LUSAS

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

Table of Contents

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.

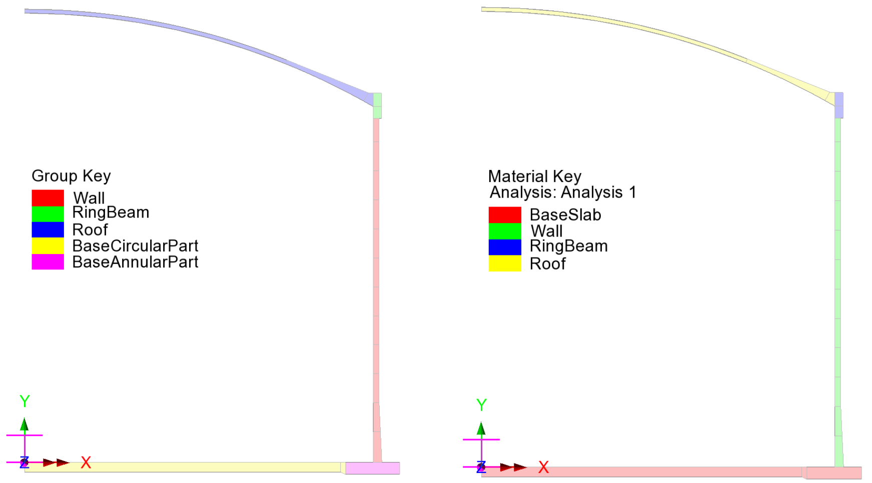


Fig 1 Group and Material Assignments for a 2D Axisymmetric Static Model

Support Condition for 2D Axisymmetric Model

Three support types are available for selection.

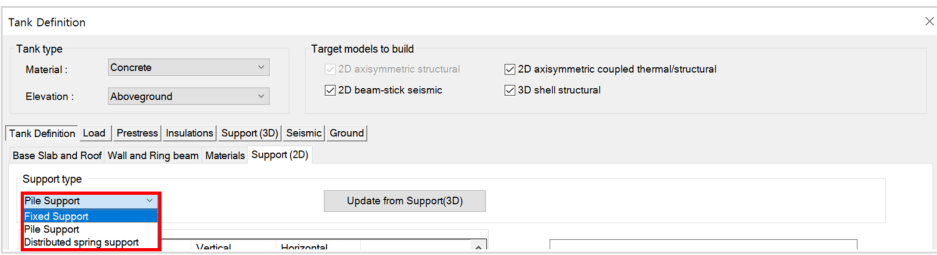


Fig 2 Support Types Available

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.

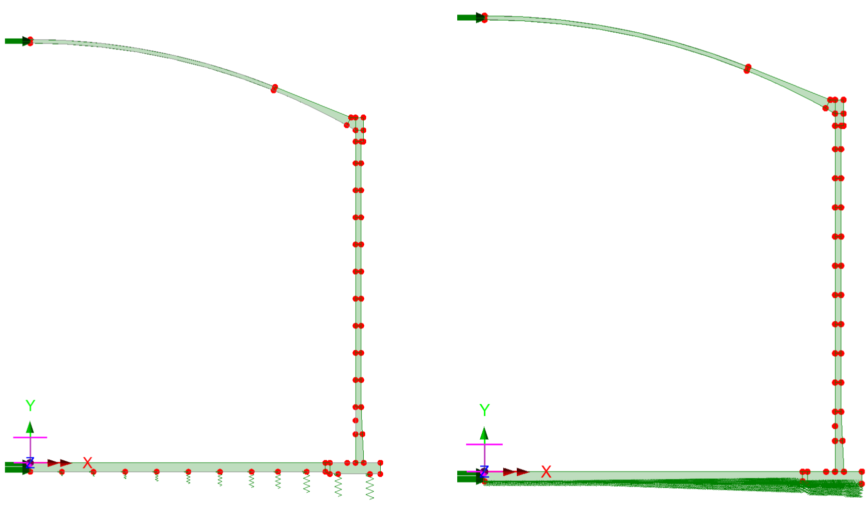


Fig 3 Support Types for a 2D Axisymmetric Static Model (Pile Support / Distributed Spring Support)

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.

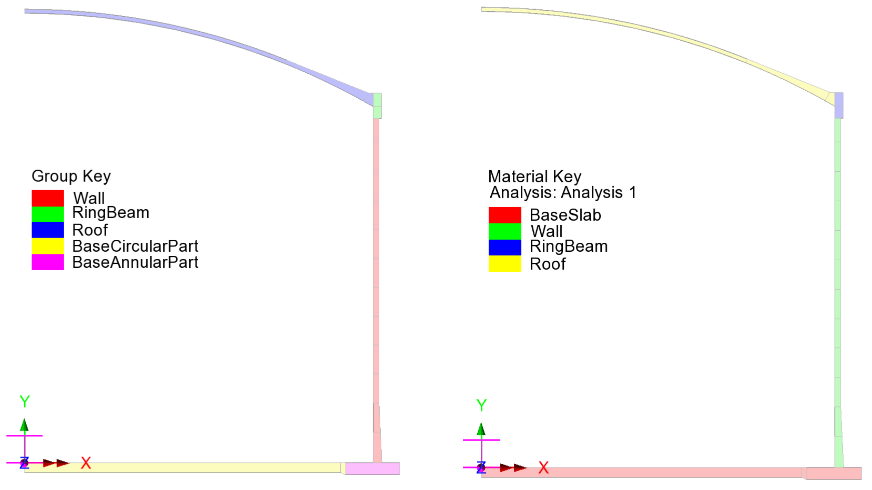


Fig 4 Group and Material Assignments in a 2D Axisymmetric Staged Construction Model

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.

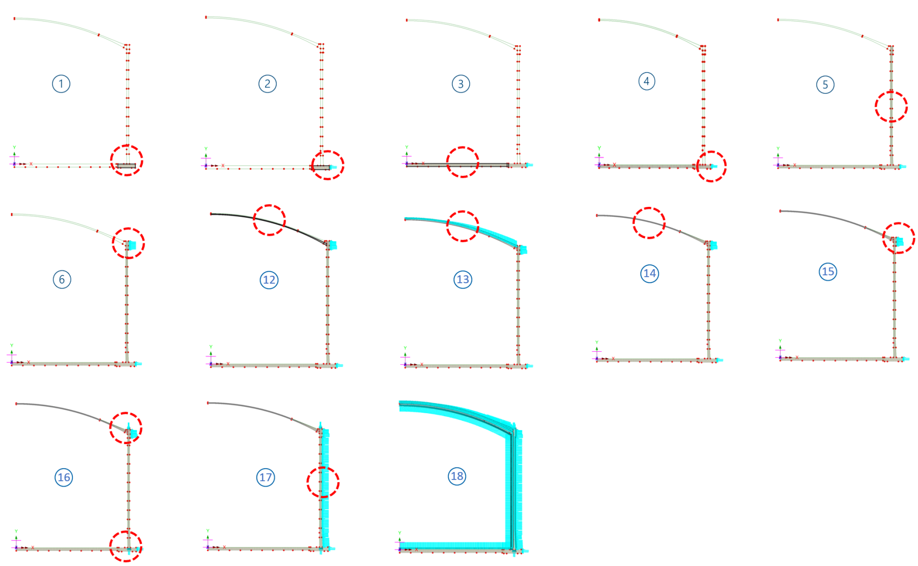


Fig 5 Activation and Deactivation in a Staged Construction Analysis Model – Layered roof option 1

Stage	Description	Note
No. 1	Annular part	
No. 2	1) + Base 1st PS	
No. 3	2) + Circular part	
No. 4	3) + Base 2nd PS	
No. 5	4) + Wall	
No. 6	5) + Ringbeam	
No. 7	6) + Ringbeam 1st 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 2nd 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

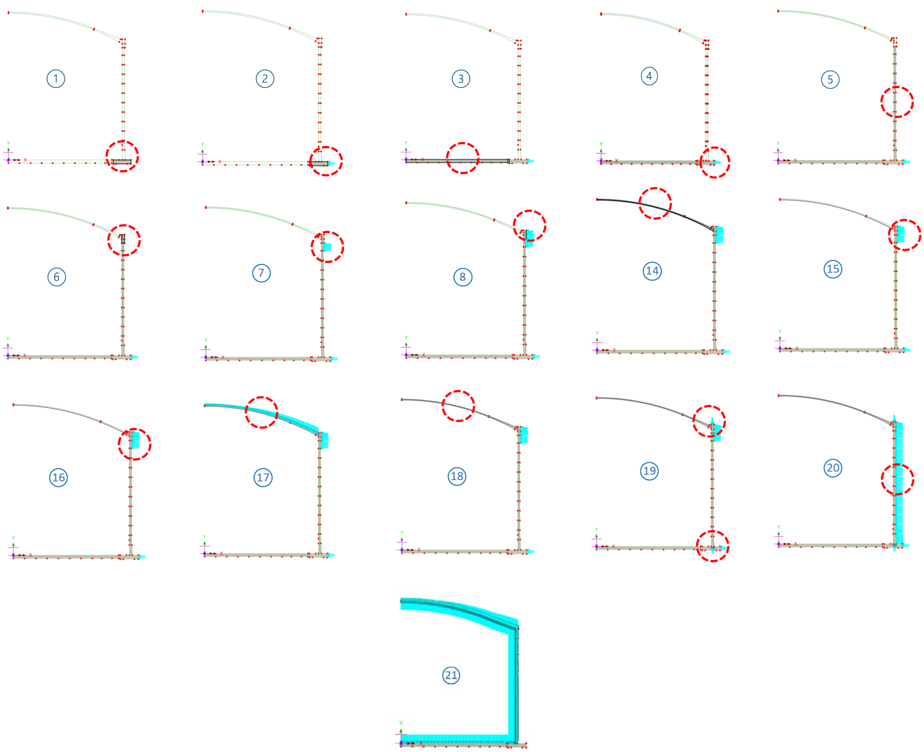


Fig 6 Activation and Deactivation in a Staged Construction Analysis Model – Layered roof option 2

Stage	Description	Note
No. 1	Annular part	
No. 2	1) + Base 1st 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 1st PS	
No. 8	7) + Ringbeam 1st 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 2nd PS	
No. 18	17) + Ringbeam 2nd PS	
No. 19	18) + Vertical PS	
No. 20	19) + Horizontal PS	
No. 21	20) + Operating Stage	Prestress short

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

Table 2 Sequence of Construction Stages – Layered roof option 2

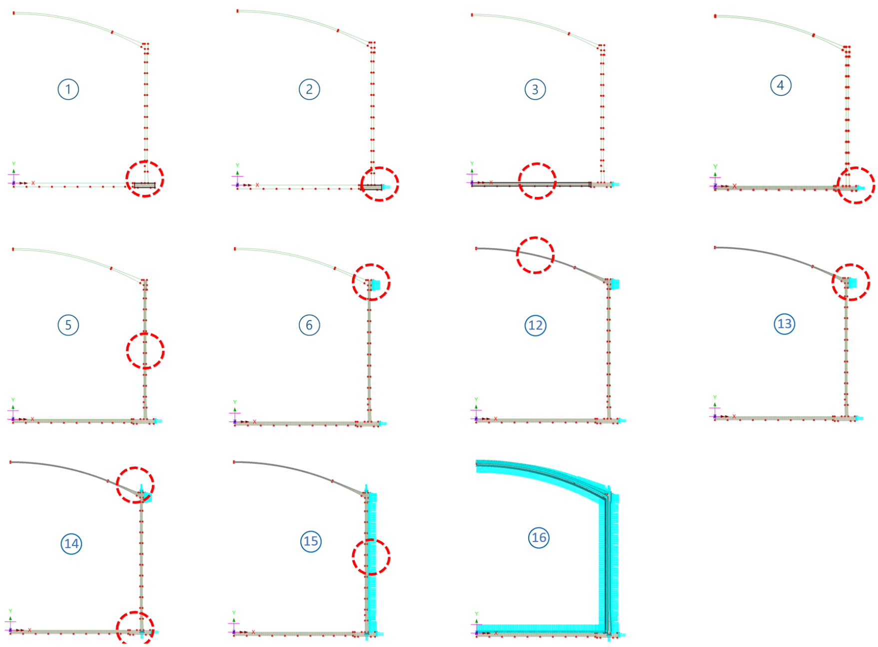


Fig 7 Activation and Deactivation in a Staged Construction Analysis Model

– Single Layered roof 1

Stage	Description	Note
No. 1	Annular part	
No. 2	1) + Base 1st 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 1st 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 2nd 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

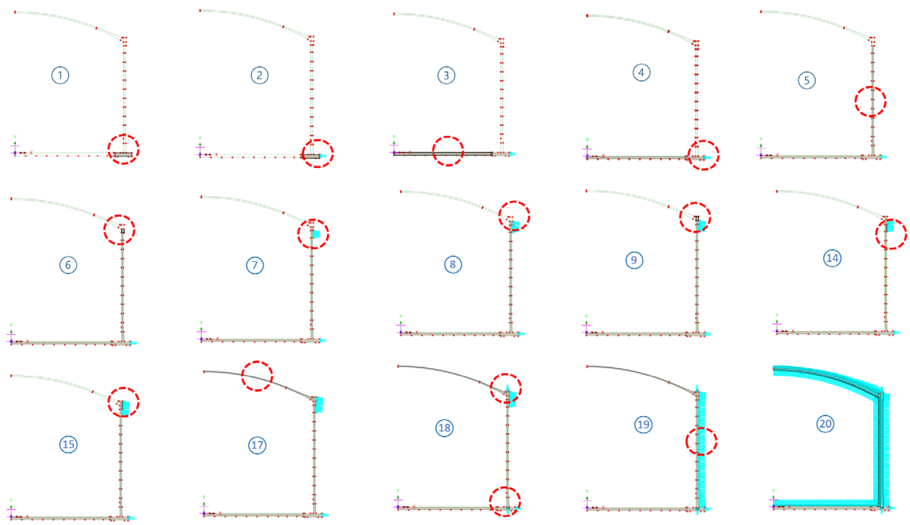


Fig 8 Activation and Deactivation in a Staged Construction Analysis Model

– Single Layered roof 2

Stage	Description	Note
No. 1	Annular part	
No. 2	1) + Base 1st PS	
No. 3	2) + Circular part	
No. 4	3) + Base 2nd PS	
No. 5	4) + Wall	
No. 6	5) + Ringbeam 1st	
No. 7	6) + Wall End 1st PS	
No. 8	7) + Ringbeam 1st 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 2nd PS	
No. 15	14) + Ringbeam 2nd 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

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.

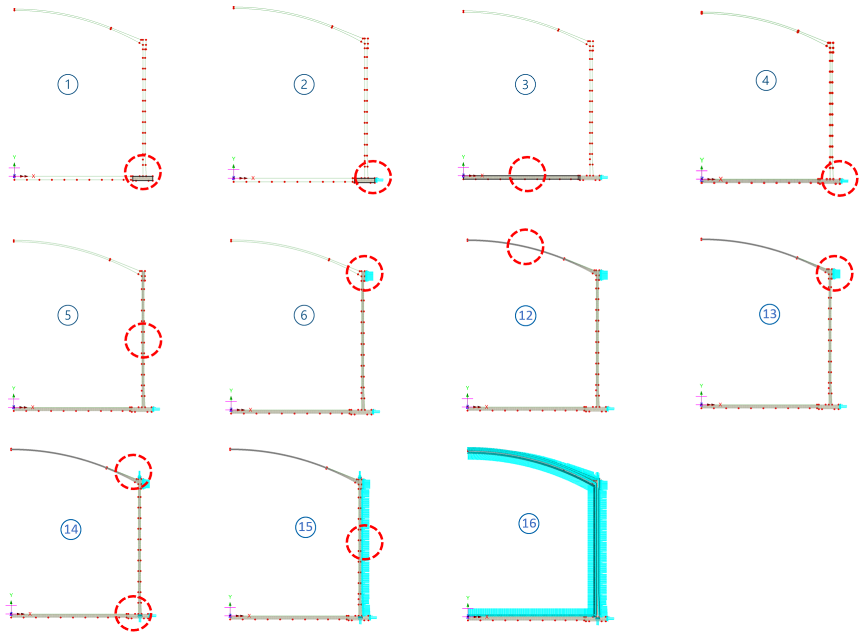


Fig 9 Birth and Death Staged Construction Analysis Model (Roof ratio for 1st built =1)

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

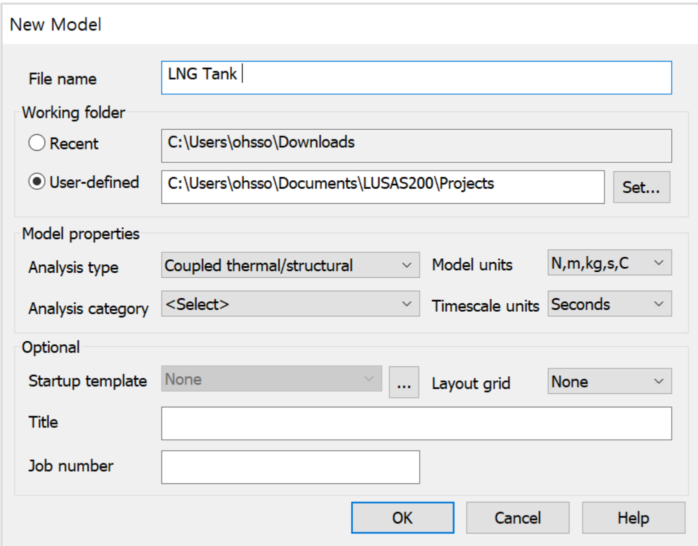


Fig 10 New Model Dialog Setting Thermal/Structural Coupled Analysis

Elements

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

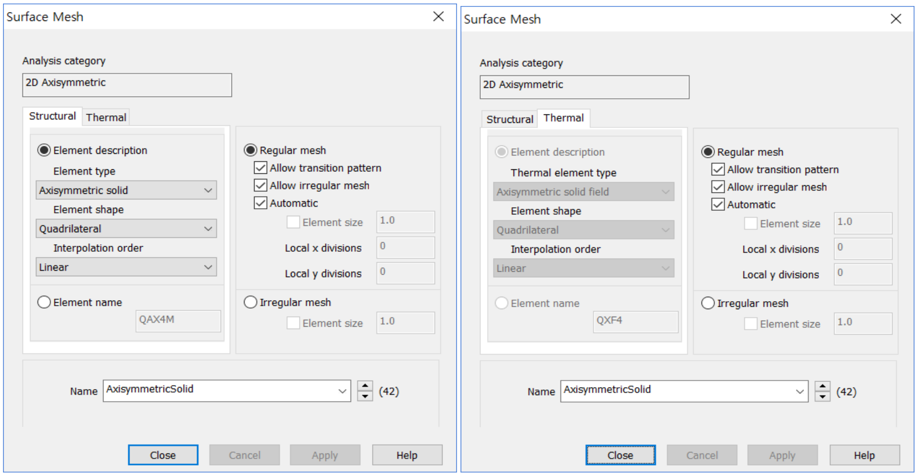


Fig 11 Element Definition for 2D Axisymmetric Thermal Analysis

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.

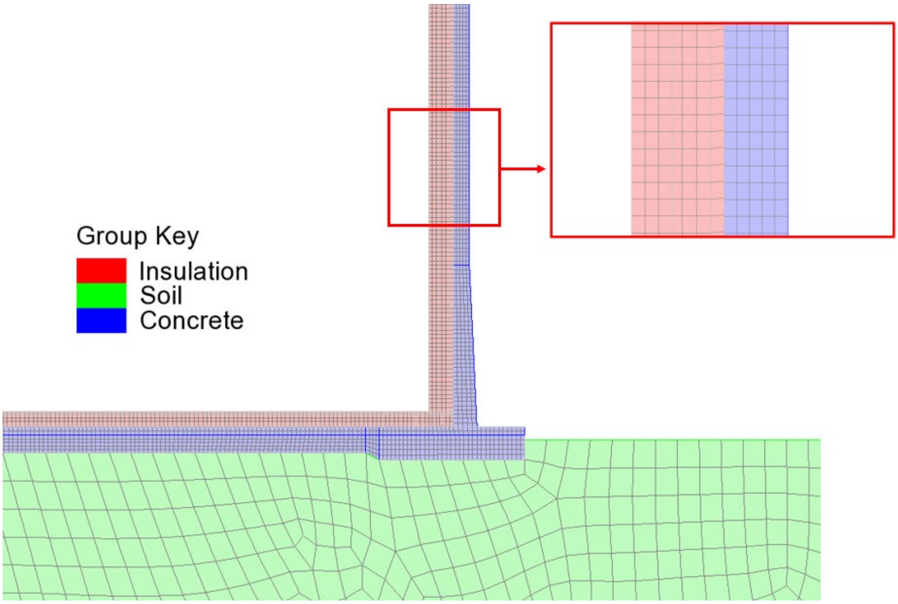


Fig 12 Insulation Elements Separated from Structure Elements

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.

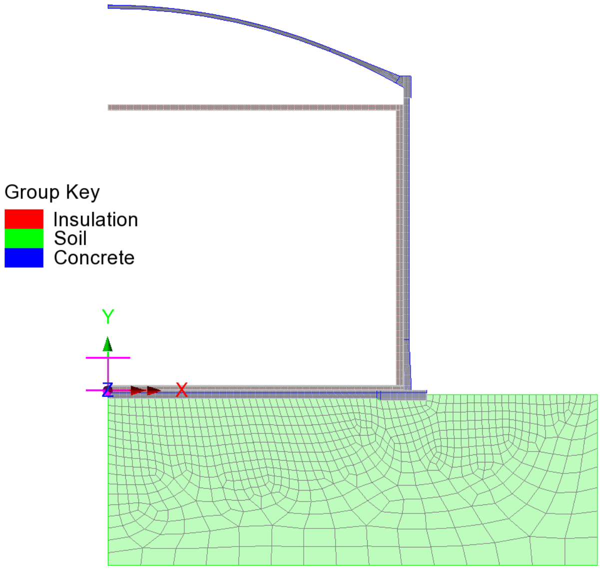


Fig 13 Mesh for 2D Axisymmetric Thermal Analysis

Groups / Materials

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

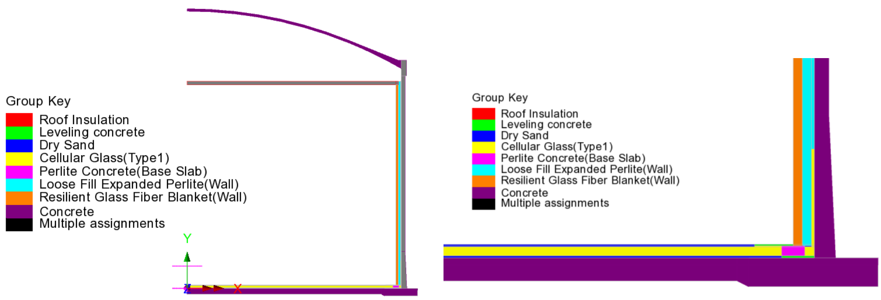


Fig 14 Group and Material Assignments in 2D Axisymmetric Thermal Analysis Model

Supports and Loading for Thermal Analysis

The 1st Loadcase

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

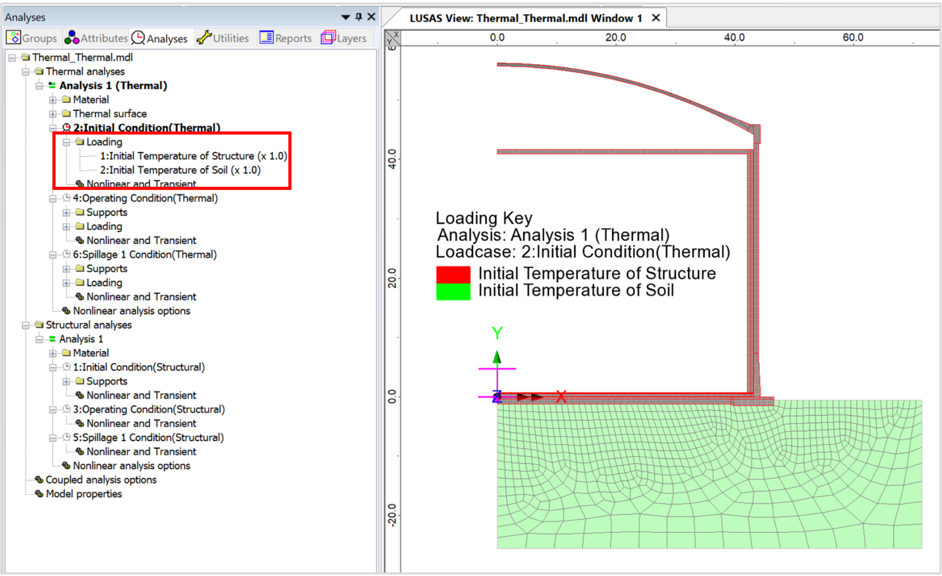


Fig 15 Thermal Analysis -1st Loadcase

The 2nd Loadcase

Liquid temperature is assigned to inner side of the insulation.

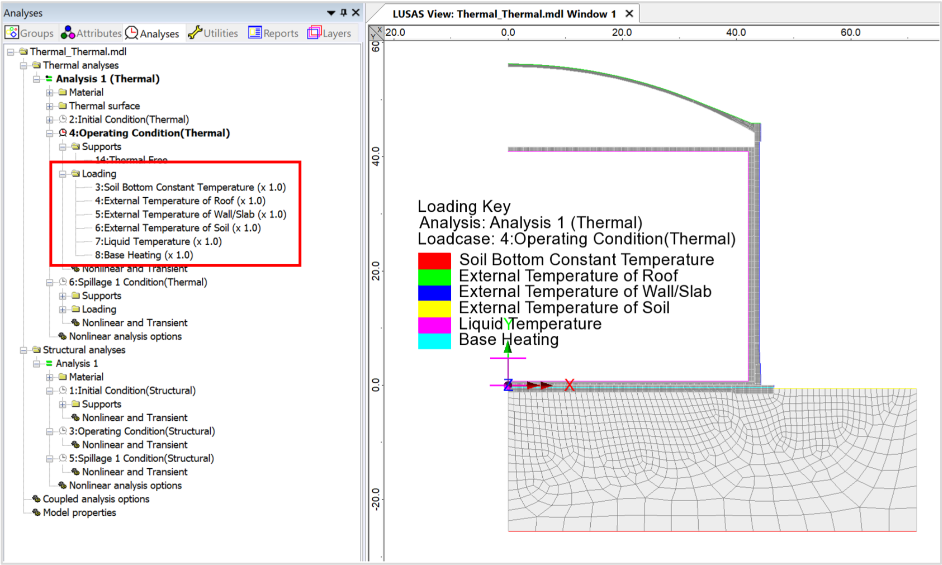


Fig 16 Thermal Analysis – 2nd Loadcase

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

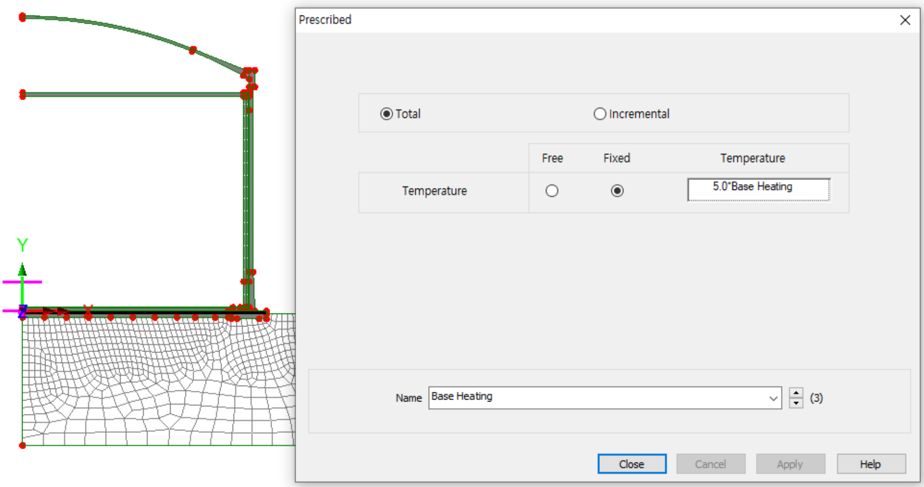


Fig 17 Base Heating Temperature in a 2D Axisymmetric Thermal Analysis Model

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.

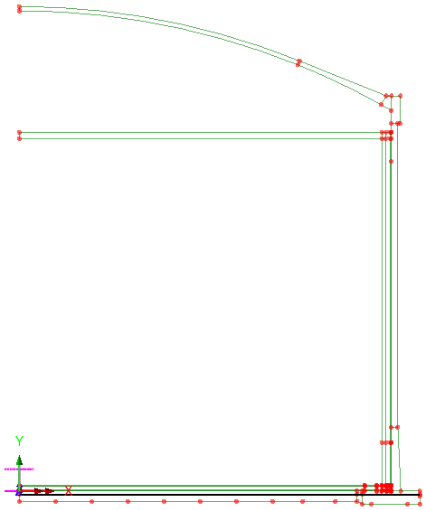


Fig 18 Base Heating Temperature in a 2D Axisymmetric Thermal Analysis Model

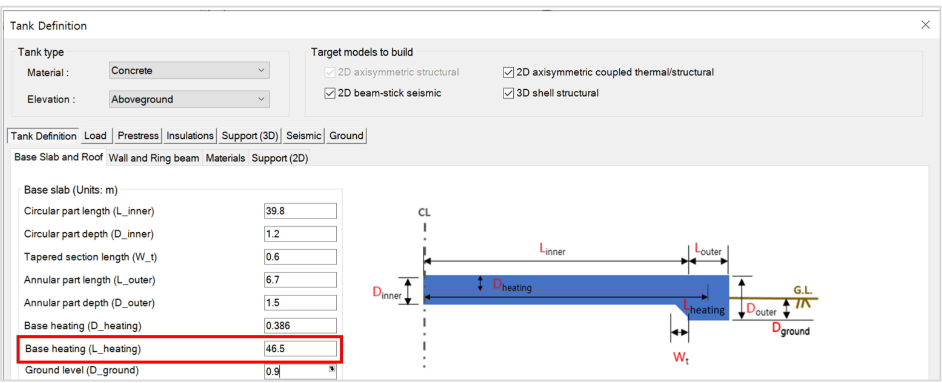


Fig 19 Base Heating Length for 2D Axisymmetric Thermal Analysis Model

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.

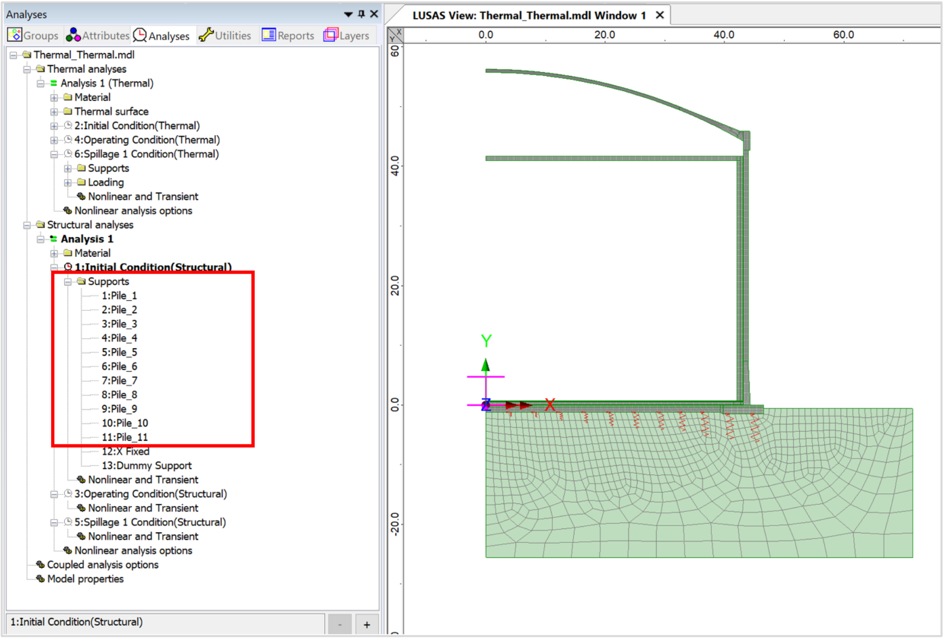


Fig 20 Pile Support for Structural Analysis following Thermal Analysis

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.

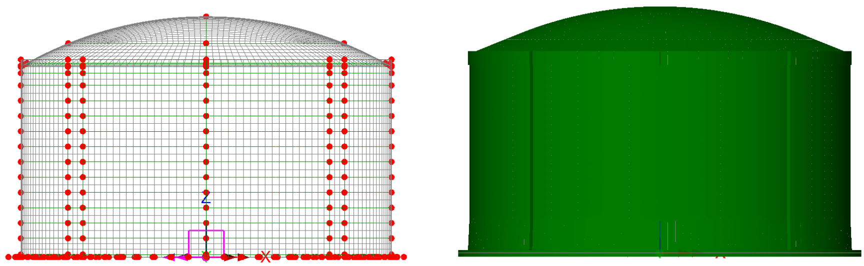


Fig 21 3D Shell Model for Static Analysis

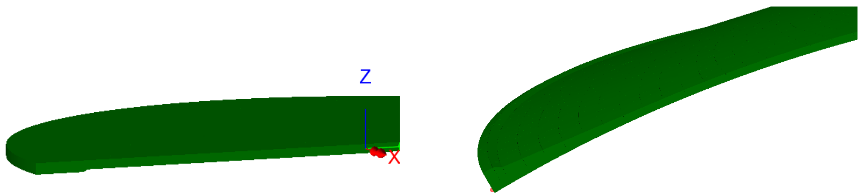


Fig 22 3D Shell Model Thickness Variation at Roof and Slab

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.

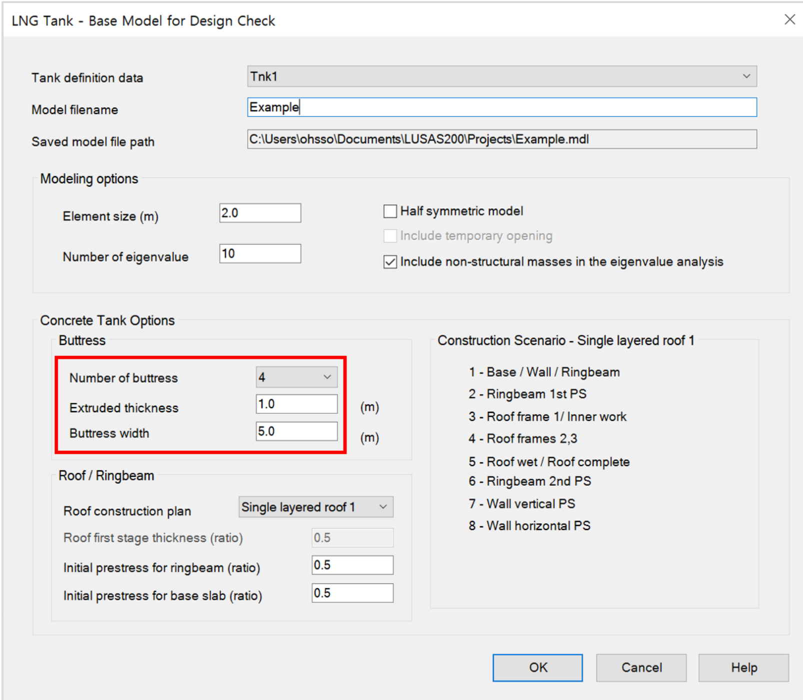


Fig 23 User Input for the Number of Buttresses in a 3D Shell Model

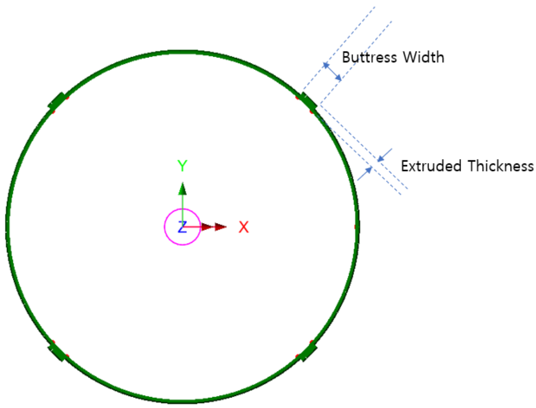


Fig 24 Buttress Definition for a 3D Shell Model

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.

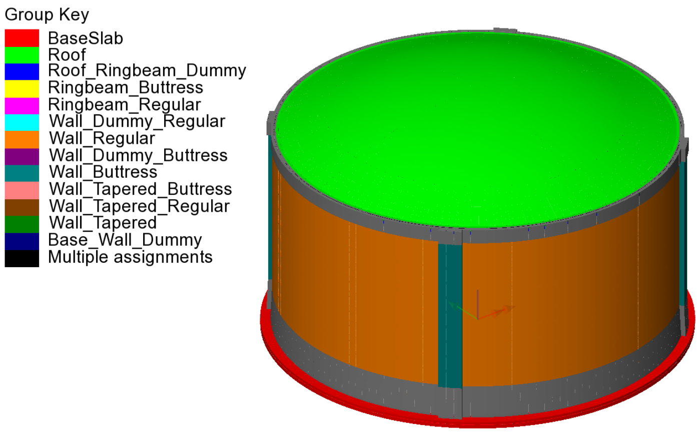


Fig 25 Groups in a 3D Shell Model

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

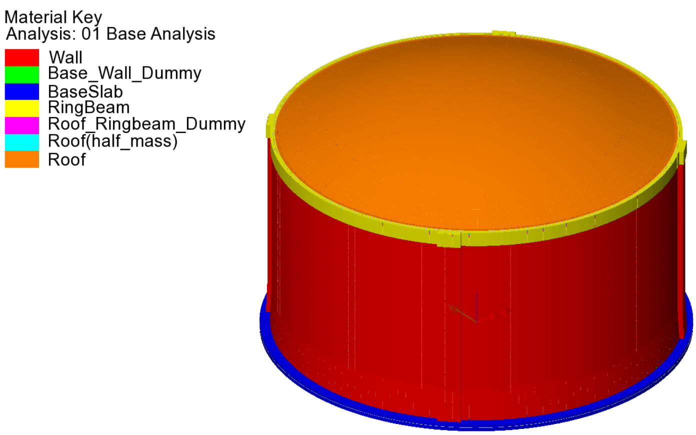


Fig 26 Material Assignments in a 3D Shell Model

Support Conditions

Three different types of support conditions can be defined.

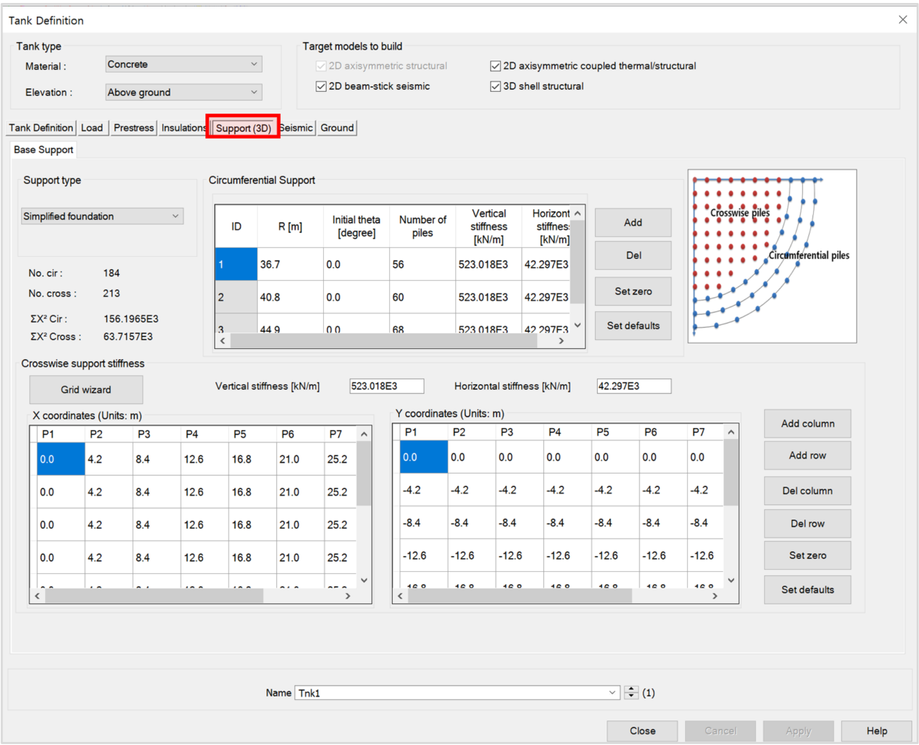
Fixed Support

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.



ID	R [m]	Initial theta [degree]	Number of piles	Vertical stiffness [kN/m]	Horizont stiffness [kN/m]
1	36.7	0.0	56	523.018E3	42.297E3
2	40.8	0.0	60	523.018E3	42.297E3
3	44.9	0.0	68	523.018E3	42.297E3

P1	P2	P3	P4	P5	P6	P7
0.0	4.2	8.4	12.6	16.8	21.0	25.2
0.0	4.2	8.4	12.6	16.8	21.0	25.2
0.0	4.2	8.4	12.6	16.8	21.0	25.2
0.0	4.2	8.4	12.6	16.8	21.0	25.2

P1	P2	P3	P4	P5	P6	P7
0.0	0.0	0.0	0.0	0.0	0.0	0.0
-4.2	-4.2	-4.2	-4.2	-4.2	-4.2	-4.2
-8.4	-8.4	-8.4	-8.4	-8.4	-8.4	-8.4
-12.6	-12.6	-12.6	-12.6	-12.6	-12.6	-12.6

Fig 27 Input for Pile Locations and Stiffnesses

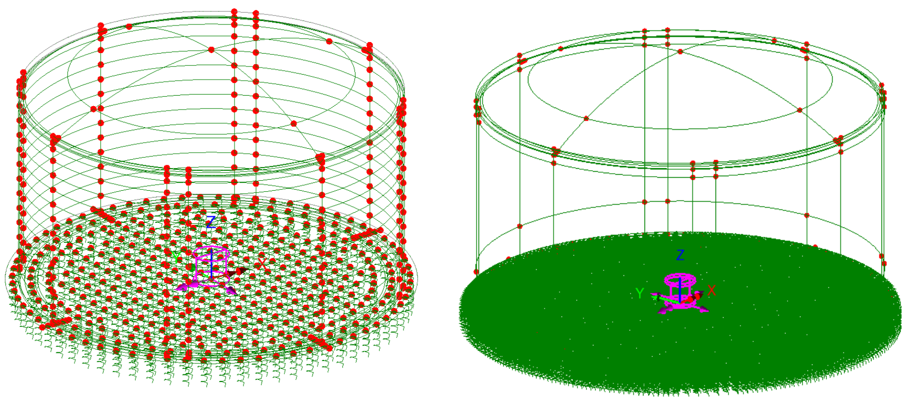


Fig 28 Support Condition for a 3D Shell Model (Pile Support / Regular Support)

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.

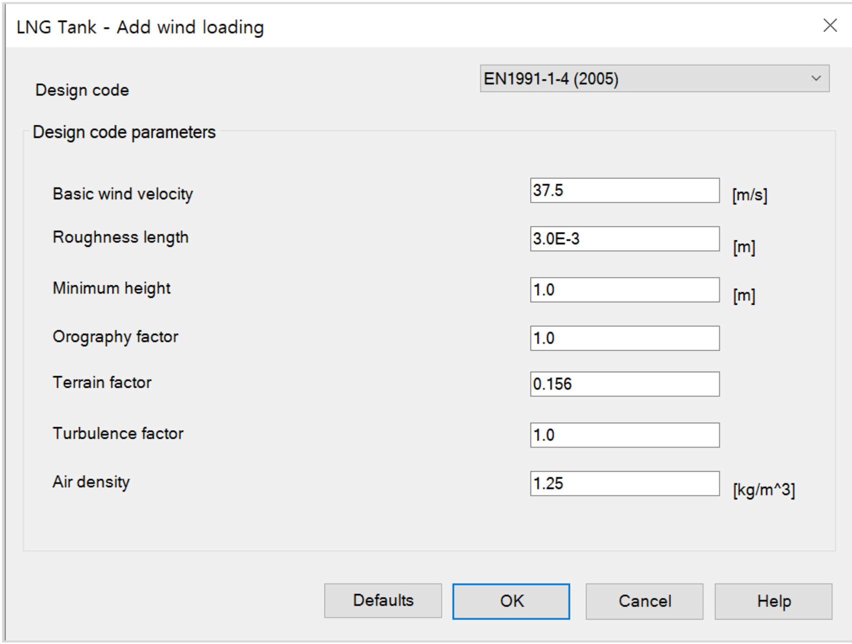


Fig 29 User Input for Wind Load for a 3D Shell Model

Other Options

Half Only Model

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

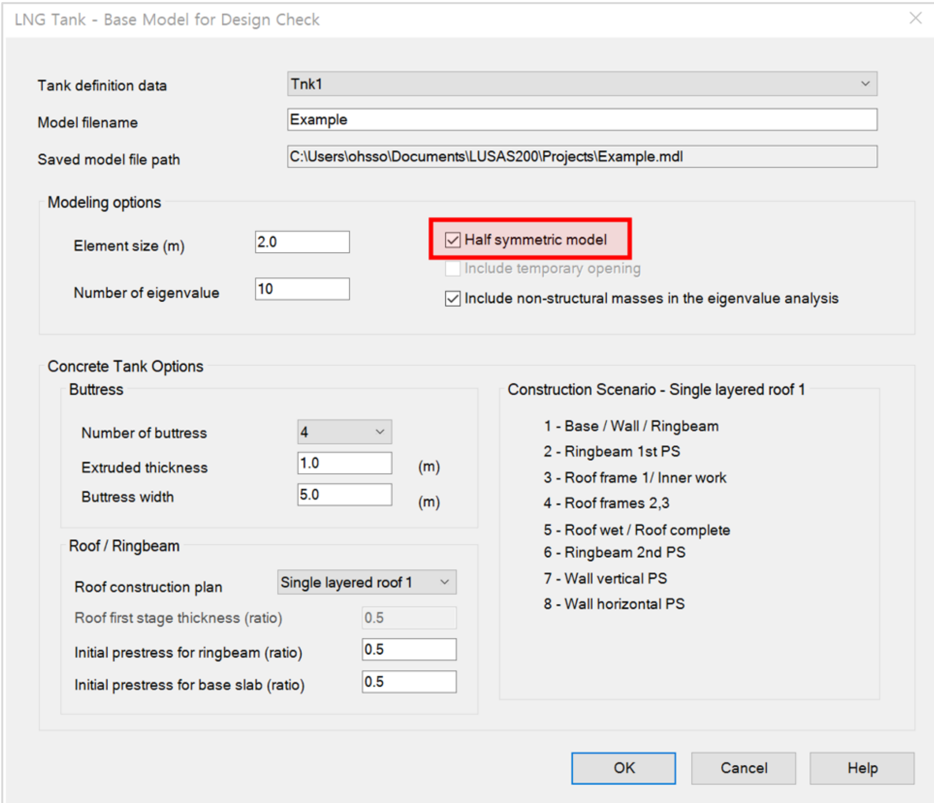


Fig 30 User Input for a 3D Shell Model (Half Model)

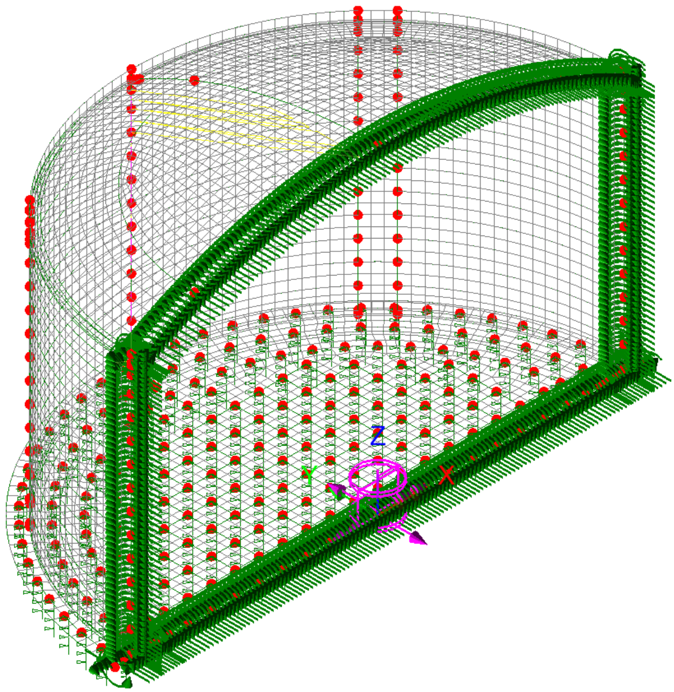


Fig 31 3D Shell Model (Half Model)

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.'

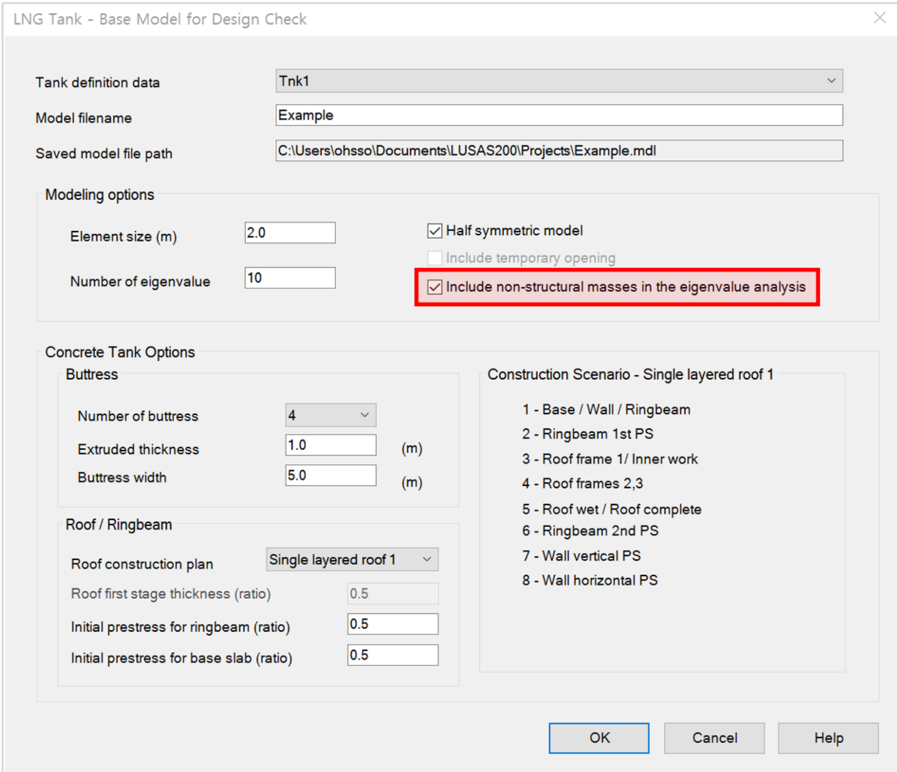


Fig 32 User Input for Eigenvalue Analysis Model including Non-Structural Masses

Summary of Mass Calculation

DIMENSION

Component	Dimension(m)
Inner Tank Radius	42.1
Tank Height	40.06
LNG Height	38.92

SUMMARY FOR MASS

Component	Volume	Unit mass	Structural mass	Total mass	Equivalent unit mass
	m^3	kg/m^3	kg	kg	kg/m^3
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)
Roof	Concrete Roof (= Roof volume * unit concrete mass)	9,917,753
	Roof liner + steel roof structure	1,400,000
	Suspended deck + insulation of the suspended ceiling	135,000
	Roof nozzles	42,000
	Roof platform	400,000
	Roof pump & crane	30,000
	Roof piping and support	103,000
	Others	-
	Total	**12,027,753**
Ring Beam	Concrete Ring Beam (= Ring Beam volume * unit concrete mass)	2,307,751
	wall barrier plate	-
	wall piping and support	-
	Others	-
	Total	**2,307,751**
Outer Concrete Wall	Concrete Wall (= Wall volume * unit concrete mass)	22,806,425
	corner protection	242,000
	wall barrier plate	494,000
	wall piping and support	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 (= Steel tank volume * steel mass)	2,479,105
	shell stiffener	45,000
	shell insulation(50%)	-
	top girder	-
	Others	275,000
	Total	**2,799,105**
LNG	LNG (= LNG volume * 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.

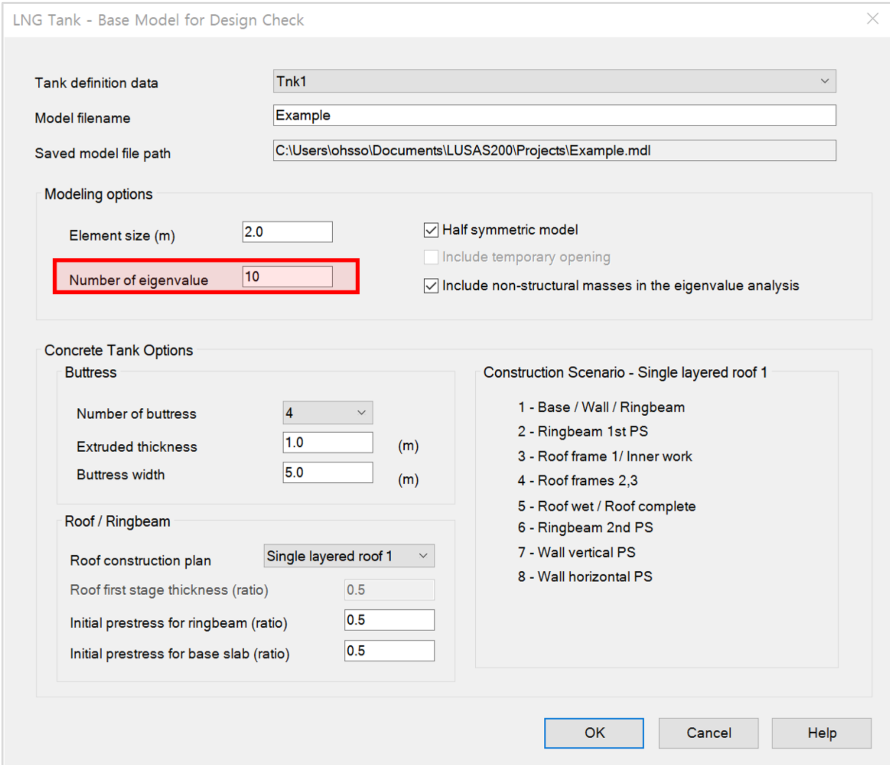


Fig 34 User Input for a 3D Shell Model for Eigenvalue Analysis

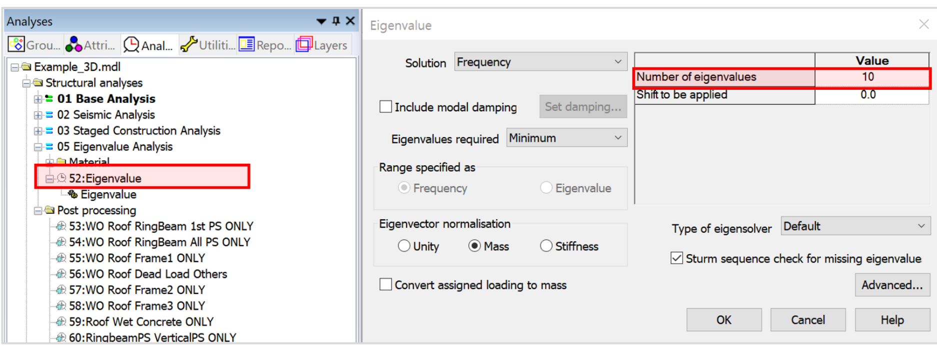


Fig 35 Eigenvalue Analysis in a 3D Shell Model

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].

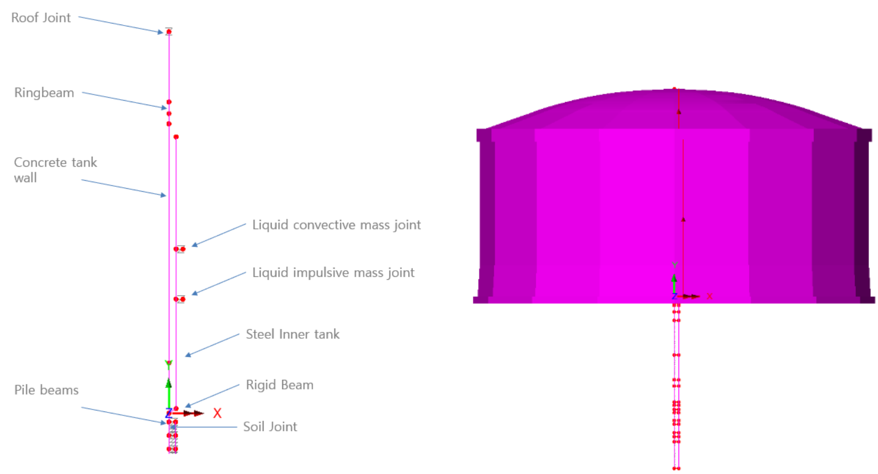


Fig 36 Beam-Stick Modelling Concept for Horizontal Actions

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].

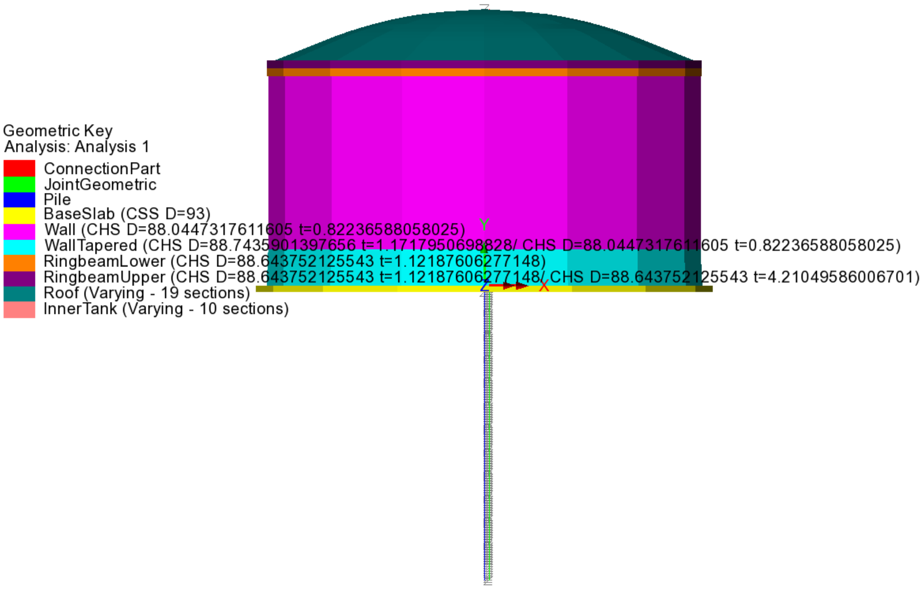


Fig 37 Geometric Properties in a Beam-Stick Horizontal Model

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])

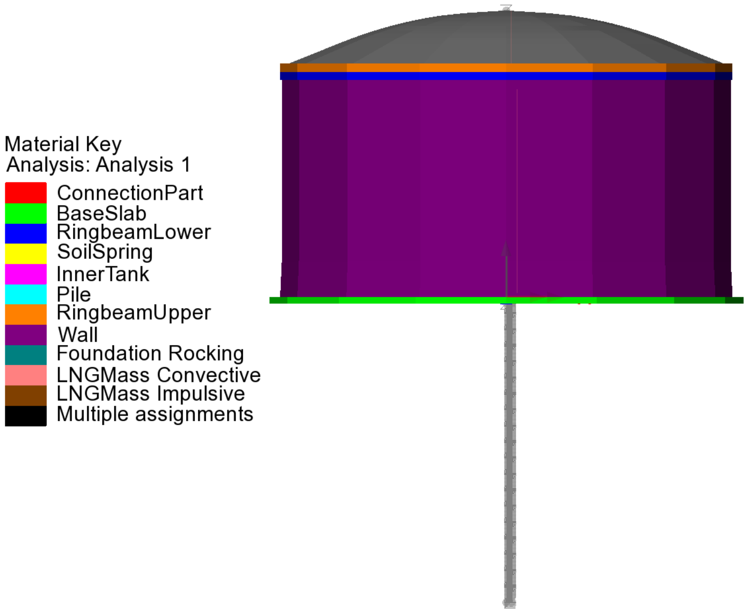


Fig 38 Material Properties in a Beam-Stick Horizontal Model

Summary of Beam-Stick Model — EN 1998-4

DIMENSION

Component	Dimension(m)
Inner Tank Radius	42.1
Tank Height	40.06
LNG Height	38.92

SUMMARY FOR MASS

Component	Volume	Unit mass	Structural mass	Total mass	Equivalent unit mass
	m^3	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

SUMMARY FOR CALCULATED PROPERTIES

1) Horizontal Model

Component	mass	Lever arm height	stiffness	Reference
	mc, Kg	hc, m	kc, N/m	
LNG Convective	50,527,854	23.53	19,974,995	EN 1998-4
LNG Impulsive	53,494,849	16.13	11,325,839,357	EN 1998-4

2) Vertical Model

Component	mass	stiffness	Reference
	m, Kg	k, N/m	
LNG Flexible	89,566,808	21,631,229,542	EN 1998-4
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	EN 1998-4

MASS DETAILS

Component	Descriptions	Mass (kg)
Roof	Concrete Roof (= Roof volume * unit concrete mass)	9,954,938
	Roof liner + steel roof structure	1,400,000
	Suspended deck + insulation of the suspended ceiling	135,000
	Roof nozzles	42,000
	Roof platform	400,000
	Roof pump & crane	30,000
	Roof piping and support	103,000
	Others	-
	Total	**12,027,753**
Ring Beam	Concrete Ring Beam (= Ring Beam volume * unit concrete mass)	2,467,751
	wall barrier plate	-
	wall piping and support	-
	Others	-
	Total	**2,467,751**
Outer Concrete Wall	Concrete Wall (= Wall volume * unit concrete mass)	24,940,428
	corner protection	242,000
	wall barrier plate	494,000
	wall piping and support	88,000
	Others	-
	Total	**25,764,428**
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 (= Steel tank volume * steel mass)	2,479,105
	shell stiffener	45,000
	shell insulation(50%)	-
	top girder	-
	Others	275,000
	Total	**2,799,105**
LNG	LNG (= LNG volume * unit LNG mass)	104,022,703
	Total	**104,022,703**

Fig 39 Mass Summary for the Beam-Stick Model

Verification for Beam-Stick Model	ACI350.3

DIMENSION

Component	Dimension(m)
Inner Tank Radius	42.1
Tank Height	40.06
LNG Height	38.92

SUMMARY FOR MASS

Component	Volume	Unit mass	Structural mass	Total mass	Equivalent unit mass
	m^3	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

CALCULATED PROPERTIES FOR HORIZONTAL MODEL

1) LNG Mass & Height

Component	mass	Lever arm height (IBP)	Lever arm height (EBP)
	mc(mi), Kg	hc(hi), m	hc(hi), m
LNG Convective	48,423,453	31.83	23.10
LNG Impulsive	52,963,803	33.36	14.60

2) LNG convective stiffness

Component	Value	Unit	Remark
g	9.8070	m/sec^2	gravitational acceleration
λ	5.8106	$m^{1/2}/s$	coefficient as defined in 9.3.4
ωc	0.6332	rad/s	circular frequency of oscillation of the first(convective)
Tc	9.9223	s	natural period of the first (convective) mode of sloshing
kc	19,417,270	N/m	

3) LNG impulsive stiffness

Component	Value	Unit	Remark
tw	29.7905	mm	average wall thickness (inner tank)
Es	2.00E+05	MPa	modulus of elasticity of inner tank
ρ_c	7.8500	$kN.s^2/m^4$	mass density of inner tank
C_w	0.1586		coefficients for determining the fundamentalfrequency
C_l	0.0422		coefficients for determining the fundamentalfrequency
ωi	5.473	rad/s	circular frequency of the impulsive mode of vibration
Ti	1.148	s	fundamental period of oscillation of the tank (plus the impulsive component of the contents)
ki	1,586,485,989	N/m	

CALCULATED PROPERTIES FOR VERTICAL MODEL

1) Roof Mass & Stiffness

Component	Value	Unit	Remark
$m_{\_roof}$	12,027,753	kg	mass of roof
f	-	Hz	fundamental frequency of oscillation of the roof
T	N/A	s	fundamental period of oscillation of the roof
$k_{\_roof}$	N/A	N/m	

2) LNG Mass & Stiffness

Component	Value	Unit	Remark
$m_{\_LNG}$	104,022,703	kg	mass of LNG
tw	29.7905	mm	average wall thickness (inner tank)
Es	2.00E+05	MPa	modulus of elasticity of inner tank
ρ_l	480.0000	kg/m^3	mass density of LNG
g	9.8070	m/sec^2	gravitational acceleration
ΥL	4.7074	kN/m^3	specific weight of contained liquid
T_v	0.4504	s	fundamental period of oscillation of the LNG
$k_{\_LNG}$	20,247,300,685	N/m	

3) Mass for Outer&Inner Tank

Component	Value	Unit	Remark
$m_{\_OuterInnerTank}$	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 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].

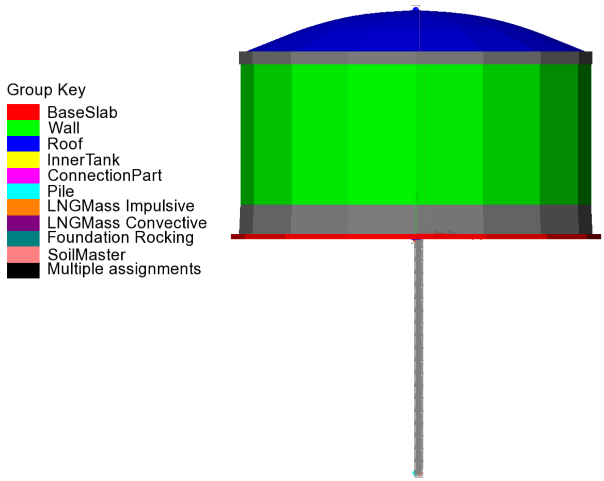
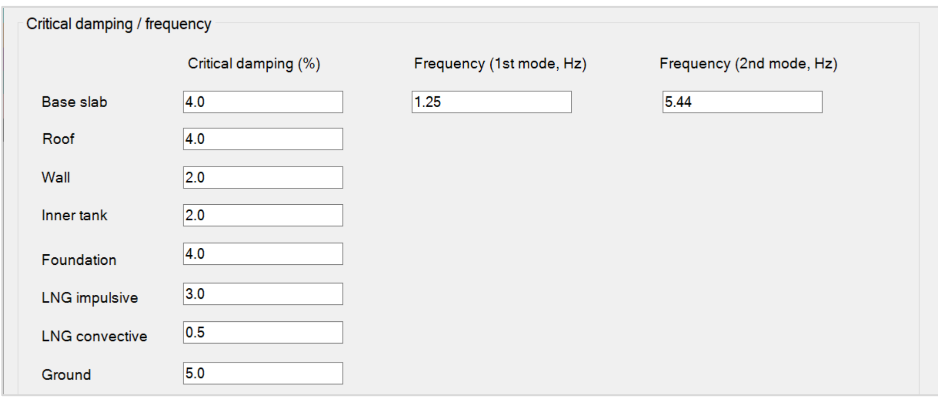


Fig 41 Groups in a Beam-Stick Horizontal Model

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 / frequency

	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.

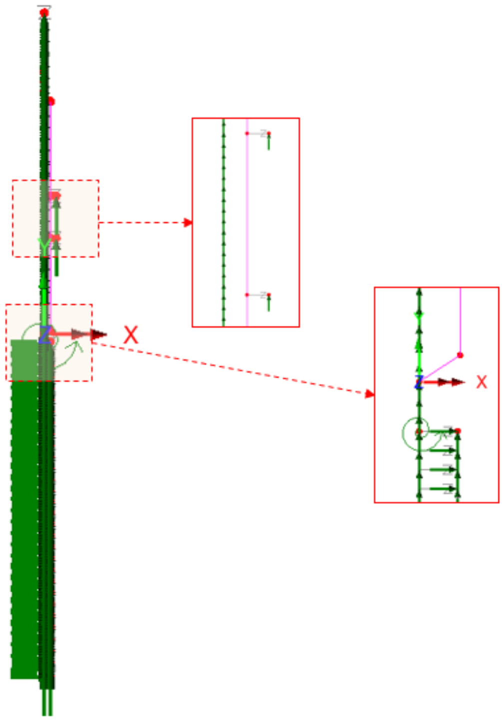


Fig 43 Support Conditions in a Beam-Stick Horizontal Model

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.

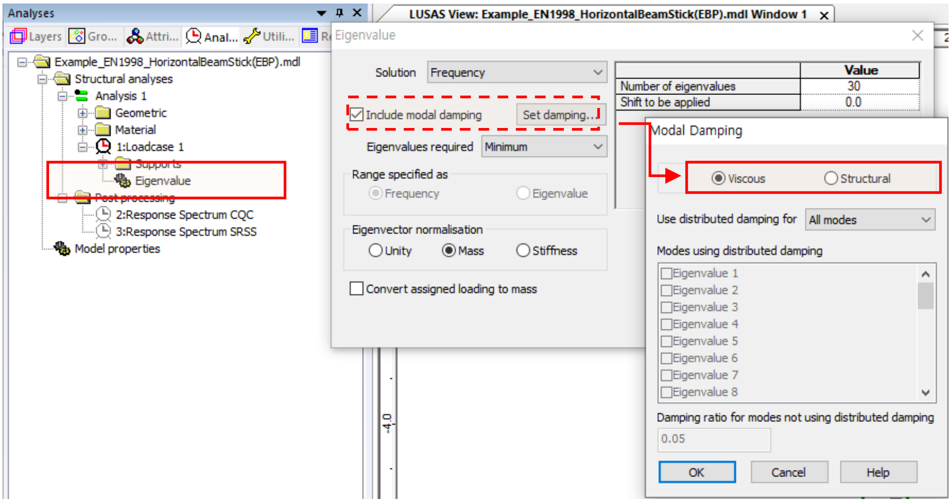


Fig 44 Eigenvalue Control for a Beam-Stick Horizontal Model

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.

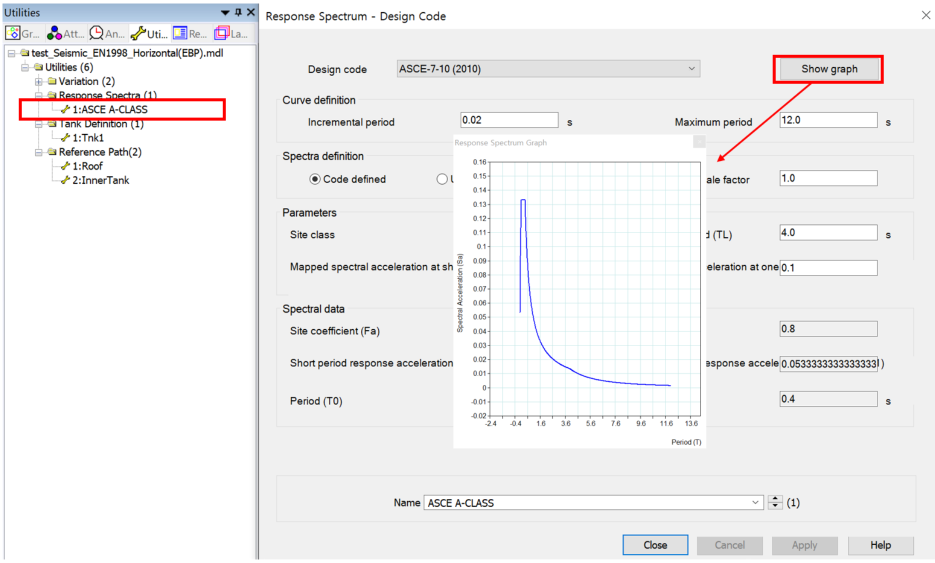


Fig 45 Default design Response Spectrum for a Beam-Stick Horizontal Model

Options for Post-Processing

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

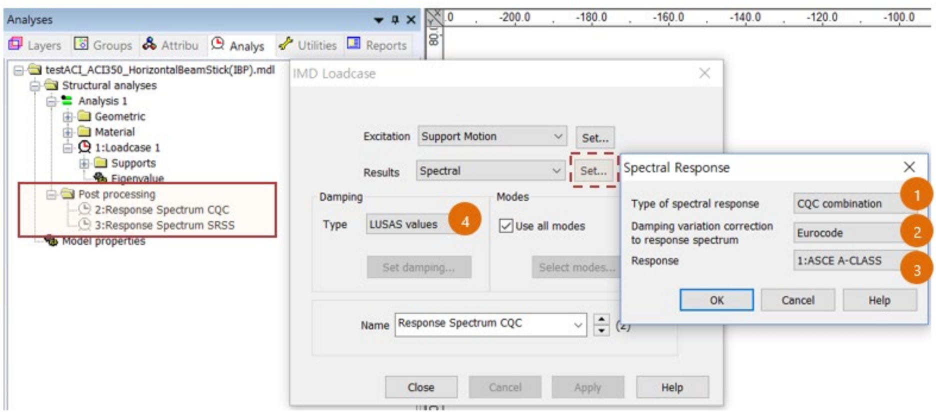


Fig 46 Post Processing Options for a Beam-Stick Horizontal Model

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].

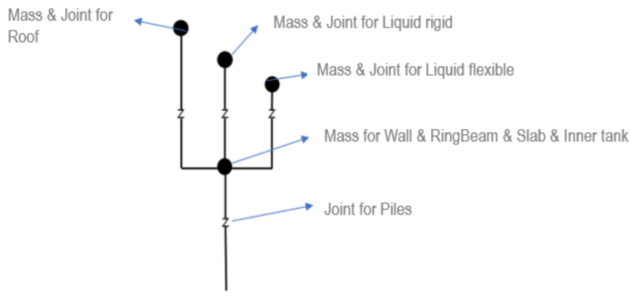


Fig 47 Beam-Stick Modelling Concept for Vertical Actions

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.

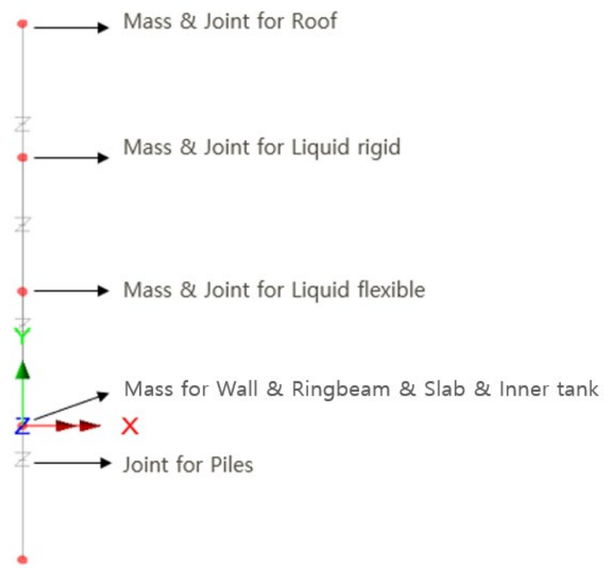


Fig 48 Beam-Stick Model for Vertical Actions

Geometric Properties

The following dataset is used.

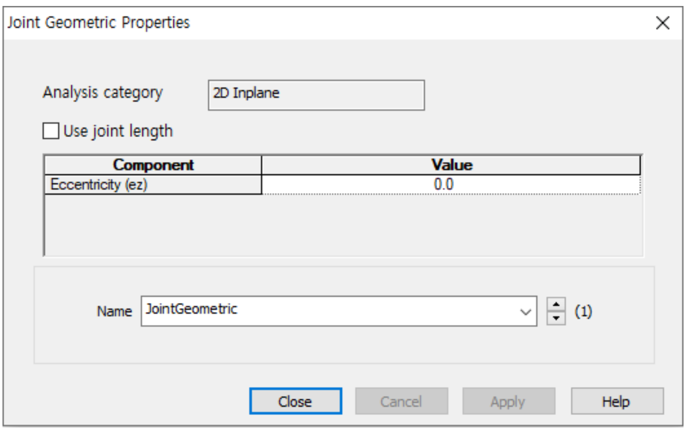


Fig 49 Geometric Properties for Joint Elements for Beam-Stick Vertical model

Material Properties

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

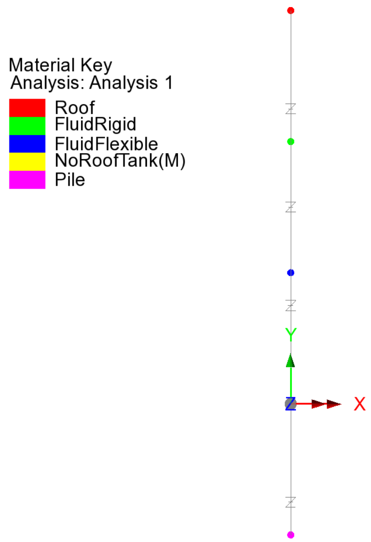


Fig 50 Material Properties in Beam-Stick Vertical Model

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
ρ_l	480.0000	kg/m^3	mass density of LNG
Es	2.00E+11	N/m^2	modulus of elasticity of inner tank material
ν	0.3		poisson ratio of steel
s(ζ)	0.0361	m	wall thickness for ζ = 1/3 (ζ = z/H_L)
f(γ)	1.0565		0.8<=γ<4 : 1.078+0.274 ln (γ) , γ<0.8 : 1 (A.41a, A41b)
P_{vf}	16,085	kg/m^2	hydrodynamic pressure on the wall base, from A.40.
$m_{\_LNG\_f}$	89,566,808	kg	mass of LNG (radial breathing), ref. A.40.
$m_{\_LNG\_r}$ (1)	52,900,941	kg	mass of LNG (rigidly moving) = sqrt($m_{\_LNG\_total}^2$-$m_{\_LNG\_f}^2$)
$m_{\_LNG\_r}$ (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		$=H_L/R$
γ_1	1.699140		=π / (2γ)
$I_0(\gamma_1)$	1.8629		bessel function order 0
$I_1(\gamma_1)$	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
$k_{\_LNG\_f}$	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&Inner Tank			
Component	Value	Unit	Remark
$m_{\_OuterInnerTank}$	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].

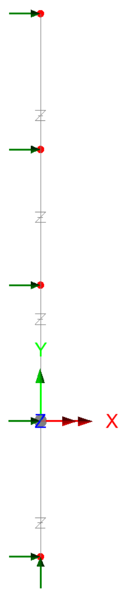


Fig 53 Supports in a Beam-Stick Vertical Model

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.

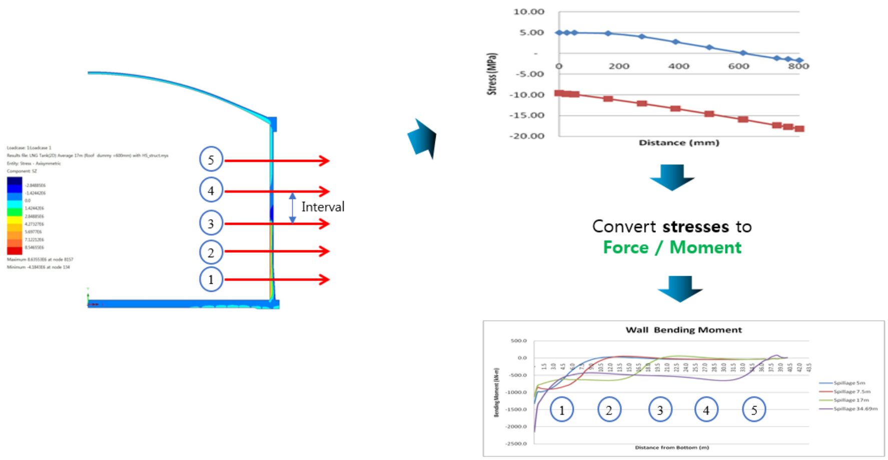


Fig 54 Converting Stress to Forces

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.

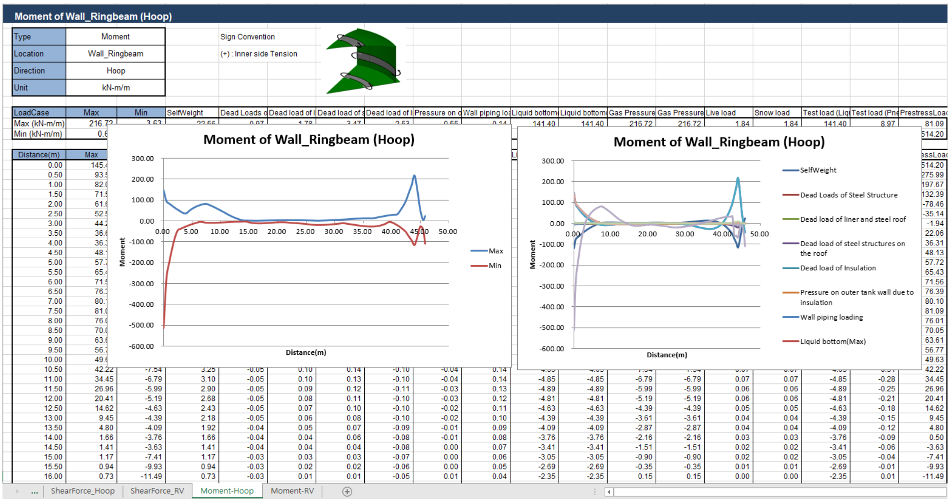


Fig 55 Section Force Spreadsheet for 2D Axisymmetric Solid Model

Roof - Exporting Forces

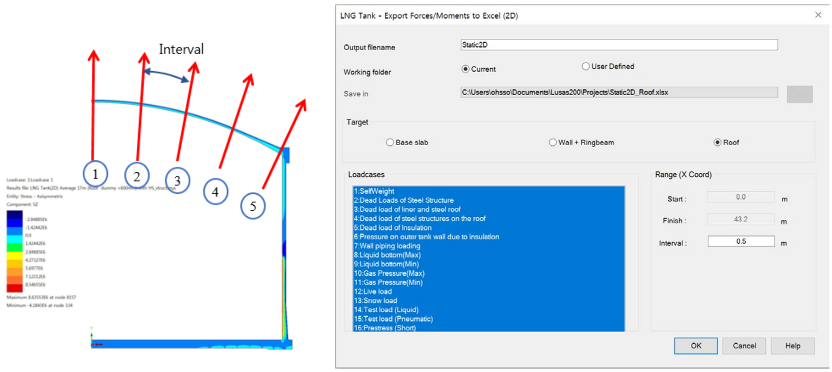


Fig 56 Exporting Forces for a 2D Axisymmetric Solid Model (Roof)

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

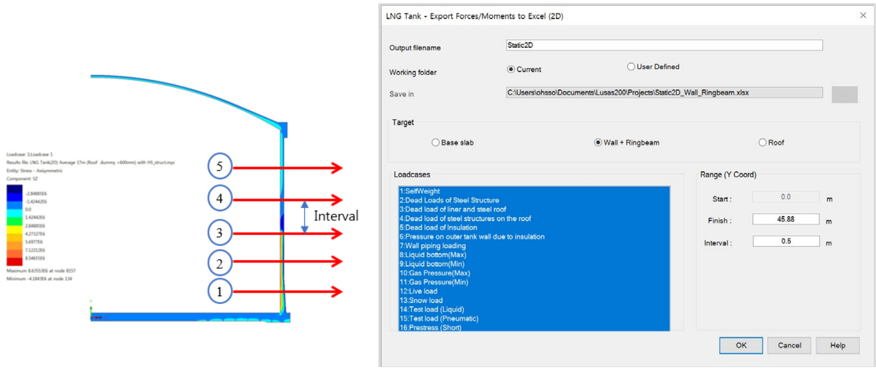


Fig 57 Exporting Forces for a 2D Axisymmetric Solid Model (Wall)

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

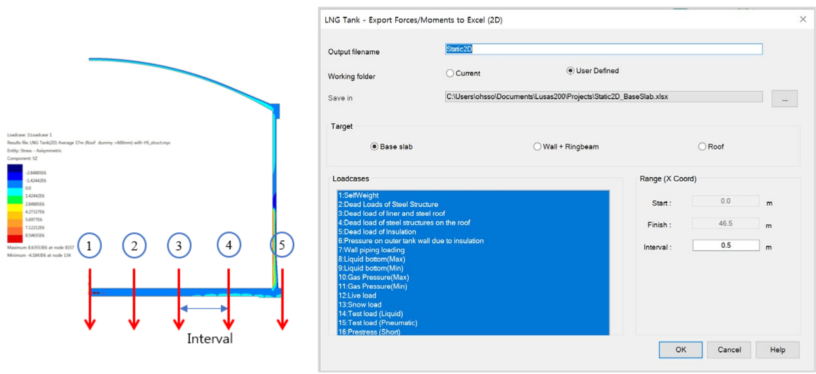


Fig 58 Exporting Forces for a 2D Axisymmetric Solid Model (Base Slab)

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.

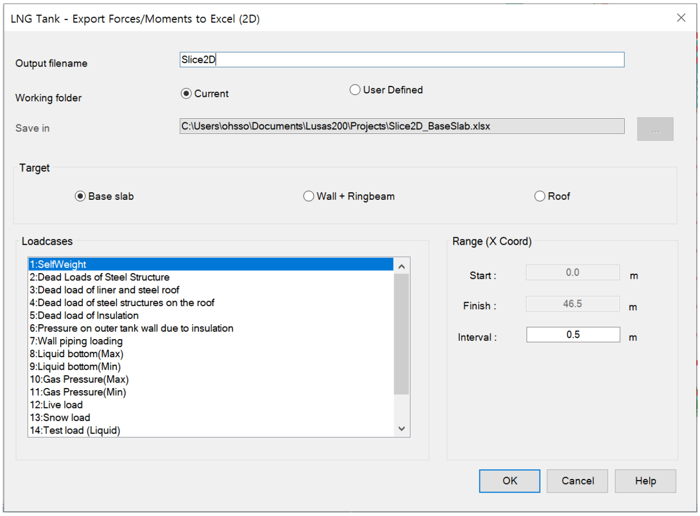


Fig 59 Exporting Forces for a 2D Axisymmetric Solid Mode

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**.

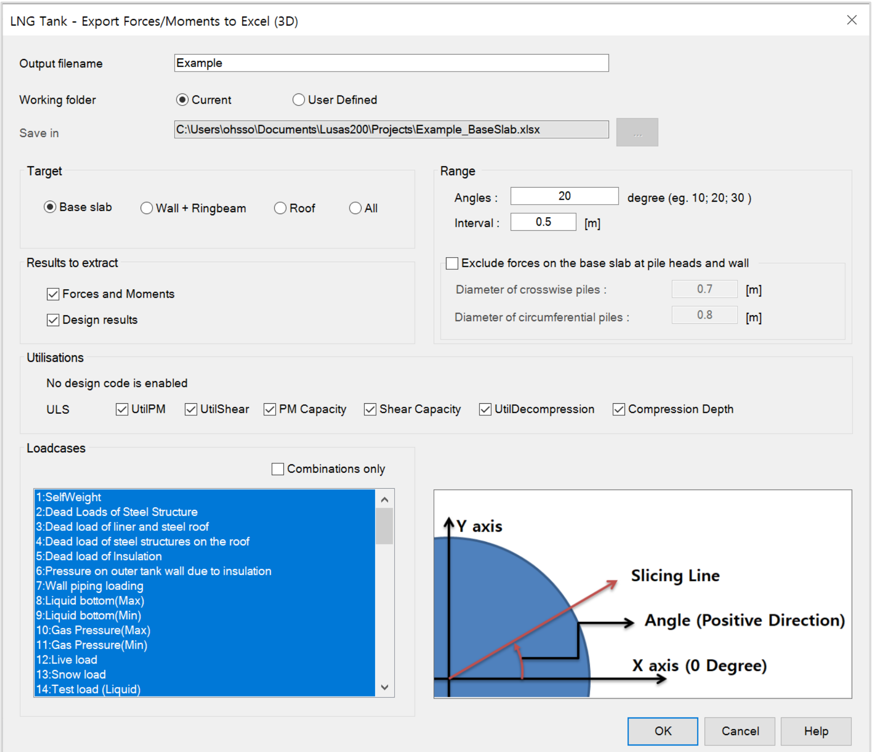


Fig 60 Exporting Forces for a 3D Shell Model

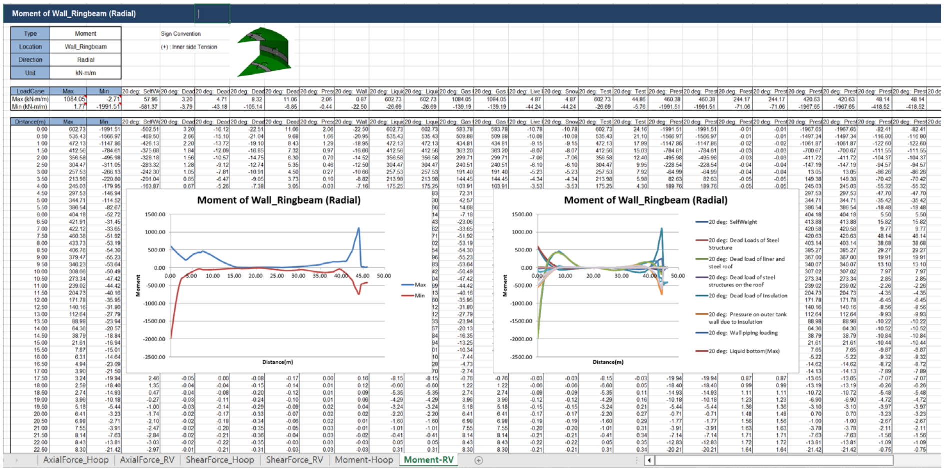


Fig 61 Section Forces Exported from a 3D Shell Model

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.

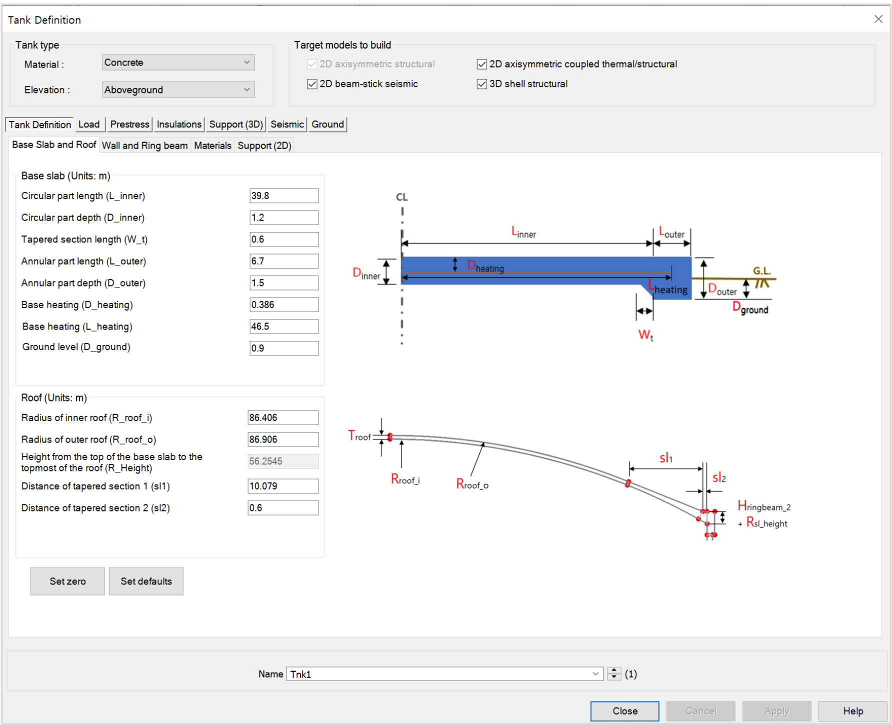


Fig 62 Tank Definition Dialog

- **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

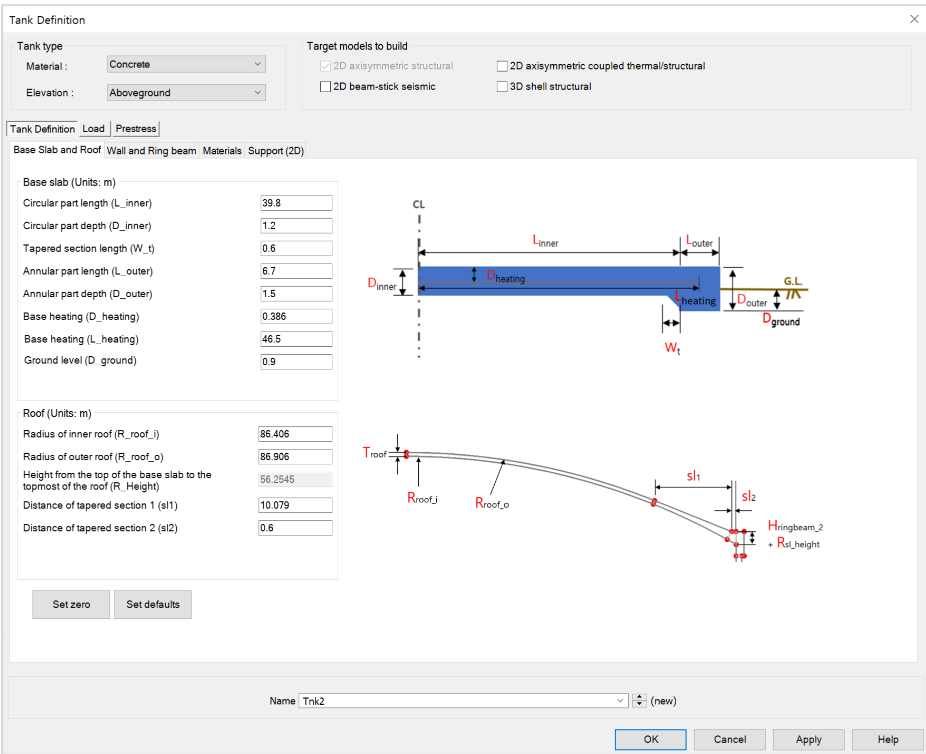
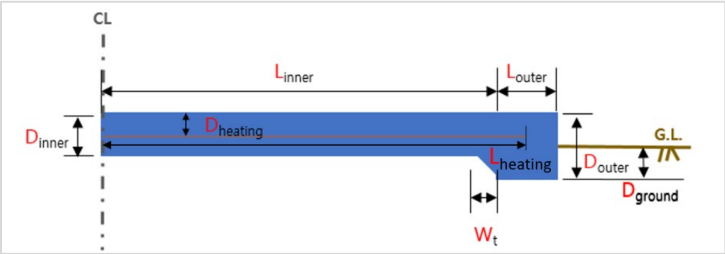


Fig 63 Tank Definition Dialog (Structure Definition/ Concrete Tank/ Base Slab and Roof)

Base Slab

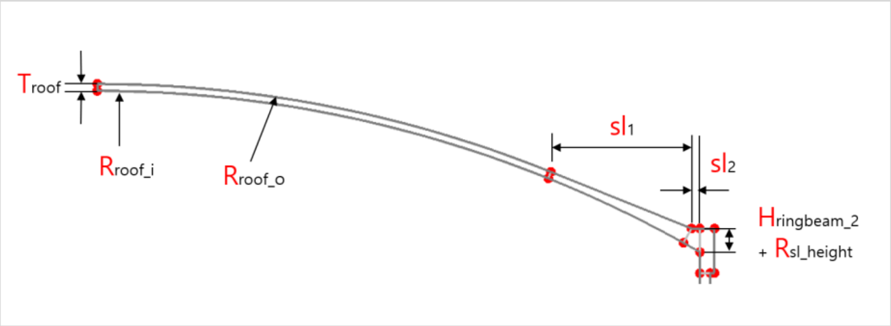
Dimensions for the Base Slab should be entered. The input value must be a positive numerical value.



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

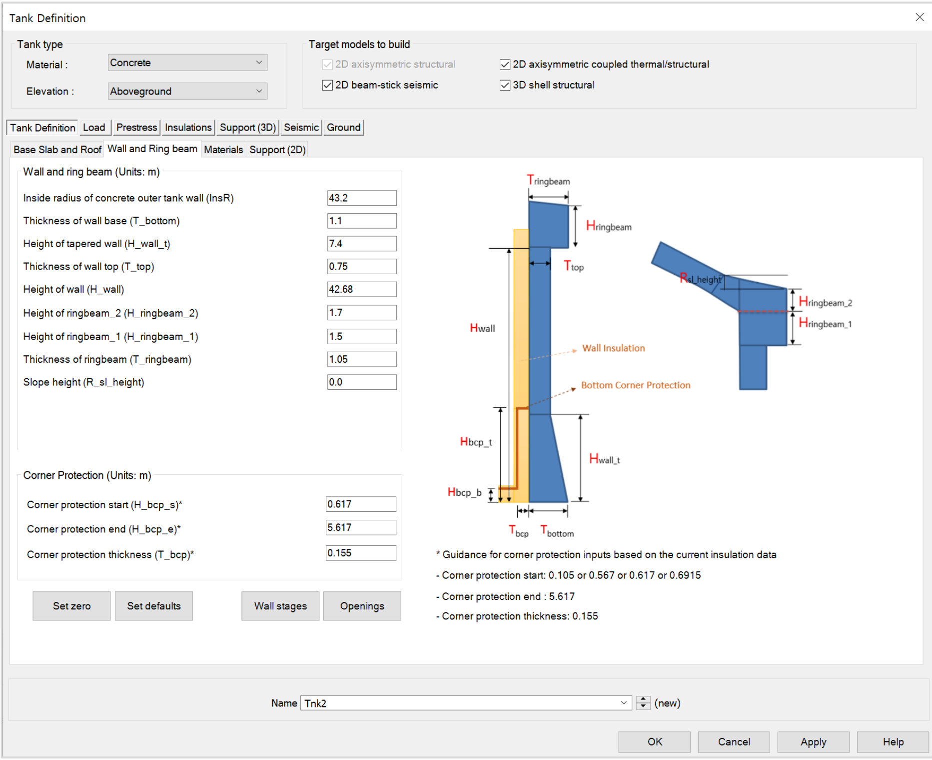
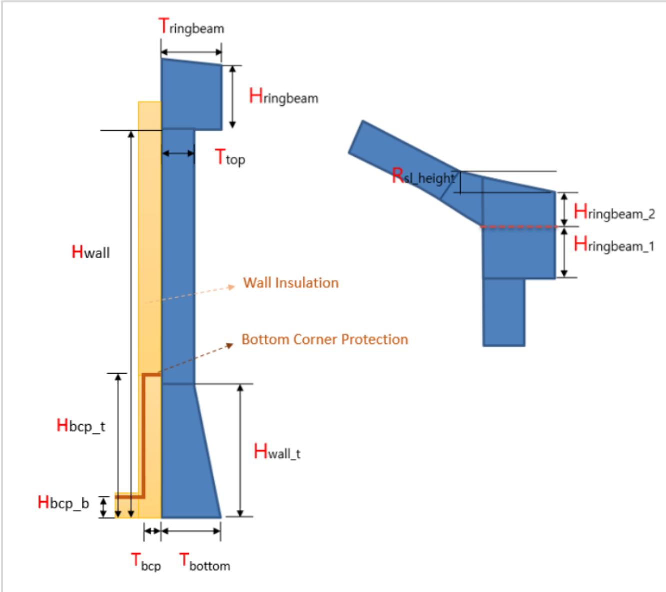


Fig 64 Tank Definition Dialog (Structure Definition/ Concrete Tank/ Wall and Ring Beam)

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.

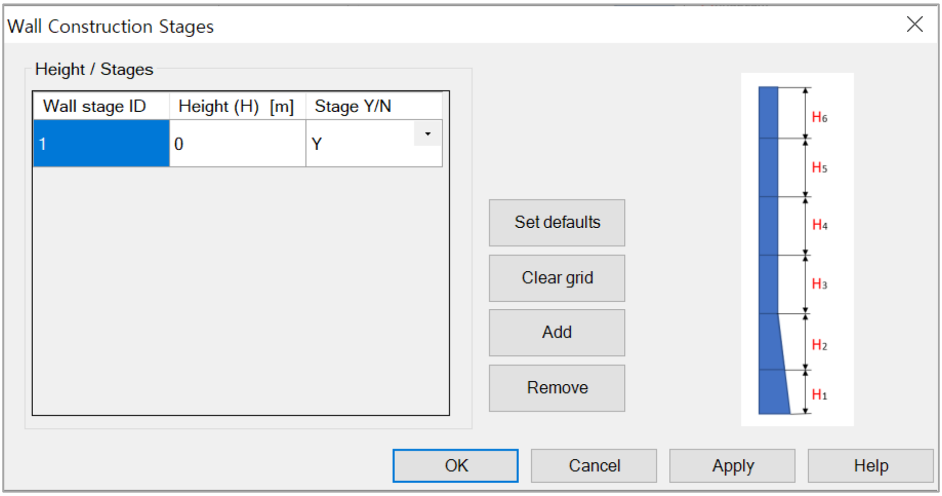
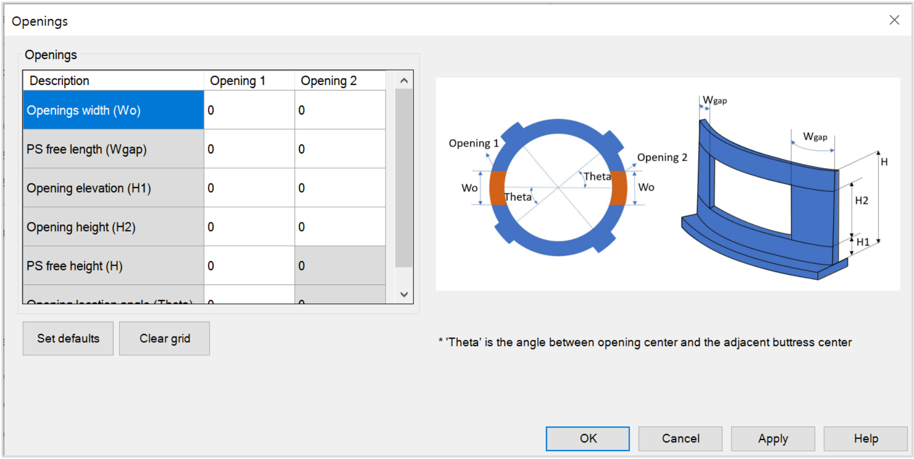


Fig 65 Wall Construction Stages Dialog (Tank Definition/ Structure Definition/ Concrete Tank/ Wall and Ring Beam)

- **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



- ❑ **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

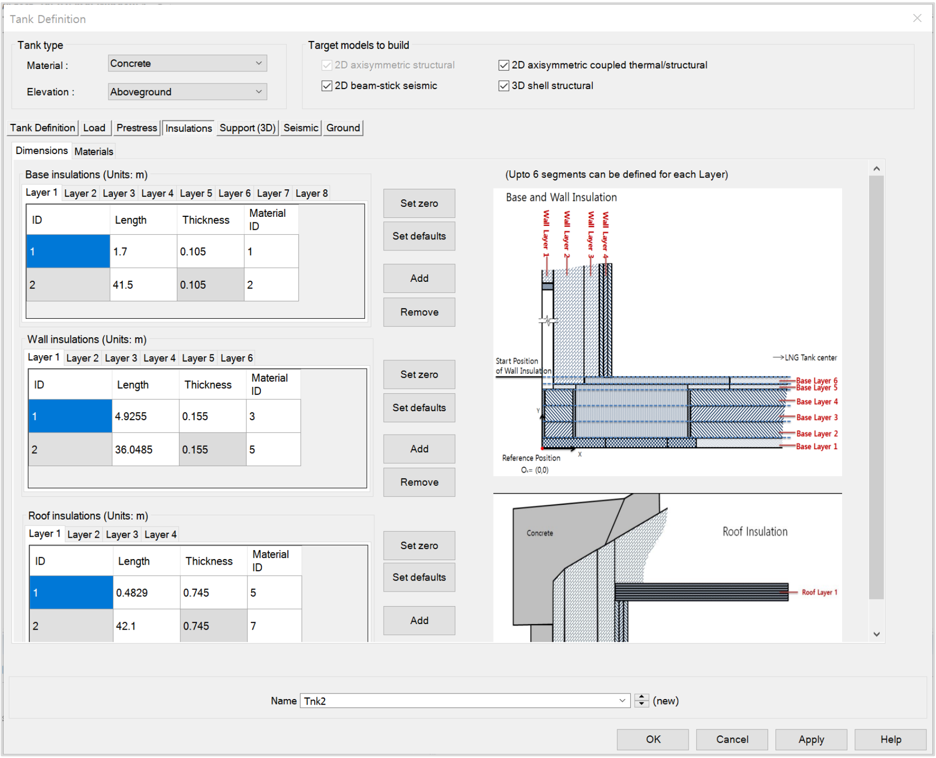


Fig 67 Tank Definition Dialog (Structure Definition/ Concrete Tank/ Insulation)

- ❑ **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.

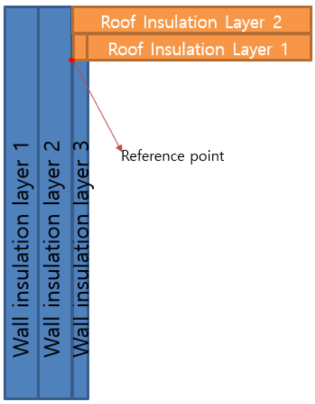
Wall Insulation

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

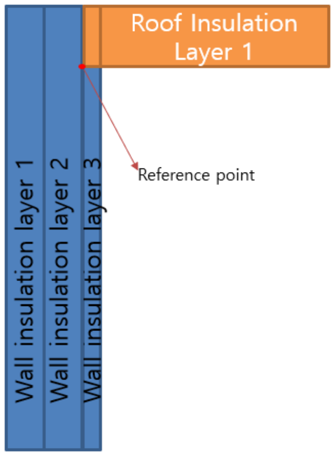
Roof insulation

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.

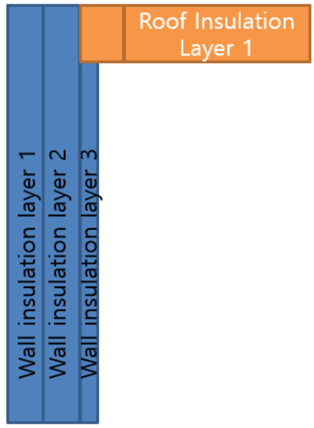
- The top-left point of the innermost layer of wall insulation is used as the ‘reference point’ for modelling the roof insulation.
- The sum of the total length of the roof insulation for a layer and the total thickness excluding the last layer for the wall insulation should be equal to inner diameter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



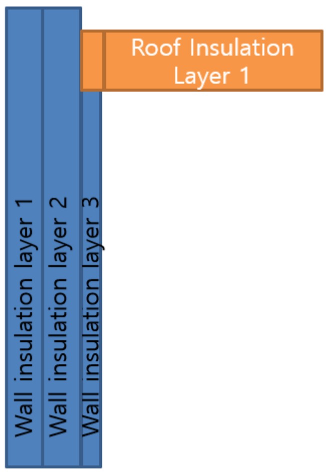
Several examples of defining wall and roof insulation follow:



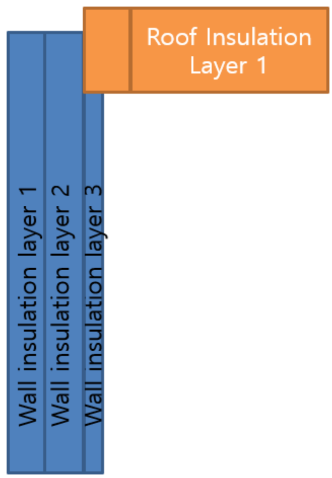
Case 1



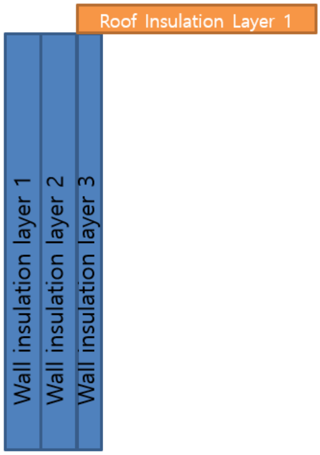
Case 2



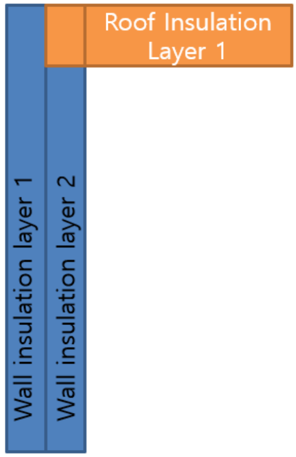
Case 3



Case 4



Case 5



Case 6

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

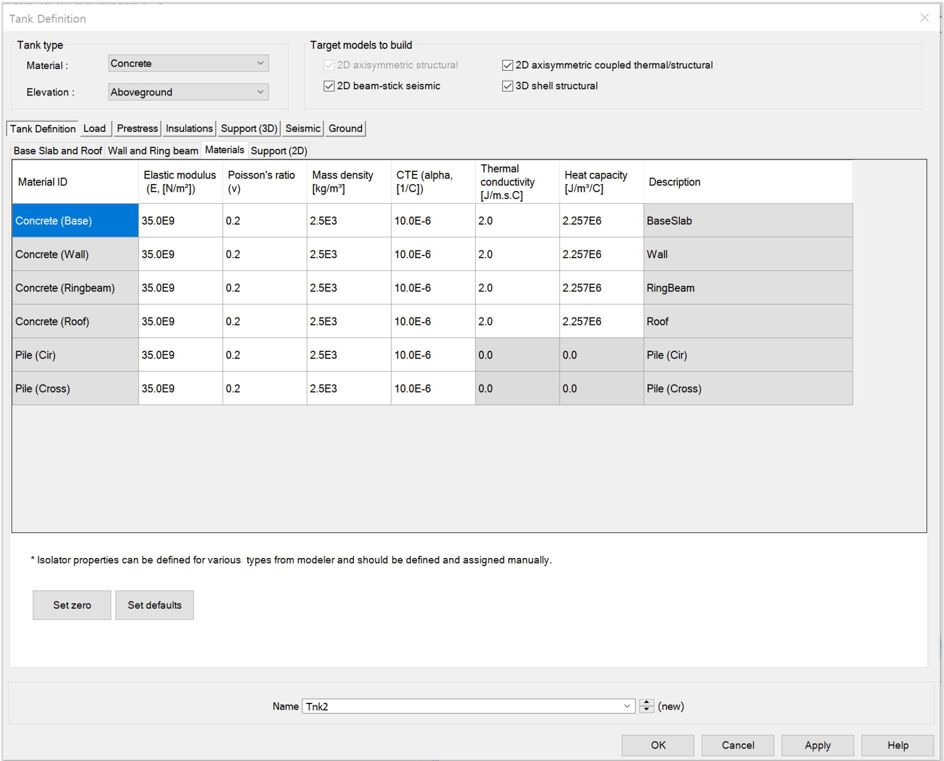
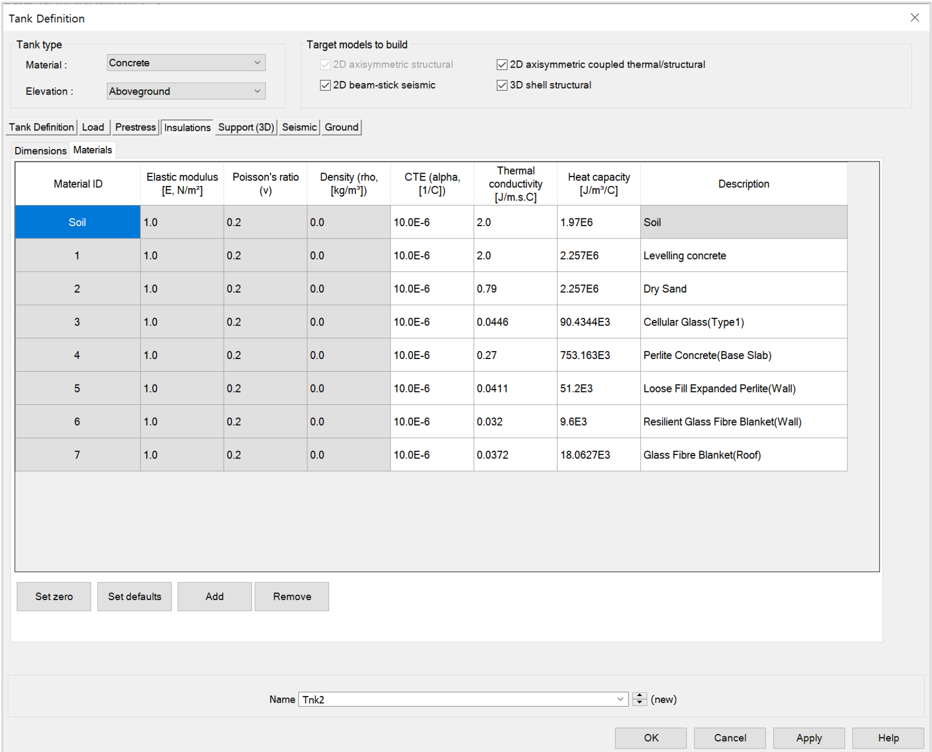


Fig 68 Tank Definition Dialog (Material Properties – Tank Materials)

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.



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)

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)

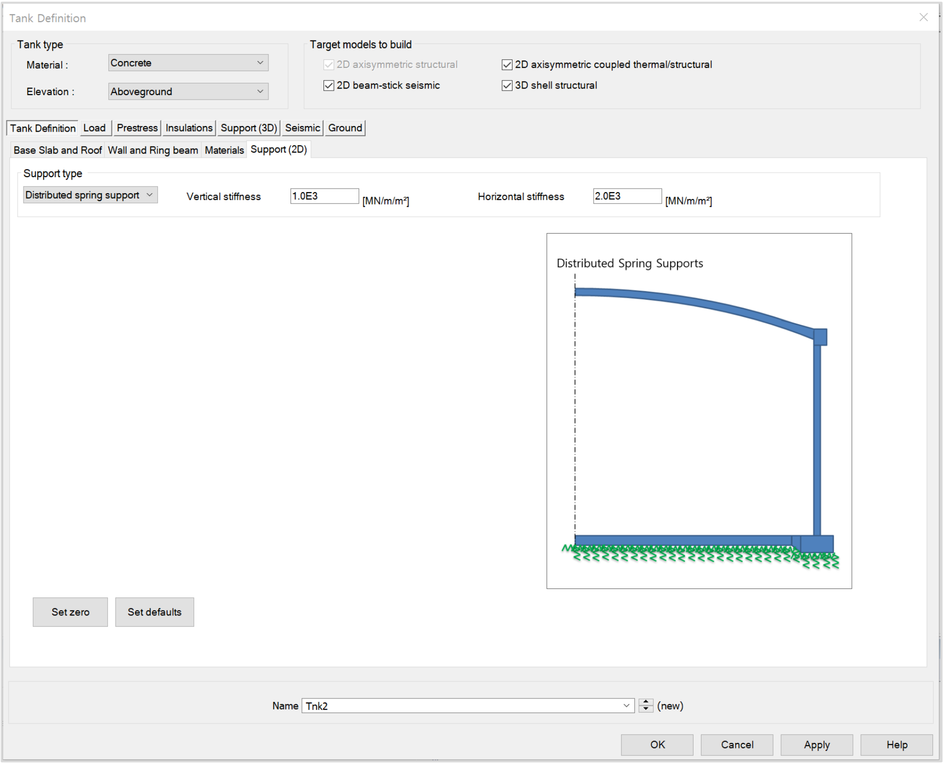


Fig 70 Tank Definition Dialog (Support (2D) – Distributed spring support)

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/m^2.

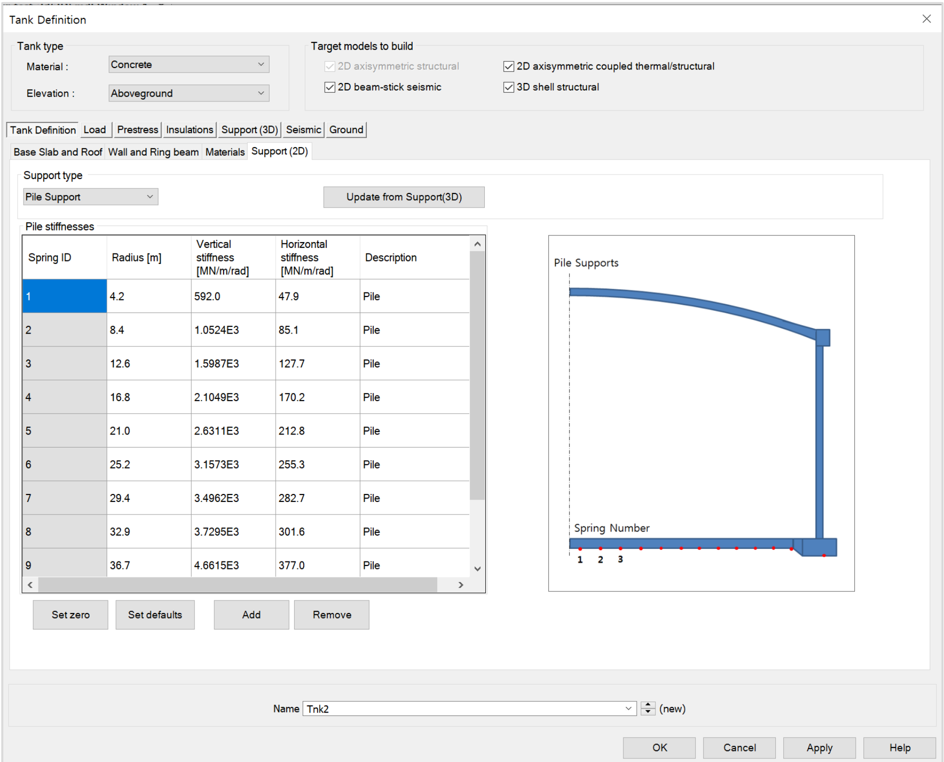


Fig 71 Tank Definition Dialog (Boundary Condition- Pile Support)

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

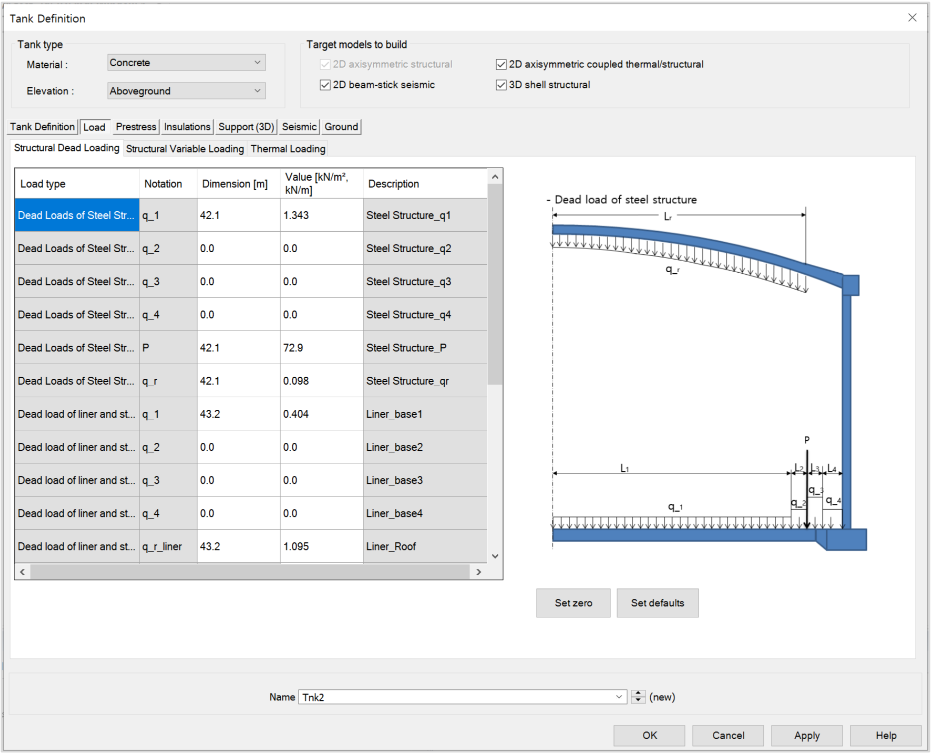


Fig 72 Tank Definition Dialog (Load – Structural Dead Loading)

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.

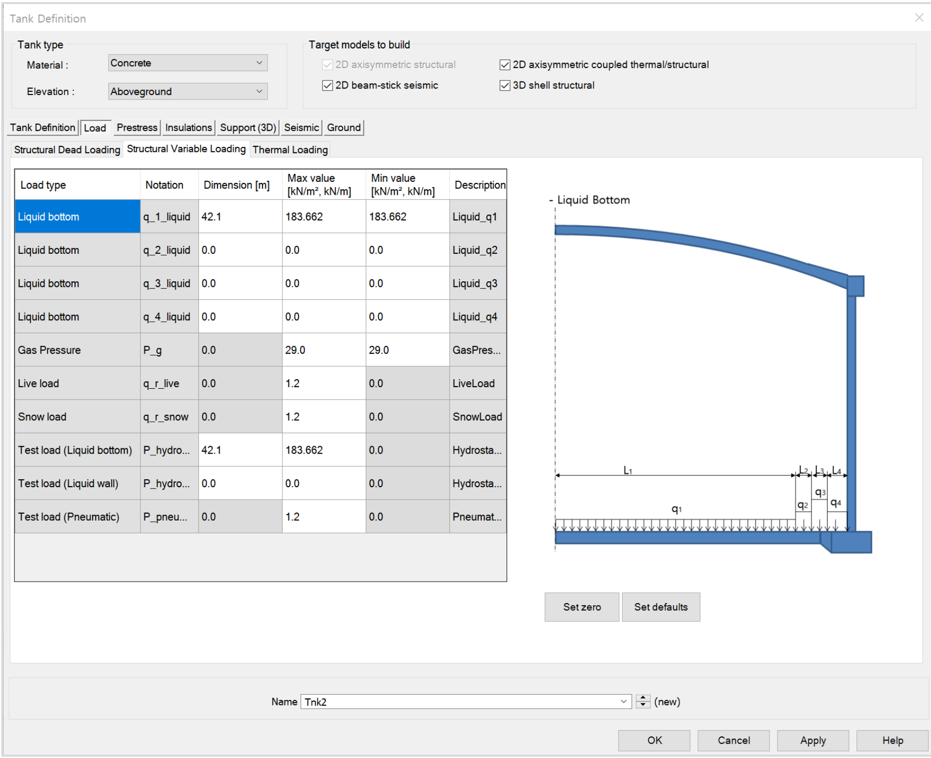


Fig 73 Tank Definition Dialog (Load – Structural Variable Loading)

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.

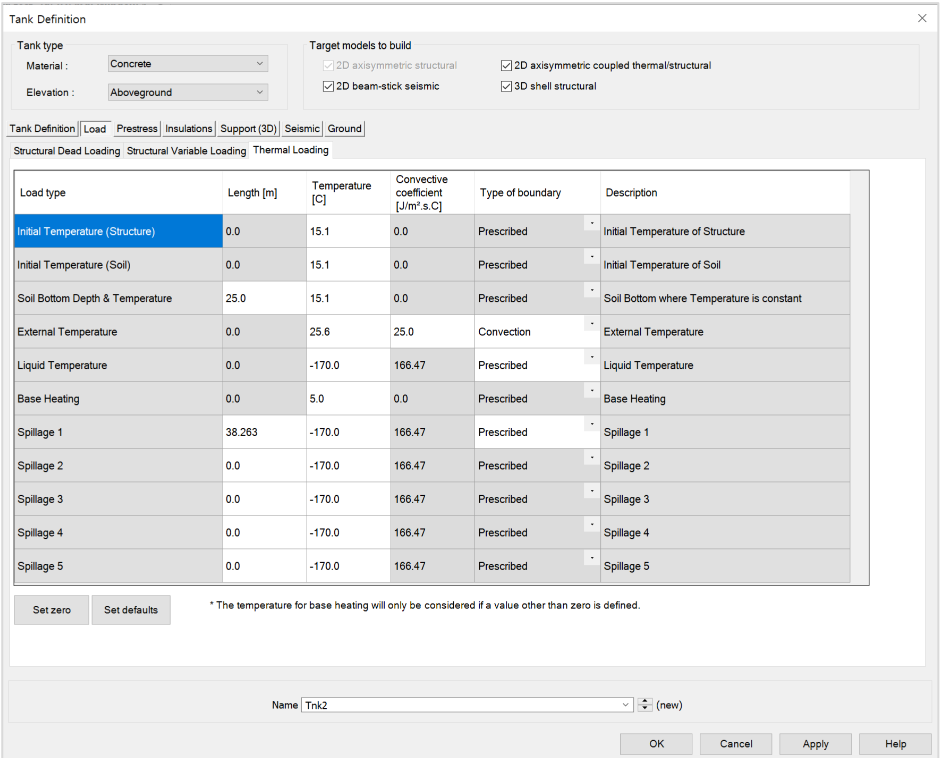


Fig 74 Tank Definition Dialog (Load – Thermal Loading)

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

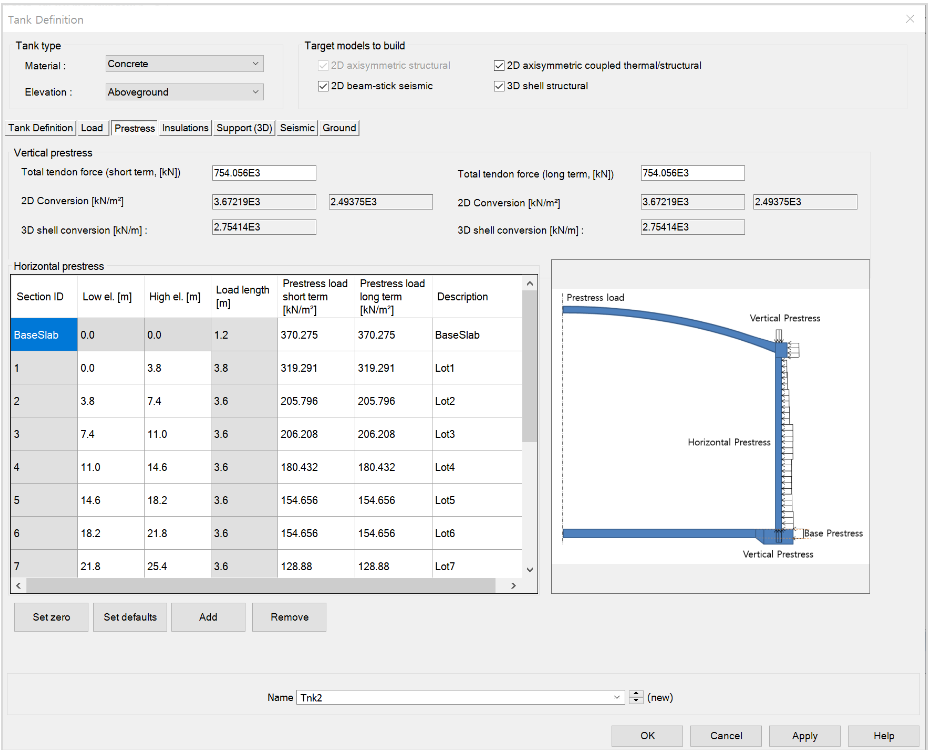


Fig 75 Tank Definition Dialog (Prestress)

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)

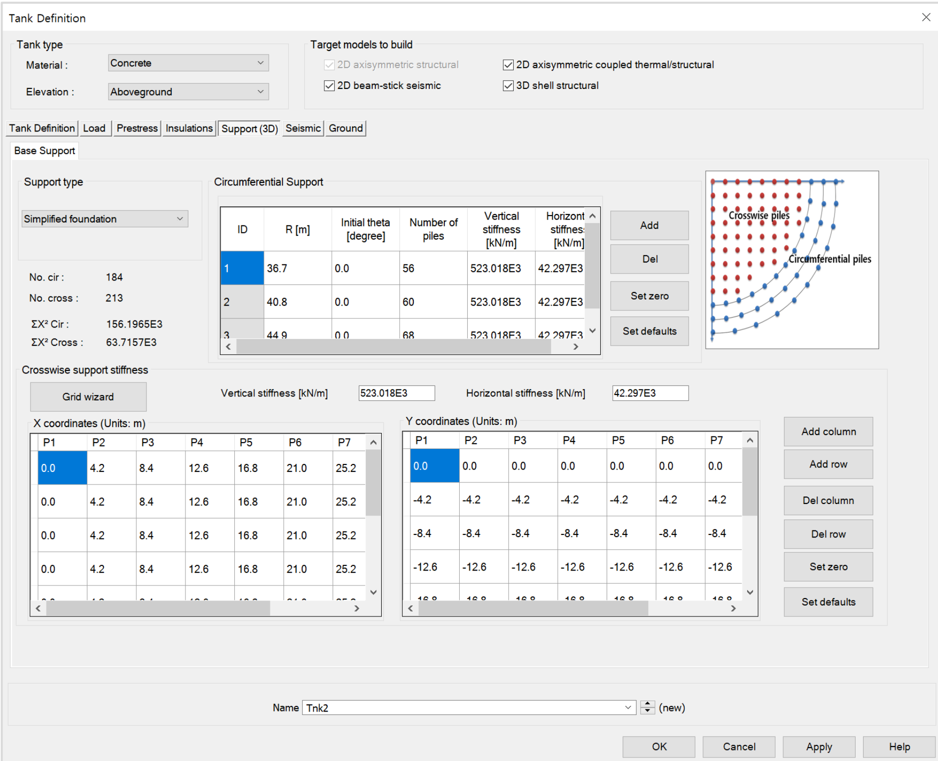


Fig 76 Tank Definition Dialog (Support (3D))

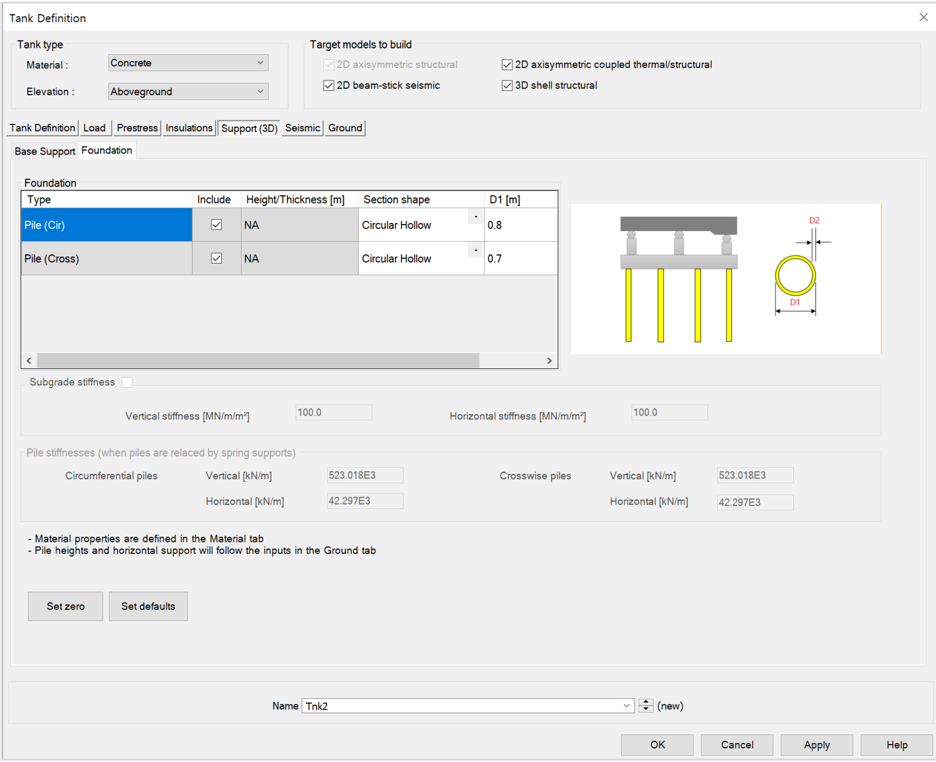


Fig 77 Tank Definition Dialog (Support (3D))

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.

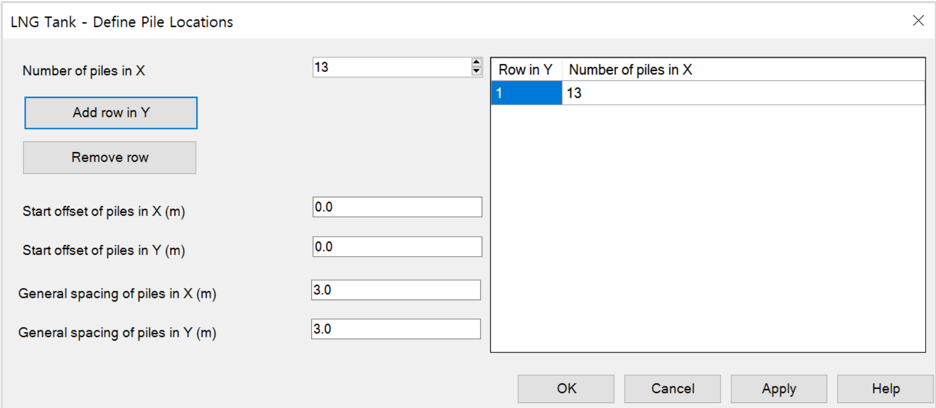


Fig 78 Tank Definition Dialog (Support(3D) – Define Pile Locations)

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

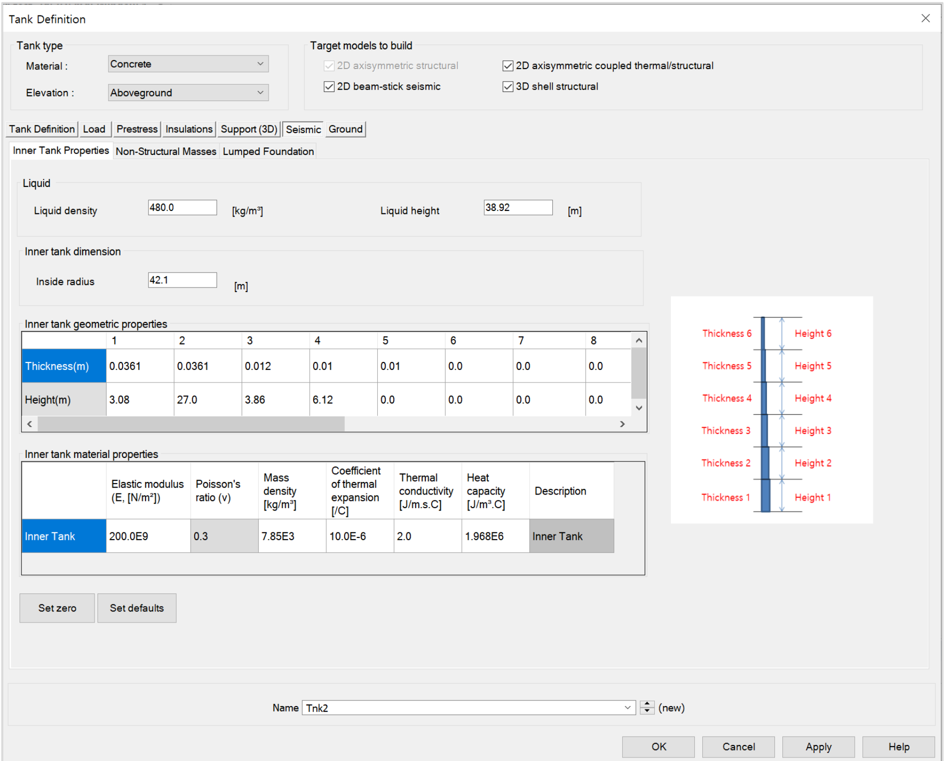


Fig 79 Tank Definition Dialog (Seismic– Inner Tank Properties)

- ❑ **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.

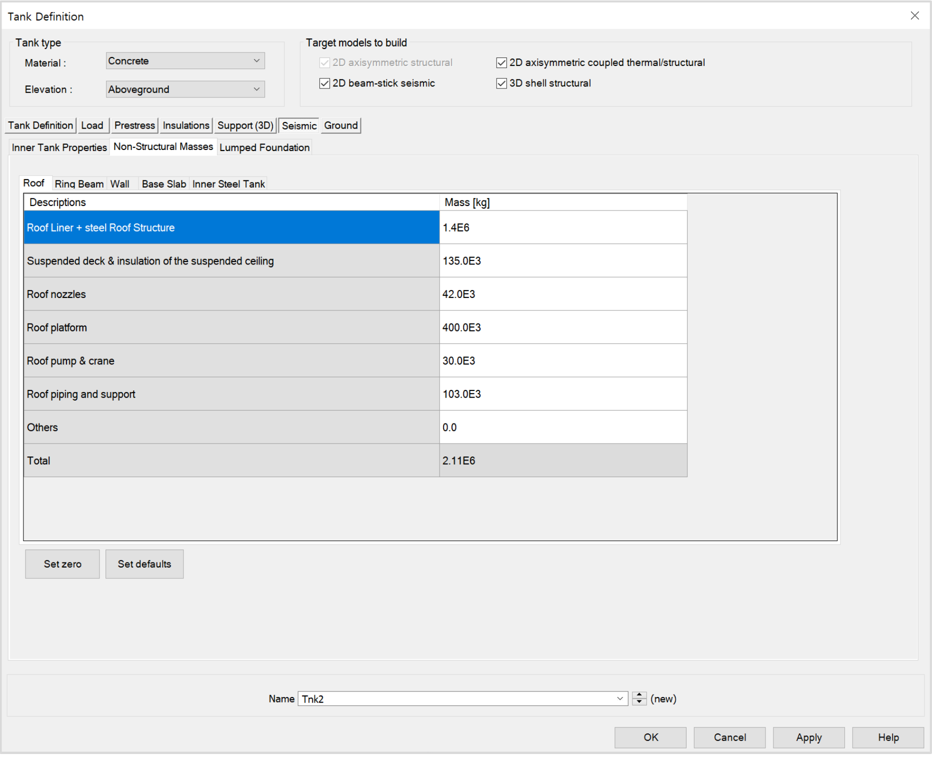


Fig 80 Tank Definition Dialog (Seismic– Non-Structural Masses)

Non-Structural Masses

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

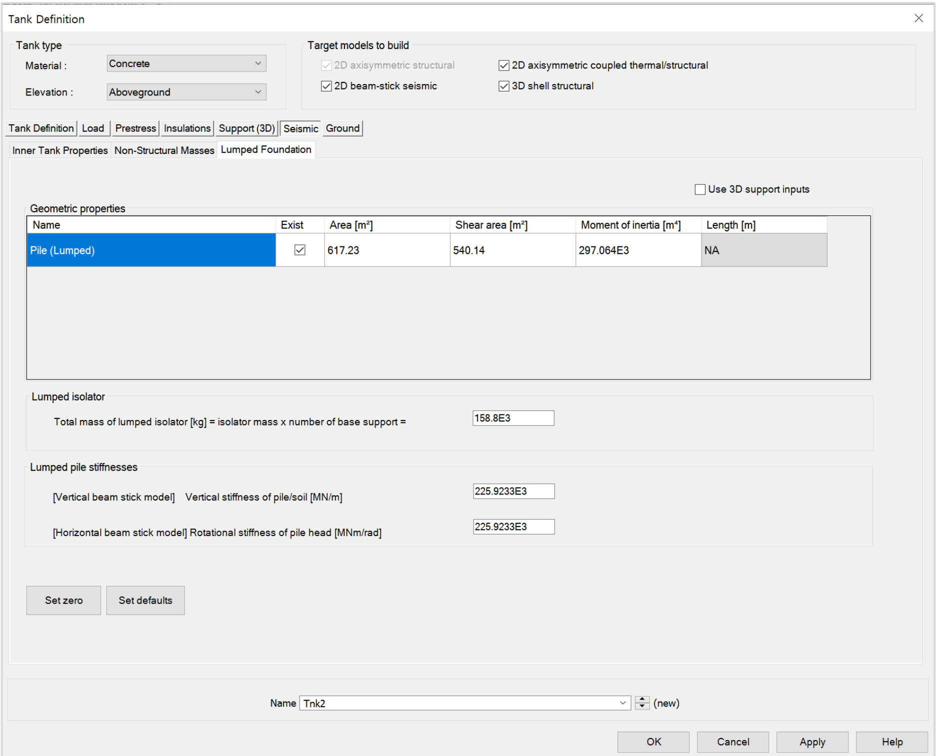


Fig 81 Tank Definition Dialog (Seismic– Lumped Foundation)

- ❑ **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

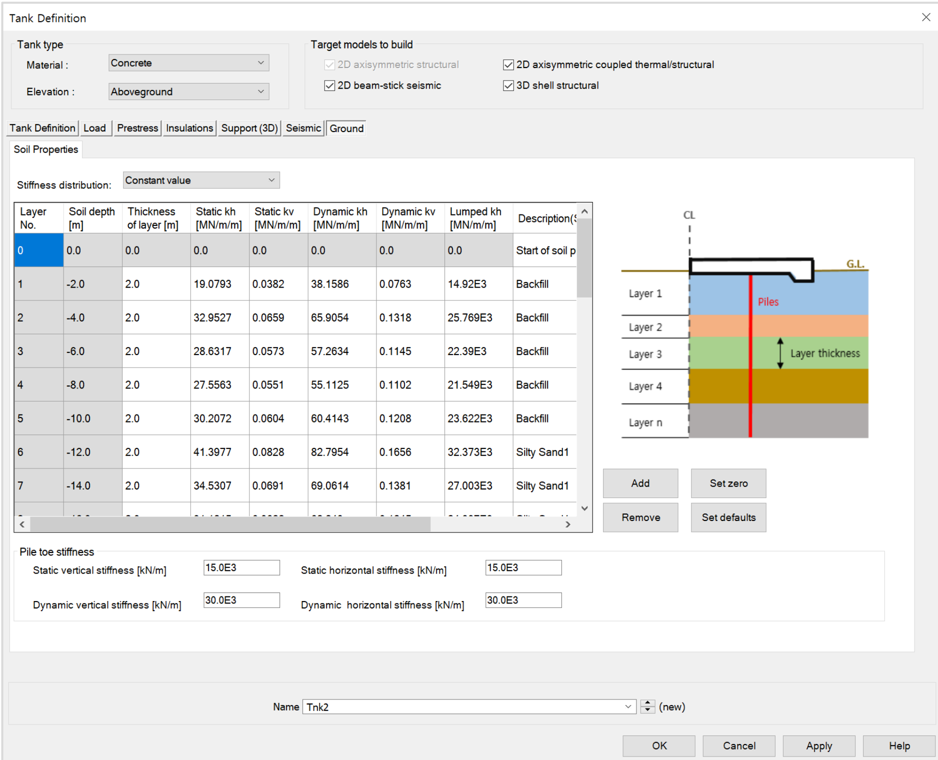


Fig 82 Tank Definition Dialog (Ground– Soil Properties)

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].

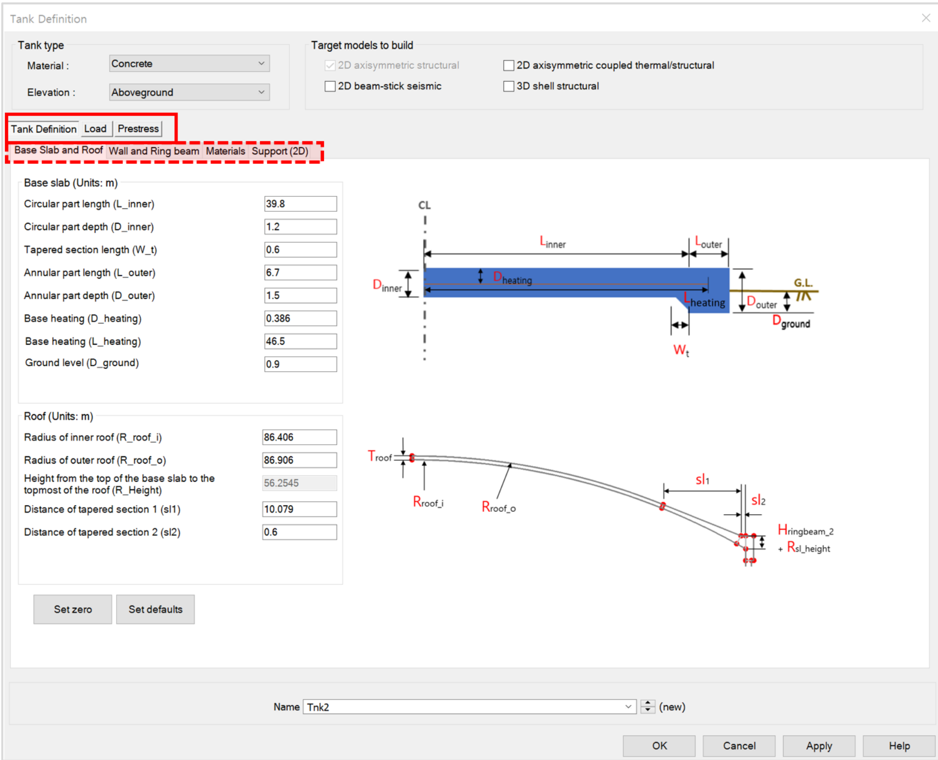


Fig 83 User Inputs for 2D Axisymmetric Static Analysis

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.

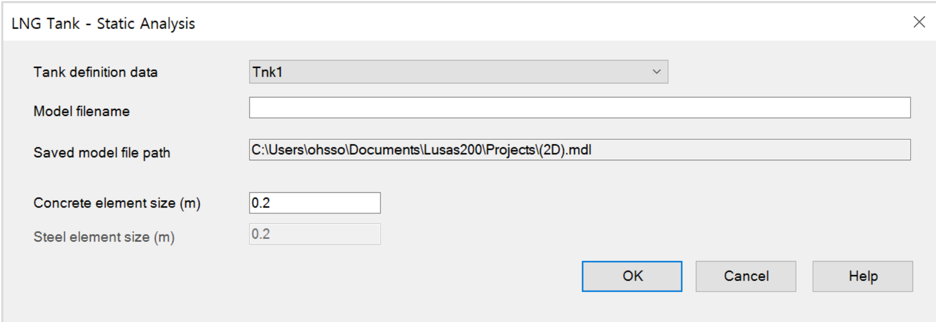


Fig 84 User Dialog for 2D Axisymmetric Static Analysis

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.

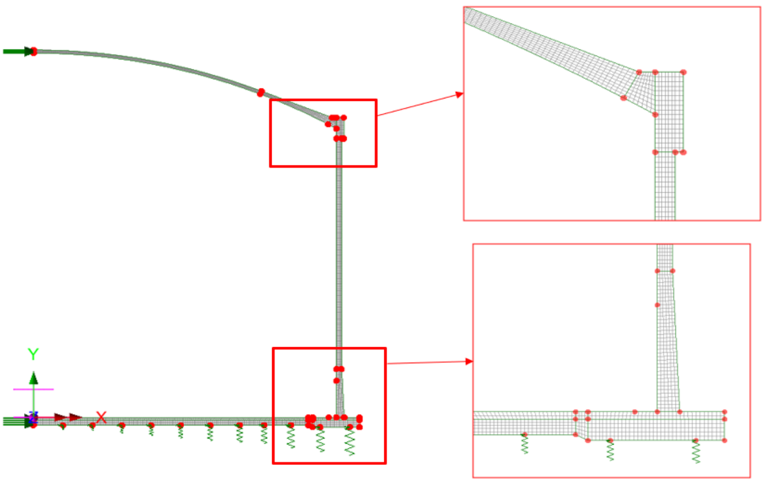


Fig 85 Mesh division for a 2D Axisymmetric Model

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.

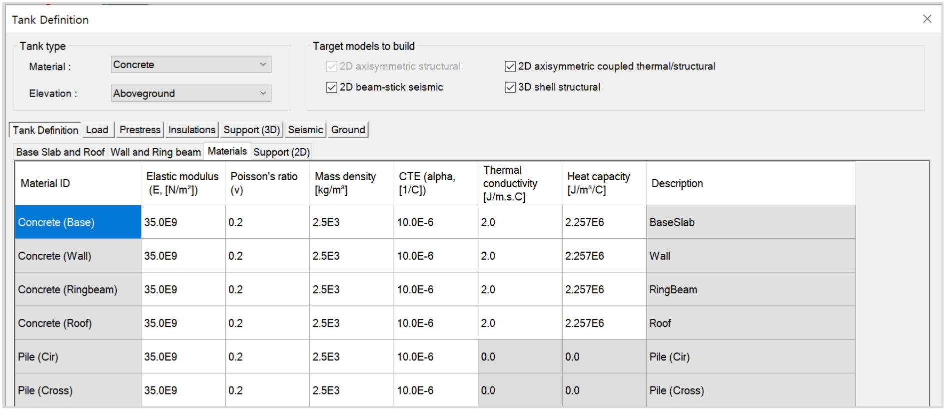


Fig 86 User Inputs for Tank Materials

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

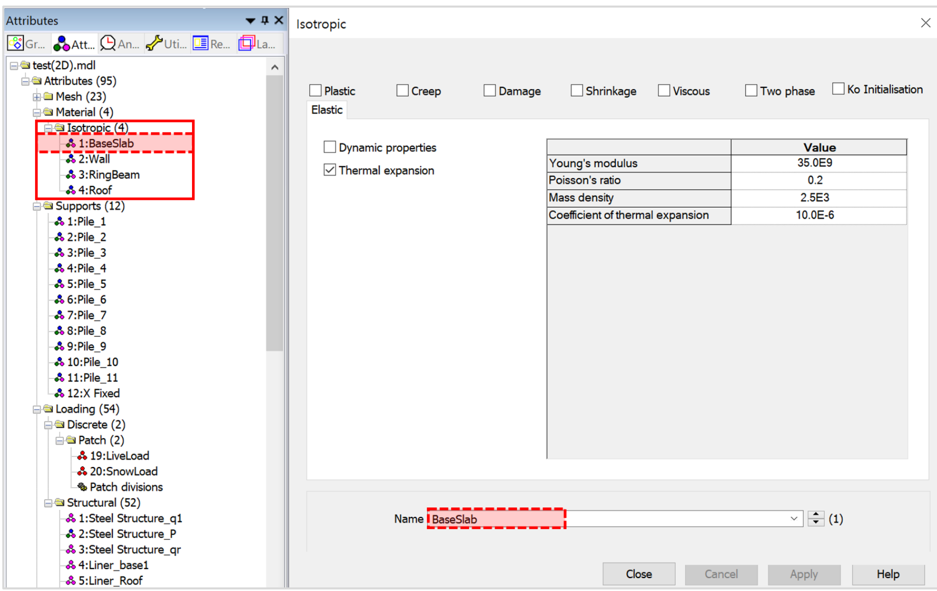


Fig 87 Material Properties for a 2D Axisymmetric Model

Support Conditions

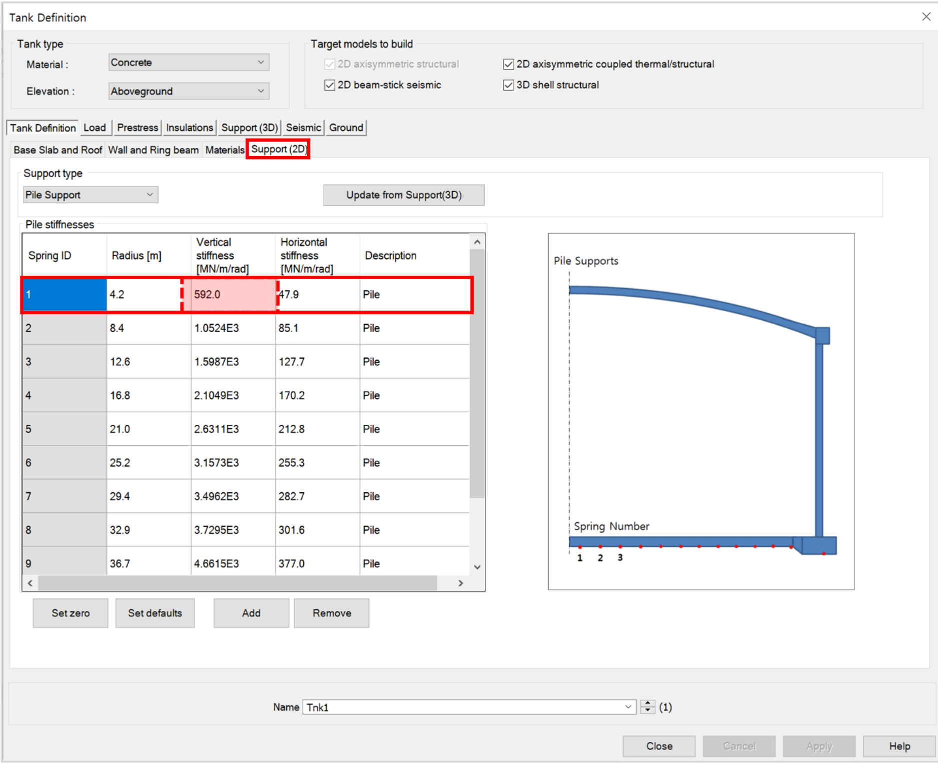


Fig 88 User Inputs for Boundary Conditions

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

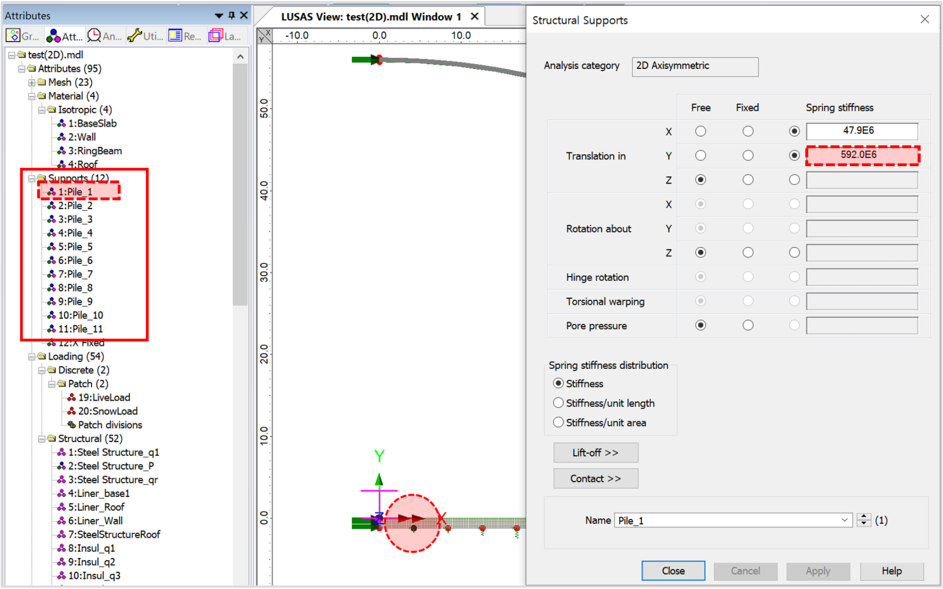


Fig 89 Pile Support for a 2D Axisymmetric Model

TEST CASE

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

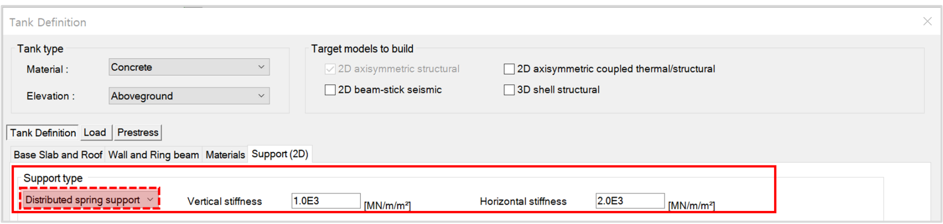


Fig 90 Test Case - Regular Support for a 2D Axisymmetric Model

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

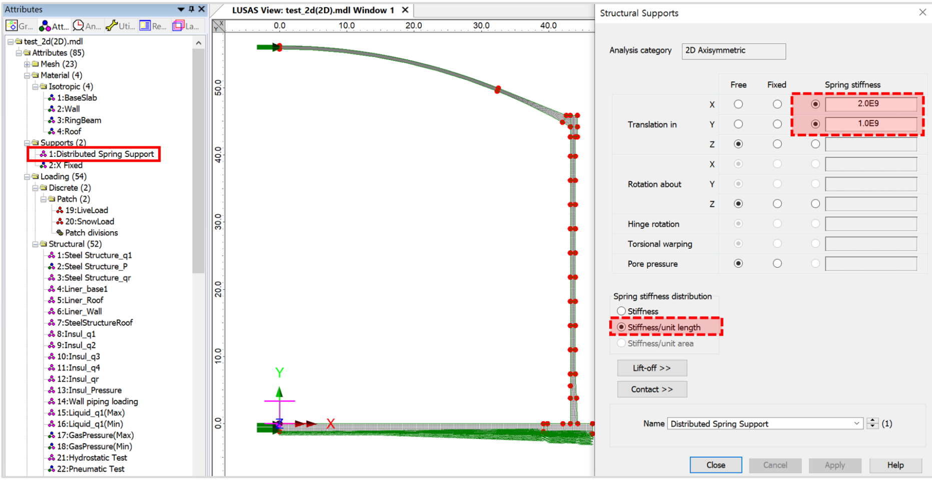


Fig 91 Test Case - Definition of a Regular Support for a 2D Axisymmetric Model

Loadings

A total of 17 loadcases is defined in the model.

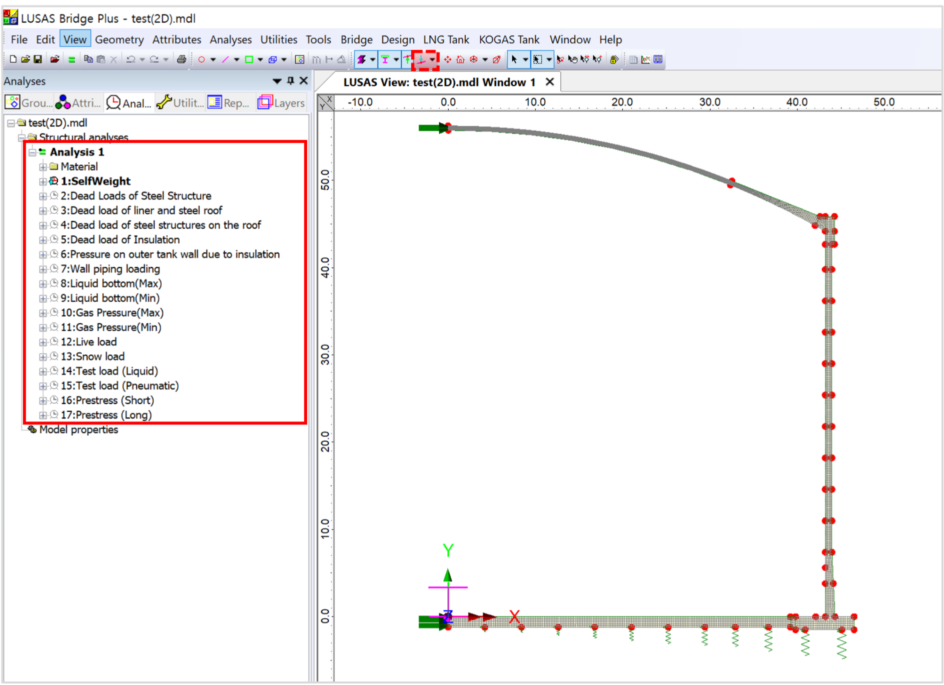


Fig 92 Loadcases available in a 2D Axisymmetric Static Analysis Model

Self Weight

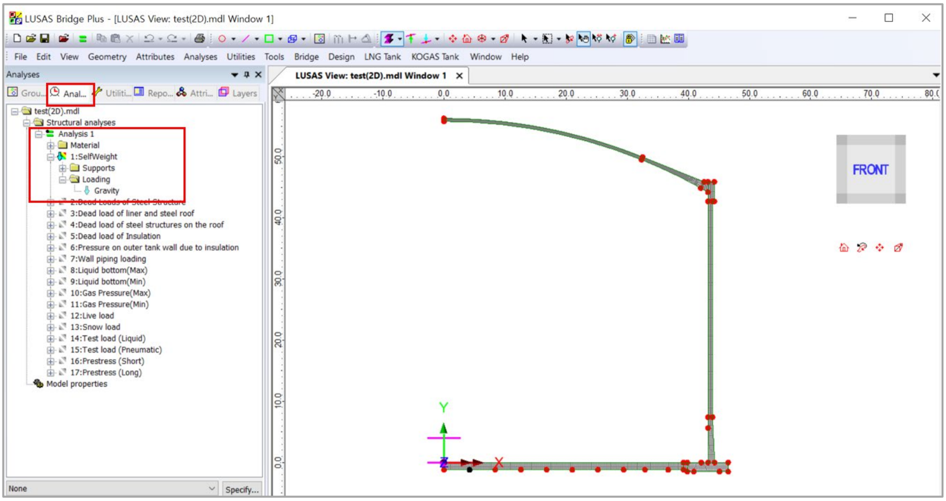


Fig 93 Self Weight in a 2D Axisymmetric Static Analysis Model

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.

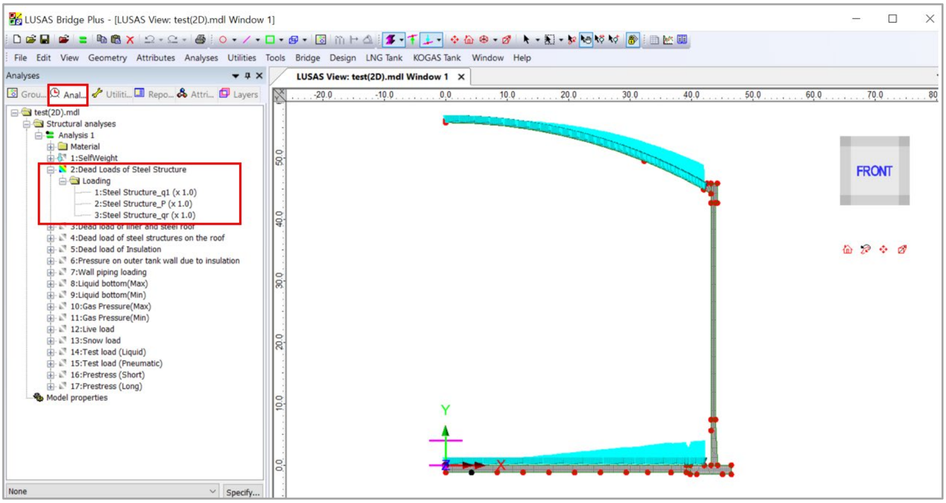


Fig 94 Dead Load for Steel Structure in a 2D Axisymmetric Static Analysis Model

Dead load of liner and steel roof

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

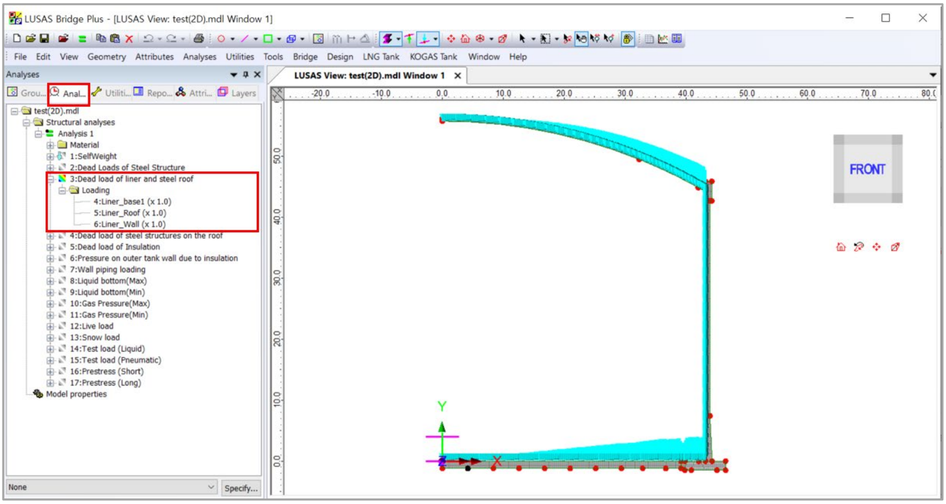


Fig 95 Dead Load of Liner and Steel Roof in a 2D Axisymmetric Static Analysis Model

Dead load of steel structures on the roof

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

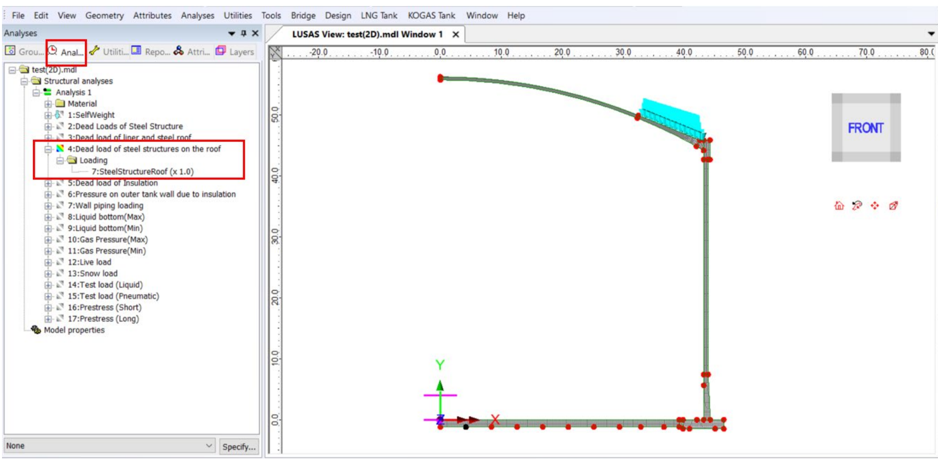


Fig 96 Dead Load of Steel Structures on the Roof in a 2D Axisymmetric Static Analysis Model

Dead load of Insulation

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

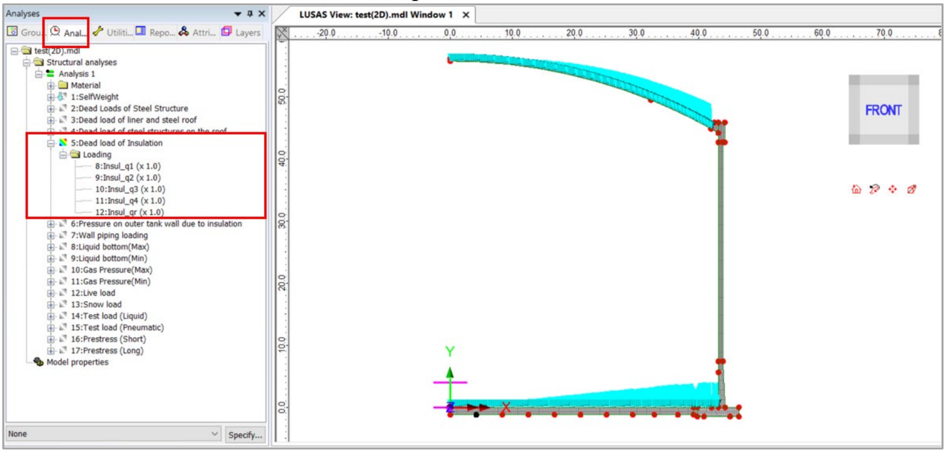


Fig 97 Dead Load of Insulation in a 2D Axisymmetric Static Analysis Model

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.

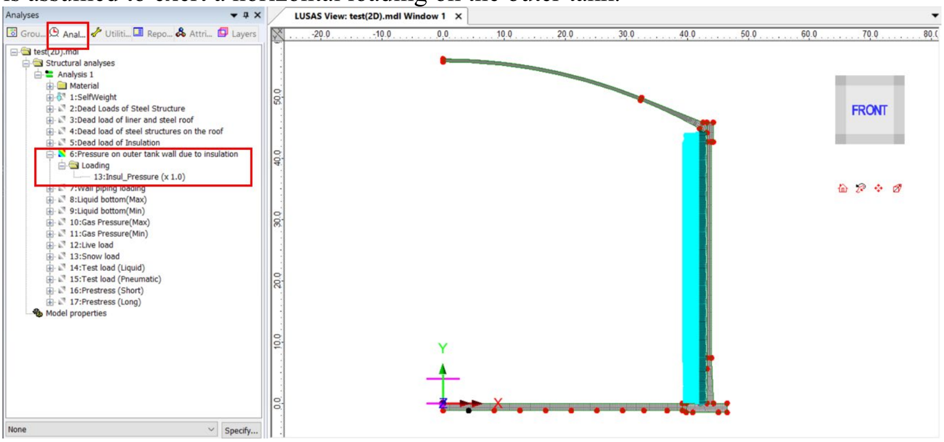


Fig 98 Insulation Pressure Load in a 2D Axisymmetric Static Analysis Model

Wall piping loading

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

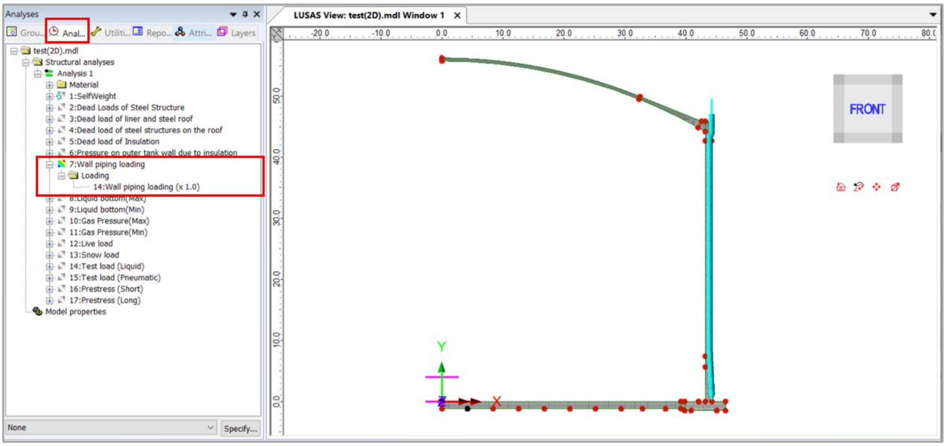


Fig 99 Wall Piping Loading in a 2D Axisymmetric Static Analysis Model

Liquid bottom (Max, Min)

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

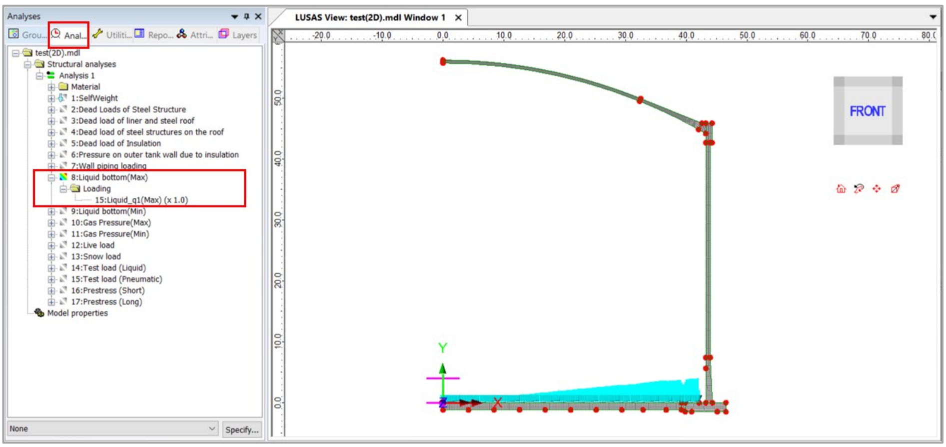


Fig 100 Liquid Bottom Loading in a 2D Axisymmetric Static Analysis Model

Gas pressure (Max, Min)

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

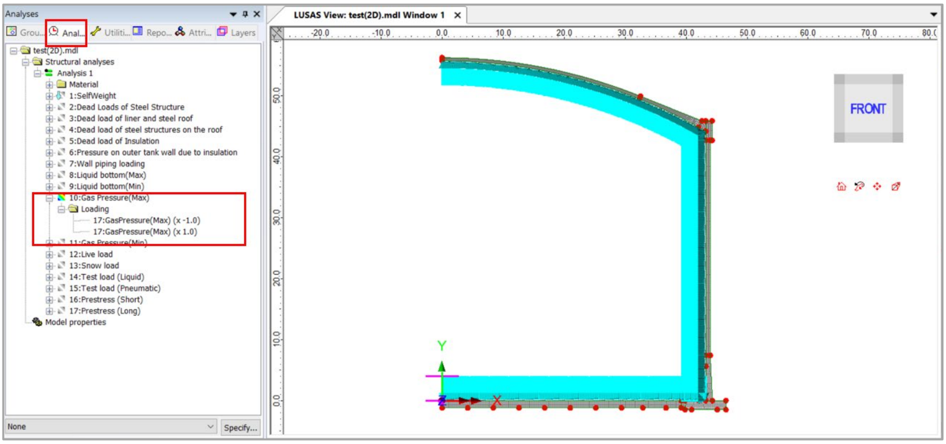


Fig 101 Gas Pressure Loading in a 2D Axisymmetric Static Analysis Model

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.

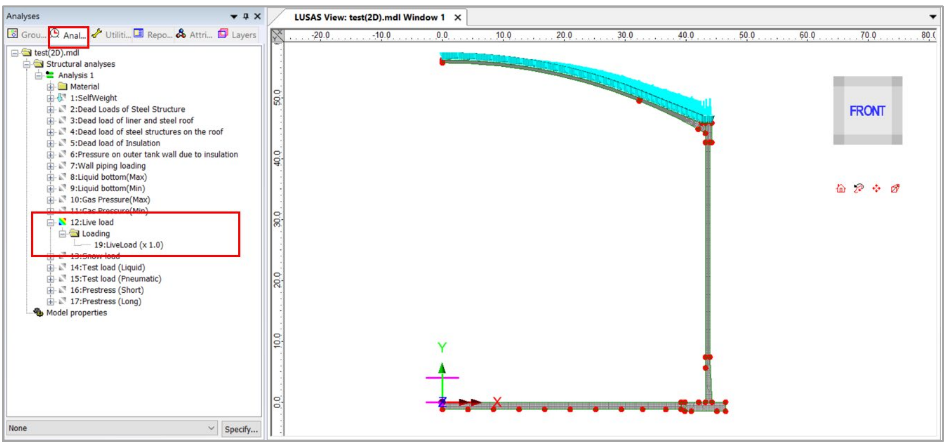


Fig 102 Live Load in a 2D Axisymmetric Static Analysis Model

Snow load

Snow load acts on the top surface of roof.

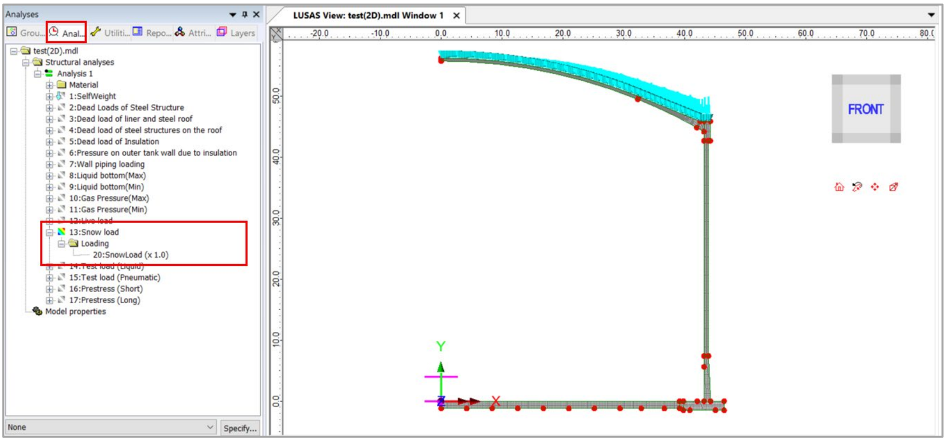


Fig 103 Snow Load in a 2D Axisymmetric Static Analysis Model

Test load (Liquid bottom)

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

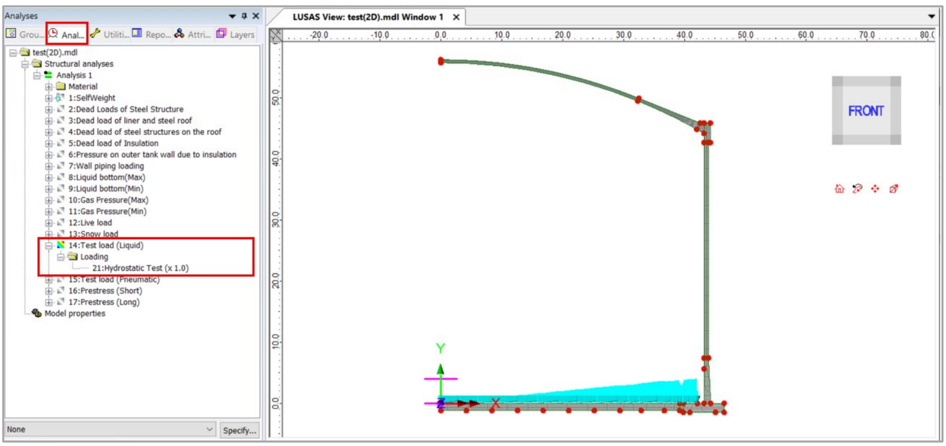


Fig 104 Test Load (Liquid Bottom) in 2D Axisymmetric Static Analysis Model

Test load (Pneumatic)

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

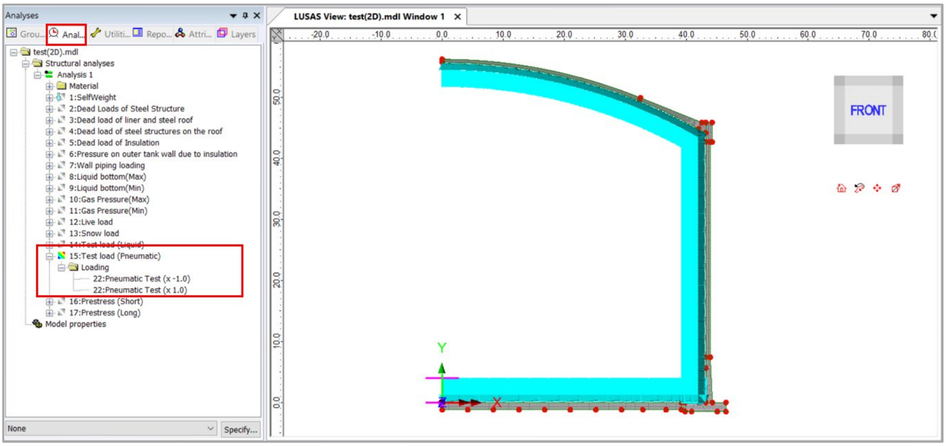


Fig 105 Test Load (Pneumatic) in a 2D Axisymmetric Static Analysis Model

Prestress Load

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

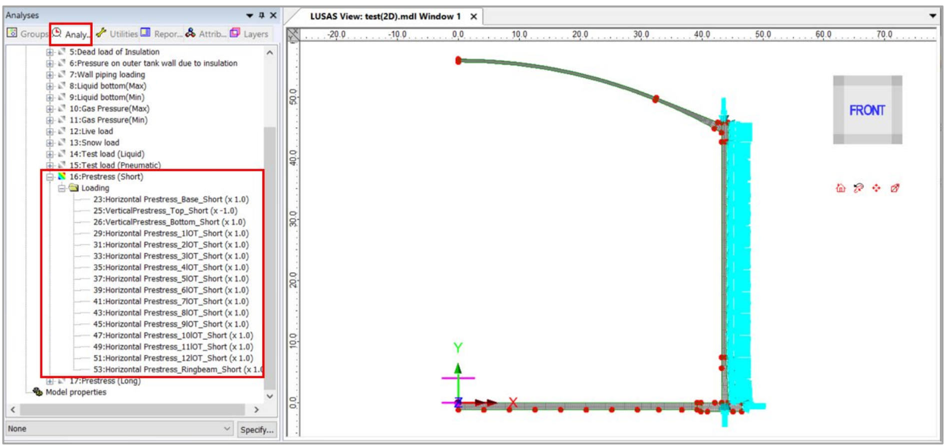


Fig 106 Prestress Load in a 2D Axisymmetric Static Analysis Model

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.

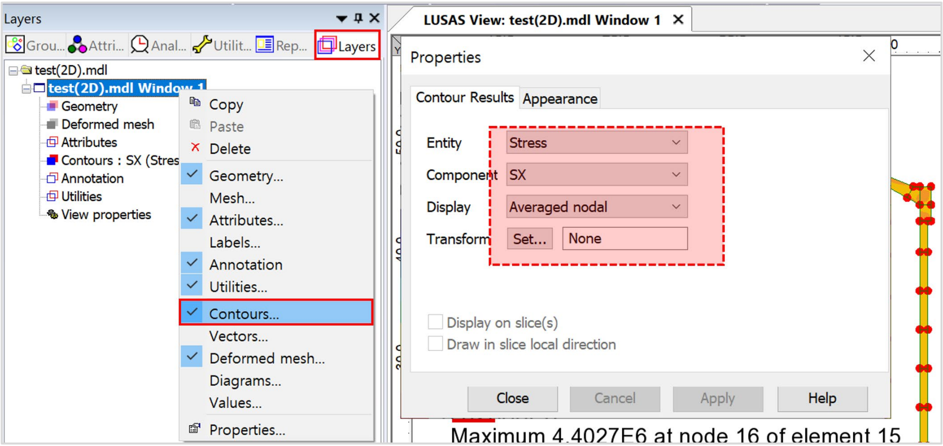


Fig 107 Selection for Contour Display in a 2D Axisymmetric Solid Model

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

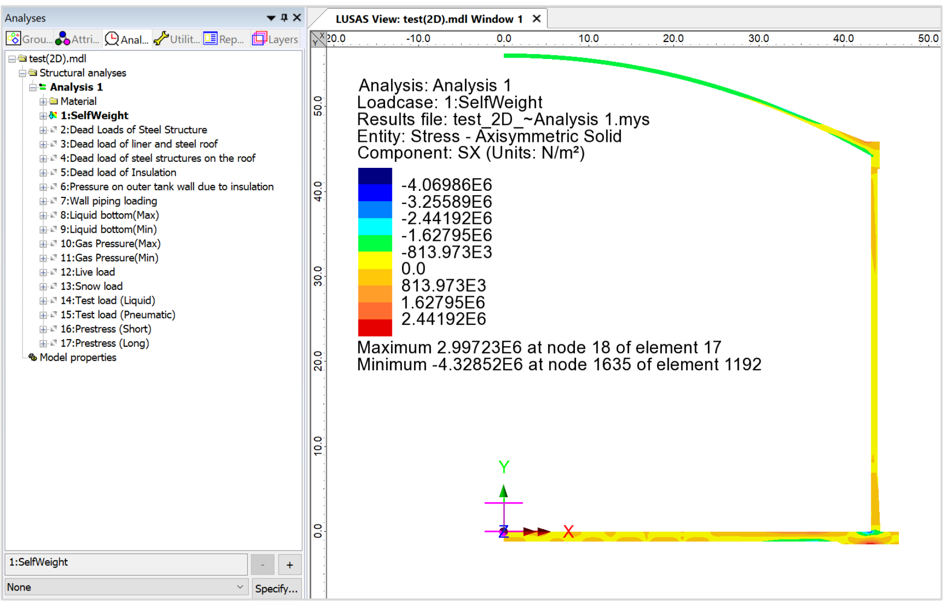


Fig 108 SX Contour for Self Weight in a 2D Axisymmetric Solid Model

Values

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

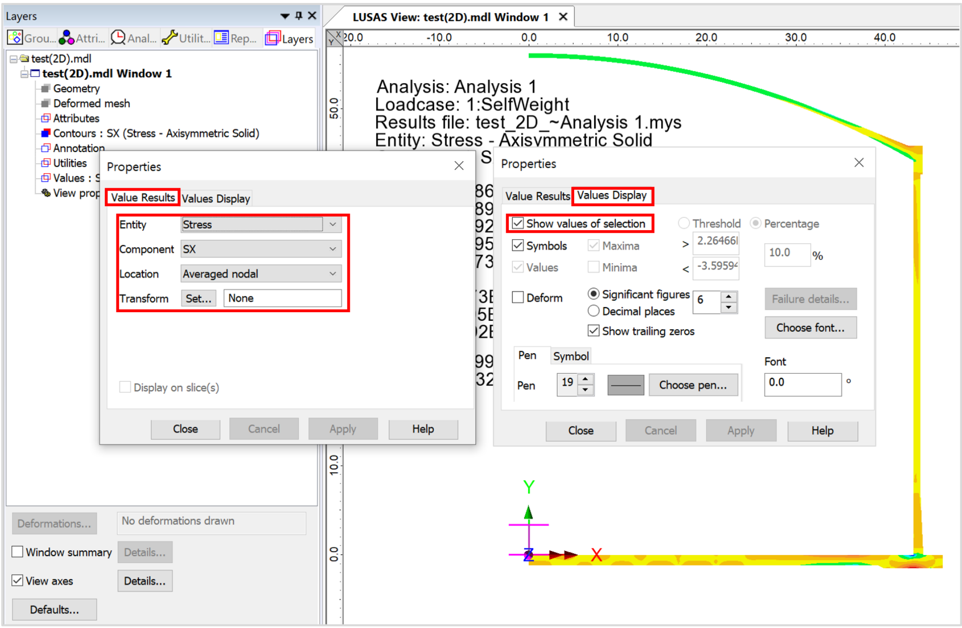


Fig 109 Value Display in a 2D Axisymmetric Solid Model

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

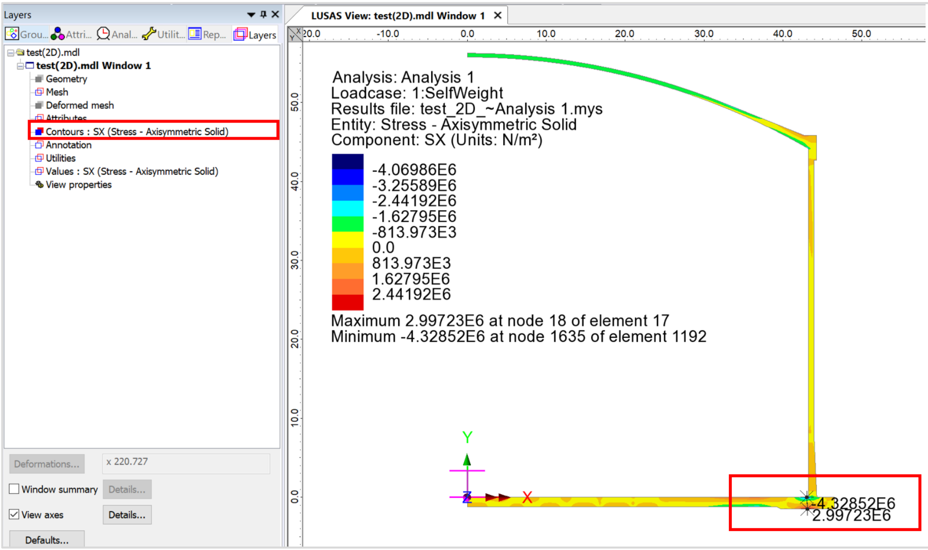


Fig 110 Values Displayed for Selected Nodes in a 3D Shell Model

Graph through 2D

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

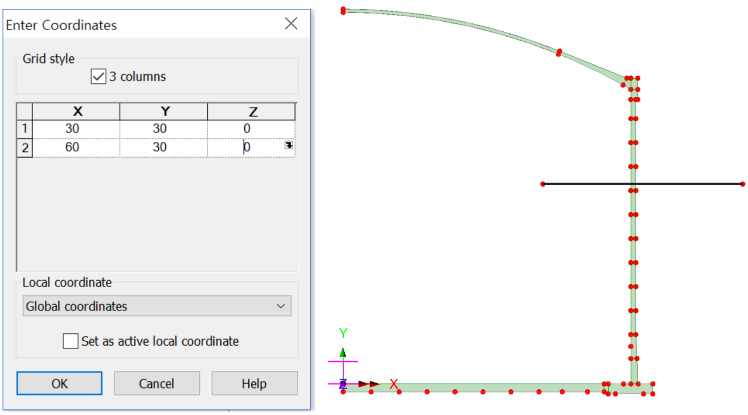


Fig 111 Line for Slicing Results in a 2D Axisymmetric Solid Model

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

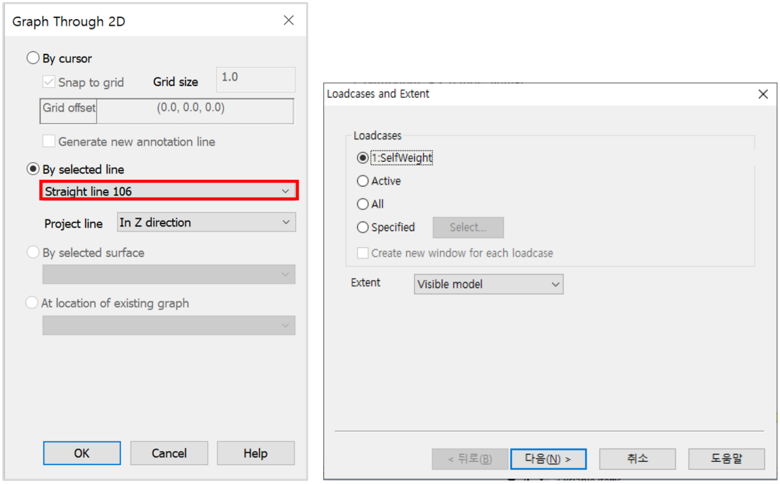


Fig 112 Graph Through 2D in a 2D Axisymmetric Solid Model (1)

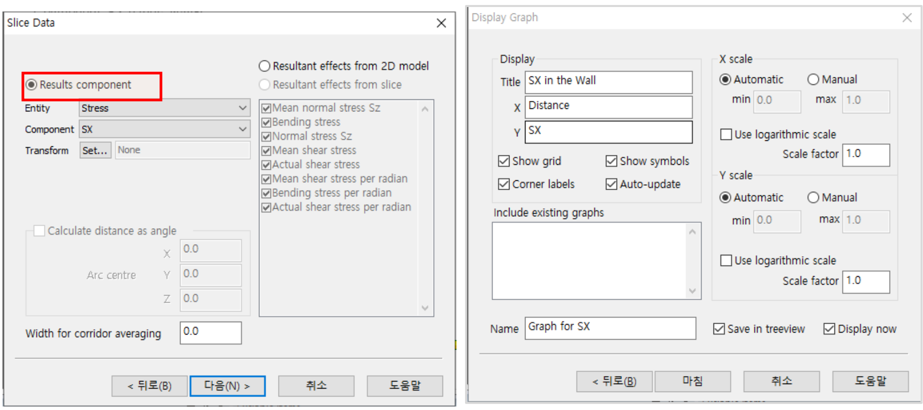


Fig 113 Graph Through 2D in a 2D Axisymmetric Solid Model (2)

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

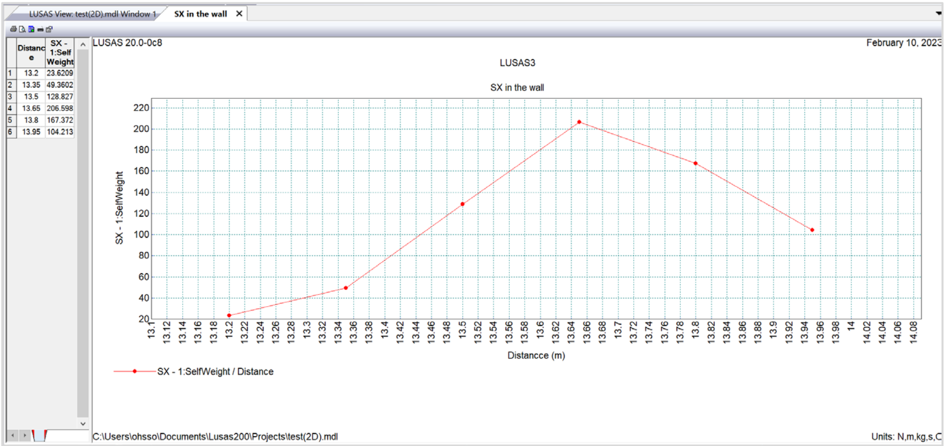


Fig 114 SX Graph for Sliced Line in a 2D Axisymmetric Solid Model

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.

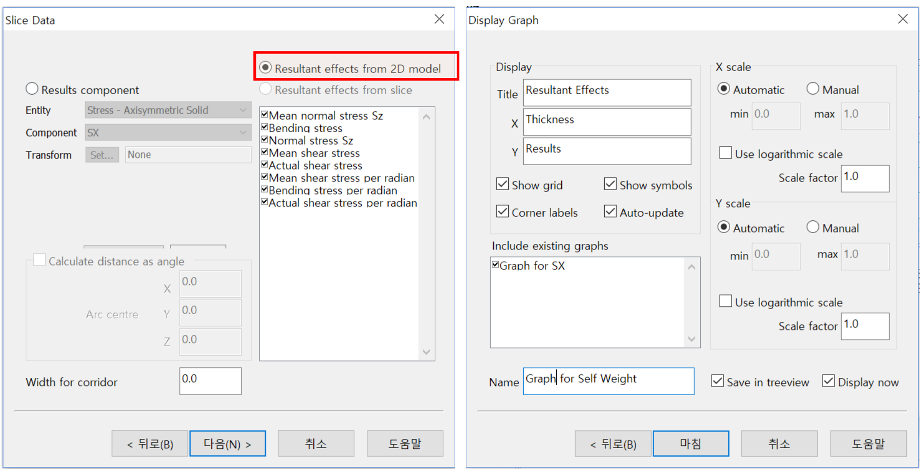


Fig 115 Graph Through 2D in a 2D Axisymmetric Solid Model (3)

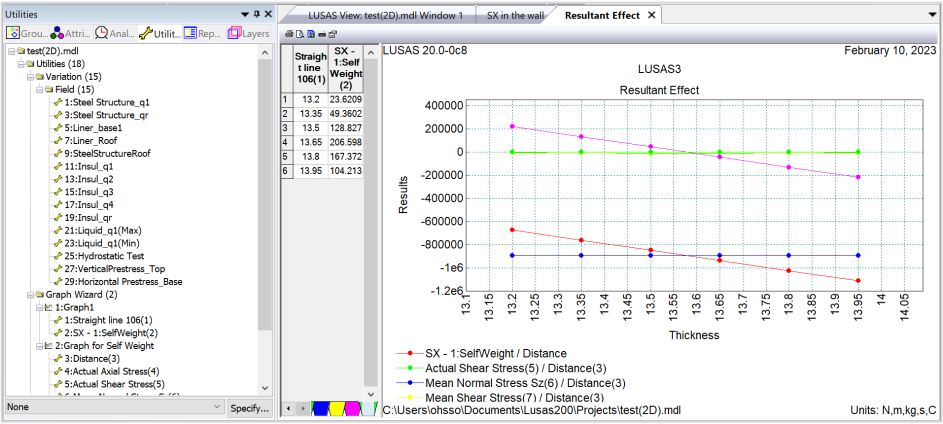


Fig 116 SX Graph for Sliced Line in a 2D Axisymmetric Solid Model

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.

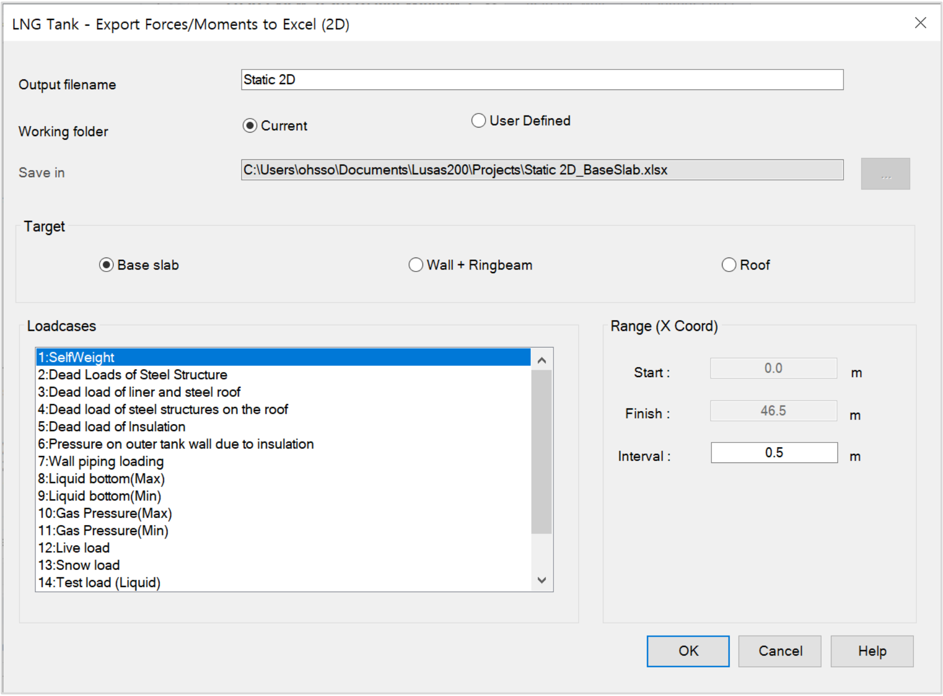


Fig 117 Export Forces for a 2D Axisymmetric Solid Model (1)

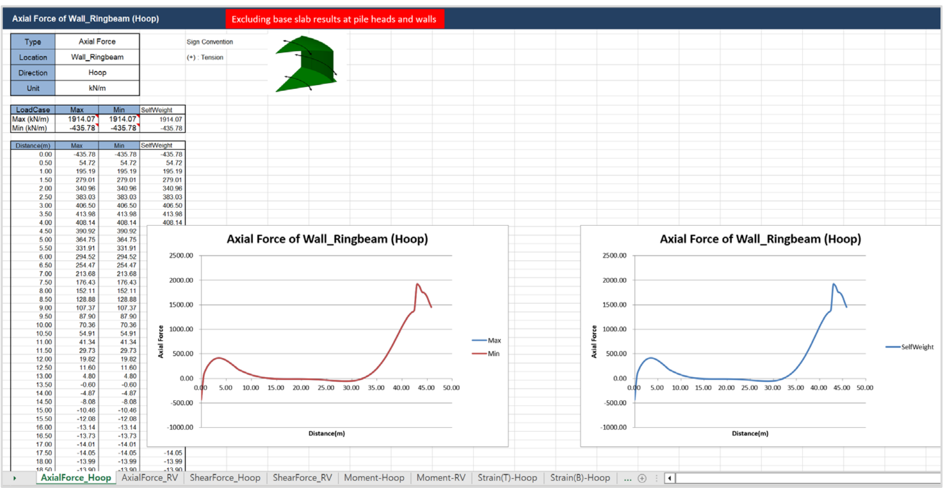


Fig 118 Section Force Spreadsheet for Self Weight

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

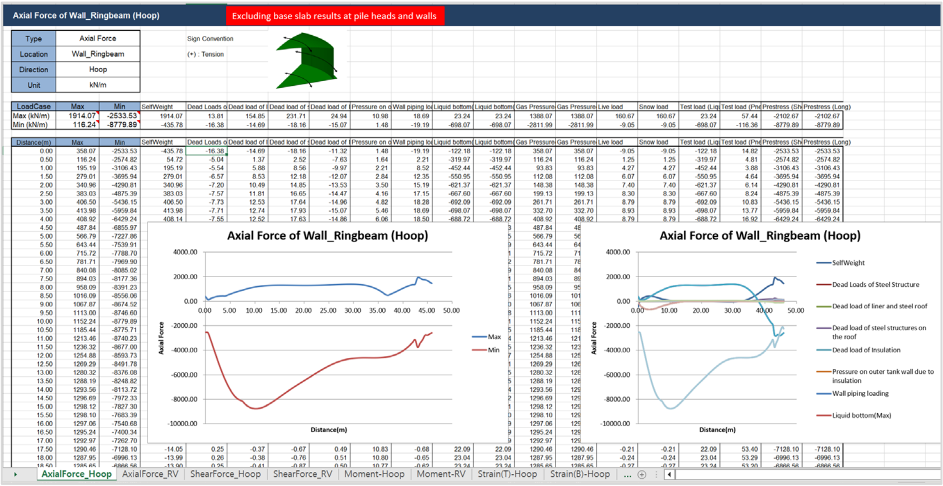


Fig 119 Section Force Spreadsheet for All Loadcases

Sign convention

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

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

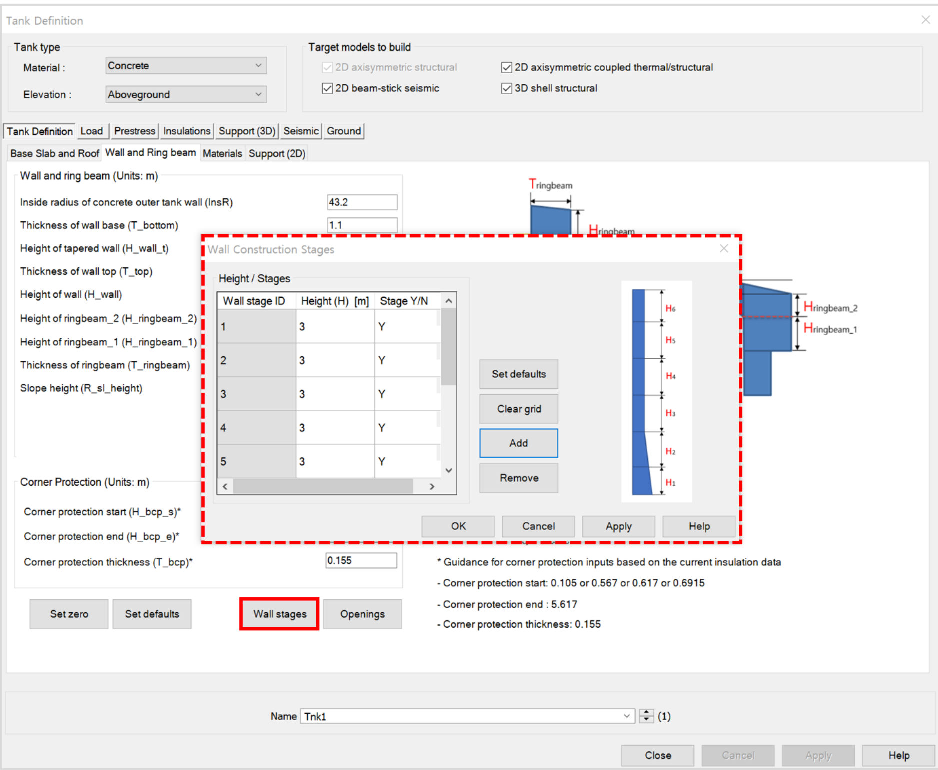
2D Axisymmetric Staged Construction Analysis

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.



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**

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

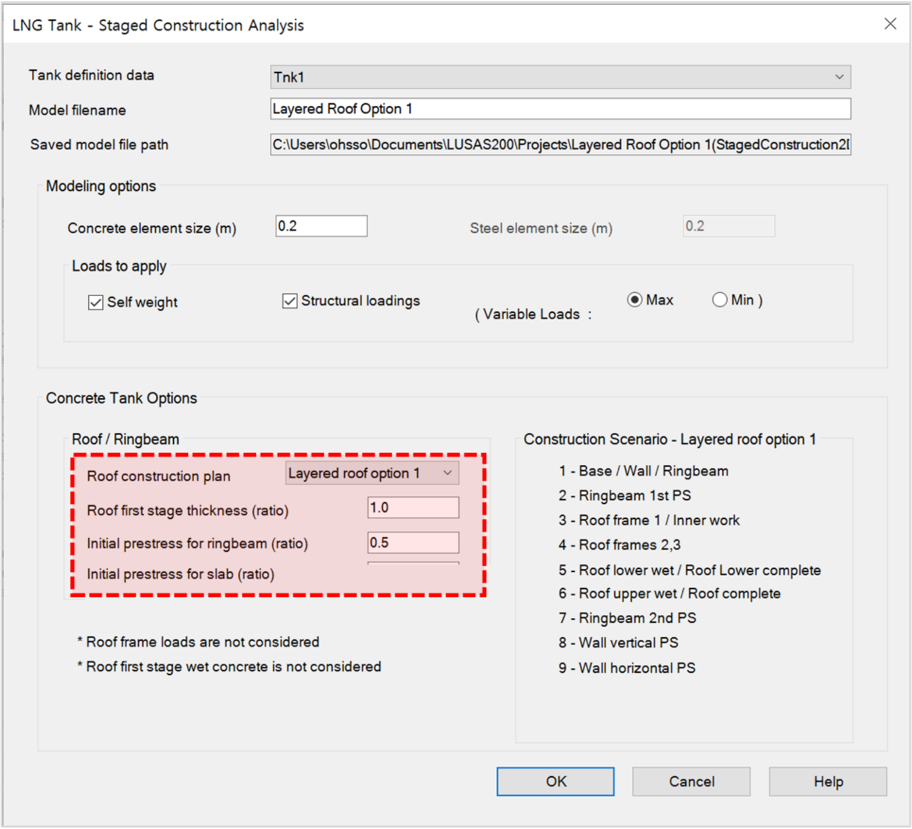


Fig 120 User Dialog for 2D Axisymmetric Staged Construction Analysis

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.

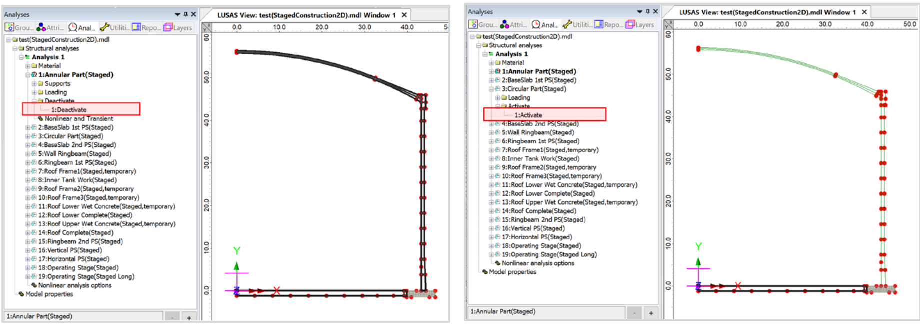


Fig 121 Activate and Deactivate Assignment in the Model

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.

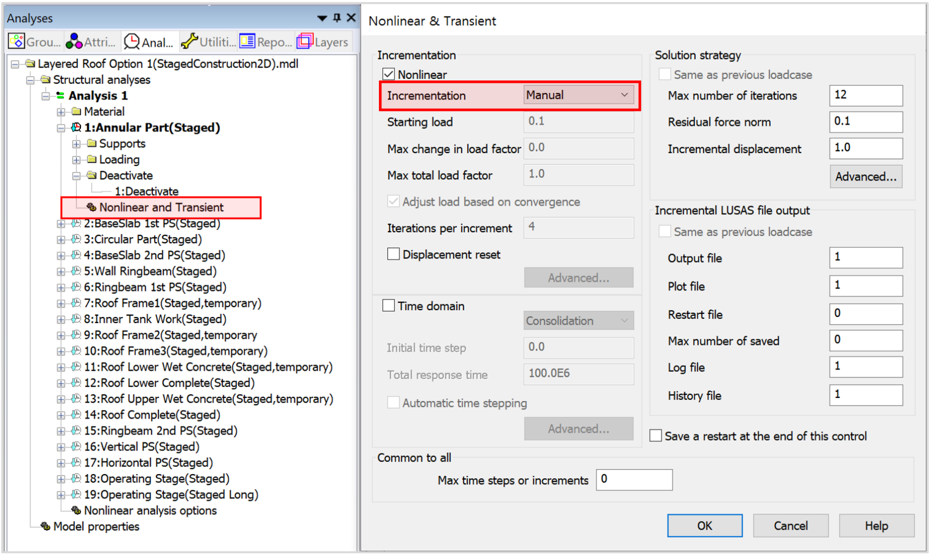


Fig 122 Nonlinear Control for a Staged Construction Analysis

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**.

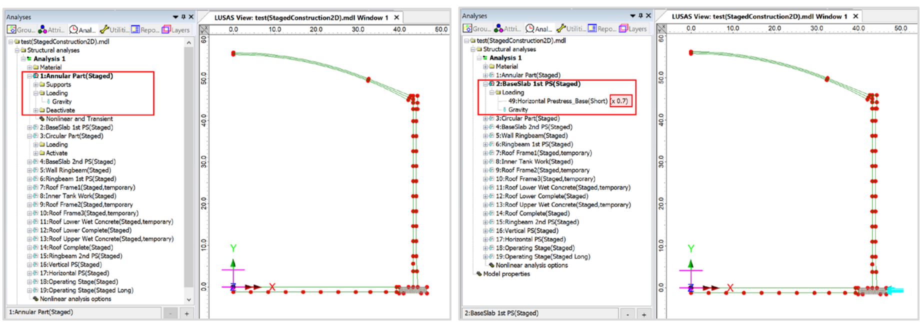


Fig 123 Loadings for Stage 1~2

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**.

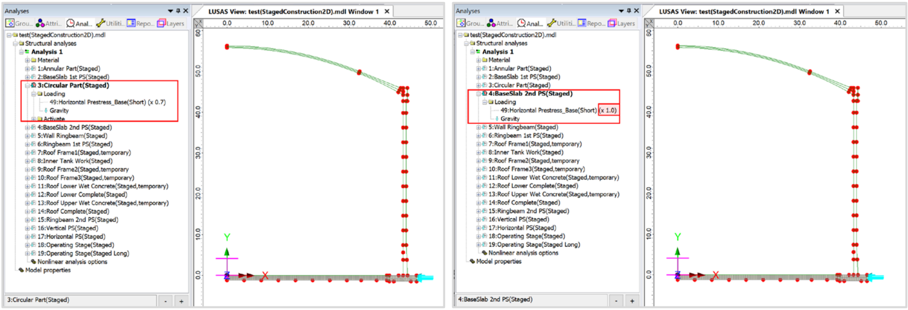


Fig 124 Loadings for Stage 3~4

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.

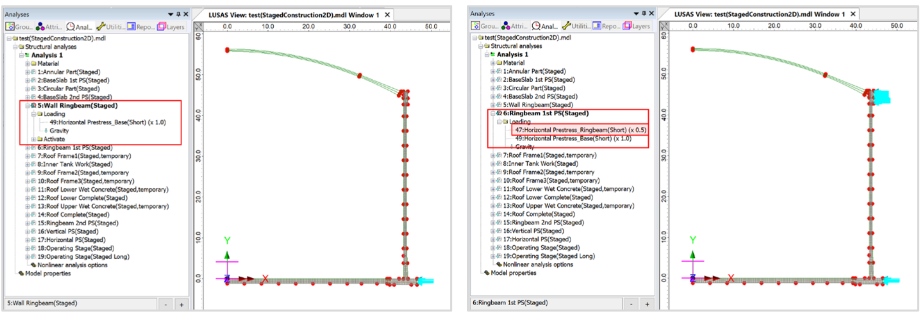


Fig 125 Loadings for Stage 5~6

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.

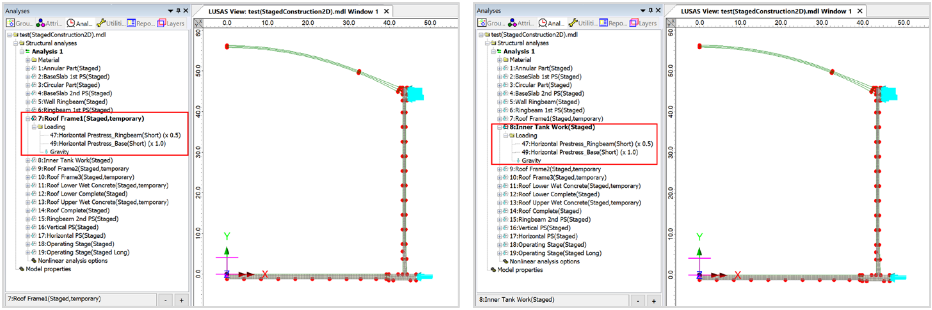


Fig 126 Loadings for Stage 7~8

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.

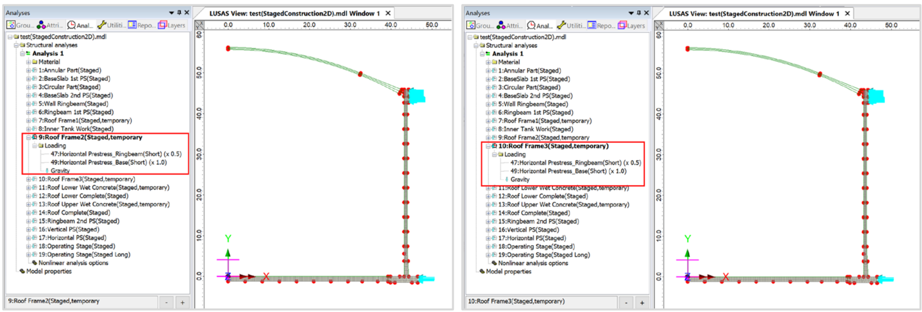


Fig 127 Loadings for Stage 9~10

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.

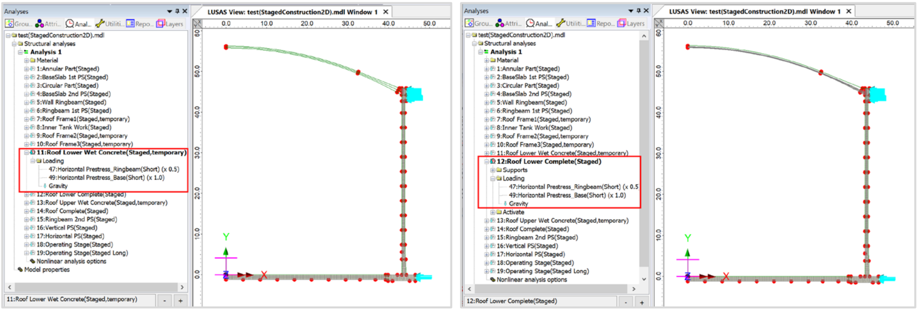


Fig 128 Loadings for Stage 11~12

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.

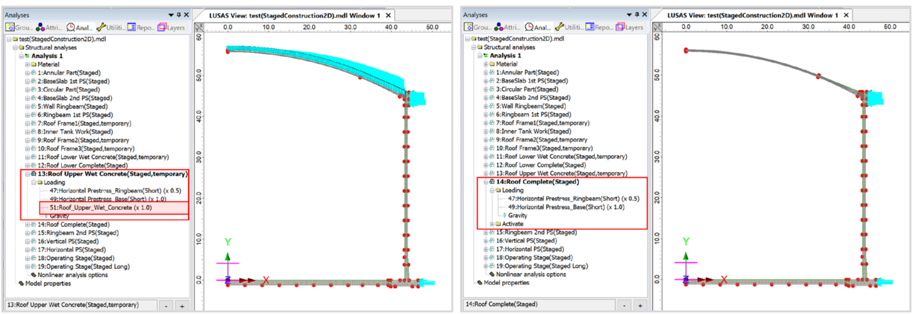


Fig 129 Loadings for Stage 13~14

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/m^2 is defined. This can be verified by assigning self weight to the upper part of the Roof and checking the reaction.

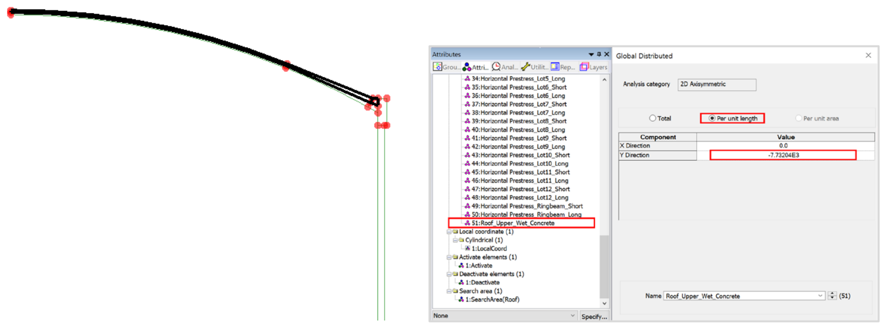


Fig 130 Load Definition for the Wet Concrete of Upper Part of the Roof

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**.

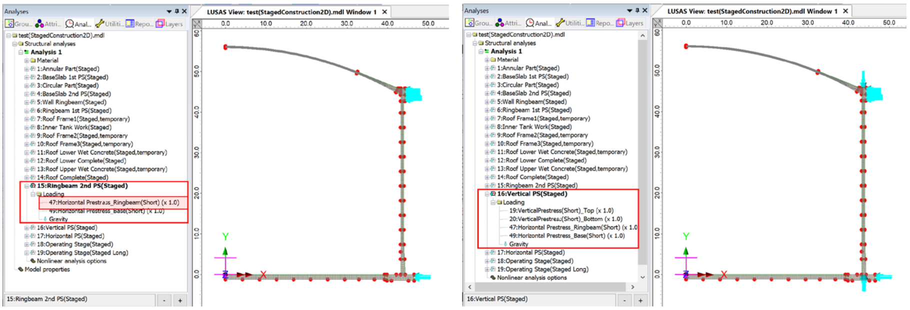


Fig 131 Loadings for Stage 15~16

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 only the short-term prestress is applied.

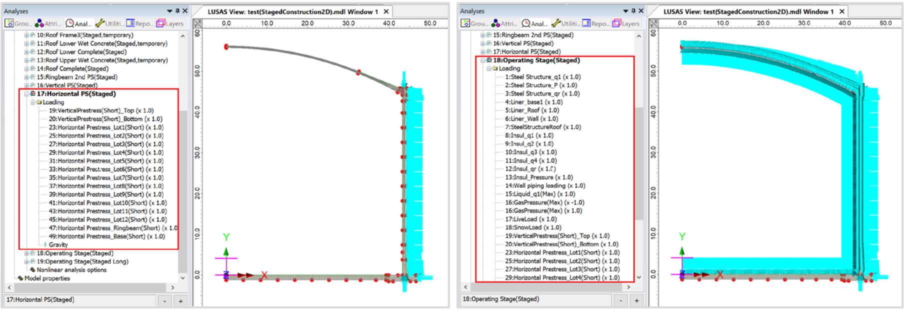


Fig 132 Loadings for Stage 17~18

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.

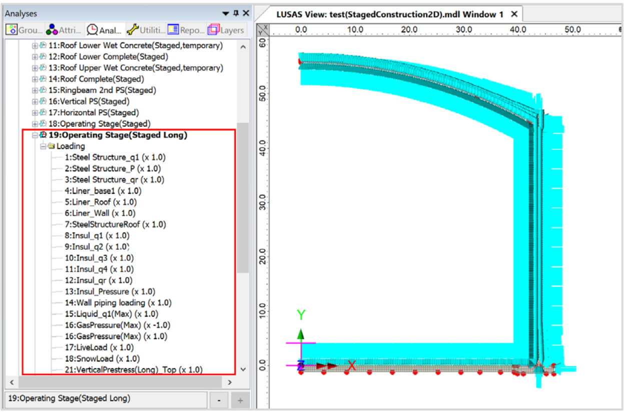


Fig 133 Loadings for Stage 19 of the Operating Stage

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

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
Description			Permanent															
			Outer tank			Others	Creep and Shrinkage		Prestress					Roof 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	Tank WO roof + RB 1st PS	1.35						1.30									
2	U-C1-2		1.35						1.00									
3	U-C1-3		1.00						1.30									
4	U-C1-4		1.00						1.00									
5	U-C2-1	Tank WO roof + RB 1st PS + Roof frame 1	1.35						1.30					1.50				
6	U-C2-2		1.35						1.00					1.50				
7	U-C2-3		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'.

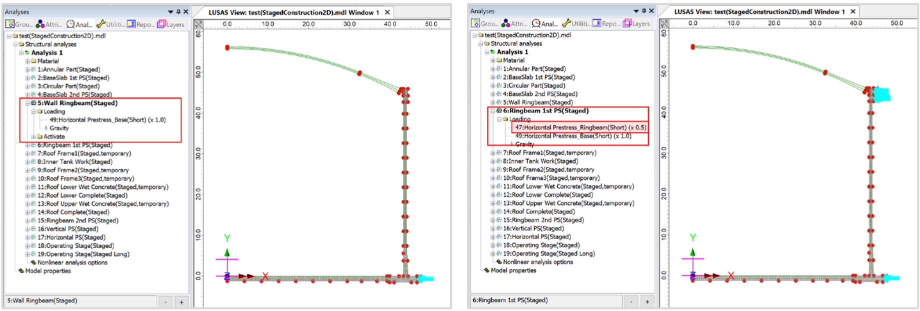


Fig 135 Loadings for Stage 5 and Stage 6

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.

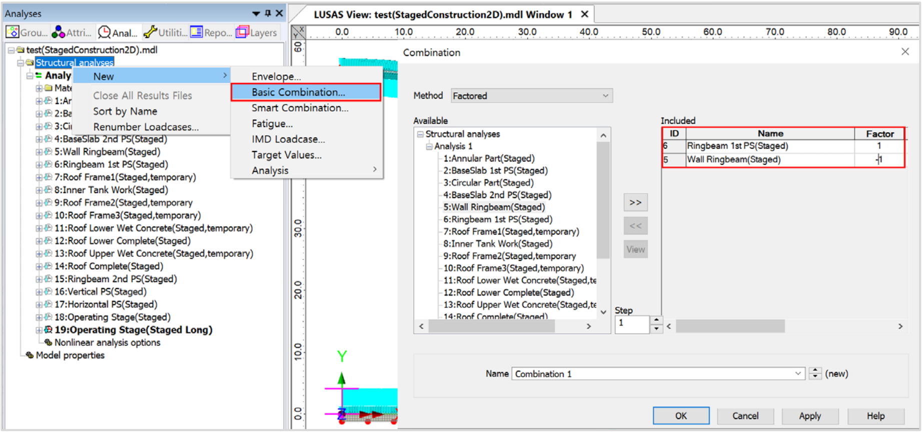


Fig 136 Definition of Load Combination for Pure PS effect

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.

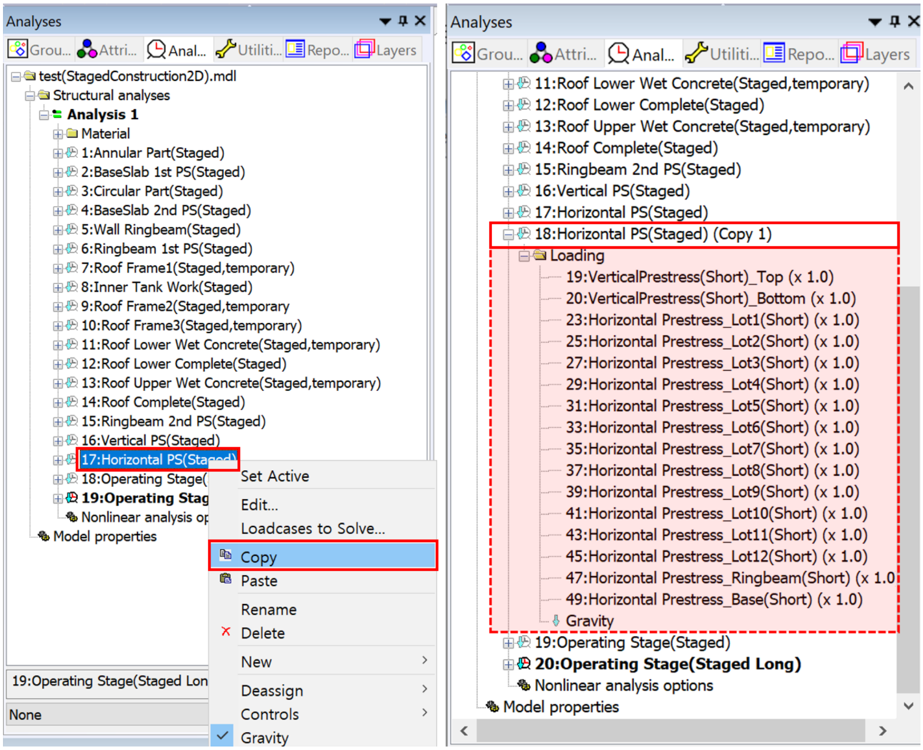


Fig 137 Adding Stages in the 2D Axisymmetric Staged Construction Analysis Model

2D Axisymmetric Thermal Analysis

User Inputs

The required user inputs for this model are as shown in [Fig 176].

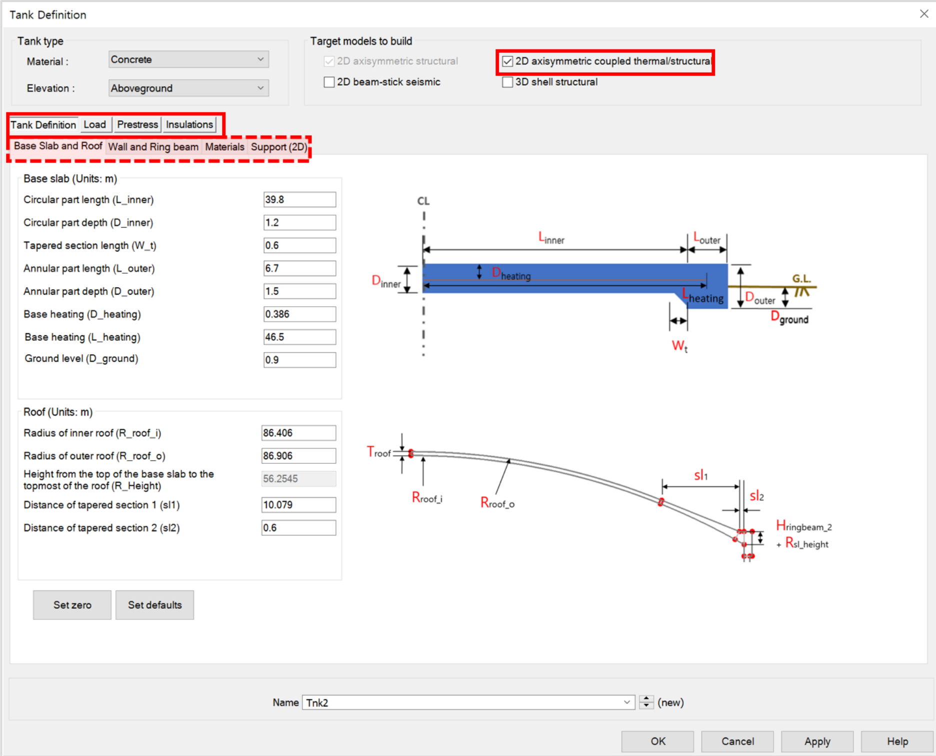


Fig 138 User Inputs for 2D Axisymmetric Thermal Analysis

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.

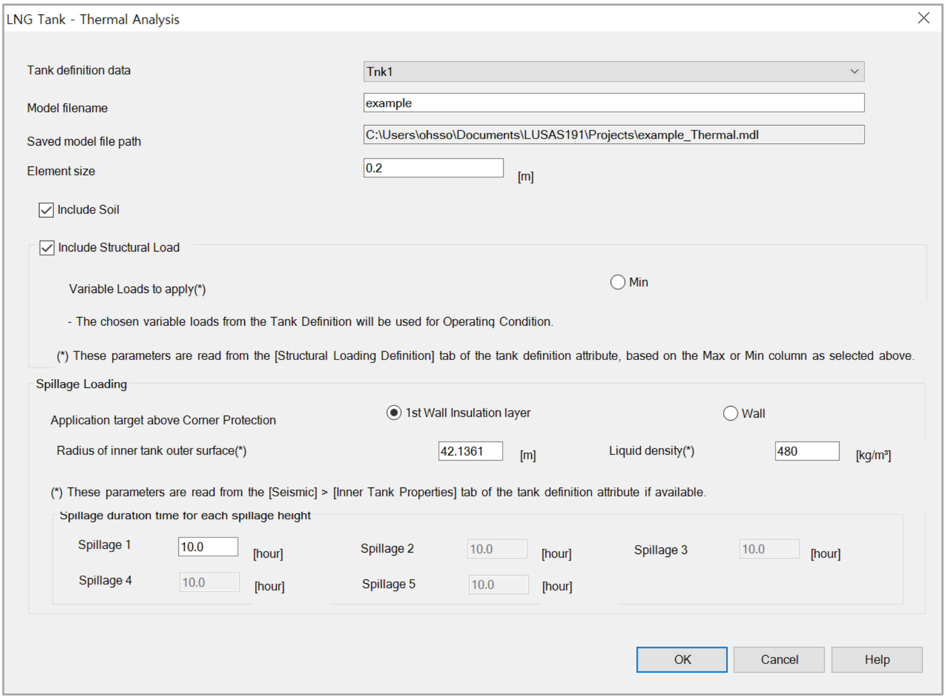


Fig 139 User Dialog for 2D Axisymmetric Thermal Analysis

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.

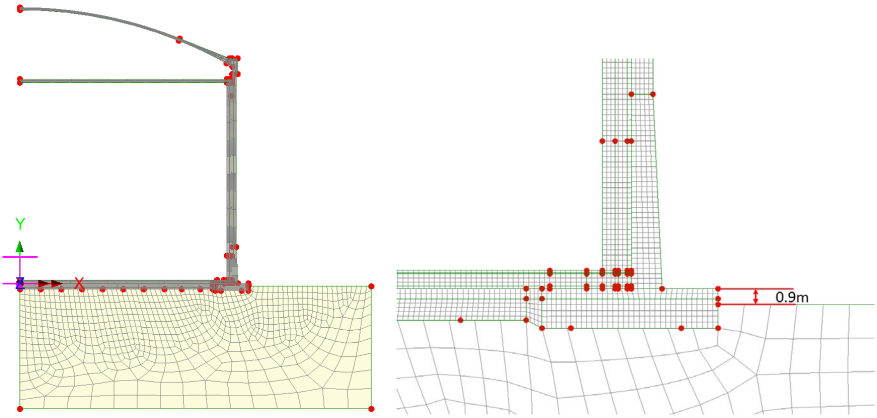


Fig 140 Elements for 2D Axisymmetric Thermal Analysis Model

Geometric Properties

No geometric properties are required for 2D axisymmetric model.

Material Properties

User defined material properties are assigned to the relevant surfaces.

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

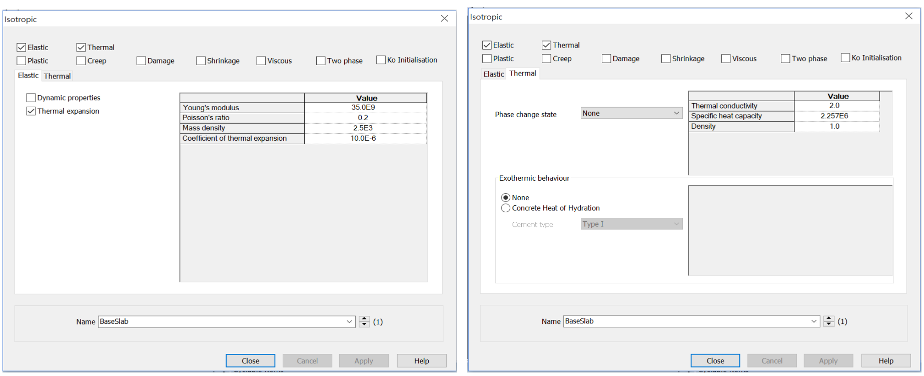


Fig 141 Material Properties of BaseSlab for a 2D Axisymmetric Thermal Analysis Model

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].

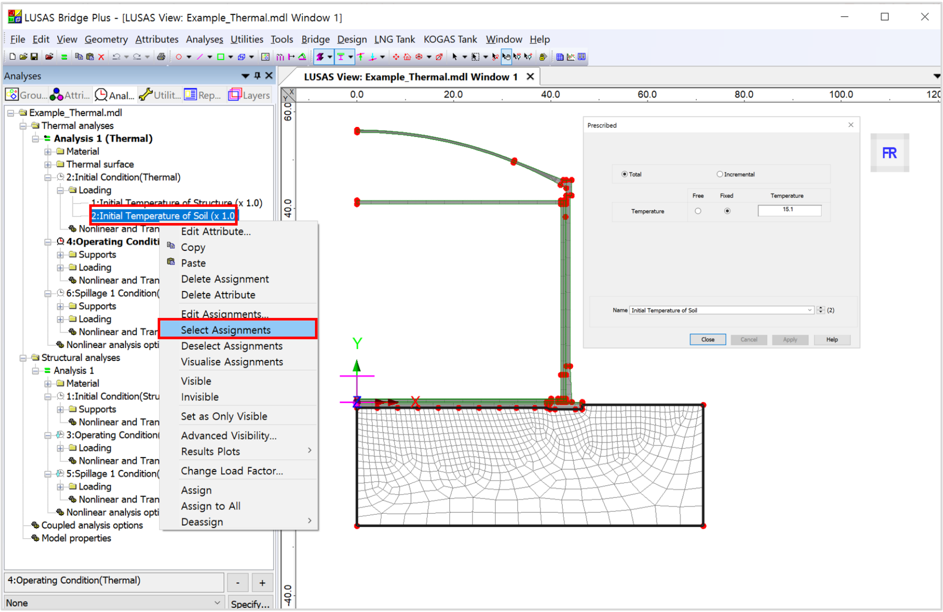


Fig 142 Initial Soil Temperature in a 2D Axisymmetric Thermal Analysis Model

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

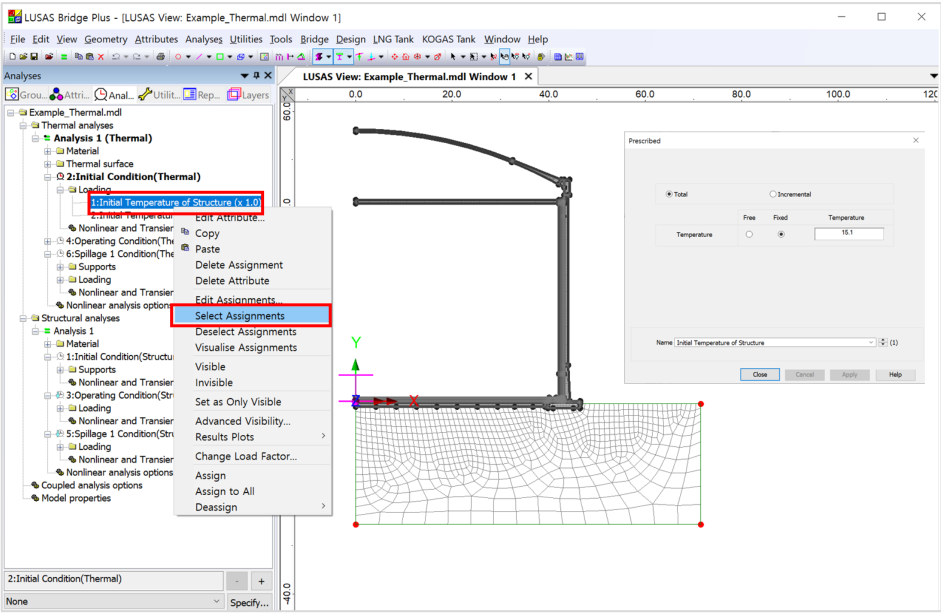


Fig 143 Initial Structure Temperature in a 2D Axisymmetric Thermal Analysis Model

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.

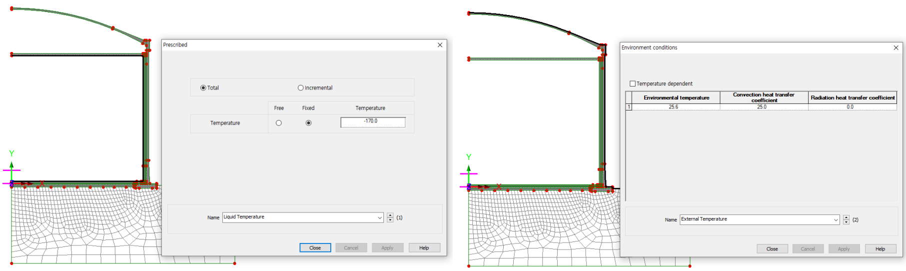


Fig 144 Operating Temperatures in a 2D Axisymmetric Thermal Analysis Model

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.

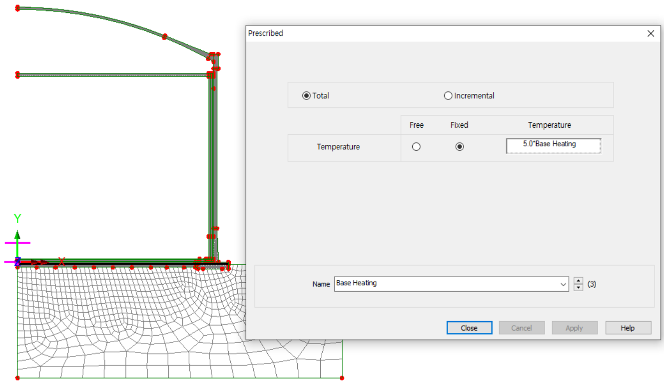


Fig 145 Base Heating Temperature in 2D Axisymmetric Thermal Analysis Model

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.

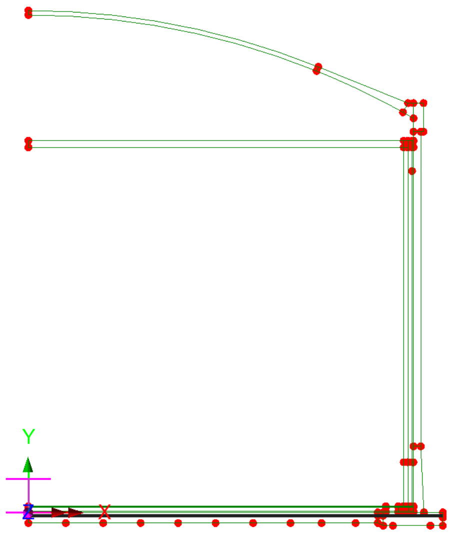


Fig 146 Base Heating Temperature in 2D Axisymmetric Thermal Analysis Model

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.

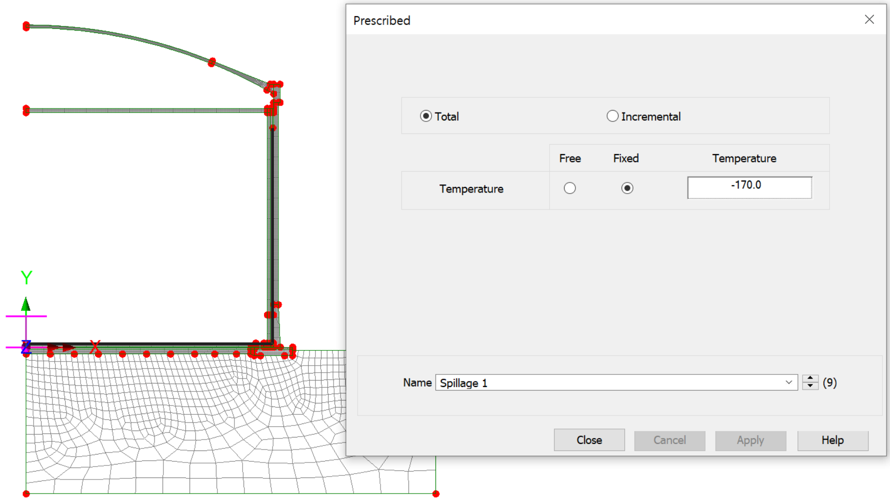


Fig 147 Spillage Temperature in 2D Axisymmetric Thermal Analysis Model

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].

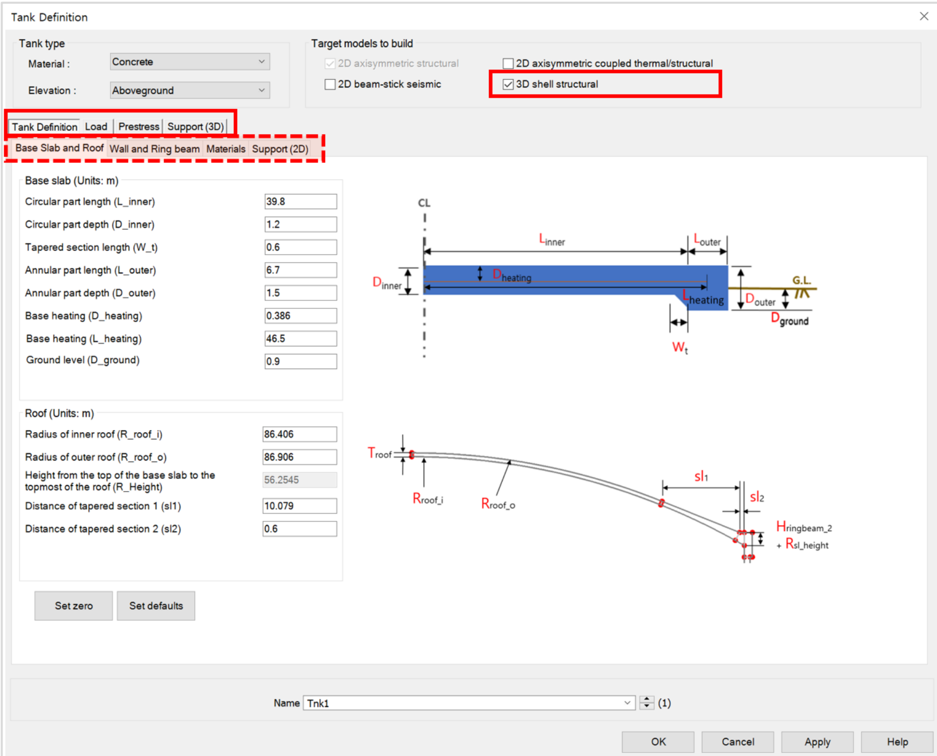


Fig 148 User Inputs for a 3D Shell Model

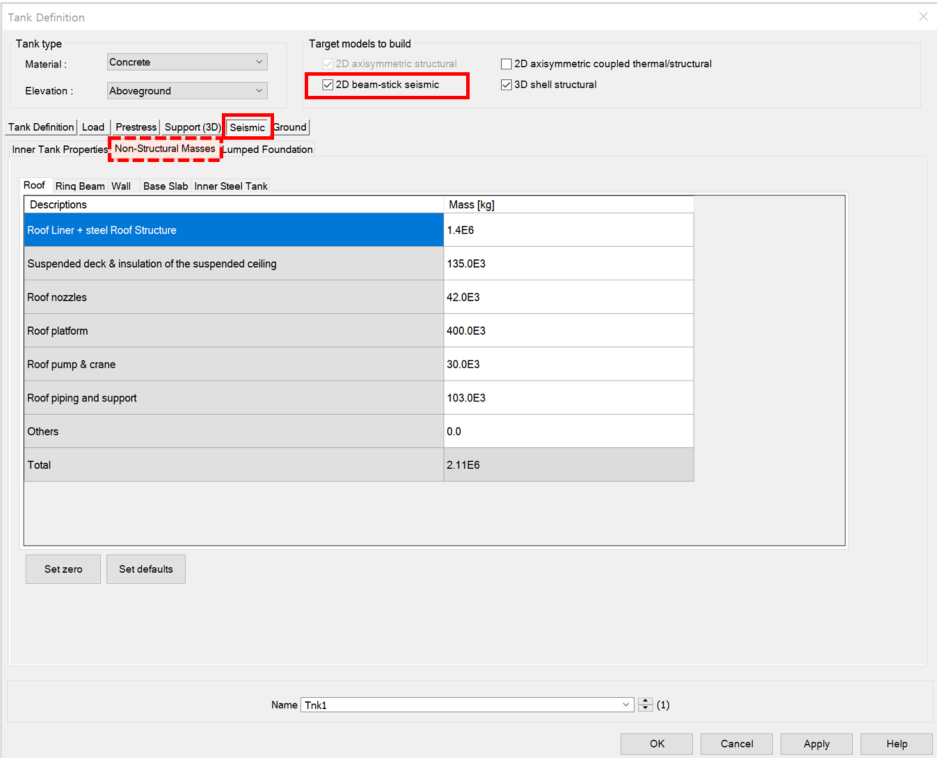


Fig 149 User Inputs for a 3D Shell Model including non-structural masses to Eigenvalue Analysis

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.**

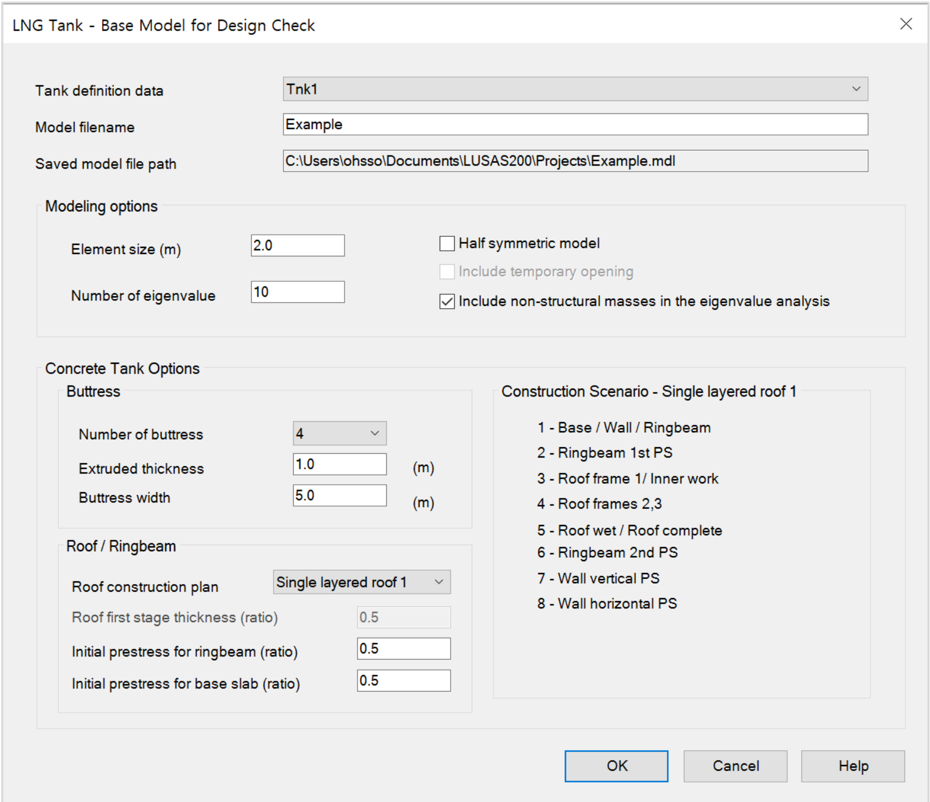


Fig 150 User Dialog for a 3D Shell Static Analysis Model

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.

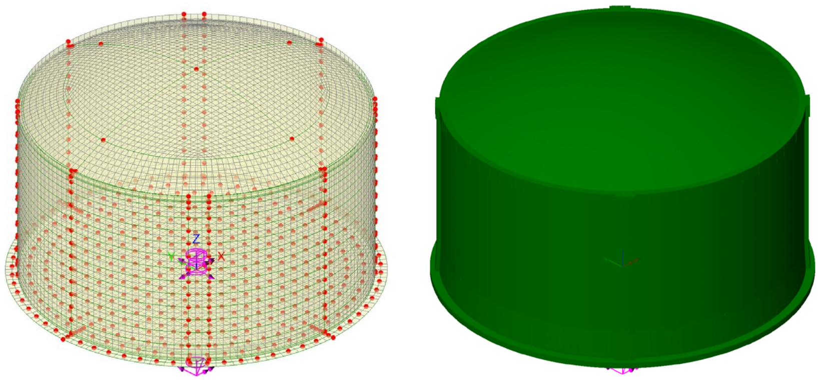


Fig 151 Mesh Arrangement and Geometric Properties for a 3D Shell Model

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.

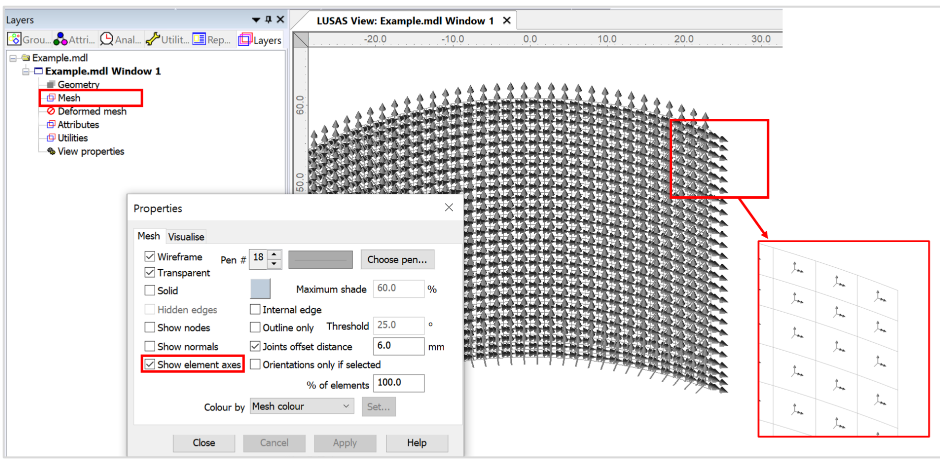


Fig 152 Element Local Axis in a 3D Shell Model

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 treeview.

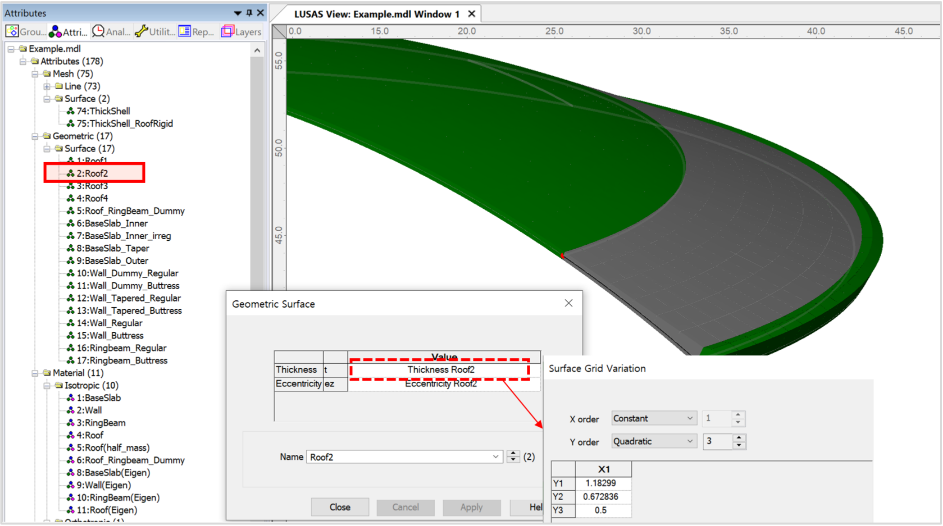


Fig 153 Geometric Properties for Roof in 3D Shell Model

TEST CASE

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

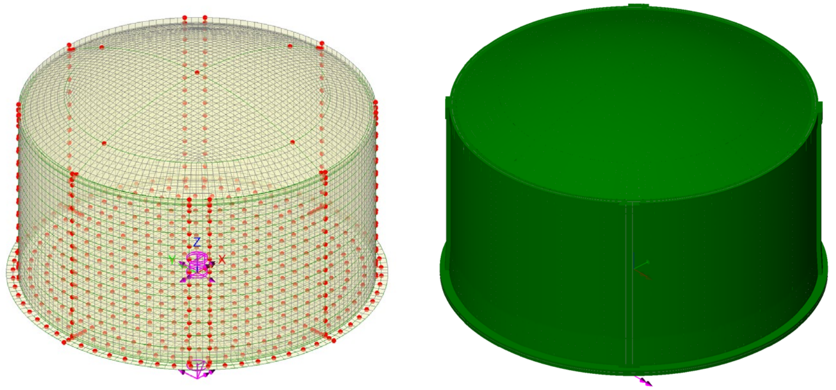


Fig 154 Mesh Arrangement and Geometric Properties for a 3D Shell Model with no Buttresses

Material Properties

Structural members

Material properties are defined and assigned as shown in [Fig 197].

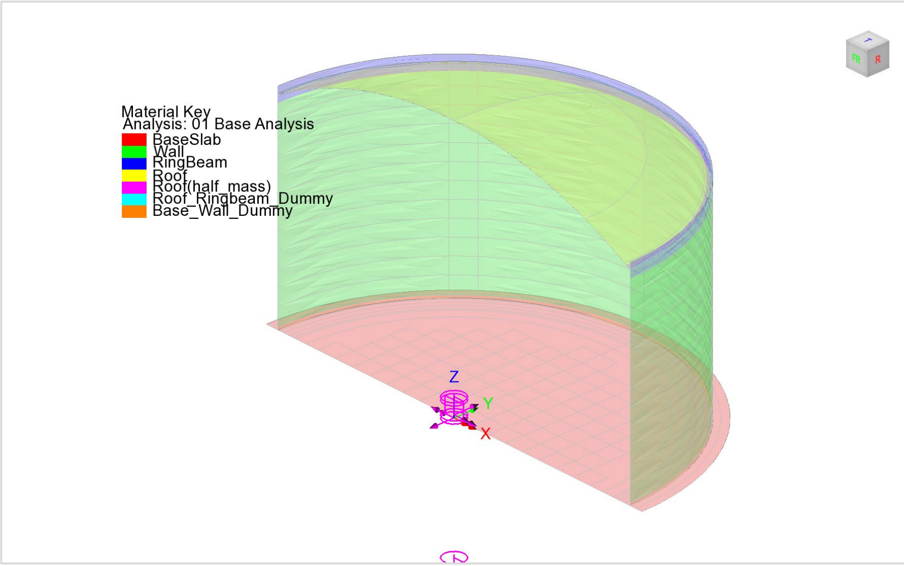
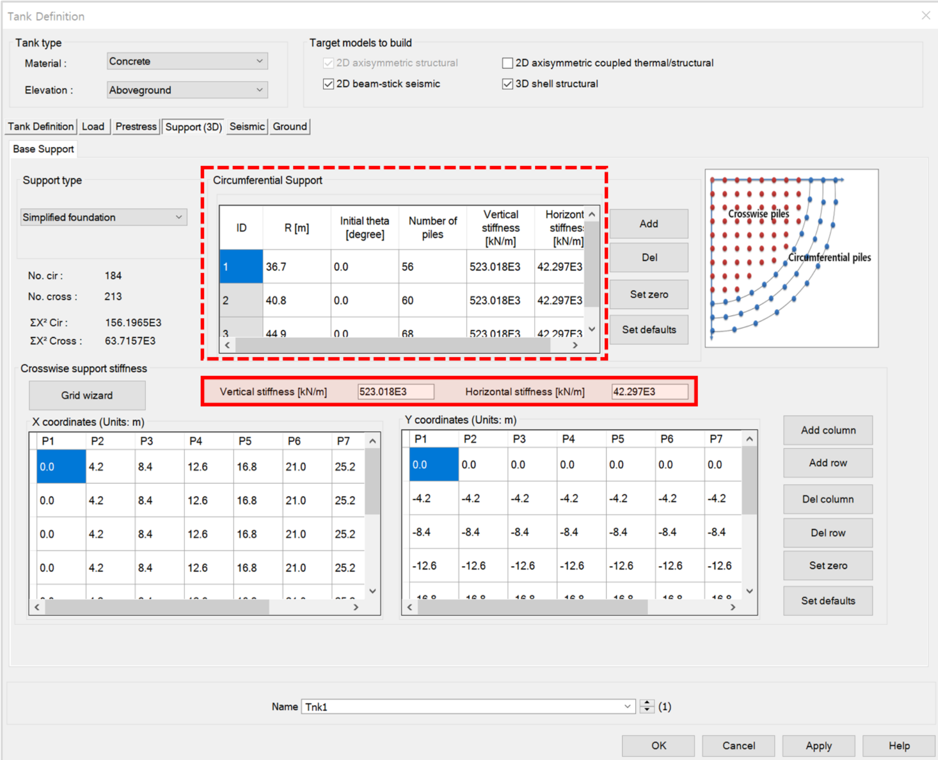


Fig 155 Material Properties in a 3D Shell Model

Support Conditions

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


Tank Definition
Tank type
Material : Concrete
Elevation : Aboveground
Target models to build
2D axisymmetric structural
2D axisymmetric coupled thermal/structural
2D beam-stick seismic
3D shell structural
Tank Definition Load Prestress Support (3D) Seismic Ground
Base Support
Support type
Simplified foundation
No. cir : 184
No. cross : 213
ΣX² Cir : 156.1965E3
ΣX² Cross : 63.7157E3
Circumferential Support
ID R [m] Initial theta [degree] Number of piles Vertical stiffness [kN/m] Horizont stiffnes [kN/m]
1 36.7 0.0 56 523.018E3 42.297E3
2 40.8 0.0 60 523.018E3 42.297E3
Add
Del
Set zero
Set defaults
Crosswise piles
Circumferential piles
Crosswise support stiffness
Grid wizard
Vertical stiffness [kN/m] 523.018E3
Horizontal stiffness [kN/m] 42.297E3
X coordinates (Units: m)
Y coordinates (Units: m)
Add column
Add row
Del column
Del row
Set zero
Set defaults
Name Tnk1 (1)
OK
Cancel
Apply
Help

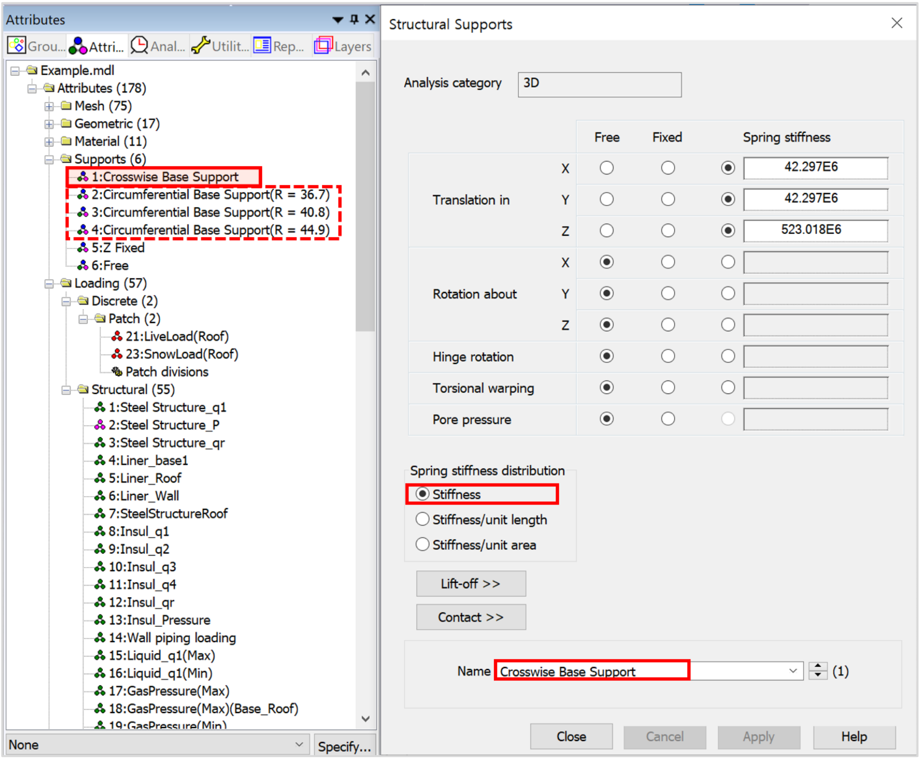


Fig 156 Support Definition in a 3D Shell Model

TEST CASE

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

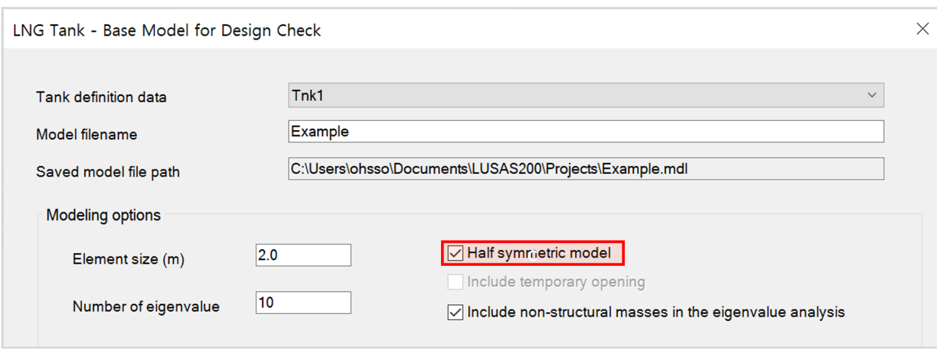


Fig 157 Option for Half Model

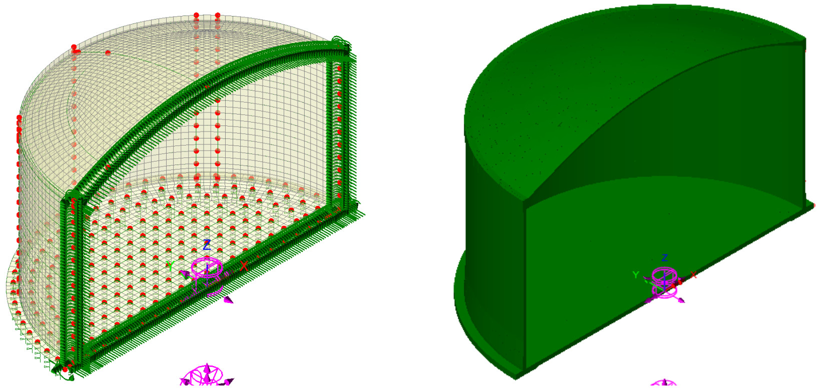


Fig 158 Half Symmetric Model

Loadings

28 loadcases are defined in the model.

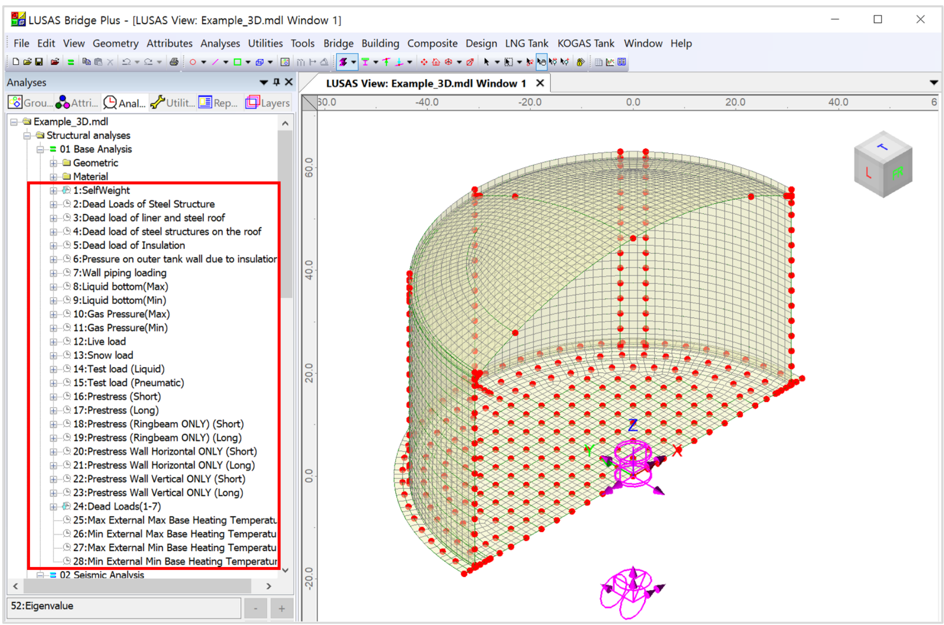


Fig 159 Loadcases Available in a 3D Shell Model

Self Weight

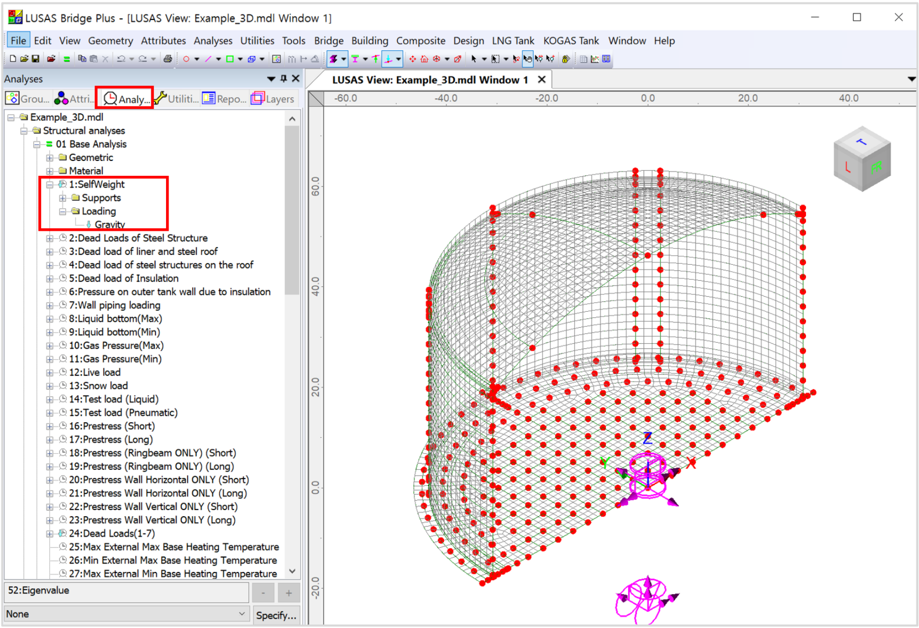


Fig 160 Self Weight in a 3D Shell Model

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.

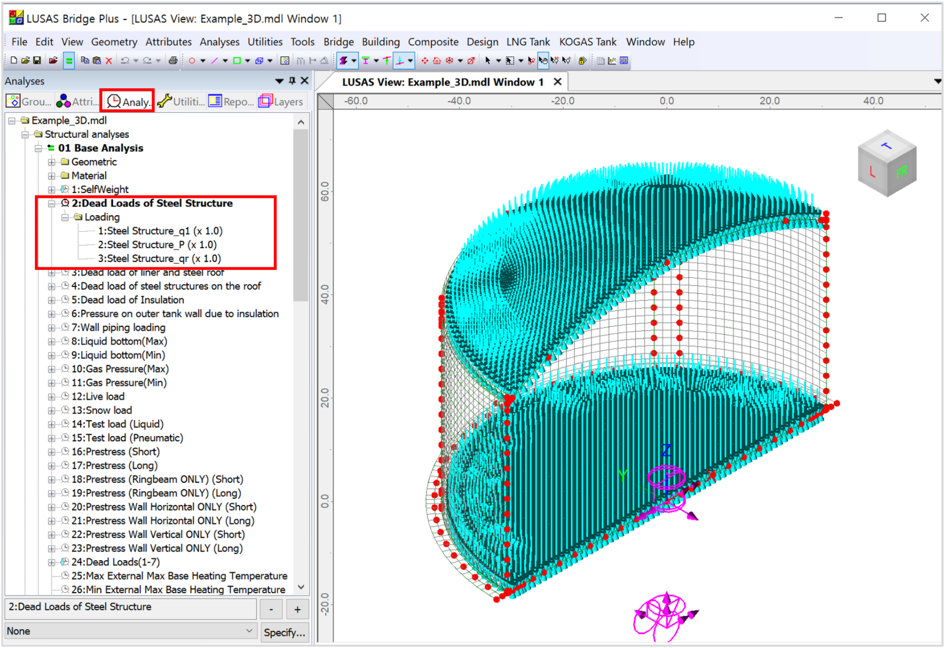


Fig 161 Dead Loads for Steel Structure in a 3D Shell Model

Dead load of liner and steel roof

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

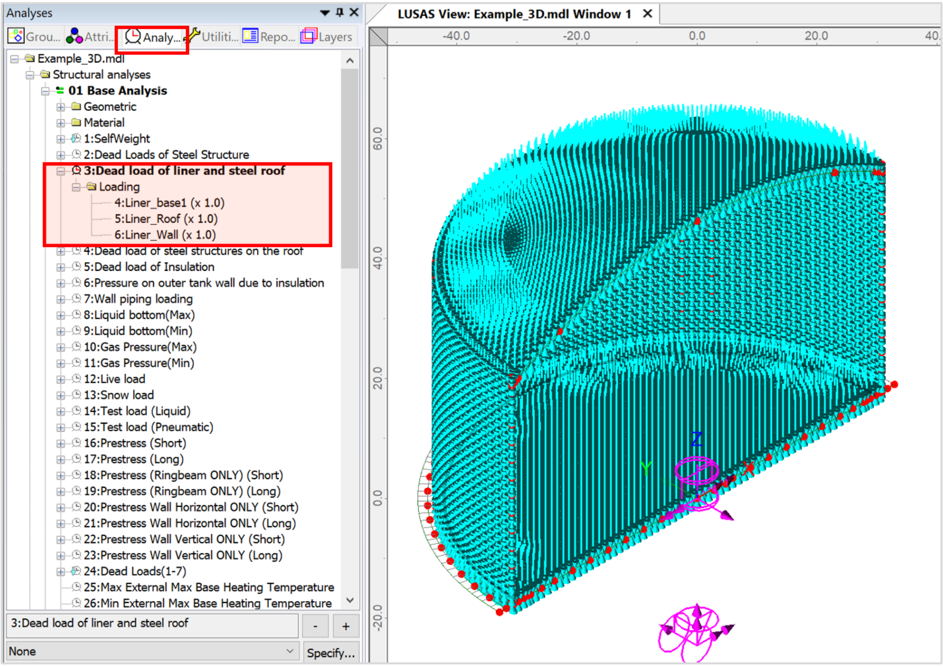


Fig 162 Dead Load of Liner and Steel Roof in a 3D Shell Model

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.

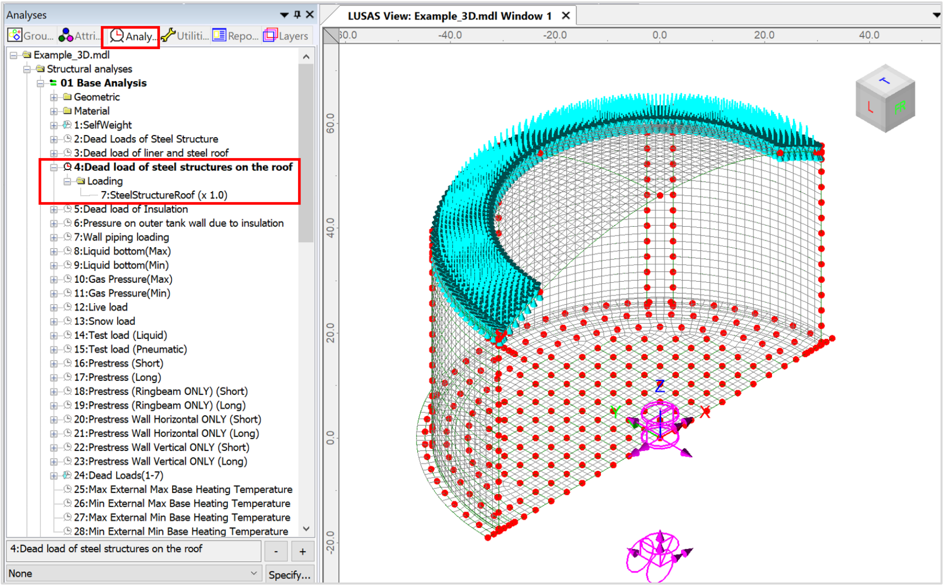


Fig 163 Dead Load of Steel Structures on the Roof in a 3D Shell Model

Dead load of Insulation

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

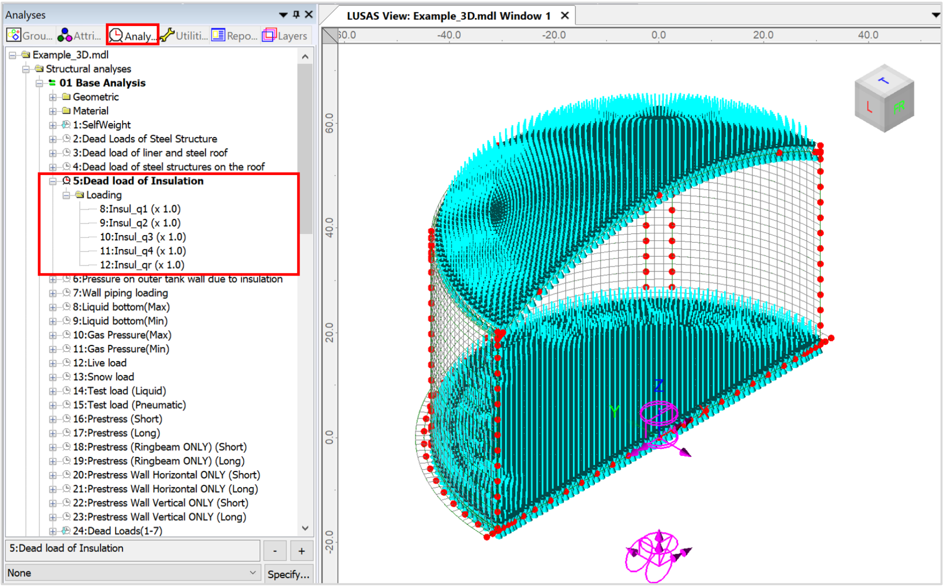


Fig 164 Dead Load of Insulation in a 3D Shell Model

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.

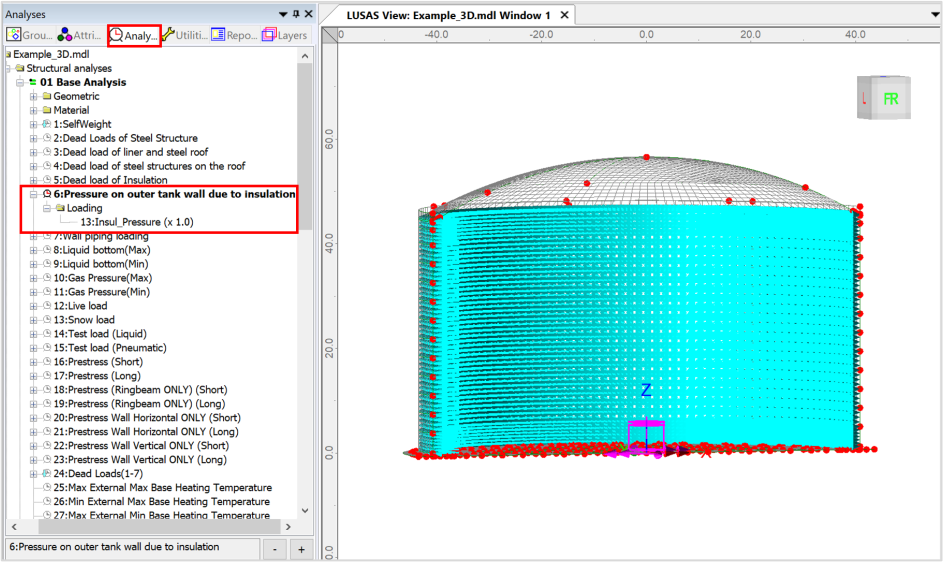


Fig 165 Insulation Pressure Load in a 3D Shell Model

Wall Piping Loading

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

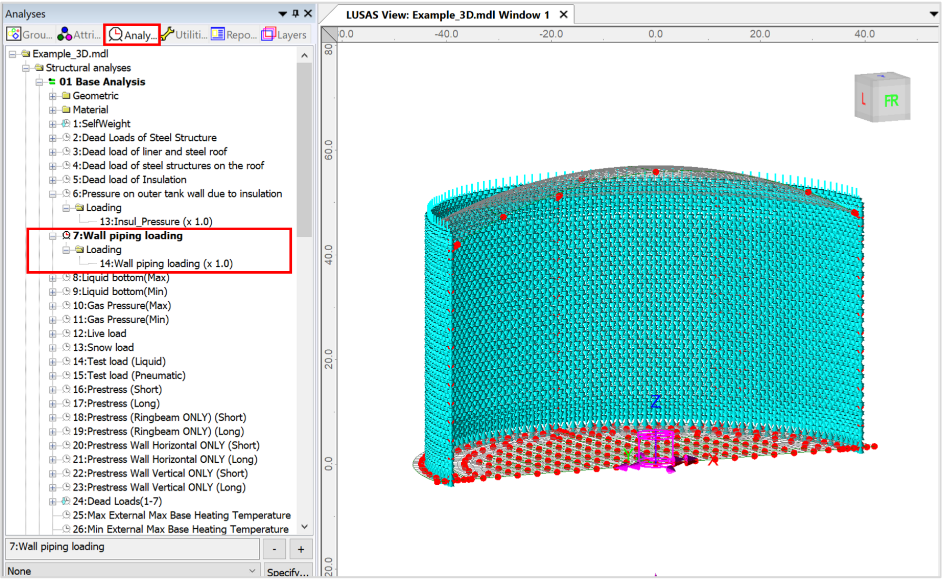


Fig 166 Wall piping loading in a 3D Shell Static Analysis Model

Liquid bottom (Max/Min)

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

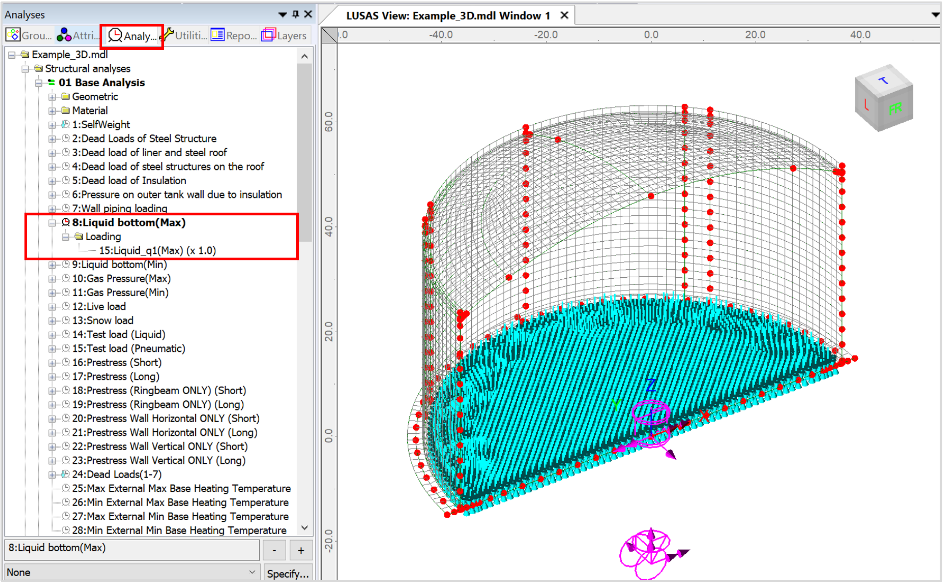


Fig 167 Liquid Bottom Loading in a 3D Shell Model

Gas Pressure(Max/Min)

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

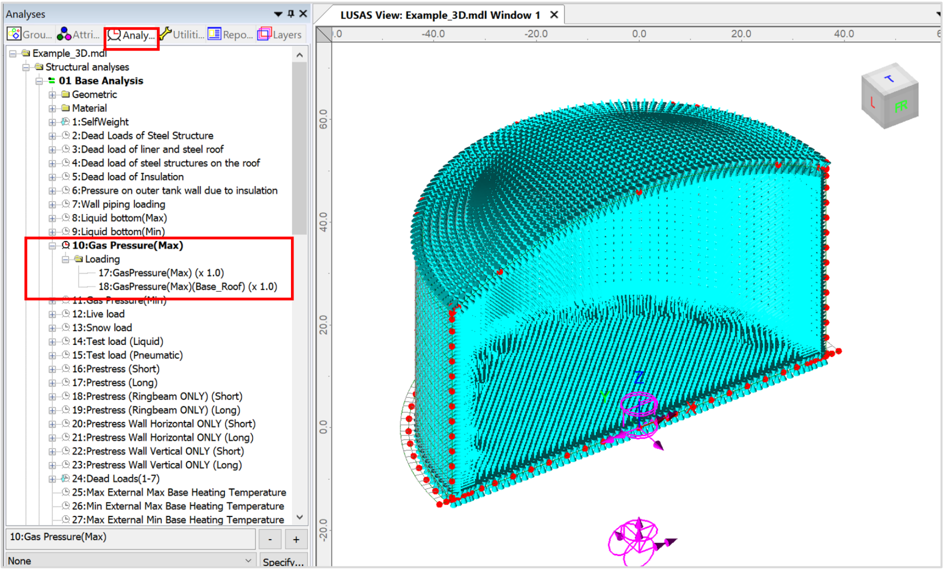


Fig 168 Gas Pressure Loading in a 3D Shell Static Analysis Model

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.

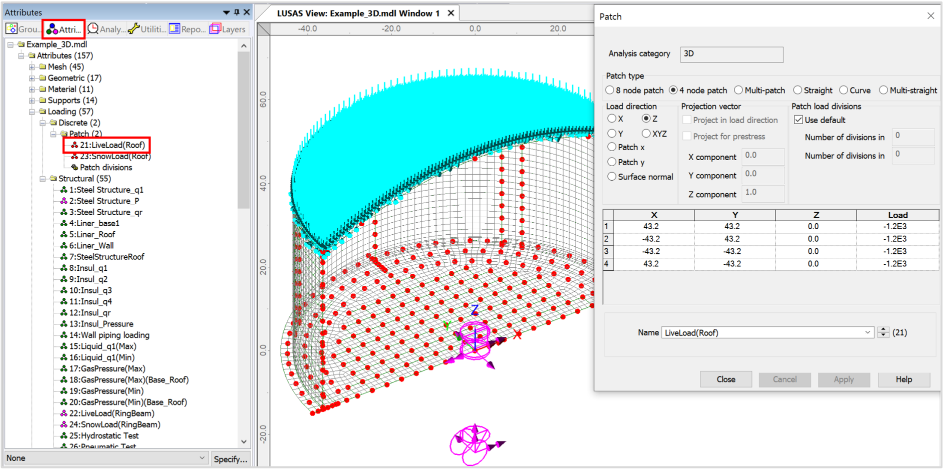
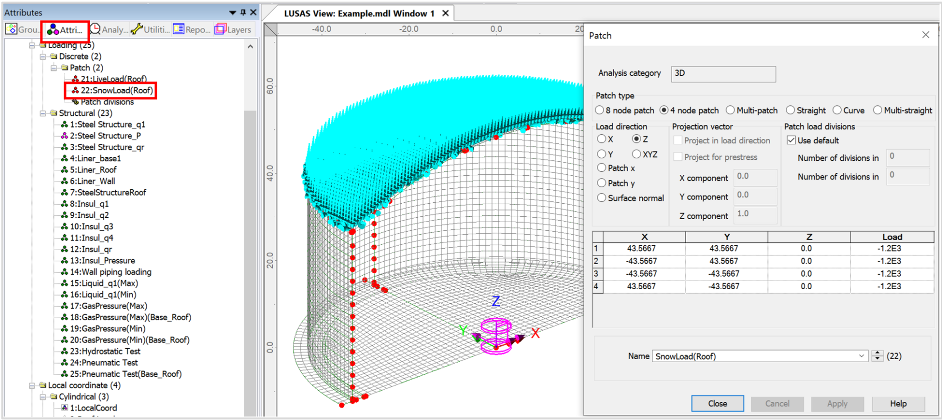


Fig 169 Live Load in a 3D Shell Static Analysis Model (Roof)

Snow load

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



	X	Y	Z	Load
1	43.5667	43.5667	0.0	-1.2E3
2	-43.5667	43.5667	0.0	-1.2E3
3	-43.5667	-43.5667	0.0	-1.2E3
4	43.5667	-43.5667	0.0	-1.2E3

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.

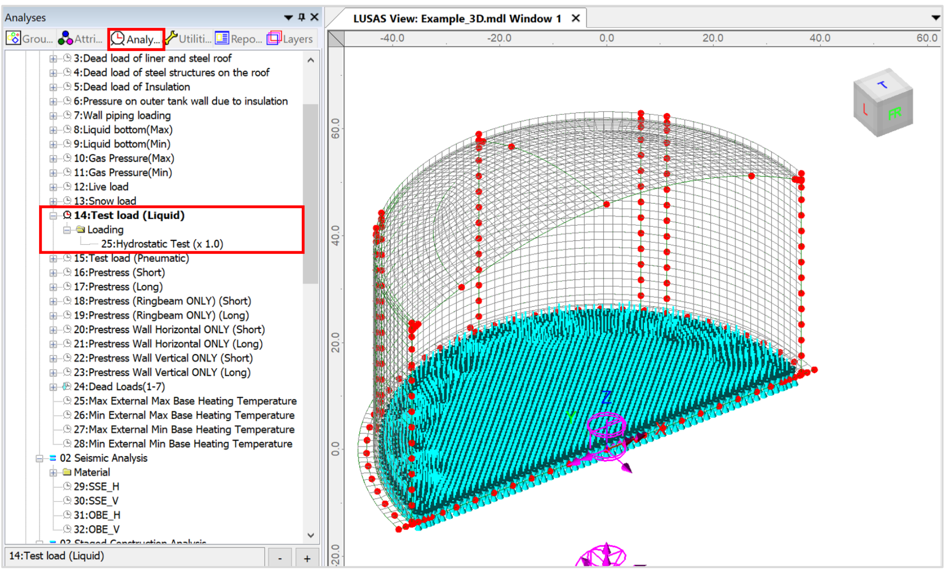


Fig 171 Test Load (Liquid Bottom) in a 3D Shell Model

Test load (Pneumatic)

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

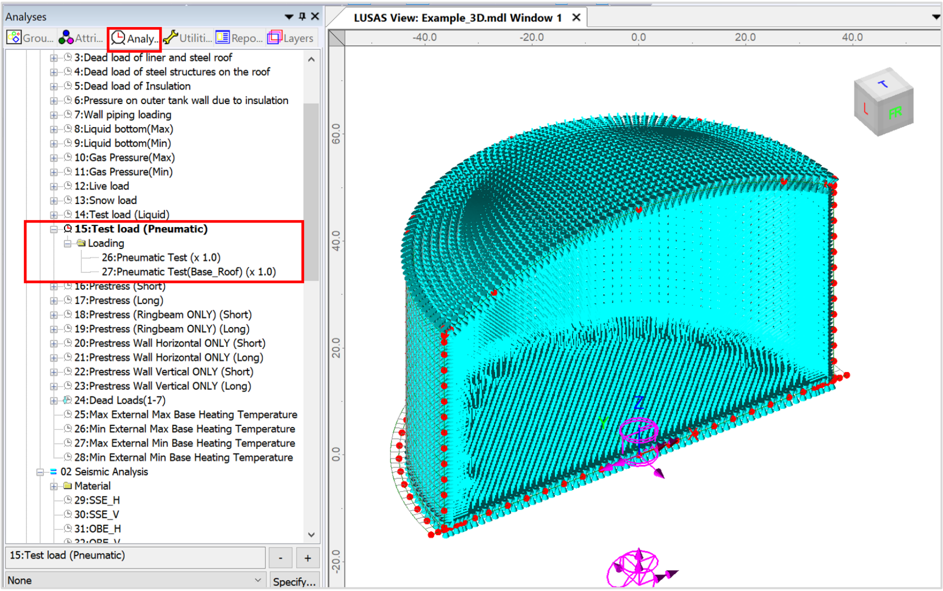


Fig 172 Test Load (Pneumatic) in a 3D Shell Model

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.

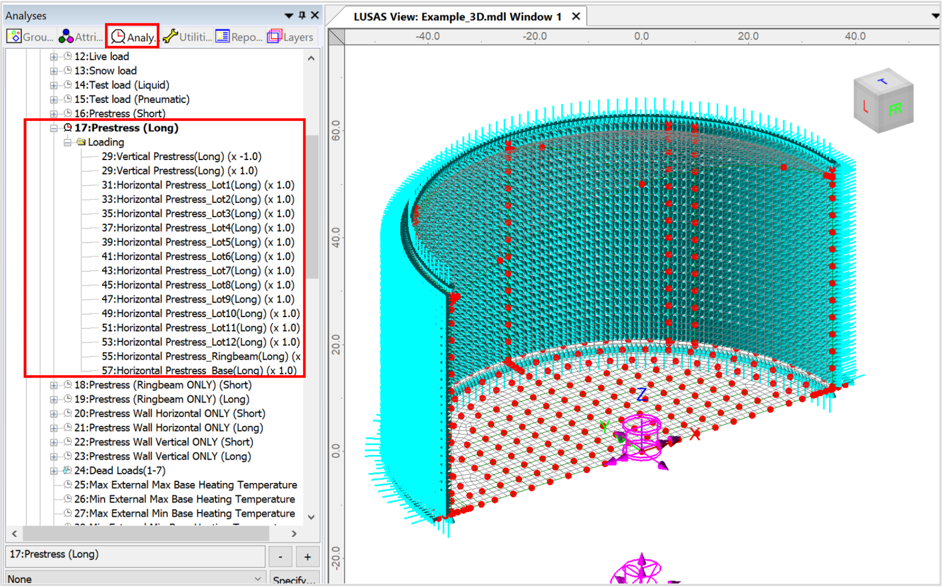


Fig 173 Prestress Load in a 3D Shell Model

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.

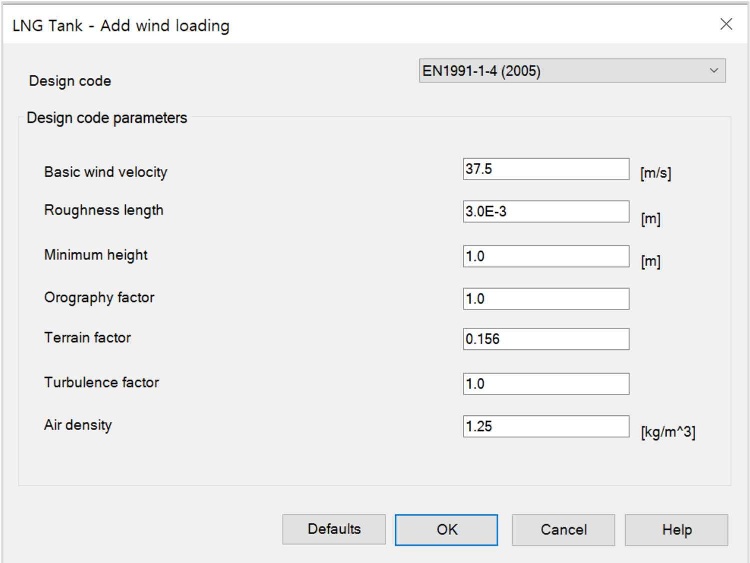


Fig 174 Wind Load in a 3D Shell Model

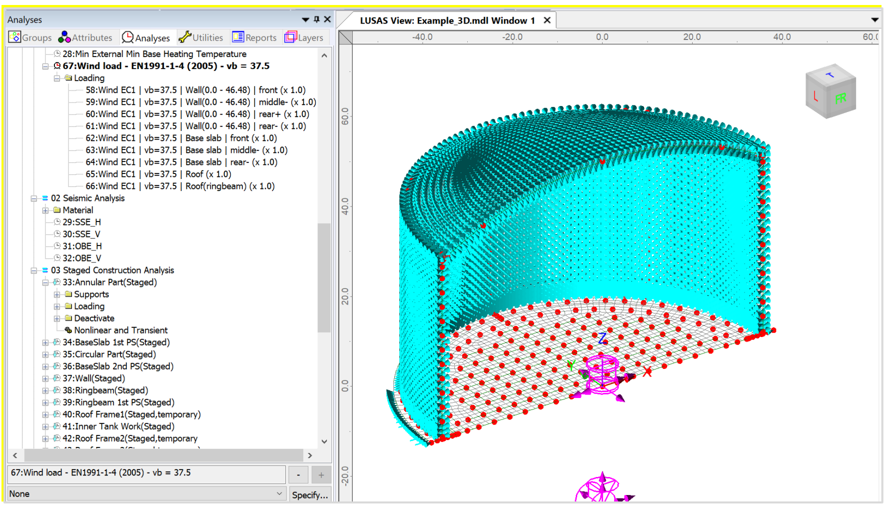


Fig 175 Wind Load in a 3D Shell Model

Viewing Results

Contours

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

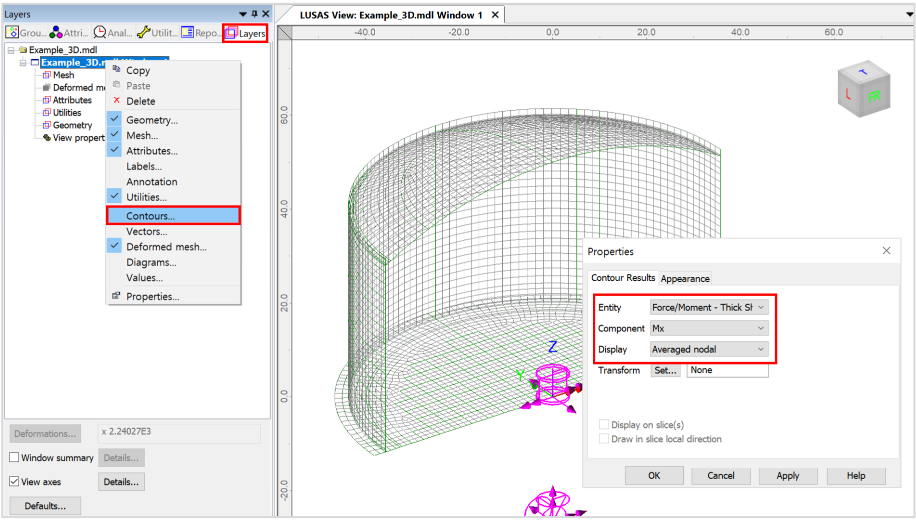


Fig 176 Selection for Contour Display in 3D Shell Model

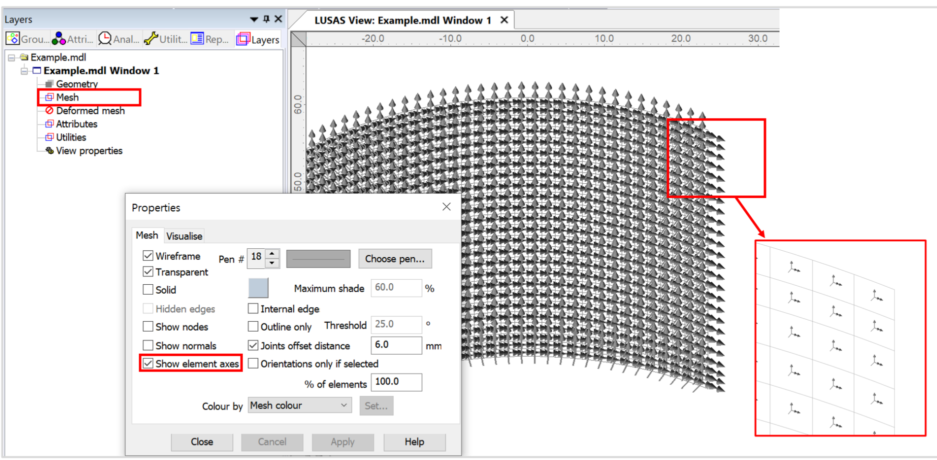


Fig 177 Element Local Axis in a 3D Shell Model

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.

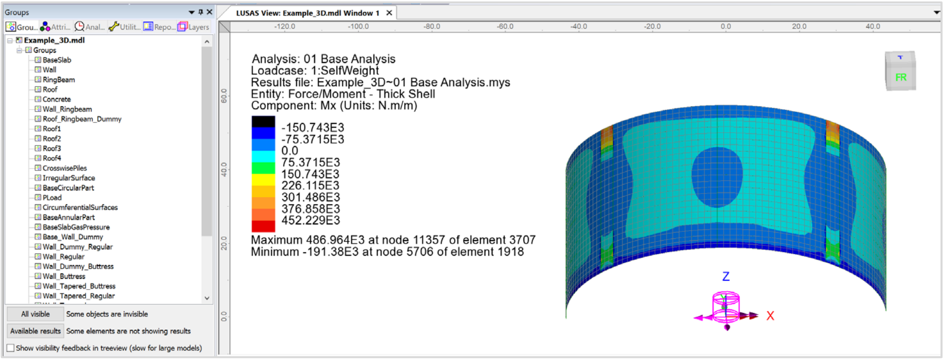


Fig 178 Mx Contour for Self Weight in a 3D Shell Model

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.

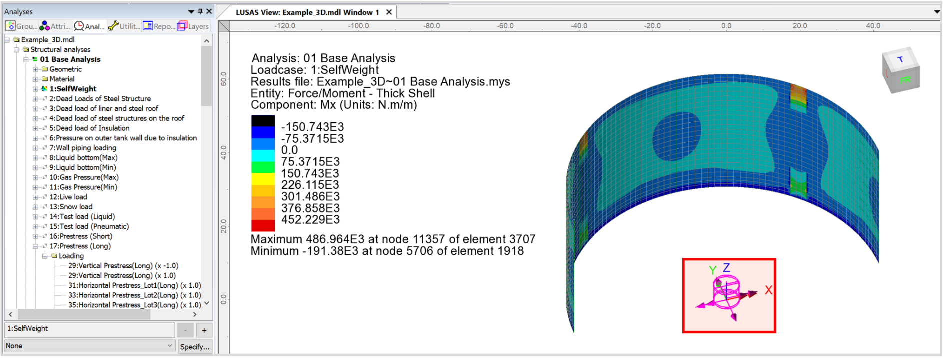


Fig 179 Local Coordinate in a 3D Shell Model

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.

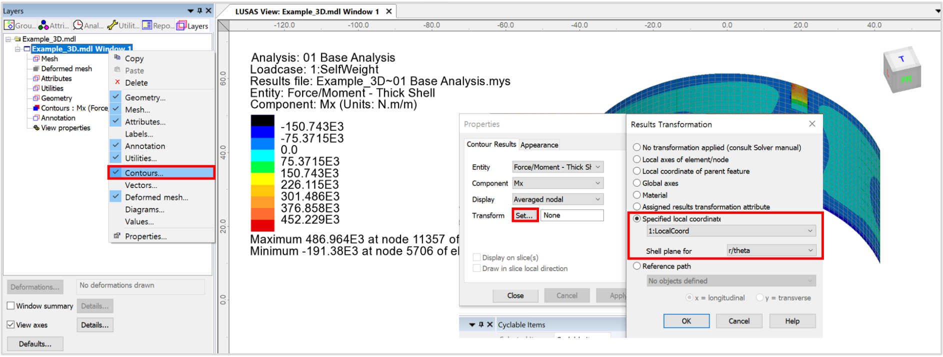


Fig 180 Contour Display using Local Coordinate in a 3D Shell Model

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.

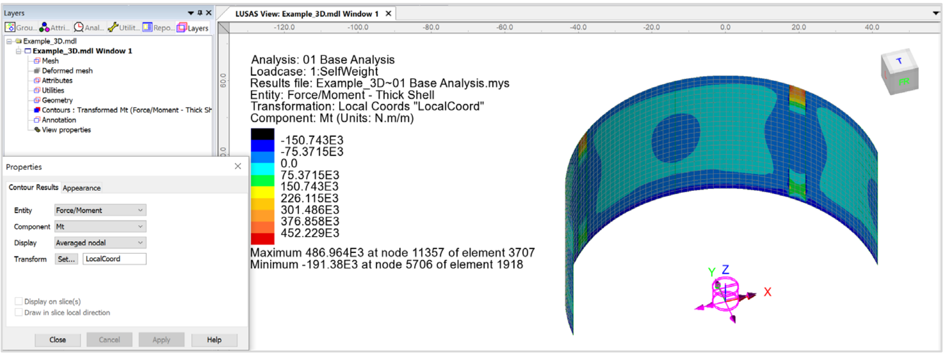
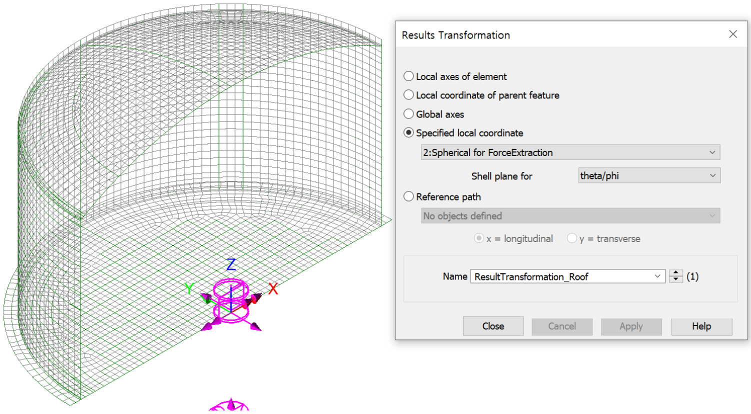
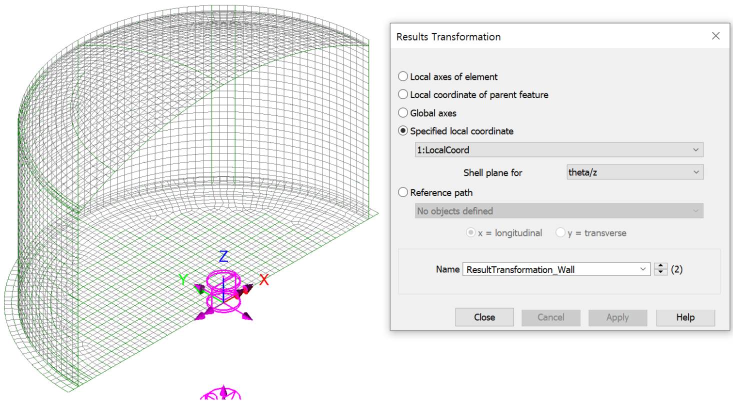
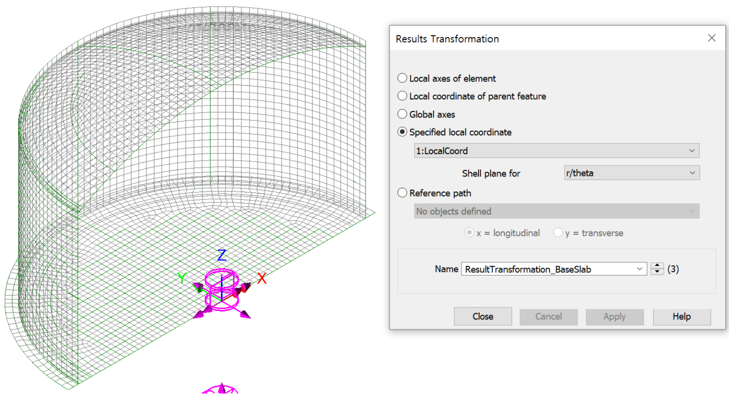


Fig 181 Mt Contour in a 3D Shell Model

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


Z
Y
X
Results Transformation
Local axes of element
Local coordinate of parent feature
Global axes
Specified local coordinate
2:Spherical for ForceExtraction
Shell plane for
theta/phi
Reference path
No objects defined
x = longitudinal
y = transverse
Name
ResultTransformation_Roof
(1)
Close
Cancel
Apply
Help


Z
Y
X
Results Transformation
Local axes of element
Local coordinate of parent feature
Global axes
Specified local coordinate
1:LocalCoord
Shell plane for
theta/z
Reference path
No objects defined
x = longitudinal
y = transverse
Name
ResultTransformation_Wall
(2)
Close
Cancel
Apply
Help


Z
Y
X
Results Transformation
Local axes of element
Local coordinate of parent feature
Global axes
Specified local coordinate
1:LocalCoord
Shell plane for
r/theta
Reference path
No objects defined
x = longitudinal
y = transverse
Name
ResultTransformation_BaseSlab
(3)
Close
Cancel
Apply
Help

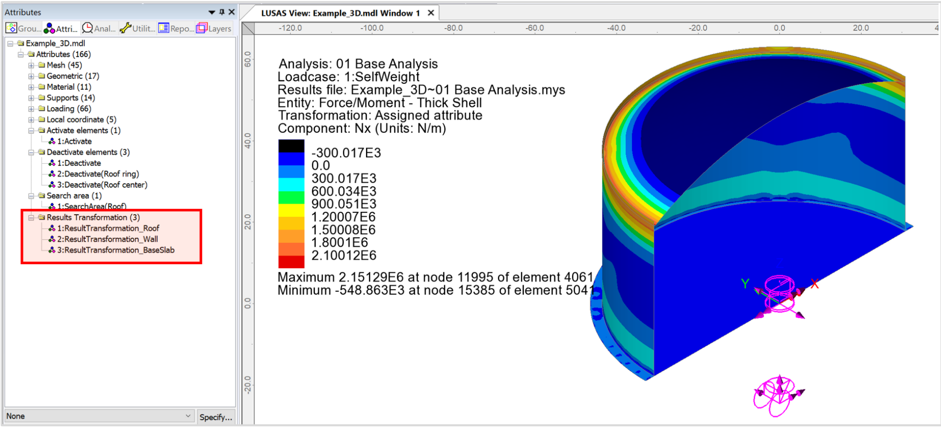


Fig 182 Results Transformation in a 3D Shell Model

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

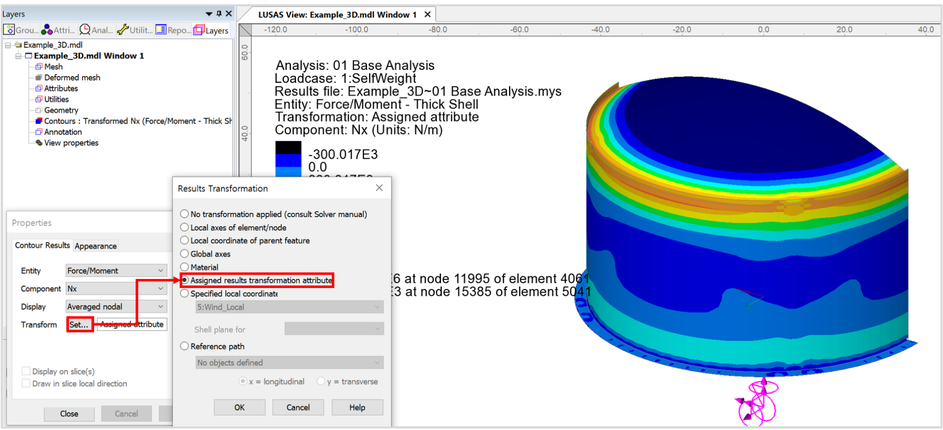


Fig 183 Contour Display using Results Transformation in a 3D Shell Model

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.

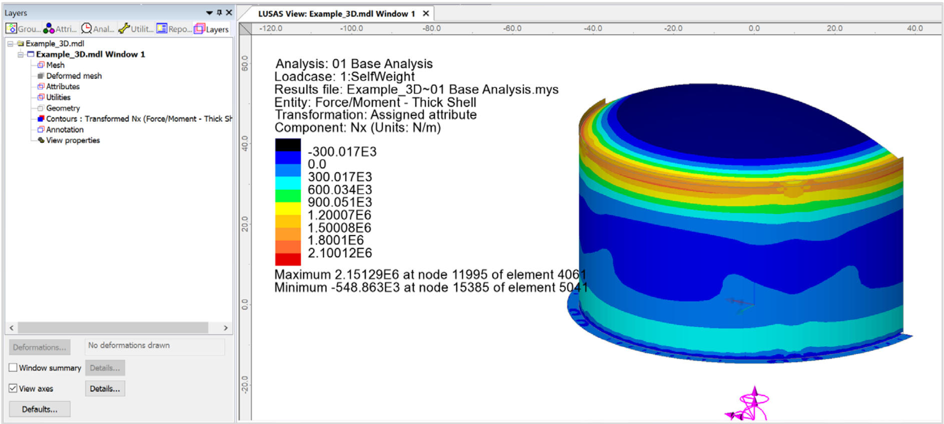


Fig 184 Nx Contours in a 3D Shell Model

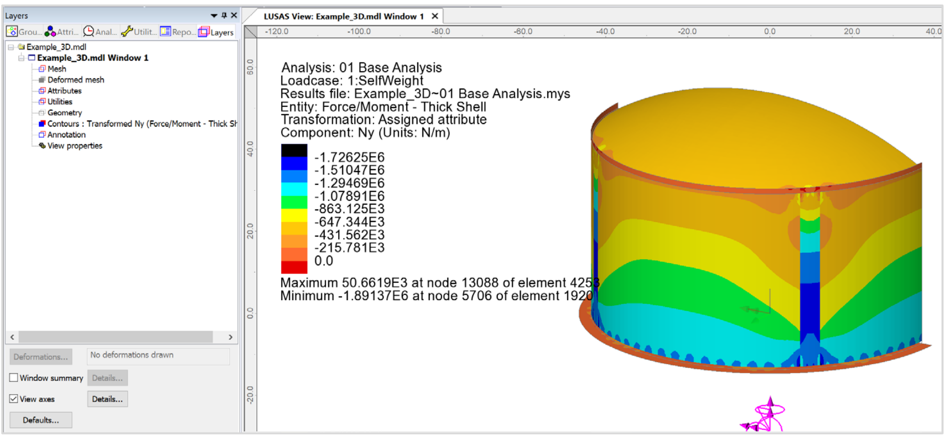


Fig 185 Ny Contours in a 3D Shell Model

Values

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

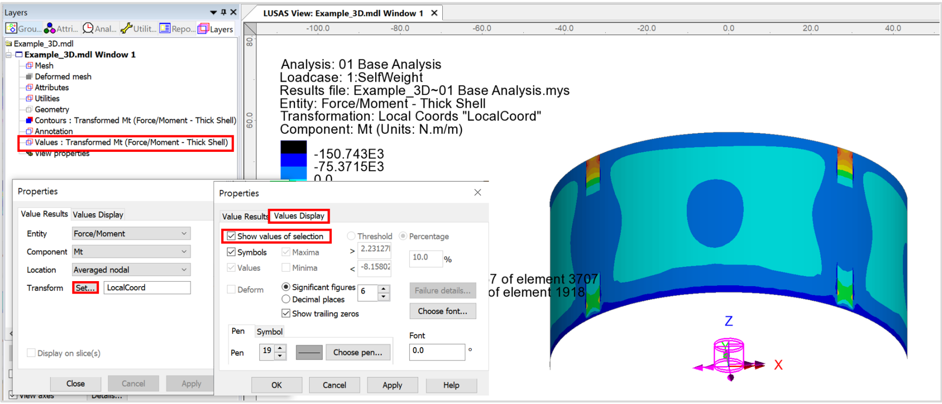


Fig 186 Value Display in a 3D Shell Model

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

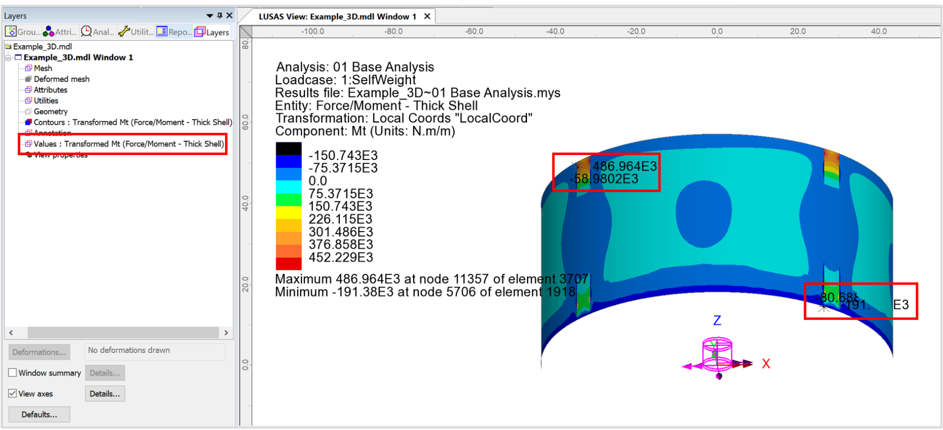


Fig 187 Values Displayed for Selected Nodes in a 3D Shell Model

Graph through 2D

Define a line from **Geometry>Line>By Coords…**

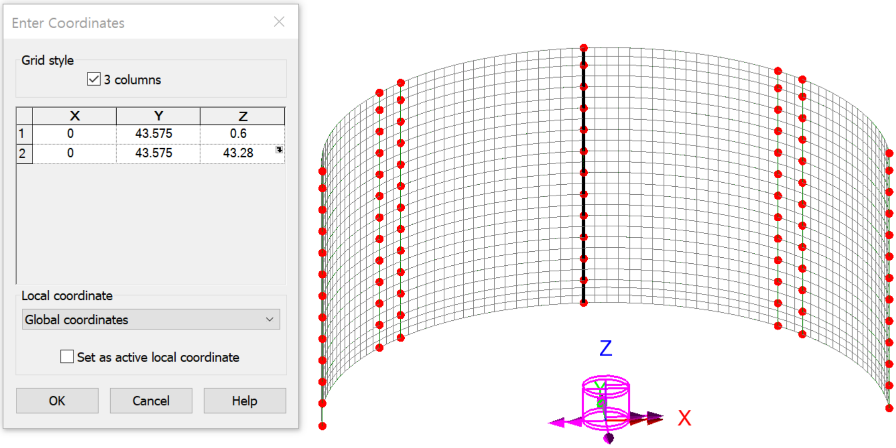


Fig 188 Line for Slicing Results in a 3D Shell Model

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

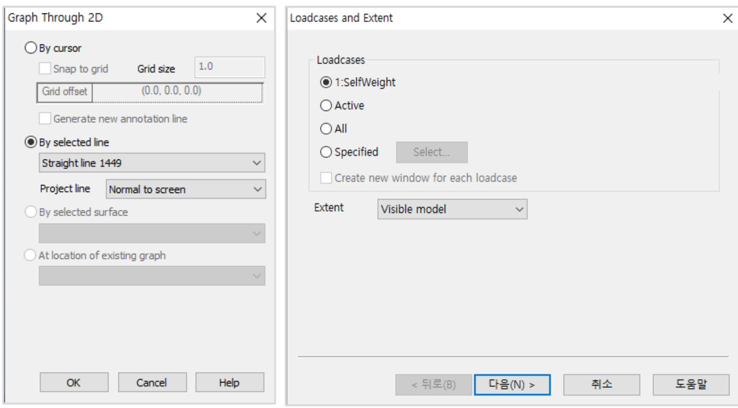


Fig 189 Graph Through 2D in a 3D Shell Model (1)

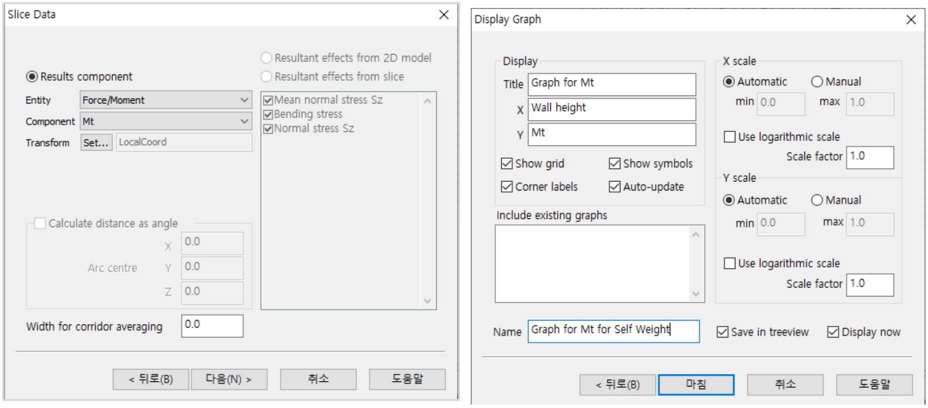


Fig 190 Graph Through 2D in a 3D Shell Model (2)

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.

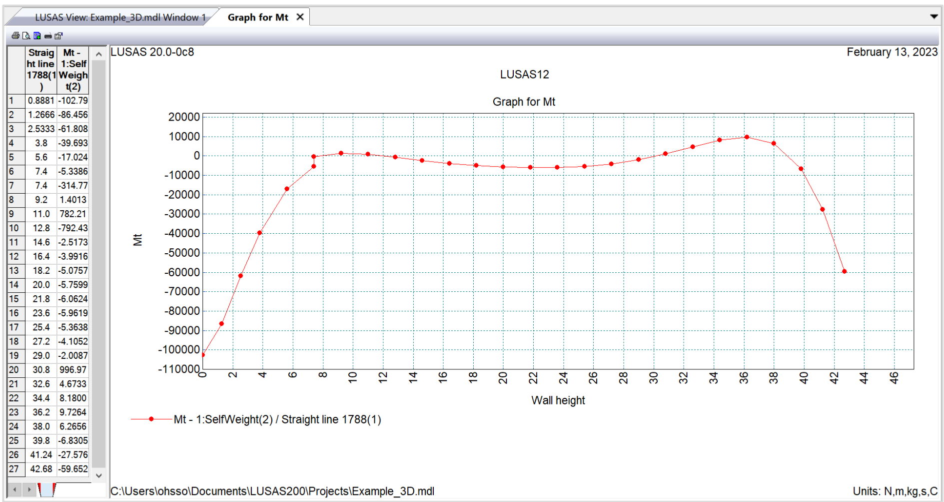


Fig 191 Mt Graph for Sliced Line in a 3D Shell Model

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.

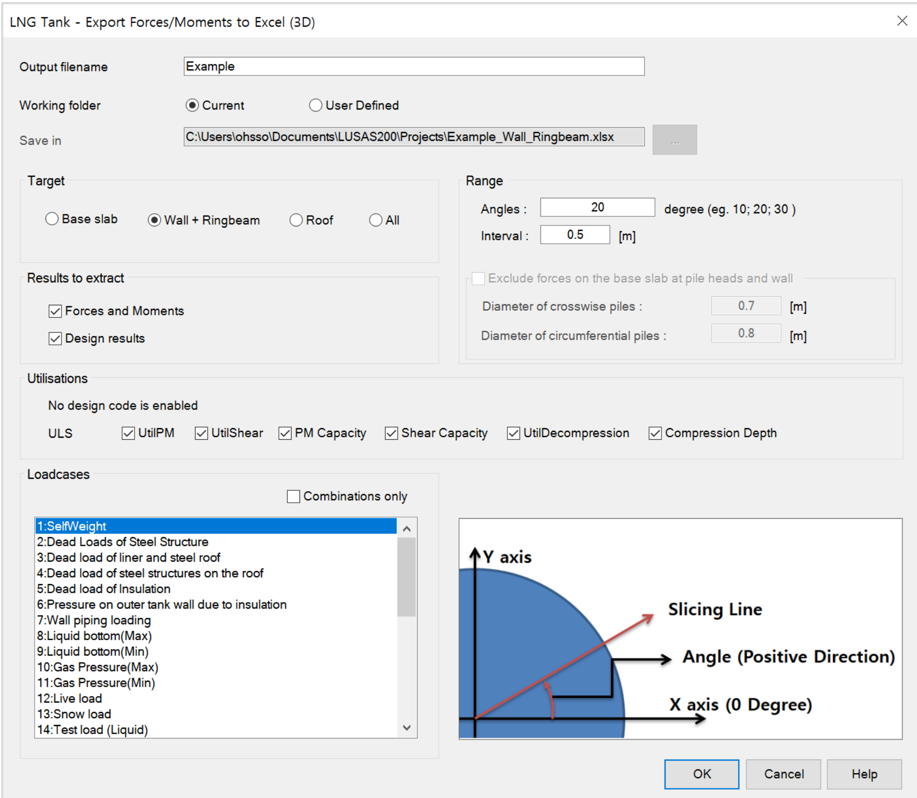


Fig 192 Export Forces for 3D Shell Model (1)

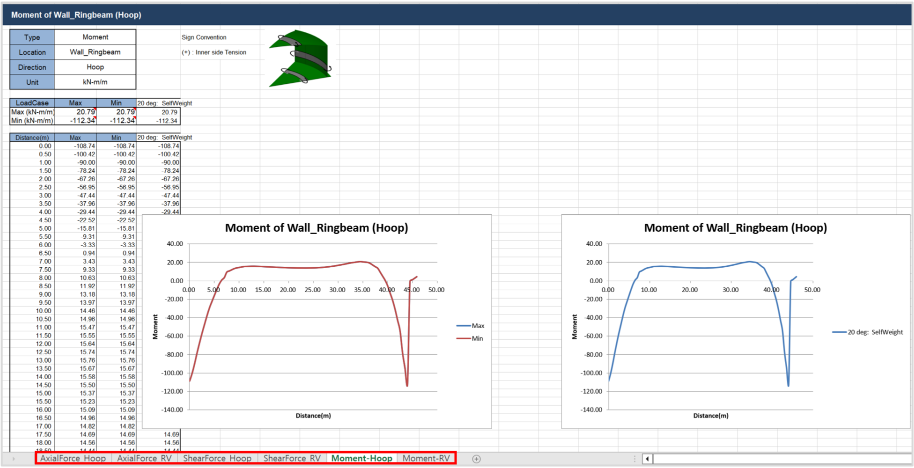


Fig 193 Section Force Spreadsheet for Self Weight

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.

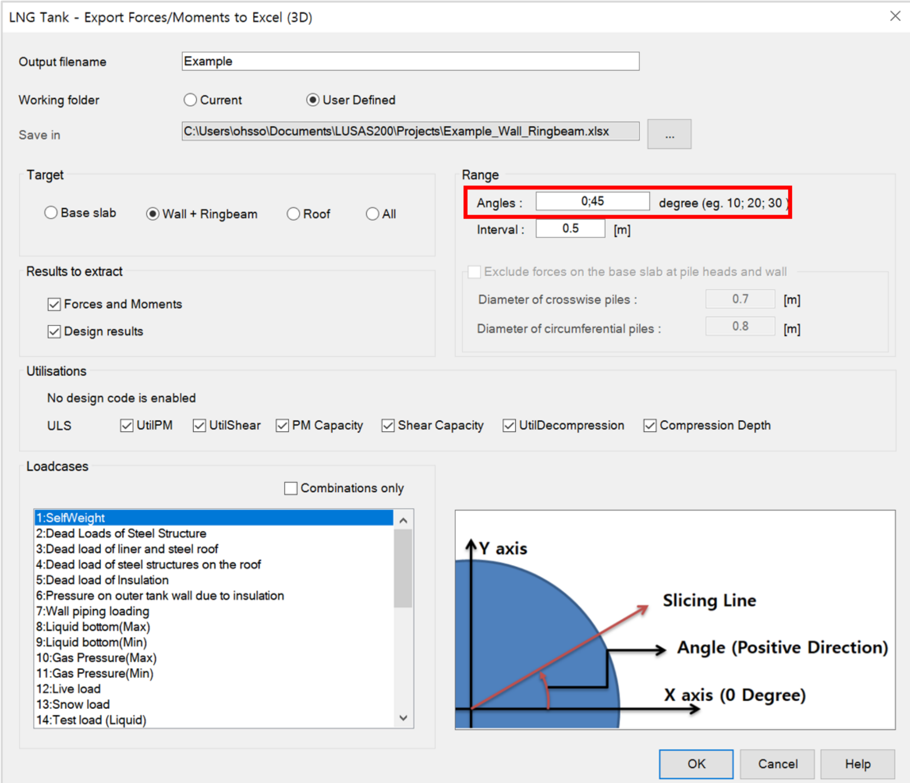


Fig 194 Export Forces for a 3D Shell Model (2)

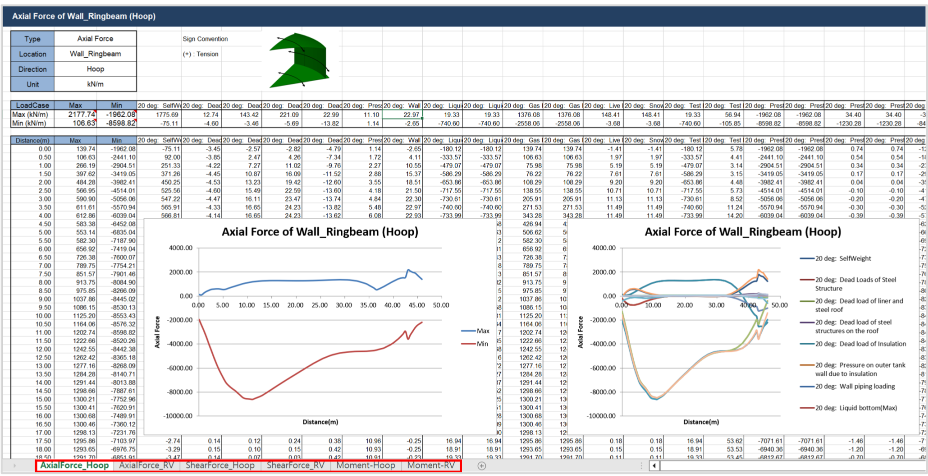


Fig 195 Section Force Spreadsheet for All Loadcases

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

Sign convention

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

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

3D Shell Eigenvalue Analysis

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

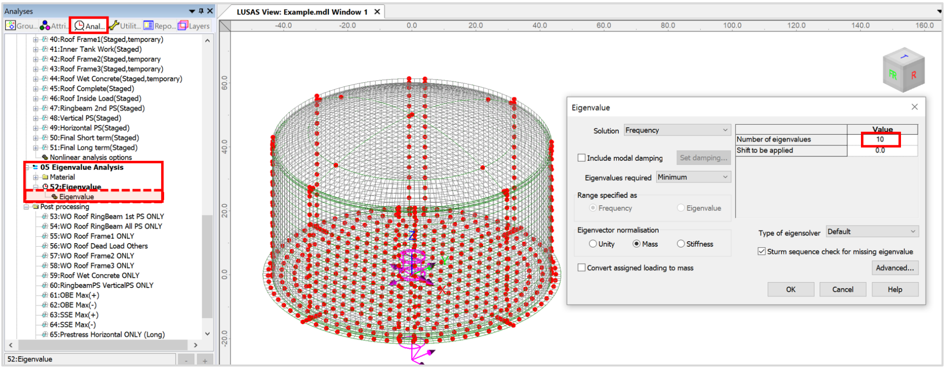


Fig 196 Eigenvalue Analysis in a 3D Shell Model

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.**

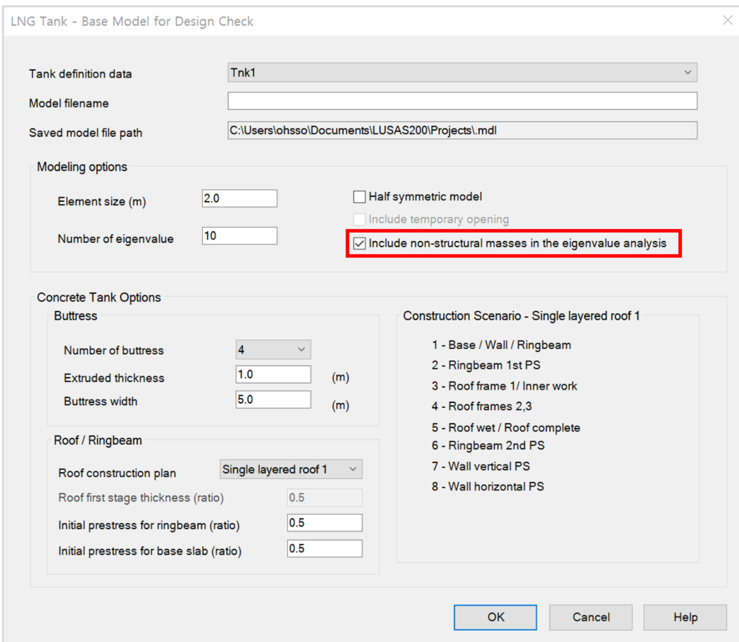


Fig 197 Dialog for a 3D Shell Eigenvalue Analysis

Summary of Mass Calculation

DIMENSION

Component	Dimension(m)
Inner Tank Radius	42.1
Tank Height	40.06
LNG Height	38.92

SUMMARY FOR MASS

Component	Volume	Unit mass	Structural mass	Total mass	Equivalent unit mass
	m^3	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.

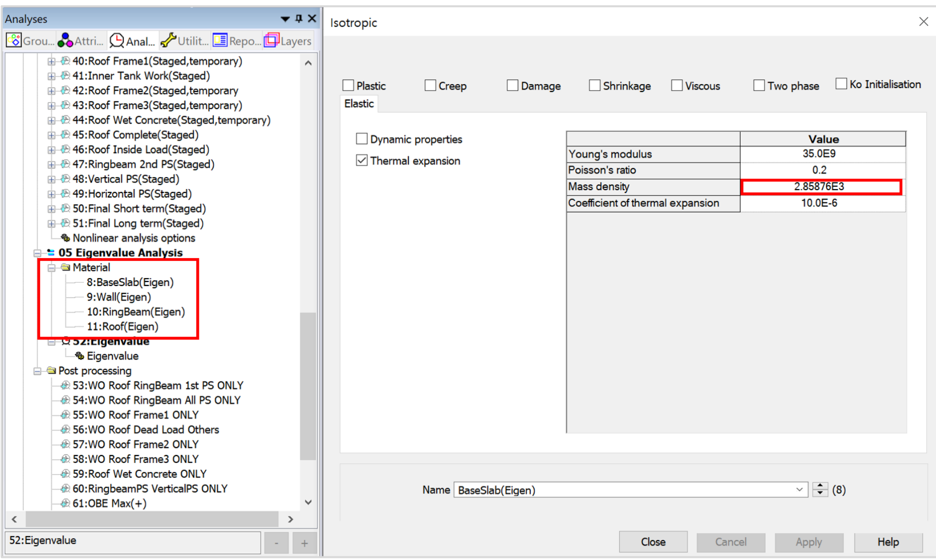


Fig 199 Mass for Eigenvalue Analysis

Viewing Results

Mode Shapes

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

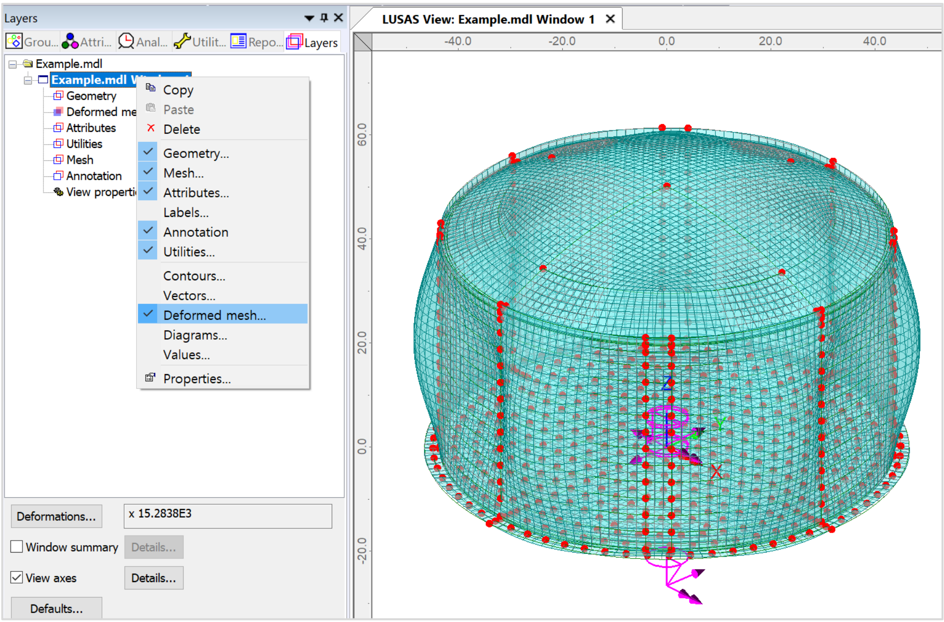
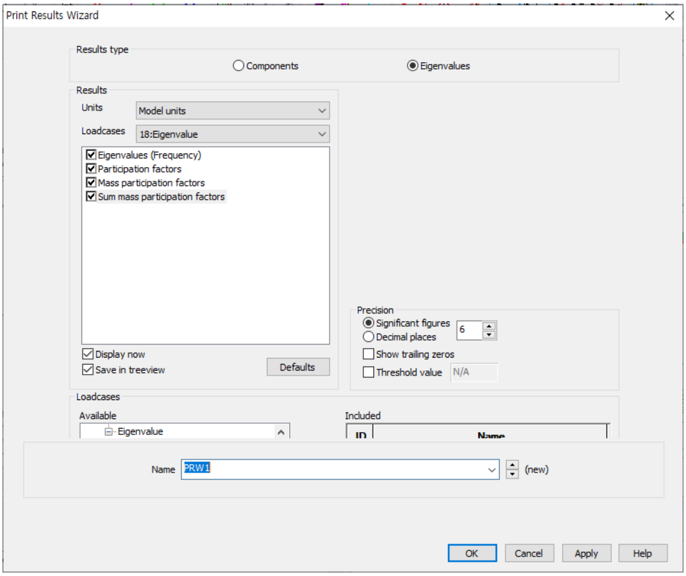


Fig 200 Mode Shape from an Eigenvalue Analysis

Natural Frequencies

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



LUSAS View: Example.mdl Window 1 | **Sum mass participation factors**

	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.

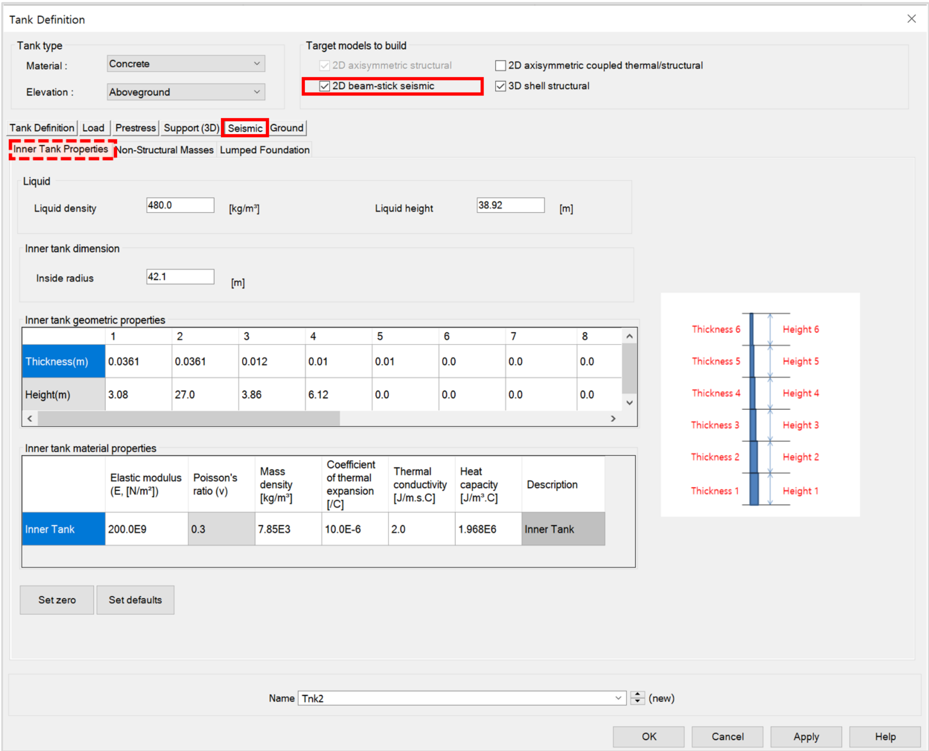


Fig 202 User Inputs 1 for Seismic Analysis

Non-Structural Masses

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

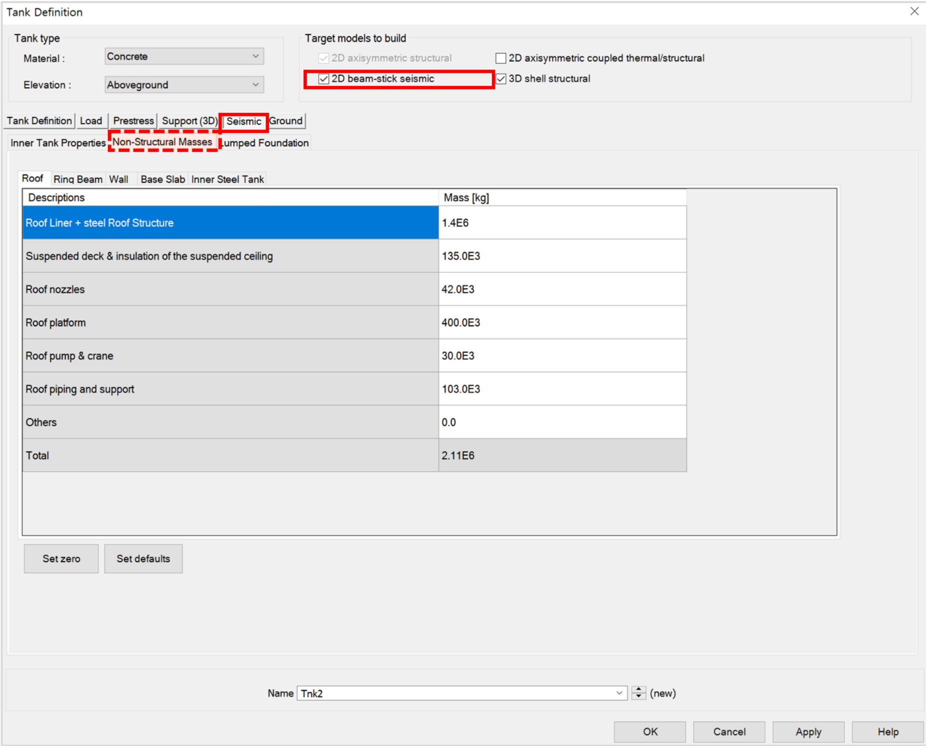


Fig 203 User Inputs 2 for Seismic Analysis

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.

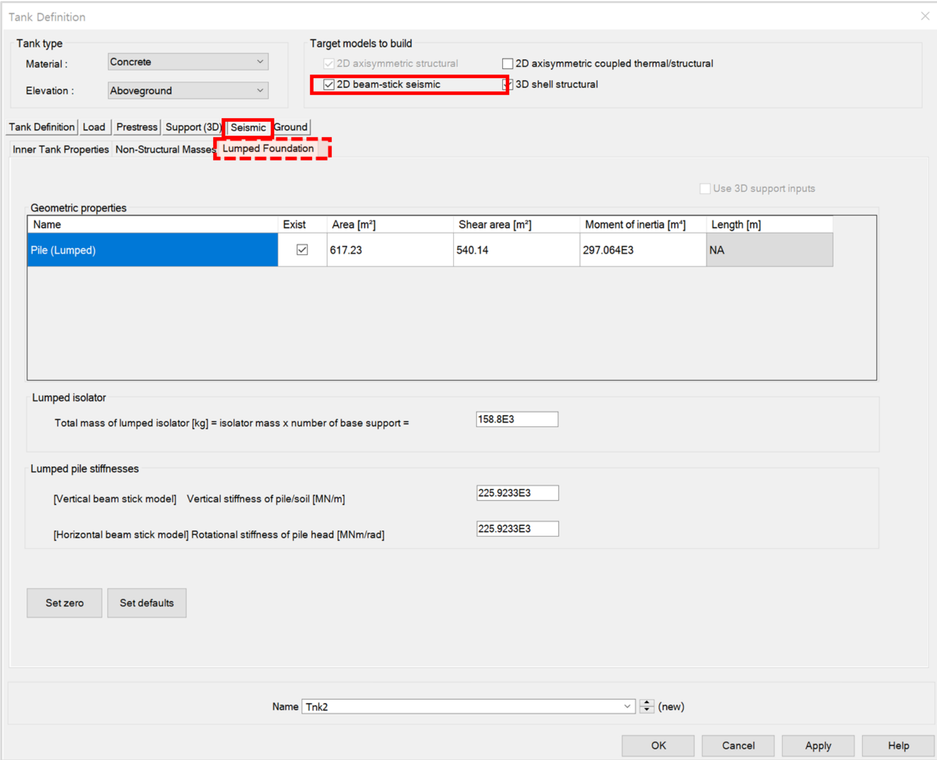


Fig 204 User Inputs 3 for Seismic Analysis

Soil Properties

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

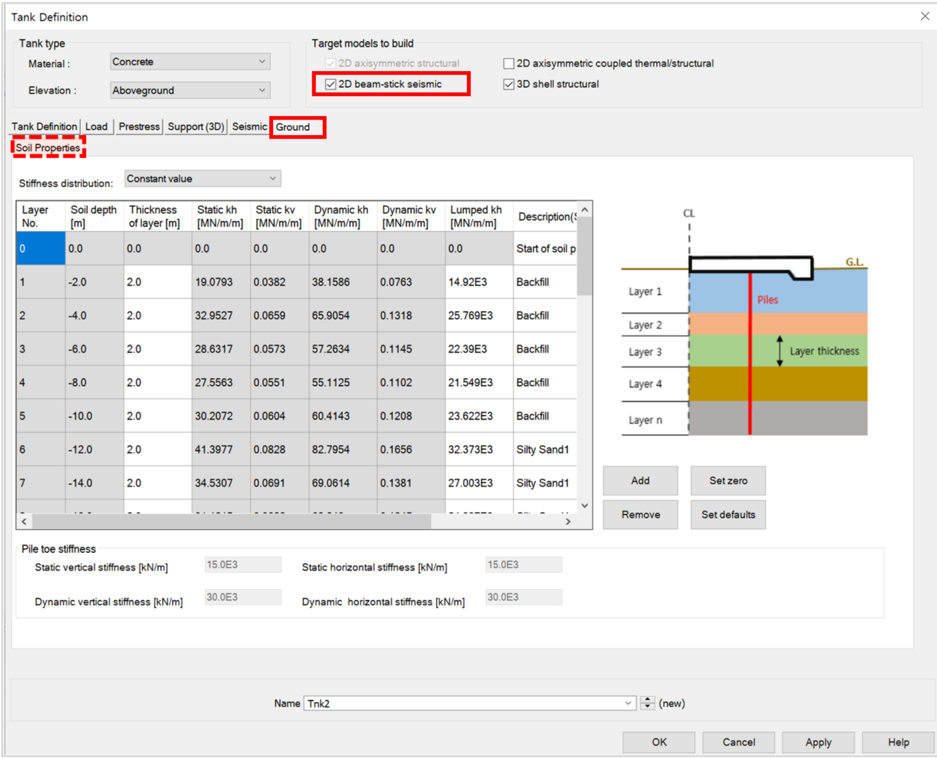


Fig 205 User Inputs 4 for Seismic Analysis

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.

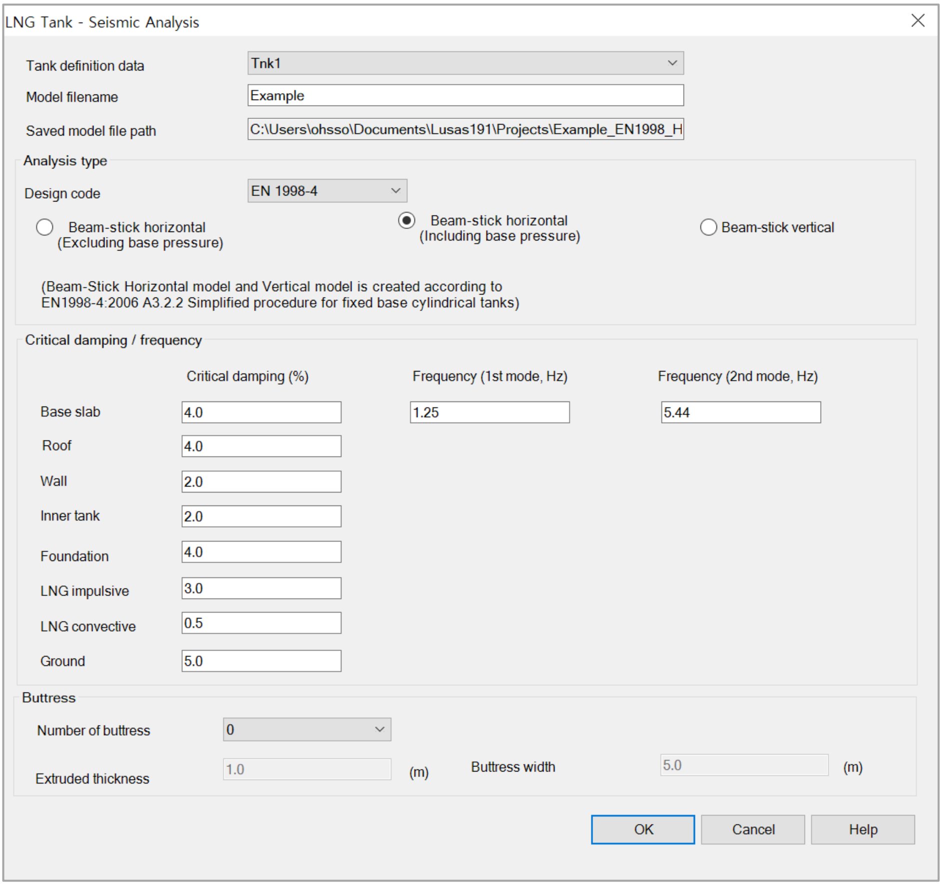


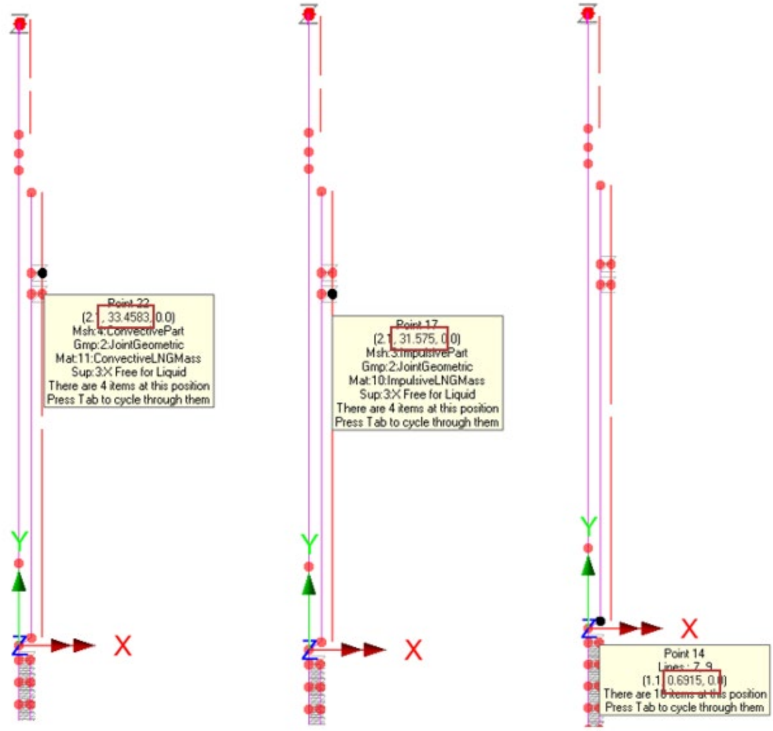
Fig 206 User Dialog for Seismic Analysis Wizard

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 PROPERTIES FOR HORIZONTAL MODEL

1) LNG Mass & Height

IBP (Including Base Pressure)

Component	H/R	$m_{(c,i)}/m$	$h'_{(c,i)}/H$	mass mc(mi), Kg	Lever arm height 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 mc(mi), Kg	Lever arm height 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-ACI350.3 | Verifications-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.

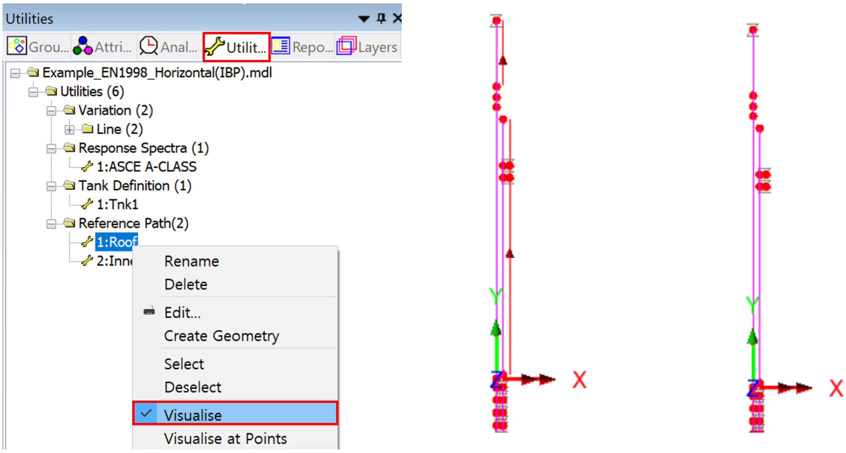


Fig 208 Hide reference path in Beam-Stick Model

Geometric Properties

Roof

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

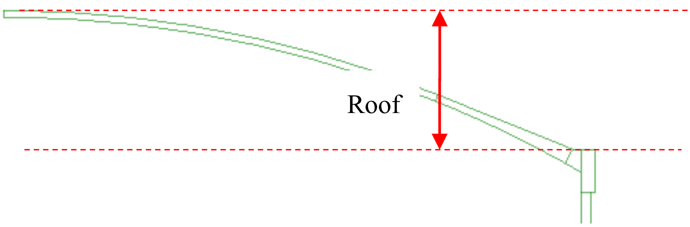


Fig 209 Roof in Beam-Stick Model

Varying Section properties are defined as shown below.

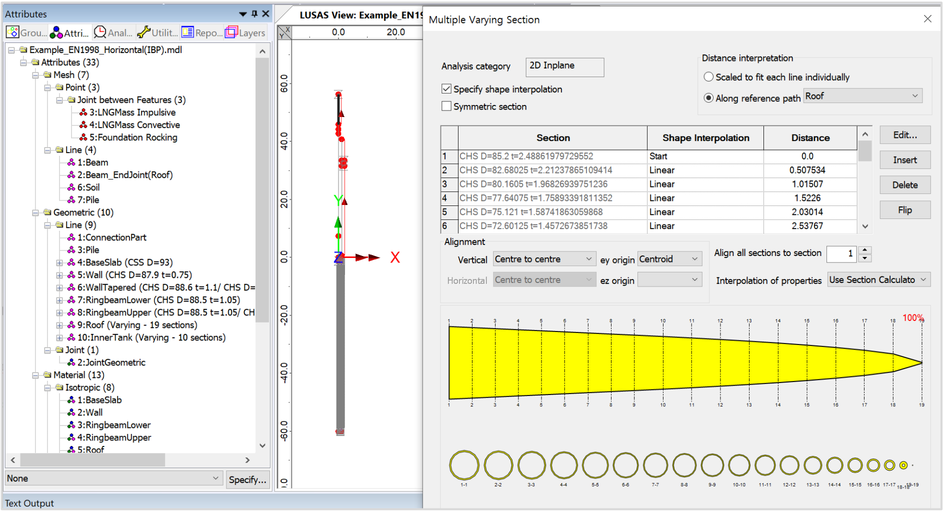


Fig 210 Geometric Properties for the Roof in a Beam-Stick Model

RingBeam Upper

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

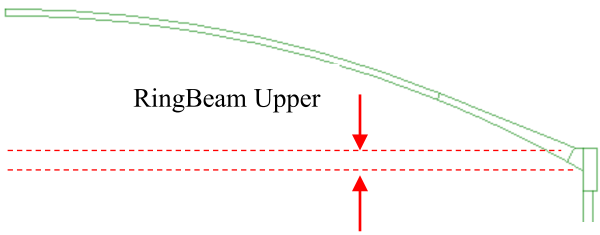


Fig 211 RingBeam Upper in Beam-Stick Model

Varying Section properties are defined as shown below.

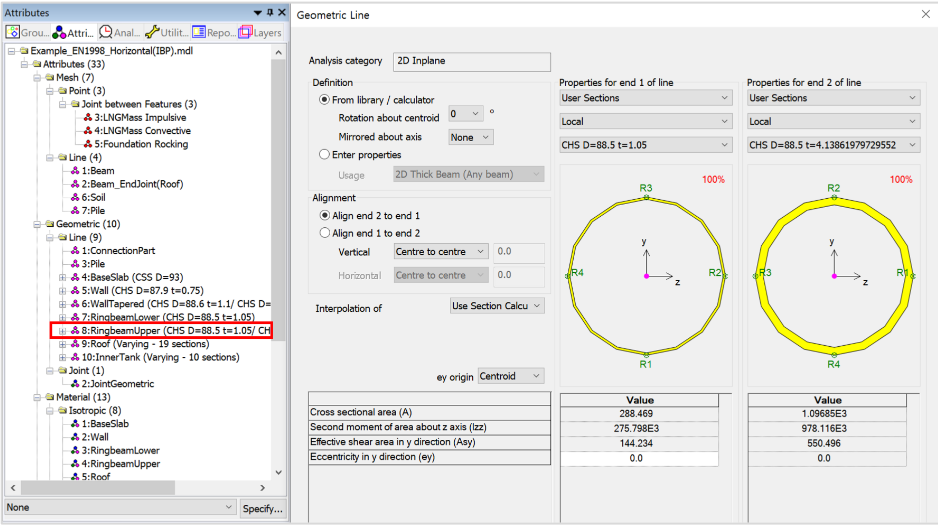


Fig 212 Geometric Properties for RingBeam Upper in a Beam-Stick Model

RingBeam Lower

The elements for RingBeam Lower represent the region shown below.

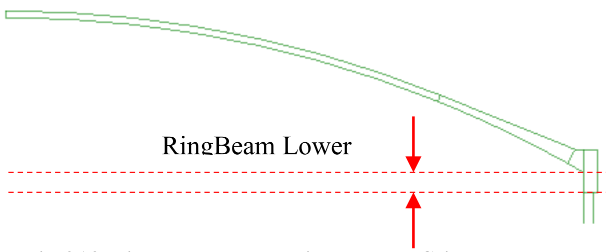


Fig 213 RingBeam Lower in a Beam-Stick Model

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**.

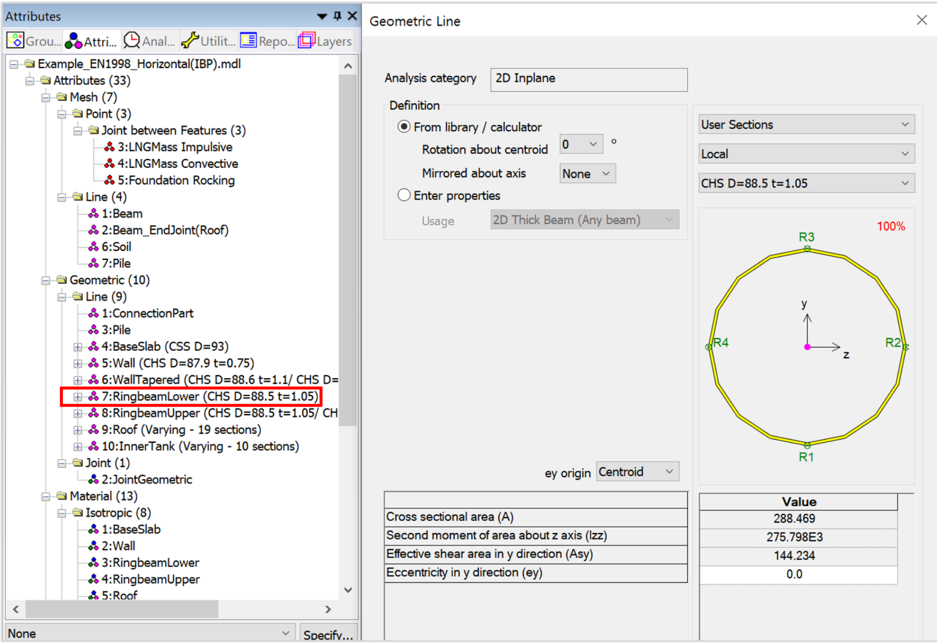


Fig 214 Geometric Properties for RingBeam Lower in a Beam-Stick Model

Wall

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

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

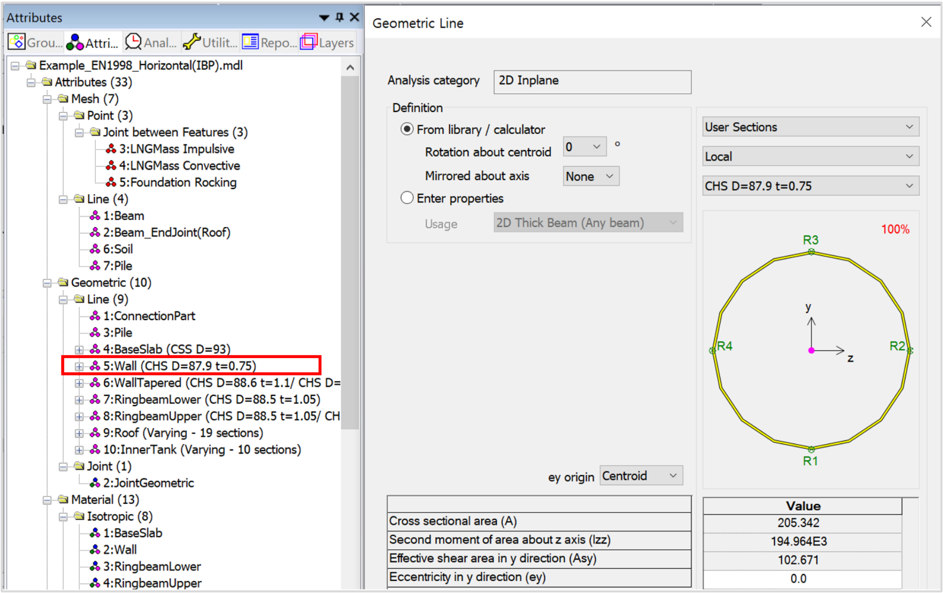


Fig 215 Geometric Properties for Wall in a Beam-Stick Model

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.

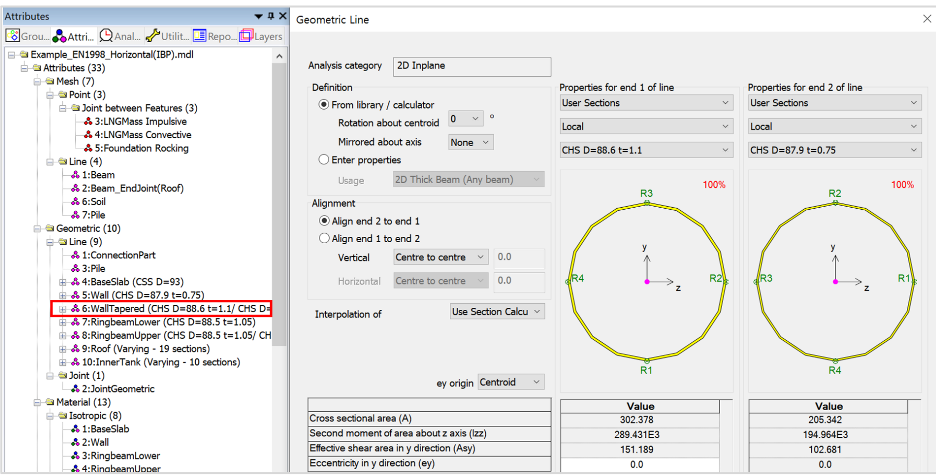
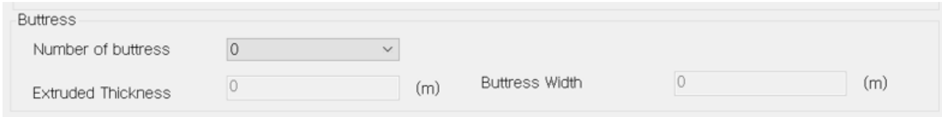


Fig 216 Geometric Properties for Wall Tapered in Beam-Stick Model

Buttresses

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



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

BaseSlab

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

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

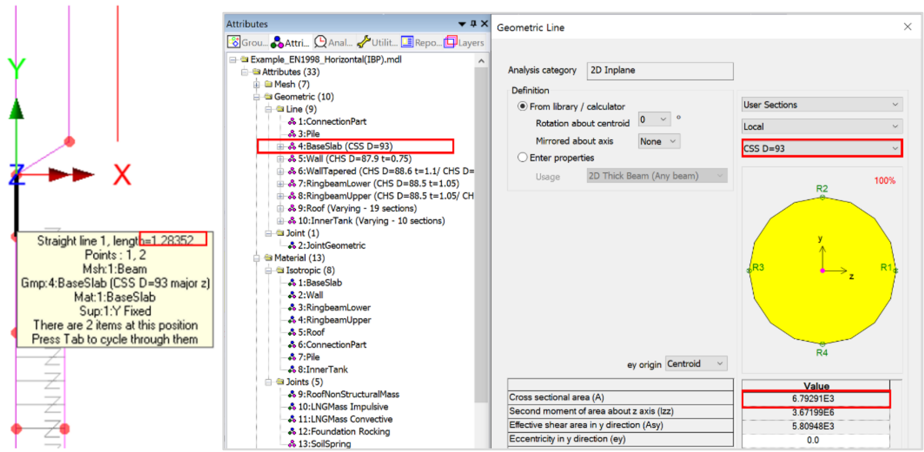


Fig 217 Section Properties for Base Slab in a Beam-Stick Model

Pile

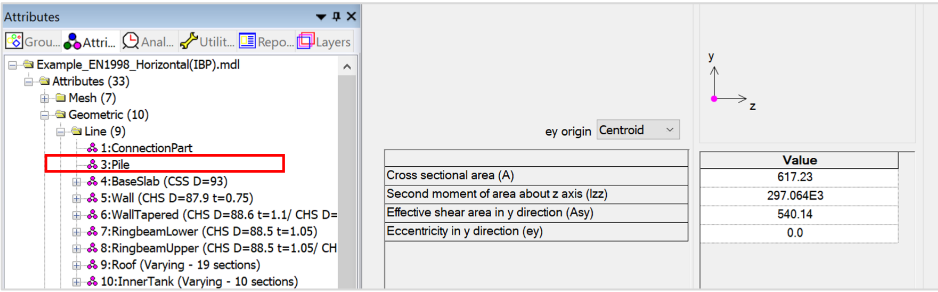


Fig 218 Section Properties for Pile in a Beam-Stick Model

Inner Tank

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

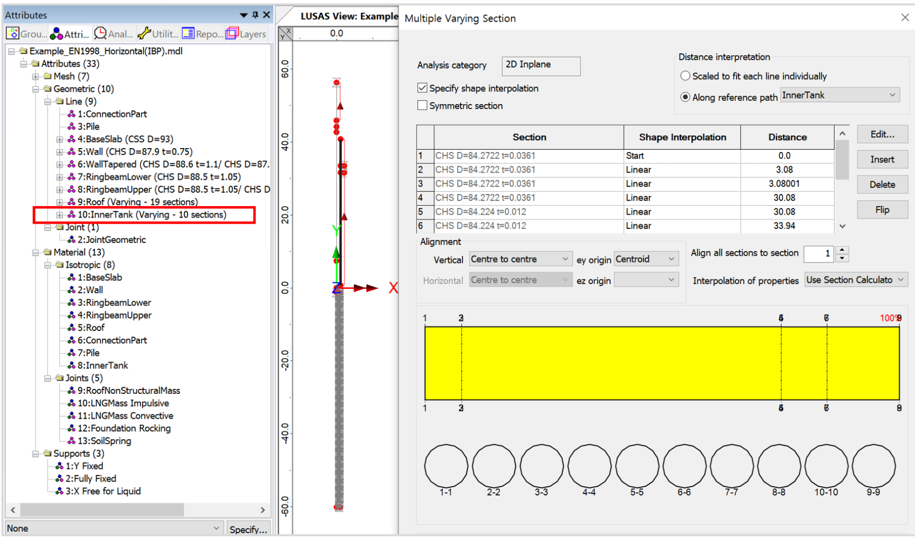


Fig 219 Section Properties for 'Inner Tank' in a Beam-Stick Model

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.

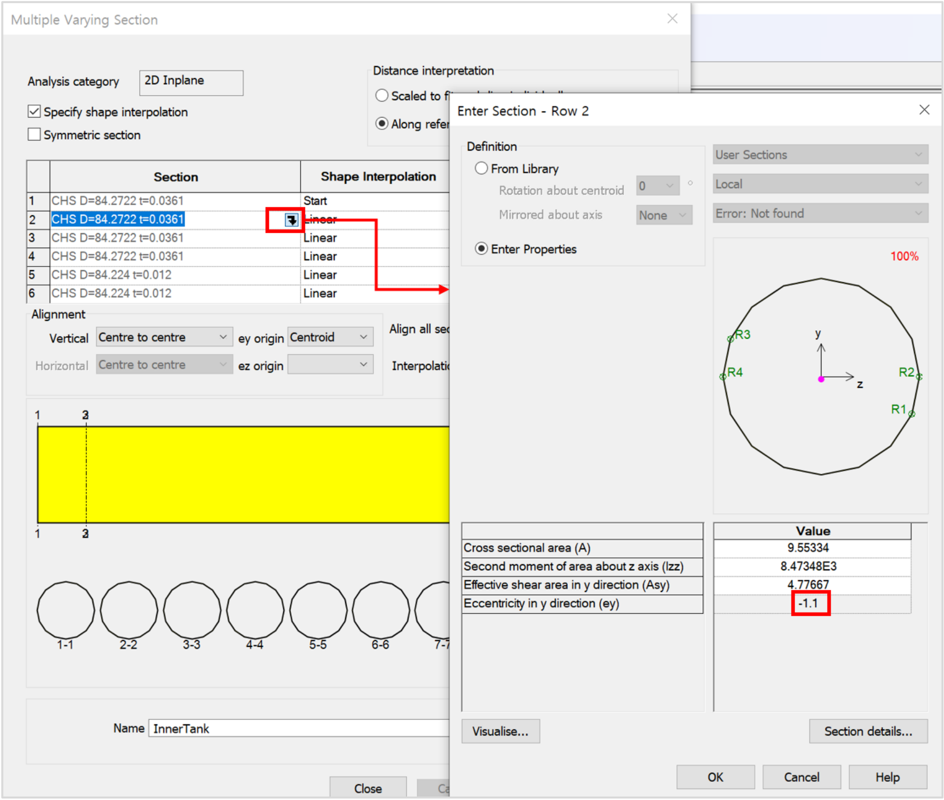


Fig 220 Section Properties for Inner Tank in a Beam-Stick Model

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.

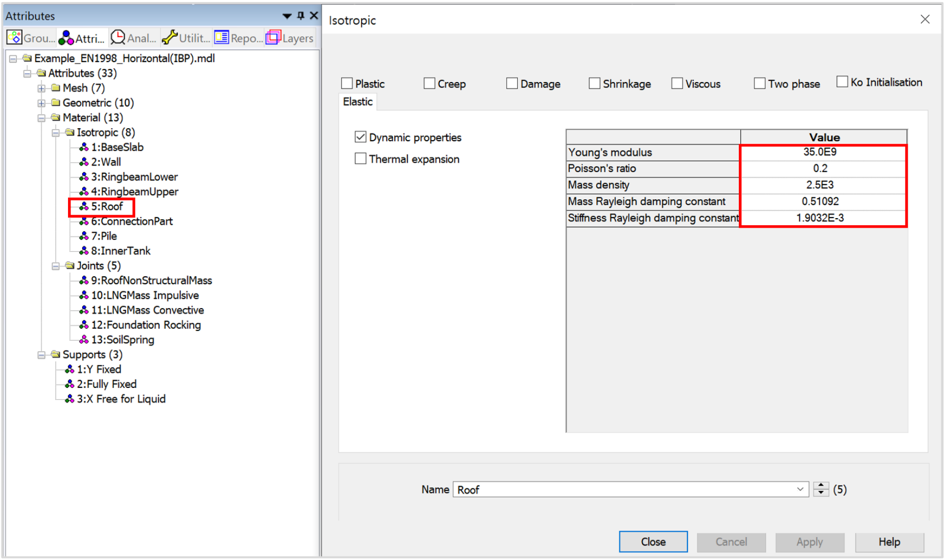


Fig 221 Material Properties for Roof in Beam-Stick Model

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].)

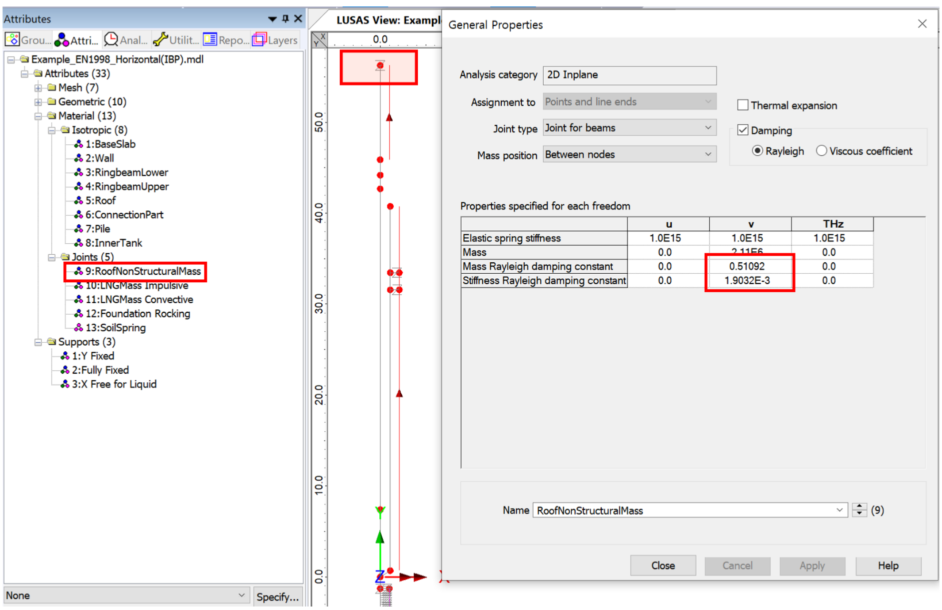


Fig 222 Non-structural mass on the Roof in a Beam-Stick Model

RingBeam Upper / RingBeam Lower

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

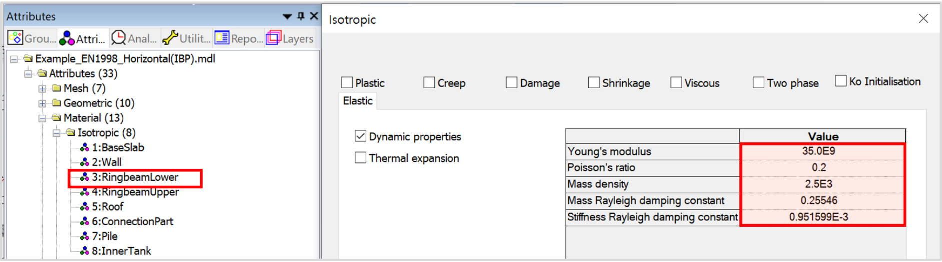


Fig 223 Material Properties for Ringbeam in a Beam-Stick Model

Wall

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

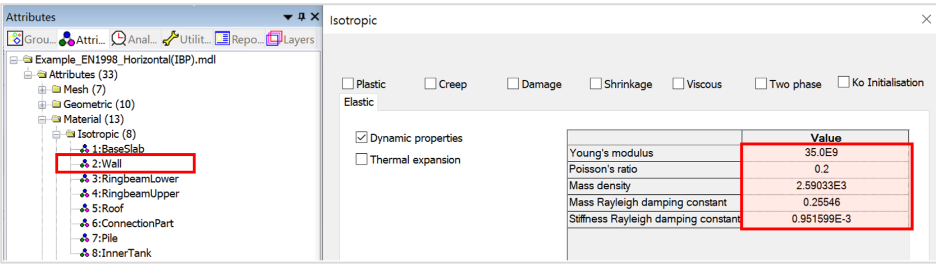


Fig 224 Material Properties for 'Wall' in a Beam-Stick Model

Base slab

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

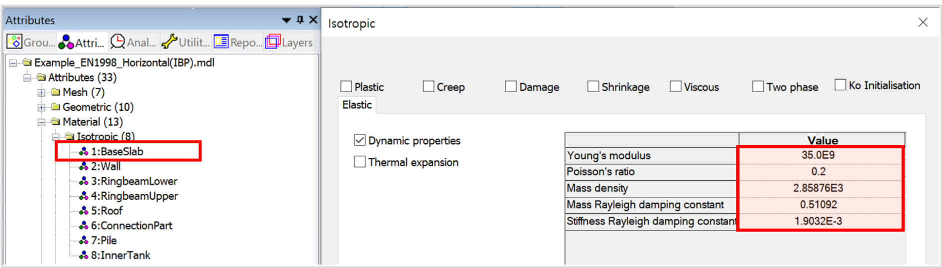
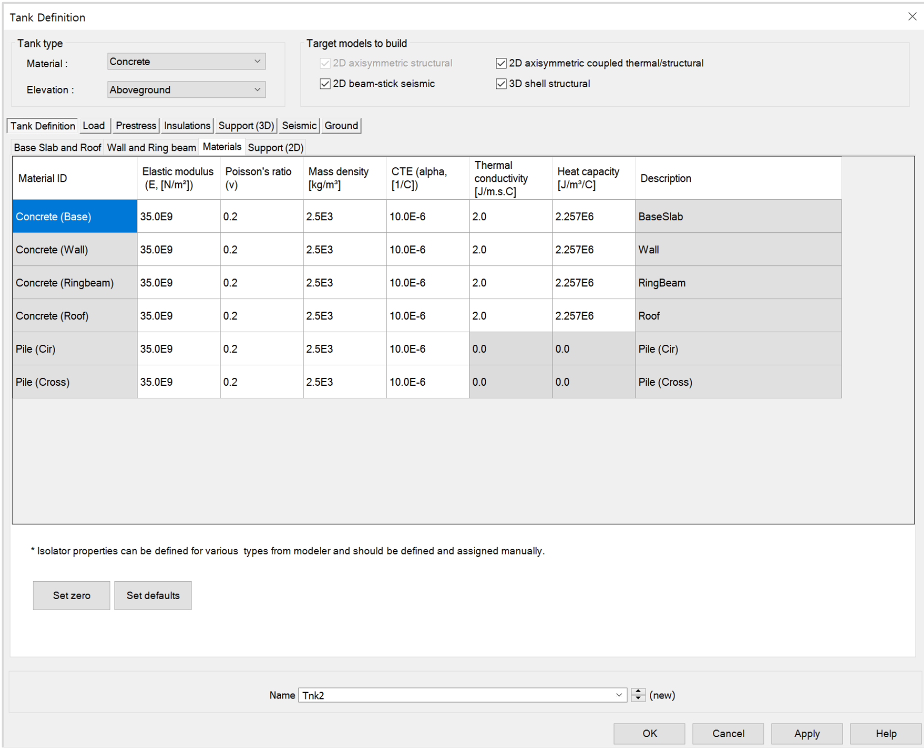


Fig 225 Material Properties for BaseSlab in Beam-Stick Model

Pile & Soil

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



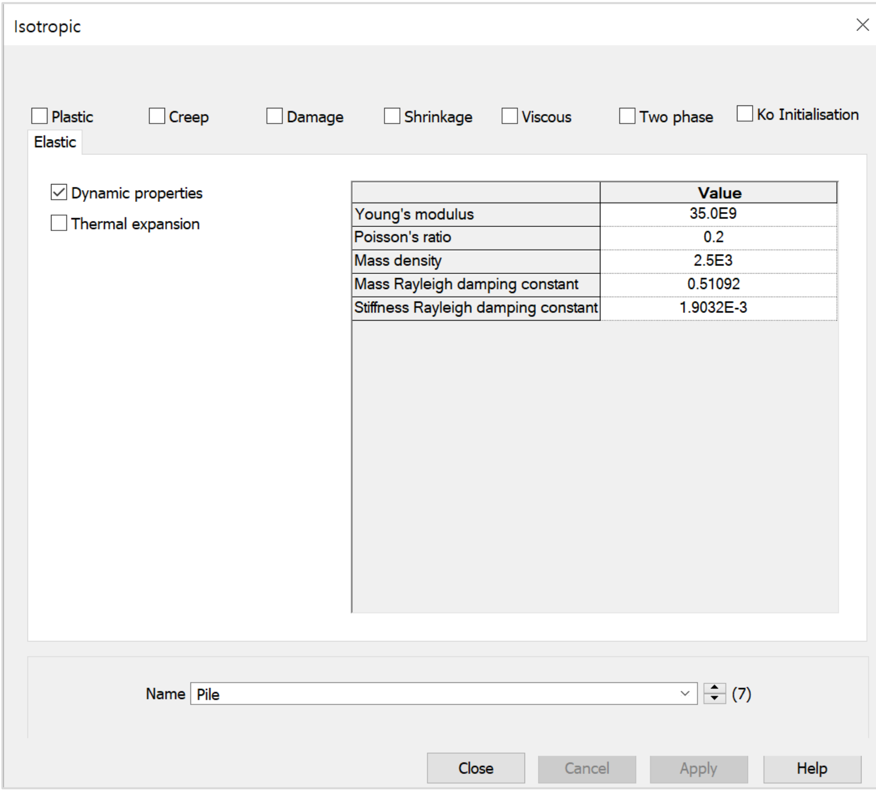
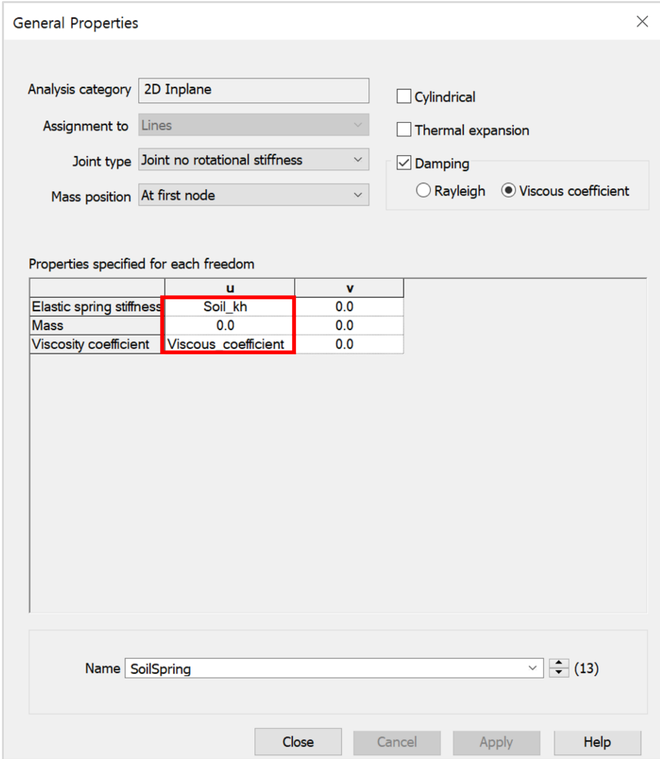


Fig 226 Material Properties for 'Pile' in a Beam-Stick Model


General Properties
Analysis category
2D Inplane
Assignment to
Lines
Joint type
Joint no rotational stiffness
Mass position
At first node
Cylindrical
Thermal expansion
Damping
Rayleigh
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
(13)
Close
Cancel
Apply
Help

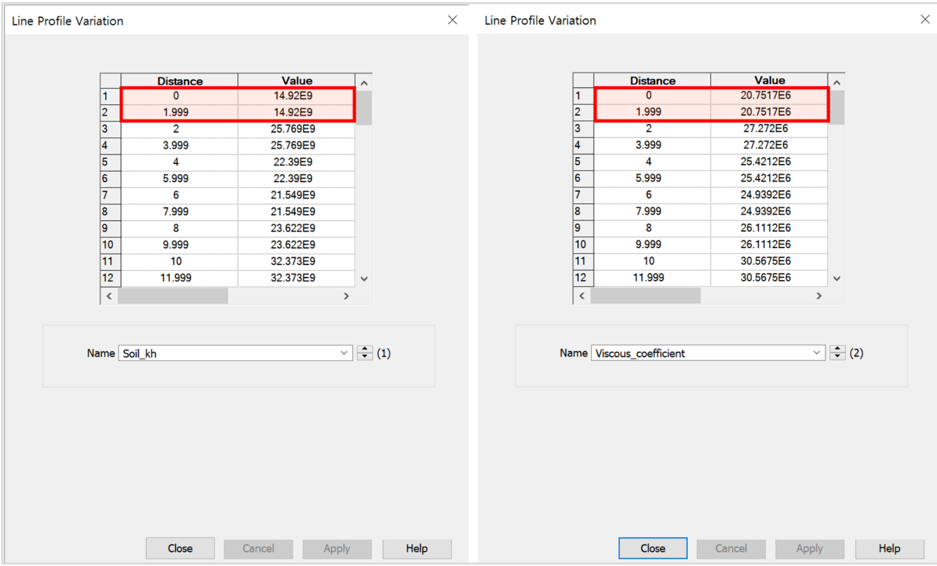


Fig 227 Material Properties for Soil Boundary in a Beam-Stick Model

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 Impulsive Mass			
Component	Value	Unit	Remark
H/R	0.92447		LNG height divided by inner tank radius
ρ_l	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
C_i	6.51359		coefficients for determining the fundamental frequency
T_{imp}	0.43182	s	fundamental period of oscillation of the tank (plus the in
ki	11,325,839,357	N/m	

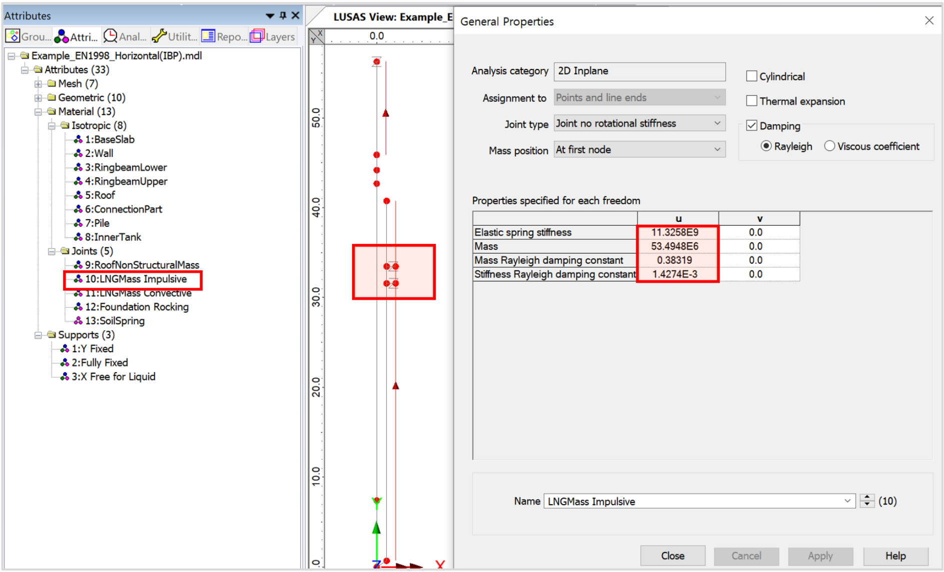


Fig 228 Material Properties for Impulsive liquid mass in Beam-Stick Model

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	

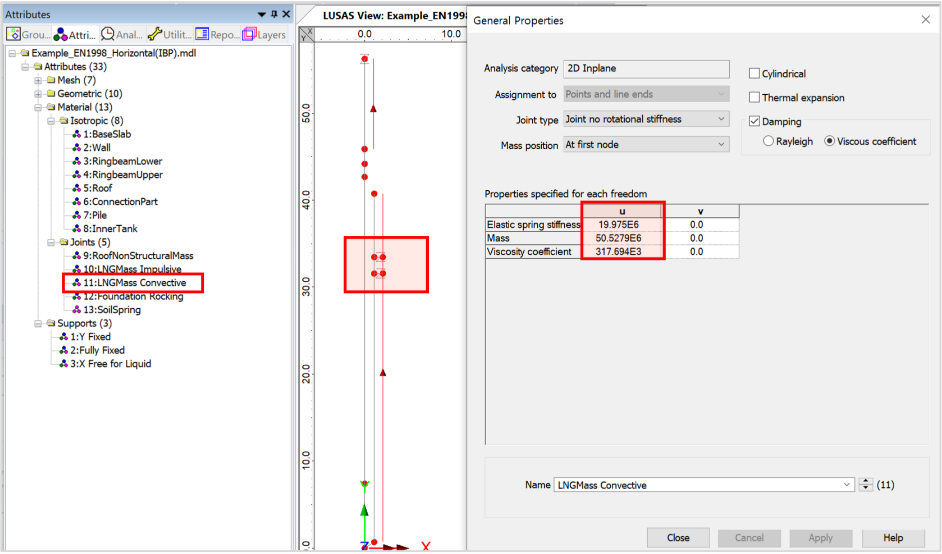


Fig 229 Material Properties for Convective Liquid Mass in a Beam-Stick Model

Inner Tank

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

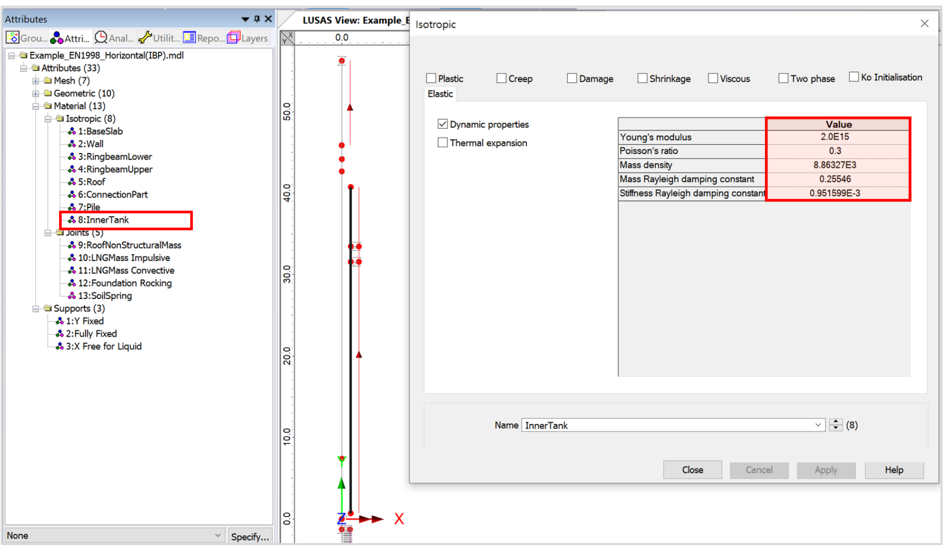


Fig 230 Material Properties for Inner Tank in a Beam-Stick Model

Viewing Results

Mode Shapes

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

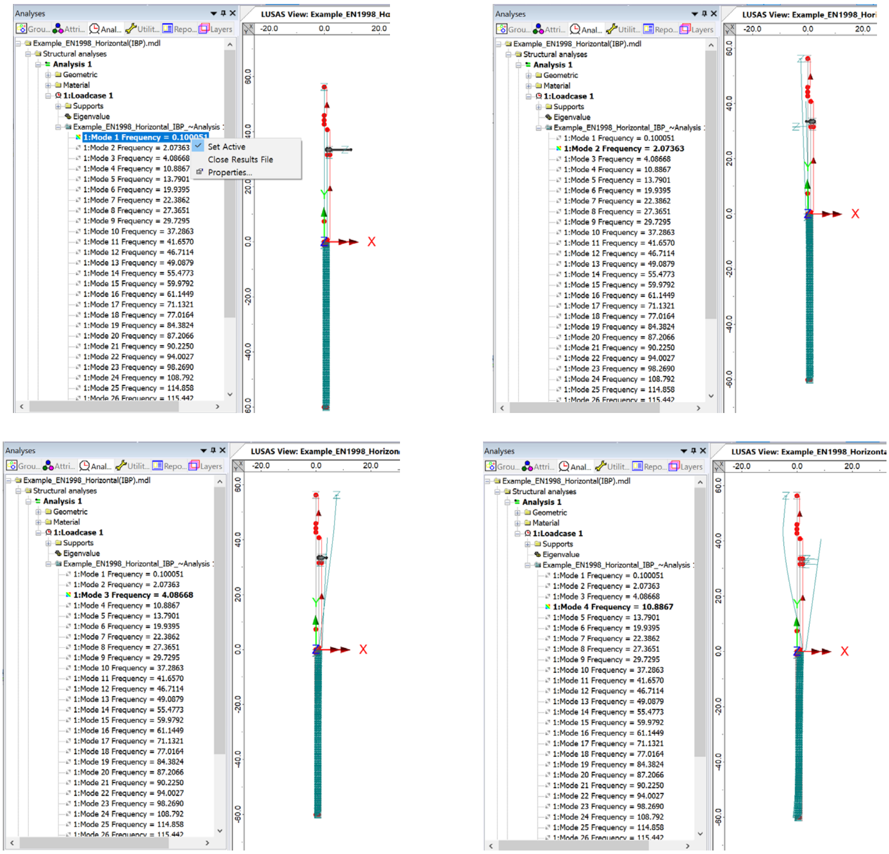
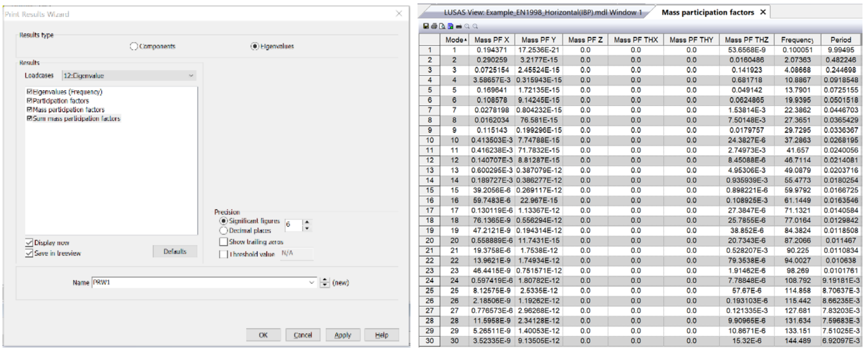


Fig 231 Mode Shapes from Eigenvalue Analysis with a Beam-Stick Model

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 1st mode is for convective liquid mass, and the subsequent modes are mixed modes. From this it would be reasonable to use the 2nd and 3rd frequencies 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.)



	Mode	Mass PF X	Mass PF Y	Mass PF Z	Mass PF THX	Mass PF THY	Mass PF THZ	Frequency	Period
1	1	0.194371	17.2536E-21	0.0	0.0	0.0	53.6568E-9	0.100051	9.99495
2	2	0.290259	3.2177E-15	0.0	0.0	0.0	0.0160486	2.07363	0.482246
3	3	0.0725154	2.45524E-15	0.0	0.0	0.0	0.141923	4.08668	0.244698
4	4	3.58657E-3	0.315943E-15	0.0	0.0	0.0	0.681718	10.8867	0.0918548
5	5	0.169641	1.72135E-15	0.0	0.0	0.0	0.049142	13.7901	0.0725155
6	6	0.108578	9.14245E-15	0.0	0.0	0.0	0.0624865	19.9395	0.0501518
7	7	0.0278198	0.804232E-15	0.0	0.0	0.0	1.53814E-3	22.3862	0.0446703
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
17	17	0.130119E-6	1.13367E-12	0.0	0.0	0.0	27.3847E-6	71.1321	0.0140584
18	18	76.1365E-9	0.556294E-12	0.0	0.0	0.0	25.7855E-6	77.0164	0.0129842
19	19	47.2121E-9	0.194314E-12	0.0	0.0	0.0	38.852E-6	84.3824	0.0118508
20	20	0.558889E-6	11.7431E-15	0.0	0.0	0.0	20.7343E-6	87.2066	0.011467
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
24	24	0.597419E-6	1.80782E-12	0.0	0.0	0.0	7.78848E-6	108.792	9.19181E-3
25	25	8.12575E-9	2.5335E-12	0.0	0.0	0.0	57.67E-6	114.858	8.70637E-3
26	26	2.18506E-9	1.19262E-12	0.0	0.0	0.0	0.193103E-6	115.442	8.66235E-3
27	27	0.776573E-6	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
29	29	5.26511E-9	1.40053E-12	0.0	0.0	0.0	10.8671E-6	133.151	7.51025E-3
30	30	3.52335E-9	9.13505E-12	0.0	0.0	0.0	15.32E-6	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.

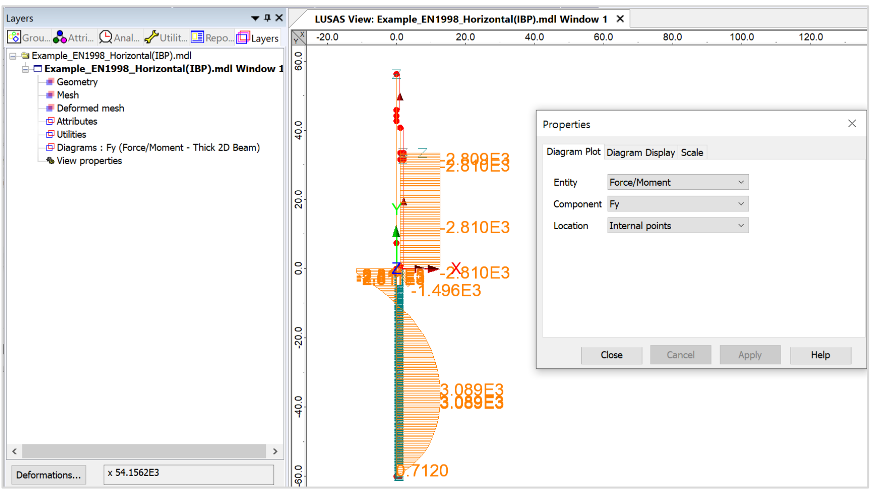


Fig 233 Shear Force Diagram from a Beam-Stick Model

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

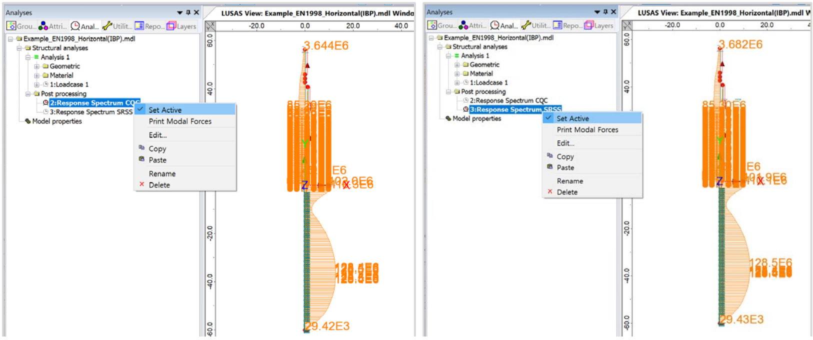


Fig 234 Shear Force Diagram for CQC & SRSS from a Beam-Stick Model

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.

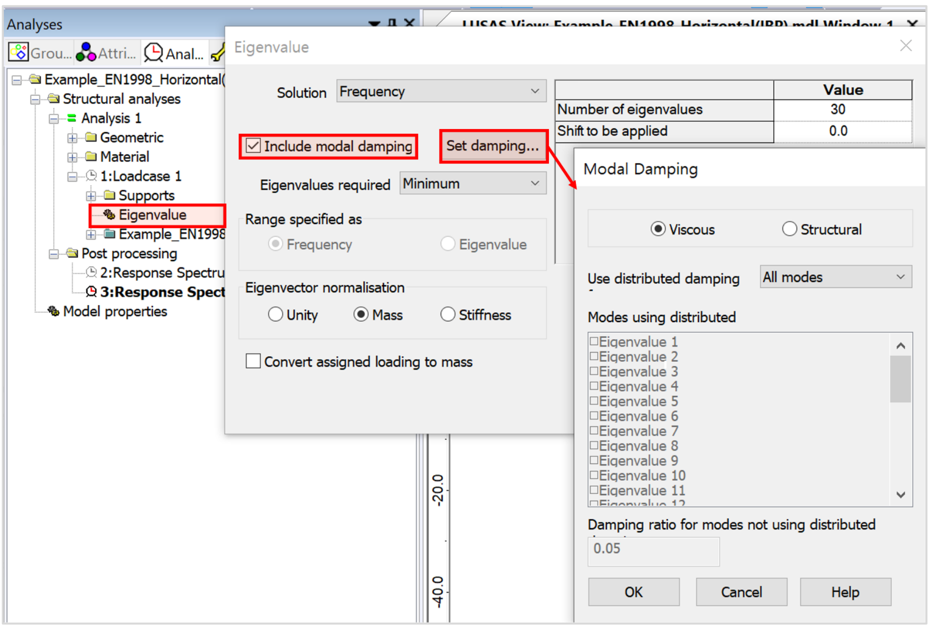
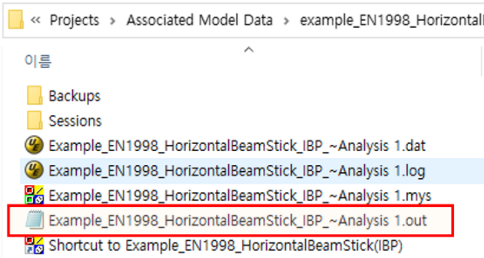


Fig 235 Eigenvalue Control for a Beam-Stick Model

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)**



M O D A L   D A M P I N G   F A C T O R S

MODE	EIGENVALUE	FREQUENCY	VISCOUS DAMPING
1	0.395184	0.100051	0.499755E-02
2	169.755	2.07363	0.240787E-01
3	659.327	4.08668	0.238845E-01
4	4679.03	10.8867	0.437176E-01
5	7507.54	13.7901	0.556754E-01
6	15695.9	19.9395	0.862933E-01
7	19784.3	22.3862	0.873847E-01
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).

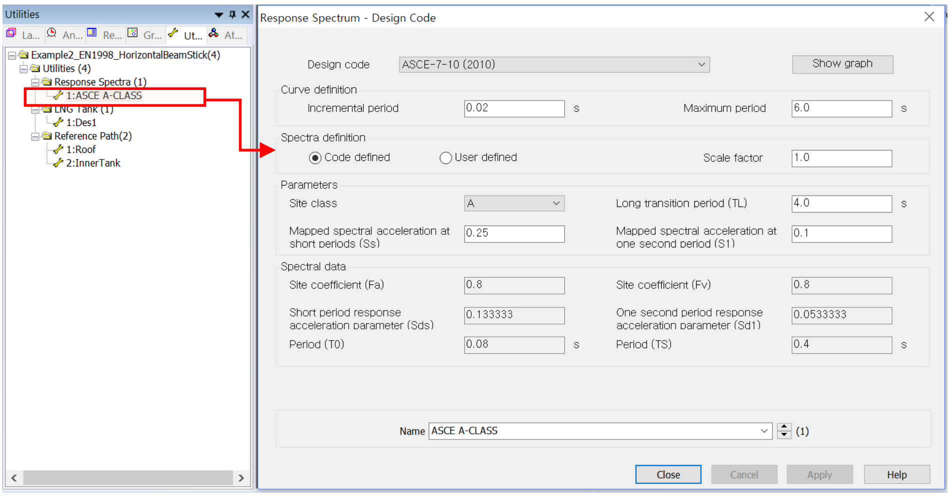


Fig 237 Design Response Spectrum used in Beam-Stick Model by default

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

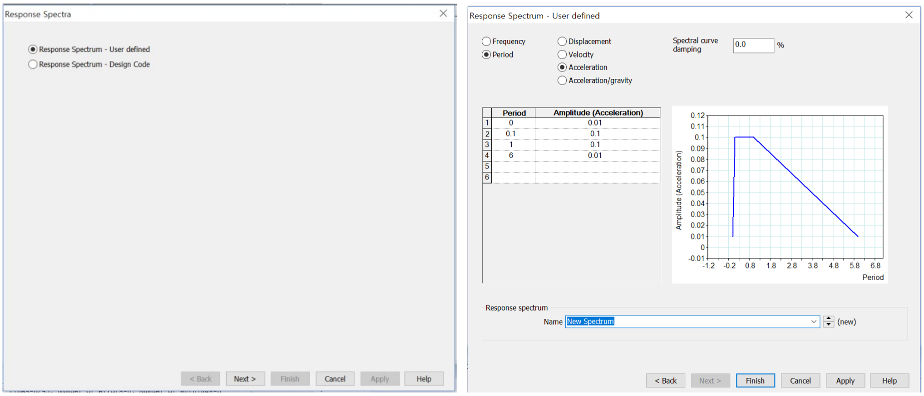


Fig 238 User-defined Response Spectrum

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

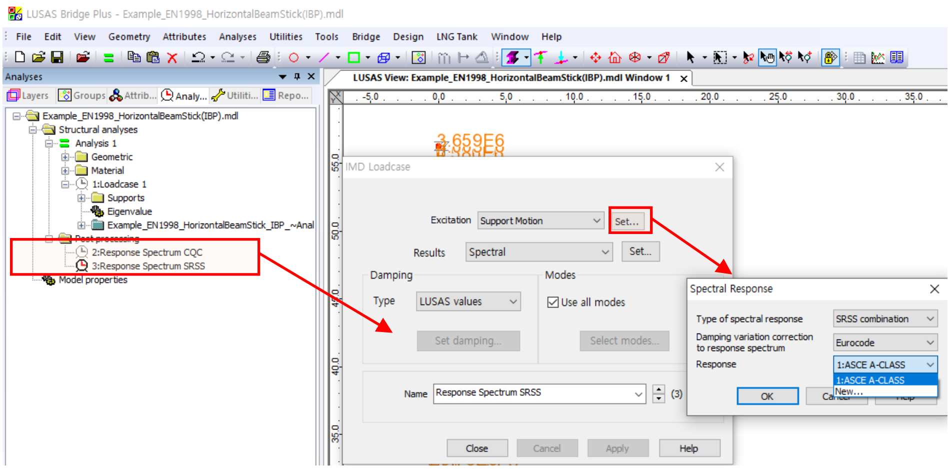


Fig 239 Change of Response Spectrum for Post-Processing

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

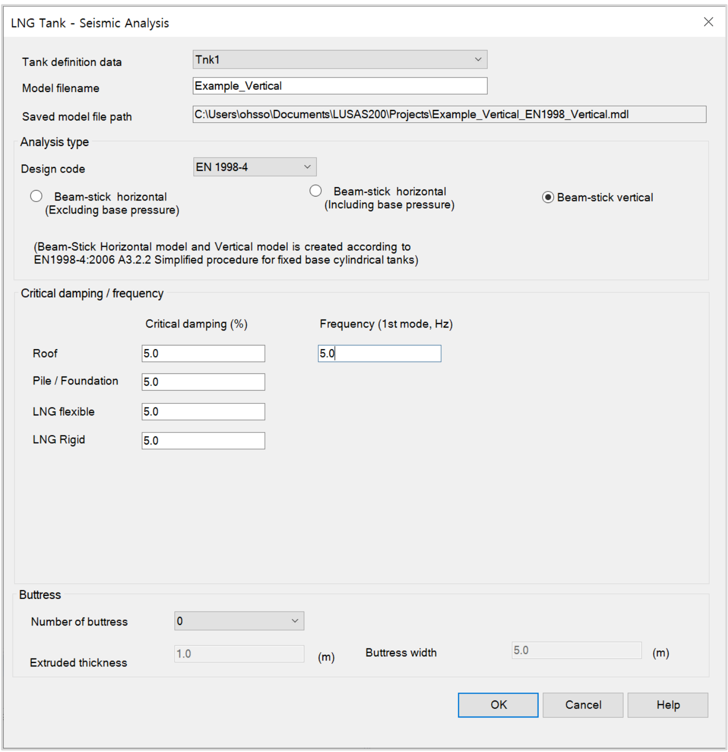


Fig 240 User Dialog for Seismic Analysis Wizard (for Vertical Actions)

Mesh

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

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

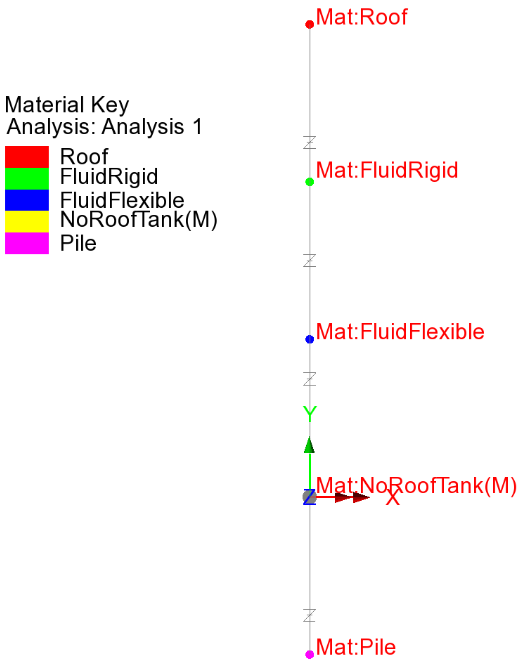


Fig 241 Mesh for a Beam-Stick Model for Vertical Actions

Material Properties

The details for computing properties are summarized in the spreadsheet.

Roof

1) Roof Mass & Stiffness			
Component	Value	Unit	Remark
m_roof	12,027,753	kg	mass of roof
f	5.0000	Hz	fundamental frequency of oscillation of the roof
T	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

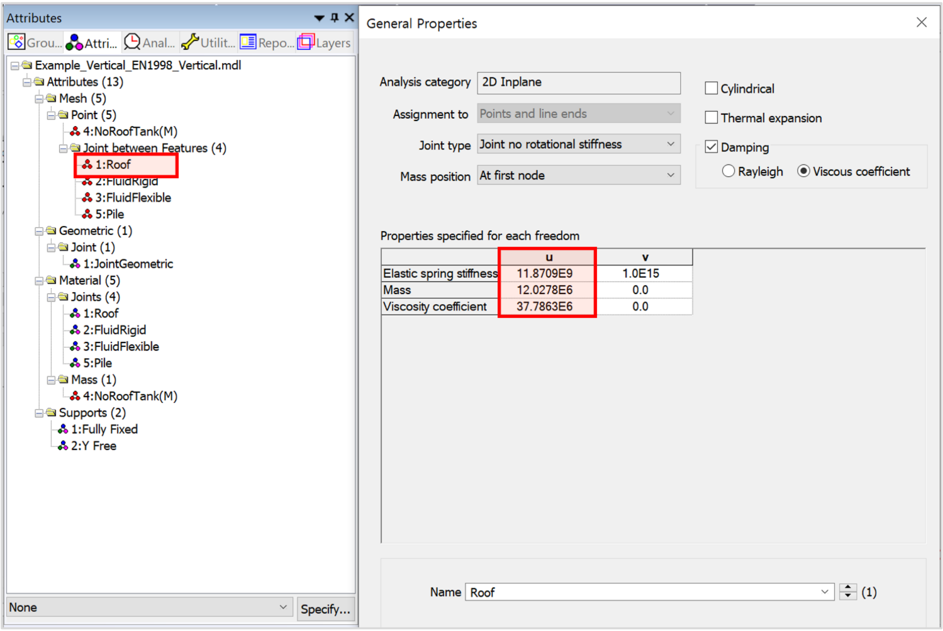


Fig 243 Roof Properties in a Beam-Stick Model for Vertical Actions

Fluid-Flexible

m_LNG_f	89,566,808	kg	mass of LNG (radial breathing), ref. A.40.
k_LNG_r	21,631,229,542,194,300	N/m	

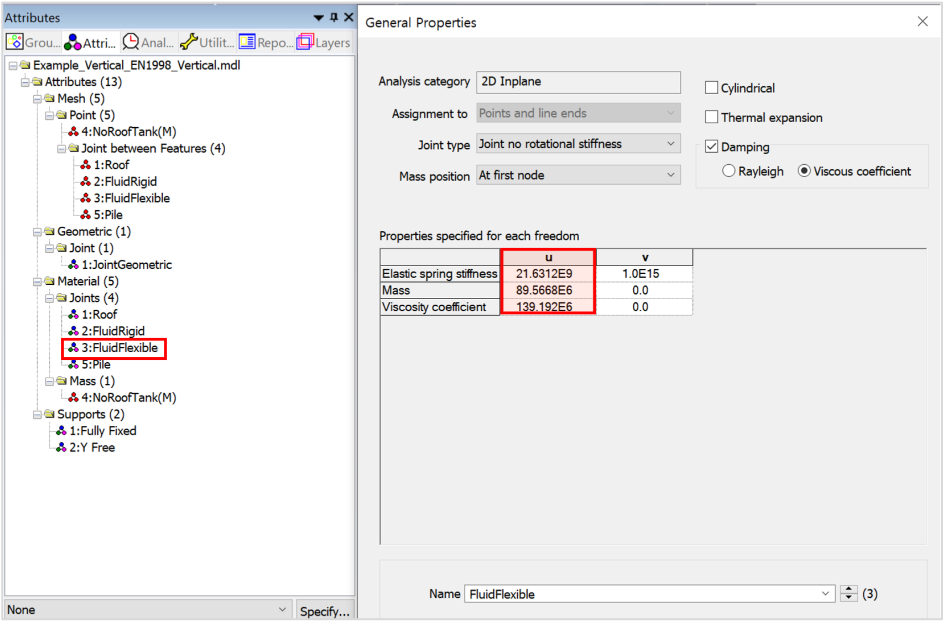


Fig 244 Material Properties for Fluid-Flexible in a Beam-Stick Model for Vertical Actions

Fluid-Rigid

m_LNG_r (1)	52,900,941	kg	mass of LNG (rigidly moving) = sqrt(m_LNG total² - m_LNG f²)
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/m²	hydrodynamic pressure on the wall, from A.17
m_LNG_r (3)	104,022,703	kg	mass of LNG (rigidly moving), ref. A.17.

k_LNG_r	21,631,229,542,194,300	N/m	

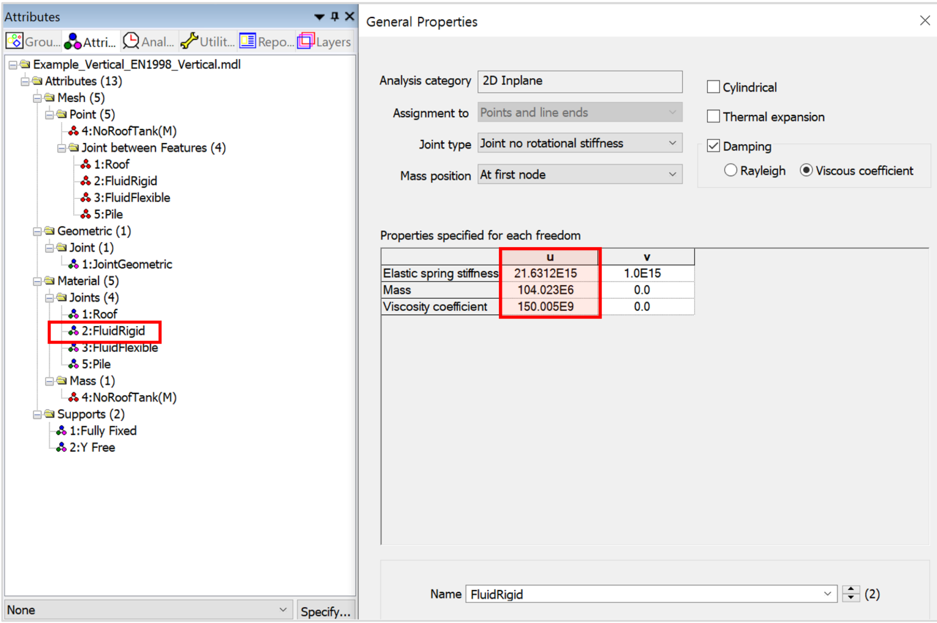


Fig 245 Fluid-Rigid Properties in a Beam-Stick Model for Vertical Actions

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	

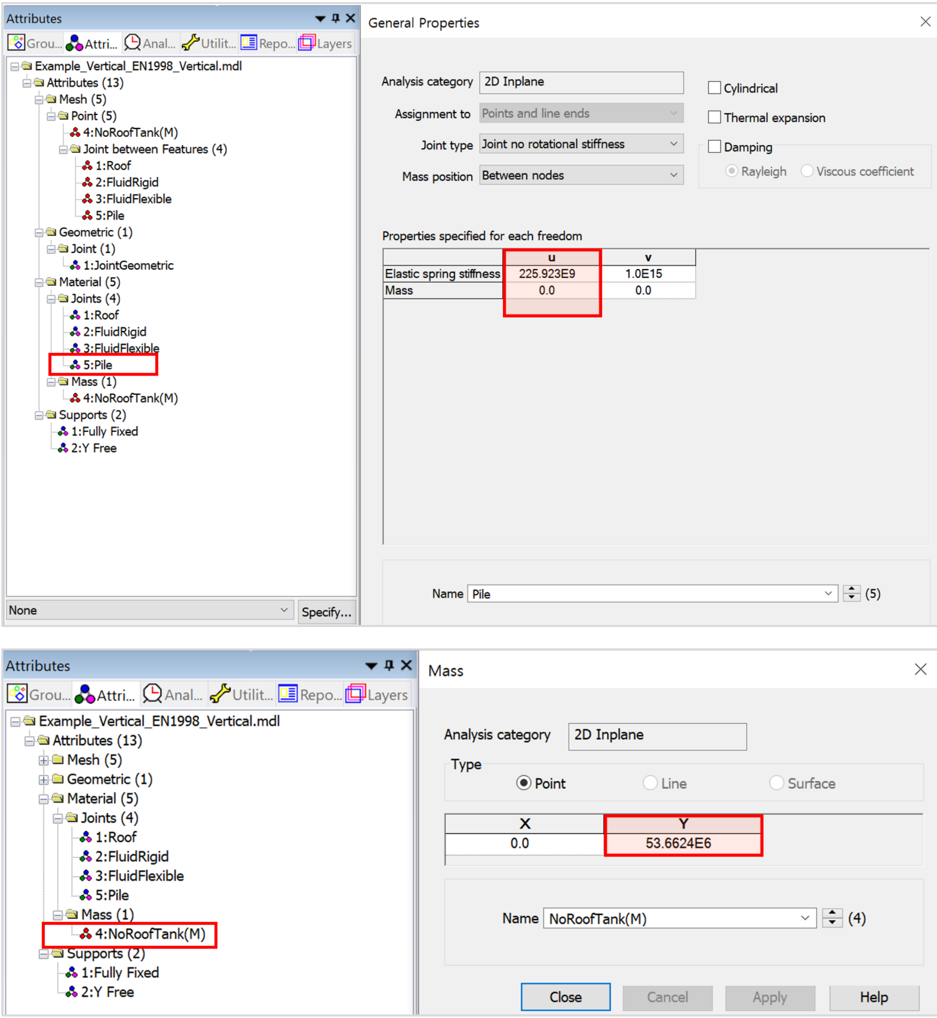


Fig 246 Material Properties for Pile(k), NoRoofTank(M) in Beam-Stick Model for Vertical Actions

Viewing Results

Value

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

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

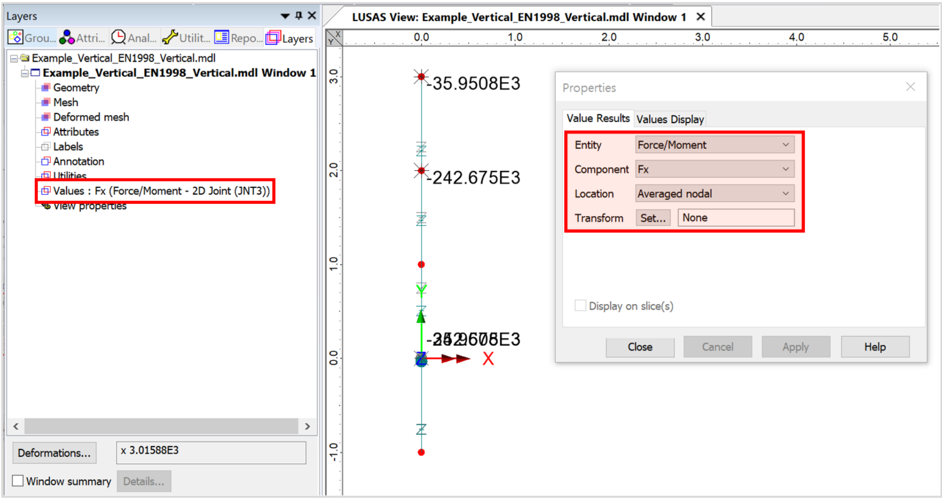


Fig 247 Axial Forces from Beam-Stick Model for Vertical Actions

- 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.

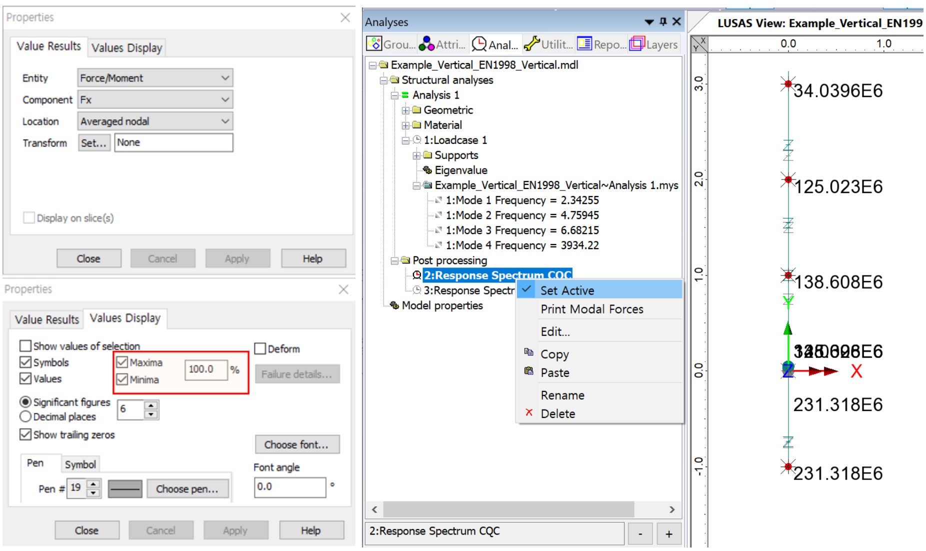


Fig 248 Axial Forces for CQC combination from Beam-Stick Model for Vertical Actions

LUSAS, 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