-8.4	Del row
0.0 4 0.0 4 0.0 4 0.0 4	8.2 8.4 8.2 8.4 8.2 8.4	12.6 12.6 12.6	16.8 16.8 16.8	21.0 21.0 21.0	25.2 25.2 25.2		-4.2 - -8.4 - -12.6 -	4.2 8.4 12.6	-4.2 -8.4 -12.6	-4.2 -8.4 -12.6	-4.2 -8.4 -12.6	-4.2 -8.4 -12.6	-8.4	Del row Set zero
0.0 4 0.0 4 0.0 4 0.0 4	1.2 8.4 1.2 8.4 1.2 8.4	12.6 12.6 12.6	16.8 16.8 16.8	21.0 21.0 21.0	25.2 25.2 25.2		-4.2 - -8.4 - -12.6 -	4.2 8.4 12.6	-4.2 -8.4 -12.6	-4.2 -8.4 -12.6	-4.2 -8.4 -12.6	-4.2 -8.4 -12.6	-8.4 -12.6	Del row Set zero Set defaults
0.0 4 0.0 4 0.0 4 0.0 4	1.2 8.4 1.2 8.4 1.2 8.4 1.2 8.4	12.6 12.6 12.6	16.8 16.8 16.8	21.0 21.0 21.0	25.2 25.2 25.2	•	-4.2 - -8.4 - -12.6 - <	4.2 8.4 12.6	-4.2 -8.4 -12.6	-4.2 -8.4 -12.6	-4.2 -8.4 -12.6	-4.2 -8.4 -12.6	-8.4 -12.6	Dei row Set zero Set defaults
0.0 4 0.0 4 0.0 4 0.0 4	1.2 8.4 1.2 8.4 1.2 8.4 1.2 8.4	12.6 12.6 12.6	16.8 16.8 16.8	21.0 21.0 21.0	25.2 25.2 25.2	,	-4.2 - -8.4 - -12.6 -	4.2 8.4 12.6	-4.2 -8.4 -12.6	-4.2 -8.4 -12.6	-4.2 -8.4 -12.6	-4.2 -8.4 -12.6	-8.4 -12.6	Del row Set zero Set defaults

Fig 27 Input for Pile Locations and Stiffnesses



Fig 28 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

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

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

Fig 29 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

	Tnk1 ~					
lodel filename	Example					
aved model file path	C:\Users\ohsso\Documents\L	USAS200\Projects\Example.mdl				
Modeling options						
Element size (m)	2.0 H	alf symmetric model				
	In	iclude temporary opening				
Number of eigenvalue	10 🗸 In	clude non-structural masses in the eigenvalue analysis				
Concrete Tank Options						
Buttress		Construction Scenario - Single layered roof 1				
Number of buttress	4 ~	1 - Base / Wall / Ringbeam				
Extruded thickness	1.0 (m)	2 - Ringbeam 1st PS				
Buttress width	5.0 ()	3 - Roof frame 1/ Inner work				
Dutiess width	(m)	4 - Root frames 2,3				
Roof / Ringbeam		6 - Ringbeam 2nd PS				
Reaf equation alon	Single lavered roof 1 V	7 - Wall vertical PS				
Root construction plan		8 - Wall horizontal PS				
Root first stage thickness (ra	tio) 0.5					
Initial prestress for ringbeam	(ratio) 0.5					
Initial prestress for base slab	(ratio) 0.5					

Fig 30 User Input for a 3D Shell Model (Half Model)



Fig 31 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 Modelling

ank definition data	Tnk1	~
odel filename	Example	
aved model file path	C:\Users\ohsso\Documents\LU	JSAS200\Projects\Example.mdl
Modeling options		
E lementains (m) 2	0 V Ha	If symmetric model
Element size (m)		lude temporary opening
Number of eigenvalue) v Inc	lude non-structural masses in the eigenvalue analysis
Concrete Tank Options		
Buttress		Construction Scenario - Single layered roof 1
Number of buttress	4 ~	1 - Base / Wall / Ringbeam
Extruded thickness	1.0 (m)	2 - Ringbeam 1st PS
Buttrees width	5.0	3 - Roof frame 1/ Inner work
Buttless width	0.0 (m)	4 - Roof frames 2,3
Roof / Ringbeam		5 - Roof wet / Roof complete 6 - Ringbeam 2nd PS
De la construction de la	Single lavered roof 1	7 - Wall vertical PS
Roof construction plan		8 - Wall horizontal PS
Roof first stage thickness (rati	0) 0.5	
Initial prestress for ringbeam (natio) 0.5	
Initial prestress for base slab	(ratio) 0.5	

Fig 32 User Input for Eigenvalue Analysis Model including Non-Structural Masses

Summary of Mass (Calculation				
DIMENSION					
Component	Dimension(m)				
Inner Tank Radius	42.1				
Tank Height	40.06				
LNG Height	38.92				
SUMMARY FOR M	ASS				
	Volumo	Linit mass	Structural mass	Total mass	Equivalent unit
Component	volume	Unit mass	Structural mass	Total mass	mass
	m³	kg/m³	kg	kg	kg/m³
Roof	3,967	2,500	9,917,753	12,027,753	Not Used
Ringbeam(upper)	490	2,500	1,225,993	1,225,993	2,500
Ringbeam(lower)	433	2,500	1,081,758	1,081,758	2,500
Wall & Buttress	9,123	2,500	22,806,425	23,630,425	2,590
BaseSlab	8,719	2,500	21,797,085	24,925,085	2,859
LNG	216,714	480	104.022.703	104.022.703	480
Inner Tank	316	7.850	2.479.105	2.799.105	8.863
		.,	_,,	_,,	-,
MASS DETAILS					
Component	Descriptions				Mass (kg)
Poof	Concrete Reef /-	Roof volume * unit	concrete march		0.017.752
Root	Roof liner + stee	1,400,000			
	Susponded dock	1,400,000			
	Boof pozzlas	+ insulation of the	suspended centing		135,000
	Root nozzies				42,000
	Roof platform				400,000
	Roof pump & cra	ne			30,000
	Root piping and	support			103,000
	Others				-
	Total				12,027,753
Ring Beam	Concrete Ring Be	eam (= Ring Beam vo	olume * unit concret	e mass)	2,307,751
	wall barrier plate	2			-
	wall piping and s	upport			-
	Others				-
	Total				2,307,751
Outer Concrete Wall	Concrete Wall (=	Wall volume * unit	concrete mass)		22,806,425
	corner protectio	n			242,000
	wall barrier plate	2			494,000
	wall piping and s	upport			88,000
	Others				-
	Total				23,630,425
Base Slab	Concrete base (=	Base slab volume *	unit concrete mass		21,797,085
	Others				3,128,000
	Total				24,925,085
Inner Steel Tank	Steel tank (= Ste	el tank volume * st	eel mass)		2,479,105
	shell stiffener				45,000
	shell insulation(50%)			-
	top girder				-
	Others				275,000
	Total				2,799,105
LNG	LNG (= LNG volu	me * unit LNG mass)		104,022,703
	Total				104.022.703

Fig 33 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.

nk definition data	Inki	×
odel filename	Example	
ved model file path	C:\Users\ohsso\Document	s\LUSAS200\Projects\Example.mdl
Iodeling options		
Element size (m)	2.0	Half symmetric model
. ,		Include temporary opening
Number of eigenvalue	10	Include non-structural masses in the eigenvalue analysis
Extruded thickness Buttress width	1.0 (m) 5.0 (m)	3 - Roof frame 1/ Inner work 4 - Roof frames 2,3 5 - Roof wet / Roof complete
Roof / Ringbeam		6 - Ringbeam 2nd PS
Roof construction plan	Single layered roof 1 \sim	7 - Wall vertical PS
Roof first stage thickness	(ratio) 0.5	8 - Wall horizontal PS
Initial prestress for ringbea	m (ratio) 0.5	
Initial prestress for base sl	ab (ratio) 0.5	

Fig 34 User Input for a 3D Shell Model for Eigenvalue Analysis



Fig 35 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 44].



Fig 36 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 50].



Fig 37 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 48].

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 detail 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 49] and [Fig 50])



Fig 38 Material Properties in a Beam-Stick Horizontal Model

Summary of Beam-	Stick Model			EN 1	998-4			
DIMENSION						MASS DETAILS		
DIMENSION						Component	Descriptions	
Component						Roof	Concrete Roof (= Roof volume * unit concrete mass)	9,954,938
Inner Tank Radius	42.1						Roof liner + steel roof structure	1,400,000
Tank Height	40.06						Suspended deck + insulation of the suspended ceiling	135,000
ING Height	38.92						Roof nozzles	42,000
and neight	50.52						Roof platform	400,000
							Roof pump & crane	30,000
							Roof piping and support	103,000
SUMMARY FOR M	ASS						Others	
_					Equivalent unit		Total	12,027,753
Component					mass	Ring Beam	Concrete Ring Beam (= Ring Beam volume * unit concrete mass)	2,467,751
	m²	kg/m²	kg	kg	kg/m²		wall barrier plate	
Roof	3,967	2,500	9,917,753	12,027,753	Not Used		wall piping and support	
Ringbeam(upper)	524	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		corner protection	242,000
LNG	216,714	480	104,022,703	104,022,703	480		wall barrier plate	494,000
Inner Tank	316	7,850	2,479,105	2,799,105	8,863		wall piping and support	88,000
							Others	
							Total	25,764,428
SUMMARY FOR CA	LCULATED PRO	PERTIES				Base Slab	Concrete base (= Base slab volume * unit concrete mass)	21,797,085
1) Horizontal Model							Others	3,128,000
Component	mass	Lever arm height	stiffness	Pofe			Total	24,925,085
component				Neie	Tence	Inner Steel Tank	Steel tank (= Steel tank volume * steel mass)	2,479,10
LNG Convective	50,527,854	23.53	19,974,995	EN 1	998-4		shell stiffener	45,000
LNG Impulsive	53,494,849	16.13	11,325,839,357	EN 1	998-4		shell insulation(50%)	
							top girder	
2) Vertical Model							Others	275,000
C		stiffness					Total	2,799,105
component		k, N/m		Reference		LNG	LNG (= LNG volume * unit LNG mass)	104,022,703
LNG Flexible	89,566,808	21,631,229,542		EN 1998-4			Total	104,022,703
LNG Rigid	104022702.7	2.16312E+16		EN 1998-4				
Roof	12,027,753	-		EN 1998-4				
Pile(K) NoRoofTank	55 956 370	225 923 300 000		FN 1998-4				

Fig 39 Mass Summary for the Beam-Stick Model

						3) LNG impulsive stiff	ness				
Verification for Rea	m-Stick Model			AC13	50.3	Component					
vermeation for bea	m-Stick Model			Acij	50.5	tw	29.7905	mm	average wall thickne	ess (inner tank)	
						Es	2.00E+05	MPa	modulus of elasticit	ty of inner tank	
DIMENSION						ρε	7.8500	kN.s ² /m ⁴	mass density of inn	nass density of inner tank	
Component	Dimension(m)					C _w	0.1586		coefficients for dete	coefficients for determining the fundamental frequency	
						C ₁	0.0422		coefficients for dete	ermining the fundam	entalfrequency
Inner Tank Radius	42.1					ωί	5.473	rad/s	circular frequency o	of the impulsive mod	e of vibration
LNG Height	40.06 38.92					ті	1.148	5	fundamental period impulsive compone	i of oscillation of the nt of the contents)	tank (plus the
						ki	1,586,485,989	N/m			
SUMIWART FOR MA	55		•		Fault shart unit		EDTIES FOR VEDTIC				
Component					Equivalent unit	1) Roof Mars & Stiffe	CRITES FOR VERTIC	ALMODEL			
component	m ³	kg/m ³	kg	ke	kg/m ³	Component	Value	Unit		Remark	
Roof	3.967	2,500	9,917,753	12.027.753	Not Used	m	12 027 753	kø	mass of roof	TICTION N	
Ringbeam(upper)	524	2,500	1,310,993	1.310.993	2.500	f		HZ	fundamental freque	ency of oscillation of	the roof
Ringbeam(lower)	463	2,500	1,156,758	1,156,758	2,500	T	N/A	s	fundamental period of oscillation of the roof		roof
Wall & Buttress	9,976	2,500	24,940,428	25,764,428	2,583	k_roof	N/A	N/m			
BaseSlab	8,719	2,500	21,797,085	24,925,085	2,859	200					
LNG	216,714	480	104,022,703	104,022,703	480	2) LNG Mass & Stiffn	855				
Inner Tank	316	7,850	2,479,105	2,799,105	8,863	Component	Value	Unit		Remark	
						m_ung	104,022,703	kg	mass of LNG		
						tw	29.7905	mm	average wall thickne	ess (inner tank)	
CALCULATED PROP	ERTIES FOR HORIZ	ONTAL MOD	EL			Es	2.00E+05	MPa	modulus of elasticit	ty of inner tank	
1) LNG Mass & Heigh	t					ρι	480.0000	kg/m ³	mass density of LNC	3	
		Lever arm	Lever arm			g	9.8070	m/sec ²	gravitational acceler	ration	
Component	melmi) Ka	height (IBP)	neight (EBP)			n	4.7074	kN/m ³	specific weight of co	ontained liquid	
LNG Convective	48 423 453	31.83	23.10			Ty	0.4504	s	fundamental period	f of oscillation of the	LNG
LNG Impulsive	52,963,803	33.36	14.60			k_lng	20,247,300,685	N/m			
2) LNG convective sti	ffness					3) Mass for Outer&In	ner Tank				
Component	Value	Unit		Remark		Component	Value	Unit		Remark	
g	9.8070	m/sec2	gravitational acceleration	ation		mOuterInnerTank	55,956,370	kg	mass at top of pile	= total mass - LNG - I	oof
λ	5.8106	m ^{1/2} /s	coefficient as defined	l in 9.3.4							
ως	0.6332	rad/s	circular frequency of	oscillation of the fir	st(convective)	4) Mass & Stiffness fo	or Pile				
Tc	9.9223	\$	natural period of the	first (convective) m	ode of sloshing	Component					
kc	19,417,270	N/m				k _{ole}	225,923,300,000	N/m			

Fig 40 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 51].



Fig 41 Groups in a Beam-Stick Horizontal Model

Damping Coefficients

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

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

Fig 42 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 43 Support Conditions in a Beam-Stick Horizontal Model

Loadings

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

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

Analysis Control

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

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

Analyses 👻	A × LUSAS View: Example_EN1998_Horiz	ontalBeamStick(EBP).mdl Window 1 🗙	
🗗 Layers 🐼 Gro 🚴 Attri 🕒 Anal 🥜 Utili 💷 Re	Eigenvalue	×	:
Example_EN1998_HorizontalBeamStick(EBP).mdl Grading Structural analyses	Solution Frequency V	Value Number of eigenvalues 30	l
	Include modal damping Set damping	Shift to be applied 0.0 Modal Damping	
11.030CdSe 1	Range specified as Frequency Eigenvalue	Viscous Structural]
SiResponse Spectrum SRSS	Eigenvector normalisation O Unity	Modes using distributed damping	<u> </u>
	Convert assigned loading to mass	Bigenvalue 1 Bigenvalue 2 Bigenvalue 3 Bigenvalue 4 Bigenvalue 5	
	0	Bigenvalue 6 Bigenvalue 7 Bigenvalue 8 Damping ratio for modes not using distributed dampin 0.05 OK Cancel	rg

Fig 44 Eigenvalue Control for a Beam-Stick Horizontal Model

Response Spectrum

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

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





Options for Post-Processing

After solving the model, the results of eigenvalue analysis will be loaded on LUSAS Modeller. The results are combined in accordance with the options available in post-processing loadcase. The options can be defined manually by selecting the **Analyses>IMD loadcase** menu item and making selections as shown in [Fig 58].

Analyses	* # X	. 0	-200.0	-180.0	-160.0	-140.0	-120.0	-100.0
🛱 Layers 🛐 Groups 🖧 Attribu 🖳 Analys 🖌	Utilities 🖪 Reports	8.						
testACL_ACI350_HorizontalBeamStick(IBP).mdl TestACL_ACI350_HorizontalBeamStick(IBP).mdl TestAnalysis 1 TestAnalysis 1 TestAnalysis 1 TestAnalysis 1	IMD Loadcase				×			
H Material	Excitation	Support Moti	on 🔻	Set				
Gupports Genvalue	Results	Spectral	v	Set	Spectral Respon	se		×
Post processing	Damping		Modes		Type of spectral r	esponse	CQC combine	ation 🚺
3:Response Spectrum SRSS	Type LUSAS vi	alues 4	Use all m	nodes	Damping variation	correction	Eurocode	2
	Set da	mping	Sele	ect modes	Response	rum	1:ASCE A-CL	LASS 3
					0	K	Cancel	Help
	Name	sponse Spectru	m CQC	~	(2)			
	C	lose	Cancel	Apply	Help			

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

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

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

The design response spectrum is chosen.

If the 'Include modal damping' option is checked from Eigenvalue analysis control dialog (see [Fig 56]), 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 59].





The model is built using four joint elements as shown in [Fig 60]. 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.





Geometric Properties

The following dataset is used.

Joint Geometric Pr	operties				×
Analysis cate <u>o</u> Use joint le	jory	2D Inplane]		
Cor Eccentricity (e	nponent		 Value 0.0		
	-,				
Name	JointGeom	etric		~ •	ω

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

Material Properties

Mass, stiffness, and damping coefficients are assigned for material properties for joint element as shown in [Fig 62].



Fig 50 Material Properties in Beam-Stick Vertical Model

Details of how masses and stiffness are calculated are summarized in a spreadsheet form as shown in [Fig 52] and [Fig 53]. 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.

2) Mass & Stiffness for	LNG				
Component	Value	Unit	Remark		
H/R	0.924		LNG height divided by inner tank radius		
ρι	480.0000	kg/m³	mass density of LNG		
Es	2.00E+11	N/m ²	modulus of elasticity of inner tank material		
ν	0.3		poisson ratio of steel		
s(ζ)	0.0361	m	wall thickness for $\zeta = 1/3$ ($\zeta = z/H_L$)		
f(γ)	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		mass of LNG (rigidly moving) = $sqrt(m_{LNG_{total}}^2 m_{LNG_{f}}^2)$		
m _{LNG_r} (2)	14,455,895	kg	mass of LNG (rigidly moving) = m_ _{LNG_total} -m_ _{LNG_f}		
P _{vr}	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)		
Ι ₀ (γ ₁)	1.8629		bessel function order 0		
Ι ₁ (γ ₁)	1.1953		bessel function order 1		
f _{vd}	2.4734	Hz	fundamental frequency of oscillation of the liquid		
T _{vd}	0.4043	S	fundamental period of oscillation of the liquid		
kf	21,631,229,542	N/m			
k_ _{LNG_r}	21,631,229,542,194,300	N/m			

Fig 51 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 67]. This mass is assumed to move rigidly vertically.

3) Mass for Outer&Inne	er Tank				
Component	Unit	Remark			
mouterInnerTank	55,956,370	kg	mass at top of pile = total mass - LNG - roof		
4) Mass & Stiffness for	Pile				
Component	Value	Unit		Remark	
k_pile	225,923,300,000	N/m			

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

Damping Coefficients

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

Support Conditions

Only vertical movement is allowed for all members.

The end of the pile joint is fully fixed. The mass considered in the pile joints comprises the mass moving as a rigid body in the vertical direction, i.e. the sum of the mass for the outer tank (excluding the roof) and the inner tank. The stiffness is defined from user input. These values are summarised in the spreadsheet shown in [Fig 67].



Fig 53 Supports in a Beam-Stick Vertical Model

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

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

See *Examples – User Inputs : 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 69]. For example, SY through the wall section can be used for computing vertical axial forces and bending moment.



Fig 54 Converting Stress to Forces

The forces for the sliced section are automatically calculated by the Wizard from LNG Tank> Excel Tools> Export Forces

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



Fig 55 Section Force Spreadsheet for 2D Axisymmetric Solid Model

Roof - Exporting Forces



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

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

Wall - Exporting Forces



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

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

Base Slab - Exporting Forces



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

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

Exporting Forces of Specific Named Groups

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

utout filonomo	Slice2D				
orking folder	Current	O User Defined			
	-				
ave in	C:\Users\ohsso\Da	cuments\Lusas200\Projects\Slice2D_E	laseSlab.xisx		
arget					
Base slab		⊖ Wall + Ringbeam		⊖ Roof	
Loadcases			Range (X Coord)	
1:SelfWeight 2:Dead Loads of Steel Struct	ure	^	Start :	0.0	m
3:Dead load of liner and stee	el roof				
4:Dead load of steel structure 5:Dead load of Insulation	es on the roof		Finish :	46.5	m
6:Pressure on outer tank wal	Il due to insulation		Interval -	0.5	m
7:Wall piping loading 8:Liquid bottom(Max)			intervar.		
9:Liquid bottom(Min)					
10:Gas Pressure(Max) 11:Gas Pressure(Min)					
12:Live load					
13:Snow load					
14. Lescioau (Liquid)		*			

Fig 59 Exporting Forces for a 2D Axisymmetric Solid Mode

Exporting Forces from the 3D Shell Model

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

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

LNG Tank - Export Forces/Moments to Excel (3D)	×
Output filename Example	
Working folder Ourrent User Defined	
Save in C:\Users\ohsso\Documents\Lusas200\Projects\	Example_BaseSlab.xlsx
Target	Range
● Base slab ─ Wall + Ringbeam ─ Roof ─ All	Angles : 20 degree (eg. 10; 20; 30) Interval : 0.5 [m]
Results to extract	Exclude forces on the base slab at pile heads and wall
✓ Forces and Moments	Diameter of crosswise piles : 0.7 [m]
✓ Design results	Diameter of circumferential piles : 0.8 [m]
ULS UtilPM UtilShear OPM Capacity Shear O	apacity 🗹 UtilDecompression 🗹 Compression Depth
1:SetWeight 2:Dead Loads of Steel Structure 3:Dead Loads of finer and steel roof 4:Dead load of finer and steel roof 5:Dead load of finualian	↑Y axis

Fig 60 Exporting Forces for a 3D Shell Model



Fig 61 Section Forces Exported from a 3D Shell Model

Examples – User Inputs

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

Tank Definition

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



Fig 62 Tank Definition Dialog

- □ Tank Type Material Two types of tanks are able to be modelled. Either 'Concrete' or 'Metallic' should be selected.
- □ Tank Type Elevation One of elevation type should be selected either 'Aboveground' or 'Elevated/Isolated'.
- □ 3D shell structural This option should be checked to define each pile location and its properties in a 3D shell model. If checked (ticked) the Pile Arrangement (3D) tab will appear
- □ 2D axisymmetric coupled thermal-mechanical 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.
- □ 2D beam-stick seismic This option should be checked for Seismic Analysis. If checked (ticked) the Seismic and Ground tabs for seismic data will appear.

Structural Definition

Concrete Tank

Tank Definition						×
Tank type Material : Elevation :	Concrete Aboveground	✓ ✓ 20	nodels to build) axisymmetric structural) beam-stick seismic	2D axisymmetric cou	pled thermal/structural	
Tank Definition Lo Base Slab and Ro	oad Prestress of Wall and Ring beam Materials	Support (2D)				
Base slab (Unii Circular part ler Circular part de Tapered sectio Annular part ler Base heating (I Base heating (I Ground level (I	ts:m) 	39.8 1.2 0.6 1.5 0.386 46.5 0.9		Linner Deating	Couter Couter	
Roof (Units: m) Radius of inner Radius of outer Height from the topmost of the i Distance of tap Distance of tap	roof (R_roof_i) roof (R_roof_o) top of the base slab to the roof (R_Height) ered section 1 (sl1) ered section 2 (sl2)	86.406 86.906 56.2545 10.079 0.6	Troot T	Recot.o	Sla Hringbeam_2 • Rsl_height	
Set zero	Set defaults	Name Tnk2			✓ (new)	
					OK Cancel Apply	Help

Fig 63 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.
- □ Base Heating (L_heating): Defines the length from the center of tank to the heating line if base heating is considered in the analysis. Base heating is installed to maintain constant temperature in base slab.
- □ Ground level (D_ground): Defines the delpth from the bottom off the outer tank to the ground level.

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_0):** 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.

nk Definition								
ank type			Target models to build					
Material :	Concrete	~	2D axisymmetric structure	ctural	2D axisymmetric	coupled thermal/structural		
Elevation :	Aboveground	~	✓ 2D beam-stick seismi	ic	☑ 3D shell structura			
nk Definition Lo	ad Prestress Insulations	Support (3D) Seisn	ic Ground					
ase Slab and Ro	of Wall and Ring beam M	aterials Support (2D)						
Wall and ring be	eam (Units: m)				Trincheam			
Inside radius of	concrete outer tank wall (li	nsR)	43.2		•+			
Thickness of wa	all base (T_bottom)		1.1		1 L	de alt anna		
Height of tapere	ed wall (H_wall_t)		7.4	-	· ·			
Thickness of wa	all top (T_top)		0.75		Ttop			
Height of wall (H	H_wall)		42.68			Rsl heig	h	
- Height of ringbe	aam 2 (H ringbeam 2)		1.7				Hringbea	m_2
- Height of ringbe	eam 1 (H ringbeam 1)		1.5	Hwal			Hringbea	m_1
 Thickness of rin	ngbeam (T ringbeam)		1.05			II Insulation	· ·	
Slope height (R	t sl height)		0.0		Ba	tom Corner Protection		
	• /							
				Hbcp_t				
						wall_t		
Corner Protection	on (Units: m)			Hhen h				
Corner protecti	ion start (H_bcp_s)*		0.617	Theorem 1				
Corner protecti	ion end (H_bcp_e)*		5.617		T _{bcp} T _{bottom}			
Corner protecti	ion thickness (T_bcp)*		0.155	* Guidance f	or corner protection in	outs based on the current i	nsulation data	
				- Corner prot	ection start: 0.105 or 0	.567 or 0.617 or 0.6915		
Set zero	Set defaults	Wall stages	Openings	- Corner prot	ection end : 5.617			
0012610	occusions		oponingo	- Corner prot	ection thickness: 0.15	5		
		Name	ink2			 (new) 		

Fig 64 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.
- □ Corner protection start(H_bcp_s): Defines the height where the corner protection start based on the top surface of base slab.

- □ Corner protection end(H_bcp_e): Defines the height where the corner protection end based on the top surface of base slab.
- □ Corner protection thickness(T_cbp): Defines the thickness for corner protection.

Wa	Il Construction S	tages							×
	Height / Stages –								
	Wall stage ID	Height (H) [m]	Stage Y/N					H6	
	1	0	Y	•				-	
								Hs	
						Set defaults		H4	
								_	
						Clear grid		Нз	
						Add		Ť	
								H ₂	
						Remove		H1	
					1				
				(OK	Cancel	Apply	H	lelp

Fig 65 Wall Construction Stages Dialog (Tank Definition/ Structure Definition/ Concrete Tank/ Wall and Ring Beam)

- □ Wall stage ID: Wall lot IDs from the bottom of the wall. This value is automatically set.
- **Height (H):** Defines the height of each wall lot. This value should be positive.
- □ Stage Y/N: Defines whether the stage should be separated at each wall section. 'Y' should be selected if a separate stage should be created for the wall lot in the model. Otherwise 'N' should be selected. However, if the input value is 'N' for wall stage ID "1", it is assumed that the wall lot 1 is activated together with the vase annular part as shown in the Figure



Fig 66 The stage of activating wall lot 1 when 'N' for 'Staged Y/N

Description Opening 1 Opening 2 Openings width (Wo) 0 0 PS free length (Wgap) 0 0 Opening elevation (H1) 0 0 Opening height (H2) 0 0 PS free height (H) 0 0 Set defaults Clear grid • *Theta' is the angle between opening center and the adjacent buttress center	Openings			
Openings width (Wo) 0 0 PS free length (Wgap) 0 0 Opening elevation (H1) 0 0 Opening height (H2) 0 0 PS free height (H) 0 0 Set defaults Clear grid • *Theta' is the angle between opening center and the adjacent buttress center	Description	Opening 1	Opening 2	
PS free length (Wgap) 0 0 0 Opening elevation (H1) 0 0 Opening height (H2) 0 0 PS free height (H) 0 0 Set defaults Clear grid * Theta' is the angle between opening center and the adjacent buttress center	Openings width (Wo)	0	0	Wgap +
Opening elevation (H1) 0 0 Opening height (H2) 0 0 PS free height (H) 0 0 Set defaults Clear grid • "Theta' is the angle between opening center and the adjacent buttress center"	PS free length (Wgap)	0	0	Opening 1 Opening 2 Wgap
Opening height (H2) 0 0 PS free height (H) 0 0 Set defaults Clear grid • • • • • • • • • • • • • • • • • • •	Opening elevation (H1)	0	0	Wo Theta Wo
PS free height (H) 0 0 0 • • • • • • • • • • • • • • • •	Opening height (H2)	0	0	
Set defaults Clear grid * 'Theta' is the angle between opening center and the adjacent buttress center	PS free height (H)	0	0	
Set defaults Clear grid * 'Theta' is the angle between opening center and the adjacent buttress center	Onening leastion angle (Thete)	^	^	
	Set defaults Clear grid			* 'Theta' is the angle between opening center and the adjacent buttress center

- **Opening width (Wo):** Defines the width of opening.
- **PS free length (Wgap):** Defines the length of prestress free zone.
- **Opening elevation (H1):** Defines elevation from the top surface of base slab.
- **Opening height (H2):** Defines the heights for each opening.
- **PS free height (H):** Defines the height of prestress free zone.
- **Opening location angle (Theta):** Defines angle to the middle of opening.

Insulation



Fig 67 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.
- □ Set defaults: Sets all the input values to default values.
- **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 6 layers of wall insulation can be defined.

Roof insulation

A maximum of 4 layers of wall insulation can be defined. Roof insulation layers are assumed to sit on top of the innermost layer of wall insulation.

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



Several examples of defining wall and roof insulation follow:





Case 1

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

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

Case 2

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

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

Case 3

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

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

Case 4

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

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

Case 5

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

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

Case 6

2 wall insulation layers and 1 roof insulation layer defined.

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

Material Properties

anktype Material: Levation: boowsground 12 Detamatick selamic 12 Detamatick selamic 12 Detamatick selamic 12 Detamatick selamic 12 Detamatick selamic 12 Detamatick selamic 12 Detamatick selamic 12 Detamatick selamic 12 Detamatick selamic 12 Detamatick selamic 12 Detamatick selamic 12 Detamatick selamic 12 Detamatick selamic 12 Detamatick selamic 12 Detamatick selamic 12 Detamatick selamic 12 Detamatick selamic	nk Definition								
Material I: Concrete Elwation I: Doveground and definition Load Prestees; Insulations' Support (3D) Setemic Concurd: and definition Load Prestees; Insulations' Support (3D) Setemic Concurd: and definition Load Prestees; Insulations' Support (3D) Setemic Concurd: and definition Load Prestees; Insulations' Support (3D) Setemic Concurd: and definition Concurse Prestees; Insulations' Support (3D) Setemic Concurd: and definition Concurse Prestees; Insulations' Support (3D) Setemic Concurd: and the concurse (Nam) bits and Roof Vall and Ring beam. Material: Support (Roof) 35.0E9 0.2 2.5E3 10.0E-6 2.0 2.5E3 10.0E-6 0.0 0.0 10.0E-7 0.0 10.0E-6 0.0 10.0E-6 0.0 0.0 10.0E-6 0.0 <p< td=""><td>ank type</td><td></td><td></td><td>Target models to</td><td>build</td><td></td><td></td><td></td><td></td></p<>	ank type			Target models to	build				
Elevation: Abovegnound Image: Comparison of the comparison	Material :	Concrete	~	🔽 2D axisym	metric structural	🗸 2D axis	ymmetric coupled t	hermal/structural	
hi bolinion Laid Preseres Insulation's Support(3D) Sesinic Ground as Siband Roof Wall and Ring beam. Materials Support(3D) Taterial ID Basic modulu (E, [Wm ⁻]) So DE9 02 02 02 02 02 02 02 02 02 02	Elevation :	evation : Aboveground ~		☑ 2D beam-s	tick seismic	☑ 3D shel	l structural		
ib Celfino Load _ Prestress _ Insulations _ Support (D) _ Seame Ground : see State and Roof Wall and Ring beam Materials : Support (D) _ for manual : Support (D) _ for manual : for onductivity _ Ums CI _ 2000 _ 25766 _ BaseSlab									
tabe and Roof Will and Hold Will and King Maam Maase Support (20) Thermal Liper Conductivity Million (20) Pescription taberial ID Elastic mondus Poisson's ratio Mass density (11/C) Thermal Units (2) Description oncrete (Base) 35.0E9 0.2 2.5E3 10.0E-6 2.0 2.257E6 BassEsiab oncrete (Ringbeam) 35.0E9 0.2 2.5E3 10.0E-6 2.0 2.257E6 RingBeam oncrete (Ringbeam) 35.0E9 0.2 2.5E3 10.0E-6 2.0 2.257E6 RingBeam oncrete (Ringbeam) 35.0E9 0.2 2.5E3 10.0E-6 2.0 2.257E6 RingBeam oncrete (Ringbeam) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pie (Cir) ie (Cross) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pie (Cross) * solator properties can be defined for various types from modeler and should be defined and assigned manually. Set zero Set defaults	nk Definition Load	Prestress Insulations	Support (3D) Seis	mic Ground					
chreche (Base) 35.0E9 0.2 25E3 10.0E-6 2.0 2257E6 BaseSlab oncrede (Wall) 35.0E9 0.2 25E3 10.0E-6 2.0 2257E6 RingBeam oncrede (Ringbeam) 35.0E9 0.2 25E3 10.0E-6 2.0 2.57E6 RingBeam oncrede (Ringbeam) 35.0E9 0.2 2.5E3 10.0E-6 2.0 2.57E6 RingBeam oncrede (Ringbeam) 35.0E9 0.2 2.5E3 10.0E-6 2.0 2.57E6 RingBeam oncrede (Ringbeam) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pile (Cir) ie (Ciros) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pile (Ciros)	faterial ID	Elastic modulus (E, [N/m²])	Poisson's ratio (v)	Mass density [kg/m ³]	CTE (alpha, [1/C])	Thermal conductivity [J/m.s.C]	Heat capacity [J/m³/C]	Description	
snarete (Wal) 35.0E9 0.2 2.5E3 10.0E-6 2.0 2.257E6 Mail snarete (Ringbeem) 35.0E9 0.2 2.5E3 10.0E-6 2.0 2.257E6 RingBeem ancrete (Ringbeem) 35.0E9 0.2 2.5E3 10.0E-6 2.0 2.257E6 RingBeem ancrete (Ring) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pie (Cir) a (Cross) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pie (Cir) a (Cross) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pie (Cir) a (Cross) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pie (Ciross) a (Storp properties) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pie (Ciross)	oncrete (Base)	35.0E9	0.2	2.5E3	10.0E-6	2.0	2.257E6	BaseSlab	
sncrete (Ringbeem) 35.0E9 0.2 2.5E3 10.0E-6 2.0 2.257E6 RingBeem e (Cr) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pie (Cr) e (Cr) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pie (Cr) e (Cross) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pie (Cross) * storage the defined for various types from works whether and about the defined and assigned manually. * storage the defined for various types from the defined and assigned manually. * storage the defined to various types from the defined and assigned manually. * storage the defined to various type to various types from the defined and assigned manually. * storage the defined to various type to various ty	oncrete (Wall)	35.0E9	0.2	2.5E3	10.0E-6	2.0	2.257E6	Wall	
oncrete (Root) 35.0E9 0.2 2.5E3 10.0E-6 2.0 2.257E6 Root le (Cir) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pie (Cir) le (Cross) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pie (Cross)	oncrete (Ringbeam	n) 35.0E9	0.2	2.5E3	10.0E-6	2.0	2.257E6	RingBeam	
le (Cir) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pile (Cir) le (Cross) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pile (Cross)	oncrete (Roof)	35.0E9	0.2	2.5E3	10.0E-6	2.0	2.257E6	Roof	
le (Cross) 35.0E9 0.2 2.5E3 10.0E-6 0.0 0.0 Pile (Cross) * Isolator properties can be defined for various types from modeler and should be defined and assigned manually. Set zero Set defaults Variant Varian	le (Cir)	35.0E9	0.2	2.5E3	10.0E-6	0.0	0.0	Pile (Cir)	
* Isolator properties can be defined for various types from modeler and should be defined and assigned manually. Set zero Set defaults	ile (Cross)	35.0E9	0.2	2.5E3	10.0E-6	0.0	0.0	Pile (Cross)	
Name Trik2 V (new)	* Isolator properti	ies can be defined for var	rious types from m	todeler and should	I be defined and a	ssigned manually.			
			Name	Tak2			~	(man)	
			ivame	THK2			~	▼ (new)	

Fig 68 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. Material properties for pile(cir) and pile(cross) are only required when piles are modelled. Thermal Conductivity and Heat capacity should be entered only when thermal analysis is carried out.

type			Target models to	build			
Material : Concrete ~		~	🔽 2D axisymn	netric structural	🗹 2D axisyı	mmetric coupled the	ermal/structural
vation :	Aboveground	~	☑ 2D beam-stick seismic		☑ 3D shell :	structural	
efinition Load	Prestress Insulations S	Support (3D) Seisr	nic Ground				
nsions Material	s						
Material ID	Elastic modulus [E, N/m²]	Poisson's ratio (v)	Density (rho, [kg/m³])	CTE (alpha, [1/C])	Thermal conductivity [J/m.s.C]	Heat capacity [J/m³/C]	Description
Soil	1.0	0.2	0.0	10.0E-6	2.0	1.97E6	Soil
1	1.0	0.2	0.0	10.0E-6	2.0	2.257E6	Levelling concrete
2	1.0	0.2	0.0	10.0E-6	0.79	2.257E6	Dry Sand
3	1.0	0.2	0.0	10.0E-6	0.0446	90.4344E3	Cellular Glass(Type1)
4	1.0	0.2	0.0	10.0E-6	0.27	753.163E3	Perlite Concrete(Base Slab)
5	1.0	0.2	0.0	10.0E-6	0.0411	51.2E3	Loose Fill Expanded Perlite(Wall)
6	1.0	0.2	0.0	10.0E-6	0.032	9.6E3	Resilient Glass Fibre Blanket(Wall)
7	1.0	0.2	0.0	10.0E-6	0.0372	18.0627E3	Glass Fibre Blanket(Roof)
et zero 💠	Set defaults Add	I Remov	re				

Fig 69 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
Support (2D)

ant Centrols Concerned to build Tark type Everation: Aboveground Tark tarks standard Tark Definition Load Preateress Insulations Support (20) Seattle Cacund Base Stab and Roof Wait and Ring beam Materials Support (20) Support type Bienbluided spring support Venical stiffness 065 (MN/mim] Horizonal stiffness 2063 (MN/mim]) Distributed Spring Supports Stat and Roof Wait and Ring beam Materials Support (20) Sector Venical stiffness 065 (MN/mim] Horizonal stiffness 2063 (MN/mim]) Stitubuted Spring Supports Stat and Roof Wait and Ring beam Materials Support (20) Sector Venical stiffness 065 (MN/mim] Horizonal stiffness 2063 (MN/mim]) Sector Venical stiffness 065 (MN/mim] Horizonal stiffness 065 (MN/mim]) Sector Venical stiffness 065 (MN/mim]) Sector Venical stiffness 065 (MN/mim]) Sector Venical stiffness 065 (MN/mim]) Material Venical Stiffness 065 (MN/mim]) Sector Venical Stiffness 065 (MN/mim])	Taali Dafisitian								~
Tank type Tank Tank	Tank Definition								
Matria Concerner de la aciegnmentio du coloral de la aciegnmentio de coupled memalitaructural 2 de beam-stok seismic 3 de bei atructural 2 de beam-stok	Tank type	a		Target models to build	_				
Evaluations Abbreground ank Definition Load Prestress Insulations Support (3D) Seismic Ground Base Slab and Roof Wall and Ring beam Materials Support (3D) Seismic Ground 20E3 [MN/m/m] Support type Destroated atiffness 1.0E3 [MN/m/m] Horizontal atiffness 2.0E3 [MN/m/m] Distributed spring support Vertical stiffness 1.0E3 [MN/m/m] Horizontal atiffness 2.0E3 [MN/m/m] Stetzero Set defaults Set defaults Set defaults Set defaults Set defaults	Material :	Concrete	~	2D axisymmetric structural	✓ 2D axisymmetric	coupled thermal/structu	iral		
ank Definition Load Preteres Insulations Support (20) Seemic Ground Base Slab and Roof Wall and Ring beam Maerials Support (20) Support type Deschued spring support Verical stiffnes 10E3 MN/m/m ² Horizontal atfines 20E3 MN/m/m ²	Elevation :	Aboveground	~	✓ 2D beam-stick seismic	☑ 3D shell structura	l			
Base Slab and Root Wall and Ring beam Materials Support (20)	Tank Definition	oad Prestress Insulations	Support (3D) Seis	mic Ground					
Support vertical stiffness 1.0E3 (MN/m/mg Horizontal stiffness 2.0E3 (MN/m/mg)	Base Slab and Re	oof Wall and Ring beam Ma	aterials Support (2D)						
Desributed spring support Vertical stiffness 1.0E3 Mi/mimil Horizontal stiffness 2.0E3 Mi/mimil Distributed Spring Supports 0 0 0 0 0 0 Set zero Set defaults Name Titl2 Vertical stiffness 2.0E3 Mi/mimil	Support type								
State	Distributed spr	ing support Vertica	al stiffness 1.0	E3 [MN/m/m²]	Horizontal stiffness	2.0E3 [MN/m/r	m²]		
Set zero Set defaults									
Set zero Set defaults					Distribu	uted Spring Support	s		
Set zero Set defaults									
Set zero Set defaultz									
Set zero Set defaults									
Set zero Set defaults									
Set zero Set defaults Name Tink2 OK OK Cancel									
Set zero Set defaults									
Set zero Set defaults									
Set zero Set defaults									
Set zero Set defaults									
Set zero Set defaults									
Set zero Set defaults									
Set zero Set defaults									
Set zero Set defaults									
Set zero Set defaults					Maga	*****	~~~~~~~~~~~	Second	
Set zero Set defaults								****	
Set zero Set defaults Name Trk2 OK Cancel Anniv Help									
Name Trik2 Cancel Anniv Helo	Set zero	Set defaults							
Name Trikz Cancel Apply Help									
Name Trik2 Cancel Aprily Help									
Name Trik2 Cancel Apply Help									
OK Cancel Apply Help			Name	Tnk2		 (new) 			
						ОК	Cancel	Apply	Help

Fig 70 Tank Definition Dialog (Support (2D) – Distributed spring support)

Support Type

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

ank type			Target m	odels to build	
Material :	Concrete	~	20	axisymmetric structural	2D axisymmetric coupled thermal/structural
Elevation :	Aboveground		✓ 2D	beam-stick seismic	☑ 3D shell structural
Lievalion .	Abovegiound				
hk Definition Lo	ad Prestress Ins	sulations Support (3D) Seismic Grou	nd	
se Slab and Ro	of Wall and Ring b	beam Materials Su	pport (2D)		
Support type					
Pile Support	~		L	Jpdate from Support(3D)	
Pile stiffnesses					
Spring ID	Radius [m]	Vertical stiffness [MN/m/rad]	Horizontal stiffness [MN/m/rad]	Description	Pile Supports
1	4.2	592.0	47.9	Pile	
2	8.4	1.0524E3	85.1	Pile	
3	12.6	1.5987E3	127.7	Pile	
4	16.8	2.1049E3	170.2	Pile	
5	21.0	2.6311E3	212.8	Pile	
6	25.2	3.1573E3	255.3	Pile	
7	29.4	3.4962E3	282.7	Pile	
В	32.9	3.7295E3	301.6	Pile	Spring Number
Э	36.7	4.6615E3	377.0	Pile	1 2 3
<				>	
Set zero	Set default	s Add	Remove		
			Name Tak2		(new)

Fig 71 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.

Load

Tank Definition						:	×
Tank type				Target mode	els to build		
Material :	Concr	ste	~	🔽 2D axi	symmetric structural	✓ 2D axisymmetric coupled thermal/structural	
Elevation :	Above	ground	~	☑ 2D be	am-stick seismic	☑ 3D shell structural	
Structural Dead Load	ding Str	ress Insulat	ons Support (3D)	Seismic Ground			
			no zodanigi mom	Makes fishting			
Load type		Notation	Dimension [m]	kN/m]	Description	- Dead load of steel structure	
Dead Loads of Ste	el Str	q_1	42.1	1.343	Steel Structure_q1	- Dear Ned Orsteer structure	
Dead Loads of Ste	el Str	q_2	0.0	0.0	Steel Structure_q2		
Dead Loads of Ste	el Str	q_3	0.0	0.0	Steel Structure_q3		
Dead Loads of Ste	el Str	q_4	0.0	0.0	Steel Structure_q4		
Dead Loads of Ste	el Str	P	42.1	72.9	Steel Structure_P		
Dead Loads of Ste	el Str	q_r	42.1	0.098	Steel Structure_qr		
Dead load of liner a	and st	q_1	43.2	0.404	Liner_base1		
Dead load of liner a	and st	q_2	0.0	0.0	Liner_base2		
Dead load of liner a	and st	q_3	0.0	0.0	Liner_base3		
Dead load of liner a	and st	q_4	0.0	0.0	Liner_base4		
Dead load of liner a	and st	q_r_liner	43.2	1.095	Liner_Roof	_ ·	
<					>	>	
						Set zero Set defaults	
			N	ame Tnk2		 (new) 	
						OK Cancel Apply Help	

Fig 72 Tank Definition Dialog (Load – Structural Dead Loading)

Structural Dead Loading

This defines the structural dead loading to consider in analysis.

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

Tank type			Target mod	lels to build						
Material : Conc	rete	~	🗸 2D a)	cisymmetric structu	iral 💽	2D axisymmetri	ic coupled	thermal/struc	tural	
Elevation : Above	eground	~	✓ 2D be	am-stick seismic	5	☑ 3D shell structural				
				1						
nk Definition Load Pres	tress Insulati	ons Support (3D)	Seismic Ground							
addara bead Loading of		ine section in granted	Manualua	Minumbus						
Load type	Notation	Dimension [m]	[kN/m ² , kN/m]	[kN/m ² , kN/m]	Description	- Liqui	d Bottom			
iquid bottom	q_1_liquid	42.1	183.662	183.662	Liquid_q1	Erqui	a bottom			
iquid bottom	q_2_liquid	0.0	0.0	0.0	Liquid_q2					
iquid bottom	q_3_liquid	0.0	0.0	0.0	Liquid_q3					
iquid bottom	q_4_liquid	0.0	0.0	0.0	Liquid_q4					
Gas Pressure	P_g	0.0	29.0	29.0	GasPres					
ive load	q_r_live	0.0	1.2	0.0	LiveLoad					
Snow load	q_r_snow	0.0	1.2	0.0	SnowLoad					
Fest load (Liquid bottom)	P_hydro	42.1	183.662	0.0	Hydrosta					
Fest load (Liquid wall)	P_hydro	0.0	0.0	0.0	Hydrosta	-	L	1		
Test load (Pneumatic)	P_pneu	0.0	1.2	0.0	Pneumat			q1		q2 q
						C-14				
						Set	2010	Set deladits		

Fig 73 Tank Definition Dialog (Load – Structural Variable Loading)

Structural Variable Loading

Defines the structural variable loadings to consider in analysis.

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

Tank Definition							>
Tank type		Target models t	o build				
Material : Concrete	~	🔽 2D axisyn	metric structural	🗹 2D axisymmet	ric c	oupled thermal/structural	
Elevation : Aboveground	~	☑ 2D beam-	stick seismic	☑ 3D shell struct	ural		
Tank Definition Load Prestress Insulation	ons Support (3D) Se	ismic Ground					
Structural Dead Loading Structural Variab	le Loading Thermal L	_oading					
Load type	Length [m]	Temperature [C]	Convective coefficient [J/m ² .s.C]	Type of boundary		Description	
Initial Temperature (Structure)	0.0	15.1	0.0	Prescribed	*	Initial Temperature of Structure	
Initial Temperature (Soil)	0.0	15.1	0.0	Prescribed	•	Initial Temperature of Soil	
Soil Bottom Depth & Temperature	25.0	15.1	0.0	Prescribed		Soil Bottom where Temperature is constant	
External Temperature	0.0	25.6	25.0	Convection	•	External Temperature	
Liquid Temperature	0.0	-170.0	166.47	Prescribed	٠	Liquid Temperature	
Base Heating	0.0	5.0	0.0	Prescribed	•	Base Heating	
Spillage 1	38.263	-170.0	166.47	Prescribed	٠	Spillage 1	
Spillage 2	0.0	-170.0	166.47	Prescribed	•	Spillage 2	
Spillage 3	0.0	-170.0	166.47	Prescribed	٠	Spillage 3	
Spillage 4	0.0	-170.0	166.47	Prescribed	•	Spillage 4	
Spillage 5	0.0	-170.0	166.47	Prescribed	•	Spillage 5	
Set zero Set defaults	* The temperature	for base heating wi	I only be considere	ed if a value other than ze	ero is	defined.	_
	Name	Tnk2				✓ → (new)	
						OK Cancel Apply	Help

Fig 74 Tank Definition Dialog (Load – Thermal Loading)

Thermal Loading

- □ Load 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

Tank Definiti	on									×
Tank type Material : Elevation :	Concre	te ground	~ ~	Target more 2D a ☑ 2D b	dels to build xisymmetric struc eam-stick seismi	tural 🗹 :	2D axisy 3D shell	rmmetric coupled t structural	hermal/structural	
Tank Definition Vertical pres Total tendo	tress n force (short te	ess Insulations	Support (3D)	Seismic Ground	d	Total tendo	n force (long term, [kN])	754.056E3	
2D Conver	sion [kN/m²] nversion [kN/m]:	3.67219E3 2.75414E3	2.493	375E3	2D Convers 3D shell co	ion (kN/	/m²] 1 [kN/m] :	2.49375E3 2.75414E3	
Section ID	estress Low el. [m]	High el. [m]	Load length [m]	Prestress load short term [kN/m²]	Prestress load long term [kN/m²]	Description	^	Prestress load	Vartical Practrace	
BaseSlab	0.0	0.0	1.2 3.8	370.275 319.291	370.275 319.291	BaseSlab				
2	3.8	7.4	3.6	205.796	205.796	Lot2				
4	11.0	11.0	3.6	180.432	180.432	Lot4			Horizontal Prestress	
5	14.6 18.2	18.2 21.8	3.6 3.6	154.656	154.656 154.656	Lot5 Lot6	-		Base Prestress	
7	21.8	25.4	3.6	128.88	128.88	Lot7	~		Vertical Prestress	
Set zero	Set defa	nults A	dd F	Remove						
			N	ame Tnk2				~	÷ (new)	
									OK Cancel Apply He	lp

Fig 75 Tank Definition Dialog (Prestress)

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.
- □ Low el. [m]: 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.
- □ **High el. [m]:** 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.

Support (3D)

Definition															
nk type				Та	arget models to	build									
Naterial :	Concrete		~		🗹 2D axisym	metric structura		🗹 2D ax	isymmetric c	oupled the	ermal/struc	tural			
Elevation :	Aboveground		~		☑ 2D beam-s	tick seismic		☑ 3D sh	ell structural						
	5														
Definition Load	Prestress In	sulations	Support (3	D) Seismic	Ground										
e Support															
															1
upport type		C	Sircumferen	tial Suppor	t.										
implified foundatio	n	~			Initial that	Number of	Ve	tical	Horizont ^			Cr	osswise p	iles 🚺 🗍	
			ID	R [m]	[degree]	piles	stiff	ness I/m1	stiffnes: [kN/m]	A	dd			- / / /	
									[(Del		:::	Circumferential piles	
No. cir : 1	184		1.	30.7	0.0	50	523.0	18E3 4	2.297E3				•		
No. cross : 2	213		2 4	40.8	0.0	60	523.0	18E3 4	2.297E3	Set	t zero		1		
ΣX ² Cir: 1	156.1965E3								~	Set	lofoulto		~	•	
ΣX ² Cross : 6	0 745750		3 4	14 9	0.0	68	523.0	18F3 4	2 297F3 🎽	Serd	leiaults				
rosswise support	stiffness		<	ffness [kN/r	n] 523.0°	18E3	Horizor	ntal stiffne:	> ss [kN/m]	42.297	E3				
rosswise support Grid wizan X coordinates (U	stiffness d Jnits: m)		<	ffness [kN/r	n] 523.0 ⁻	18E3	Horizor	ntal stiffne: : m)	> ss [kN/m]	42.297	E3			Add ook me	
Grid wizan X coordinates (L	stiffness d Jnits: m)	P4	Vertical sti	ffness [kN/r P6	n] 523.01	Y coordinat	Horizor tes (Units P2	ntal stiffne: : m) P3	> ss [kN/m] P4	42.297	E3	P7	^	Add column	J
Grid wizan X coordinates (U P1 P2 0.0 4.2	Jnits: m) P3 8.4	P4 12.6	Vertical stil P5 16.8	ffness [kN/r P6 21.0	n] 523.0 P7 ^ 25.2	Y coordinal P1 0.0	Horizor tes (Units P2 0.0	ntal stiffne: : m) P3 0.0	> ss [kN/m] P4 0.0	42.297 P5 0.0	E3 P6 0.0	P7 0.0	^	Add column Add row	
Crid wizar Crid wizar X coordinates (U P1 P2 0.0 4.2 0.0 4.2	d Jnits: m) P3 8.4 8.4	P4 12.6 12.6	Vertical stil P5 16.8 16.8	Ffness [kN/r P6 21.0 21.0	n] <u>523.0</u> P7 25.2 25.2	18E3 Y coordinat P1 0.0 -4.2	Horizor P2 0.0 -4.2	ntal stiffne: : m) P3 0.0 -4.2	> ss [kN/m] P4 0.0 -4.2	42.297 P5 0.0 -4.2	E3 P6 0.0 -4.2	P7 0.0 -4.2		Add column Add row Del column	
Prosswise support Grid wizan X coordinates (L P1 P2 0.0 4.2 0.0 4.2 0.0 4.2	stiffness d Jnits: m) P3 8.4 8.4 8.4	P4 12.6 12.6 12.6	Vertical still P5 16.8 16.8 16.8	Frness [kN/r P6 21.0 21.0 21.0	n] 523.0 P7 ^ 25.2 25.2 25.2 25.2	18E3 Y coordinat P1 0.0 -4.2 -8.4	Horizor P2 0.0 -4.2 -8.4	ntal stiffne: : m) P3 0.0 -4.2 -8.4	> ss [kN/m] P4 0.0 -4.2 -8.4	42.297 P5 0.0 -4.2 -8.4	E3 P6 0.0 -4.2 -8.4	P7 0.0 -4.2 -8.4		Add column Add row Del column Del row	
Prosswise support Grid wizan X coordinates (L P1 P2 0.0 4.2 0.0 4.2 0.0 4.2 0.0 4.2 0.0 4.2	Jnits: m) P3 8.4 8.4 8.4 8.4	P4 12.6 12.6 12.6 12.6	 Vertical still P5 16.8 16.8 16.8 16.8 	Freess [kN/r P6 21.0 21.0 21.0 21.0 21.0	n] 523.0 P7 25.2 25.2 25.2 25.2 25.2	I8E3 Y coordinal P1 0.0 -4.2 -8.4 -12.6	Horizor P2 0.0 -4.2 -8.4 -12.6	P3 0.0 -4.2 -8.4 -12.6	> ss [kN/m] P4 0.0 -4.2 -8.4 -12.6	42.297 P5 0.0 -4.2 -8.4 -12.6	E3 P6 0.0 -4.2 -8.4 -12.6	P7 0.0 -4.2 -8.4 -12.6		Add column Add row Del column Del row Set zero	

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

Examples – User Inputs

Fank Definition								×
Tank time		Ternet m	odels to build					
Material Concret	e e		aviewmetric structural	2D avievmmetric coun	led thermal/structure			
Waterial.		20	heam-stick seismic	2D shell structural	ieu alemanaudule			
Elevation : Aboveg	round	~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~	beam-stick seisinic					
Tank Definition Load Prestre	ess Insulations Sup	port (3D) Seismic Grou	nd					
Base Support Foundation								
Foundation								
Туре	Include	Height/Thickness [m]	Section shape	D1 [m]				
Pile (Cir)		NA	Circular Hollow	0.8			D2	
Pile (Cross)		NA	Circular Hollow	0.7				
							D1	
						· ·	• •	
						U		
<				>				
Subgrade stiffness								
Verti	ical stiffness [MN/m/r	m²] 100.0	Horizo	ontal stiffness [MN/m/m²]	100.0			
Pile stiffnesses (when pile	s are relaced by spri	ng supports)						
Circumferential pil	les Vertica	I [kN/m] 523.0	18E3	Crosswise piles V	ertical [kN/m]	523.018E3		
	Hariza	atol [[/b]/m] 42.20	752		evizentel [kh/m]	40.00750		
	Horizor	42.23	723		orizontali [kiv/m]	42.297E3		
- Material properties are o	defined in the Materi	al tab						
- Pile heights and horizon	tal support will follow	w the inputs in the Grour	d tab					
Set zero Set d	efaults							
		Name Tnk2			✓ (new)			
					OK	Cancel	Apply	Help
					OK	Cancel	~Pbiy	Help

Fig 77 Tank Definition Dialog (Support (3D))

Support Type

Options are 'Use support (2D) conditions', 'Simplified foundation', or 'Detailed foundation'. If 'Detailed foundation' is selected, 'Foundation' tab will be appeared (See Fig 92)

Crosswise piles X Coordinates

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

Crosswise piles Y Coordinates

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

LNG Tank - Define Pile Locations						>
Number of piles in X Add row in Y Remove row	13	Row 1	r in Y Numb 13	per of piles in X		
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					
			ОК	Cancel	Apply	Help

Fig 78 Tank Definition Dialog (Support(3D) – 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.

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

Inner Tank Properties

Tank Definition												×
Tank type Material : Elevation :	Concrete Aboveground		~	Target models to build 2 D axisymmetric structural 2 D beam-stick seismic 3 D beam-stick seismic				ric coupleo ural	I thermal/s	tructural		
Tank Definition Load	Prestress In Non-Structur	al Masses Lun	port (3D) Seis	smic Ground								
Liquid Liquid density	480.0) [k	g/m³]		Liquid height	38.92	[m]					
Inner tank dimens	sion 42.1	[1	n]									
Inner tank geome	tric properties	2	3	4	5	6	7	8	^	Thickness 6	Height 6	
Thickness(m)	0.0361	0.0361	0.012	0.01	0.01	0.0	0.0	0.0		Thickness 5	Height 5	
Height(m)	3.08	27.0	3.86	6.12	0.0	0.0	0.0	0.0	~	Thickness 4	Height 4	
<								>		Thickness 3	Height 3	
Inner tank materia	Elastic modul (E, [N/m²])	lus Poisson's ratio (v)	Mass density [kg/m³]	Coefficier of therma expansion [/C]	nt Thermal I conductiv [J/m.s.C]	ity Heat capacity [J/m³.C]	Descriptio	n		Thickness 2 Thickness 1	Height 2 Height 1	
Inner Tank	200.0E9	0.3	7.85E3	10.0E-6	2.0	1.968E6	Inner Tank					
Set zero	Set defaults			·								
			Name	Tnk2					🗘 (nev	v)		
									ок	Cancel	Apply	Help

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

- □ Liquid density This defines the LNG density for convective and impulsive mass in seismic analysis.
- **Liquid height** This defines the LNG height from the top of the base slab.
- □ 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 geometric properties 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.

nk Definition				>
Tank type Material : Concrete Elevation : Aboveground	Target models to build 2D axisymmetric structura 2D beam-stick seismic		ral	
nk Definition Load Prestress Insulations Support Iner Tank Properties Non-Structural Masses Lumped	(3D) Seismic Ground Foundation			
Descriptions	IK N	lass [kg]		
Roof Liner + steel Roof Structure	1.	4E6		
Suspended deck & insulation of the suspended ce	iling 1	35.0E3		
Roof nozzles	4.	2.0E3		
Roof platform	44	00.0E3		
Roof pump & crane	34	0.0E3		
Roof piping and support	11	03.0E3		
Others	0.	0		
Total	2.	11E6		
Set zero Set defaults				
	Name Tnk2	 (new) 		
		ОК	Cancel Apply	Help

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

Non-Structural Masses

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

Examples – User Inputs

Tank Definition						×
Tank type	Ta	rget models to build				
Material : Concrete ~		2D axisymmetric structur	al 🔽 2D axisymme	tric coupled thermal/structu	ral	
Elevation : Aboveground ~		✓ 2D beam-stick seismic	✓ 3D shell struc	tural		
Tank Definition Load Prestress Insulations Support (3D)	Seismic	Ground				
Inner Tank Properties Non-Structural Masses Lumped Four	ndation					
				ſ	Use 3D support inputs	
Geometric properties						
Name	Exist	Area [m ²]	Shear area [m ²]	Moment of inertia [m ⁴]	Length [m]	
Pile (Lumped)		617.23	540 14	297 064F3	NA	
The (compety)		011120	010.11	207.00420		
Lumped isolator						
Lamportovator			150 052			
Total mass of lumped isolator [kg] = isolator mass	x numbe	r of base support =	100.0E3			
Lumped pile stiffnesses						
			225 9233E3			
[Vertical beam stick model] Vertical stiffness of p	ile/soil [M	N/m]	220.020000			
			225 9233E3			
[Horizontal beam stick model] Rotational stiffness	of pile he	ad [MNm/rad]	220.320020			
Sat zoro Sat defaulta						
Serveradits						
Na	ame Tnk	2		~ 🗘 (new)		
				ОК	Cancel Apply	Help
					16.0	

Fig 81 Tank Definition Dialog (Seismic-Lumped Foundation)

- □ Geometric properties Define geometric properties for piles which will be used to build a model for seismic analysis. Piles are to be modelled with a series of elements in a single line. Values for area, inertia and stiffness for 'Pile' should be for the total of all piles acting as a group.
- Lumped isolator Defines the total mass of lumped isolator in units of kg which will be used to build a model for seismic analysis.
- □ **Lumped pile stiffness** Defines the vertical stiffness of pile/soil in units of MN per metre and rotational stiffness of pile head in units of MN per metre rad which will be used to build a model for seismic analysis.

Ground

Lieval		boveground											
il Prop	erties	Prestress	ations Supp	on (3D) Seis	mic Ground								
tiffness	distribution:	Constant valu	e	~									
ayer lo.	Soil depth [m]	Thickness of layer [m]	Static kh [MN/m/m]	Static kv [MN/m/m]	Dynamic kh [MN/m/m]	Dynamic kv [MN/m/m]	Lumped kh [MN/m/m]	Description((CL.			
	0.0	0.0	0.0	0.0	0.0	0.0	0.0	Start of soil p				G.L.	
	-2.0	2.0	19.0793	0.0382	38.1586	0.0763	14.92E3	Backfill	Layer 1		Piles		
2	-4.0	2.0	32.9527	0.0659	65.9054	0.1318	25.769E3	Backfill	Layer 2				
3	-6.0	2.0	28.6317	0.0573	57.2634	0.1145	22.39E3	Backfill	Layer 3		1 L	ayer thickness.	
ł	-8.0	2.0	27.5563	0.0551	55.1125	0.1102	21.549E3	Backfill	Layer 4				
;	-10.0	2.0	30.2072	0.0604	60.4143	0.1208	23.622E3	Backfill	Layer n				
5	-12.0	2.0	41.3977	0.0828	82.7954	0.1656	32.373E3	Silty Sand1					
7	-14.0	2.0	34.5307	0.0691	69.0614	0.1381	27.003E3	Silty Sand1	Add	Set zero			
								····· · · · · · · · · · · · · · · · ·	Remove	Set default	s		

Fig 82 Tank Definition Dialog (Ground-Soil Properties)

Soil Properties

- □ Soil depth[m]: Defines the level (elevation) of a soil layer with respect to the pile head which is at a location of zero. The value should be negative.
- **Thickness of Layer:** Defines the thickness of each layer. The value should be positive.
- **Static Kh:** Defines the static horizontal soil spring stiffness per unit length.
- **Static Kv:** Defines the static vertical soil spring stiffness per unit length.
- **Dynamic Kh:** Defines the dynamic horizontal soil spring stiffness per unit length.
- **Dynamic Kv:** Defines the dynamic vertical soil spring stiffness per unit length.
- **Lumped Kh:** Defines the lumped horizontal soil spring stiffness per unit length.
- □ Static vertical stiffness: Defines the static vertical stiffness which is applied to pile toe.

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

2D Axisymmetric Static Structural Analysis

User Inputs

The required user inputs for this model are as shown in [Fig 98].

Ik Definition Ank type Material : Concrete Elevation : Aboveground Ak Definition Load Prestress Sab and Roof Wall and Ring beam Mat Sase slab (Units: m) Dircular part length (L_inner)	Target models to build 2 D axisymmetric structura 2 D beam-stick seismic erials Support (20)	2D axisymmetric coupled thermal/structural 3D shell structural
ank type Material : Concrete Elevation : Aboveground k Definition Load Prestress] se Slab and Roof Wall and Ring beam Mat Sase slab (Units: m) Dircular part length (L_inner)	Target models to build D axisymmetric structura 2D beam-stick seismic	2D axisymmetric coupled thermal/structural 3D shell structural
Material : Concrete Elevation : Aboveground the Definition Load Prestress] se Siab and Roof Wall and Ring beam Mat Sase Siab (Units: m) Dircular part length (L_inner)	v 2D axisymmetric structura 20 beam-stick seismic erais Support(20)	2D axisymmetric coupled thermal/structural 3D shell structural
Elevation : Aboveground k Definition Load Prestress se Slab and Roof Wall and Ring beam Mail 3ase slab (Units: m) Sircular part length (L_inner)	arials Support (2D)	☐ 3D shell structural
nk Definition Load Prestress Se Slab and Roof Wall and Ring beam Mat Sase slab (Units: m) Circular part length (L_inner)	erials Support (2D)	
ik Definition Load Prestress se Slab and Roof Wall and Ring beam Mar Base slab (Units:m) Circular part length (L_inner)	erials Support (2D)	
se slab and Roof Wall and Ring beam. Ma Base slab (Units: m) Dircular part length (L_inner)	enais Support (2D)	
Base slab (Units: m) Circular part length (L_inner)		
Sircular part length (L_inner)		
	39.8 CL	
Sircular part depth (D_inner)	1.2	
apered section length (W_t)	0.6	Linner
nnular part length (L_outer)	6.7	Deating
Annular part depth (D_outer)	1.5	
lase heating (D_heating)	0.386	
Base heating (L_heating)	46.5	
Ground level (D_ground)	0.9	Wt
loof (Units: m) tadius of inner roof (R_roof_)) tadius of outer roof (R_roof_o) teght from the top of the base sale to the spmost of the roof (R_Height) bistance of tapered section 1 (a11) bistance of tapered section 2 (a/2) Set zero Set defaults	86.406 86.306 56.2545 10.079 0.6	Rroot_o
	Name Trik2	~ [.≎] (new)

Fig 83 User Inputs for 2D Axisymmetric Static Analysis

The user dialog is available from LNG Tank>Create 2D Model > Structural as shown in [Fig 99].

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

Examples – User Inputs

LNG Tank - Static Analysis		×
Tank definition data	Tnk1 ~	
Model filename]
Saved model file path	C:\Users\ohsso\Documents\Lusas200\Projects\(2D).mdl	
Concrete element size (m)	0.2	
Steel element size (m)	0.2	
	OK Cancel Help	

Fig 84 User Dialog for 2D Axisymmetric Static Analysis

Meshing

Element Type

LUSAS elements 'QAX4M', which are suitable for a 2D axisymmetric model, are defined and assigned.

Element Size

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



Fig 85 Mesh division for a 2D Axisymmetric Model

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

Geometric Properties

No geometric properties are required for 2D axisymmetric model.

Material Properties

User defined material properties are assigned to the relevant surfaces.

Tank Definition								
Tank type Material : Conc Elevation : Abov	rete eground	~	Target models to	build metric structural tick seismic	☑ 2D axisy ☑ 3D shell	mmetric coupled ti structural	nermal/structural	
Tank Definition Load Pres	stress Insulations	Support (3D) Seis	smic Ground					
Base Slab and Roof Wall a	nd Ring beam Mat	erials Support (2D)					
Material ID	Elastic modulus (E, [N/m ²])	Poisson's ratio (v)	Mass density [kg/m²]	CTE (alpha, [1/C])	Thermal conductivity [J/m.s.C]	Heat capacity [J/m²/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	
Pile (Cir)	35.0E9	0.2	2.5E3	10.0E-6	0.0	0.0	Pile (Cir)	
Pile (Cross)	35.0E9	0.2	2.5E3	10.0E-6	0.0	0.0	Pile (Cross)	



This can be found from LUSAS Modeller as shown in [Fig 104].

Attributes 🔷 🕈 🕈	Isotropic	×
SGr & Att (C. An & Uti IRe (D. La test(2D).mdl Attributes (95) Mesh (23) Material (4)	□Plastic □Creep □Damage □Shrinkage □Viscous Elastic	Two phase Ko Initialisation
Gustropic (1) A ::BasoSlab A ::BasoSlab A ::Roof A ::Roof Gustroports (12) A ::Rie 1 A ::Rie 1 A ::Pile 1 A ::Pile 2 A ::Pile 2 A ::Pile 3 A ::Pile 4 A ::Pile 5 A ::Pile 9 A ::Pile 10 A :::Pile 10 A :::Pile 10 A :::X: Fixed Coacting (54) Coacting (54) Coacting (54) Coacting (54) Coacting (54) Coacting (54) A ::Pitchad A ::Pitchad	□ Dynamic properties ☑ Thermal expansion Poisson's ratio Mass density Coefficient of thermal expansion	Value 35.0E9 0.2 2.5E3 10.0E-6
 Action/Action Structure (52) A: Steel Structure, q1 A: Steel Structure, q1 A: Steel Structure, q1 A: Steel Structure, q1 A: Liner, Dass1 A: Liner, Dass6 	Name BaseSlab	(1)

Fig 87 Material Properties for a 2D Axisymmetric Model

Support Conditions

Tank Definition					×
Tank type			Target mo	dels to build	
Material :	Concrete	~	🗹 2D a	axisymmetric structural	2D axisymmetric coupled thermal/structural
Elevation :	Aboveground	~	☑ 2D t	peam-stick seismic	☑ 3D shell structural
				al.	
Rese Sieh and Res	Moll and Ring ba	am Materiala Sup	D) Seismic Groun	a	
Support type	vivali and King be	ann Materials Oup	port (2D)		
Pile Support	~		Up	odate from Support(3D)	
Pile stiffnesses					
Spring ID	Radius [m]	Vertical stiffness [MN/m/rad]	Horizontal stiffness [MN/m/rad]	Description	Pile Supports
1	4.2	592.0	47.9	Pile	
2	8.4	1.0524E3	85.1	Pile	
3	12.6	1.5987E3	127.7	Pile	
4	16.8	2.1049E3	170.2	Pile	
5	21.0	2.6311E3	212.8	Pile	
6	25.2	3.1573E3	255.3	Pile	
7	29.4	3.4962E3	282.7	Pile	
8	32.9	3.7295E3	301.6	Pile	Spring Number
9	36.7	4.6615E3	377.0	Pile	1 2 3
<		_		>	
Set zero	Set defaults	Add	Remove		
			Name Tnk1		~) 📚 (1)
					Close Cancel Apply Help

Fig 88 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 89 Pile Support for a 2D Axisymmetric Model

TEST CASE

If support type 'Regular Support' is chosen as shown in [Fig 107], the support definition will be as shown in [Fig 108].

Tank Definition				×
Tank type Material : Elevation :	Concrete ~ Aboveground ~	Target models to build	2D axisymmetric coupled thermal/structural 3D shell structural	
Tank Definition Load	Wall and Ring beam Materials Suppor	t (2D)		
Support type Distributed spring	support \vee Vertical stiffness	1.0E3 [MN/m/m ²]	Horizontal stiffness 2.0E3 [MN/m/m ²]	

Fig 90 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'.)

Attributes	▲ ů ×	LUSAS Vi	ew: test_2d(2)	D).mdl Window	1 X			Structural Supports					×
🐻 Gr 💑 Att 🚇 An 🥜 Uti 🛄 Re 🚺	La	×	0.0	10.0	20.0	30.0	40.0						
Attributes (85)	^	-	**					Analysis category 2D	Axisymr	netric			
Material (4)		0.0								Free	Fixed	Spri	ng stiffness
A 1:BaseSlab		8							v				2.059
- 🖧 2:Wall									^				2.02.0
- 3 3:RingBeam								Translation in	Y		0	•	1.0E9
-# 4:Root							· · · ·		-		-		
-& 1:Distributed Spring Support		8							4	۲			
- A 2:X Fixed		4					T		х	۲		0	
🖶 😂 Loading (54)								Barrier stars					
😑 📾 Discrete (2)								Rotation about	T	0			
😑 🖼 Patch (2)									z	۲	0	0	
• 19:LiveLoad		0											
Datch divisions		8						Hinge rotation		۲			
Structural (52)								Torsional warping				0	
& 1:Steel Structure g1													
A 2:Steel Structure_P								Pore pressure		۲			
-& 3:Steel Structure_qr													
-& 4:Liner_base1		0											
-& 5:Liner_Roof								spring sumess distribut	udon				
								Stiffness	_				
-& 7:SteelStructureRoof								Stiffness/unit lengt	th				
-& 8:Insul_q1													
-& 9:Insul_q2		0						Stiffness/unit area					
-# 10:Insul_q3		문)											
A 11:Insul_q9			Y					Lift-off >>					
-& 12:Insul_qr								Contraction					
A 13:Insul_Pressure			A					Contact >>					
- 45 14: Wall piping loading							II.						
Istuguid_q1(Max)		0	-	v				Name Distrik	And Cas				v 🖿 (1)
* 17 CarDenny (Mar)		•		^				Name Distribu	uteu Spr	ing supp	Int		• • (1)
 46 17:GasPressure(Max) 8 19:GasPressure(Min) 						2011-00-00 (04:00-00)	No. of Contraction						
 35 10:GasPressure(Pill) 21 Hudeostatic Test 													
* 22:Proumatic Test									Close		Cancel	App	ly Help
ee 22:Preumatic Test		1											



Loadings

A total of 17 loadcases is defined in the model.





Self Weight



Fig 93 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.





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 96 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 97 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 98 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 99 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 100 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 101 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.





Snow load

Snow load acts on the top surface of roof.



Fig 103 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 104 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 105 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 106 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	→ ‡ ×	× LUSAS View: test(2D).mdl Window 1 ×	
Grou 🖧 Attri 🕒 Anal	🦨 Utilit 💷 Rep 🗇 Layers	^{rs} Properties × 0	
test(2D).mdl Windo	le Copy	Contour Results Appearance	
Deformed mesh Deformed mesh Deformed mesh Contours : SX (Stress	PasteX Delete	Entity Stress	
- Annotation - Utilities	Geometry Mesh		
See View properties	 Attributes Labels 	d Transform Set None	Ī
	AnnotationUtilities	j	
	 Contours Vectors Deformed mesh 	Display on slice(s)	
	Diagrams Values	Close Cancel Apply Help	
	Properties	Maximum 4,4027E6 at node 16 of element 15	

Fig 107 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.





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.

Layers 👻	X LUSAS	View: test(2D).md	l Window 1 ×				
😵 Grou 💑 Attri 🕒 Anal 🥓 Utilit 🛄 Rep 🗊 Lay	ers ¥20.0	-10.0	0.0	10.0	20.0	30.0	40.0
© test(20).mdl ● test(20).mdl ● Contours : SX (Stress - Axisymmetric Solid) ● Annotation ● Utilize ● Values : S ● View prof Value Results Value Display Ently Ently Stress Component SX Location Averaged nodal Transform Set None		nalysis: Ana badcase: 1:5 esults file: tt ktity: Stress 8 8 9 9 7 7 3 5 2	Iysis 1 SelfWeight sst 2D_~Ari - Axisvmme Properties Value Results 2	Values Display Values Display es of selection Maxima Minima © Significant fit © Significant fit © Show trailing Sol	ys Threshold > 2.264661 < -3.59594 gures 6 -	Percentage 10.0 % Failure details Choose font End	×
Display on slice(s)		3	2 _{Pen} 19	÷ (Choose pen	0.0 °	
Close Cancel	Apply	Help	C	ose Can	cel Appl	y Help	1
Deformations No deformations drawn ○ Window summary Details ✓ View axes Details Defaults	0.0			×			

Fig 109 Value Display in a 2D Axisymmetric Solid Model

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



Fig 110 Values Displayed for Selected Nodes in a 3D Shell Model

Graph through 2D

Define a line from Geometry>Line>By Coords.



Fig 111 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 ×		
O By cursor ✓ Snap to grid Grid size 1.0	Loadcases and Extent	×
Grid offset (0.0, 0.0, 0.0) Generate new annotation line By selected line Straight line 106 Project line In Z direction	Loadcases ©[f::SelfWeight] O Active O All O Specified Select	
 By selected surface At location of existing graph 	Create new window for each loadcase Extent Visible model	
OK Cancel Help	< 뒤로(b) 다음(b) > 취소 도응말	

Fig 112 Graph Through 2D in a 2D Axisymmetric Solid Model (1)

B Barrilla anno ant	Resultant effects from 2D model Desultant effects from alice	Display Title SX in the Wall	X scale Automatic O Manual
Calculate distance as angle Arc centre Z 0.0	Meantaint effect and in ance Meantaint effect and in ance effect and	x Distance y SX ☑ Show grid ☑ Show symbols ☑ Comer labels ☑ Auto-update	min 0.0 max 1.0 Use logarithmic scale Scale factor 1.0 Y scale Manual min 0.0 max 1.0 Use logarithmic scale Scale factor 1.0
Width for corridor averaging 0.0		Name Graph for SX	Save in treeview Display no



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 114 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.

Slice Data	×	Display Graph	×
Calculate distance as angle Arc centre X 0.0 2 0.0 2 0.0	Resultant effects from 2D model Resultant effects from slice Mean normal stress Sz Bending stress Schutzer stress Actual shear stress Mean stress per radian Actual shear stress per radian Actual shear stress per radian	Display Title Resultant Effects X Thickness V Results Show grid Show symbols Corner labels Auto-update Include existing graphs ØGraph for SX	X scale Automatic Manual min 0.0 max 1.0 Use logarithmic scale Scale factor 10 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 now
< 뒤로(B) 다음(N)	> 취소 도움말	< 뒤로(B) 마침	취소 도움말

Fig 115 Graph Through 2D in a 2D Axisymmetric Solid Model (3)



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

Export Forces to Excel (2D)

Forces calculated can be exported using LNG Tank > Excel Tools> Export Forces.

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.

Examples – User Inputs

LNG Tank - Export Forces/Mome	ents to Excel (2D)				×
Output filename Working folder Save in	Static 2D Current User Defined C:\Users\ohsso\Documents\Lusas200\Projects\Static 2D_BaseSlab.xlsx				
Target		◯ Wall + Ringbeam		⊖ Roof	
Loadcases 1:SelWeight 2:Dead Loads of Steel Structure 3:Dead load of liner and steel rood 4:Dead load of steel structures or 5:Dead load of Insulation 6:Pressure on outer tank wall due 7:Wall piping loading 8:Liquid bottom(Max) 9:Liquid bottom(Max) 11:Gas Pressure(Max) 11:Gas Pressure(Max) 11:Ga	f the roof ≥ to insulation	^ ~	Range (X Coord Start : Finish : Interval :) 0.0 46.5 0.5	m m m
			ОК	Cancel	Help

Fig 117 Export Forces for a 2D Axisymmetric Solid Model (1)



Fig 118 Section Force Spreadsheet for Self Weight

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



Fig 119 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. However, if wall stages should be considered in staged construction analysis, 'Wall stages' should be defined in Tank Definition. If any wall stages are not defined, it is assumed that the wall is built all at once.

Tank Definition								×
Tank type Material : Elevation :	Concrete	~	Target models 2D axisy 2D beam	to build mmetric structural h-stick seismic	☑ 2D axisymmetric c ☑ 3D shell structural	oupled thermal/structural		
Tank Definition Lo Base Slab and Ro Wall and ring b Inside radius of Thickness of w	ad Prestress Insulations of Wall and Ring beam M eam (Units: m) concrete outer tank wall (I all base (T_bottom)	Support (3D) Se laterials Support (2 InsR)	43.2 1.1		Tringbeam	ricotean		
Thickness of w Height of wall (I Height of ringbe Height of ringbe Thickness of rir Slope height (R	st var ("_ver) all top (T_top) 4_wall) ham_2 (H_ringbeam_2) ham_1 (H_ringbeam_1) ggbeam (T_ringbeam) _sl_height)	Height / Stages Wall stage ID 1 2 3 4 5	Height (H) [m] 3 3 3 3 3 3 3	Stage Y/N ^ Y Y Y Y Y Y	Set defaults Clear grid Add	His His His His His	Hringbeam,2 Hringbeam,1	
Corner Protecti Corner protect Corner protect Corner protect Set zero	on (Units: m) ion start (H_bcp_s)* ion end (H_bcp_e)* ion thickness (T_bcp)* Set defaults	< Wall stag	0.155 Opening	OK Guidi Com gs Com	Remove Cancel ance for corner protection inp er protection start: 0.105 or 0. er protection end : 5.617 er protection thickness: 0.155	Apply Help uts based on the current insult 567 or 0.617 or 0.6915	tion data	
		Name	e Tnk1			✓ ★ (1) Close Car	icel Apply	Help

The user dialog is available by selecting the menu item LNG Tank> Create 2D Model> Staged Construction as shown in [Fig 159].

• Enter a model filename, set the element size to **0.2** m, check 'Self weight' and 'Structural loadings' for loads to apply. Set roof construction plan to 'Layered roof option 1', set 'Roof first stage thickness (ratio)' to 0.5, set 'Initial

	Tnk1	×					
del filename	Layered Roof Option 1						
ved model file path	C:\Users\ohsso\Documents	C:\Users\ohsso\Documents\LUSAS200\Projects\Layered Roof Option 1(StagedConstruction21					
Modeling options							
Concrete element size (m)	0.2	Steel element size (m) 0.2					
Loads to apply							
Self weight	Structural loadings	(Variable Loads : ○ Max ○ Min)					
Roof / Ringbeam		Construction Scenario - Layered roof option 1					
Roof / Ringbeam Roof construction plan Roof first stage thickness (Initial prestress for ringbea Initial prestress for slab (ra	Layered roof option 1 (ratio) 1.0 im (ratio) 0.5 tio)	Construction Scenario - Layered roof option 1 1 - Base / Wall / Ringbeam 2 - Ringbeam 1st PS 3 - Roof frame 1 / Inner work 4 - Roof frames 2,3 5 - Roof lower wet / Roof Lower complete 6 - Roof upper wet / Roof complete					

prestress for ringbeam (ratio)' to 0.5, 'Initial prestress for base (ratio)' to 0.5 and press **OK** to build the model.

Fig 120 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 3rd loadcase, the 'Activate' attribute is assigned to the circular part of the Base Slab.



Fig 121 Activate and Deactivate Assignment in the Model

The construction scenario is printed on the 'Staged construction analysis' dialog according to defined 'Roof construction plan', 'Roof 1st stage thickness (ratio)', 'Initial Prestress for Ringbeam (ratio) and 'Initial Prestress for slab (ratio).

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 161]. 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.

	Nonlinear & Transient			
Grou 🚱 Attri 💆 Anal 🖋 Utiliti 📑 Repo 🛄 Layers	Incrementation		Solution strategy	
a Structural analyses	Nonlinear		Same as previous loadcase	
🖨 🖿 Analysis 1	Incrementation	Manual ~	Max number of iterations	12
🖽 😑 Material	Starting load	0.1	Residual force porm	0.1
□-@ 1:Annular Part(Staged)	Starting load	0.1	Residual force florin	0.1
Supports	Max change in load factor	0.0	Incremental displacement	1.0
	May total land factor	1.0		A.d
- 1:Deactivate		210		Advanced.
- Nonlinear and Transient	Adjust load based on c	onvergence	Incremental LUSAS file output	
⊞-@ 2:BaseSlab 1st PS(Staged)	Iterations per increment	4		
⊞ 12 3:Circular Part(Staged)	tterations per increment		Same as previous loadcase	
⊞ 2 4:BaseSlab 2nd PS(Staged)	Displacement reset		Output file	1
⊞		Advanced		
⊞- ⊕ 6:Ringbeam 1st PS(Staged)			Plot file	1
	Time domain	Construction of the second	Restart file	0
- P 9: Poof Frame? (Staged temporan)		Consolidation		
- 10:Roof Frame3(Staged,temporary)	Initial time step	0.0	Max number of saved	0
11:Roof Lower Wet Concrete(Staged.temporary)		100.055	Log file	1
⊞-@ 12:Roof Lower Complete(Staged)	l otal response time	100.026		
B-@ 13:Roof Upper Wet Concrete(Staged,temporary)	Automatic time steppin	a	History file	1
⊞ 14: Roof Complete(Staged)				
⊞-@ 15:Ringbeam 2nd PS(Staged)		Advanced	Save a restart at the end of the	nis control
H-W 16:Vertical PS(Staged)	Common to all			
Horizontal PS(Staged) Page 12: Operating Stage (Staged)				
m-w 19.0perating Stage(Staged Long)	Max time steps of	or increments		
Nonlinear analysis options				
Model properties			OK Cancel	Help

Fig 122 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 : BaseSlab 1st PS

Self weight is assigned by using 'Gravity' loading. The initial prestress loading to the BaseSlab is added in Stage 2. If no prestress is defined for the slab, **Stage 2** will be the same as **Stage 1**.



Fig 123 Loadings for Stage 1~2

Stage 3 : Circular Part ~ Stage 4 : BaseSlab 2nd PS

2nd prestress loading to the BaseSlab is added in **Stage 4**. If no prestress is defined for the slab, **Stage 4** will be the same as **Stage 3**.



Fig 124 Loadings for Stage 3~4

Stage 5 : Wall Ringbeam ~ Stage 6 : Ringbeam 1st PS

At **Stage 5** Wall and Ringbeam are completed. The loading is the same with **Stage 4**. Initial Horizontal Prestress for the RingBeam is added in **Stage 6**, 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 **'Initial Prestress for Ringbeam (ratio)'** from the dialog input.



Fig 125 Loadings for Stage 5~6

Stage 7 : Roof Frame 1 ~ Stage 8 : Inner Tank Work

Stage 7 assumed that there could be a temporary load as temporary frame is installed for preparing the roof 1st concrete pour. The loading for Roof Frame 1 should be defined and assigned manually by the user.

Stage 8 assumed that there could be loadings while building inner tank. However, this loading should be defined and assigned manually by the user.



Fig 126 Loadings for Stage 7~8

Stage 9 : Roof Frame 2 ~ Stage 10 : Roof Frame 3

Stage 9 and **Stage 10** assumed that there could be temporary loads as temporary frame is installed for preparing the roof 1st concrete pour. The loading for 'Roof Frame 2' and **'Roof Frame 3'** should be defined and assigned manually by the user.



Fig 127 Loadings for Stage 9~10

Stage 11 : Roof Lower Wet Concrete ~ Stage 12 : Roof Lower Complete

Stage 11 assumes that the roof is being built and the poured concrete is acting as a loading on the ringbeam.

Stage 12 assumes that the lower part of the Roof is completed. At this stage the roof lower wet concrete loading assigned at **Stage 11** is removed and replaced with the body force of the lower part of the Roof.



Fig 128 Loadings for Stage 11~12

Stage 13 : Roof Upper Wet Concrete ~ Stage 14 : Roof Complete

Stage 13 assumes that the upper part of the roof is being built and the poured concrete is acting as a loading on the Lower part of the roof.

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



Fig 129 Loadings for Stage 13~14

The weight of the upper part of the roof is computed by the Wizard from the geometry as marked in [Fig 169]. 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 130 Load Definition for the Wet Concrete of Upper Part of the Roof

Stage 15 : Ringbeam 2nd PS ~ Stage 16 : Vertical PS

The remained RingBeam prestress is added at **Stage 15**. (Load factor is changed from 0.5 to 1.0). The structure is fully built at **Stage 15**, and the additional loading of the Vertical Prestress is added to **Stage 16**.



Fig 131 Loadings for Stage 15~16

Stage 17 : Horizontal PS ~ Stage 18 : Operating Stage

Horizontal Prestress is added to **Stage 17**. **Stage 18** models the operating (in-service) stage. All the loadings used in the 2D Axisymmetric Static Analysis Model are all included in this stage. The prestress loadings are defined with the values obtained from User Input dialog and <u>only the short-term prestress is applied</u>.



Fig 132 Loadings for Stage 17~18

Stage 19 : Operating Stage (Long)

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

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



Fig 133 Loadings for Stage 19 of the Operating Stage

The prestress loadings are defined with the values obtained from User Input dialog and <u>only the long-term prestress is applied</u>.

Load Combination

Looking at U-C1-1 from the sample design load combination at [Fig 172], 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
				1				Perma	nent									
	De	escription		Outer tank	:	Others	Cree Shri	ep and nkage		F	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 Frame 1	Roof Frame 2	Roof Frame 3	1st layer concrete	2nd layer concrete
1	U-C1-1		1.35		1				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	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]	1.00						1.00					1.50				

Fig 134 Example of a Design Load Combination

The 1st PS is introduced at Stage 6, hence the pure effect of 1st PS can be obtained by defining a load combination for 'Stage 6 -Stage 5'.



Fig 135 Loadings for Stage 5 and Stage 6

This can be achieved by specifying a load factor of -1 for Stage 5, and 1 for Stage 6 as illustrated in [Fig 175]. 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 136 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 176]. Other attributes such as 'Activate' and 'Loading' are also copied.



Fig 137 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 176].



Fig 138 User Inputs for 2D Axisymmetric Thermal Analysis

The user dialog is available from LNG Tank>Create 2D Model> Coupled Thermal /Structural as shown in [Fig 178].

• 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\LUSAS191\Projects\example_The	ermal.mdl					
Element size	0.2						
Lionon alzo	[m]						
✓ Include Soil							
✓ Include Structural Load							
	◯ Min						
Variable Loads to apply(")	<u> </u>						
- The chosen variable loads from the Tank Definition v	vill be used for Operating Condition.						
(*) These parameters are read from the [Structural Load	ng Definition] tab of the tank definition attribute, based on the N	Max or Min column as selected above.					
Spillage Loading							
Application target above Corner Protection	 1st Wall Insulation layer 	◯ Wall					
Radius of inner tank outer surface(*)	42.1361 [m] Liquid density(*)	480 [kg/m³]					
(*) These parameters are read from the [Seismic] > [Inne	Tank Properties] tab of the tank definition attribute if available.						
Spillage duration time for each spillage height							
Spillage 1 10.0 [hour]	Spillage 2 10.0 [hour] Spillage 3	10.0 [hour]					
Spillage 4 10.0 Ibourt	Spillage 5 10.0 (hour)	[]					
[loui]							
	ОК	Cancel Help					
	L						

Fig 139 User Dialog for 2D Axisymmetric Thermal Analysis

Meshing

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



Fig 140 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.

Isotropic		×	Isotropic	×
Elastic Thermal Plastic Creep Damage Elastic Thermal	Shrinkage Viscous	Two phase Ko Initialisation	Elastic Thermal Plastic Creep Damage Shrinkage Viscous Two phase Elastic Thermal	Ko Initialisation
☐ Dynamic properties ✓ Thermal organison Dia Dia Dia Dia Dia Dia Dia Dia	ing's modulas servir into s stimativ influent of thermal expansion	Volao 30.69 02 2.553 10.06-6	Phase dunge state Mate Phase dunge state Phase dunge state Phase dunge state Phase dunge state Phase dunge	/aluo 20 5756 10
Name BaseSlab		~ (1)	Name [BsseSlab 🗸	5 (1)
	Close Can	cel Apply Help	Close Cancel App	aly Hdp

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

Fig 141 Material Properties of BaseSlab for a 2D Axisymmetric Thermal Analysis Model

Support Conditions

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

Loadings

Thermal Analysis > Initial Conditions

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



Fig 142 Initial Soil Temperature in a 2D Axisymmetric Thermal Analysis Model

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



Fig 143 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 144 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 145 Base Heating Temperature in 2D Axisymmetric Thermal Analysis Model

The Base heating temperature is assigned to selected lines as shown in [Fig 187]. A separate line is defined according to length of base heating defined in Tank Definition.



Fig 146 Base Heating Temperature in 2D Axisymmetric Thermal Analysis Model Refer to the section entitled *Examples – User Inputs* :

2D Axisymmetric Thermal Analysis for more information.

Thermal Analysis > Spillage Conditions

Spillage temperature is defined as a Prescribed Temperature and assigned to the inner face of the corner protection up to the 'Corner protection end' and to the inner surface of the 1st layer of the wall insulation above the corner protection. The same temperature loadings are assigned as well in spillage conditions.



Fig 147 Spillage Temperature in 2D Axisymmetric Thermal Analysis Model

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 190].

ank Definition		
Tank type	Target models to build	
Material : Concrete Elevation : Aboveground	ZD axisymmetric structural 2D beam-stick seismic	2D axisymmetric coupled thermal/structural 3D shell structural
ank Definition Load Prestress Support (3) asse Slab and Roof Wall and Ring beam Mi Base slab (Units: m) Circular part length (L_Inner) Circular part depth (D_Inner) Tapered section length (W_1) Annular part length (L_outer) Annular part depth (D_outer)	D) aterials Support (2D) 39.8 1.2 0.6 6.7 1.5 Dinner	Linner Louter Desting GL Desting Doute T
Base heating (D_heating) Base heating (L_heating) Ground level (D_ground) Roof (Units: m)	0.386	↔ Dground Wt
Radius of outer roof (R_rood_)) Radius of outer roof (R_rood_o) Height from the top of the base siab to the topmost of the roof (R_Height) Distance of tapered section 1 (sl1) Distance of tapered section 2 (sl2)	B6.406 B6.906 56.2545 10.079 0.6 Troot ↓ Rroot ↓ Rroot ↓	Rroot,o
Set zero Set defaults		
	Name Tnk1	~ (1)
		OK Cancel Apply Help

Fig 148 User Inputs for a 3D Shell Model

nk type	0	Target models to build		
Material :	Concrete	 2D axisymmetric structural 	2D axisymmetric coupled thermal/structural	
Elevation :	Aboveground	✓ 2D beam-stick seismic	☑ 3D shell structural	
Definition	.oad Prestress Support (3D)	ismic Ground		
er Tank Prope	erties Non-Structural Masses Lum	ped Foundation		
Roof Ring Be	eam Wall Base Slab Inner Stee	I Tank		
Descriptions		Mass	[kg]	
Roof Liner + s	steel Roof Structure	1.4E6		
Suspended d	deck & insulation of the suspender	d ceiling 135.0	E3	
Roof nozzles		42.0E	3	
Roof platform	1	400.01	E3	
Roof pump &	crane	30.0E	3	
Roof piping ar	ind support	103.0	E3	
Others		0.0		
Total		2.11E	6	
Set zero	Set defaults			
		Name Tnk1	~ 🗘 (1)	

Fig 149 User Inputs for a 3D Shell Model including non-structural masses to Eigenvalue Analysis

The user dialog is available from the LNG Tank>Create 3D Shell Model menu item.

- Enter the model file name, and set the element size to **2.0**, and the other values as shown in [Fig 192].
- Enter **10** for Number of Eigenvalues. Thick the 'Include non-structural masses' checkbox to include non-structural masses to eigenvalue analysis.
- Select 4 for Number of buttresses, input 1.0 (m) for Extruded thickness and 5.0(m) for Buttress width.

LNG Tank - Base Model for Design Check	×
Tank definition data Tnk1 Model filename Example Saved model file path C:\Users\ohsso Modeling options C:\Users\ohsso	V Documents\LUSAS200\Projects\Example.mdl
Element size (m) 2.0 Number of eigenvalue 10	 ☐ Half symmetric model ☐ Include temporary opening ✓ Include non-structural masses in the eigenvalue analysis
Concrete Tank Options Buttress Number of buttress Extruded thickness Buttress width 5.0 Roof / Ringbeam Roof construction plan Single layered ro Roof first stage thickness (ratio) 0.5 Initial prestress for ingbeam (ratio) 0.5	Construction Scenario - Single layered roof 1 1 - Base / Wall / Ringbeam 2 - Ringbeam 1st PS 3 - Roof frame 1/ Inner work 4 - Roof frames 2,3 5 - Roof wet / Roof complete 6 - Ringbeam 2nd PS of 1 V 7 - Wall vertical PS 8 - Wall horizontal PS
	OK Cancel Help

Fig 150 User Dialog for a 3D Shell Static Analysis Model

Mesh

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



Fig 151 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.



Fig 152 Element Local Axis in a 3D Shell Model

Geometric Properties

Geometric properties are defined as per user inputs. [Fig 195] illustrates how properties were defined for varying sections at the edge of the roof. The **variation dataset** can be reviewed from the Utilities **d** treeview.



Fig 153 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 154 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 197].



Fig 155 Material Properties in a 3D Shell Model

Support Conditions

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

				Та	irget mod	dels to b	uild							
aterial :	Concrete		~						ixisymmetric coupled thermal/structural					
levation :	Abovegrour	ıd	~		☑ 2D beam-stick seismic			☑ 3D shell structural						
Definition L	oad Prestress	Support (3D)	Seismic	Ground										
e Support														
upport type		с	ircumferen	itial Suppor							1			
Simplified foundation ~		~	ID	R [m]	Initia [de	l theta gree]	Number of piles	Vertical stiffness [kN/m]		Horizont ^ stiffnes: [kN/m]	Add		Crosswi	se piles
lo. cir :	184		1	36.7	0.0		56	523.01	8E3 4	2.297E3	(Del		Circumferential piles
lo. cross :	213		2	40.8	0.0		60	523.01	8E3 4	2.297E3	Set zero			
EX ² Cir : EX ² Cross :	156.1965E3 63.7157E3		3	44 9	0.0		68	523 01	8F3 4	2 297F3 ~	Setd	efaults	Fr	
osswise sur	nort stiffness										-			
osswise sup Grid v	oport stiffness vizard		Vertical sti	ffness (kN/n	n] [523.018	E3	Horizon	tal stiffnes	ss [kN/m]	42.297	3		
osswise sup Grid v X coordinate	oport stiffness vizard es (Units: m)		Vertical sti	ffness [kN/n	n] [523.018	E3 Y coordinate	Horizon	tal stiffnes m)	ss [kN/m]	42.297	E3		
Grid v Grid v X coordinate	vizard es (Units: m) P2 P3	P4	Vertical sti	ffness [kN/n P6	n] [523.018	E3 Y coordinate	Horizon es (Units: P2	tal stiffnes m) P3	ss [kN/m] P4	42.2971 P5	E3	P7 ^	Add column
Grid v Grid v X coordinate P1 0.0	vizard P2 P3 4.2 8.4	P4 12.6	P5 16.8	P6 21.0	n] [P7 25.2	523.018	E3 Y coordinate P1 0.0	Horizon es (Units: P2 0.0	m) P3 0.0	P4 0.0	42.2971 P5 0.0	P6 0.0	P7 ^ 0.0	Add column Add row
Grid v Grid v X coordinate P1 0.0 0.0	vizard P2 P3 4.2 8.4 4.2 8.4	P4 12.6 12.6	P5 16.8 16.8	P6 21.0 21.0	n] P7 25.2 25.2	523.018	Y coordinate P1 0.0 -4.2	Horizon es (Units: P2 0.0 -4.2	m) P3 0.0 -4.2	P4 0.0 -4.2	42.2971 P5 0.0 -4.2	P6 0.0 -4.2	P7 ^ 0.0 -4.2	Add column Add row Del column
Grid v Grid v X coordinate P1 0.0 0.0 0.0	popot stiffness vizard es (Units: m) P2 P3 4.2 8.4 4.2 8.4 4.2 8.4	P4 12.6 12.6 12.6	Vertical still P5 16.8 16.8 16.8	Ffness [kN/n P6 21.0 21.0 21.0	P7 25.2 25.2 25.2	523.018	E3 Y coordinate P1 0.0 -4.2 -8.4	Horizon es (Units: P2 0.0 -4.2 -8.4	m) P3 0.0 -4.2 -8.4	P4 0.0 -4.2 -8.4	42.2971 P5 0.0 -4.2 -8.4	P6 0.0 -4.2 -8.4	P7 ^ 0.0 -4.2 -8.4	Add column Add row Del column Del row
Grid v Grid v X coordinate P1 0.0 0.0 0.0 0.0 0.0	P2 P3 4.2 8.4 4.2 8.4 4.2 8.4 4.2 8.4	P4 12.6 12.6 12.6 12.6 12.6	P5 16.8 16.8 16.8 16.8 16.8	P6 21.0 21.0 21.0 21.0 21.0 21.0	P7 25.2 25.2 25.2 25.2 25.2	523.018	E3 Y coordinate P1 0.0 -4.2 -8.4 -12.6	Horizon P2 0.0 -4.2 -8.4 -12.6	m) P3 0.0 -4.2 -8.4 -12.6	P4 0.0 -4.2 -8.4 -12.6	42.2971 P5 0.0 -4.2 -8.4 -12.6	P6 0.0 -4.2 -8.4 -12.6	P7 ^ 0.0 - -4.2 - 8.4 - 12.6	Add column Add row Del column Del row Set zero
Grid v Grid v X coordinate P1 0.0 0.0 0.0 0.0	poport stiffness vizard P2 P3 4.2 8.4 4.2 8.4 4.2 8.4 4.2 8.4 4.2 8.4	P4 12.6 12.6 12.6 12.6 12.6	P5 16.8 16.8 16.8 16.8	P6 21.0 21.0 21.0 21.0 21.0	P7 25.2 25.2 25.2 25.2 25.2 25.2	523.018	E3 Y coordinate P1 0.0 -4.2 -8.4 -12.6 +2.0 -12.6 +2.0 -2.4	Horizon P2 0.0 -4.2 -8.4 -12.6	m) P3 0.0 -4.2 -8.4 -12.6	P4 0.0 -4.2 -8.4 -12.6 1e e	42.2971 P5 0.0 -4.2 -8.4 -12.6 16 %	P6 0.0 -4.2 -8.4 -12.6 1e o	P7 0.0 4.2 -8.4 -12.6 4 c o >	Add column Add row Del column Del row Set zero Set defaults
Grid v Grid v X coordinate P1 0.0 0.0 0.0 0.0 0.0 0.0	P2 P3 4.2 8.4 4.2 8.4 4.2 8.4 4.2 8.4 4.2 8.4	P4 12.6 12.6 12.6 12.6	P5 16.8 16.8 16.8 16.8 16.8	P6 21.0 2	P7 25.2 25.2 25.2 25.2 25.2 25.2 25.2	523.018	T coordinate P1 0.0 -4.2 -8.4 -12.6 -4.2 -8.4 -12.6 -4.2 -6 -4.2	Horizon es (Units: P2 0.0 -4.2 -8.4 -12.6 40 0	m) P3 0.0 -4.2 -8.4 -12.6	P4 0.0 -4.2 -8.4 -12.6	P5 0.0 -4.2 -8.4 -12.6 48.0	P6 0.0 -4.2 -8.4 -12.6 4e o	P7 ^ 0.0 -4.2 -8.4 -12.6 *	Add column Add row Del column Del row Set zero Set defaults
Grid v Grid v P1 0.0 0.0 0.0 0.0 0.0	vizard stiffless es (Units: m) P2 P3 4.2 8.4 4.2 8.4 4.2 8.4 4.2 8.4 4.2 8.4	P4 12.6 12.6 12.6 12.6	P5 16.8 16.8 16.8 16.8 16.8	P6 21.0 21.0 21.0 21.0 21.0	P7 25.2 25.2 25.2 25.2 25.2 25.2 25.2	523.018	T coordinate P1 0.0 -4.2 -8.4 -12.6 -12.6 -12.6 -12.6	Horizon es (Units: P2 0.0 -4.2 -8.4 -12.6 4e o	m) P3 0.0 -4.2 -8.4 -12.6 4e o	P4 0.0 -4.2 -8.4 -12.6 4e o	42.2971 P5 0.0 -4.2 -8.4 -12.6 16 °	F6 0.0 -4.2 -8.4 -12.6 4€ 0	P7 ∧ 0.0 -4.2 -8.4 -12.6 	Add column Add row Del column Del row Set zero Set defaults
Coordinate Coordi	yopt stiffres es (Units: m) P2 P3 4.2 8.4 4.2 8.4 4.2 8.4 4.2 8.4 4.2 8.4 4.2 0.1	P4 12.6 12.6 12.6 12.6	Vertical still P5 16.8 16.8 16.8 16.8	P6 21.0 2	P7 25.2 25.2 25.2 25.2 25.2 25.2 25.2	523.018	E3 Y coordinate P1 0.0 -4.2 -8.4 -12.6 -12.6 <	Horizon es (Units: P2 0.0 -4.2 -8.4 -12.6 40 0	P3 0.0 -4.2 -8.4 -12.6 10 °	P4 0.0 -4.2 -8.4 -12.6 4.6 o	42.2971 P5 0.0 -4.2 -8.4 -12.6 1e e	P6 0.0 -4.2 -8.4 -12.6 4¢ °	P7 0.0 -4.2 -8.4 -12.6 4 c o v	Add column Add row Del column Del row Set zero Set defaults

Examples - User Inputs

ributes 🗸 🗸 🖓 🗸 🗸	Structural Supports					
Grou Arttri UAnal Utilit IRep ULayers Example.mdl Arthoutes (178) Head Mesh (75)	Analysis category 3D					
🗃 🖻 Geometric (17)			Free	Fixed	5.	ring stiffnoss
Material (11)			Free	Fixed	Sp	nng sumness
Supports (6)		х	\bigcirc	\bigcirc	•	42.297E6
2:Circumferential Base Support (R = 36.7)	Translation in	v	\bigcirc	\bigcirc		42 297E6
🚓 3:Circumferential Base Support(R = 40.8)		· -	0	0		42.20720
• 4:Circumferential Base Support(R = 44.9)		z	\bigcirc	\bigcirc		523.018E6
🔸 5:2 Fixed		х	۲	\bigcirc	0	
e-@ Loading (57)	Rotation about	Y	۲	0	0	
→ Siscrete (2)		-		0		
\$ 21:LiveLoad(Roof)		2	•	0		
- & 23:SnowLoad(Roof)	Hinge rotation		۲	$^{\circ}$		
	Torsional warping		۲	0	0	
- \$ 1:Steel Structure_q1	Dava nyaasuwa			0		
& 2:Steel Structure_P	Pore pressure			0		
 4:Liner_base1 4:Liner_koof 5:Liner_Roof 6:Liner_Wall 7:SteelStructureRoof 8:Insul_q1 4:Linsul_q2 10:Insul_q3 4:Linsul_q4 12:Insul_q4 4:2:Insul_qa 4:Vall_piping loading 4:5:Liquid_q1(Max) 4:6:Liquid_q1(Max) 4:3:CasPressure(Max)(Base_Roof) 4:1:CasPressure(Max)(Base_Roof) 	Spring stiffness distributio Stiffness Stiffness/unit length Stiffness/unit area Lift-off >> Contact >> Name Crosswise	Dn	Support			× 🗘 (1)

➡

Fig 156 Support Definition in a 3D Shell Model

TEST CASE

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

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

Fig 157 Option for Half Model



Fig 158 Half Symmetric Model

Loadings

28 loadcases are defined in the model.



Fig 159 Loadcases Available in a 3D Shell Model

Self Weight



Fig 160 Self Weight in a 3D Shell 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 161 Dead Loads for Steel Structure in a 3D Shell 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 162 Dead Load of Liner and Steel Roof in a 3D Shell 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 163 Dead Load of Steel Structures on the Roof in a 3D Shell Model

Dead load of Insulation

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



Fig 164 Dead Load of Insulation in a 3D Shell 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 165 Insulation Pressure Load in a 3D Shell Model

Wall Piping Loading

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



Fig 166 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 167 Liquid Bottom Loading in a 3D Shell Model
Gas Pressure(Max/Min)

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



Fig 168 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 169 Live Load in a 3D Shell Static Analysis Model (Roof)

Snow load

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



Fig 170 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 171 Test Load (Liquid Bottom) in a 3D Shell Model

Test load (Pneumatic)

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



Fig 172 Test Load (Pneumatic) in a 3D Shell 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 173 Prestress Load in a 3D Shell Model

Wind Load

Wind loading can be added to 3D shell model using LNG Tank > Add Loading> Wind... The wall and roof is computed based on the selected design code. EN 1991-1-4 (2005), GB50009(2012) and ASCE 7-16 for design code are available. For the wall, separate loading datasets are defined for approximately each 1.0 m rise in height.

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

Fig 174 Wind Load in a 3D Shell Model



Fig 175 Wind Load in a 3D Shell Model

Viewing Results

Contours

The Layers 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 176 Selection for Contour Display in 3D Shell Model

Layers 🗸 🕈 🗸	LUSAS View: Example.mdl Wi	indow 1 ×		
🕼 Grou 🚴 Attri 🕒 Anal 🥓 Utilit 🛄 Rep 🔲 Layers	-20.0 -10.	0 0.0 10.0	20.0 30.0	
Cample mdl	0.0 000			
Properties	×			
Mesh Visualise Viterfarme Pen # 18 ÷ Viterfarme Pen # 10 ÷ Viterf	Choose pen Maximum shade 60.0 % hal edge e only Threshold 25.0 ° offset distance 6.0 mm tations only if selected % of elements 100.0 loar ✓ Set			
Close Cance	I Apply Help			

Fig 177 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 178 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 179 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.

Layers 🗸 🗸 🗸	LUSAS View: Example_3D.mdl Window 1 ×			
😨 Grou 🗞 Attri 🚇 Anal 🥓 Utilit 🛄 Repo 📴 Layers 👔	-120.0 -100.0 -80.0	-60.0 -40.0		40.0
Cample 3D.md Comple 3D.md C	Analysis: 01 Base Analysis Loadcase: 1:SelfWeight Results file: Example: 3D-01 Base Ana Entity: Fore/Moment - Thick Shell Component: Mx (Units: N.m/m)	lysis.mys	Benilis Transformation X	
view projekter Labels. Annotation Villites. Vectors. Vectors. Vectors. Vectors. Vectors. Volues. Villites.	175 3175E3 07 150 745E3 220 745E3 220 745E3 301 486E3 376 858E3 452 229E3 Maximum 486 964E3 at node 11357 of Minimum -191 38E3 at node 5706 of el	Contour Results Appearance Entity Farce/Moment - Thick SI Compound Mix Upplay Averaged model Transform Cetter Rome Dapping on disc(s) Dopain yos local direction	b to transformation applied (corouth Solver manual) out ass of element/node out coordinate of parent feature obtain ass default ass	
Deformations No deformations drawn	e.	Close Cancel App	No objects defined v (n) x = longitudinal v = transverse	
Window summary Details View axes Details Defaults		▼ # × Cyclable Items	OK Cancel Help	

Fig 180 Contour Display using Local Coordinate in a 3D Shell Model

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



Fig 181 Mt Contour in a 3D Shell Model

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





Fig 182 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 183 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 184 Nx Contours in a 3D Shell Model



Fig 185 Ny Contours in a 3D Shell Model

Values

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



Fig 186 Value Display in a 3D Shell Model

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



Fig 187 Values Displayed for Selected Nodes in a 3D Shell Model

Graph through 2D

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



Fig 188 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	K Loadcases and Extent	\times
By cursor Snap to grid Grid size 1.0 Grid offset (0.0.0.0.0) Generate new annotation line By selected line Straight line 1449 Project line Normal to screen By selected ourface At location of existing graph	Loadcases	
OK Cancel Help	< 뒤로(B) 다음(N) > 취소 도움말	

Fig 189 Graph Through 2D in a 3D Shell Model (1)

Slice Data			Resultant effects from 2D model	Dis	isplay Graph
Results	component		Resultant effects from slice		Tu Croch for Mt
Entity	Force/Moment	~	✓Mean normal stress Sz ✓ ✓ ✓ ✓ Bending stress		X Wall height Min 0.0 max 1.0
Component Transform	Nt Set LocalCoord late distance as angle X 0.0	~	⊠Normal stress Sz		v Mt Use logarithmic scale Show grid Show symbols Comer labels Auto-update Include existing graphs Manual min 0.0 max 1.0
Width for	Arc centre Y 0.0 Z 0.0 corridor averaging 0.0		~		Use logarithmic scale Scale factor 1.0 Name Graph for Mt for Self Weight ✓ Save in treeview ✓ Display now
	< 뒤로(B) 다음	(N) >	취소 도움말	-	< 뒤로(B) 마침 취소 도용말

Fig 190 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 191 Mt Graph for Sliced Line in a 3D Shell Model

Export Forces to Excel (3D)

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

- **Output file name** is for the name of the result spreadsheet.
- **Target** is for selecting members from which the results will be exported.

- □ Angles defines where slices should be taken in the model. Multiple angles can be defined by using a semi-colon (;) as a separator. (e.g. 10;20;30)
- □ Interval defines the distance between each value.

With **Self Weight** 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.

IS Tank - Export Forces/Moments to Excel (3D) Duput filename Example Vorking folder © Current User Defined Save in C:USerSichasolDocuments/LUSAS200/Projects/Example_Wall_Ringbeam.xtex Target Range Results to extract © Base slab @ Wall + Ringbeam @ Roof All Results to extract © Forces and Moments © Design results Utilisations No design code is enabled ULS © UtilPM © UtilShear © PM Capacity © Shear Capacity © UtilDecompression Depth Leadcases © Combinations only TSelf/Veight 2:Dead Loads of Sheel Structure 3:Dead Loads of Sheel Structure 3	IG Tank - Export Forces/Moments to Excel (3D)	
Dutput filename Example Vorking folder Current User Defined Save in CiUsersiohssolDocuments/LUSAS200Projects/Example_Wall_Ringbeam.xtsx Target Range Angles: 20 degree (eg. 10; 20; 30) Interval: 0.5 m Results to extract Forces and Moments Orseign results Utilisations No design code is enabled ULS ULS Combinations only Image: Combinations only Image: Combinations only Image: Combinations only Image: Combinations only Image: Combinations only Image: Image: Combinations only Image: Combinatio		
Vorking folder Current User Defined Save in C:UsersiohssolDocuments/LUSAS200/Projects/Example_Wall_Ringbeam.xisx Target Base slab Wall + Ringbeam Roof All Results to extract Forces and Moments Design results Utilisations No design code is enabled ULS UtiliSeat Combinations only Combinations o	Output filonomo	
Working folder • Current • Currents • Current	Zuput mename Example	
Save in C:UsersichssolDocuments/LUSAS200/Projects/Example_Wall_Ringbeam.xlsx Target Base slab Wall + Ringbeam Results to extract ✓ Forces and Moments ✓ Design results Utilisations No design code is enabled ULS ✓ Util/PM Combinations only 1:SetWeight 2:Dead load of finer and steel roof 5:Dead load of finer and steel roof 6:Dead load of finer and steel roof 8:Dead load of finer and steel roof 8:Dead load of finer and steel roof 8:Dead load of finer and steel roof 9:Dead load of finer and steel roof	Norking folder	
Target Base slab Wall + Ringbeam Roof All Angles: 20 degree (eg. 10; 20; 30) Interval: 0.5 (m) Exclude forces on the base slab at pile heads and wall Diameter of crosswise piles: 0.7 (m) Utilisations No design code is enabled ULS ULS Combinations only 12Deed Loads of Steel Structure 3Dead load of finer and steel roof 4Dead load of finer and steel roof <th>C:\Users\ohsso\Documents\LUSAS200\Project</th> <td>s\Example_Wall_Ringbeam.xlsx</td>	C:\Users\ohsso\Documents\LUSAS200\Project	s\Example_Wall_Ringbeam.xlsx
Base slab ● Wall + Ringbeam Roof All Angles: 20 degree (eg. 10; 20; 30) Interval: 0.5 [m] Results to extract 0.5 [m] ✓ Forces and Moments 0.7 [m] ✓ Design results 0.7 [m] Utilisations 0.8 [m] No design code is enabled 0.8 [m] ULS UtilShear PM Capacity Shear Capacity UtilDecompression Compression Depth Loadcases Combinations only Image: Image: Slicing Line 12:Dead load of fiser and steel roof Eload load of fiser and steel roof Eloads of of fisulation 0:Dead coad of fiser and steel roof Eloads of of fisulation Pressure on outer tank wall due to insulation ??/Wall piping loading Slicing Line Slicing Line 8.Liquid bottom(Min) Slicing Line Angle (Positive Direction 10:Gas Pressure(Max) Xaxis (0 Degree) Xaxis (0 Degree) 14:Test load Image (Liquid) Xaxis (0 Degree)	Target	Range
Results to extract Exclude forces on the base slab at pile heads and wall Design results Utilisations No design code is enabled ULS UtilPM UtilShear PM Capacity Shear Capacity UtilDecompression Depth Loadcases Combinations only ISeMVeight 2:Dead Loads of Steel Structure 3:Dead load of liner and steel roof 4:Dead load of steel structures on the roof 5:Dead load of Steel Structure 3:Iticias Pressure(Max) 11:Gas Pre	- ◯ Base slab	Angles : 20 degree (eg. 10; 20; 30) Interval : 0.5 [m]
✓ Forces and Moments Diameter of crosswise piles : 0.7 [m] ✓ Design results Diameter of crosswise piles : 0.8 [m] Utilisations No design code is enabled 0.8 [m] ULS ✓ UtilShear > PM Capacity ✓ Shear Capacity ✓ UtilDecompression Decompression Depth Loadcases ○ Combinations only ○	Results to extract	Exclude forces on the base slab at pile heads and wall
Diameter of circumferential piles : 0.8 [m] Utilisations No design code is enabled ULS VUtilPM VUtilShear VPM Capacity V Shear Capacity VUtilDecompression Depth Loadcases Combinations only ISelfWeight 2.Dead Load of Sheel Structure 3.Dead load of Insulation 8.Pressure (Max) 9.Liquid botom(Max) 9.Liquid botom(Max) 9.Liquid botom(Max) 11.Gas Pressure(Max) 11.Gas Pre	✓ Forces and Moments	Diameter of crosswise piles : 0.7 [m]
Utilisations No design code is enabled ULS UtiliShear PM Capacity Shear Capacity UtiliDecompression Depth Loadcases Combinations only ISelWeight 2:Dead Loads of Steel Structure 3:Dead Loads of Steel Structure 3:Dead Loads of Steel Structure 3:Dead Loads of Steel Structures on the roof 5:Dead load of Inset and steel roof 4:Dead load of Steel Structures on the roof 5:Dead load of Inset and steel roof 5:Dead load of Inset Inst wall due to insulation 7:Wall piping loading 8:Liquid bottom(Max) 9:Liquid bottom(Max) 11:Gas Pressure(Max) 11:Gas Pressure(Min) 12:Live load 13:Snow load 13:Snow load 13:Snow load 13:Snow load 13:Snow load 13:Snow load 14:Test load (Liquid)	└─ Design results	Diameter of circumferential piles : 0.8 [m]
1:SelfWeight 2:Dead Load of Steel Structure 3:Dead Load of Steel Structure 3:Dead Load of Steel Structures on the roof 4:Dead load of Steel Structures on the roof 5:Pressure on outer tank wall due to insulation 7:Wall piping loading 8:Liquid bottom(Max) 9:Liquid bottom(Min) 10:Gas Pressure(Min) 12:Live load 13:Snow load 14:Test load (Liquid)		Capacity VIIIDecompression Compression Depth
	ULS UtilPM UtilShear PM Capacity Shear	Capacity 🗹 UtilDecompression 🔽 Compression Depth
	ULS UtilPM UtilShear PM Capacity Shear Loadcases Combinations only ISelWaight 2:Dead Loads of Steel Structure 3:Dead loads of isel atructures on the roof 5:Dead ioad of iseluation 6:Pressure on outer tank wall due to insulation 6:Pressure on outer tank wall due to insulation 7:Wall piping loading 8:Liquid bottom(Max) 9:Liquid bottom(Min) 10:Gas Pressure(Max) 11:Gas Pressure(Max) 11:Gas Pressure(Max) 12:Live load 13:Snow load 14:Test load (Liquid)	Capacity UtilDecompression Compression Depth

Fig 192 Export Forces for 3D Shell Model (1)



Fig 193 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.

NG Tank - Export Forces/Moments to Excel (3D)	
	· · · · · · · · · · · · · · · · · · ·
Outout flances	
Cutput filename Example	
Norking folder O Current O User Defined	
C:\Users\ohsso\Documents\LUSAS200\Projects\	Example_Wall_Ringbeam.xlsx
Target	Range
	Angles · 0:45 degree (eg. 10: 20: 30.)
⊖ Base slab	Interval : 0.5 [m]
Results to extract	Exclude forces on the base slab at pile heads and wall
✓ Forces and Moments	Diameter of crosswise piles : 0.7 [m]
	Diameter of circumferential piles : 0.8 [m]
v boligi robulo	
Loadcases	
Seal Loads of Steel Structure 2:Dead Loads of Steel Structure 3:Dead load of liner and steel roof 4:Dead load of steel structures on the roof 5:Dead load of Insulation 6:Pressure on outer tank wall due to insulation 7:Wall piping loading 8:Liquid bottom(Max) 9:Liquid bottom(Min) 10:Gas Pressure(Max) 11:Gas Pressure(Min) 12:Live load	Y axis Slicing Line Angle (Positive Direction)
1:SelWeight ^ 2:Dead Loads of Steel Structure	Y axis Slicing Line Angle (Positive Direction) X axis (0 Degree)

Fig 194 Export Forces for a 3D Shell Model (2)





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 Model.



Fig 196 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.**

ank delinition data	Tnk1	×
/lodel filename		
Saved model file path	C:\Users\ohsso\Documents\	LUSAS200\Projects\.mdl
Modeling options		
Element size (m) 2.0	F	lalf symmetric model
	Ir	nclude temporary opening
Number of eigenvalue 10		nclude non-structural masses in the eigenvalue analysis
Number of buttress Extruded thickness Buttress width	4 ~ (m) 5.0 (m)	1 - Base / Wall / Ringbeam 2 - Ringbeam 1st PS 3 - Roof frame 1 / Inner work 4 - Roof frames 2,3 5 - Roof wet / Roof complete
Roof / Ringbeam		6 - Ringbeam 2nd PS
Roof construction plan	Single layered roof 1 🛛 👻	7 - Wall vertical PS
Roof first stage thickness (ratio)	0.5	5 - Wainforzontarr 5
Initial prestress for ringbeam (rat	io) 0.5	
	tio) 0.5	

Fig 197 Dialog for a 3D Shell Eigenvalue Analysis

Summary of Mass C	Immary of Mass Calculation										
DIMENSION											
Component	Dimension(m)										
Inner Tank Radius	42.1										
Tank Height	40.06										
LNG Height	38.92										
SUMMARY FOR M	ASS										
Component	Volume	Unit mass	Structural mass	Total mass	Equivalent unit mass						
	m³	kg/m ³	kg	kg	kg/m ³						
Roof	3,967	2,500	9,917,753	12,027,753	Not Used						
Ringbeam(upper)	524	2,500	1,310,993	1,310,993	2,500						
Ringbeam(lower)	463	2,500	1,156,758	1,156,758	2,500						
Wall & Buttress	9,976	2,500	24,940,428	25,764,428	2,583						
BaseSlab	8,719	2,500	21,797,085	24,925,085	2,859						
LNG	216,714	480	104,022,703	104,022,703	480						
Inner Tank	316	7,850	2,479,105	2,799,105	8,863						

Fig 198 Mass Summary for an Eigenvalue Analysis

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

Analyses 👻	φ×	Isotropic X
Image: State of the state	vers	Plestic Creep Damage Shrinkage Viscous Two phase Ko Initialisation
⊕. # 45:Roof Complete(Staged) ⊕. # 46:Roof Inside Load(Staged) ⊕. # 47:Ringbeam 2nd PS(Staged) ⊕. # 49:Vertical PS(Staged) ⊕. # 49:Vertical PS(Staged) ⊕. # 50:Final Short term(Staged) ⊕. # 51:Final Long term(Staged) ⊕. Monlinear analysis options		□ Dynamic properties Value ☑ Thermal expansion 35.0E9 Poisson's ratio 0.2 Mass density 2.85876E3 Coefficient of thermal expansion 10.0E-6
	+	Name BaseStab(Eigen) S (8)

Fig 199 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 the TreeView.



Fig 200 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.

			 Eigenvalues
Results			
Units	Model units	~	
Loadcases	18:Eigenvalue	~	
Sum ma	auon nacuor s articipation factors iss participation factors	Defailts	Precision (e) Significant figures Decimal places I show trailing zeros
✓ Display	reeview	berdarb	Threshold value N/A
Save in			
Display Save in Loadcases Available			Included
Display Save in Loadcases Available Eig	jenvalue	^	Induded Name

	LUSAS View: Example.mdl Window 1 Sum mass participation factors ×												
8	a] ∰ [b] am q q												
	Mode*	Sum Mass X	Sum Mass Y	Sum Mass Z	Sum Mass THX	Sum Mass THY	Sum Mass THZ	Frequency	Period				
1	1	0.487244E-9	0.780641E-9	0.26361E-18	5.75525E-12	3.59156E-12	0.108516	2.17985	0.458747				
2	2	0.606336	0.306421	0.856113E-15	2.29076E-3	4.53287E-3	0.108516	2.18599	0.457459				
3	3	0.912758	0.912757	1.0427E-15	6.82365E-3	6.82363E-3	0.108516	2.18599	0.457459				
4	4	0.912758	0.912757	35.8402E-12	6.82365E-3	6.82363E-3	0.108516	4.50674	0.22189				
5	5	0.913092	0.938632	36.4364E-12	0.0224938	7.02617E-3	0.108516	4.50793	0.221831				
6	6	0.938965	0.938966	40.1325E-12	0.0226963	0.0226955	0.108516	4.50793	0.221831				
7	7	0.938965	0.938966	0.104454E-9	0.0226963	0.0226955	0.108516	4.53905	0.22031				
8	8	0.938965	0.938966	0.487768	0.0226963	0.0226955	0.108516	4.56403	0.219105				
9	9	0.945443	0.987015	0.487768	0.0621274	0.0280112	0.108516	4.58906	0.21791				
10	10	0.993492	0.993492	0.487768	0.067443	0.0674431	0.108516	4.58906	0.217909				

Fig 201 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** and **Ground** 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													
ank type				Tarc	et models	to build							
Material :	Concrete		~		2D axisymmetric structural 2D axisymmetric coupled the						tructural		
Elevation :	Aboutogroupd								tural				
Elevation .	Aboveground												
nk Definition Loa	d Prestress S	upport (3D)	Seismic G	iround									
ier Tank Properti	es Non-Structur	al Masses Lu	mped Fou	Indation									
Liquid													
Liquid density	480.0	1 (ka/m³l			Liquid height	38.92	ſn	h				
Eiquid denoity						Elquid Holgin			4				
Inner tank dimer	nsion												
Inside radius	42.1												
monae radida			[m]										
nner tank geom	etric properties										. Т	1	
	1	2	3	4		5	6	7	8	^	Thickness 6	Height 6	
Thickness(m)	0.0361	0.0361	0.012	0.01	1	0.01	0.0	0.0	0.0		Thickness 5	Height 5	
Height(m)	3.08	27.0	3.86	6.12	,	0.0	0.0	0.0	0.0		Thickness 4	Height 4	
<										, ×	-	*	
											Thickness 3	Height 3	
nner tank mater	rial properties				Coefficient				-		Thickness 2	Height 2	
	Elastic modul	lus Poisson'	's Ma den	ss isity	of thermal	Conductiv	vity capacity	Descripti	on		Thickness 1	Height 1	
	(E, [wiii])	1000 (*)	[kg	(m³]	[/C]	[J/m.s.C]	[J/m ³ .C]		_				
Inner Tank	200.0E9	0.3	7.85	E3	10.0E-6	2.0	1.968E6	Inner Tar	ik				
Set zero	Set defaults												
			N	ame Tnk?						× = (ne	(v)		
				unie TTIKZ						(iie	,		

Fig 202 User Inputs 1 for Seismic Analysis

Non-Structural Masses

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

nk type	Target models to build			
Material : Concrete ~	2D axisymmetric structural	2D axisymmetric coupled thermal/structu	ural	
	✓ 2D beam-stick seismic	✓ 3D shell structural		
elevation : Aboveground ~				
Definition Load Prestress Support (3D) Seismic	Ground			
er Tank Properties Non-Structural Masses Lumped I	oundation			
Roof Ring Beam Wall Base Slab Inner Steel Tan				
Descriptions	Mass	[kg]		
Roof Liner + steel Roof Structure	1.4E6			
Suspended deck & insulation of the suspended ceil	ng 135.0E	3		
Roof nozzles	42.0E3	1		
Roof platform	400.0E	3		
Roof pump & crane	30.0E3			
Roof piping and support	103.0E	3		
Oth				
Others	0.0			
Total	2.11E6	i		
]	
Set zero Set defaults				

Fig 203 User Inputs 2 for Seismic Analysis

Lumped Properties

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

ank Definition								>
Tank type		T	arget models to build					
Material :	Concrete	~	2D axisymmetric	structural 2D axisym	metric coupled thermal/struct	ural		
Elevetien (Abovernmed		2D beam-stick se	eismic 🔡 3D shell st	tructural			
Elevation :	Aboveground	×						
ank Definition Lo	ad Prestress Support (3D) Seisr	nic Ground						
nner Tank Propert	ies Non-Structural Masses Lumpe	d Foundation	1					
						Use 3D support in	iputs	
Geometric pro	perties							-
Name		Exist	Area [m²]	Shear area [m ²]	Moment of inertia [m ⁴]	Length [m]	_	
Pile (Lumped)		\checkmark	617.23	540.14	297.064E3	NA		
Total ma:	ss of lumped isolator [kg] = isolator	mass x numb	er of base support =	158.8E3				
Lumped pile s	tiffnesses							
[Vertical b	eam stick model] Vertical stiffne	ss of pile/soil [f	/IN/m]	225.9233E3				
				225 022252				
[Horizont	al beam stick model] Rotational stif	ffness of pile h	ead [MNm/rad]	220.020020				
Catalan	Catilation							
Set Zero	Set defaults							
		Name Tn	k2		 (new) 			
					ОК	Cancel	Apply	Help

Fig 204 User Inputs 3 for Seismic Analysis

Soil Properties

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

Material : Con Elevation : Abo nk Definition Load Pre- oil Properties Stiffness distribution: C Layer Soil depth 0.0 00	cretee voveground vestress Support (3D) Constant value Thickness Static k (MN/m/ 0.0 0.0	Seismic Groun	d Dynamic kh	ymmetric struct	tural] 2D axisymmetric cc] 3D shell structural	upled thermal/str	uctural	
Elevation : Abo nk Definition Load Pre- oil Properties Stiffness distribution: Layer Soil depth No. Soil depth O 0.0 C	Constant value Thickness Static k of layer [m] 0.0 0.0 0.0	Seismic Groun	d Dynamic kh	n-stick seismic		☑ 3D shell structural			
ank Definition Load Pro- tiol Properties Stiffness distribution: Layer Soil depth No. (m) 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	restress Support (3D) Constant value Thickness Static k of layer [m] 0.0	Seismic Groun	d Dynamic kh						
Ank Definition Load Pro- coll Properties Stiffness distribution: C Layer Soil depth No. 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Constant value Thickness Static k of layer [m] 0.0 0.0	h Static kv m [MN/m/m]	d Dynamic kh						
Soil Properties	Constant value Thickness Static k of layer [m] [MN/m/ 0.0 0.0	h Static kv m] [MN/m/m]	Dynamic kh						
Stiffness distribution:	Constant value Thickness Static k of layer [m] 0.0 0.0	h Static kv m] [MN/m/m]	Dynamic kh						
Layer Soil depth No. [m] 0 0.0 0	Thickness Static k of layer [m] [MN/m/ 0.0 0.0	h Static kv m] [MN/m/m]	Dynamic kh						
Layer Soil depth No. [m] 0 0.0 0 1 -2.0 2	of layer [m] [MN/m/ 0.0 0.0	m] [MN/m/m]	Dynamic kn	Description in the second seco	Louis and the				
0 0.0 0	0.0 0.0		[MN/m/m]	[MN/m/m]	[MN/m/m]	Description(CL I	
1 -20 2		0.0	0.0	0.0	0.0	Start of soil p			
1 _20 2									G.L.
	2.0 19.0793	0.0382	38.1586	0.0763	14.92E3	Backfill	Layer 1		
2 -4.0 2	2.0 32.9527	0.0659	65.9054	0.1318	25.769E3	Backfill			Plies
							Layer 2		+
3 -6.0 2	2.0 28.6317	0.0573	57.2634	0.1145	22.39E3	Backfill	Layer 3		Layer thickness
4 -8.0 2	2.0 27.5563	0.0551	55.1125	0.1102	21.549E3	Backfill	Layer 4		
F 40.0 (2.0 20.207	0.0004	60.4440	0.4000	22 62252	Dealifi			
5 -10.0 2	2.0 30.2072	0.0004	00.4143	0.1208	23.022E3	Backmi	Layer n		
6 -12.0 2	2.0 41.3977	0.0828	82.7954	0.1656	32.373E3	Silty Sand1			
7 -14.0 2	2.0 34.5307	0.0691	69.0614	0.1381	27.003E3	Silty Sand1	Add	Set zero	
						×	Remove	Set defaults	

Fig 205 User Inputs 4 for Seismic Analysis

Seismic Analysis Wizard

The user dialog is available from LNG Tank> Create 2D Model> Seismic... as shown in [Fig 272].

- Enter the model file name and select Design Code to EN1998-4, model type of Beam-Stick Horizontal.
- Set the number of buttresses 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.

Examples - User Inputs

ank definition data	Tnk1		~						
lodel filename	Example	Example							
aved model file path	C:\Users\ohss	NDocuments\Lusas191\Projects\Example	e_EN1998_H						
nalysis type									
esign code	EN 1998-4	\sim							
 Beam-stick horizo (Excluding base pre 	ntal ssure)	 Beam-stick horizontal (Including base pressure) 	O Beam-stick vertical						
(Beam-Stick Horizonta EN1998-4:2006 A3.2.2	al model and Vertical mode Simplified procedure for f	el is created according to ixed base cylindrical tanks)							
ritical damping / frequer	ncy								
	Critical damping (%)	Frequency (1st mode, Hz)	Frequency (2nd mode, Hz)						
Base slab	4.0	1.25	5.44						
Roof	4.0								
Wall	2.0								
Inner tank	2.0								
Foundation	4.0								
LNG impulsive	3.0								
I NG convective	0.5								
Ground	5.0								
ittress									
Number of buttress	0	~							
	-	Puttross width	5.0 (m)						
Extruded thickness	1.0	(m) Dulless width	0.0 (iii)						

Fig 206 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	RIZON		IODEL						
1) LNG Mass & Heigh	nt									
IBP (Including Base Pressure)										
Component	H/R	m _{(c}	_{:.i)} /m	h' _(c.ī) /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				
EBP (Exluding 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	0.60	50,527,854	23.53				
LNG Impulsive	0.924		0.51	0.41	53,494,849	16.13				
Summary Verifications-ACI35			Verifi	cations-EN1998-	4 (+)					

Fig 207 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.



Fig 208 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 209 Roof in Beam-Stick Model



Varying Section properties are defined as shown below.

Fig 210 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.



Fig 211 RingBeam Upper in Beam-Stick Model



Varying Section properties are defined as shown below.

Fig 212 Geometric Properties for RingBeam Upper in a Beam-Stick Model

RingBeam Lower

The elements for RingBeam Lower represent the region shown below.



Fig 213 RingBeam Lower in a Beam-Stick Model

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.





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 215 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.

Attributes 🗸 🕈 🗙	Geometric Line		×
😨 Grou 🗞 Attri 😧 Anal 🥜 Utilit 🛄 Repo 🔂 Layers			
Attributes (33)	Analysis category 2D Inplane		
⊟ = Mesh (7)	Definition	Properties for end 1 of line	Properties for end 2 of line
Point (3)	From library / calculator	User Sections	User Sections
3:1 NGMass Impulsive	Potation about controld 0 ~ °		
4:LNGMass Convective		Local	Local
4 5: Foundation Rocking	Mirrored about axis None ~	CHS D=88.6 t=1.1	CHS D=87.9 t=0.75
😑 🔤 Line (4)	 Enter properties 		
	Usage 2D Thick Beam (Any beam)	100%	100%
- & 2:Beam_EndJoint(Roof)		R3	R2
	Alignment		
Geometric (10)	Align end 2 to end 1		
🖨 🛳 Line (9)	Align end 1 to end 2		· · · ·
	Vertical Centre to centre v 0.0	, ,	· (^)
-& 3:Pile		P4 . P2	P3 . P1
4:BaseSlab (CSS D=93) 6 F-W-H (CUC D= 07.0 + 0.75)	Horizontal Centre to centre V		\rightarrow_z
			1
# & 7:RingbeamLower (CHS D=88.5 t=1.05)	Interpolation of Use Section Calcu V		
& 8:RingbeamUpper (CHS D=88.5 t=1.05/ CH			
# & 9:Roof (Varying - 19 sections)			
# 4 10:InnerTank (Varying - 10 sections)		P1	P4
😑 🔤 Joint (1)	ev origin Centroid ~	N1	134
- S 2:JointGeometric	-,	1	
E-G Patenia (13)	0	Value	Value
- 1:BaseSlab	Cross sectional area (A)	302.378	205.342
	Second moment of area about 2 axis (izz)	289.431E3	194.964E3
	Elective shear area in y ulrection (Asy)	151.189	102.681
A 4:RindbeamUnder	Eccentricity in y direction (ey)	0.0	0.0

Fig 216 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 $6,792.91 \text{ m}^2$), and total length of 1.2835 m.



Fig 217 Section Properties for Base Slab in a Beam-Stick Model
Pile

Attributes 🗸 🗘 X		
Gorou GAttrin. QAnal. Vullit. Arepo. Layers Attributes (33) Mesh (7) Geometric (10)		y ↓ → z
	ey origin Centrola 🗸	Value
	Cross sectional area (A)	617.23
B & 5:Wall (CHS D=87.9 t=0.75)	Second moment of area about z axis (lzz)	297.064E3
🕀 🐣 6:WallTapered (CHS D=88.6 t=1.1/ CHS D=	Effective shear area in y direction (Asy)	540.14
	Eccentricity in y direction (ey)	0.0

Fig 218 Section Properties for Pile in a Beam-Stick Model

Inner Tank

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



Fig 219 Section Properties for 'Inner Tank' in a Beam-Stick Model

The inner tank is modelled at X=1.1 (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 -1.1 was used in the geometric property definition. The properties for the bottom of the inner tank are as shown below.

Aultiple Varying Sec Analysis category	2D Inplane	Distance interp	pretation	×		
Specify shape in	terpolation on	Along refer	Enter Section - Row 2 Definition		Harr Cashing	×
1 CHS D=84.272	Section 2 t=0.0361	Shape Interpolation Start	O From Library Rotation about centroid	0 × °	Local	
2 CHS D=84.272 3 CHS D=84.272 4 CHS D=84.272 5 CHS D=84.224	2 t=0.0361 2 t=0.0361 2 t=0.0361	Linear Linear	Mirrored about axis	None ~	Error: Not found	100%
6 CHS D=84.224 Alignment Vertical Centre Horizontal Centre 1 3	t=0.012 e to centre ✓ ey origin e to centre ✓ ez origin	Linear Centroid ~ Align all sec Interpolati			R4 z	R2 R1
	2 33 44)	Cross sectional area (A) Second moment of area about Effective shear area in y direction Eccentricity in y direction (ey)	z axis (lzz) on (Asy)	Value 9.5534 8.47348E3 4.77667 -1.1	
Nar	ne InnerTank		Visualise		Section	details
		Close Ca			OK Cancel	Help

Fig 220 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 🗸 🕈 🗙	sotronic	×
Grou. ♣ Attri. ♀ Anal. ♀ Utilt. ■ Repo. ♀ Layers ♥ ■ Kample. FN1998. Horizotal(IBP).mdl ♥ Horizotal(IBP).mdl ♥ ■ Keht (7) ⊕ ■ Generic (10) ● ■ Metrial (13) ● ■ Schopic (8) ● ■ Schopic (8) ● ▲ 2:Wall ● ■ Schopic (8) ● ▲ 3:RingbeamUpper ● ▲ 3:RingbeamUpper ● 3:Sindof ● ■ Schopic (7) ● ■ Schopic (7) ● ■ Schopic (8) ● ■ Schopic (8) ● ■ Schopic (8) <td>□ Plastic □ Creep □ Demage □ Shrinkage □ Viscous Elastic □ □ Oynamic properties □ □ □ Thermal expansion □ Poisson's ratio □ □ ■ Mass Rayleigh damping constant Stiffness Rayleigh damping constant</td> <td>☐ Two phase Ko Initialisation 35.0E9 0.2 0.2E3 0.51092 1.9032E-3 0.51092</td>	□ Plastic □ Creep □ Demage □ Shrinkage □ Viscous Elastic □ □ Oynamic properties □ □ □ Thermal expansion □ Poisson's ratio □ □ ■ Mass Rayleigh damping constant Stiffness Rayleigh damping constant	☐ Two phase Ko Initialisation 35.0E9 0.2 0.2E3 0.51092 1.9032E-3 0.51092
	Name Roof	~ * (5)
	Close Cance	Apply Help

Fig 221 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 269].)



Fig 222 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.

otropic		×
Plastic Creep I	Damage Shrinkage Viscous	Two phase Ko Initialisation
Dynamic properties Thermal symposium	Young's modulus	Value 35.0E9
	Poisson's ratio	0.2
	Mass Rayleigh damping constant	0.25546
	Stiffness Rayleigh damping constant	0.951599E-3
	Plastic Creep I	Plastic Creep Damage Shrinkage Viscous Elastic Voung's modulus Dynamic properties Young's modulus Thermal expansion Poisson's ratio Mass density Mass Reyleigh damping constant Stiffness Rayleigh damping constant

Fig 223 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 🗸 🗘 🗸	Isotropic		×
ConnectionPart Attri_QAnal_	Plastic Creep Elastic ✓ Dynamic properties ☐ Thermal expansion	Damage Shrinkage Viscous Young's modulus Poisson's rato Mass density Mass Rayleigh damping constant Stiffness Rayleigh damping constant	Two phase Ko Initialisation

Fig 224 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.

Attributes 🗸 🕈 🛪	Isotropic			×
Grou. ♣Attri. QAnal. ✤Utilit. ■Repo. □Layers ■ Example_EN1998_Horizontal(IBP).mdl ⊕ Attributes (33) ⊕ ■ Wesh (7) ⊕ ■ Geometric (10)	Plastic Cree	ep 🗌 Damage	Shrinkage Viscous	Two phase Ko Initialisation
→ a Material (13) → a Material (13) → A 118aesSlab → & 2:Wall → & 3:RingbeamLower → & 4:RingbeamLower → & 4:RingbeamUpper → & 5:Roof → & 5:ConnectionPart → & 5:ConnectionPart → & 5:InnerTank	✓ Dynamic propertie	S	Young's modulus Poisson's ratio Mass density Mass Rayleigh damping constant Stiffness Rayleigh damping constant	Value 35.0E9 0.2 2.85876E3 0.51092 1.9032E-3

Fig 225 Material Properties for BaseSlab in Beam-Stick Model

Pile & Soil

The Input data was used for elastic modulus, Poisson's ratio, and mass density. Damping constants are computed and added.

Tank type								
			Target models t	o build				
Material :	Concrete	~	🖂 2D axisym	metric structural	🗹 2D axis	symmetric coupled t	thermal/structural	
Elevation :	Aboveground	~	✓ 2D beam-	stick seismic	☑ 3D she	ell structural		
ank Definition Lo	ad Prestress Insulations	Support (3D) Seis	mic Ground					
Base Slab and Roo	of Wall and Ring beam Ma	terials Support (2D))					
Material ID	Elastic modulus (E, [N/m²])	Poisson's ratio (v)	Mass density [kg/m²]	CTE (alpha, [1/C])	Thermal conductivity [J/m.s.C]	Heat capacity [J/m²/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 (Ringbe	am) 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	
Pile (Cir)	35.0E9	0.2	2.5E3	10.0E-6	0.0	0.0	Pile (Cir)	
Pile (Cross)	35.0E9	0.2	2.5E3	10.0E-6	0.0	0.0	Pile (Cross)	
* Isolator propr	erties can be defined for va	rious types from m	nodeler and shoul	d be defined and :	assigned manually			
* Isolator propr Set zero	erties can be defined for va	rious types from m	nodeler and shoul	d be defined and a	assigned manually	٨.		
* Isolator propr	erties can be defined for va	rious types from m	hodeler and shoul	d be defined and a	assigned manually	<i>.</i>	C (new)	

Isotropic		×
Plastic Creep Damage	e Shrinkage Viscous	Two phase Ko Initialisation
Dynamic properties	Young's modulus Poisson's ratio Mass density Mass Rayleigh damping constant Stiffness Rayleigh damping constant	Value 35.0E9 0.2 2.5E3 0.51092 1.9032E-3
Name Pile		~) 🗢 (7)
	Close Cance	el Apply Help

Fig 226 Material Properties for 'Pile' in a Beam-Stick Model

Analysis category 2D Inplane Cylindrical Assignment to Ines Thermal expansion Joint type Joint no rotational stiffness Damping Rayleigh Oviscous coefficient Properties specified for each freedom Elastic spring stiffness Soil_kh 0.0 Mass 0.0 0.0 Viscosity coefficient Viscous coefficient 0.0 Name SoilSpring (13)					
Analysis category 2D Inplane Cylindrical Cylindrical Thermal expansion Joint type Joint no rotational stiffness Category Analysis position At first node Category Rayleigh Viscous coefficient Coefficient Viscous coefficient Viscous coefficient Viscous coefficient Viscous coefficient Viscous coefficient Category Category (13)	eneral Properties				
Analysis category 2D Inplane Cylindrical Cylindrical Thermal expansion Joint type Joint no rotational stiffness Category Reyleigh OViscous coefficient Reyleigh OViscous coefficient Specified for each freedom Elastic spring stiffness O.0 0.0 Viscous coefficient Viscous coefficient Viscous coefficient O.0 Viscous coefficient Viscous coefficient O.0 Viscous coefficient Viscous coefficient O.0 Viscous coefficient Viscous Coeff					
Analysis category 2D Inplane Cylindrical Cylindrical Cylindrical Internal expansion Joint type Joint no rotational stiffness Damping Rayleigh Oviscous coefficient Rayleigh Viscous coefficient Society coefficient Viscous Coeffi					
Assignment to Lines Joint type Joint no rotational stiffness Mass position At first node Properties specified for each freedom Elastic spring stiffness Soil_kh 0.0 Mass 0.0 0.0 Viscosity coefficient Viscous coefficient 0.0 Name SoilSpring Close Cancel Apply Help	Analysis category	2D Inplane		Cylindrical	
Joint type Joint no rotational stiffness ✓ Damping Mass position At first node ✓ Damping Properties specified for each freedom Elastic spring stiffness Soil kh 0.0 Viscosity coefficient Viscous coefficient 0.0 Viscosity coefficient Viscous coefficient 0.0 Name SoilSpring ✓ Close Cancel Apply Help	Assignment to	Lines	~		
Joint type Joint no rotational stiffness	Assignment to	LINGS		I hermal exp	ansion
Mass position At first node Rayleigh Viscous coefficient Properties specified for each freedom Elastic spring stiffness Soil kh 0.0 Viscosity coefficient Viscous coefficient 0.0 Viscosity coefficient Viscous coefficient 0.0 Name SoilSpring Image: Close Cancel Apply Help	Joint type	Joint no rotational stiffnes	s ~	✓ Damping	
Mass O O Viscosity coefficient Viscosity coefficient 0.0 Name SoilSpring Image: Close Cancel Apply Help					Viscous coefficient
Properties specified for each freedom u v Elastic spring stiffness Soil_kh 0.0 Mass 0.0 0.0 Viscosity coefficient Viscous coefficient 0.0 Name SoilSpring Image: SoilSpring Close Cancel Apply	Mass position	At first node	~		Viscous coefficient
Properties specified for each freedom u v Elastic spring stiffness Soil kh 0.0 Mass 0.0 0.0 Viscosity coefficient Viscous coefficient 0.0 Name SoilSpring Close Cancel Apply Help 					
Properties specified for each freedom u v Elastic spring stiffness Soil kh 0.0 Viscosity coefficient Viscous coefficient 0.0					
u v Elastic spring stiffness Soil kh 0.0 Mass 0.0 0.0 Viscosity coefficient Viscous coefficient 0.0 Name SoilSpring Image: SoilSpring Close Cancel Apply	Properties specifie	d for each freedom			
Elastic spring stiffness Soil_kh 0.0 Mass 0.0 0.0 Viscosity coefficient Viscous coefficient 0.0 Name SoilSpring Viscous Coefficient (13)		u	v		
Mass 0.0 0.0 Viscosity coefficient Viscous coefficient 0.0 Name SoilSpring 	Elastic spring stiff	ness Soil_kh	0.0		
Viscosity coefficient 0.0 Name SoilSpring	Mass	0.0	0.0		
Name SoilSpring	Viscosity coefficie	nt Viscous coefficient	0.0		
Name SoilSpring V (13) Close Cancel Apply Help					
Name SoilSpring					
Name SoilSpring V (13) Close Cancel Apply Help					
Name SoilSpring					
Name SoilSpring V 🗘 (13)					
Name SoilSpring (13) Close Cancel Apply Help					
Name SoilSpring V 🗘 (13)					
Name SoilSpring (13) Close Cancel Apply Help					
Name SoilSpring V (13) Close Cancel Apply Help					
Name SoilSpring (13) Close Cancel Apply Help					
Name SoilSpring (13) Close Cancel Apply Help					
Name SoilSpring (13) Close Cancel Apply Help					
Name SoilSpring					
Name SoilSpring V (13) Close Cancel Apply Help					
Name SoilSpring (13) Close Cancel Apply Help					
Name SoilSpring V (13)					
Name SoilSpring (13) Close Cancel Apply Help					
Name SoilSpring (13) Close Cancel Apply Help					
Name SoilSpring					
Name SoilSpring V (13)					
Close Cancel Apply Help	Nama	oilSpring			× (12)
Close Cancel Apply Help		olisping			· · · · · · · · · · · · · · · · · · ·
Close Cancel Apply Help	Name S				
Close Cancel Apply Help	Name S				
Close Cancel Apply Help	Name S				
	Name S				

	Distance	Malua			Distance	Value	
1	Distance	14 92E9	^	1	O	20 7517E6	- î
2	1 999	14.92E9		2	1 999	20.7517E6	
3	2	25 769E9	-	3	2	27.272E6	
4	3 999	25.769E9		4	3.999	27.272E6	
5	4	22.39E9		5	4	25.4212E6	
6	5,999	22.39E9		6	5,999	25.4212E6	
7	6	21.549E9		7	6	24.9392E6	
8	7.999	21.549E9		8	7.999	24.9392E6	
9	8	23.622E9		9	8	26.1112E6	
10	9.999	23.622E9		10	9.999	26.1112E6	
11	10	32.373E9		11	10	30.5675E6	
12	11.999	32.373E9	~	12	11.999	30.5675E6	~
<			>	<			>
Name	Soil_kh		× 🗘 (1)	Name Vis	cous_coefficient		~ 🗘 (2)

Fig 227 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 Impu	sive 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
G	6.51359		coefficients for determining the fundamental frequency
T _{imp}	0.43182	s	fundamental period of oscillation of the tank (plus the i
ki	11,325,839,357	N/m	



Fig 228 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					

2) Convective stiffness for Liquid

,	•						
Component	Value	Unit	Remark				
H/R	0.924		Liquid height divided by inner tank radius				
Cc	1.54	s/m ^(1/2)	coefficients for determining the fundamentalfrequency				
T _{conv}	9.993	s	natural period of the first (convective) mode of sloshing				
kc	19,974,995	N/m					



Fig 229 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:

Attributes 🗸 🕈 Attributes		LUSAS View: Example_E	sotropic ×
🔞 Grou 💑 Attri 🚇 Anal 🦨 Utilit 🛄 Repo 🗊 Layers	X	0.0	
	0		Plastic Creep Damage Shrinkage Viscous Two phase Ko Initialisation Elastic
🖨 🔤 Isotropic (8)	8		Dynamic properties Value
1:BaseSlab			Thermal expansion Young's modulus 2.0E15
- 3:RingbeamLower		I I	Poisson's ratio 0.3
- 4:RingbeamUpper			Mass density 8.86327E3
- 5:Roof - 5:ConnectionPart	0	† 1	Stiffness Ravielich damping constant 0.25040
-\$ 7:Pile	4		
A SilmerTank A SilmerTank A SiRcoffloorStucturalMass A UNKMass Impublie A 111.NKMass Convective A 111.NKMass Convective A 113.SolSpring G Support (3)	30.0	0.0 0.0	
- & 11 Flow - & 2:Ully Fixed - & 3:X Free for Liquid	20.0	•	
	0.0		Name InnerTank
			Close Cancel Apply Help
None × Sport/	0		
speciry	-		A 7



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 231 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.)

Print Results Wizard			×		LUSAS	View: Example	EN1998_Horiz	ontal(IBP).mdl	Window 1	Mass participat	ion factors ×		
				80	D. 🖬 🖬 🕬	1.0.							
Results type					Mode+	Mass PF X	Mass PF Y	Mass PF Z	Mass PF THX	Mass PF THY	Mass PF THZ	Frequency	Period
	Components	(e) Eigenvalues		1	1	0.194371	17.2536E-21	0.0	0.0	0.0	53.6568E-9	0.100051	9.99495
Regits				2	2	0.290259	3.2177E-15	0.0	0.0	0.0	0.0160486	2.07363	0.482246
Londrated 1247 million				3	3	0.0725154	2.45524E-15	0.0	0.0	0.0	0.141923	4.08668	0.244698
Colocases 12.Liganatie				4	4	3.58657E-3	0.315943E-15	0.0	0.0	0.0	0.681718	10.8867	0.0918548
REgenvalues (Frequency)				5	5	0.169641	1.72135E-15	0.0	0.0	0.0	0.049142	13.7901	0.0725155
PiPertidgetion factors				6	6	0.108578	9.14245E-15	0.0	0.0	0.0	0.0624865	19.9395	0.0501518
PMass pertidipation factors				7	7	0.0278198	0.804232E-15	0.0	0.0	0.0	1.53814E-3	22.3862	0.0446703
PSum mass perticipation for	actors			8	8	0.0162034	76.581E-15	0.0	0.0	0.0	7.50148E-3	27.3651	0.0365429
				9	9	0.115143	0.199296E-15	0.0	0.0	0.0	0.0179757	29.7295	0.0336367
				10	10	0.413503E-3	7.74788E-15	0.0	0.0	0.0	24.3827E-6	37.2863	0.0268195
				11	11	0.416238E-3	71.7832E-15	0.0	0.0	0.0	2.74973E-3	41.657	0.0240056
				12	12	0.140707E-3	8.81287E-15	0.0	0.0	0.0	8.45088E-6	46.7114	0.0214081
				13	13	0.600295E-3	0.387079E-12	0.0	0.0	0.0	4.95306E-3	49.0879	0.0203716
				14	14	0.189727E-3	0.386277E-12	0.0	0.0	0.0	0.935939E-3	55.4773	0.0180254
				15	15	39.2056E-6	0.269117E-12	0.0	0.0	0.0	0.898221E-6	59.9792	0.0166725
				16	16	59.7483E-6	22.967E-15	0.0	0.0	0.0	0.108925E-3	61.1449	0.0163546
		Precision		17	17	0.130119E-6	1.13367E-12	0.0	0.0	0.0	27.3847E-6	71.1321	0.0140584
		 Significant figures 	0	18	18	76.1365E-9	0.556294E-12	0.0	0.0	0.0	25.7855E-6	77.0164	0.0129842
		O Decimal places		19	19	47.2121E-9	0.194314E-12	0.0	0.0	0.0	38.852E-6	84.3824	0.0118508
Display may		Show trailing zeros		20	20	0.558889E-6	11.7431E-15	0.0	0.0	0.0	20.7343E-6	87.2066	0.011467
See in Inerview	Defaults	Liberholdunhus N/A		21	21	19.3758E-6	1.7538E-12	0.0	0.0	0.0	0.528207E-3	90.225	0.0110834
			_	22	22	13.9621E-9	1.74934E-12	0.0	0.0	0.0	79.3538E-6	94.0027	0.010638
				23	23	46.4415E-9	0.751571E-12	0.0	0.0	0.0	1.91462E-6	98.269	0.0101761
Name PRV	V1		✓ 1⇒1 (new)	24	24	0.597419E-8	1.80782E-12	0.0	0.0	0.0	7.78848E-6	108.792	9.19181E-3
			(I) to any	25	25	8.12575E-9	2.5335E-12	0.0	0.0	0.0	57.67E-6	114.858	8.70637E-3
				28	28	2.18508E-9	1.19262E-12	0.0	0.0	0.0	0.193103E-6	115.442	8.66235E-3
				27	27	0.776573E-8	2.96268E-12	0.0	0.0	0.0	0.121335E-3	127.681	7.83203E-3
				28	28	11.5958E-9	2.34128E-12	0.0	0.0	0.0	9.90965E-6	131.634	7.59683E-3
		CM	d Analy Inda	29	29	5.26511E-9	1.40053E-12	0.0	0.0	0.0	10.8671E-6	133.151	7.51025E-3
		OK Galo	a Short Den	30	30	3.52335E-9	9.13505E-12	0.0	0.0	0.0	15.32E-8	144.489	6.92097E-3

Fig 232 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.



Fig 233 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 234 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.

Analyses	¥ II X		Evampla EN1009 Harizanta	I/IRD) mdl \\/indow	1 Y
😵 Grou 🖧 Attri 🕒 Anal 🖌	Eigenvalue				\times
Example_EN1998_Horizontal(Solution Frequency	~	Number of eigenvalues	Value 30	
Geometric Geometrial Geometrial Geometrial Geometrial Geometrial	✓ Include modal damping Eigenvalues required Mi	Set damping	Modal Damping	0.0	
Eigenvalue ⊕ ■ Example_EN1998 ⊟ − ■ Post processing	Range specified as Frequency 	O Eigenvalue	Viscous	◯ Structural	
	Eigenvector normalisation		Use distributed damping	All modes	~
Model properties	O Unity Mass	◯ Stiffness	Modes using distributed		
	Convert assigned loading	to mass	Eigenvalue 1 Eigenvalue 2 Eigenvalue 3 Eigenvalue 3 Eigenvalue 4 Eigenvalue 6 Eigenvalue 7		^
		-20.0	Eigenvalue 8 Eigenvalue 9 Eigenvalue 10 Eigenvalue 11		~
			Damping ratio for modes 0.05	not using distributed	
		40.0	OK Cano	el Help	

Fig 235 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)

Section 2012 - Associated Model Data > example_EN1998_Horizontall	MODAL DAMPING FACTORS
이름 ^	
Backups Sessions	MODE EIGENVALUE FREQUENCY VISCOUS DAMPING 1 0.395184 0.100051 0.499755E-02 2 169.755 2.07363 0.240787E-01 2 169.755 2.07363 0.240787E-01 2 2 2
Example_EN1998_HorizontalBeamStick_IBP_~Analysis 1.dat	3 659.327 4.08668 0.238845E-01 4 4679.03 10.8867 0.437176E-01
Example_EN1998_HorizontalBeamStick_IBP_~Analysis 1.log Example_EN1998_HorizontalBeamStick_IBP_~Analysis 1.mys	6 15695.9 19.3901 0.556754E-01 7 19784.3 22.3862 0.873847E-01
Example_EN1998_HorizontalBeamStick_IBP_~Analysis 1.out Shortcut to Example_EN1998_HorizontalBeamStick(IBP)	8 29563.4 27.3651 0.112492 9 34892.6 29.7295 0.106041 10 54885.5 37.2863 0.150798

Fig 236 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								
💷 La 🖄 An 🍱 Re 🖄 Gr 📌 Ut Å At									
Example2_EN1998_HorizontalBeamStick(4)	Design code ASCE-7-10	Show graph							
LING TARK (1)	Incremental period	0.02 s	Maximum period	6.0	s				
a Reference Path(2)	Spectra definition								
2:InnerTank	Code defined	User defined	Scale factor	1.0					
	Parameters								
	Site class	A ~	Long transition period (TL)	4.0	s				
	Mapped spectral acceleration at short periods (Ss)	0.25	Mapped spectral acceleration at one second period (S1)	0.1					
	Spectral data								
	Site coefficient (Fa)	0.8	Site coefficient (Fv)	0.8					
	Short period response acceleration parameter (Sds)	0.133333	One second period response acceleration parameter (Sd1)	0.0533333					
	Period (T0)	0.08 s	Period (TS)	0.4	s				
	Name ASCE	A-CLASS	V Close Cancel	(1) Apply H	elp				
	L		Cancer	rippit 11	cip				

Fig 237 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	Χ.	esponse Spectrum - User defined	×
		Frequency Oisplacement Spe Ovelocity Ovelocity Ovelocity Ovelocity Ovelocity Ovelocity Ovelocity	schal curve 0.0 %
		0 0	012 014 016 000 000 000 000 000 000 000 000 000
		Response spectrum Name Stew/Spistrum	v 💽 (new)
< Back Next > Finish Cascel Apply Help		< Back	Next > Finish Cancel Apply Help

Fig 238 User-defined Response Spectrum

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

🔀 LUSAS Bridge Plus - Example_EN1998_HorizontalBeamStick(IBP).mdl
File Edit View Geometry Attributes Analyses Utilities T	ools Bridge Design LNG Tank Window Help
: D 🖆 🖶 🖆 😑 👒 🔞 🗙 🖸 - 🗠 - 🚳 i 🔿	/ - 🗋 - 😥 - 🔞 ííí - 🍐 : 🌋 - 🌾 🎍 - 💠 🏠 🐵 - 🎢 📐 - 😥 - 😥 😽 🕸 🐼 🐼 👔 - 🔛
Analyses 👻 🕈 🗙	LUSAS View: Example_EN1998_HorizontalBeamStick(IBP).mdl Window 1 x
🔁 Layers 🔣 Groups 歳 Attrib 🕒 Analy 🥓 Utiliti 💶 Repo	X5.0 0,0 5,0 10.0 15,0 20,0
Example EN1993_HorizontaBeamStok(BP).md Garden Stok(BP).md Garden Stok(BP).md	A STAR
B Supports Supports Supports Supports Supports Supports Supports Supports Support Sup	Excitation Support Motion Set
	Spectral Response X
	Type of spectral response SRSS combination ∨
	Set damping Set damping Select modes Damping variation correction to response spectrum Eurocode
	Response 1:ASCE A-CLASS V
	Name Response Spectrum SRSS V + (3)
	0 97 Close Cancel Apply Heb

Fig 239 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> Create 2D Model>Seismic...

• 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.

ank definition data		Tnk1				\sim			
lodel filename		Example_Vertic	al						
Saved model file path		C:\Users\ohsso\Documents\LUSAS200\Projects\Example_Vertical_EN1998_Vertical.mdl							
nalysis type									
)esign code		EN 1998-4	~	·					
 Beam-stick hori (Excluding base p 	izontal pressure)			O Bea (Inclu	m-stick horizontal ding base pressure)		eam-st	ick vertical	
(Beam-Stick Horizon EN1998-4:2006 A3.	ntal mode 2.2 Simpli	I and Vertical mo fied procedure for	del is c r fixed b	reated a	ccording to drical tanks)				
ritical damping / frequ	uency								
	Critical o	damping (%)		Freque	ncy (1st mode, Hz)				
Roof	5.0			5.0					
Pile / Foundation	5.0								
LNG flexible	5.0								
LNG Rigid	5.0								
uttress									
Number of buttress	0		~						
Extruded thickness	1.	0		(m)	Buttress width	5.0)	(m)	



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.





Material Properties

The details for computing properties are summarized in the spreadsheet.

Roof

1) Roof Mass & Stiffn	less		
Component	Value	Unit	Remark
m_ _{roof}	12,027,753	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}	11,870,916,725	N/m	

Fig 242 Roof Properties for Beam-Stick Model for Vertical Actions

And the second se					
Attributes	₩ # X	General Properties			×
Grou. Attri (Anal Utilit Repo Layers Example. Vertical_EN1998_Vertical.mdl Attributes (13) Attributes (13) Attributes (13) Attributes (13) Attributes (13) Attributes (13) Attributes (14) Attributes (14) At		Analysis category Assignment to Joint type Mass position	2D Inplane Points and line ends Joint no rotational stiff At first node	Cylindrical Thermal expansion Damping Rayleigh OViscous coefficient	
→ S:Pile Geometric (1) → Joint (1) → Joints (4) → 1:Roof → 2:FluidRigid → 3:FluidRigid → 3:FluidRigid → 4:NoRoofTank(M) ⊖ Supports (2) → 4:NiRoof	 Joint Between Features (4) JiRoof JiRoof Semetric (1) Joint Commercic Geometric (1) Joints (4) A: FluidRigid J:Roof A: Roof A: Roof A: Sindi Rigid A: Sindi Rigid		d for each freedom u uess 11.8709E9 12.0278E6 nt 37.7863E6 005	v 1.0E15 0.0 0.0	 ✓ (1)
None ~	Specify				

Fig 243 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.			
· ·	1					
k_ _{LNG_r}	21,631,229,542,194,300	N/m				

Attributes 🗸 🗸	• × General Propertie	s			×
Grout Antri QAnal Utilit Repo Attributes EN1998_Vertical.mdl Attributes (13) Mesh (5) Point (5) Attributes (13) Doint between Features (4) AttRoof AttRoof AttRoof Science Science Science Science AttRoof Science Science Science	Analysis category Assignment to Joint type Mass position	2D Inplane Points and line ends Joint no rotational stif At first node	∽ fness ∽ ∽	Cylindrical Thermal expan	sion • Viscous coefficient
Geometric (1) Geometric (1) Material (5) Geometric (1) Material (5) Geometric (1) Material (1)	Properties specific Elastic spring stiff Mass Viscosity coefficient	ed for each freedom u hess 21.6312E9 89.5668E6 ant 139.192E6	v 1.0E15 0.0 0.0		
None Y Spec	tify				• (3)

Fig 244 Material Properties for Fluid-Flexible in a Beam-Stick Model for Vertical Actions

Fluid-Rigid

k_lng_r

m_ _{LNG_r} (1)	52,900,941	kg	mass of LNG (rigidly moving) = $sqrt(m_{_UNS_total}^2 - m_{_UNS_f}^2)$
m_{LNG_r} (2)	14,455,895	kg	mass of LNG (rigidly moving) = $m_{_LNG_total}$ - $m_{_LNG_f}$
Pur	18,681,6000	ka/m ²	hydrodynamic pressure on the wall, from A.17
1.10	,	149/111	
m_ _{LNG_r} (3)	104,022,703	kg	mass of LNG (rigidly moving), ref. A.17.

N/m

21,631,229,542,194,300

Attributes 🗸 🕈 🗙	General Properties	s			×
Grou. Attri QAnal. Vulit Repo. Layer Example. Vertical. EN1998. Vertical.mdl Attributes (13) Point (5) Point (5) A 1:NoofTank(M) Joint between Features (4) A 1:Roof A 2:FluidRajd A 3:FluidRaid A 3:FluidRaid A 3:FluidRaid A 3:FluidRaid	Analysis category Assignment to Joint type Mass position Properties specifie	2D Inplane Points and line ends Joint no rotational stiffness At first node	Cylindrical ☐ Thermal expansion ☑ Damping ○ Rayleigh		
Soint (1) A 1: Joint Cemetric Material (5) Joints (4) A 1: Roof A 2: FluidRigid A 3: Fluid Rigid A 4: NokooFrank(M) Supports (2) A 1: Fully Fixed A 2: Y Free	Elastic spring stiff Mass Viscosity coefficie	u hess 21.6312E15 104.023E6 ant 150.005E9	v 1.0E15 0.0 0.0		× 🗘 (2)
None ~ Specify					

Fig 245 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 for Pile						
Component	Value	Unit	Remark			
k_pile	225,923,300,000	N/m				

Attributes 🗸 🕈 🗸	General Properties	×
🐻 Grou 💑 Attri 🕒 Anal 🦨 Utilit 🛄 Repo 🗊 Layers		
Example_Vertical_EN1998_Vertical.mdl	Analysis category 2D Inplane	Cylindrical
Ba Mesh (5) Ba Point (5)	Assignment to Points and line ends	Thermal expansion
4:NoRoofTank(M)	laint trans loint no rotational stiffness	
ial joint between Features (4)	Joint type Joint no rotational sumess	
-& 2:FluidRigid	Mass position Between nodes	Rayleigh Viscous coefficient
- ♣ 3:FluidFlexible		
	Properties specified for each freedom	
in a literation		/
→ A 1:JointGeometric	Elastic spring stiffness 225.923E9 1.0E	15
⊨⊜ Joints (4)	Mass 0.0 0.0	
- \$ 1:Roof		
- 3:FluidFlexible		
S:Pile		
B Supports (2)		
-\$ 1:Fully Fixed		
- A 2:Y Free		
]	
	Name Pile	~ 🗧 (5)
None Specify		
Attributes	→ ↓ × Mass	×
🕼 Grou, , 🖧 Attri., 🕒 Anal., 🖉 Utilit., 🛄 Repo., 🗖	Lavers	
Evample Vertical EN1998 Vertical mdl		
Attributes (13)	Analysis category 2D Inplane	
⊕	Туре	
🕀 🚍 Geometric (1)	Point	Line Surface
🖨 🖾 Material (5)		
ia Joints (4)	X	Y
	0.0	53.6624E6
\$ 5:Pile		
🖃 📾 Mass (1)	Name NoRoofTank(M)	~ (4)
4:NoRoofTank(M)		
⊟⊜ Supports (2)		
3 1:Fully Fixed		
+++ 2:T Free	Close	Cancel Apply Help



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 🗸 🕈 🗙		LUSAS View: Example_Vertical_EN19	98_Vertical.mdl	Window 1 ×		
🐼 Grou 🗞 Attri 🚇 Anal 🦨 Utilit 💷 Repo 🗊 Layers	YX	0.0 1.0	2.0	3.0	4.0	5.0
Stample_Vertical_EN1998_Vertical.mdl Geometry Mesh	3.0	[≭] 35.9508E3	Properties	_		×
Beformed mesh Cathering Annotation Initiase Values : Fx (Force/Moment - 2D Joint (JNT3)) view properties	2.0	≆ [≭] -242.675E3 ≆	Value Results Entity Component Location Transform	Values Display Force/Moment Fx Averaged nodal Set None	>	
	1.0	¥ X	Display o	n slice(s)		
	0.0	-2522608E3 ★ ►► X		Close Cancel	Apply	Help
<pre>> Deformations x 3.01588E3</pre>	-1.0	z				

Fig 247 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.



Fig 248 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