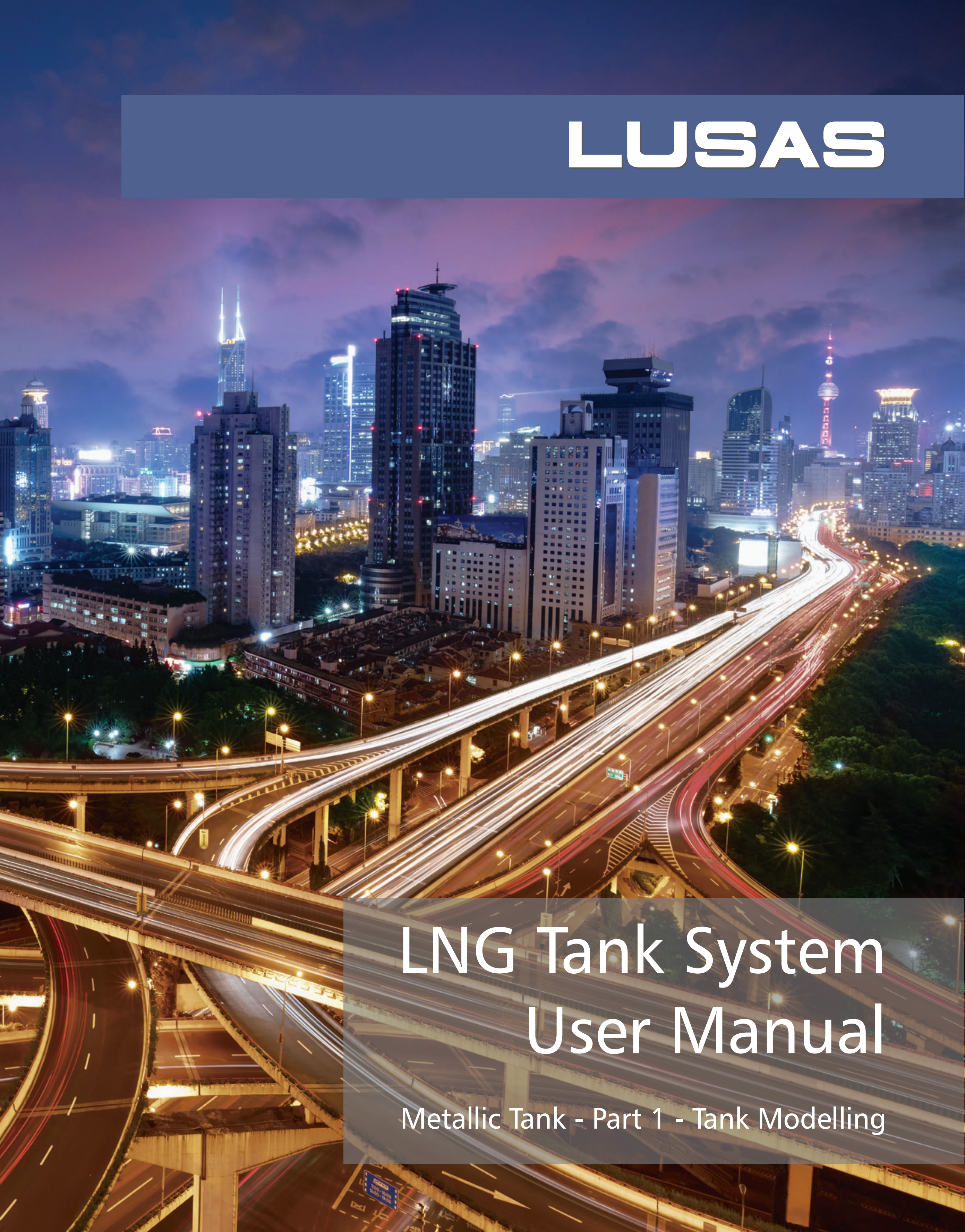


LUSAS

LNG Tank System User Manual

Metallic Tank - Part 1 - Tank Modelling

LNG Tank System

User Manual (Metallic Tank)

Part 1 – Tank Modelling

LUSAS

Forge House, 66 High Street, Kingston upon Thames,

Surrey, KT1 1HN, United Kingdom

Tel: +44 (0)20 8541 1999

Fax +44 (0)20 8549 9399

Email: info@lusas.com

http://www.lusas.com

Distributors Worldwide

Table of Contents

LNG Tank Modelling 1

Overview 1

2D Axisymmetric Static Structural Analysis 3

2D Axisymmetric Construction Stage Analysis 5

2D Axisymmetric Thermal Analysis 8

3D Shell Static Structural Analysis 18

3D Shell Eigenvalue Analysis 31

2D Beam-Stick FSSI Seismic Analysis 33

Exporting Forces from the 2D Axisymmetric Model 48

Exporting Forces from the 3D Shell Model 52

Examples – User Inputs 55

Tank Definition 55

2D Axisymmetric Static Structural Analysis 86

2D Axisymmetric Staged Construction Analysis 119

2D Axisymmetric Thermal Analysis 126

3D Shell Analysis 132

LNG Tank Modelling

Overview

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

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

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

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

Capabilities

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

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

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

2D Axisymmetric Static Structural Analysis

Elements

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

Groups / Materials

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

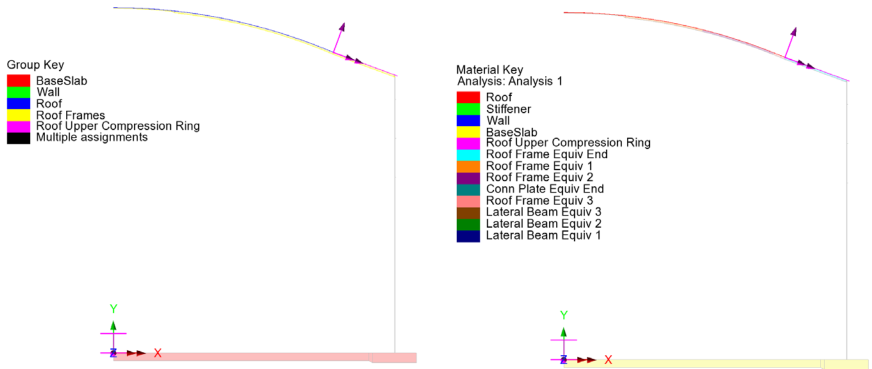


Fig 1 Group and Material Assignments for a 2D Axisymmetric Static Model

Support Condition for 2D Axisymmetric Model

Three support types are available for selection.

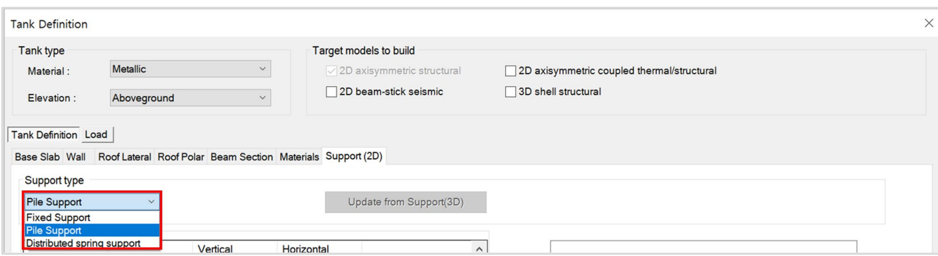


Fig 2 Support Types Available

Fixed Support

Fully fixed supports are assigned to the base slab.

Pile Support

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

Regular Support

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

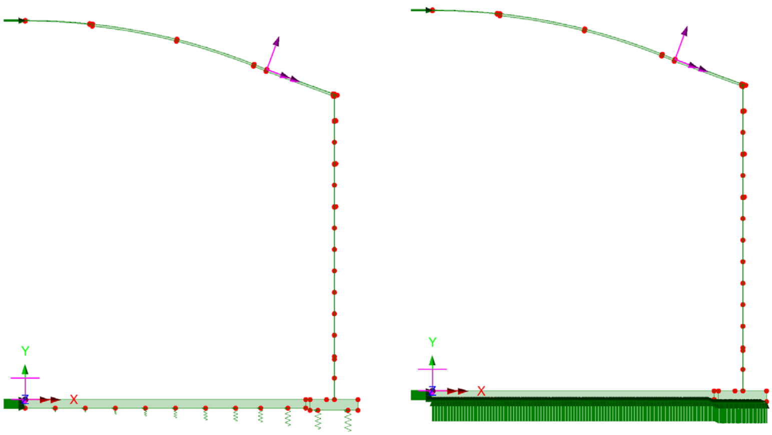


Fig 3 Support Types for a 2D Axisymmetric Static Model (Pile Support / Regular Support)

Loadings

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

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

2D Axisymmetric Construction Stage Analysis

Elements

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

Groups / Materials

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

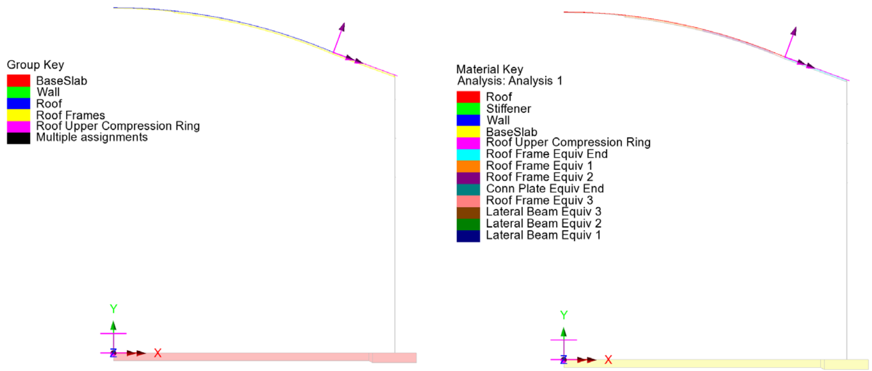


Fig 4 Group and Material Assignments in a 2D Axisymmetric Staged Construction Model

Support Condition

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

Construction Stages

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

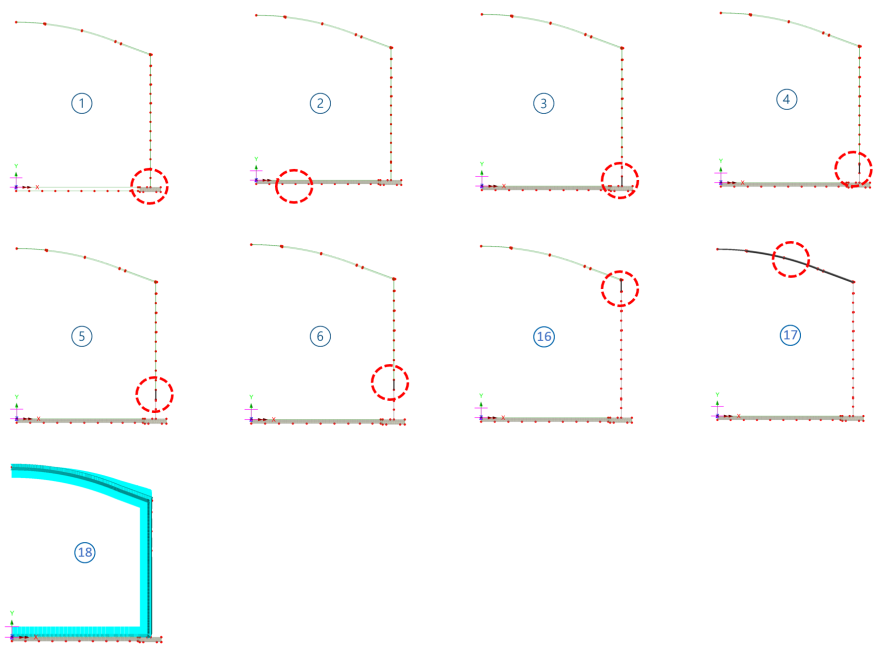


Fig 5 Activation and Deactivation in a Staged Construction Analysis Model

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

Table 1 Sequence of Construction Stages

Loadings

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

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

2D Axisymmetric Thermal Analysis

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

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

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

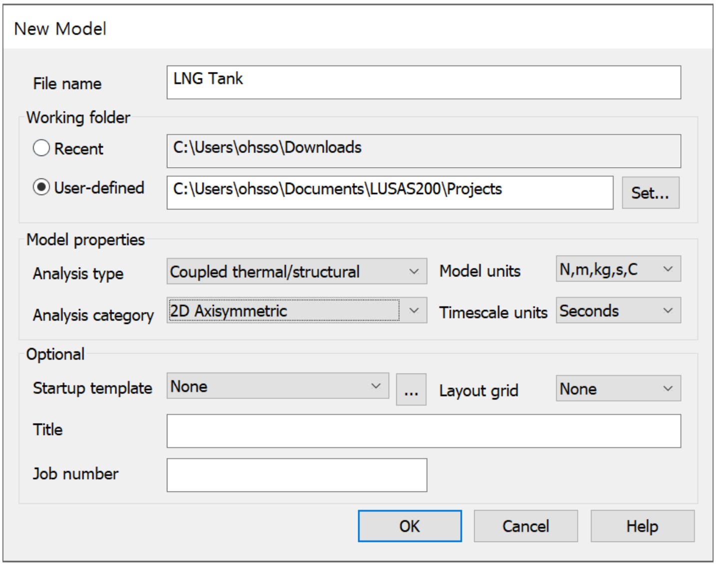


Fig 6 New Model Dialog Setting Thermal/Structural Coupled Analysis

Elements

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

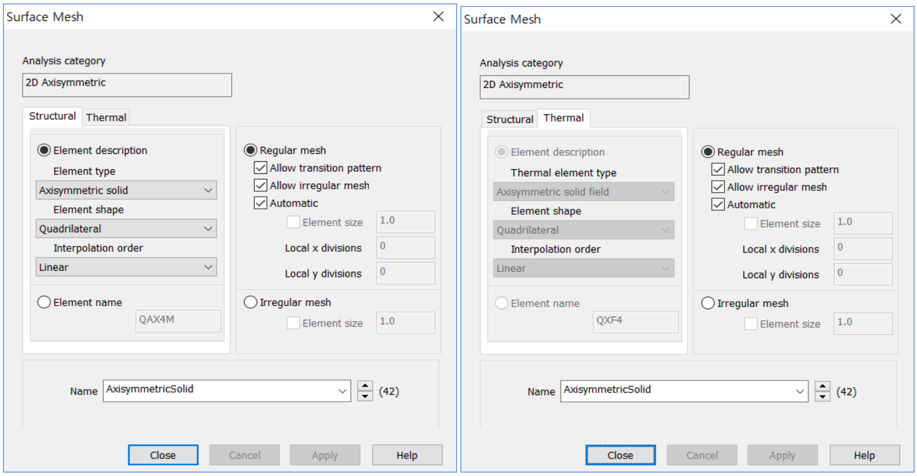


Fig 7 Element Definition for 2D Axisymmetric Thermal Analysis

Insulation

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

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

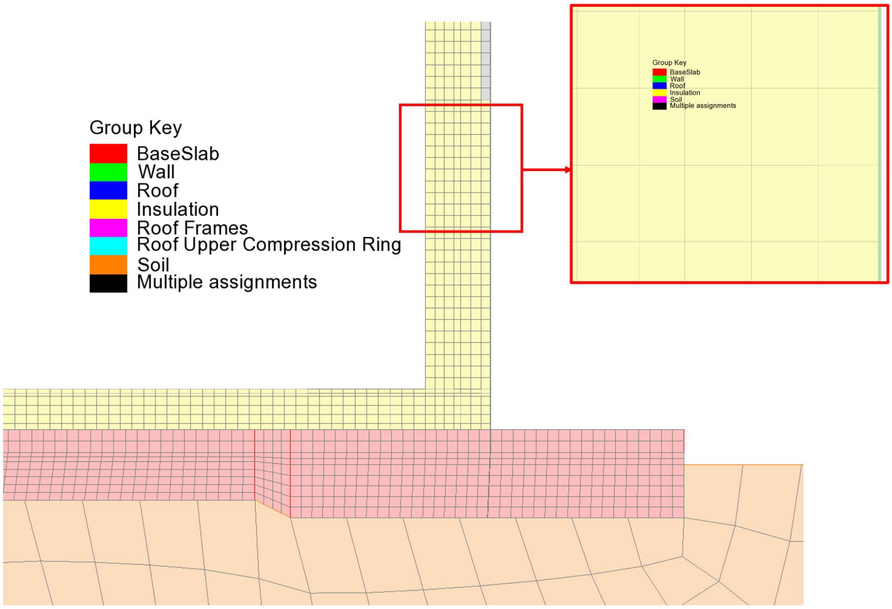


Fig 8 Insulation Elements Separated from Structure Elements

Ground (Soil)

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

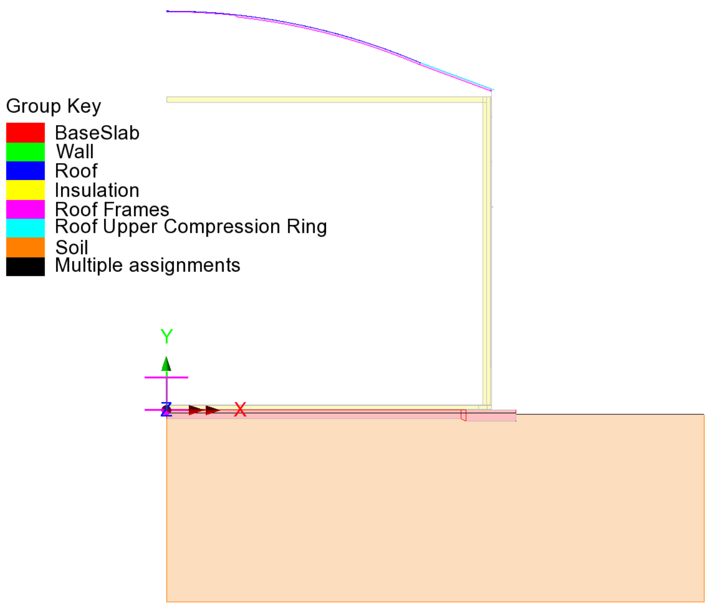


Fig 9 Mesh for 2D Axisymmetric Thermal Analysis

Groups / Materials

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

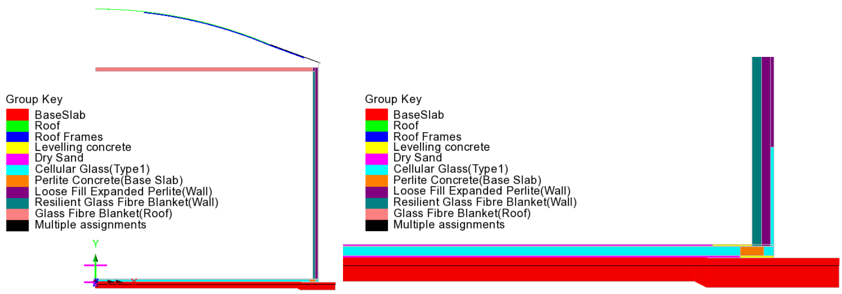


Fig 10 Group Assignments in 2D Axisymmetric Thermal Analysis Model

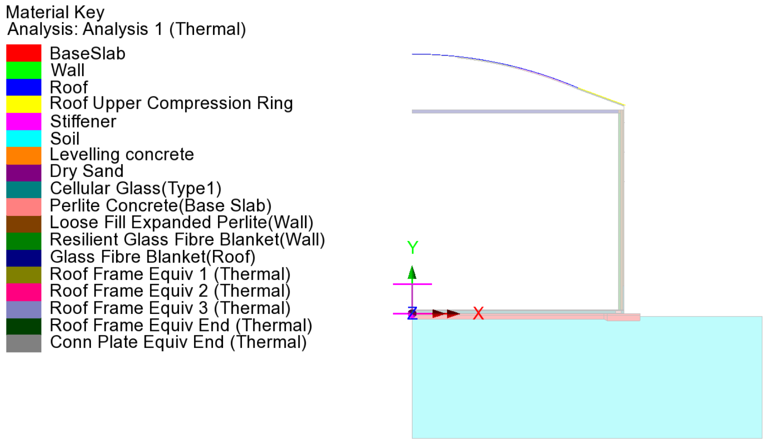


Fig 11 Material Assignments in 2D Axisymmetric Thermal Analysis Model

Supports and Loading for Thermal Analysis

The 1st Loadcase

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

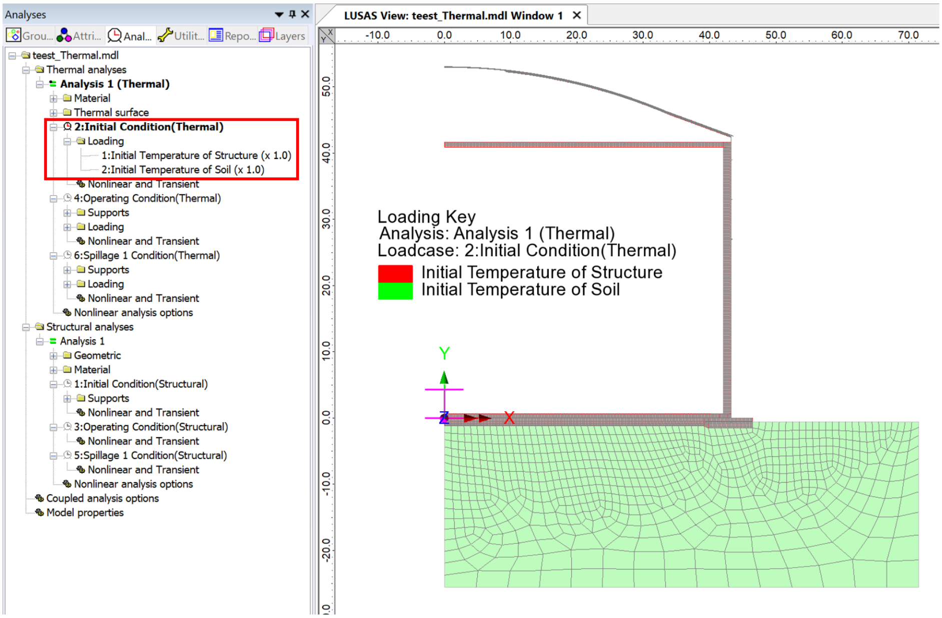


Fig 12 Thermal Analysis -1st Loadcase

The 2nd Loadcase

Liquid temperature is assigned to inner side of the insulation.

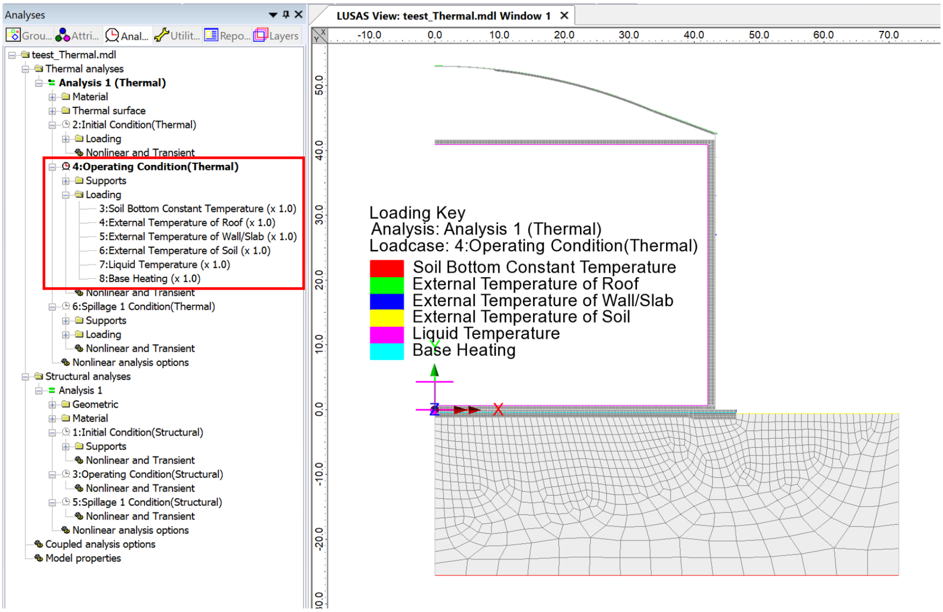


Fig 13 Thermal Analysis – 2nd Loadcase

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

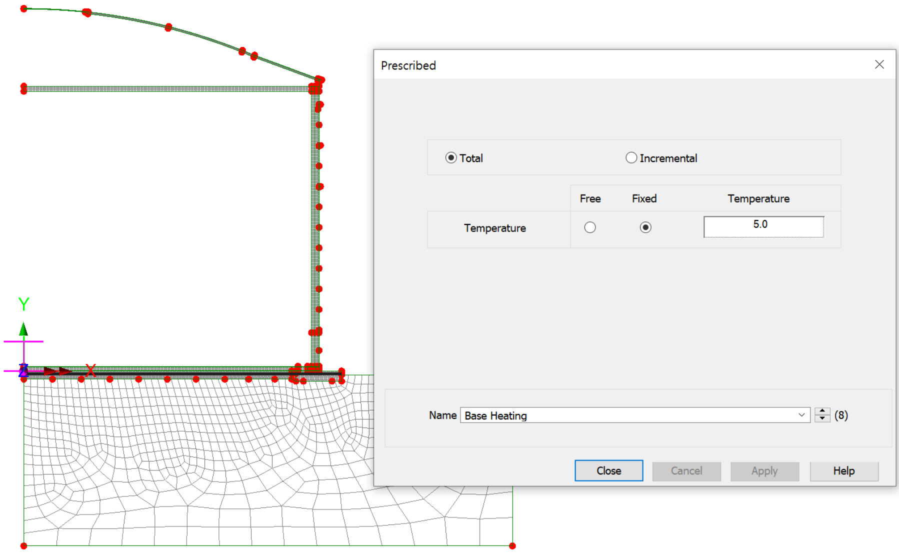


Fig 14 Base Heating Temperature in a 2D Axisymmetric Thermal Analysis Model

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

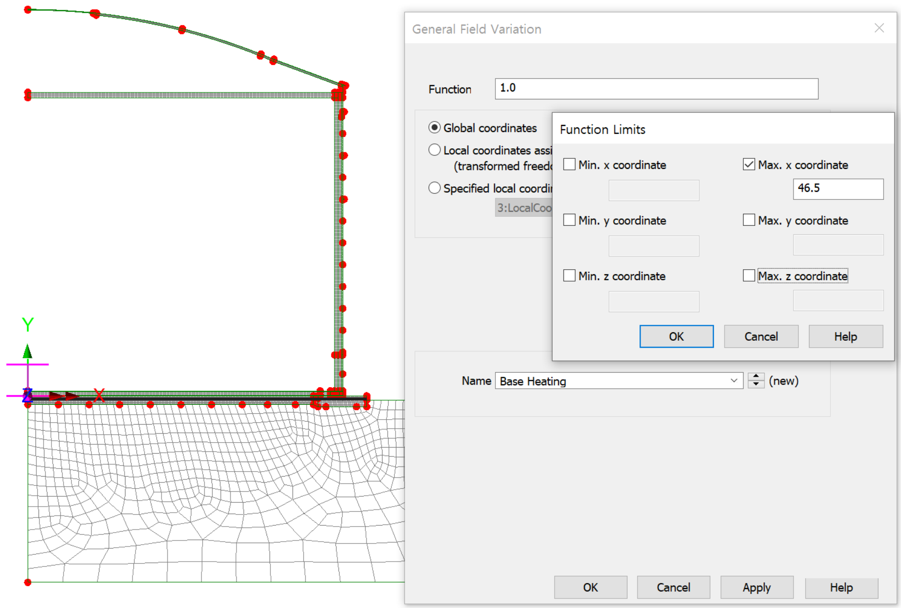


Fig 15 Base Heating Temperature Variation in a 2D Axisymmetric Thermal Analysis Model

Supports and Loadings for Structural Analysis

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

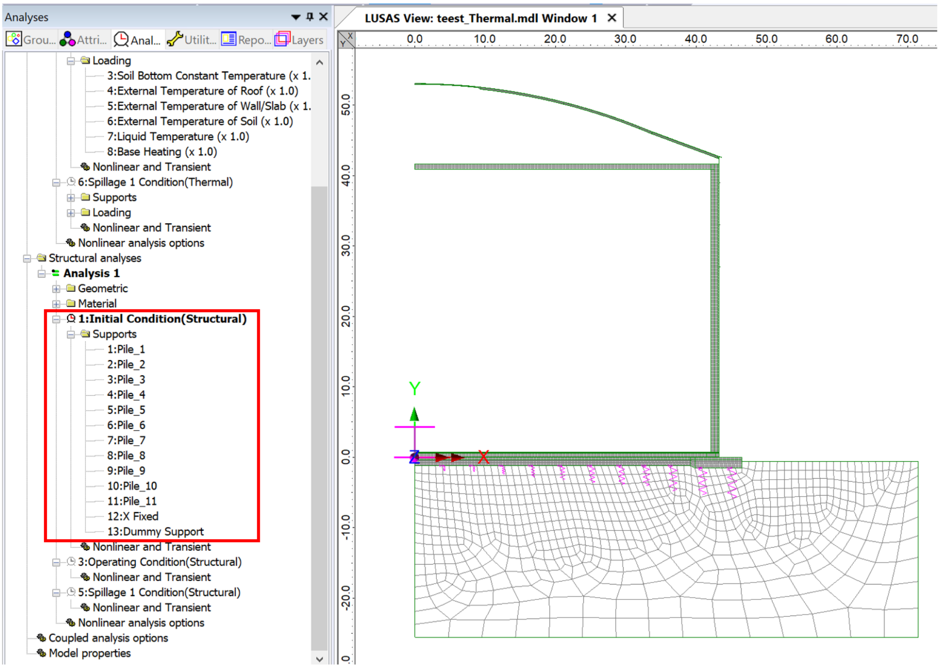


Fig 16 Pile Support for Structural Analysis following Thermal Analysis

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

3D Shell Static Structural Analysis

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

Elements & Geometric Properties

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

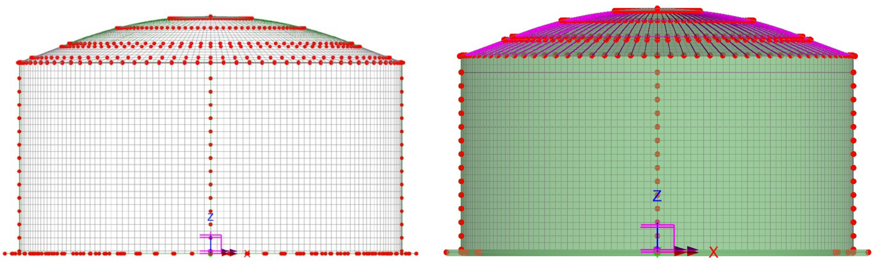


Fig 17 3D Shell Model for Static Analysis

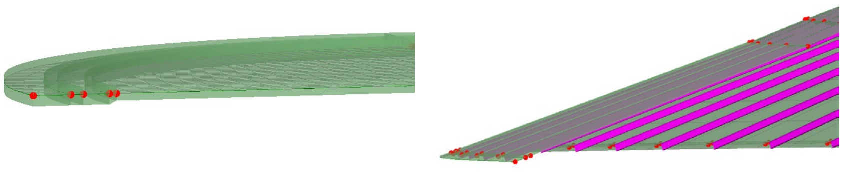


Fig 18 3D Shell Model Thickness Variation at Roof and Slab

Groups and Materials

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

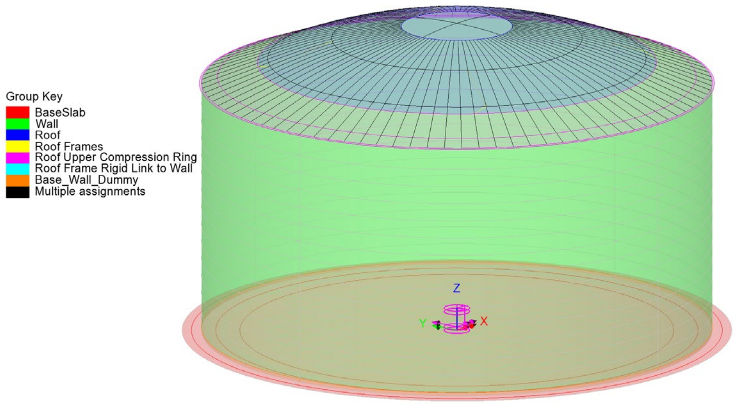


Fig 19 Groups in a 3D Shell Model

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

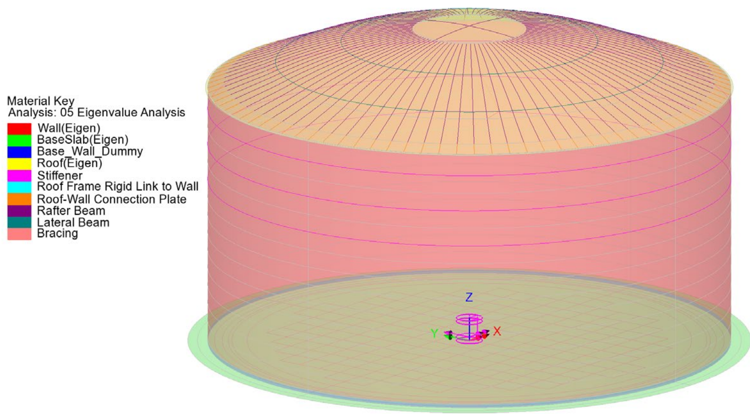


Fig 20 Material Assignments in a 3D Shell Model

Support Conditions

Three different types of support conditions can be defined.

Fixed Support

Fully fixed supports are assigned to the base slab.

Pile Support

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

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

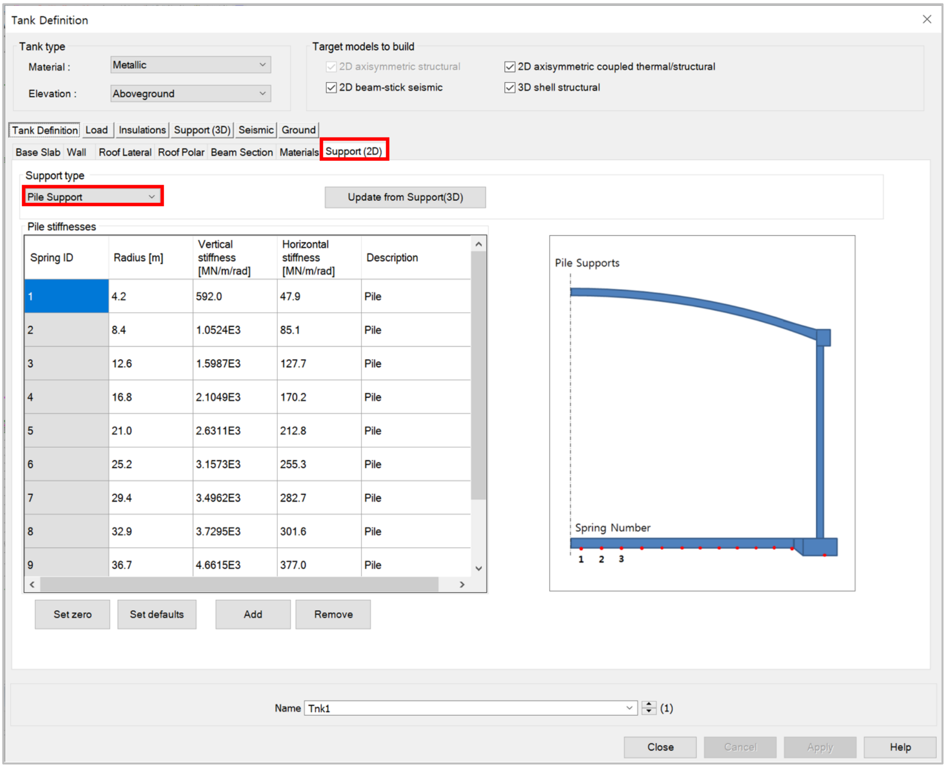


Fig 21 Input for Pile Locations and Stiffnesses

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

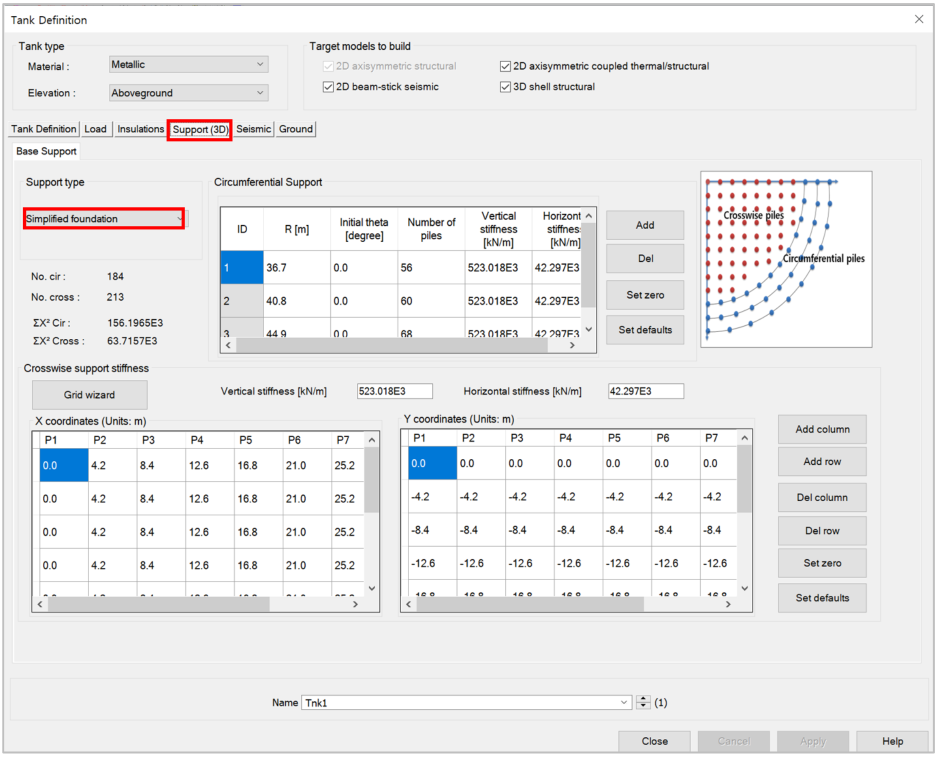


Fig 22 Input for Pile Locations and Stiffnesses (Simplified foundation)

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

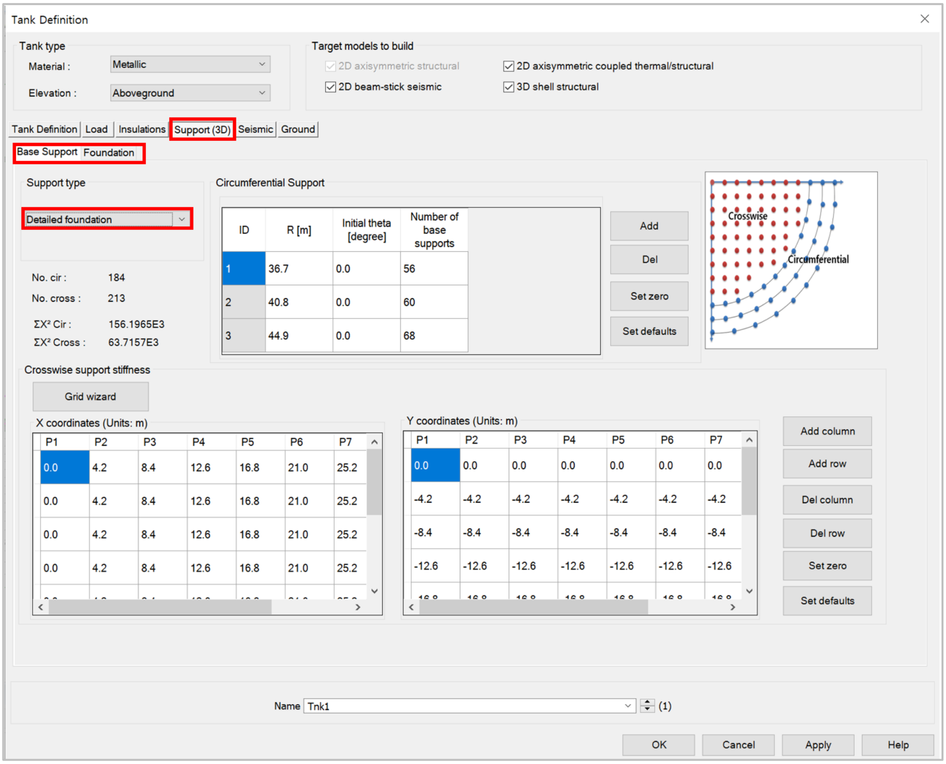


Fig 23 Input for Pile Locations and Stiffnesses (Simplified foundation)

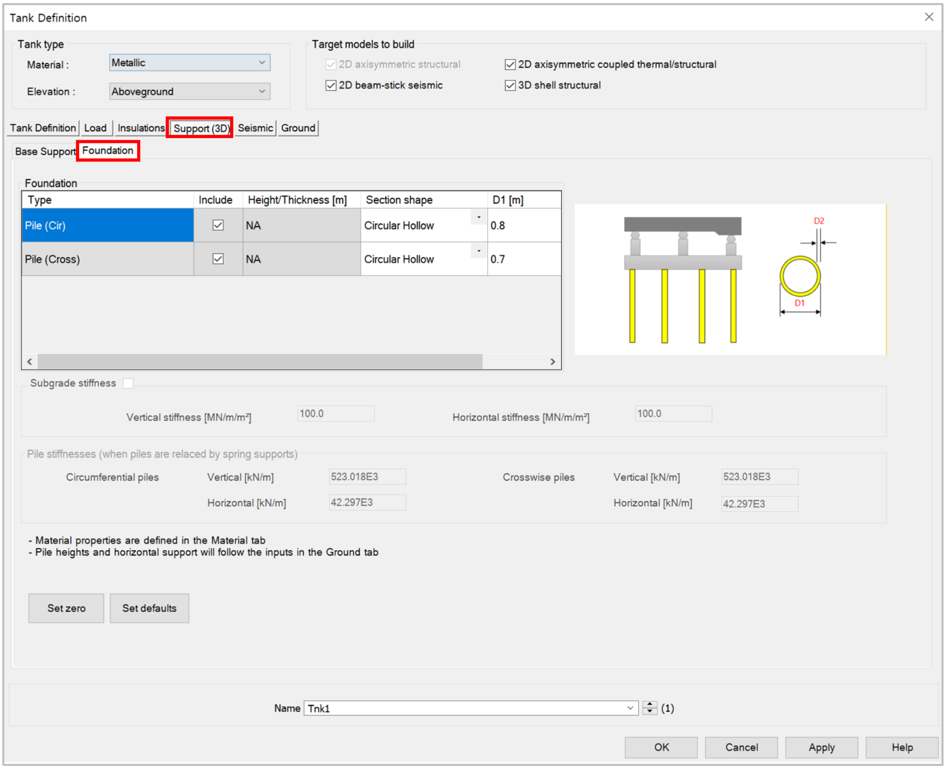


Fig 24 Input for Pile Dimensions (Detailed foundation)

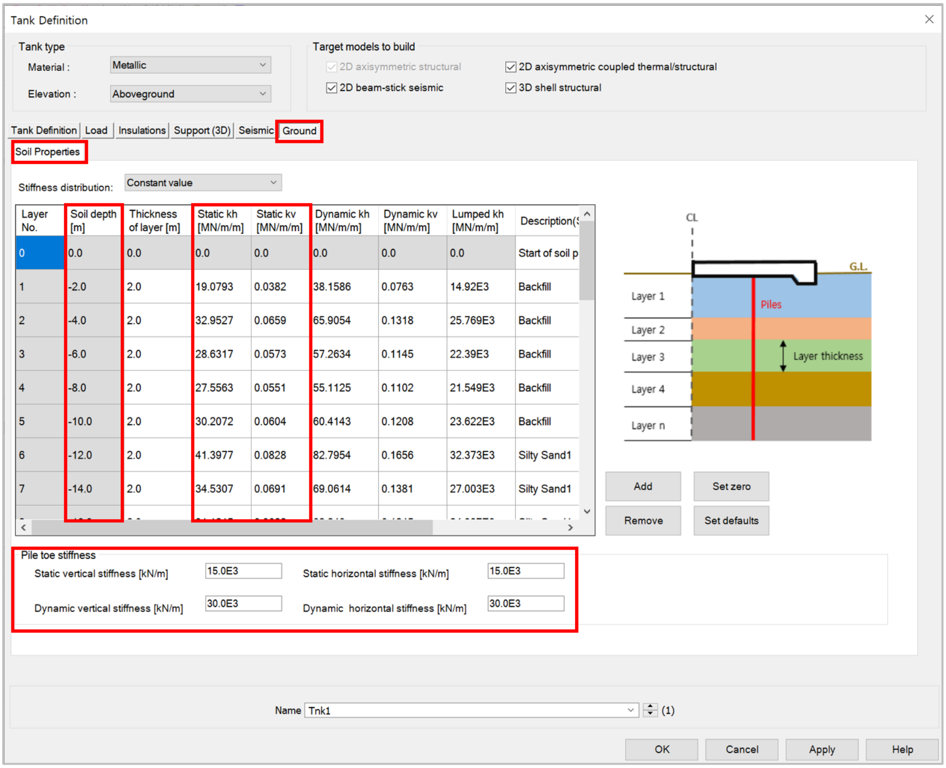


Fig 25 Input for Pile Dimensions (Detailed foundation)

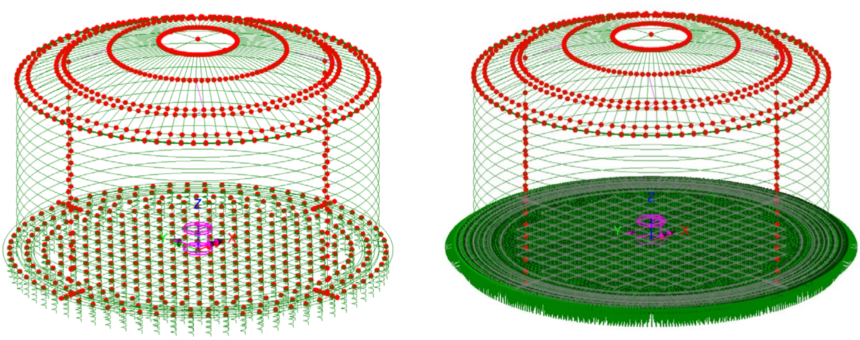


Fig 26 Support Condition for a 3D Shell Model
(Pile Support(Simplified foundation) / Regular Support)

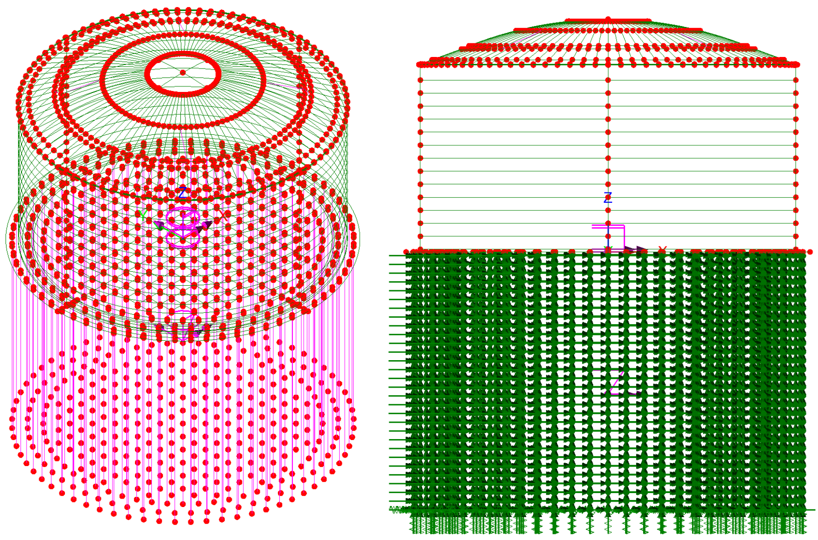


Fig 27 Support Condition for a 3D Shell Model (Pile Support(Detailed foundation))

Regular Support

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

Loadings

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

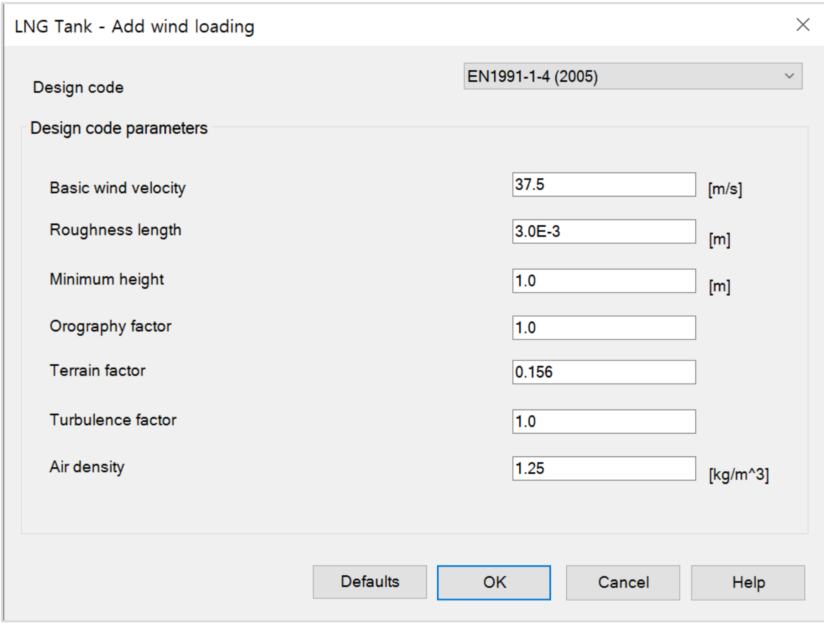


Fig 28 User Input for Wind Load for 3D Shell Model

Other Options

Half Only Model

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

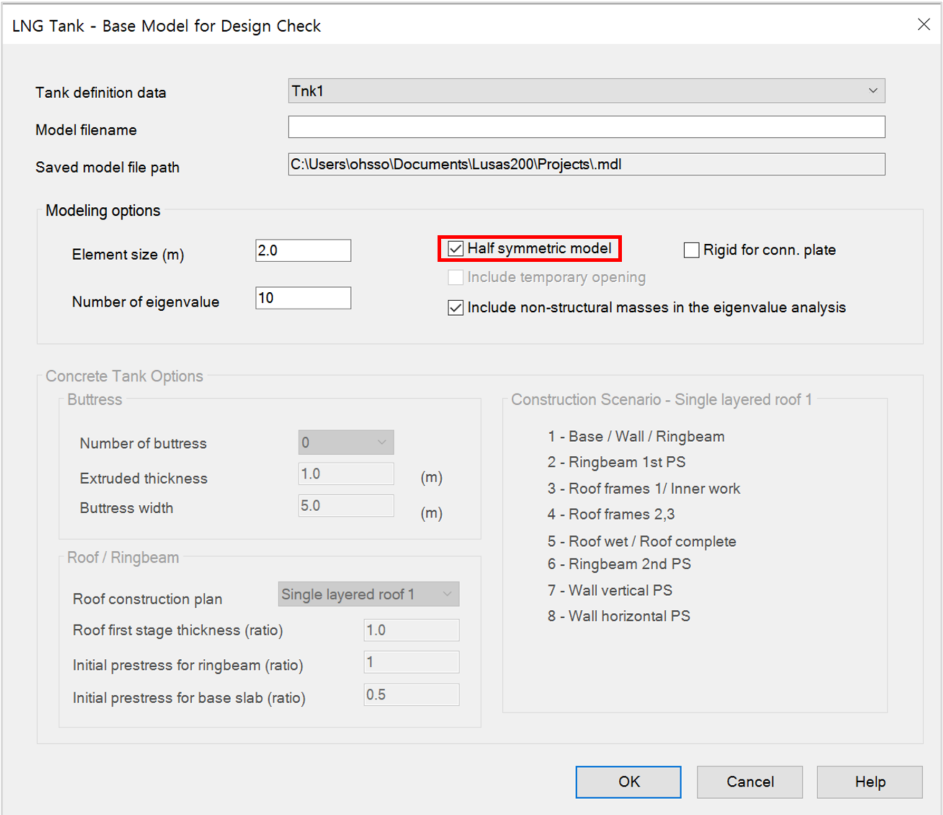


Fig 29 User Input for a 3D Shell Model (Half Model)

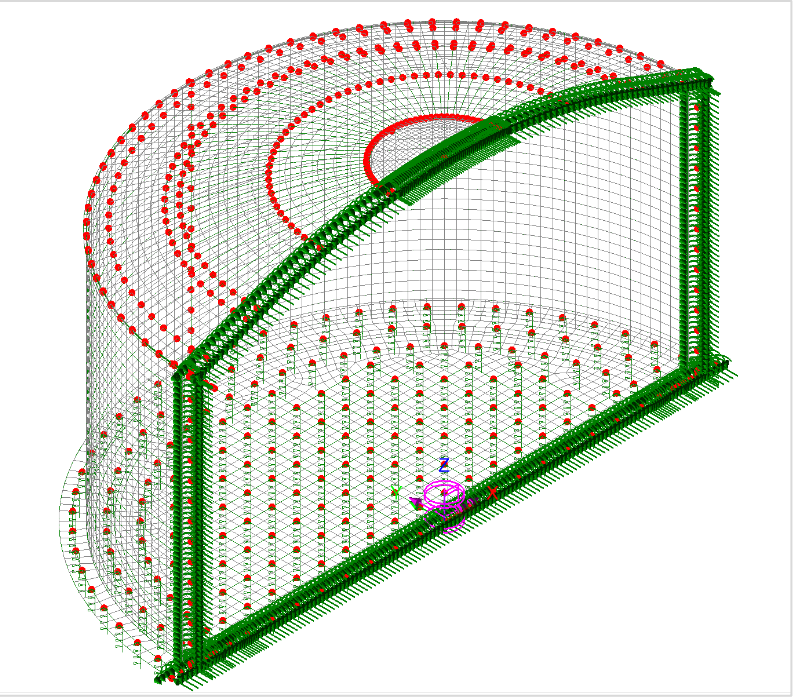


Fig 30 3D Shell Model (Half Model)

Include non-structural masses

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

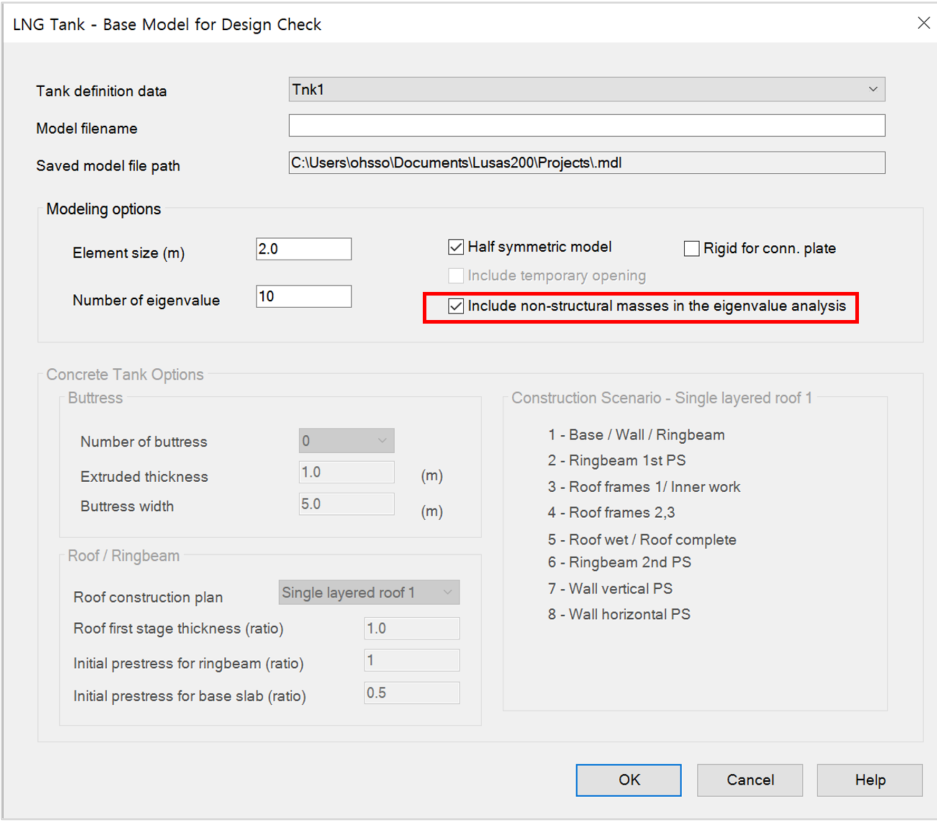


Fig 31 User Input for Eigenvalue Analysis Model including Non-Structural Masses

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 plate	106	7,800	825,740	2,935,740	27,731
Lateral beam	3.93E-01	7,800	3.07E+03	3,066	7,800
Polar beam	9.11E+00	7,800	7.11E+04	71,086	7,800
Bracing	1.29E-01	7,800	1.00E+03	1,003	7,800
Wall	117	7,800	912,893	1,736,893	14,840
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	Roof plate (= Roof plate volume * unit steel mass)	825,740
	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	**2,935,740**
Ring Beam	Ring Beam (= Ring Beam volume * unit steel mass)	-
	wall barrier plate	-
	wall piping and support	-
	Others	-
	Total	**-**
Outer Wall	Wall (= Wall volume * unit steel mass)	912,893
	corner protection	242,000
	wall barrier plate	494,000
	wall piping and support	88,000
	Others	-
	Total	**1,736,893**
Base Slab	Base (= Base slab volume * unit steel mass)	21,797,085
	Others	3,128,000
	Total	**24,925,085**
Inner Steel Tank	Inner 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 32 Summary of Mass Calculation for Eigenvalue Analysis including Non-structural Masses

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

3D Shell Eigenvalue Analysis

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

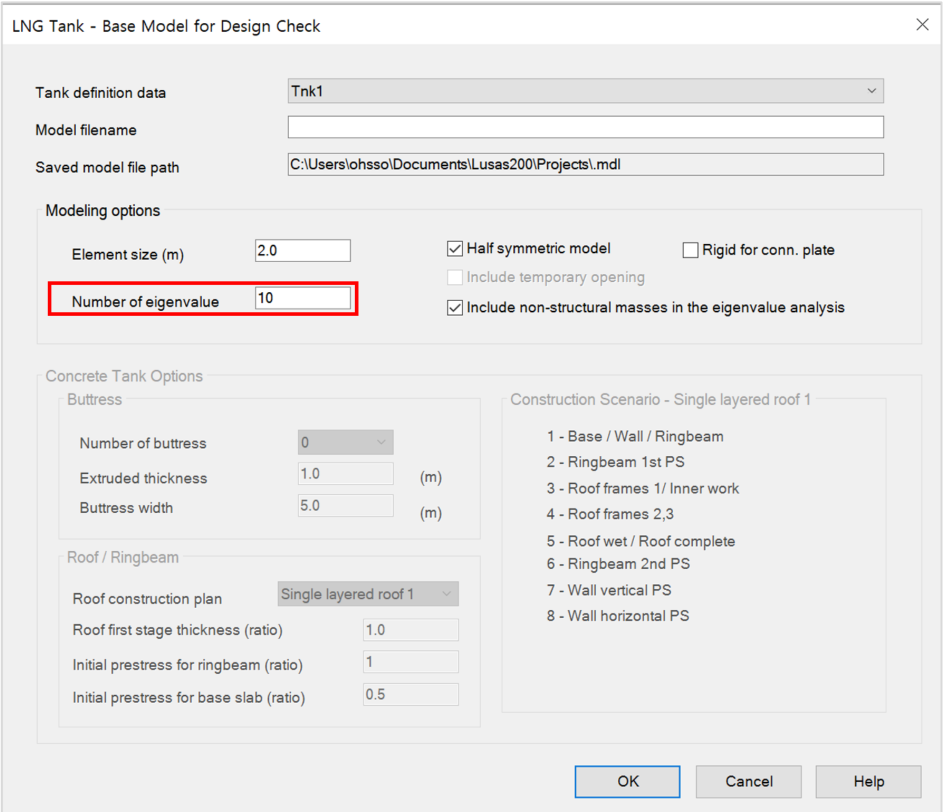


Fig 33 User Input for a 3D Shell Model for Eigenvalue Analysis

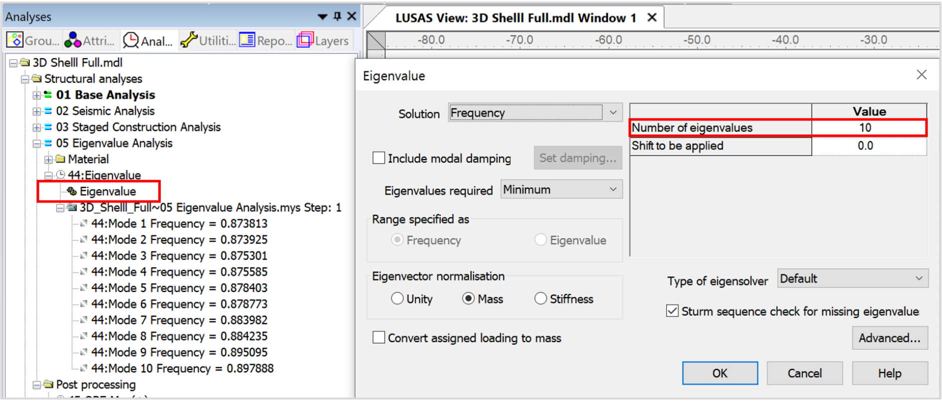


Fig 34 Eigenvalue Analysis in a 3D Shell Model

2D Beam-Stick FSSI Seismic Analysis

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

The beam-stick model includes:

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

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

Model for horizontal actions

Elements

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

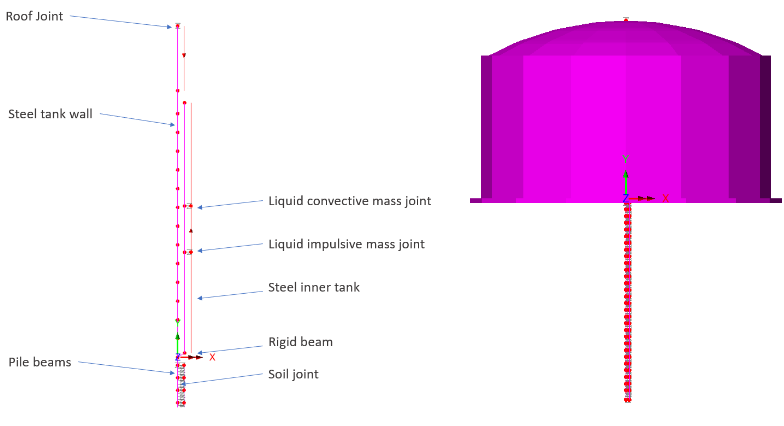


Fig 35 Beam-Stick Modelling Concept for Horizontal Actions

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

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

Geometric Properties

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

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

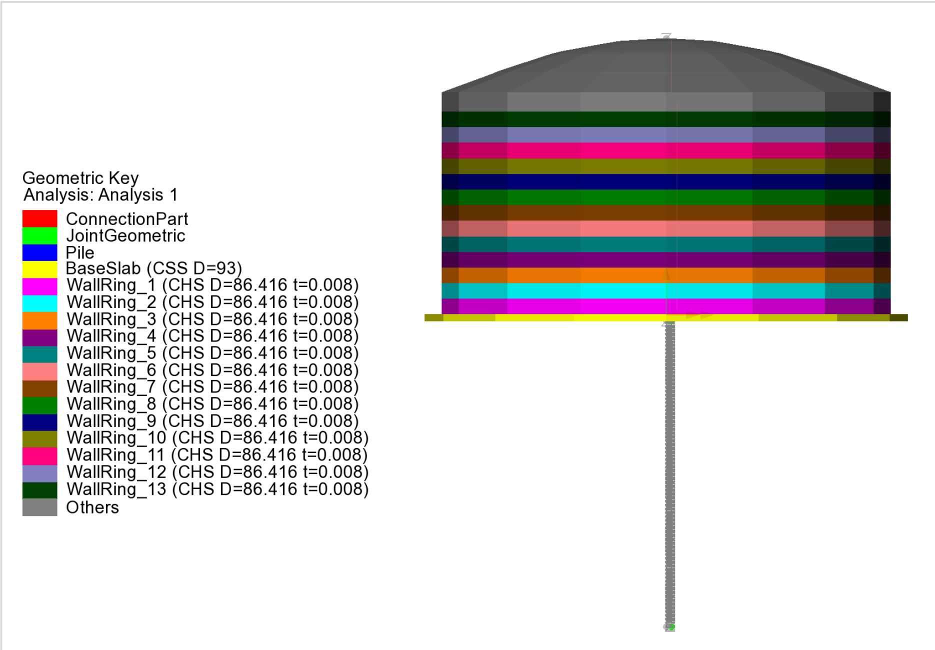


Fig 36 Geometric Properties in a Beam-Stick Horizontal Model

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

Material Properties

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

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

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

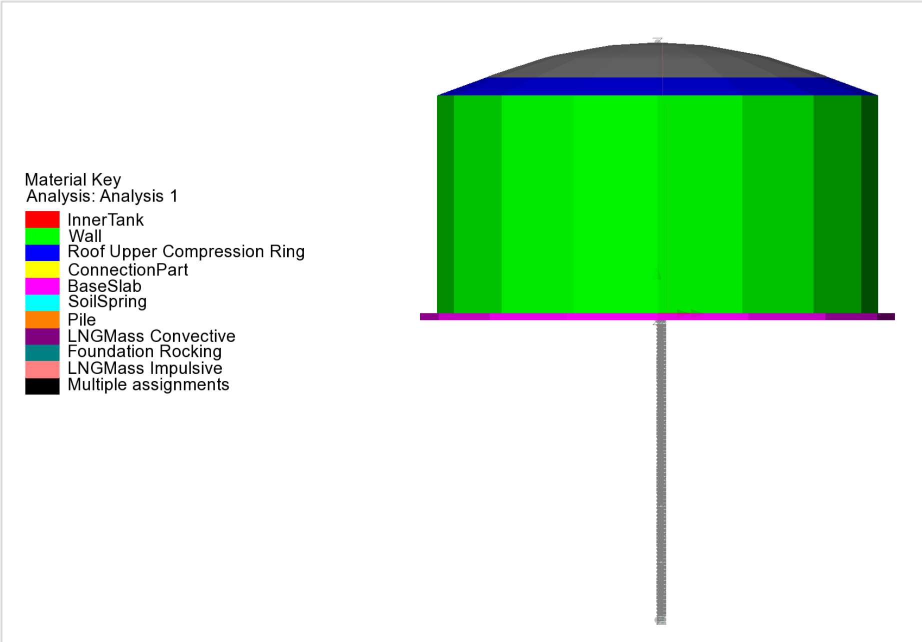


Fig 37 Material Properties in a Beam-Stick Horizontal Model

Verification for Beam-Stick Model — EN1998-4

DIMENSION

Component	Dimension(m)
Inner Tank Radius (R)	42.1
Tank Height (H)	40.06
LNG Height (H_L)	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	103	7,800	825,740	3,015,865	8,812
Lateral Beam	0	7,800	3,066	-	-
Polar Beam	9	7,800	71,086	-	-
Bracing	0	7,800	1,003	-	-
Connection plate	1	7,850	4,970	-	-
Wall & Buttress	117	7,800	912,893	1,736,893	14,840
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

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

EBP (Excluding Base Pressure)

Component	H/R	$m_{(c,i)}/m$	$h_{(c,i)}/H$	mass	Lever arm height
				mc(mi), Kg	hc(hi), m
LNG Convective	0.924	0.49	0.60	50,527,854	23.53
LNG Impulsive	0.924	0.51	0.41	53,494,849	16.13

2) LNG convective stiffness

Component	Value	Unit	Remark
H/R	0.924		LNG 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	

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 impul
ki	11,325,839,357	N/m	

CALCULATED PROPERTIES FOR VERTICAL MODEL

1) Mass & Stiffness for Roof

Component	Value	Unit	Remark
$m_{\_roof}$	3,015,865	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) 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	

3) Mass for Outer&Inner Tank

Component	Value	Unit	Remark
$m_{\_OuterInnerTank}$	29,461,083	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 38 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	103	7,800	825,740	3,015,865	8,812
Lateral beam	0	7,800	3,066	-	-
Rafter beam	9	7,800	71,086	-	-
Bracing	0	7,800	1,003	-	-
Connection plate	1	7,850	4,970	-	-
Wall & Buttress	117	7,800	912,893	1,736,893	14,840
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
$m_{\_roof}$	3,015,865	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
$m_{\_OuterInnerTank}$	29,461,083	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 39 Computation Summary of Liquid Masses for the Beam-Stick Model

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

Groups

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

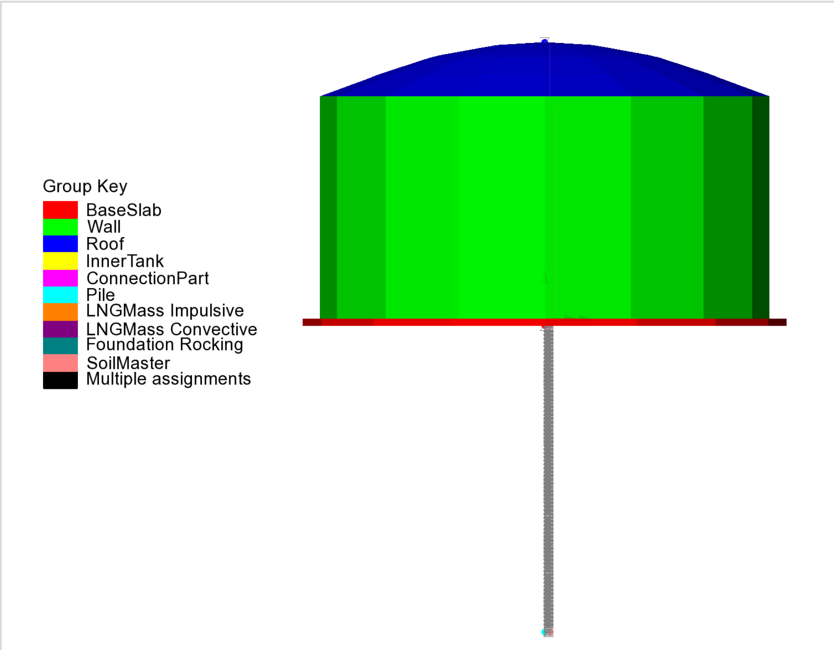


Fig 40 Groups in a Beam-Stick Horizontal Model

Damping Coefficients

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

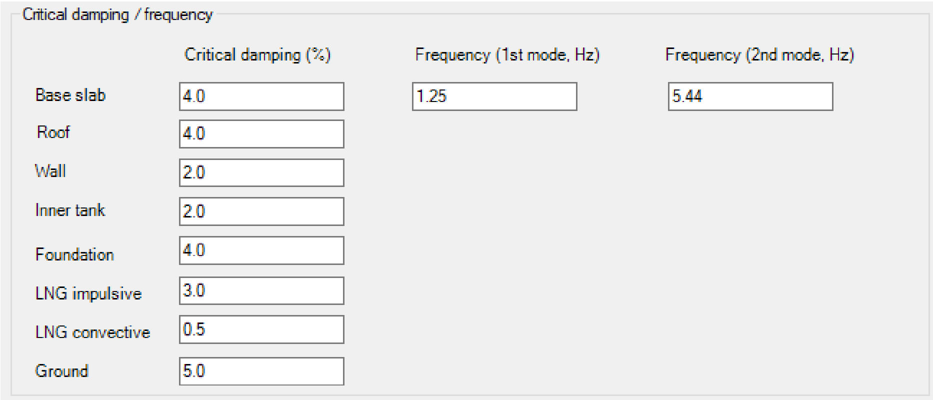


Fig 41 User Inputs for Damping for Seismic Analysis

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

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

Support Conditions

Vertical supports are assigned to all members.

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

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

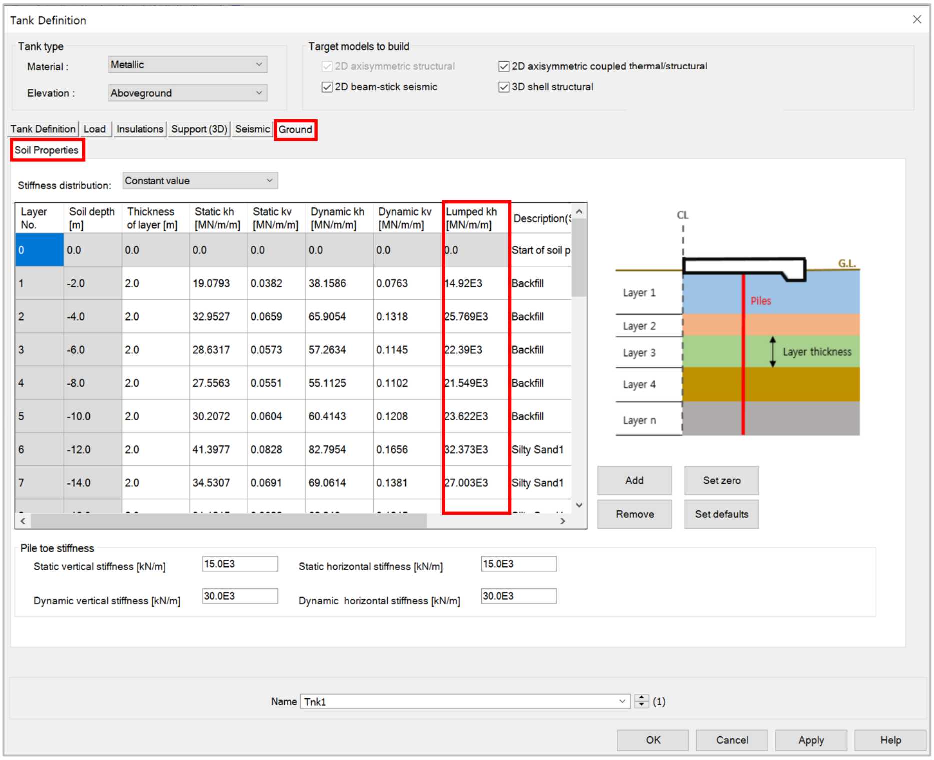


Fig 42 lumped stiffness for Soil Springs for Seismic Analysis

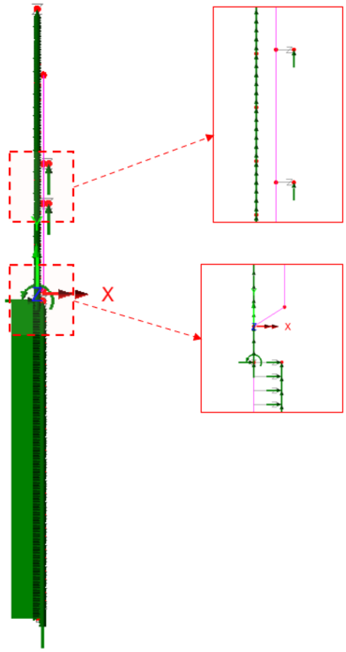


Fig 43 Support Conditions in a Beam-Stick Horizontal Model

Loadings

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

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

Analysis Control

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

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

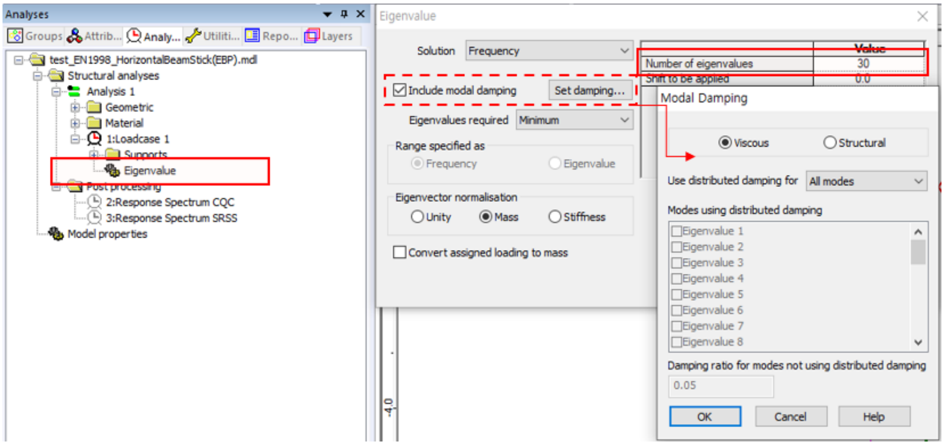


Fig 44 Eigenvalue Control for a Beam-Stick Horizontal Model

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.

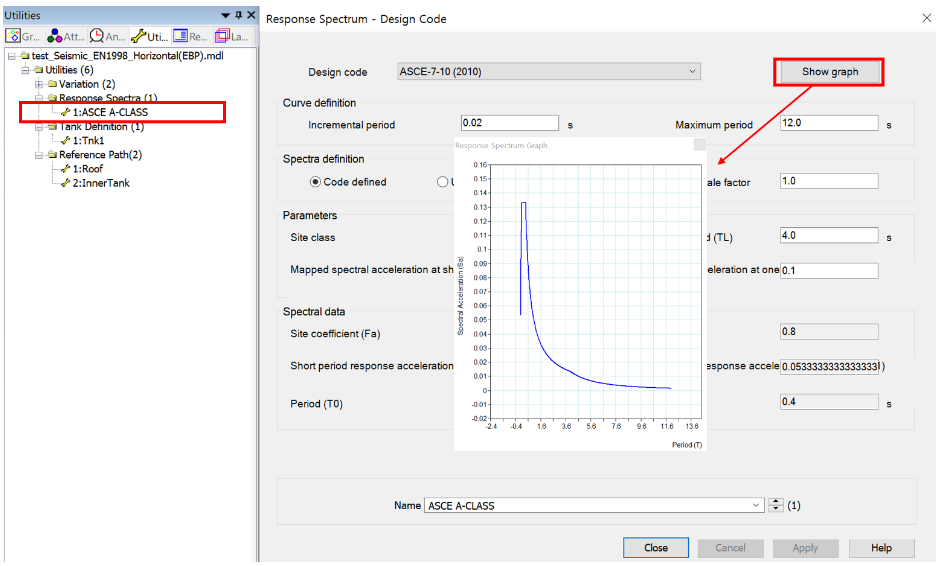


Fig 45 Default design Response Spectrum for a Beam-Stick Horizontal Model

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 63].

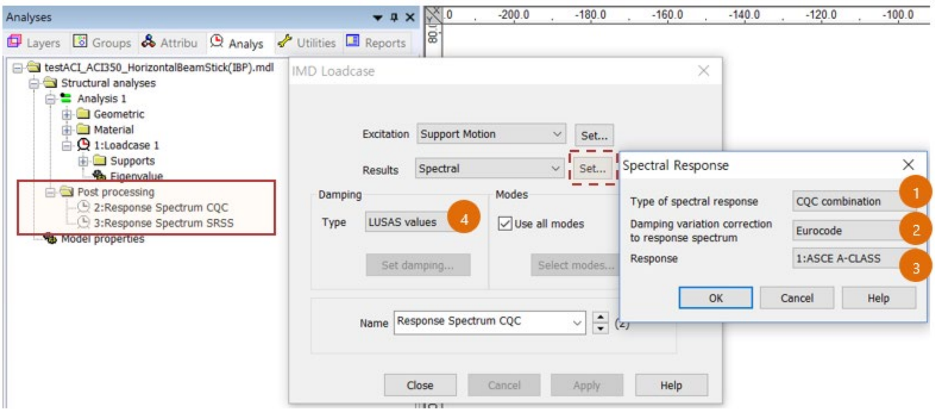


Fig 46 Post Processing Options for a Beam-Stick Horizontal Model

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

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

The design response spectrum is chosen.

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

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

Model for vertical actions

Elements

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

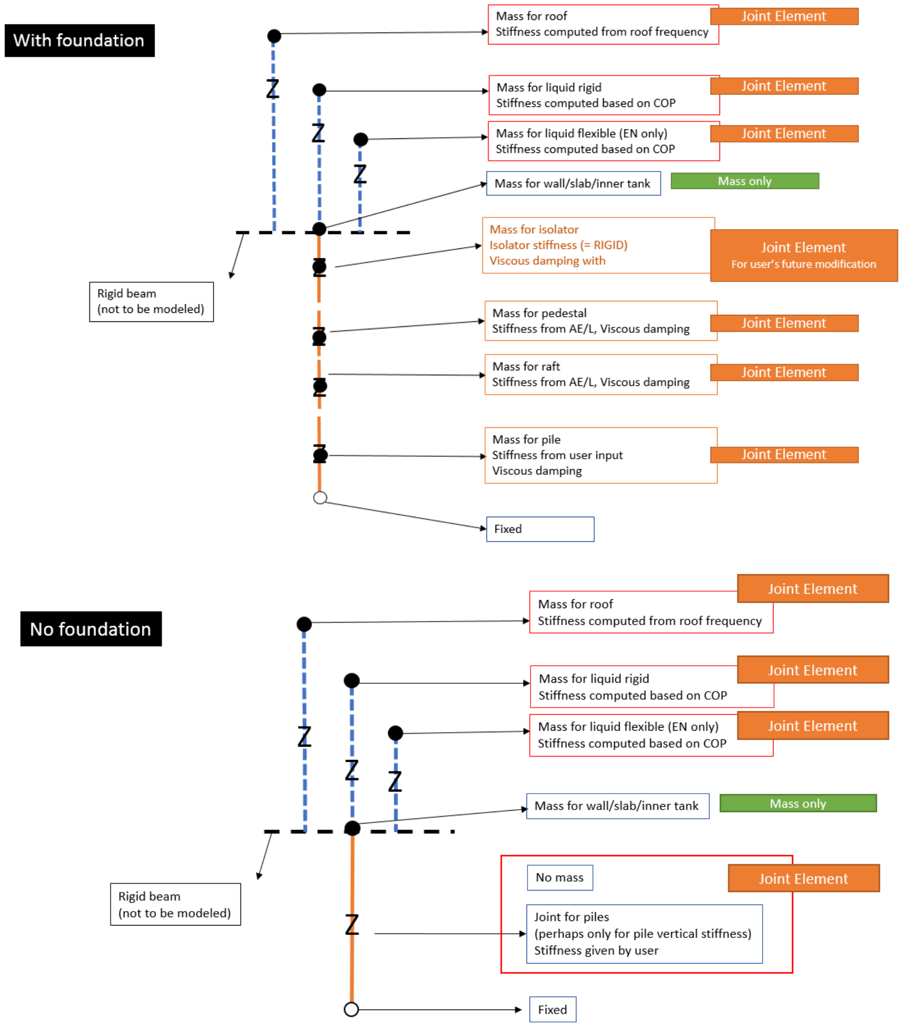


Fig 47 Beam-Stick Modelling Concept for Vertical Actions

The model is built using four joint elements as shown in [Fig 65]. Four joint elements share the node at the location of 'Mass for Wall & Slab & Inner tank'. The length of joint elements does not affect the analysis result. Different joint lengths are shown here only for visualization purposes.

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

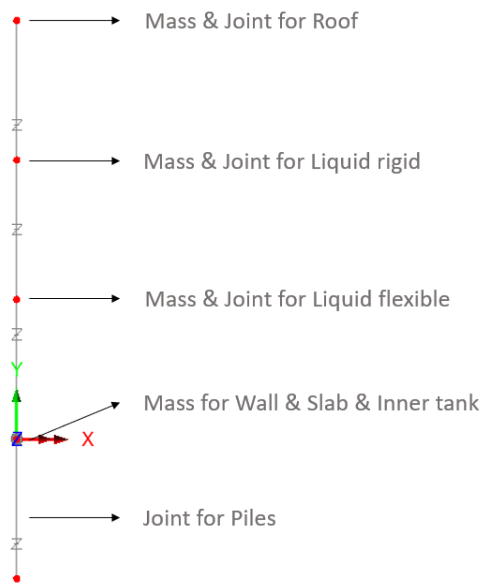


Fig 48 Beam-Stick Model for Vertical Actions

Geometric Properties

The following dataset is used.

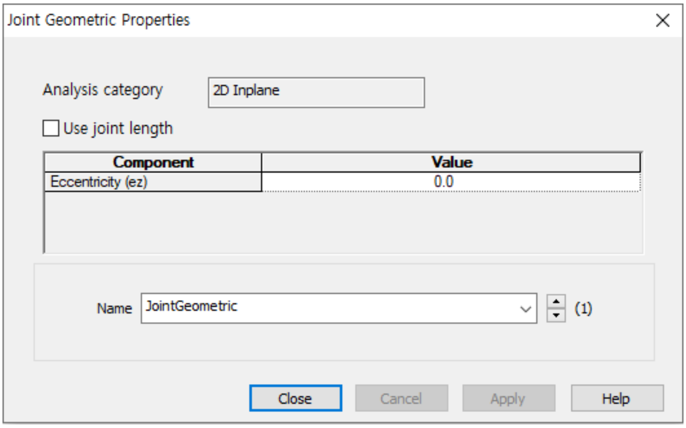


Fig 49 Geometric Properties for Joint Elements for Beam-Stick Vertical model

Material Properties

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

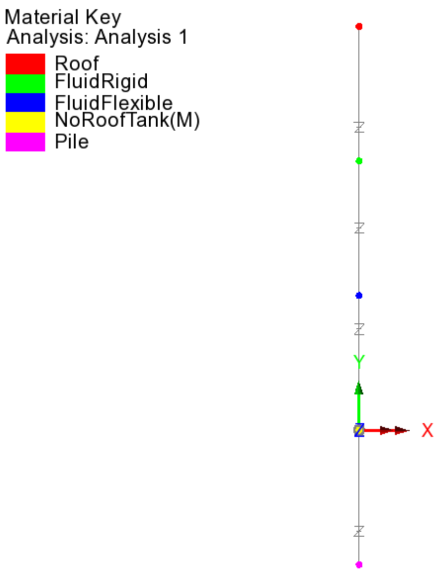


Fig 50 Material Properties in Beam-Stick Vertical Model

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

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	0.2		poisson ratio of steel
s(ζ)	0.0361	m	wall thickness for ζ = 1/3 (ζ = z/H_L)
f(γ)	1.0565		0.8<=γ<4 : 1.078+0.274 ln (γ) , γ<0.8 : 1 (A.41a, 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}	2.4081	Hz	fundamental frequency of oscillation of the liquid
T_{vd}	0.4153	s	fundamental period of oscillation of the liquid
$k_{\_LNG\_f}$	20,504,603,004	N/m	
$k_{\_LNG\_r}$	20,504,603,003,538,400	N/m	

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

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

3) Mass for Outer&Inner Tank

Component	Value	Unit	Remark
$m_{\_OuterInnerTank}$	29,461,083	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 69].

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

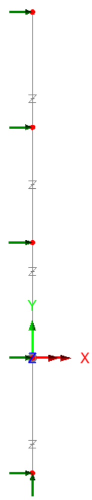


Fig 53 Supports in a Beam-Stick Vertical Model

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

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

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

Exporting Forces from the 2D Axisymmetric Model

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

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

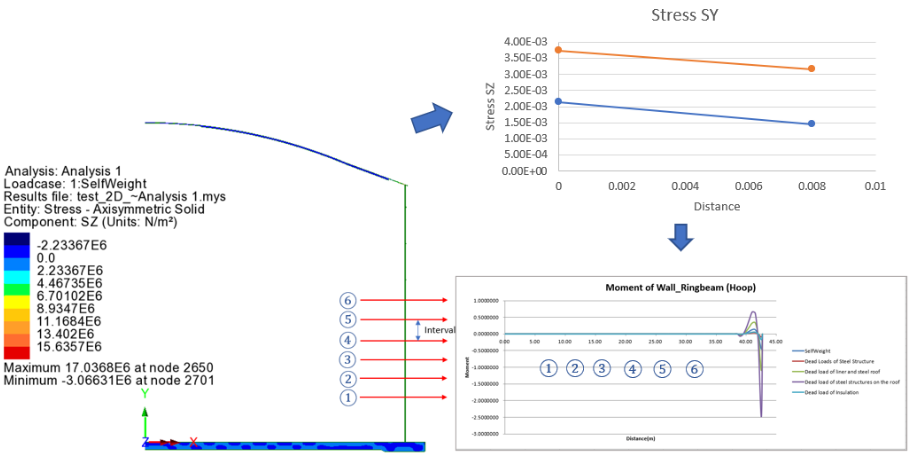


Fig 54 Converting Stress to Forces

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

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

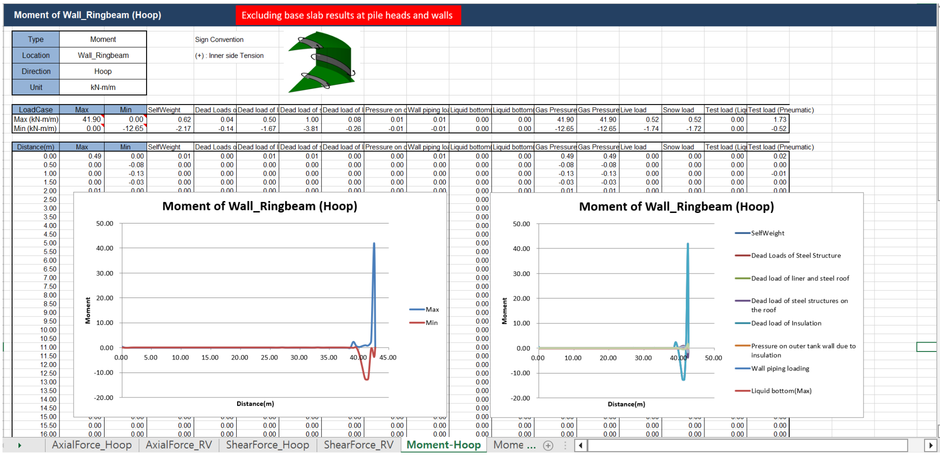


Fig 55 Section Force Spreadsheet for 2D Axisymmetric Solid Model

Roof - Exporting Forces

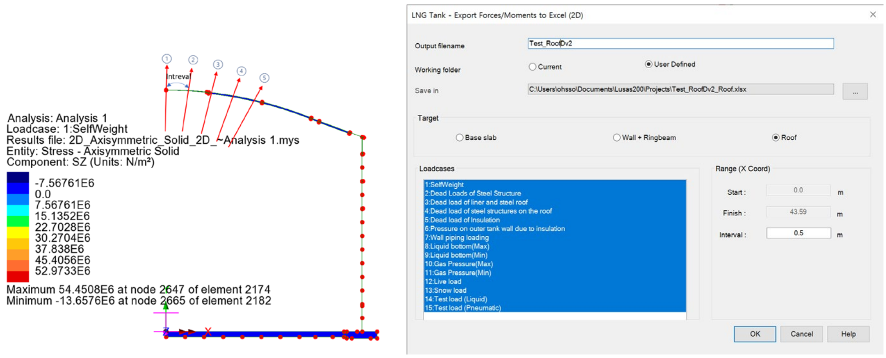


Fig 56 Exporting Forces for a 2D Axisymmetric Solid Model (Roof)

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

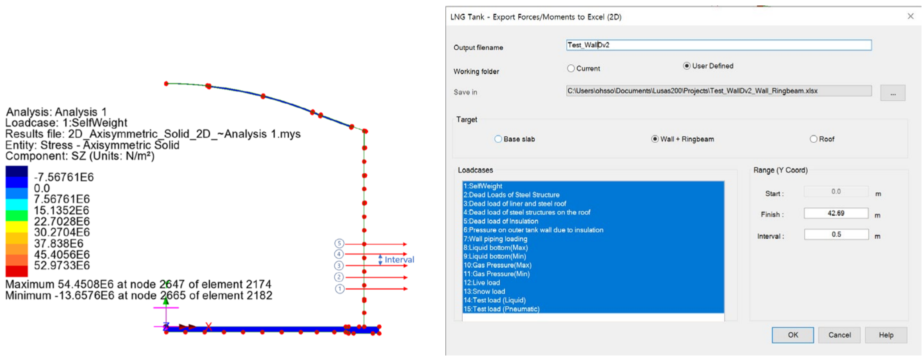


Fig 57 Exporting Forces for a 2D Axisymmetric Solid Model (Wall)

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

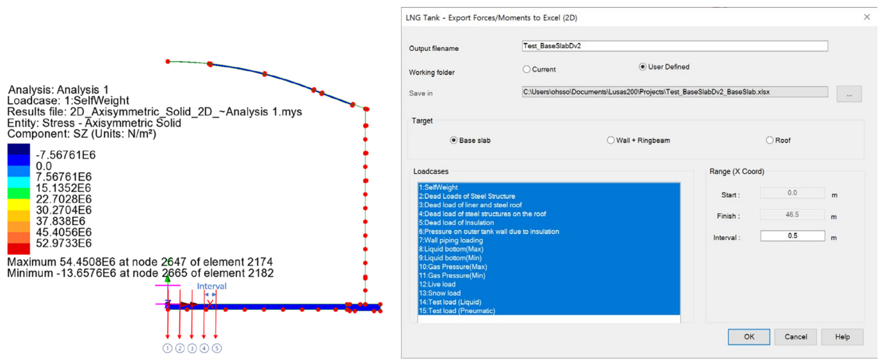


Fig 58 Exporting Forces for a 2D Axisymmetric Solid Model (Base Slab)

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.

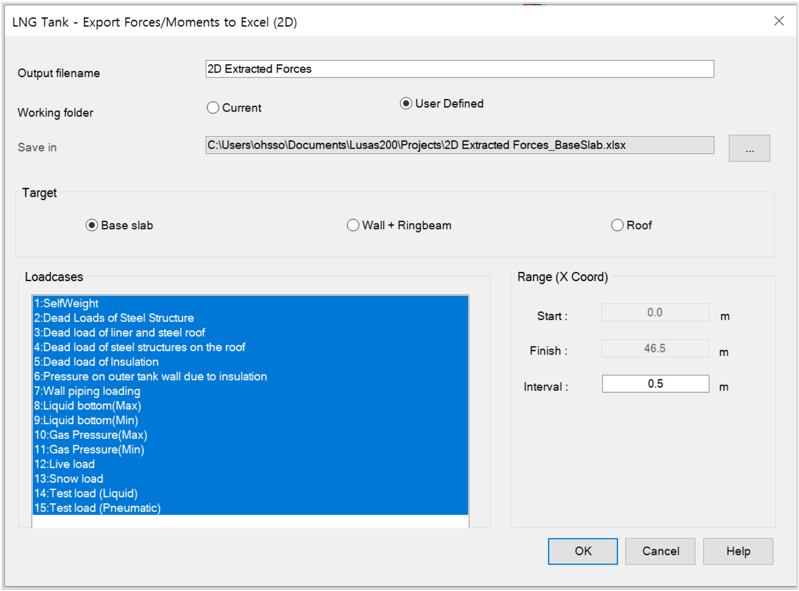


Fig 59 Exporting Forces for a 2D Axisymmetric Solid Mode

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**.

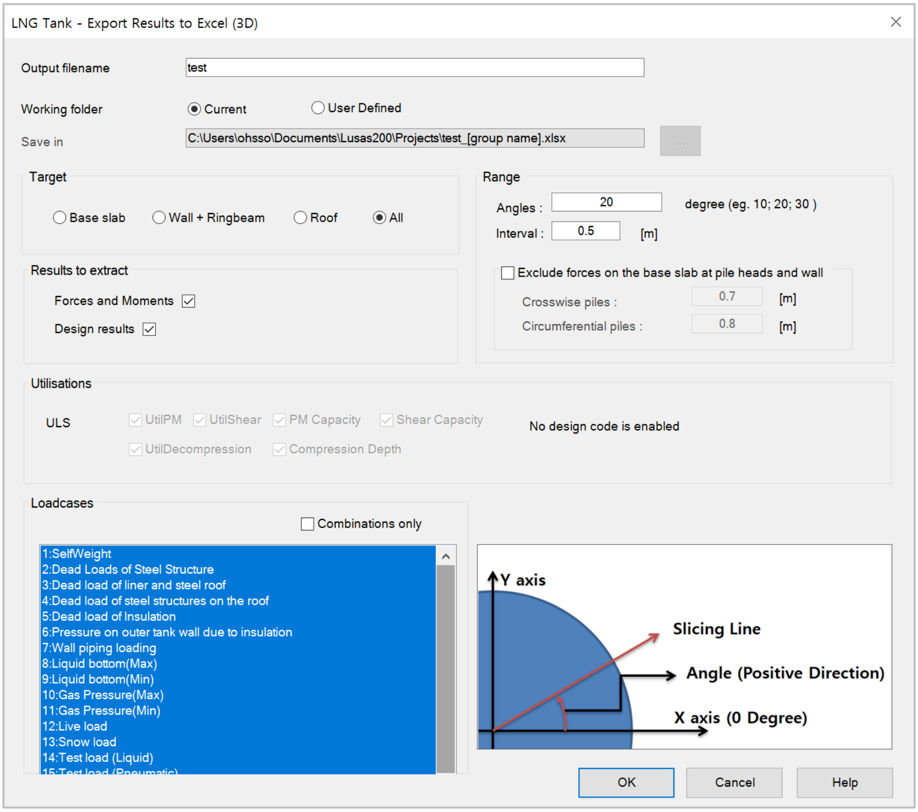


Fig 60 Exporting Forces for a 3D Shell Model

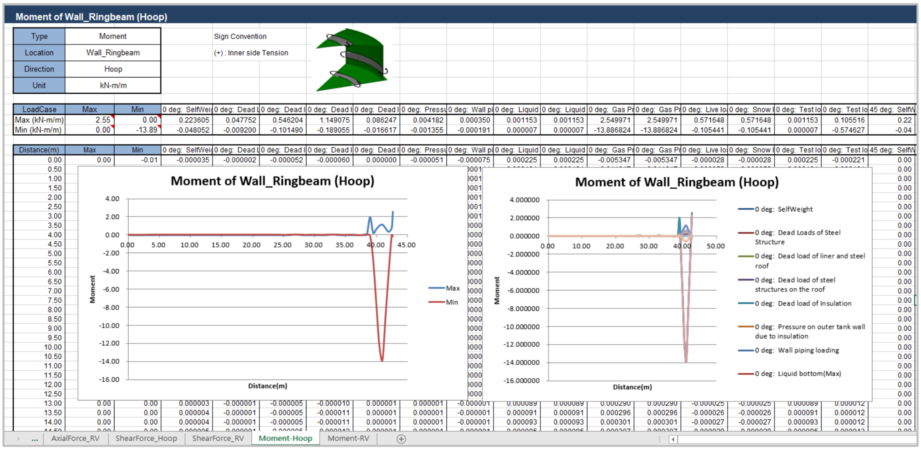


Fig 61 Section Forces Exported from a 3D Shell Model

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.

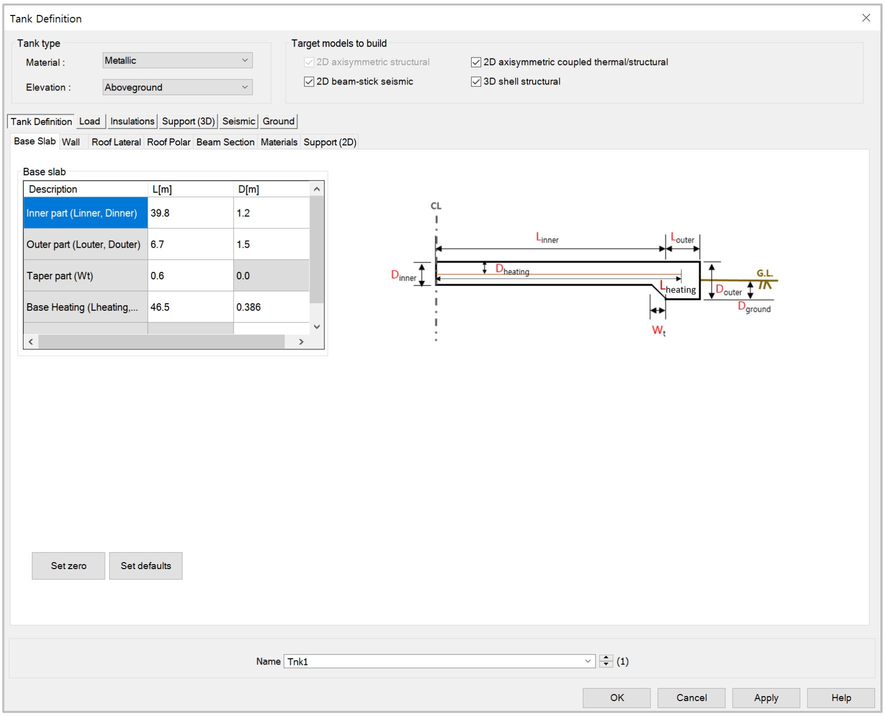


Fig 62 Tank Definition Dialog

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

Structural Definition

Metallic Tank

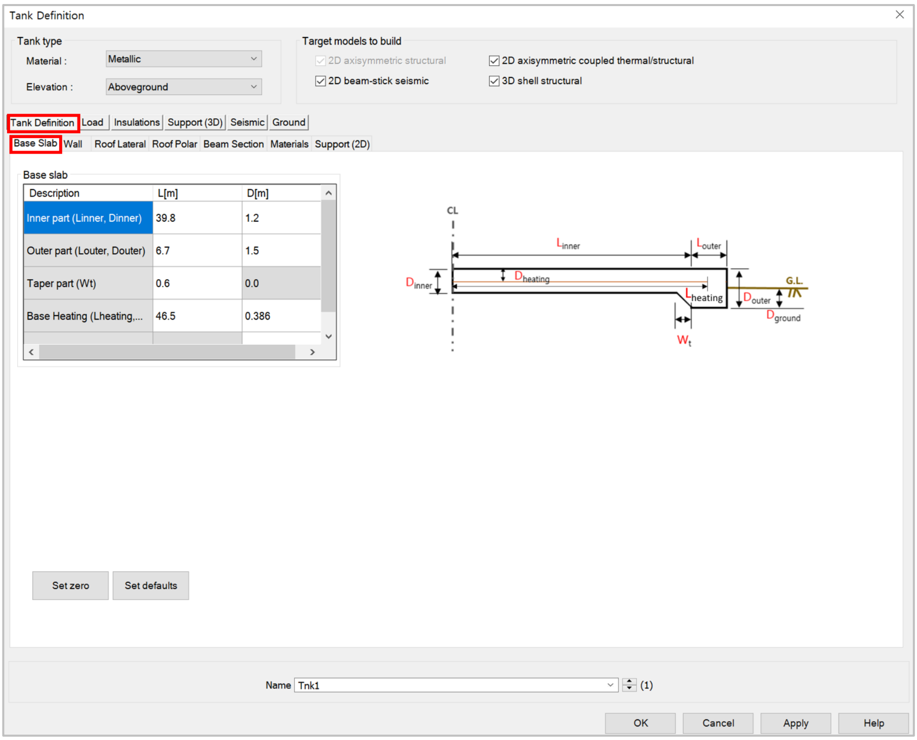
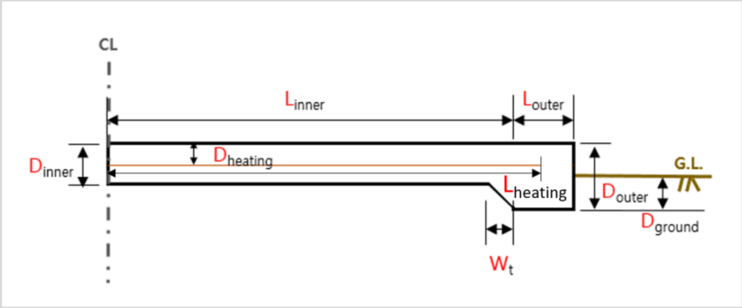


Fig 63 Tank Definition Dialog (Tank Definition/ Base Slab)

Base Slab

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



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

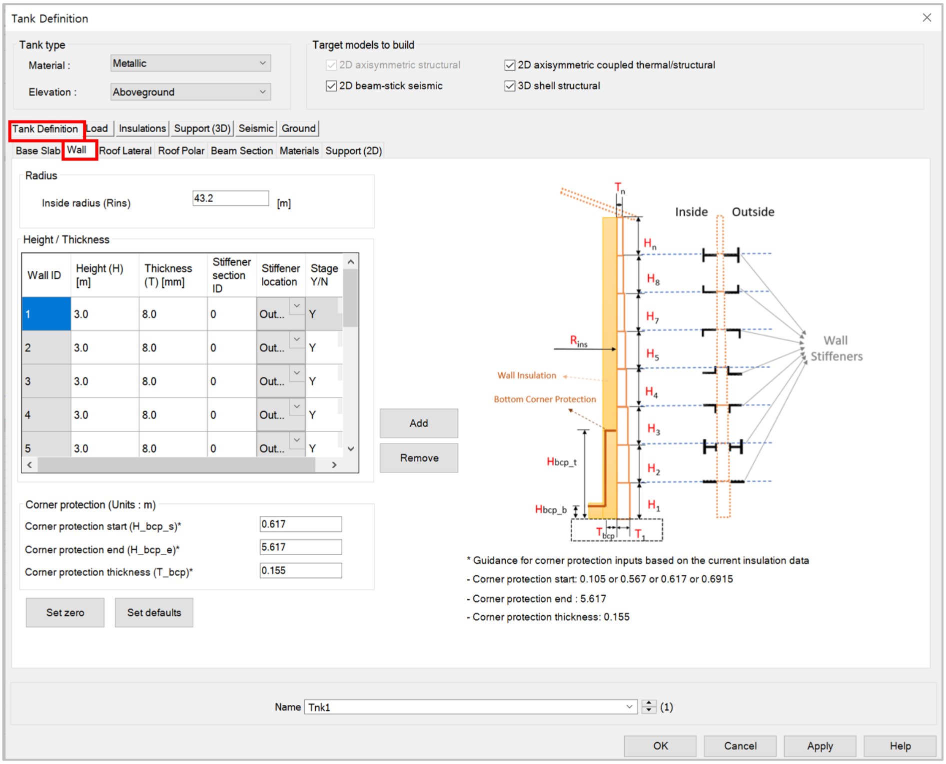
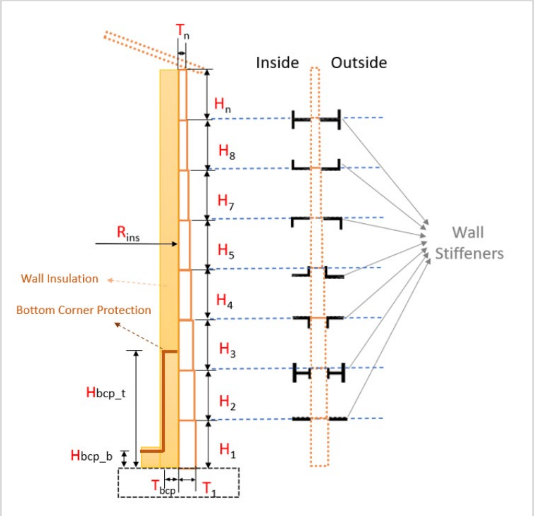


Fig 64 Tank Definition Dialog (Tank Definition/ Wall)

Wall

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



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

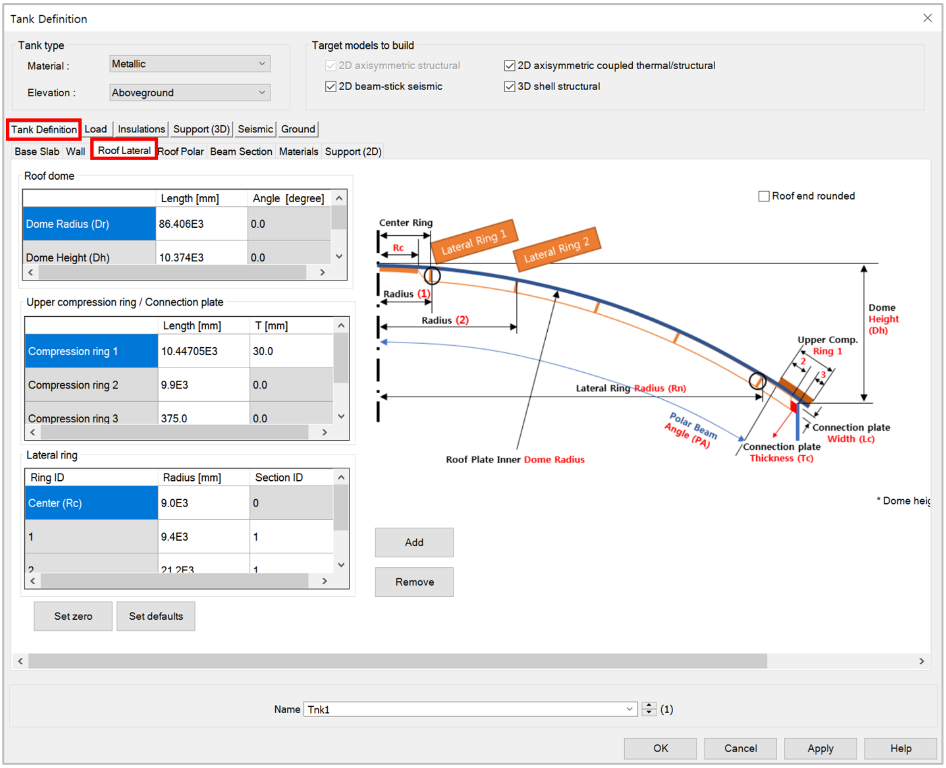
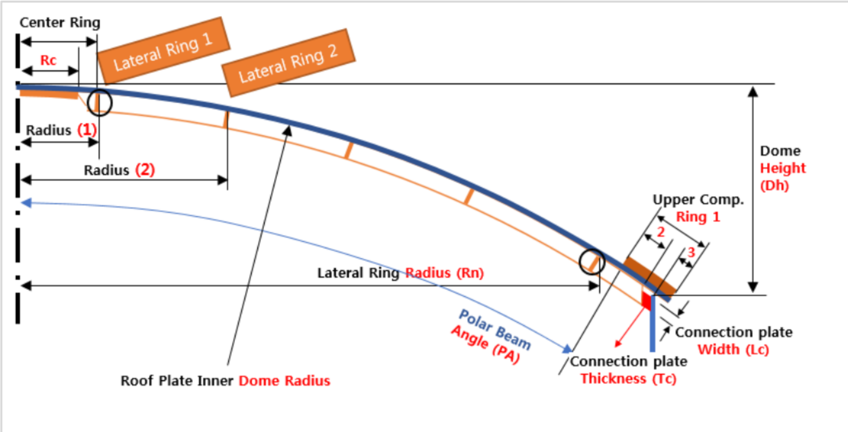


Fig 65 Tank Definition Dialog (Tank Definition/ Roof Lateral)

Roof Lateral

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



Roof dome

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

Upper compression ring/Connection plate

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

Lateral Ring

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

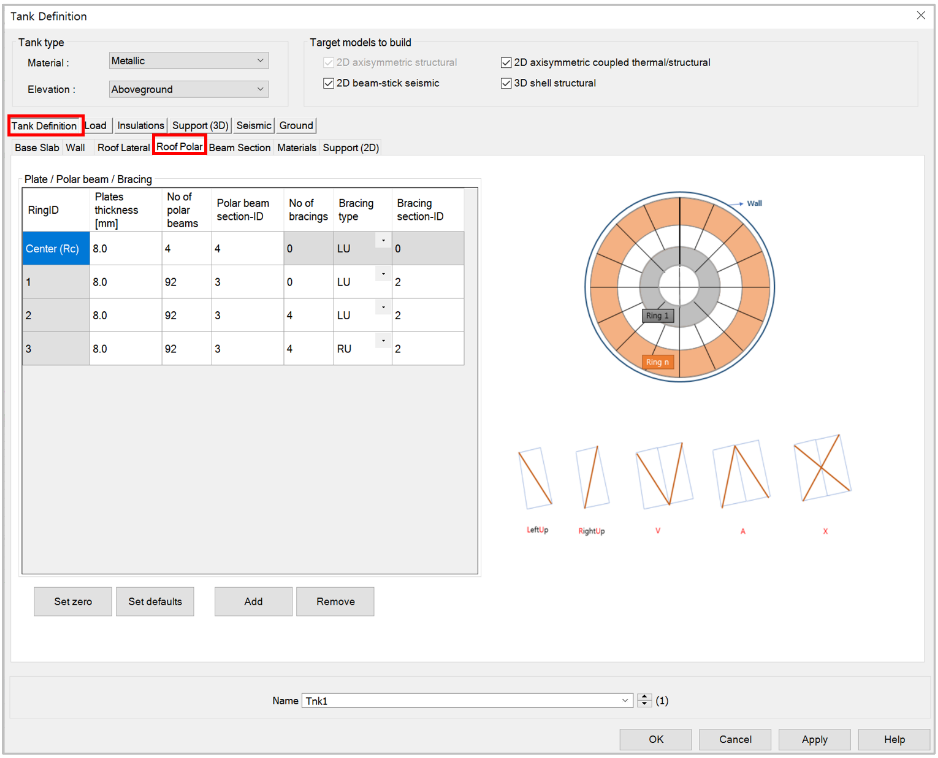
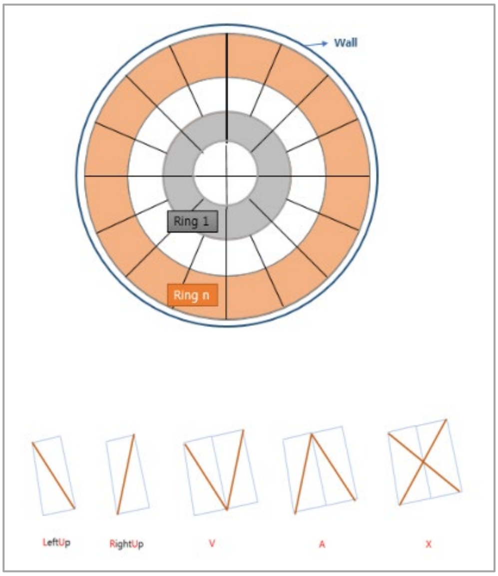


Fig 66 Tank Definition Dialog (Tank Definition/ Roof Polar)

Roof Polar

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



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

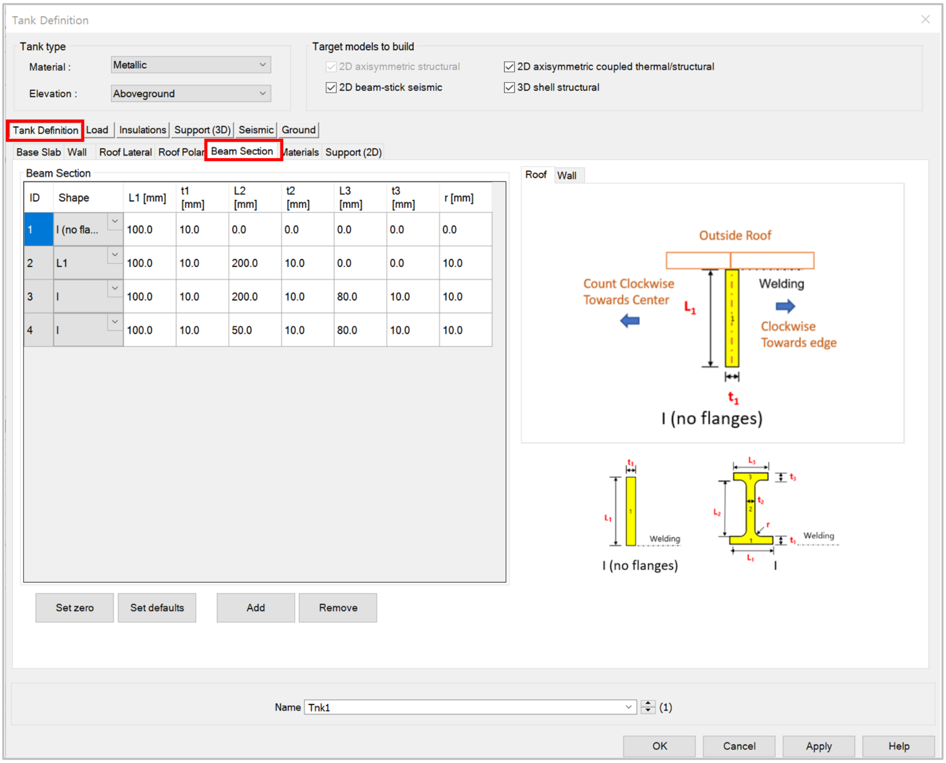
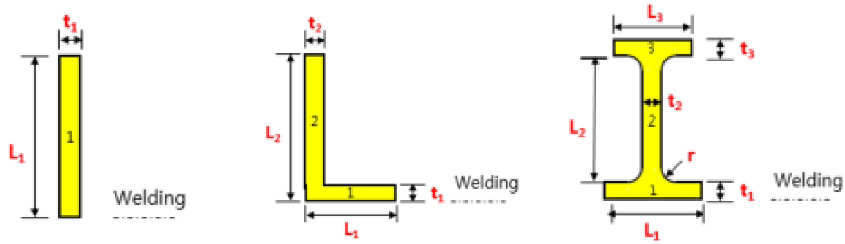


Fig 67 Tank Definition Dialog (Tank Definition/ Beam Section)

Beam Section

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



I without flange L I with flange

- **Shape:** Defines the shape of section

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

Insulation

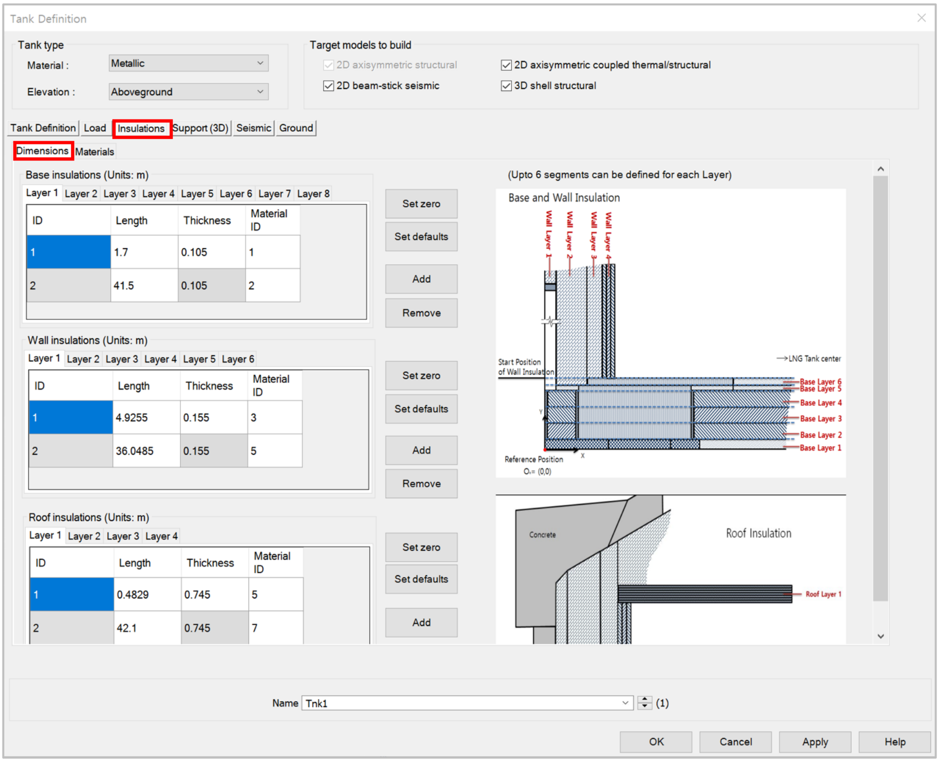


Fig 68 Tank Definition Dialog (Tank Definition/ Insulation)

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

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

Base Insulation

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

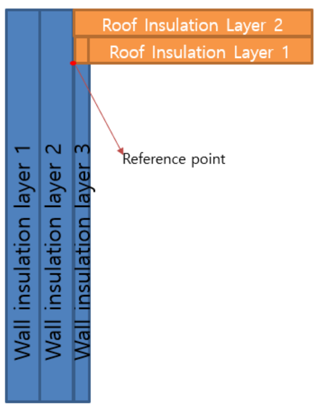
Wall Insulation

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

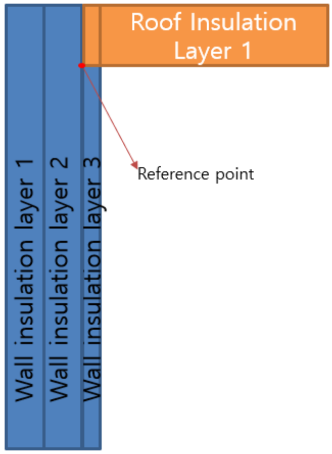
Roof insulation

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

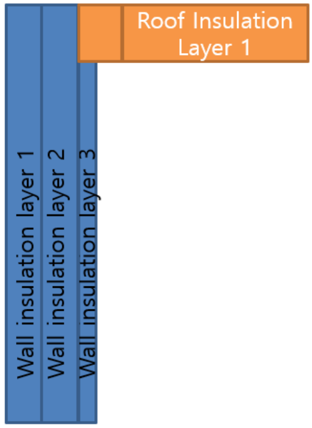
- The top-left point of the innermost layer of wall insulation is used as the 'reference point' for modelling the roof insulation.
- The sum of the total length of the roof insulation for a layer and the total thickness excluding the last layer for the wall insulation should be equal to inner diameter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



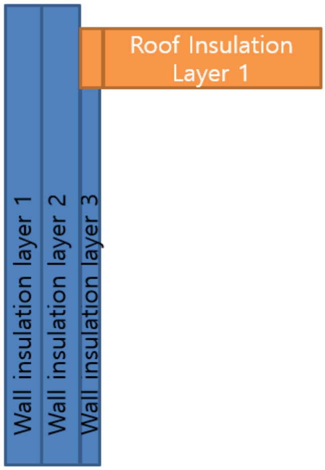
Several examples of defining wall and roof insulation follow:



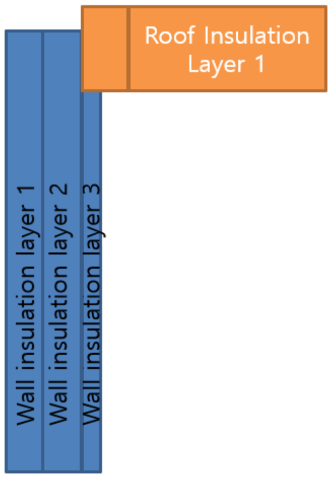
Case 1



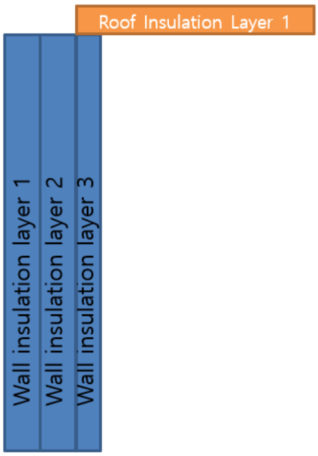
Case 2



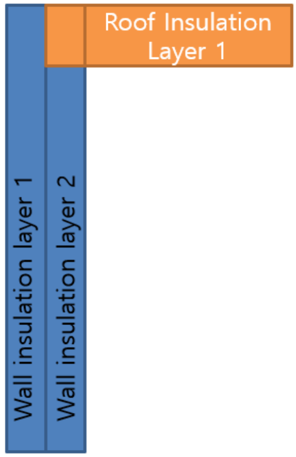
Case 3



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

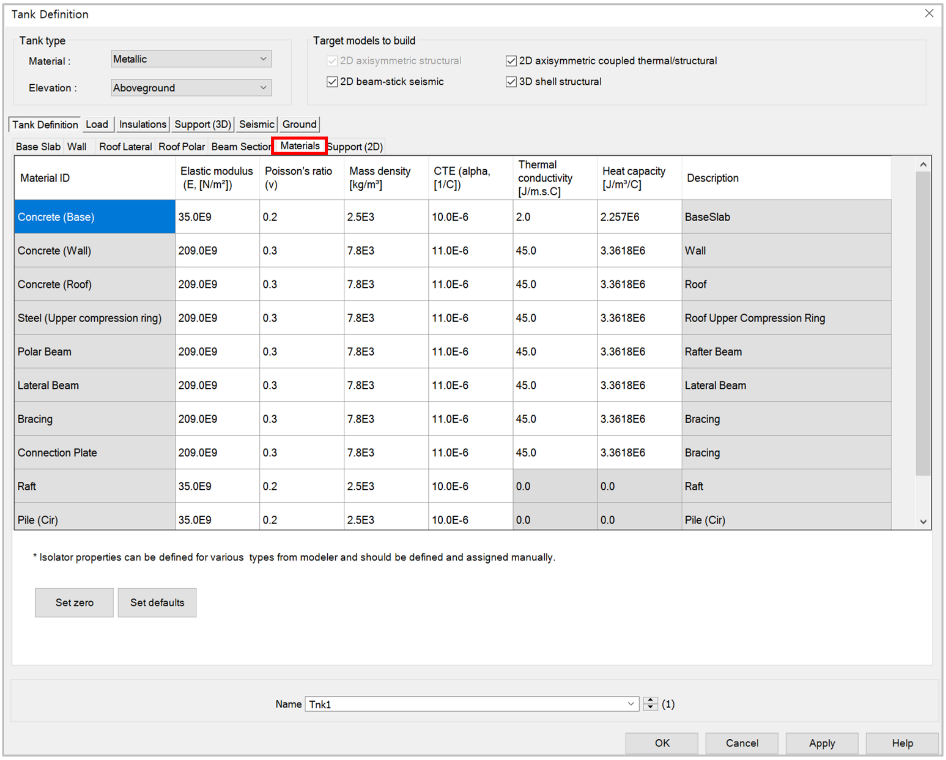
- Length of wall insulation layer1/Layer2 and Layer3 are identical
- Thickness of wall insulation layer3 < the length of the 1st segment of roof insulation layer1

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



Material ID	Elastic modulus (E, [N/m²])	Poisson's ratio (v)	Mass density [kg/m³]	CTE (alpha, [1/C])	Thermal conductivity [J/m.s.C]	Heat capacity [J/m³/C]	Description
Concrete (Base)	35.0E9	0.2	2.5E3	10.0E-6	2.0	2.257E6	BaseSlab
Concrete (Wall)	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Wall
Concrete (Roof)	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Roof
Steel (Upper compression ring)	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Roof Upper Compression Ring
Polar Beam	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Rafter Beam
Lateral Beam	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Lateral Beam
Bracing	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Bracing
Connection Plate	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Bracing
Raft	35.0E9	0.2	2.5E3	10.0E-6	0.0	0.0	Raft
Pile (Cir)	35.0E9	0.2	2.5E3	10.0E-6	0.0	0.0	Pile (Cir)

Fig 69 Tank Definition Dialog (Tank Definition–Materials)

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

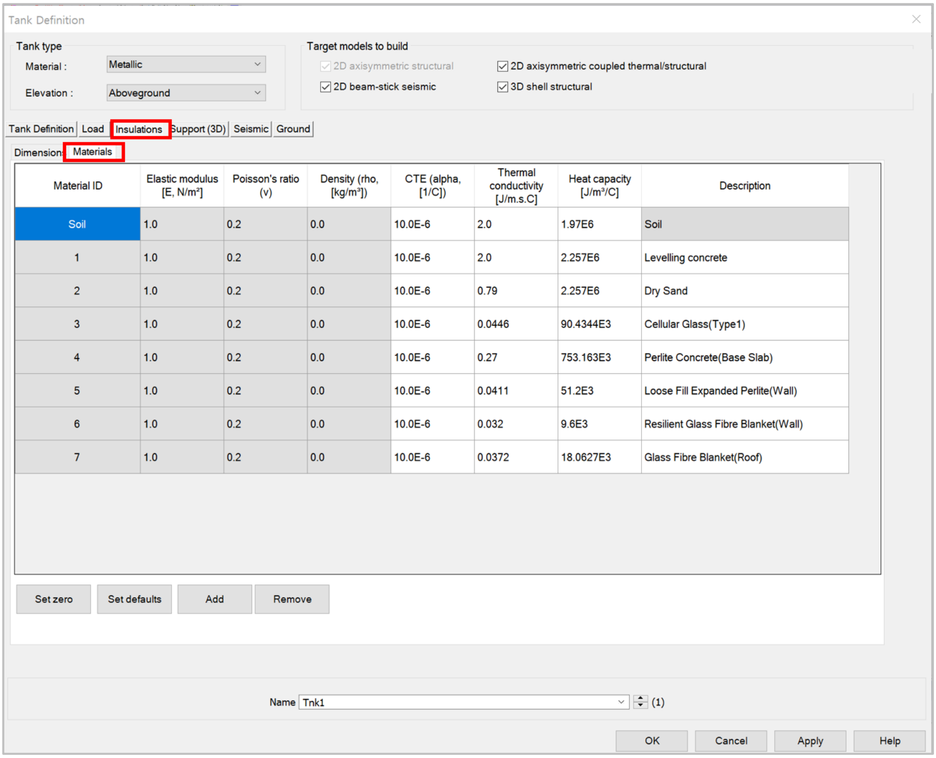


Fig 70 Tank Definition Dialog (Insulations / Materials)

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

Boundary Conditions

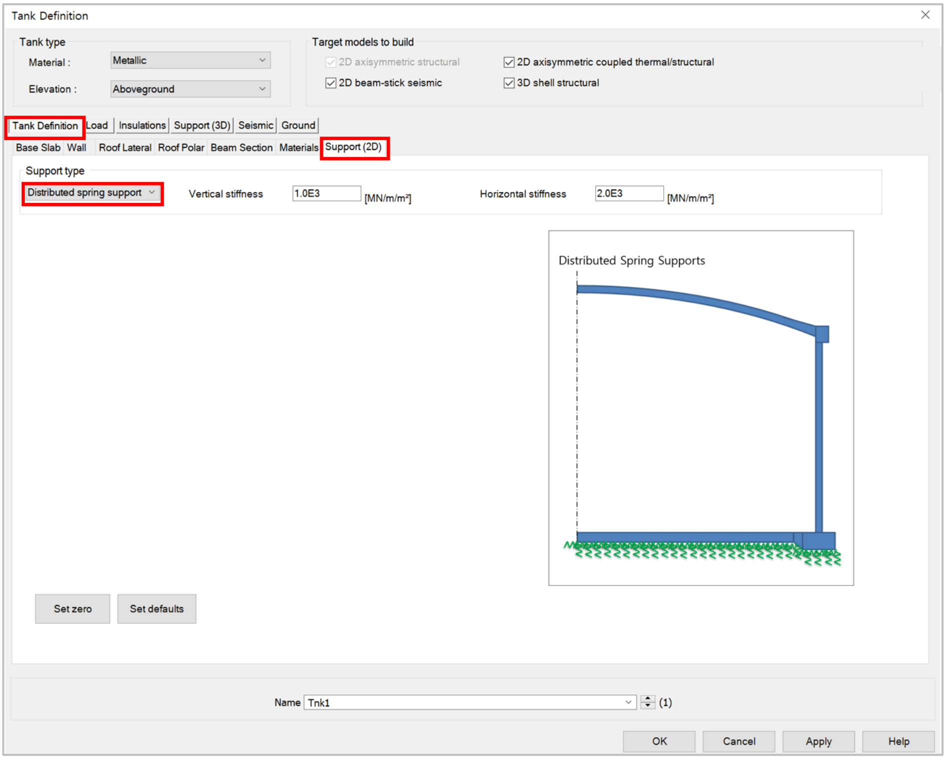


Fig 71 Tank Definition Dialog (Support(2D)- Distributed spring support)

Support Type

This tab defines the support type for the bottom of the base slab. Options are: 'Fixed Support', 'Pile Support' or 'Distributed spring support'. If 'Pile Support' is selected, the pile stiffness for spring supports should be defined and the unique ID numbers must include all of the pile stiffness. If 'Distributed spring support' is selected, one vertical and one horizontal stiffness should be defined. The stiffnesses should be a positive value in $MN/m/m^2$.

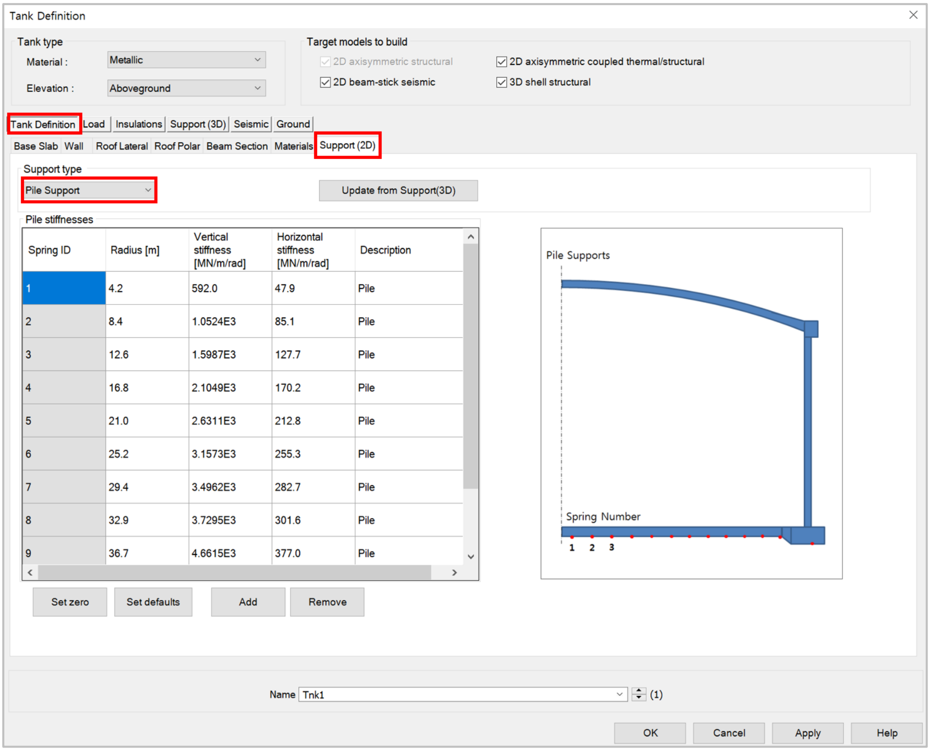


Fig 72 Tank Definition Dialog (Support(2D)- Pile Support)

Spring Stiffness for Piles

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

Loading

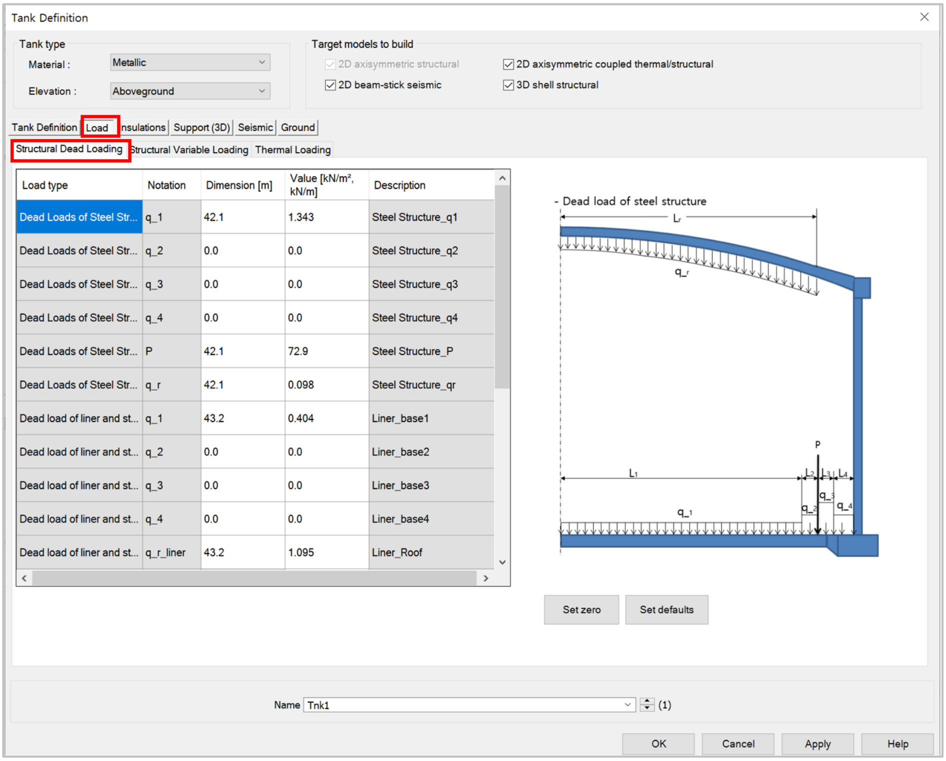


Fig 73 Tank Definition Dialog (Load– Structural Dead Loading)

Structural Dead Loading

This defines the structural dead loading to consider in analysis.

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

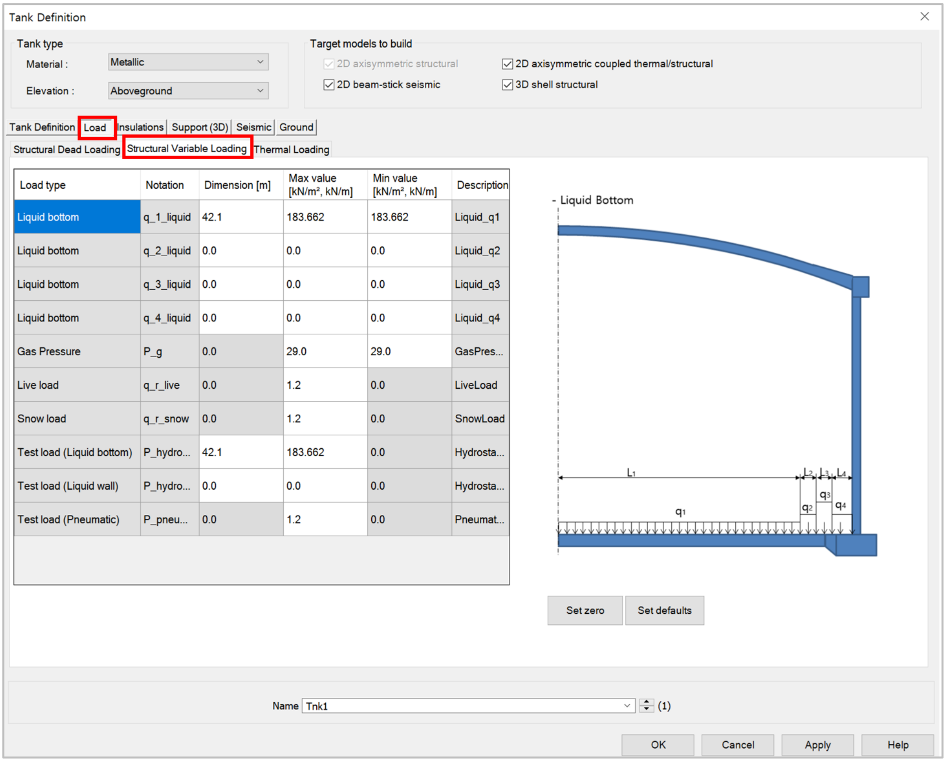


Fig 74 Tank Definition Dialog (Load – Structural Variable Loading)

Structural Variable Loading

Defines the structural variable loadings to consider in analysis.

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

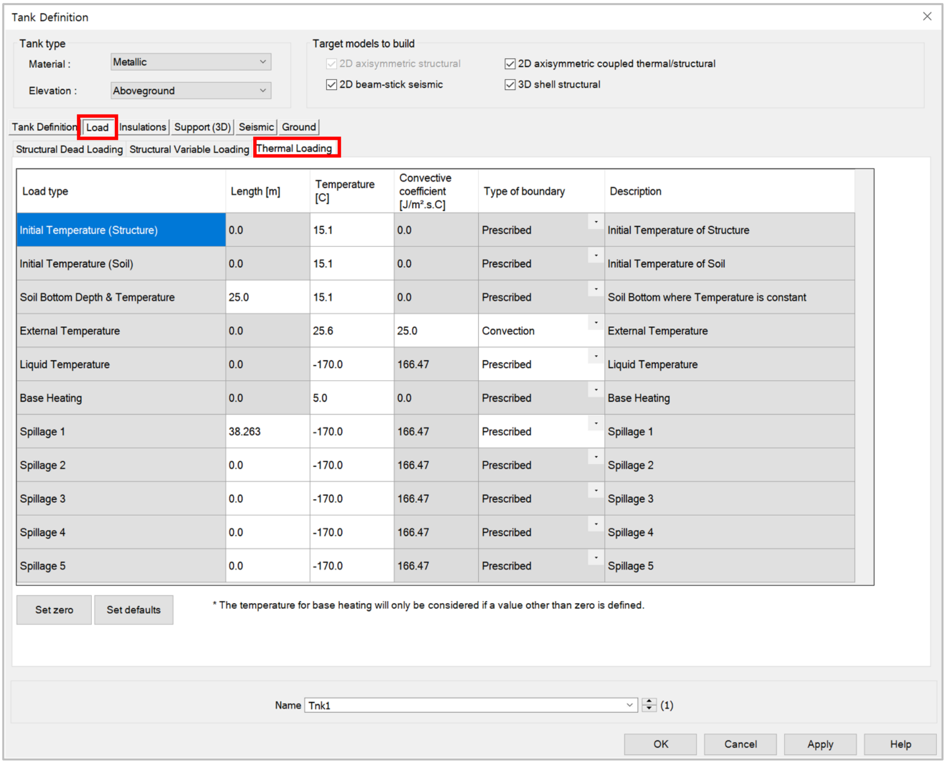


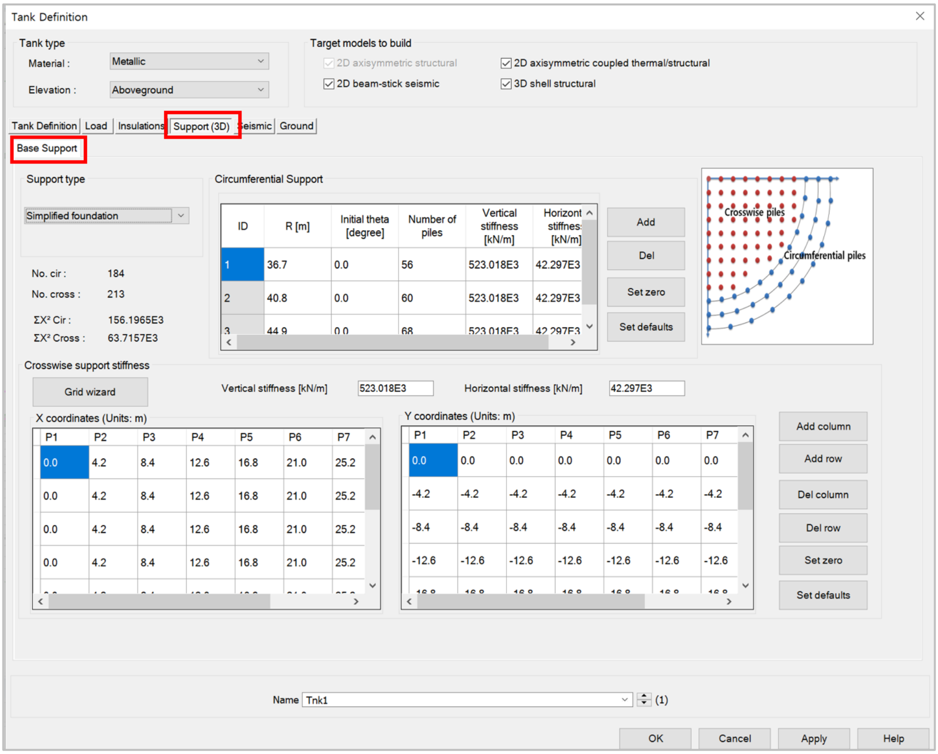
Fig 75 Tank Definition Dialog (Load – Thermal Loading)

Thermal Loading

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

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

Support (3D)



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 76 Tank Definition Dialog (Support (3D))

Support Type

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

Crosswise piles X Coordinates

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

Crosswise piles Y Coordinates

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

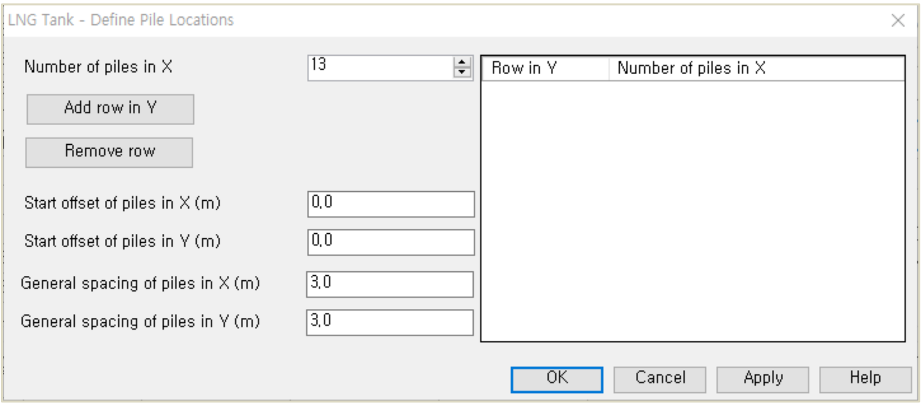


Fig 77 Tank Definition Dialog (Pile Arrangement – Define Pile Locations)

Define pile locations

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

Crosswise piles stiffness

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

Circumferential piles

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

Foundation

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

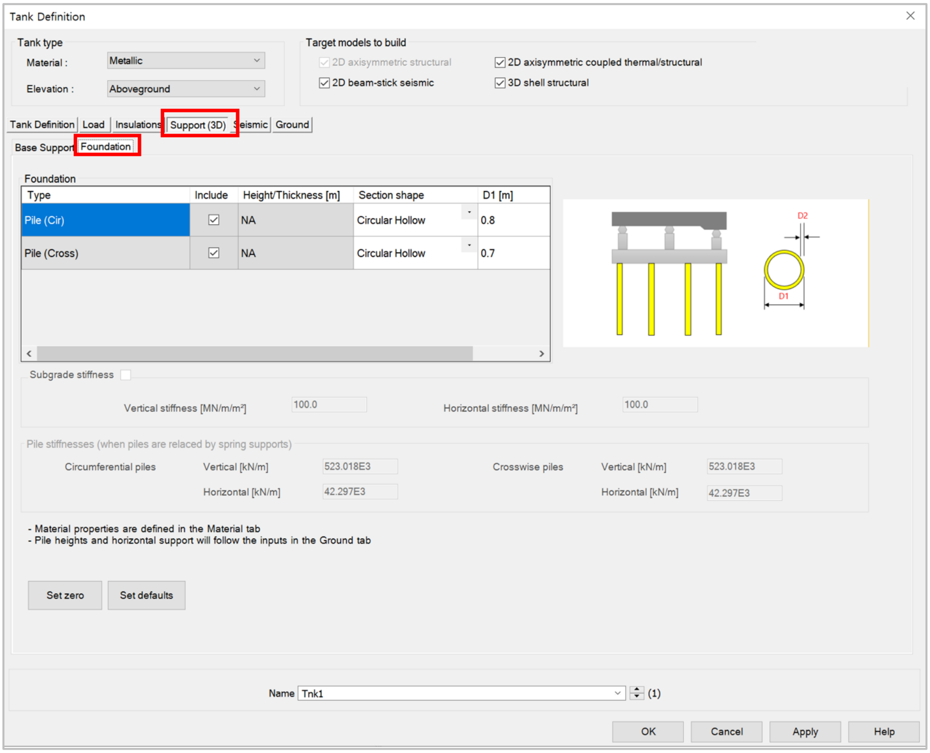


Fig 78 Tank Definition Dialog (Support (3D)>Foudation)

- ❑ **Foundation :** Defines the section shape and the dimensions of piles (Crosswise, Circumferential)

Seismic

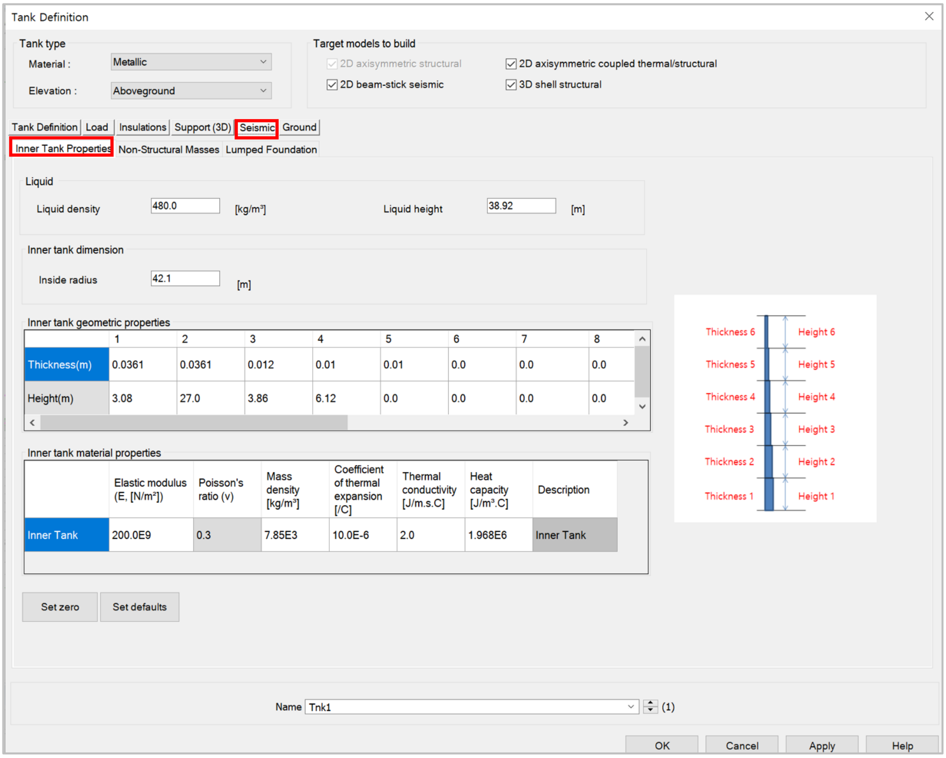


Fig 79 Tank Definition Dialog (Seismic – Inner Tank Properties)

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

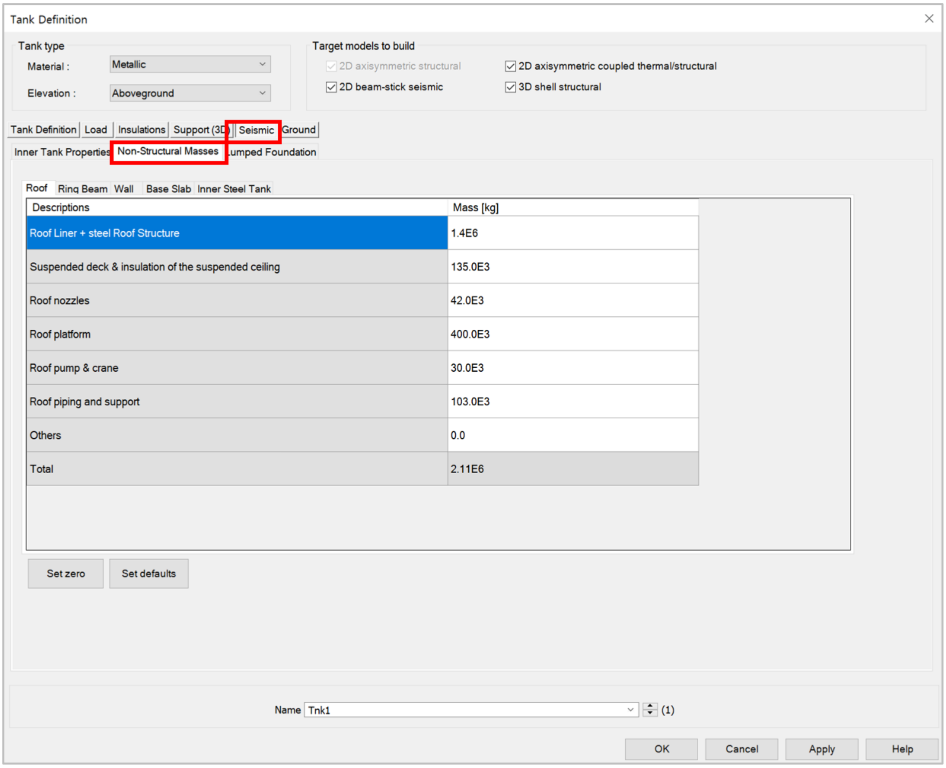


Fig 80 Tank Definition Dialog (Seismic - Non-Structural Masses)

Non-Structural Masses

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

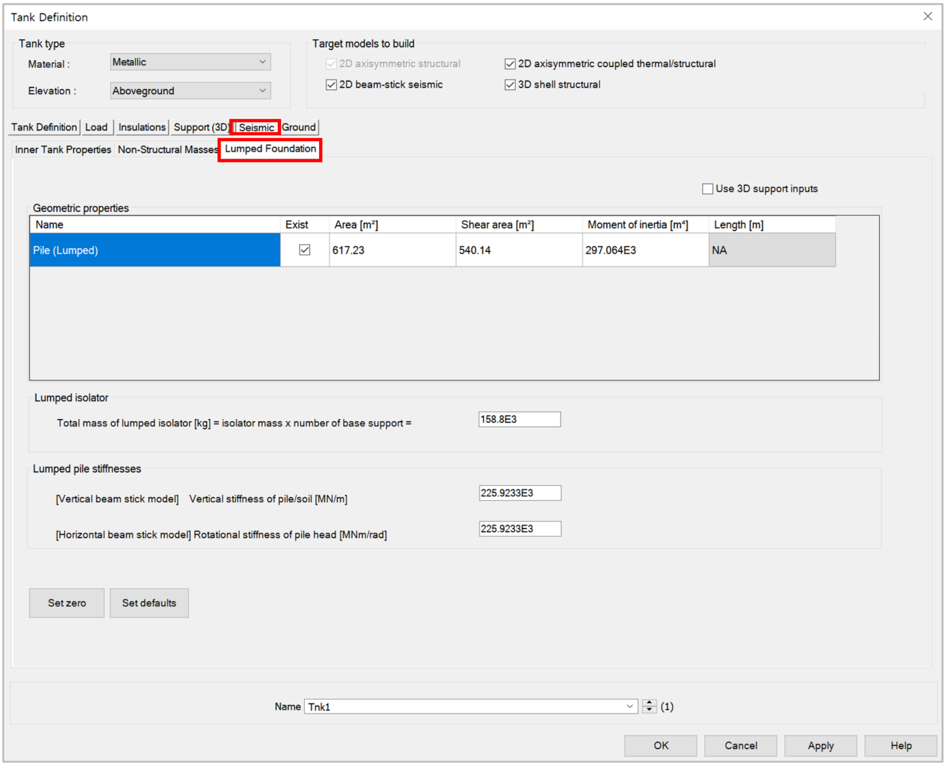


Fig 81 Tank Definition Dialog (Seismic – Limped Foundation)

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

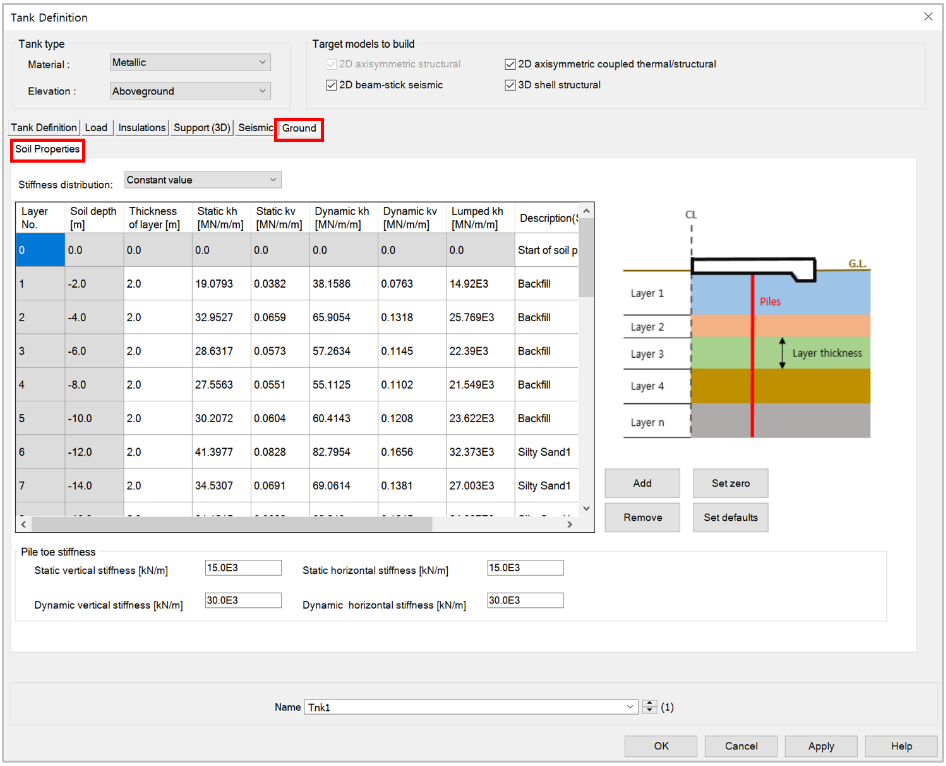


Fig 82 Tank Definition Dialog (Ground – Soil Properties)

Soil Properties

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

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

2D Axisymmetric Static Structural Analysis

User Inputs

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

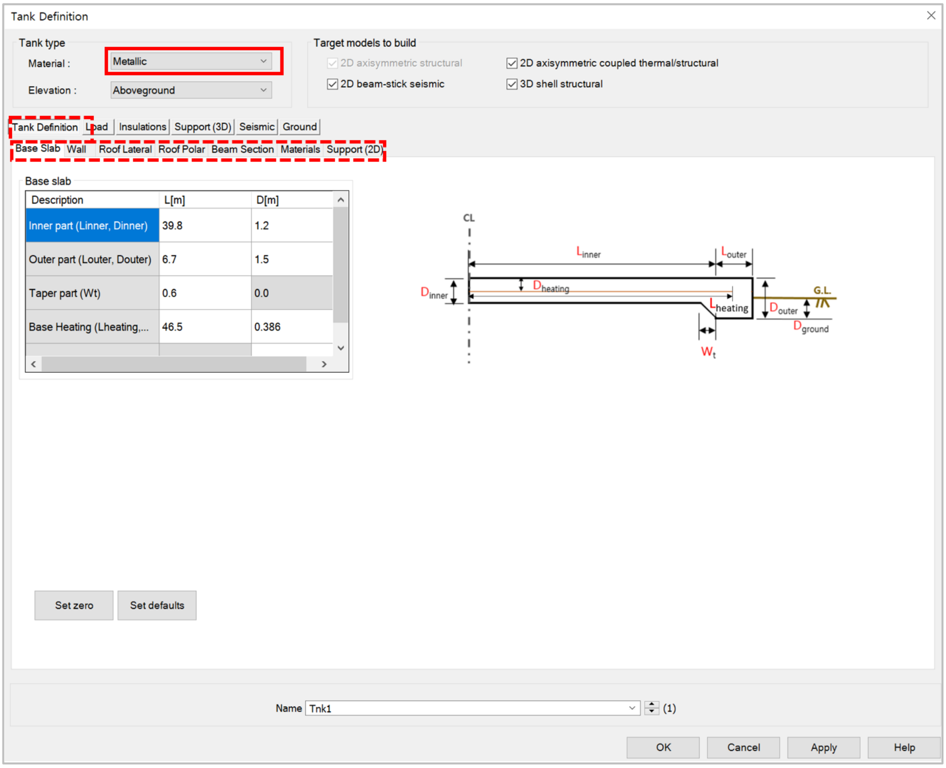


Fig 83 User Inputs for 2D Axisymmetric Static Analysis

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

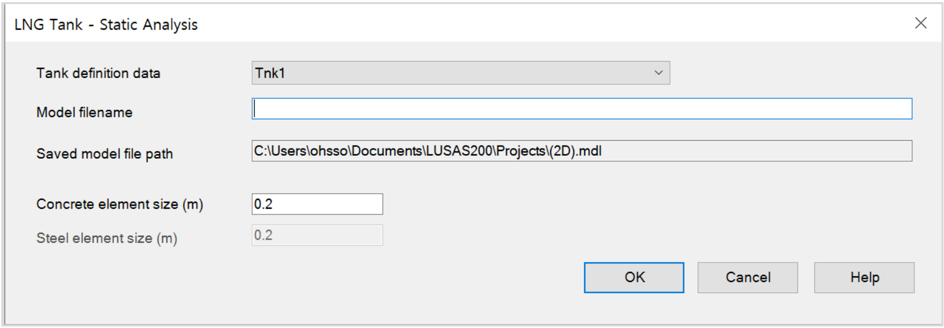


Fig 84 User Dialog for 2D Axisymmetric Static Analysis

Meshing

Element Type

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

Element Size

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

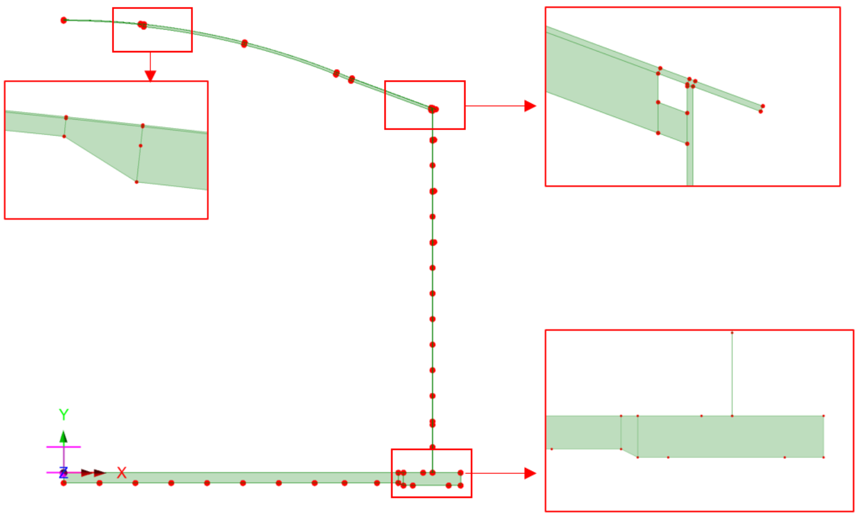


Fig 85 Mesh division for a 2D Axisymmetric Model

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

Geometric Properties

Roof Plate

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

Polar Beams

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

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

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

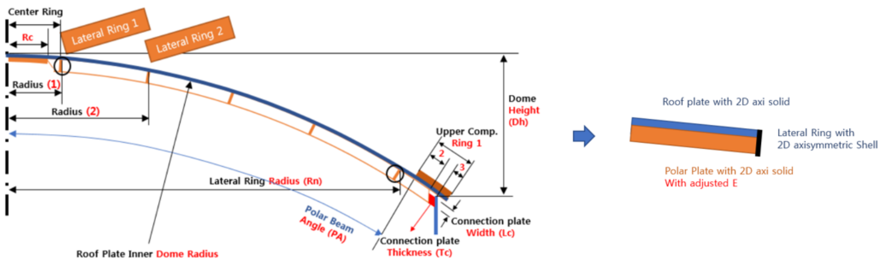


Fig 86 Polar beams for a 2D Axisymmetric Model

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

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

Bracing

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

Wall Stiffener

Wall stiffeners are modelled as defined in Tank Definition.

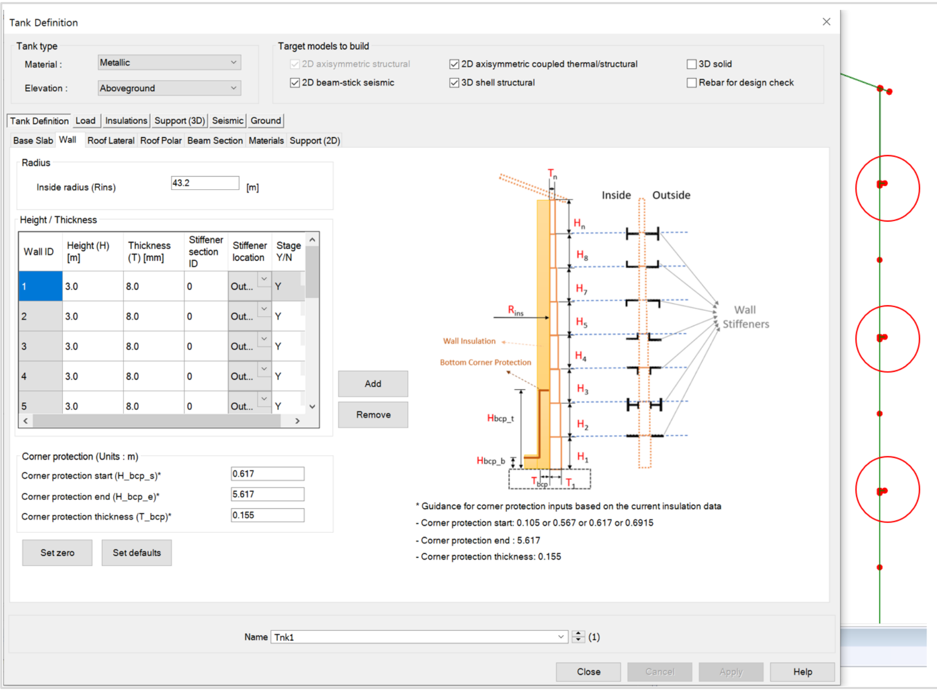


Fig 87 User input for Wall Stiffener for a 2D Axisymmetric Model

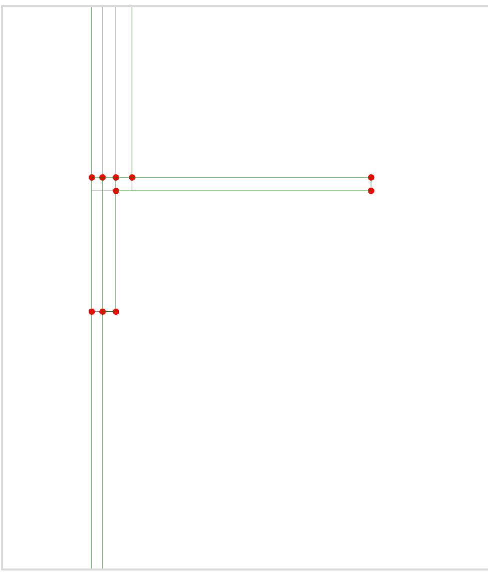
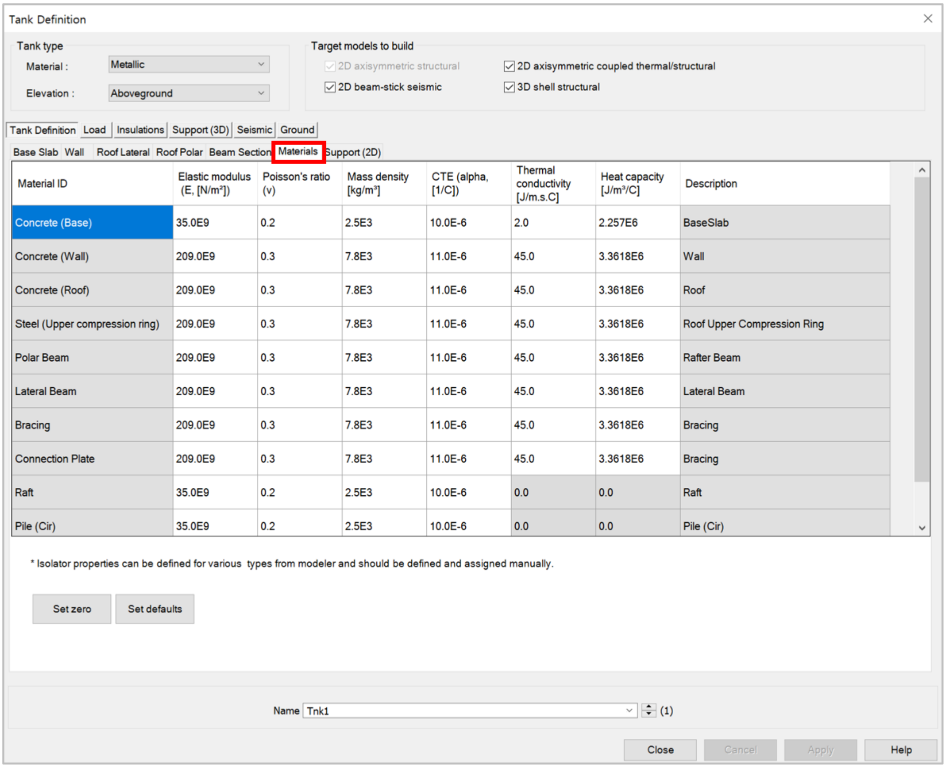


Fig 88 Geometry of Wall stiffener for a 2D Axisymmetric Model

Material Properties

User defined material properties are assigned to the relevant surfaces.



Material ID	Elastic modulus (E, [N/m²])	Poisson's ratio (v)	Mass density [kg/m³]	CTE (alpha, [1/C])	Thermal conductivity [J/m.s.C]	Heat capacity [J/m³/C]	Description
Concrete (Base)	35.0E9	0.2	2.5E3	10.0E-6	2.0	2.257E6	BaseSlab
Concrete (Wall)	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Wall
Concrete (Roof)	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Roof
Steel (Upper compression ring)	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Roof Upper Compression Ring
Polar Beam	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Rafter Beam
Lateral Beam	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Lateral Beam
Bracing	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Bracing
Connection Plate	209.0E9	0.3	7.8E3	11.0E-6	45.0	3.3618E6	Bracing
Raft	35.0E9	0.2	2.5E3	10.0E-6	0.0	0.0	Raft
Pile (Cir)	35.0E9	0.2	2.5E3	10.0E-6	0.0	0.0	Pile (Cir)

Fig 89 User Inputs for Tank Materials

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

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

Total mass of polar beams per Ring

The total mass of polar beams per ring is computed from

1) Volume of polar beams in a ring

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

2) Total mass of polar beam in a ring

   Mp = Vp * mass density of polar beam

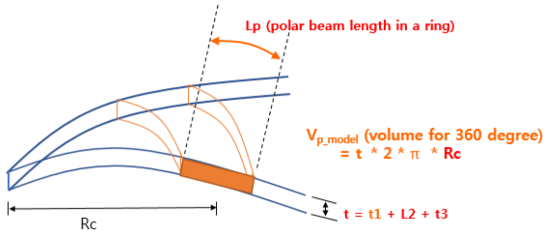
3) Volume of bracings in a ring
   Vb = No. of bracing per ring * Area of bracing section * length of bracing

4) Total mass of bracings in a ring
   Mb = Vb * mass density of bracing

Therefore, the total mass will be **Mp + Mb.**

Equivalent mass density of polar beams per Ring

The equivalent mass for the polar beam member incorporating bracings is computed as below;
mass_pb = (Mp + Mb) / (Vp, Total volume modelled)



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

Length of bracing beams

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

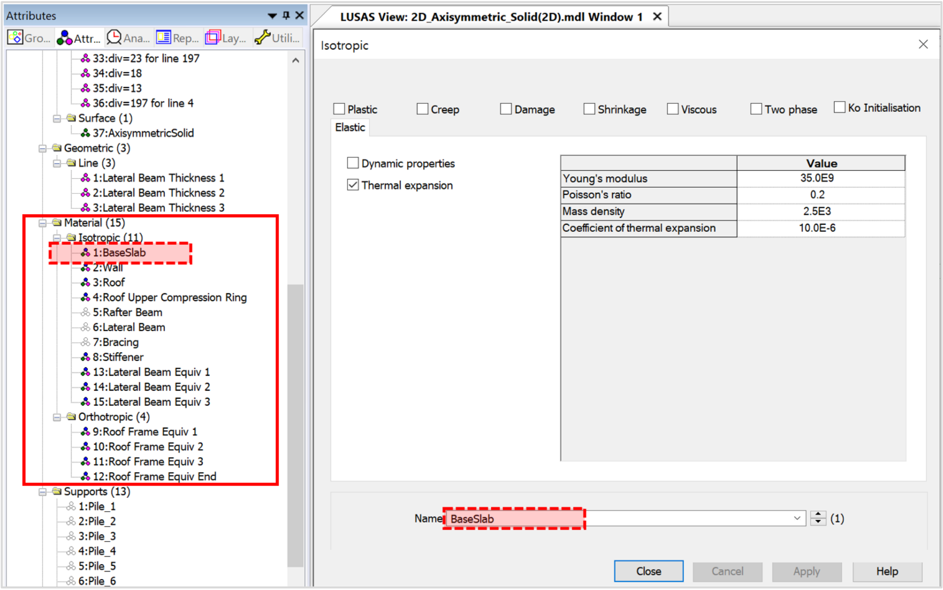


Fig 90 Material Properties for a 2D Axisymmetric Model

Support Conditions

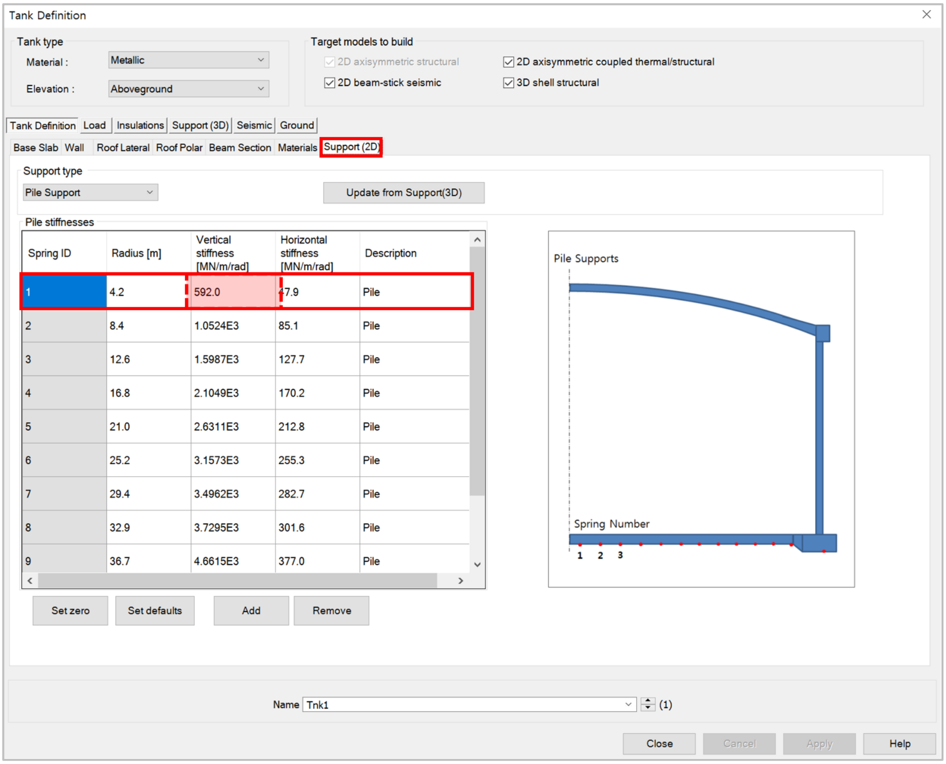


Fig 91 User Inputs for Boundary Conditions

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

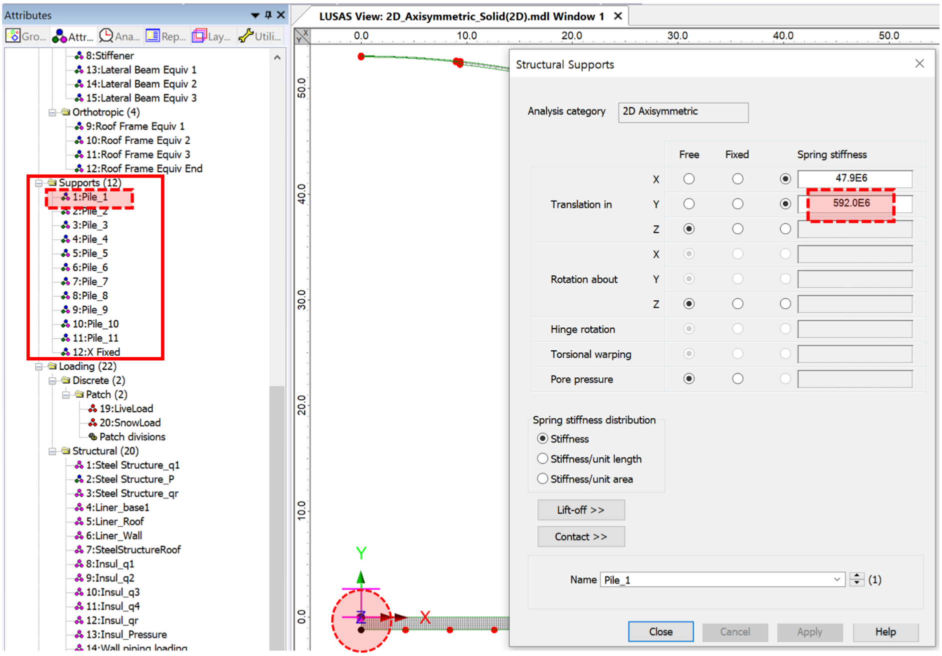


Fig 92 Pile Support for a 2D Axisymmetric Model

TEST CASE

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

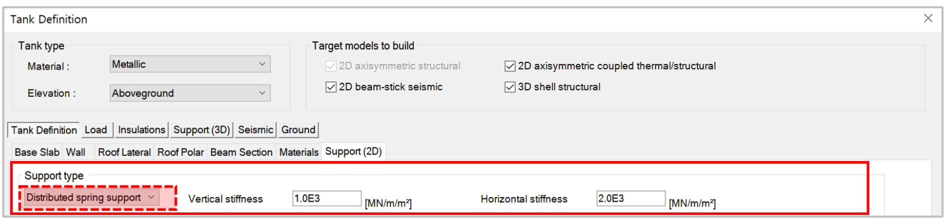


Fig 93 Test Case - Regular Support for a 2D Axisymmetric Model

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

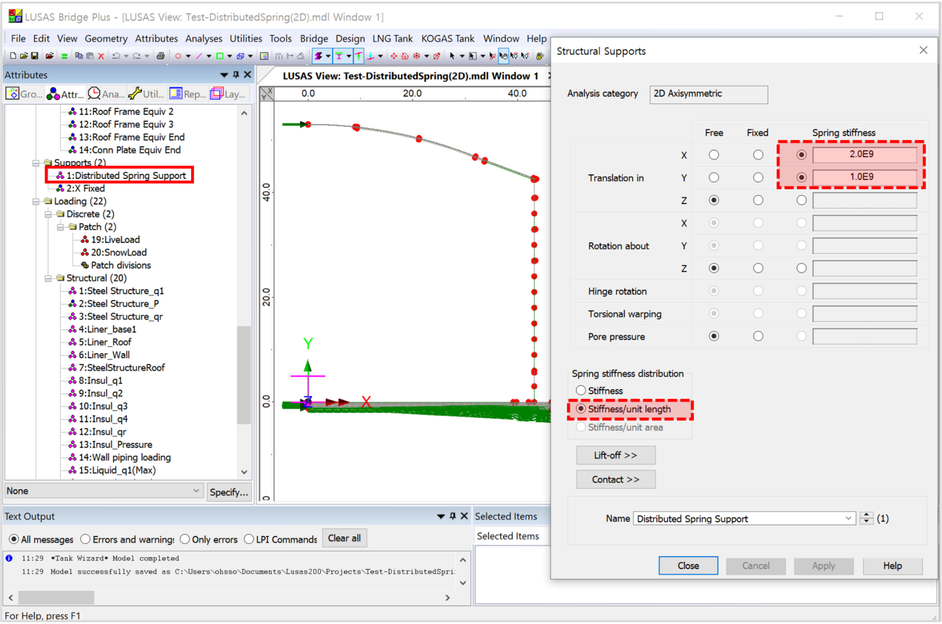


Fig 94 Test Case - Definition of a Regular Support for a 2D Axisymmetric Model

Loadings

A total of 15 loadcases is defined in the model.

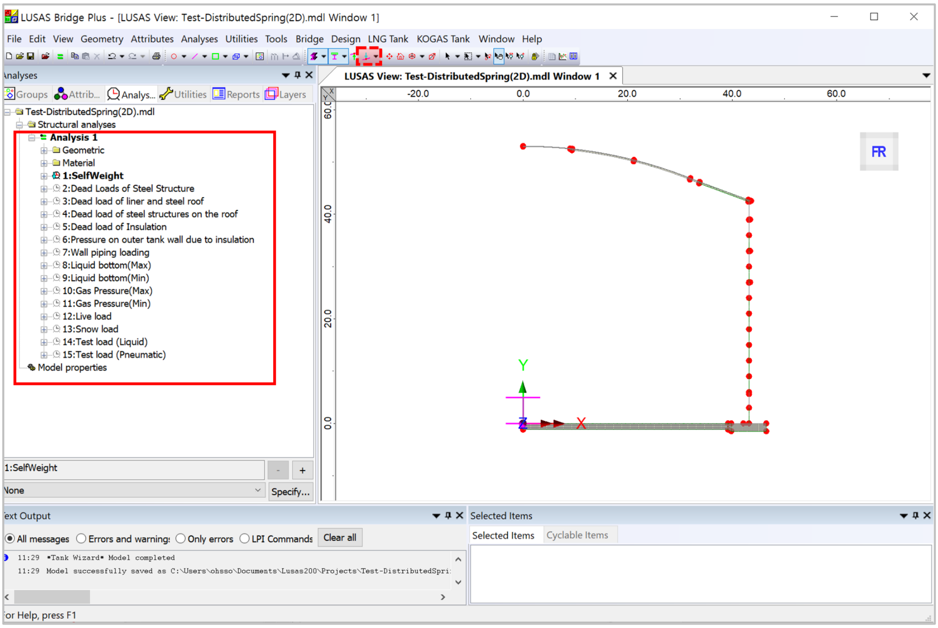


Fig 95 Loadcases available in a 2D Axisymmetric Static Analysis Model

Self Weight

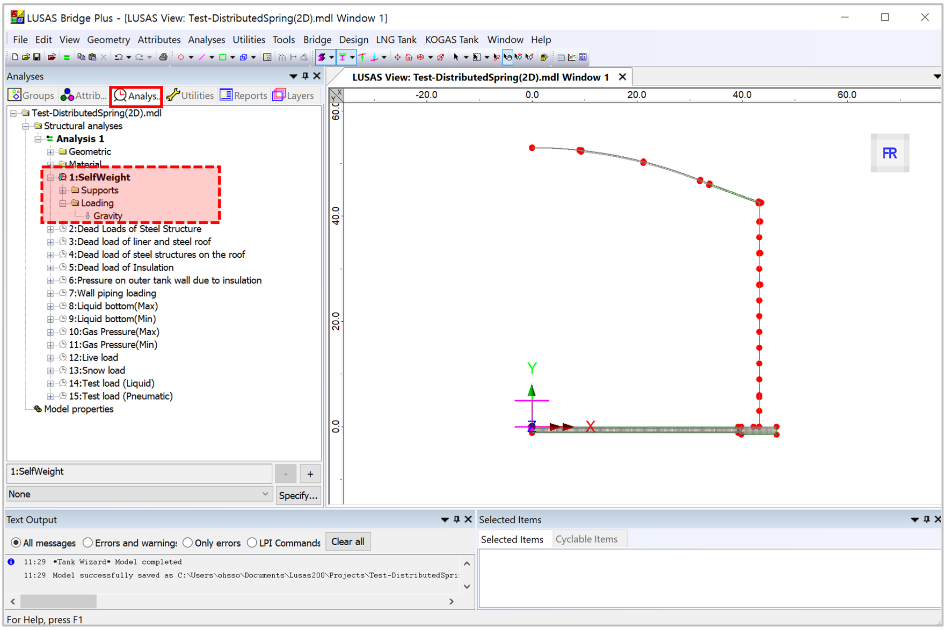


Fig 96 Self Weight in a 2D Axisymmetric Static Analysis Model

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.

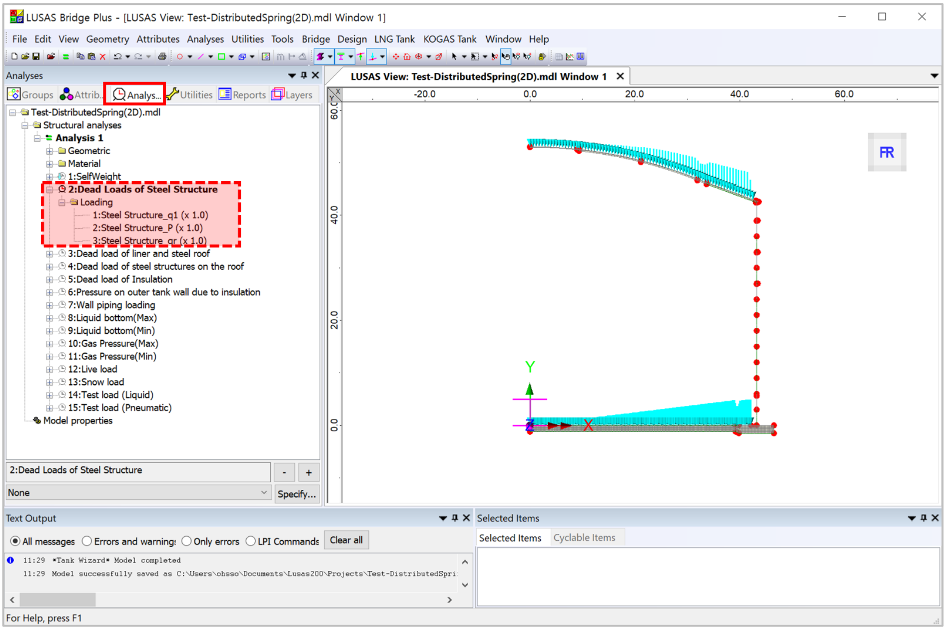


Fig 97 Dead Load for Steel Structure in a 2D Axisymmetric Static Analysis Model

Dead load of liner and steel roof

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

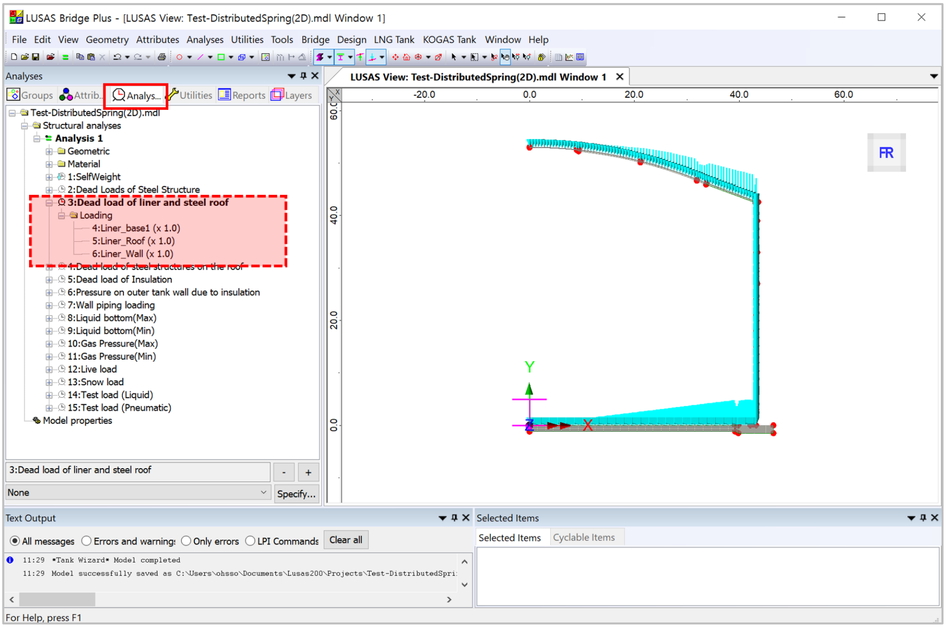


Fig 98 Dead Load of Liner and Steel Roof in a 2D Axisymmetric Static Analysis Model

Dead load of steel structures on the roof

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

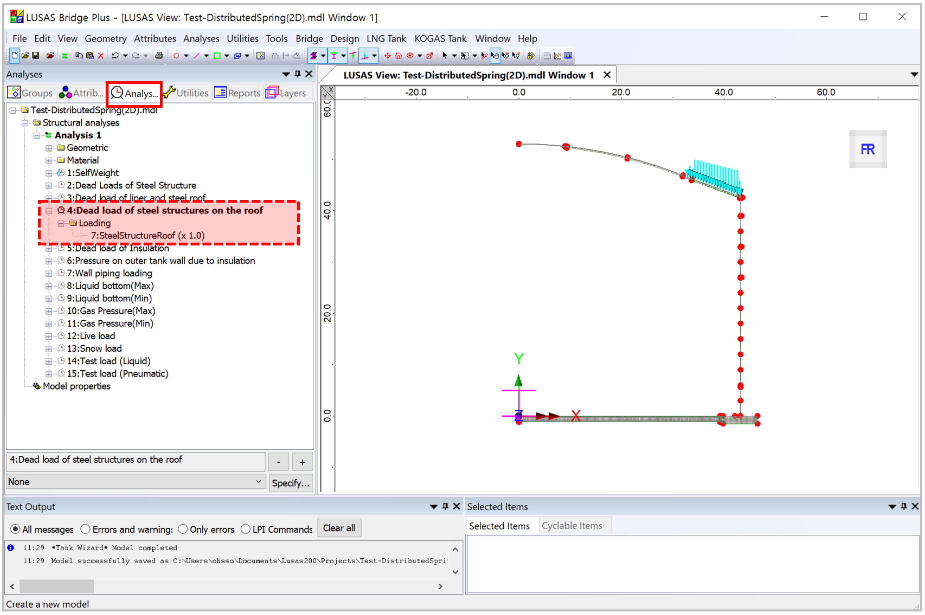


Fig 99 Dead Load of Steel Structures on the Roof in a 2D Axisymmetric Static Analysis Model

Dead load of Insulation

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

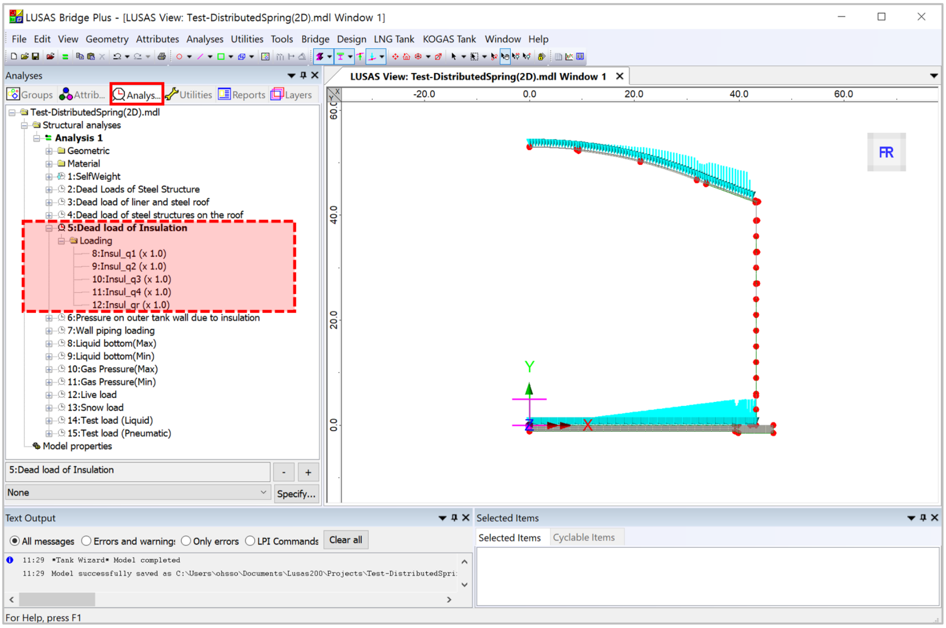


Fig 100 Dead Load of Insulation in a 2D Axisymmetric Static Analysis Model

Pressure on outer tank wall due to insulation

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

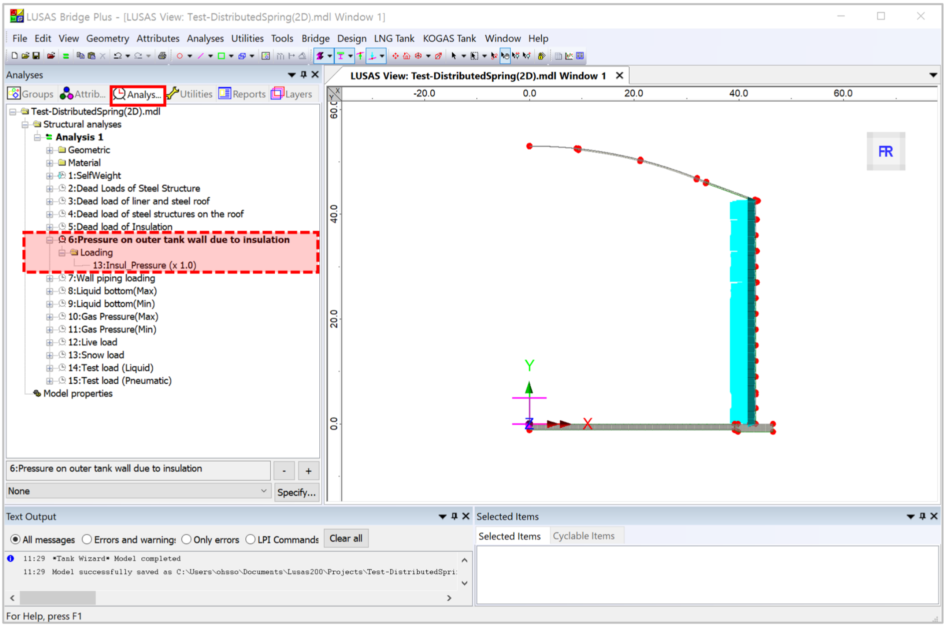


Fig 101 Insulation Pressure Load in a 2D Axisymmetric Static Analysis Model

Wall piping loading

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

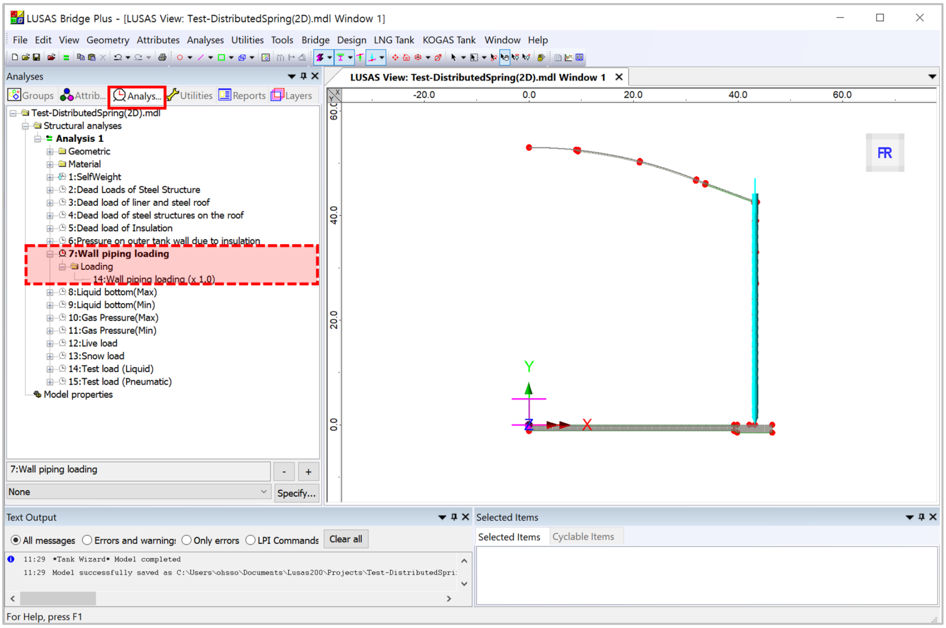


Fig 102 Wall Piping Loading in a 2D Axisymmetric Static Analysis Model

Liquid bottom (Max, Min)

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

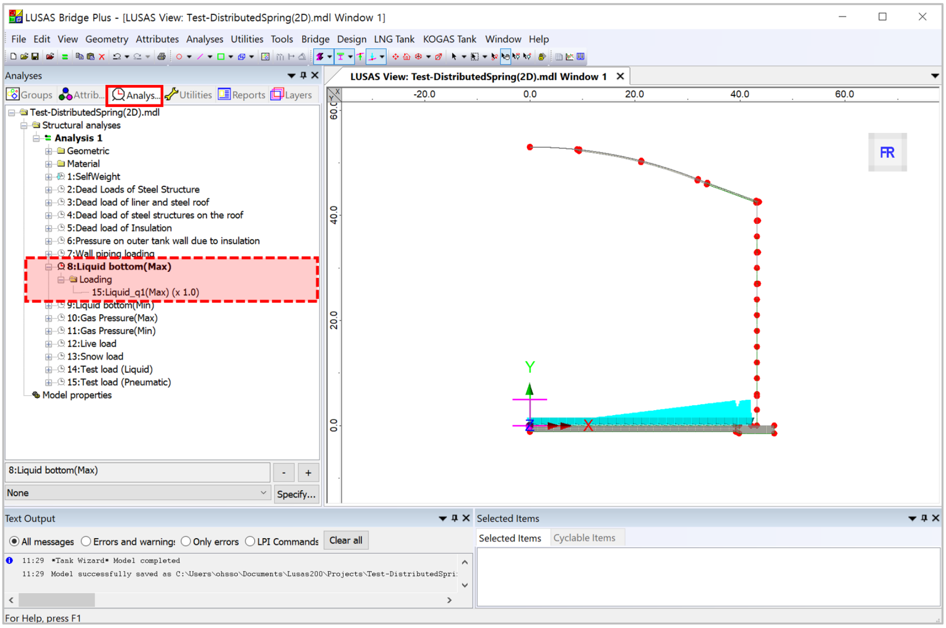


Fig 103 Liquid Bottom Loading in a 2D Axisymmetric Static Analysis Model

Gas pressure (Max, Min)

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

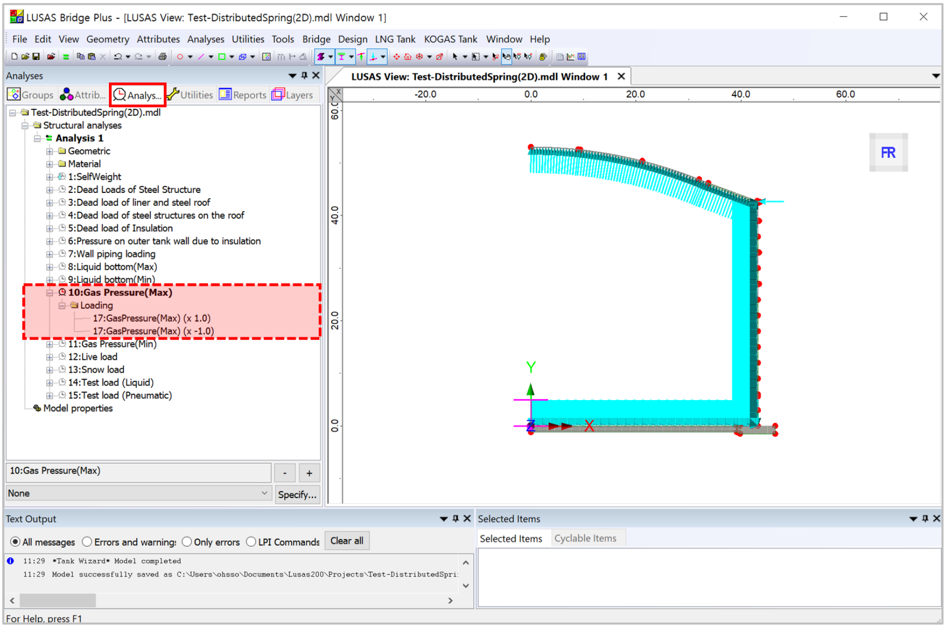


Fig 104 Gas Pressure Loading in a 2D Axisymmetric Static Analysis Model

Live load (Imposed Load on the roof)

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

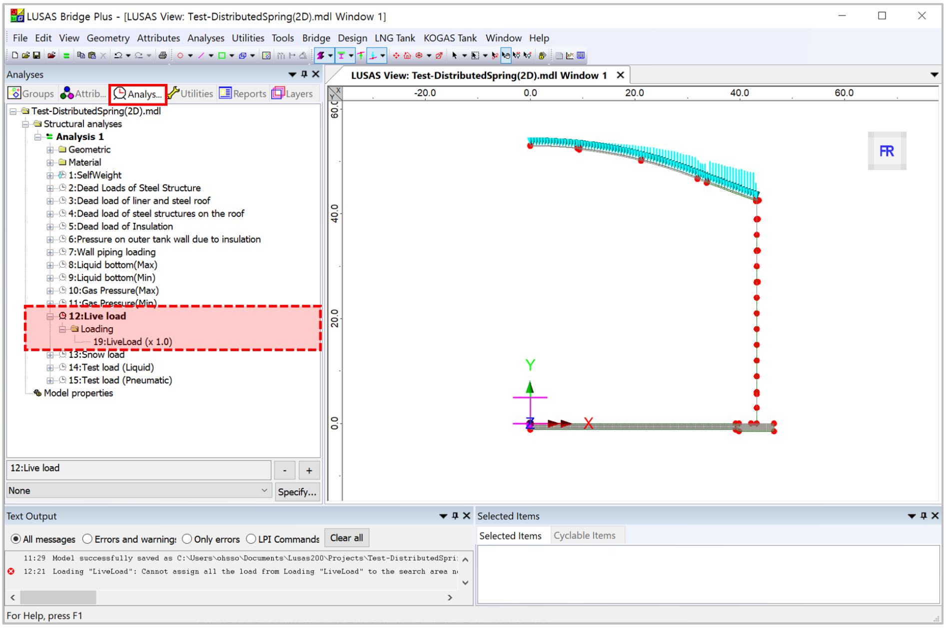


Fig 105 Live Load in a 2D Axisymmetric Static Analysis Model

Snow load

Snow load acts on the top surface of roof.

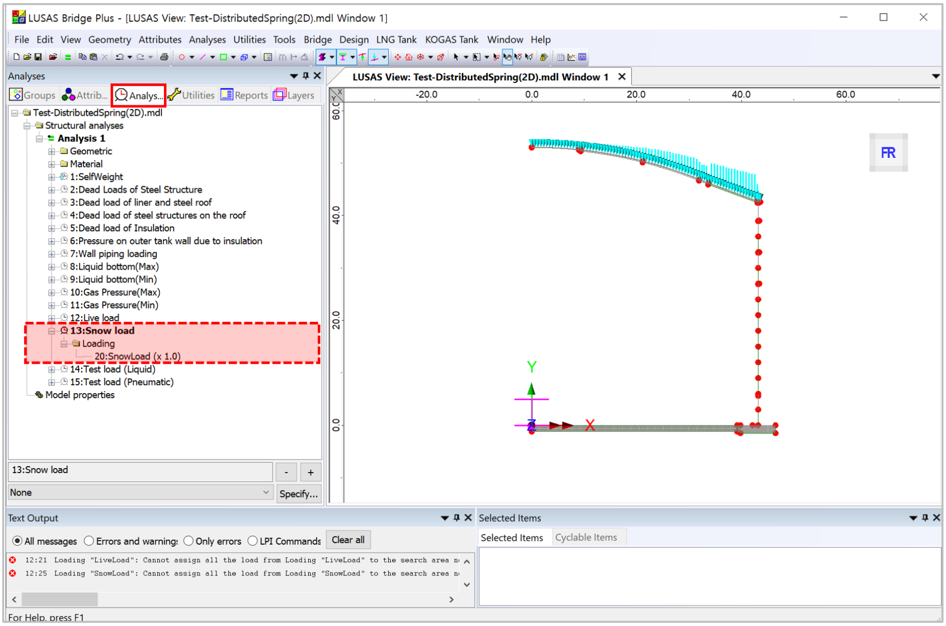


Fig 106 Snow Load in a 2D Axisymmetric Static Analysis Model

Test load (Liquid bottom)

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

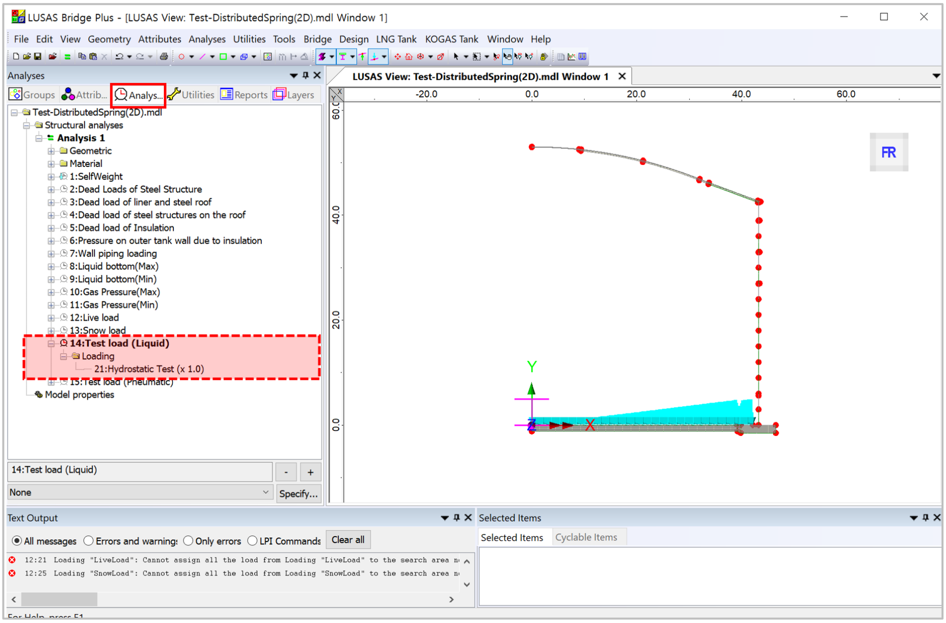


Fig 107 Test Load (Liquid Bottom) in 2D Axisymmetric Static Analysis Model

Test load (Pneumatic)

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

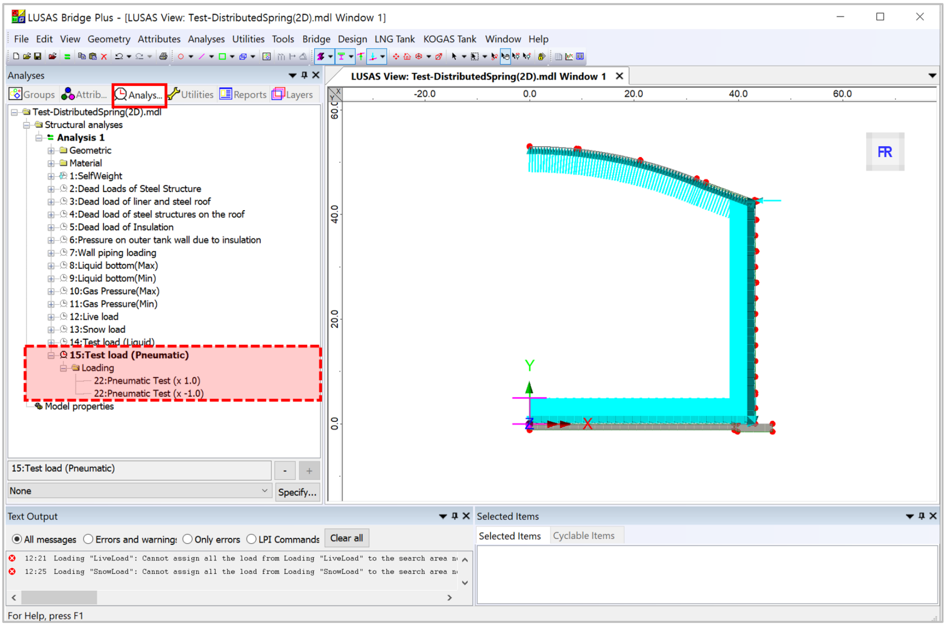


Fig 108 Test Load (Pneumatic) in a 2D Axisymmetric Static Analysis Model

Viewing Results

Contours

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

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

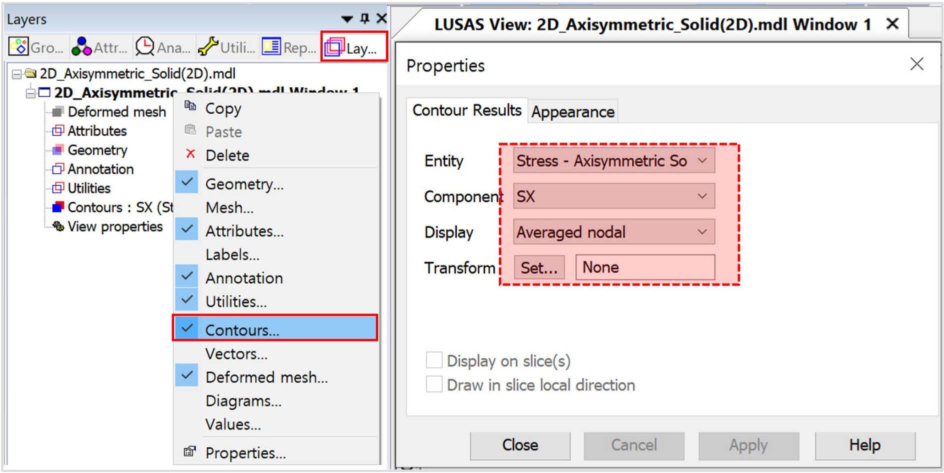


Fig 109 Selection for Contour Display in a 2D Axisymmetric Solid Model

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

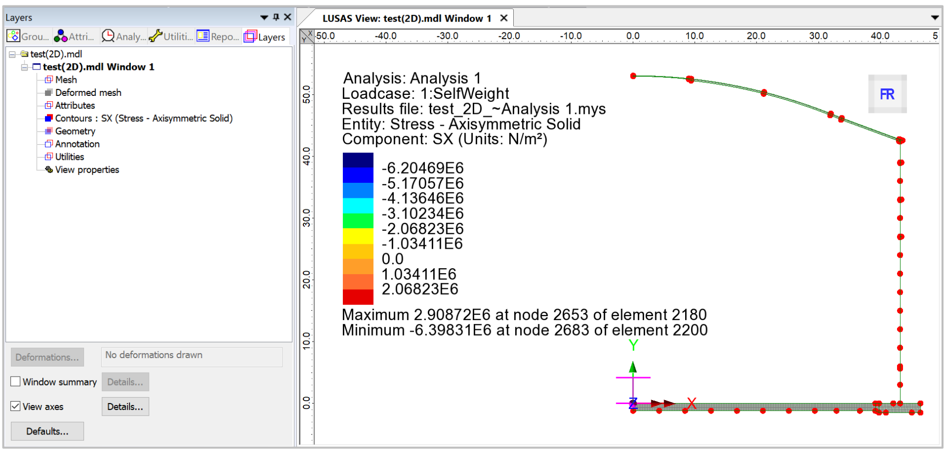


Fig 110 SX Contour for Self Weight in a 2D Axisymmetric Solid Model

Values

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

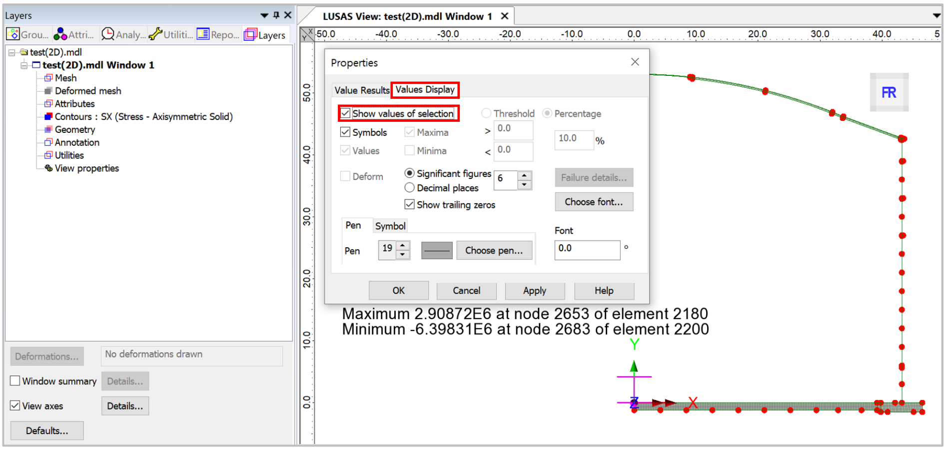


Fig 111 Value Display in a 2D Axisymmetric Solid Model

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

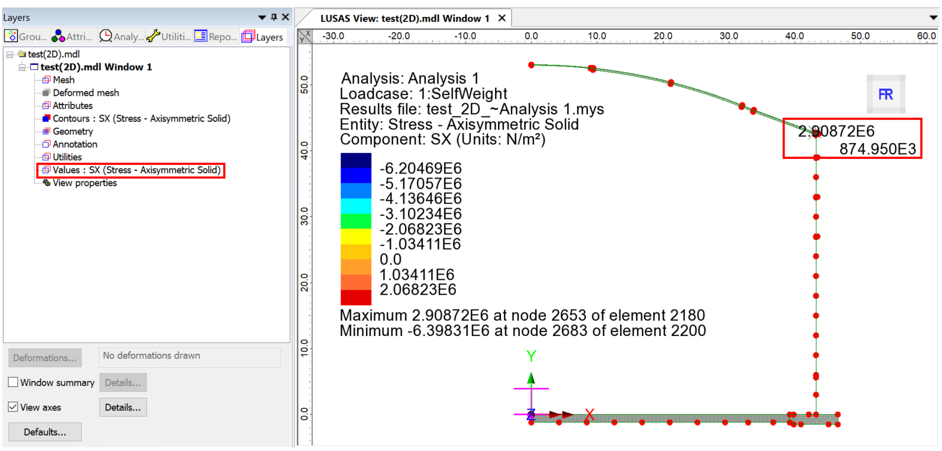


Fig 112 Values Displayed for Selected Nodes in a 3D Shell Model

Graph through 2D

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

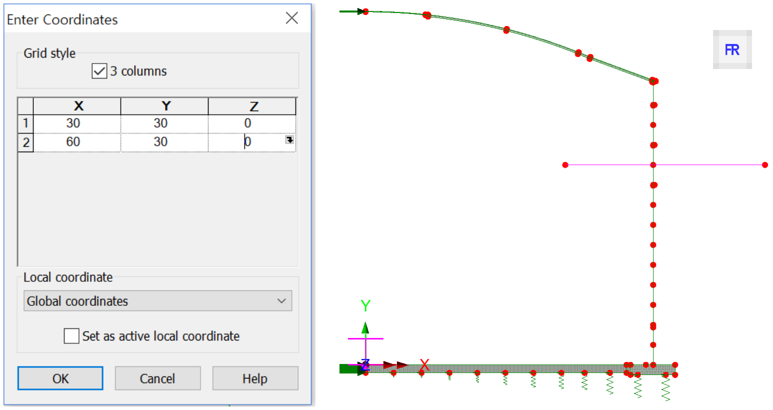


Fig 113 Line for Slicing Results in a 2D Axisymmetric Solid Model

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

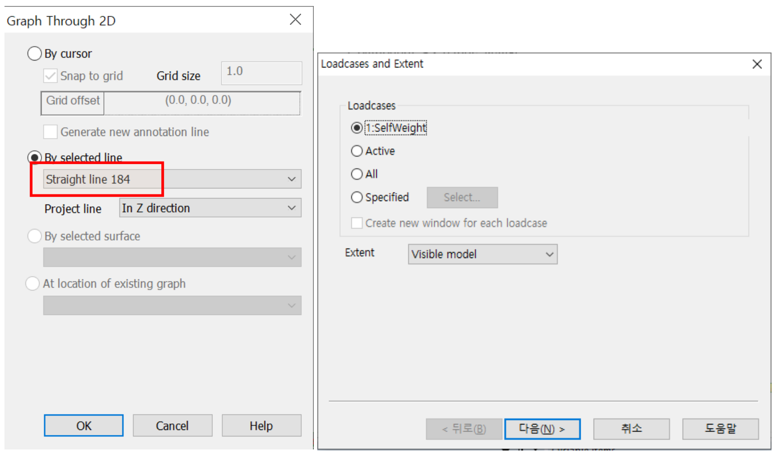


Fig 114 Graph Through 2D in a 2D Axisymmetric Solid Model (1)

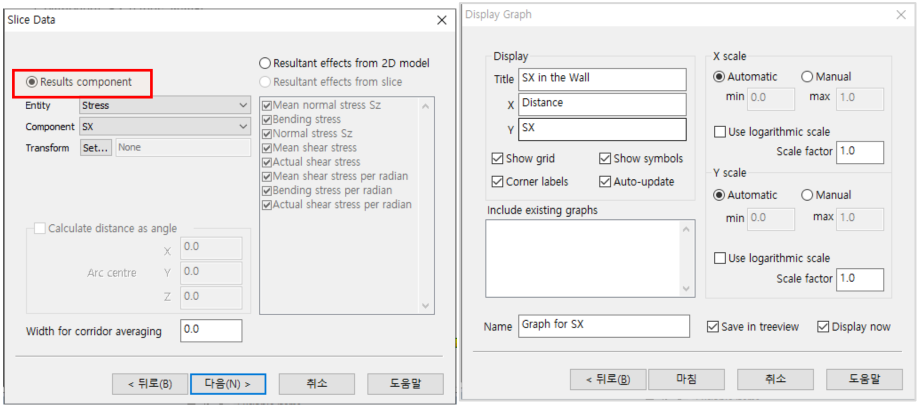


Fig 115 Graph Through 2D in a 2D Axisymmetric Solid Model (2)

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

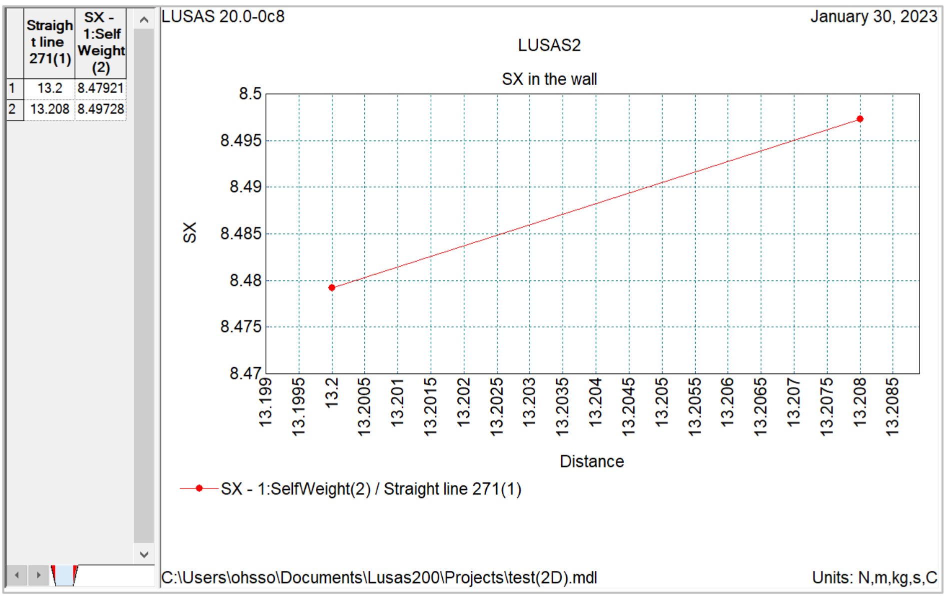


Fig 116 SX Graph for Sliced Line in a 2D Axisymmetric Solid Model

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

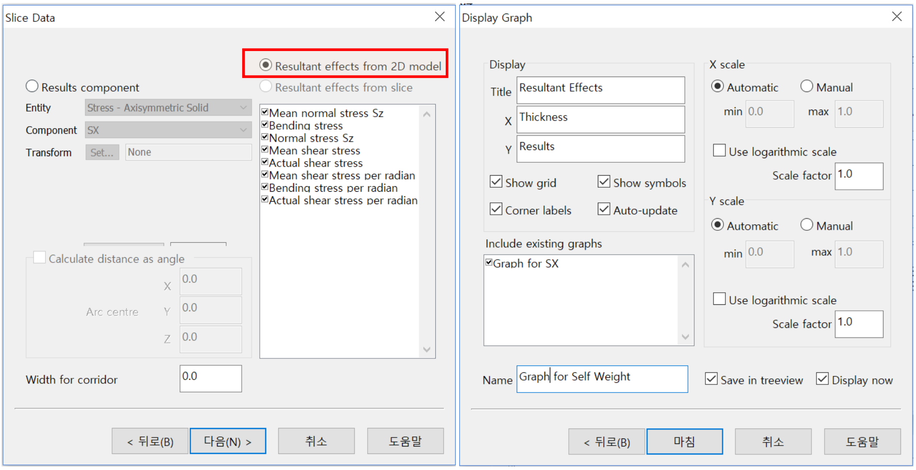


Fig 117 Graph Through 2D in a 2D Axisymmetric Solid Model (3)

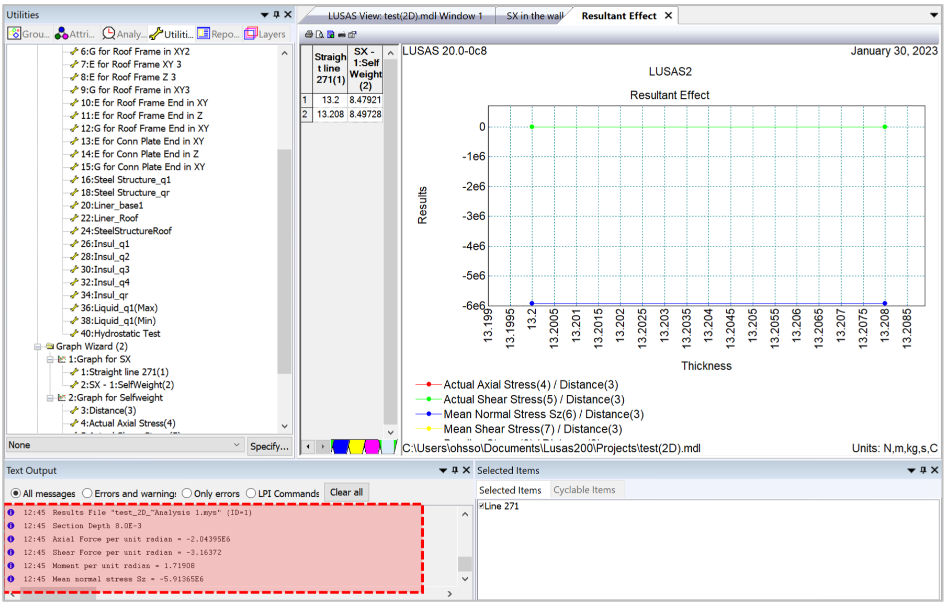


Fig 118 SX Graph for Sliced Line in a 2D Axisymmetric Solid Model

Export Forces to Excel (2D)

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

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

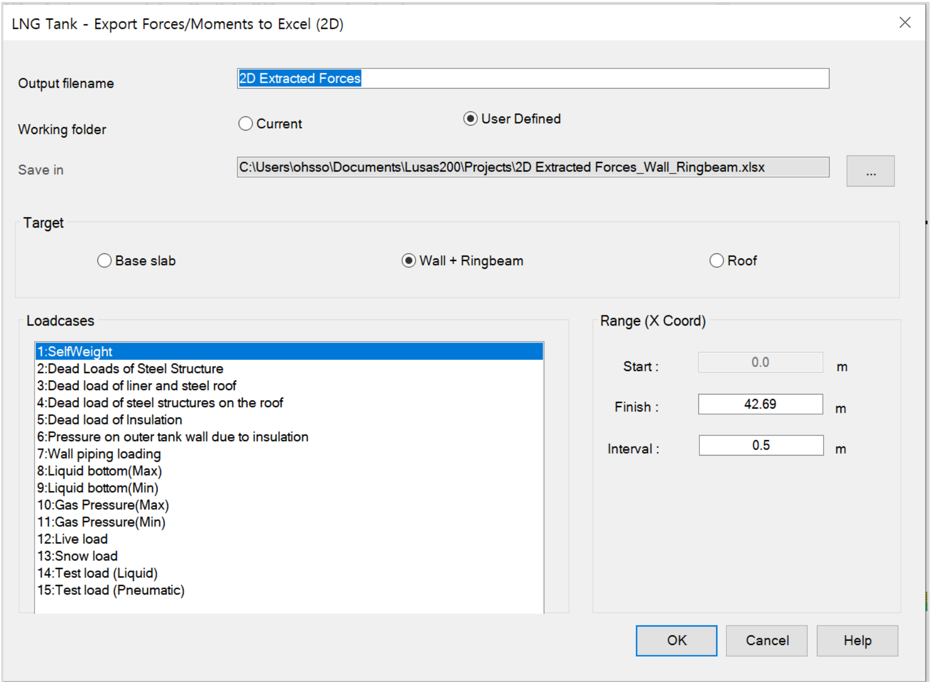


Fig 119 Export Forces for a 2D Axisymmetric Solid Model (1)

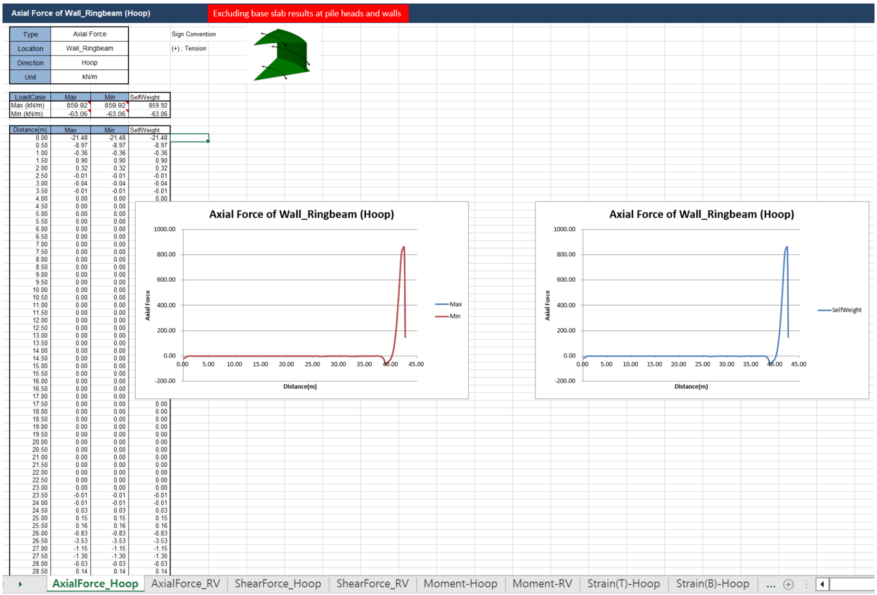


Fig 120 Section Force Spreadsheet for Self Weight

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

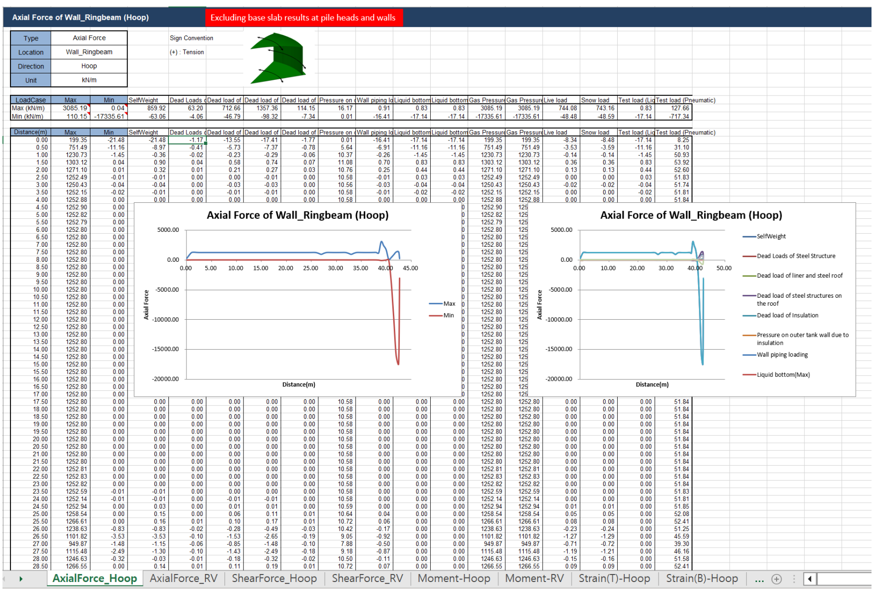


Fig 121 Section Force Spreadsheet for All Loadcases

Sign convention

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

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

2D Axisymmetric Staged Construction Analysis

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

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

User Inputs

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

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

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

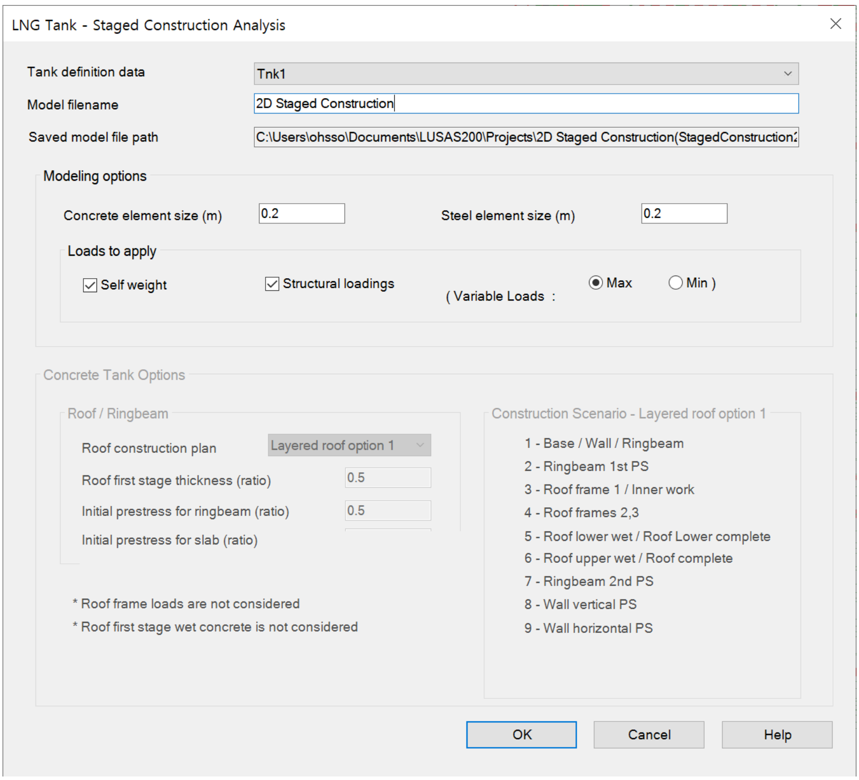


Fig 122 User Dialog for 2D Axisymmetric Staged Construction Analysis

Meshing / Geometric Properties / Material Properties / Support Conditions

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

Activation and Deactivation

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

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

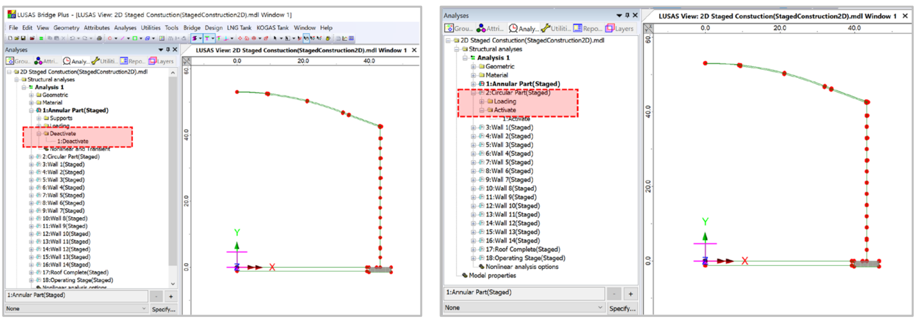


Fig 123 Activate and Deactivate Assignment in the Model

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

Control for Nonlinear Analysis

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

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

- 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.

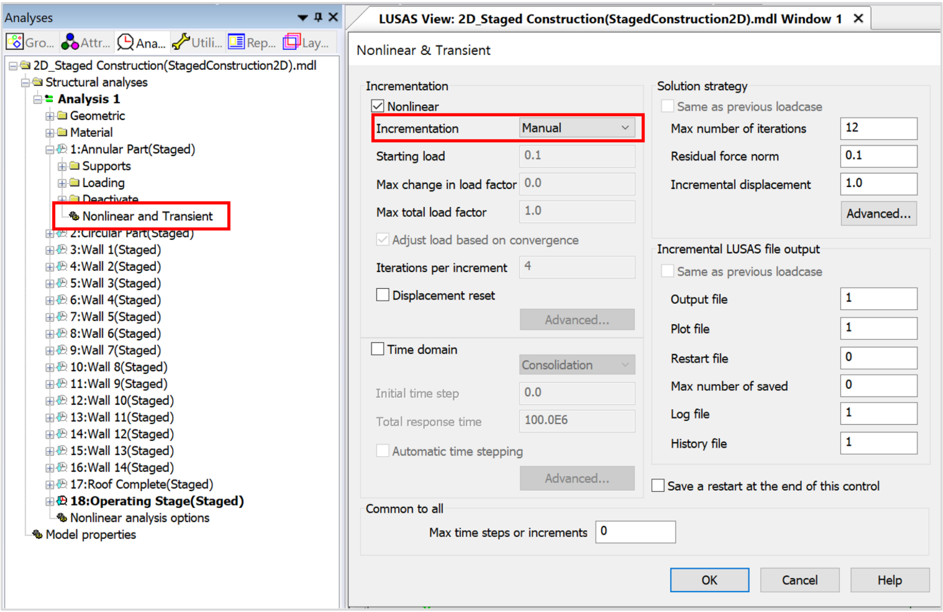


Fig 124 Nonlinear Control for a Staged Construction Analysis

Loading

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

Stage 1 : Annular Part ~ Stage 2 : Circular Part

Self weight is assigned by using 'Gravity' loading.

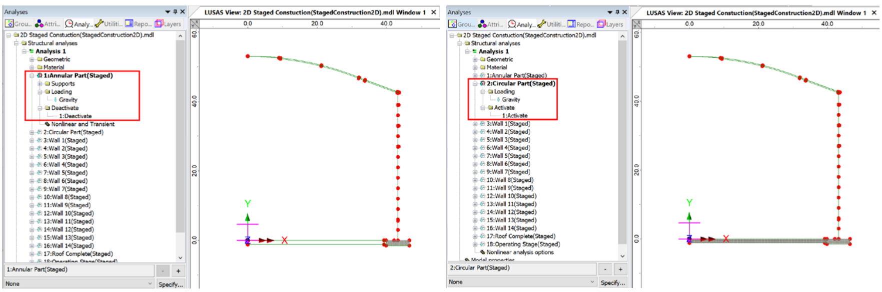


Fig 125 Loadings for Stage 1~2

Stage 3 : Wall 1 ~ Stage 16 : Wall 14

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

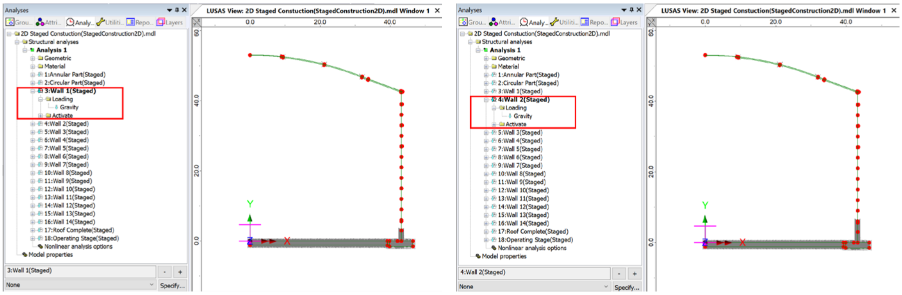


Fig 126 Loadings for Stage 3~4

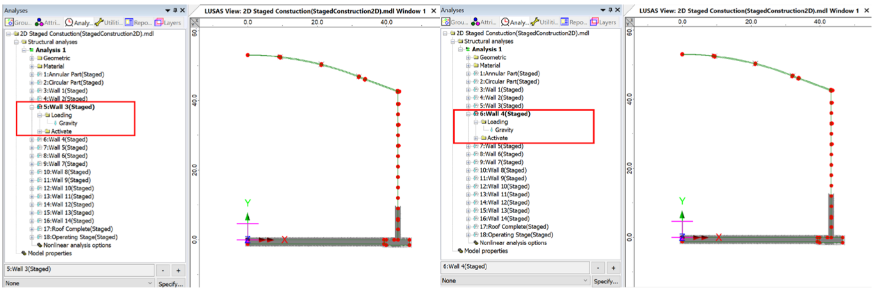


Fig 127 Loadings for Stage 5~6

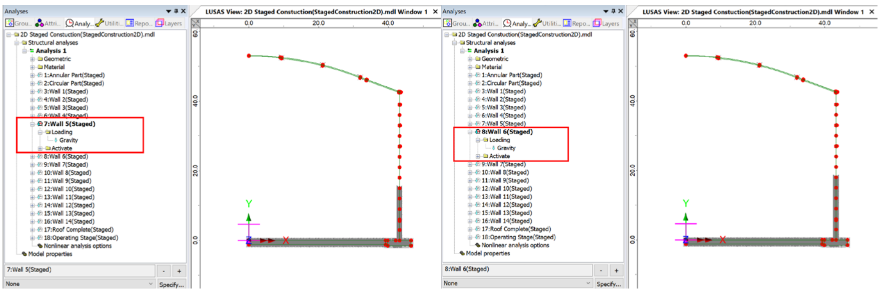


Fig 128 Loadings for Stage 7~8

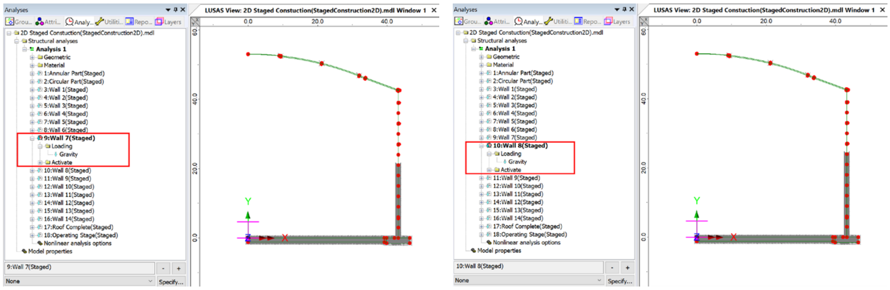


Fig 129 Loadings for Stage 9~10

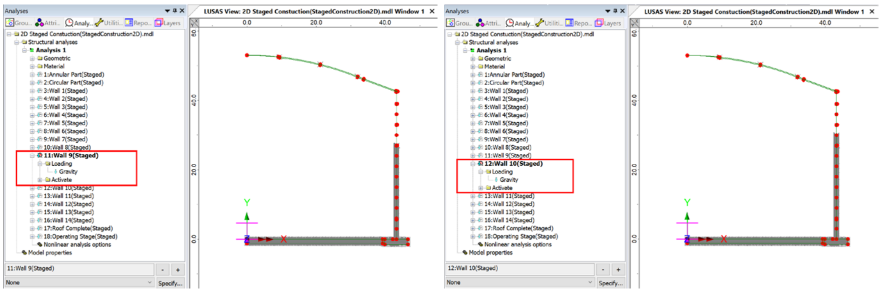


Fig 130 Loadings for Stage 11~12

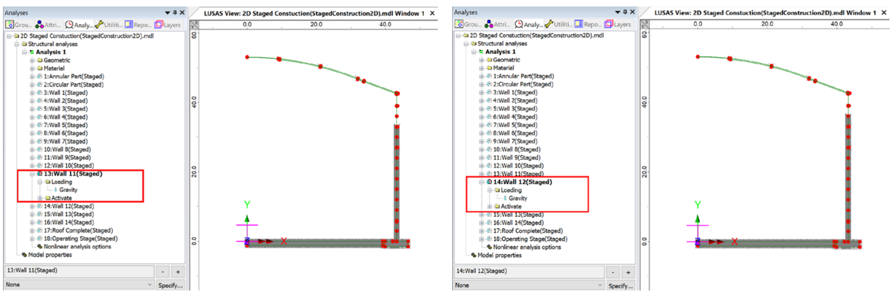


Fig 131 Loadings for Stage 13~14

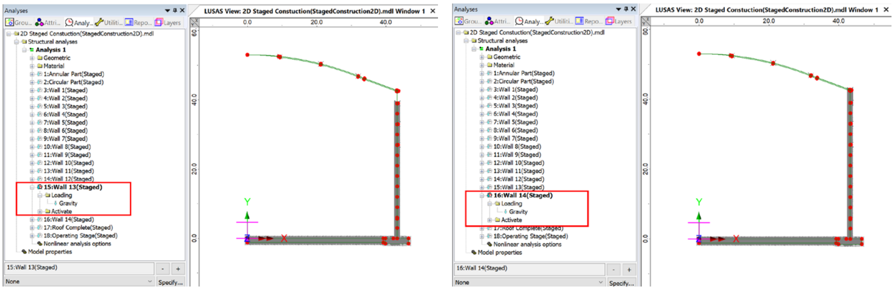


Fig 132 Loadings for Stage 15~16

Stage 17 : Roof Complete ~ Stage 18 : Operating Stage

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

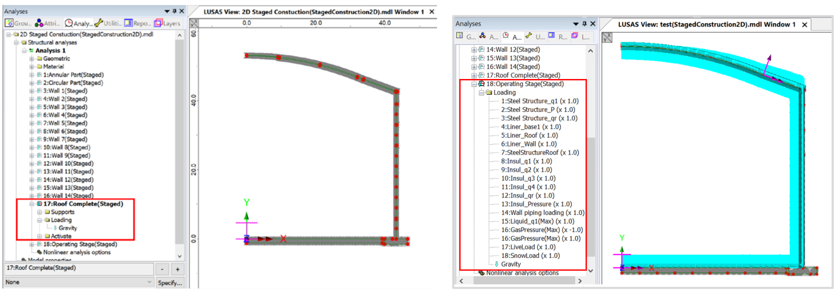


Fig 133 Loadings for Stage 17~18

Adding Extra Stages

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

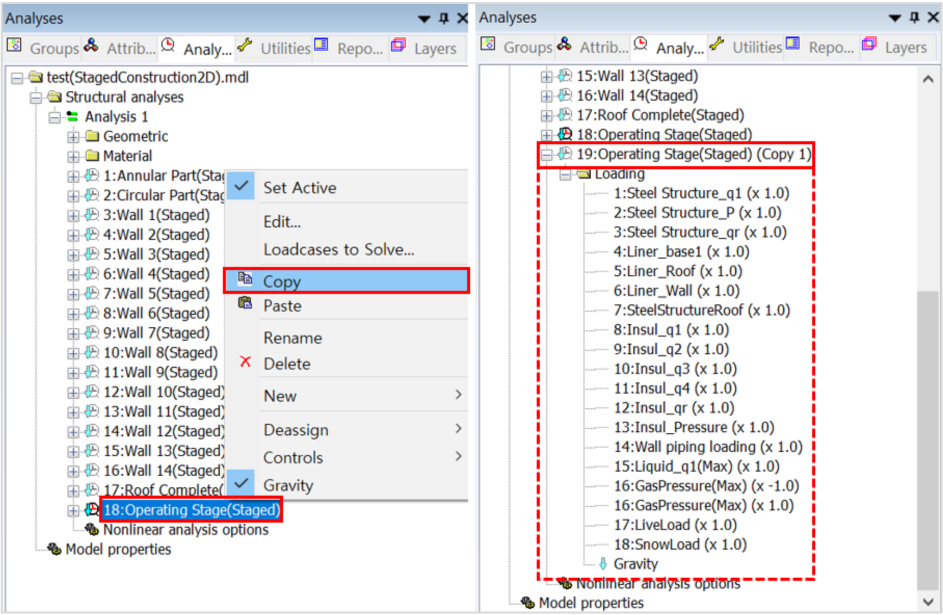


Fig 134 Adding Stages in the 2D Axisymmetric Staged Construction Analysis Model

2D Axisymmetric Thermal Analysis

User Inputs

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

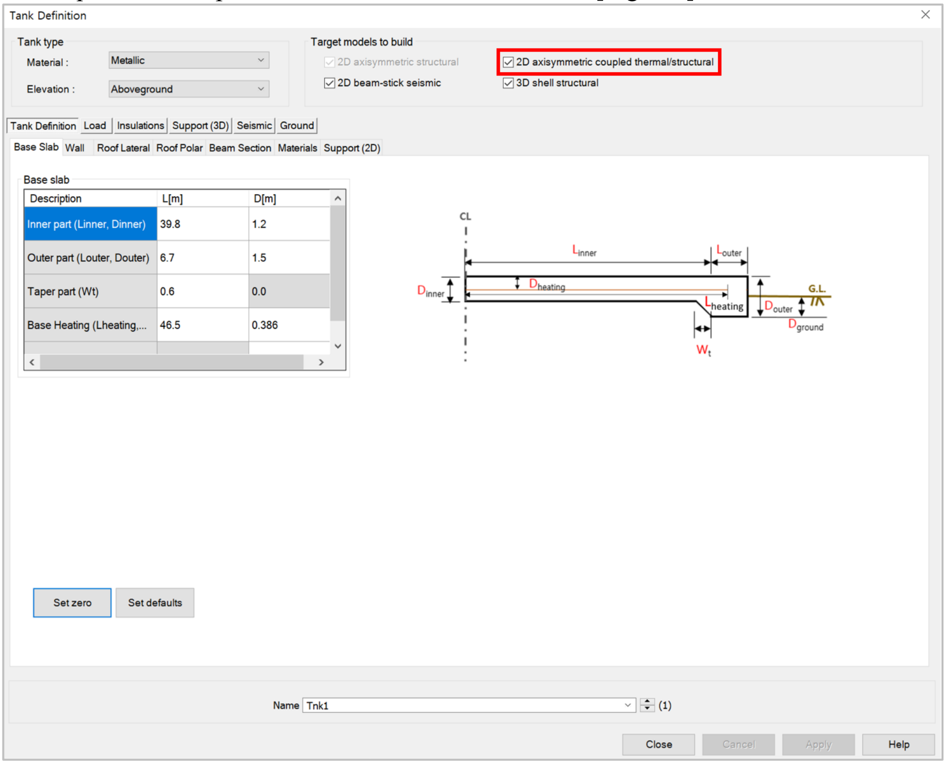


Fig 135 User Inputs for 2D Axisymmetric Thermal Analysis

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

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

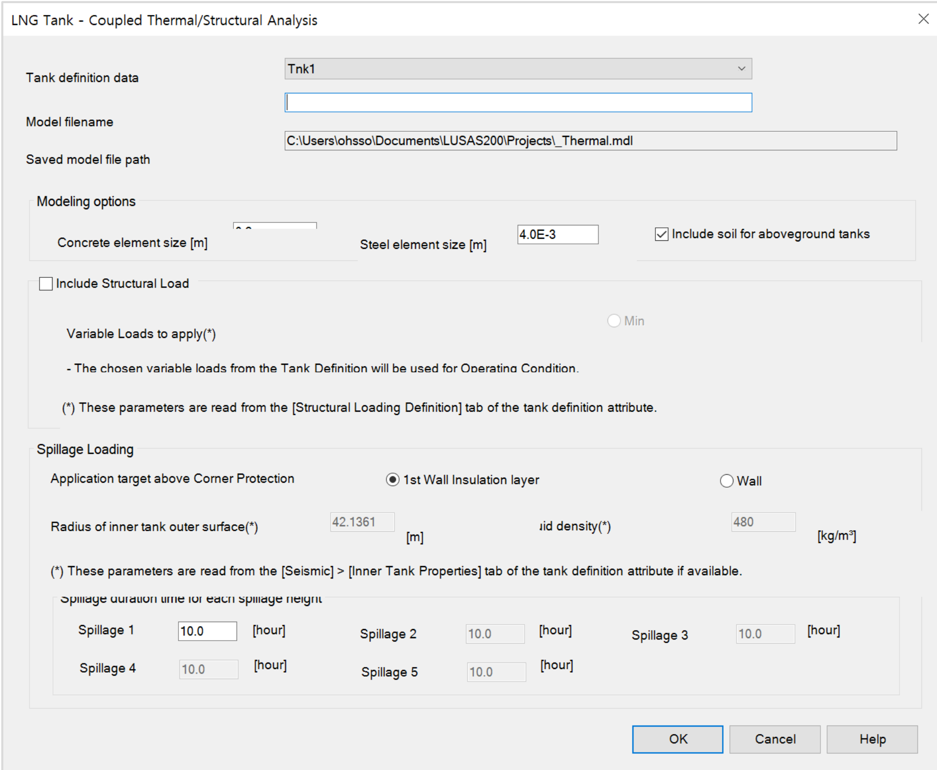


Fig 136 User Dialog for 2D Axisymmetric Thermal Analysis

Meshing

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

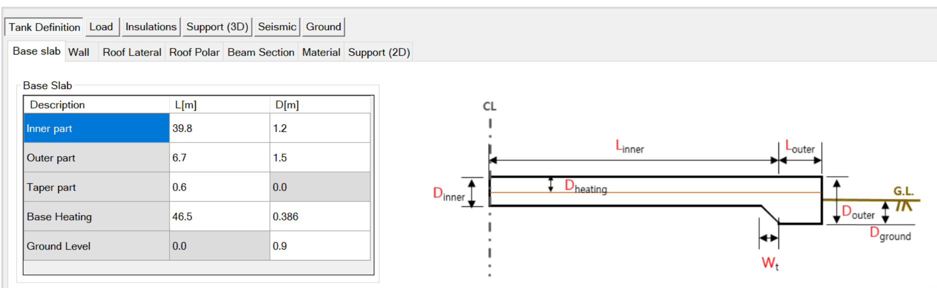


Fig 137 ‘Ground Level’ in Tank Definition for Thermal Analysis Model

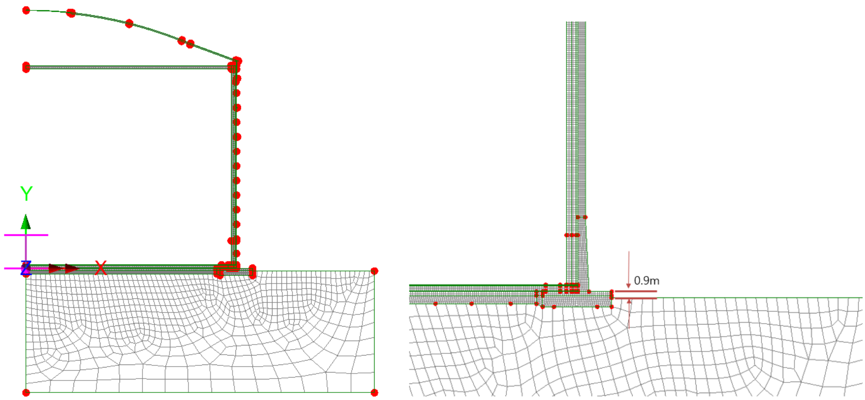


Fig 138 Elements for 2D Axisymmetric Thermal Analysis Model

Geometric Properties

No geometric properties are required for 2D axisymmetric model.

Material Properties

User defined material properties are assigned to the relevant surfaces.

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

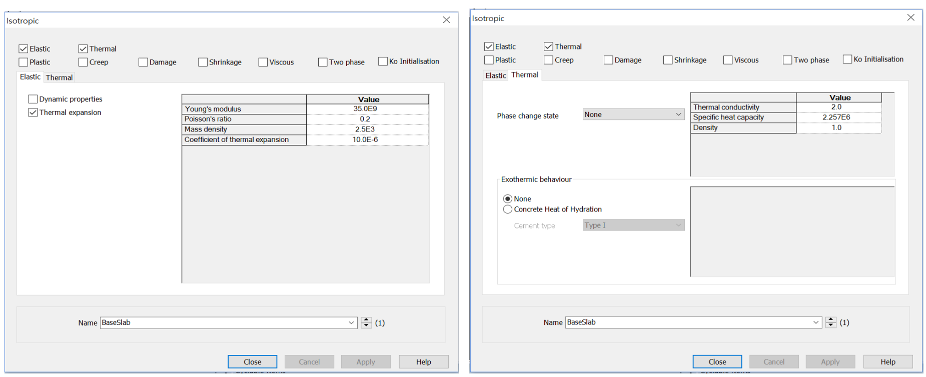


Fig 139 Material Properties of BaseSlab for a 2D Axisymmetric Thermal Analysis Model

Support Conditions

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

Loadings

Thermal Analysis > Initial Conditions

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

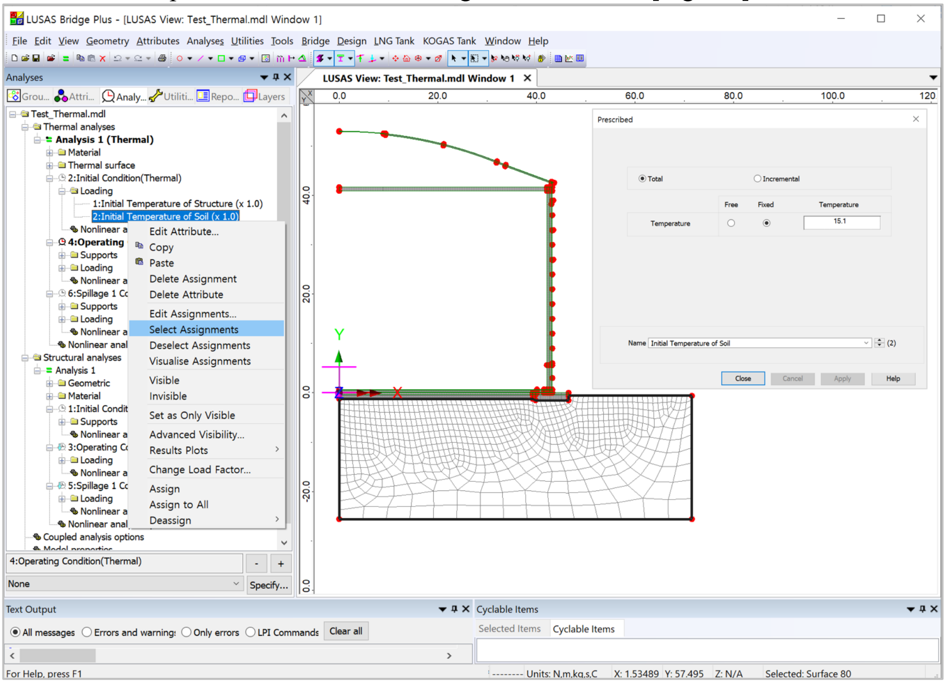


Fig 140 Initial Soil Temperature in a 2D Axisymmetric Thermal Analysis Model

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

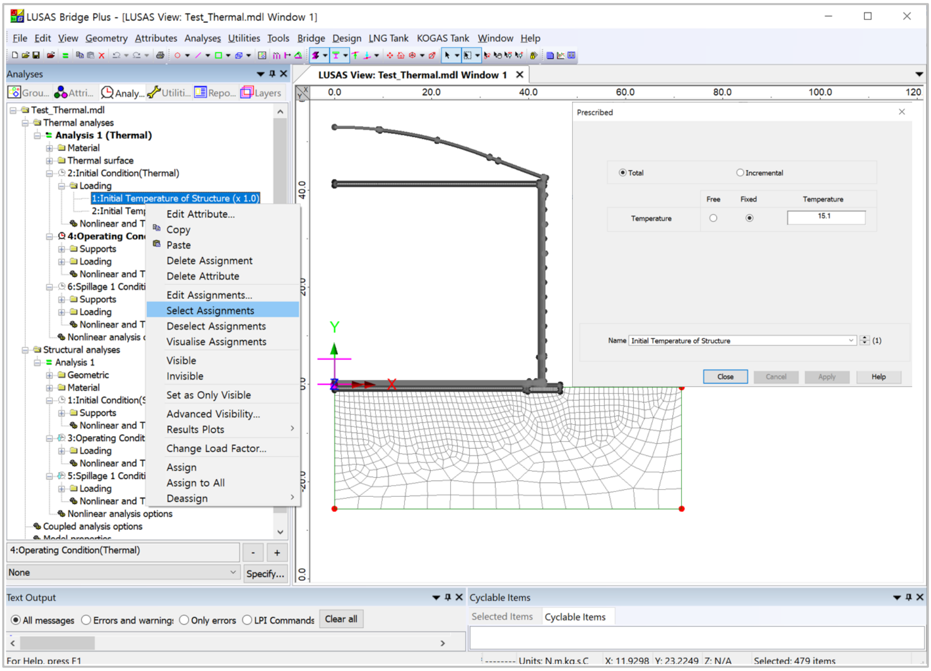


Fig 141 Initial Structure Temperature in a 2D Axisymmetric Thermal Analysis Model

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.

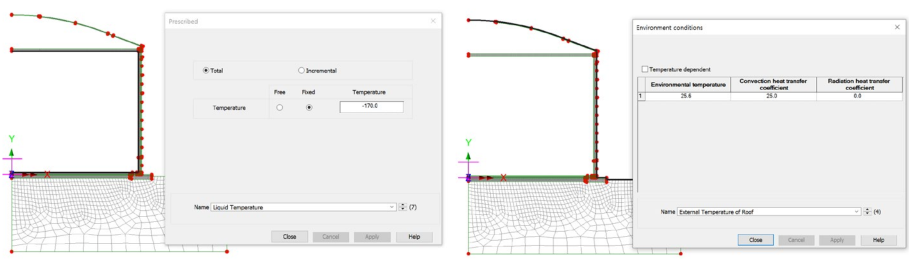


Fig 142 Operating Temperatures in a 2D Axisymmetric Thermal Analysis Model

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

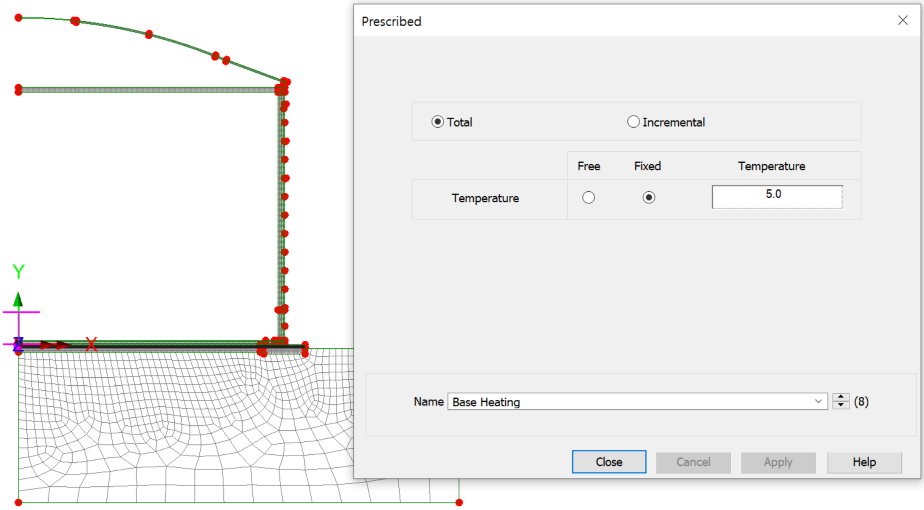


Fig 143 Base Heating Temperature in 2D Axisymmetric Thermal Analysis Model

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

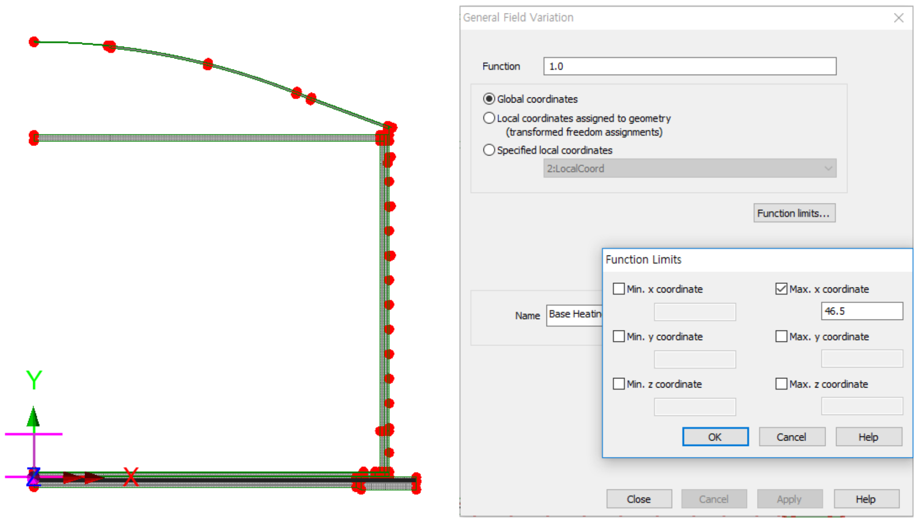


Fig 144 Base Heating Temperature Variation in 2D Axisymmetric Thermal Analysis Model

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

3D Shell Analysis

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

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

User Inputs

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

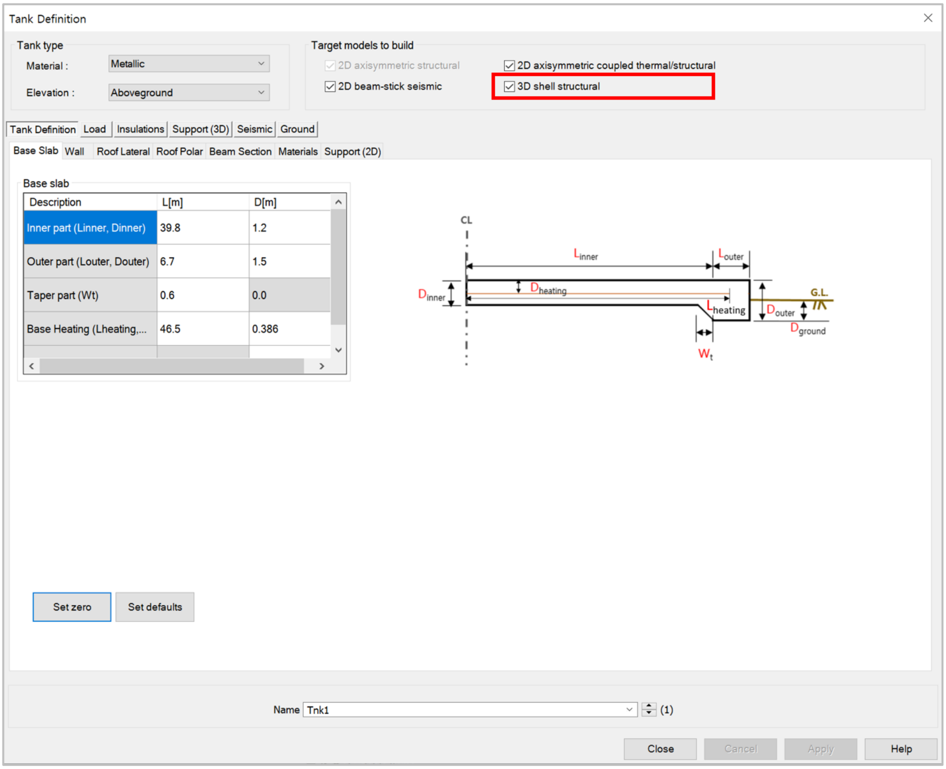


Fig 145 User Inputs for a 3D Shell Static Analysis

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

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

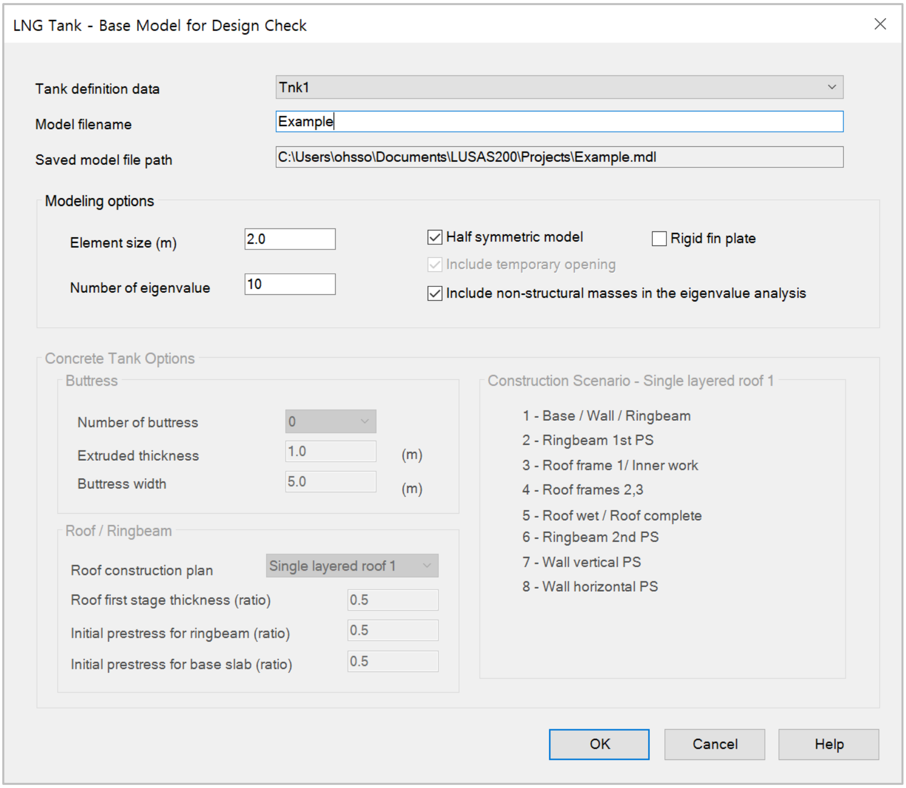


Fig 146 User Dialog for a 3D Shell Static Analysis Model

Mesh

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

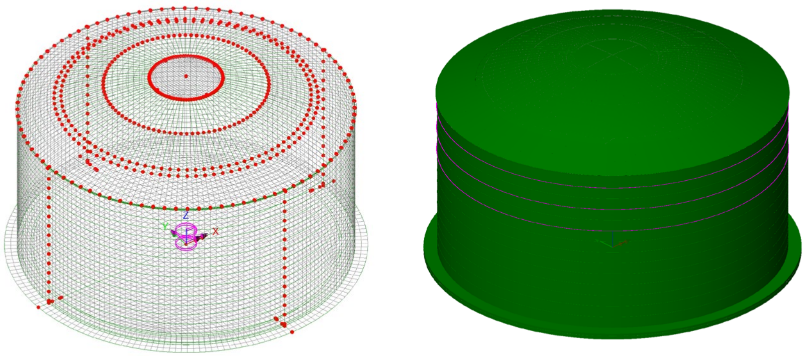


Fig 147 Mesh Arrangement and Geometric Properties for a 3D Shell Model

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

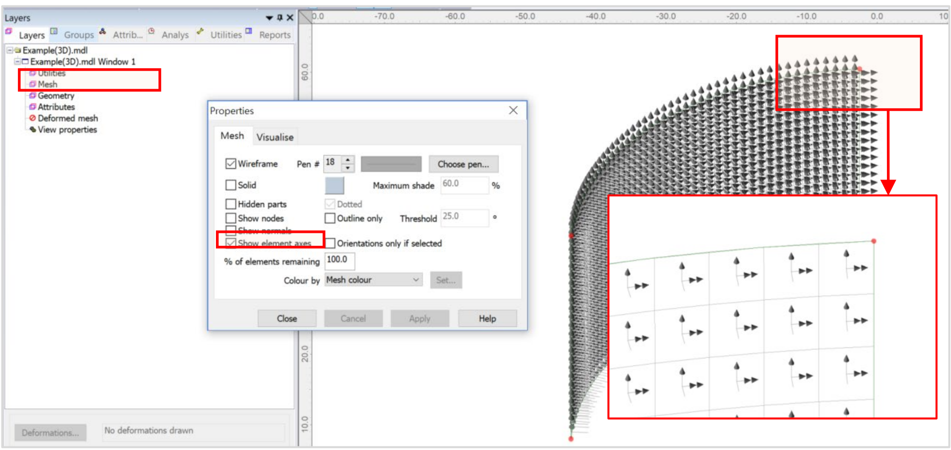


Fig 148 Element Local Axis in a 3D Shell Model

Material Properties

Structural members

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

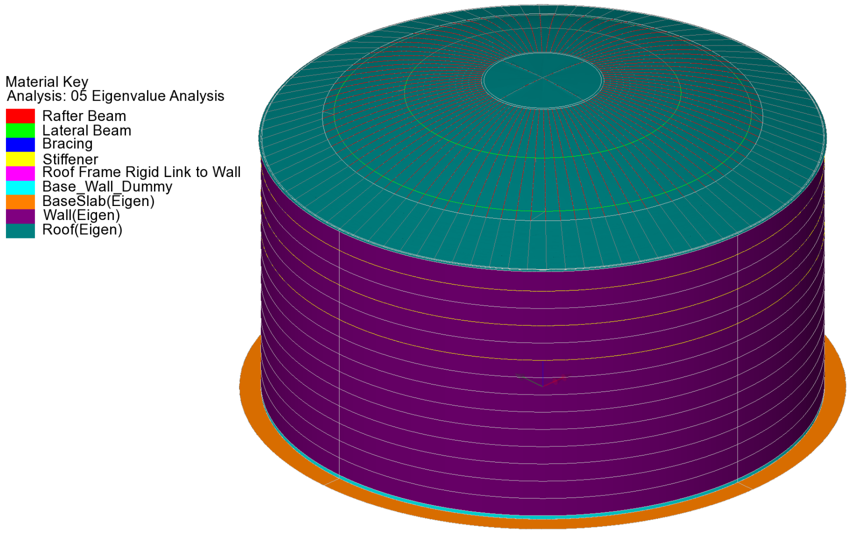


Fig 149 Material Properties in a 3D Shell Model

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.

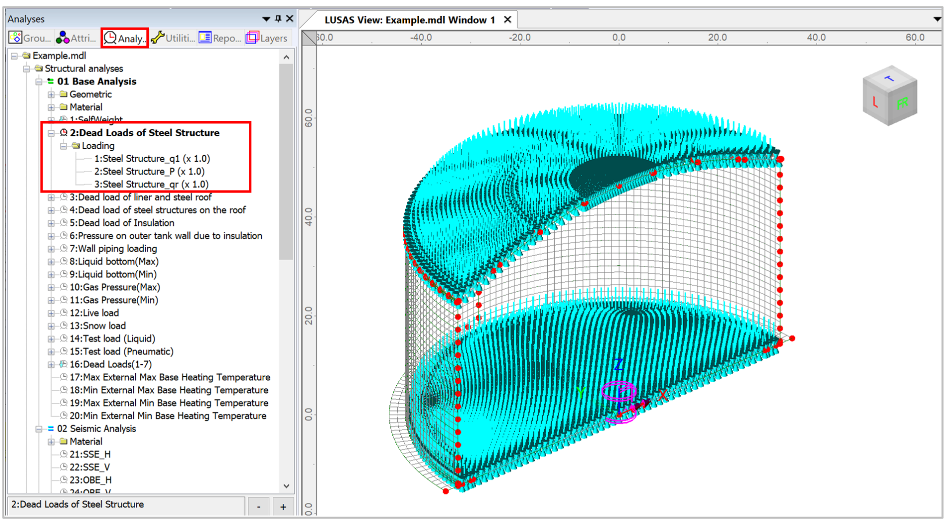


Fig 150 Dead Loads for Steel Structure in a 3D Shell Static Analysis Model

Dead load of liner and steel roof

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

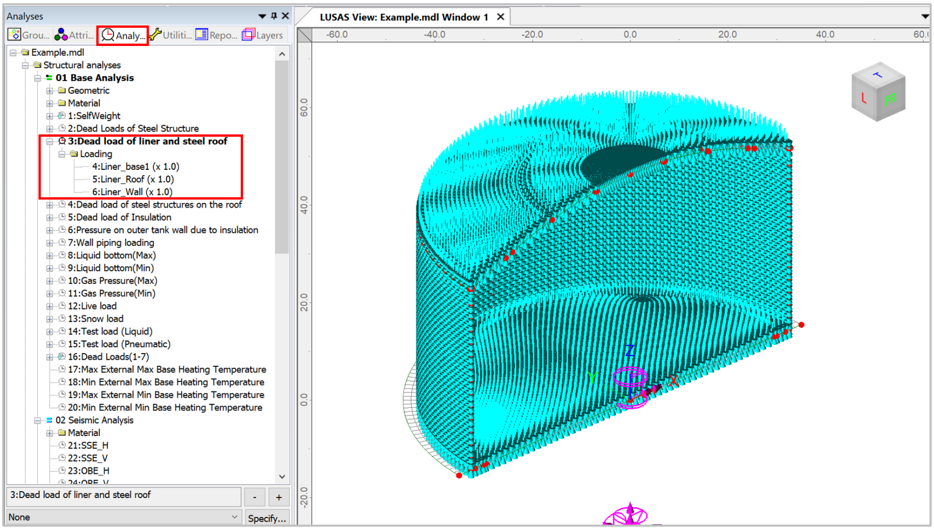


Fig 151 Dead Load of Liner and Steel Roof in a 3D Shell Static Analysis Model

Dead load of steel structures on the roof

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

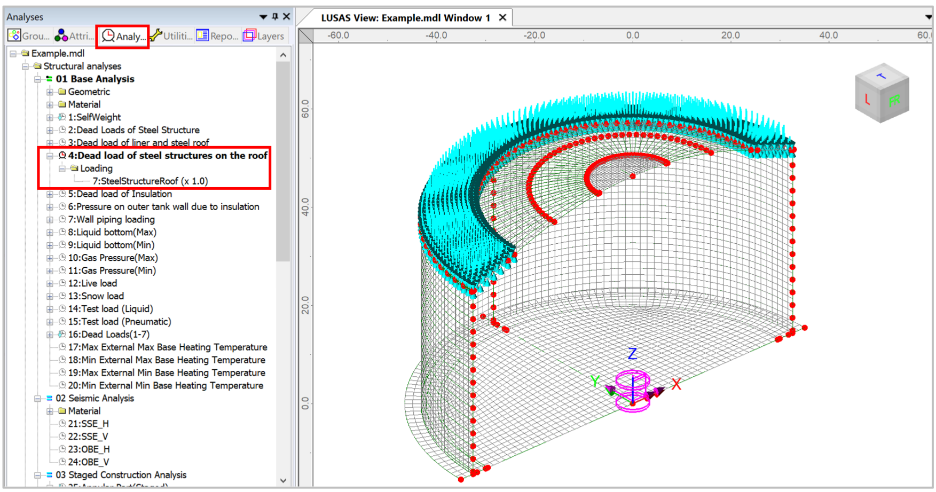


Fig 152 Dead Load of Steel Structures on the Roof in a 3D Shell Static Analysis Model

Dead load of Insulation

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

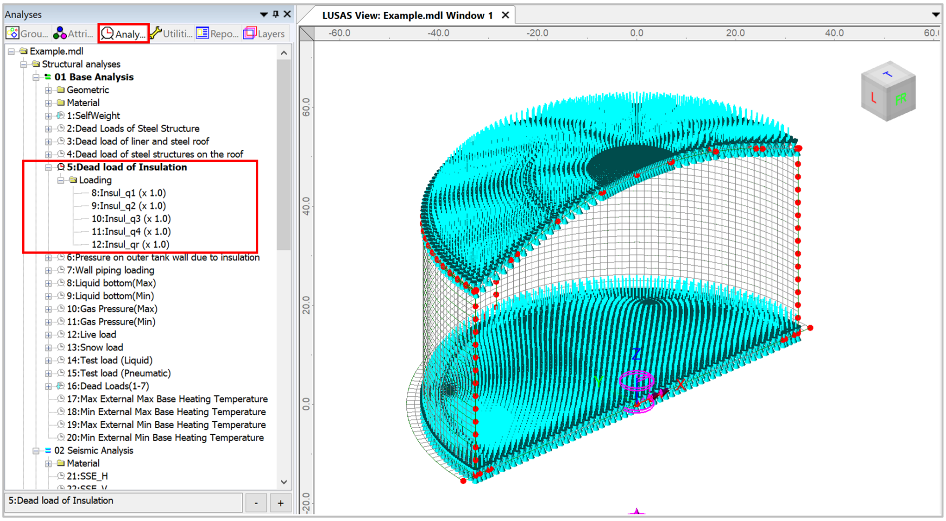


Fig 153 Dead Load of Insulation in a 3D Shell Static Analysis Model

Pressure on outer tank wall due to insulation

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

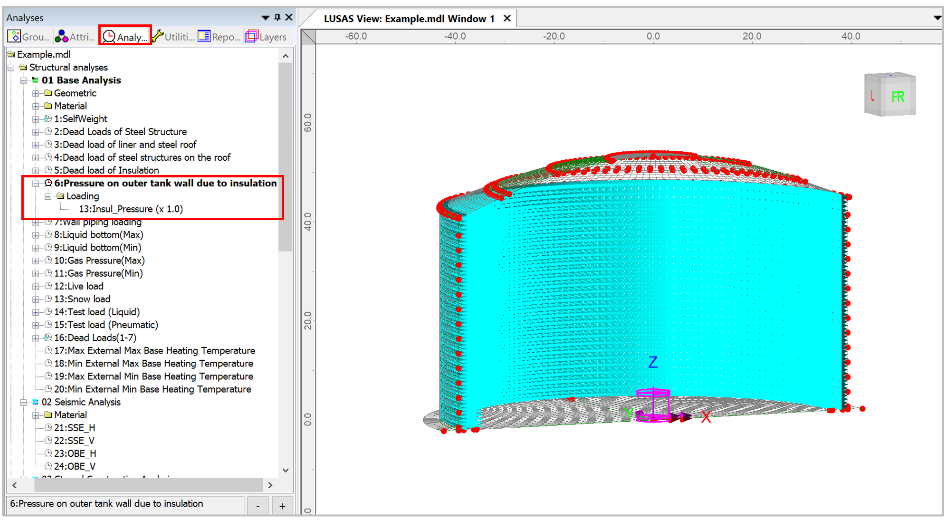


Fig 154 Insulation Pressure Load in a 3D Shell Static Analysis Model

Wall Piping Loading

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

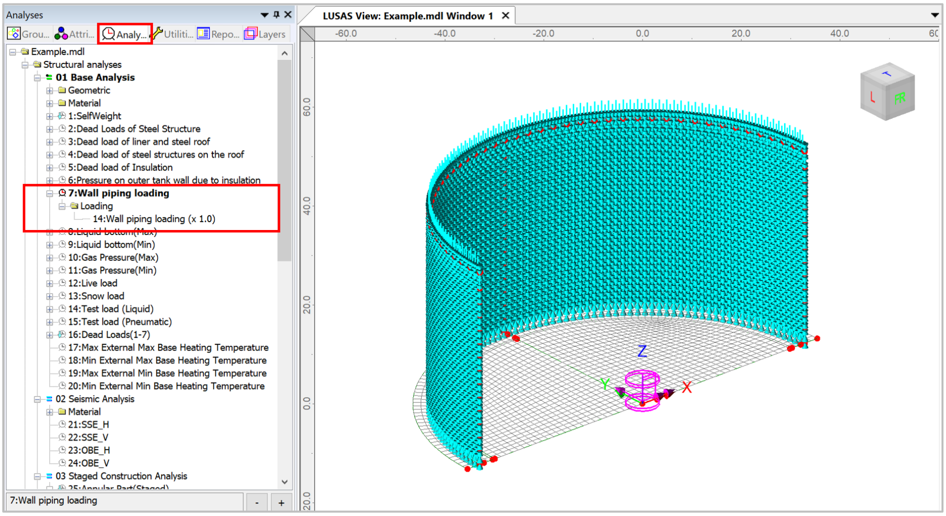


Fig 155 Wall piping loading in a 3D Shell Static Analysis Model

Liquid bottom (Max/Min)

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

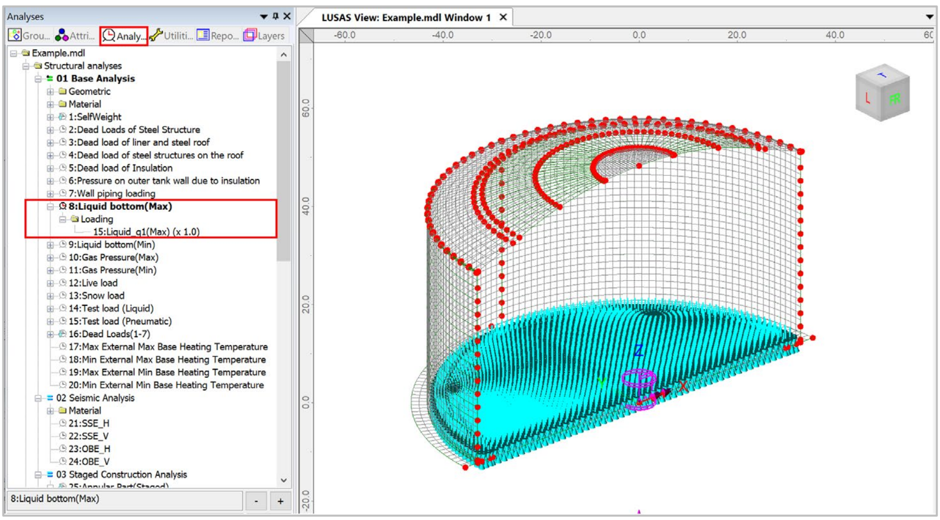


Fig 156 Liquid Bottom Loading in a 3D Shell Static Analysis Model

Gas Pressure(Max/Min)

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

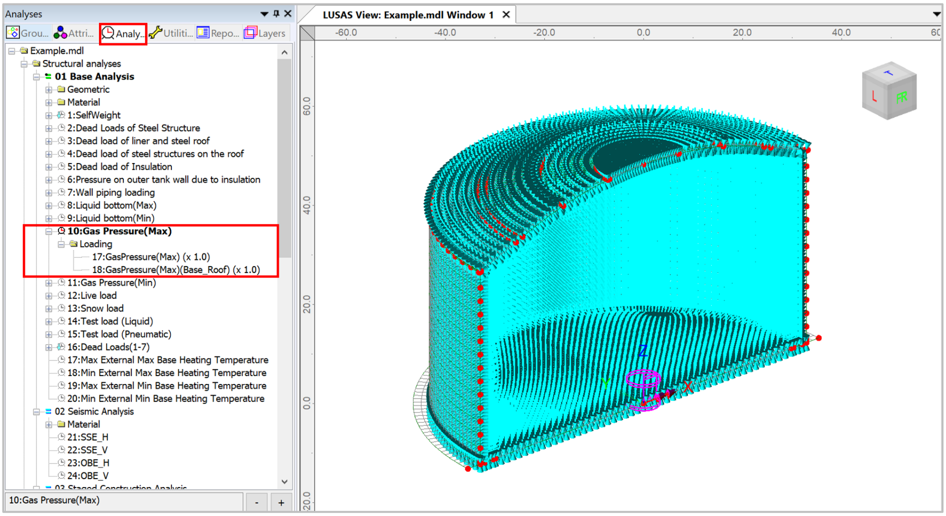


Fig 157 Gas Pressure Loading in a 3D Shell Static Analysis Model

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(R = 0 ~ R = 43.23m).

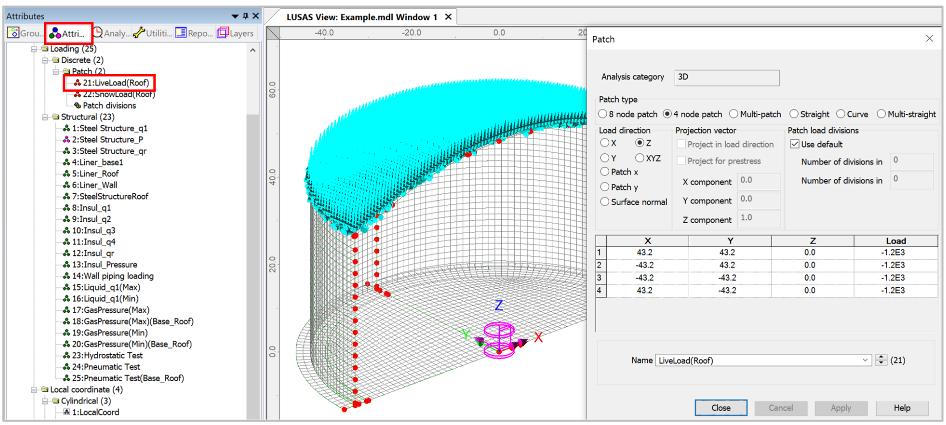


Fig 158 Live Load in a 3D Shell Static Analysis Model (Roof)

Snow load

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

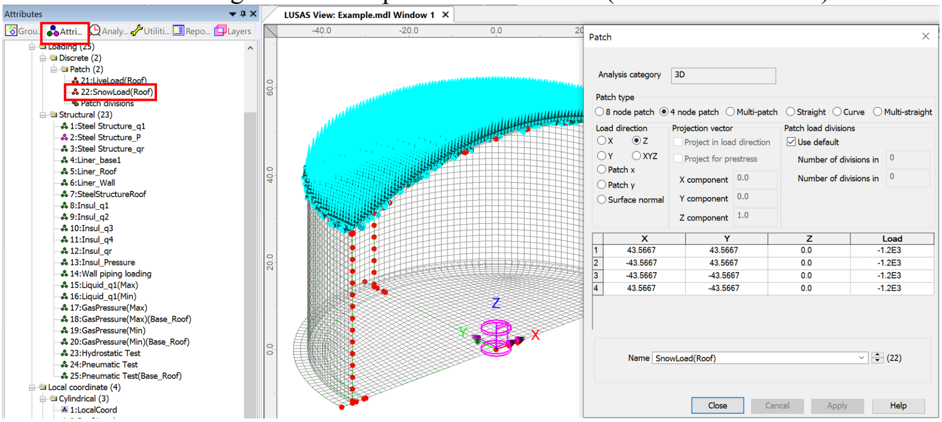


Fig 159 Snow Load in a 3D Shell Static Analysis Model (Roof)

Test load (Liquid bottom)

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

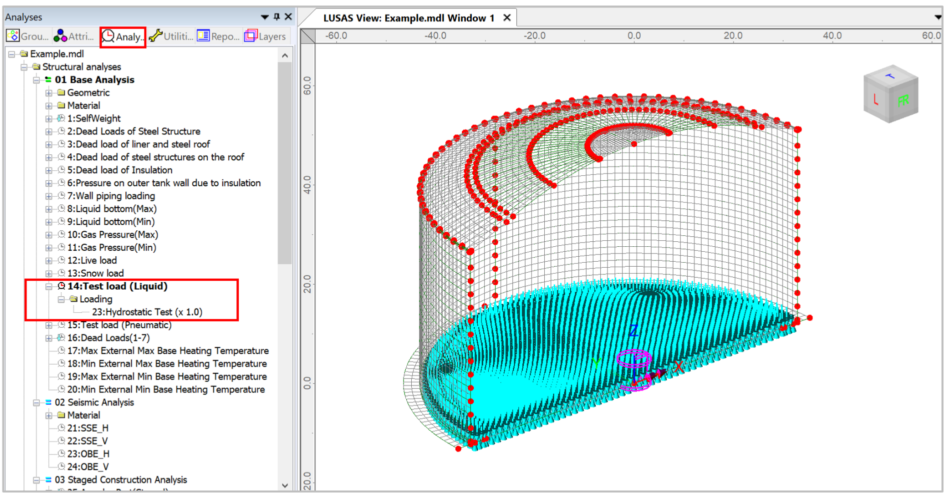


Fig 160 Test Load (Liquid Bottom) in a 3D Shell Static Analysis Model

Test load (Pneumatic)

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

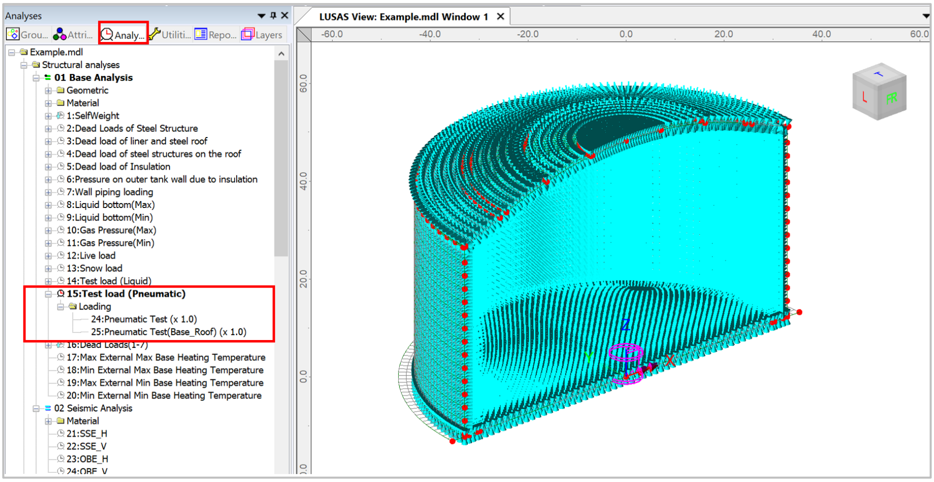
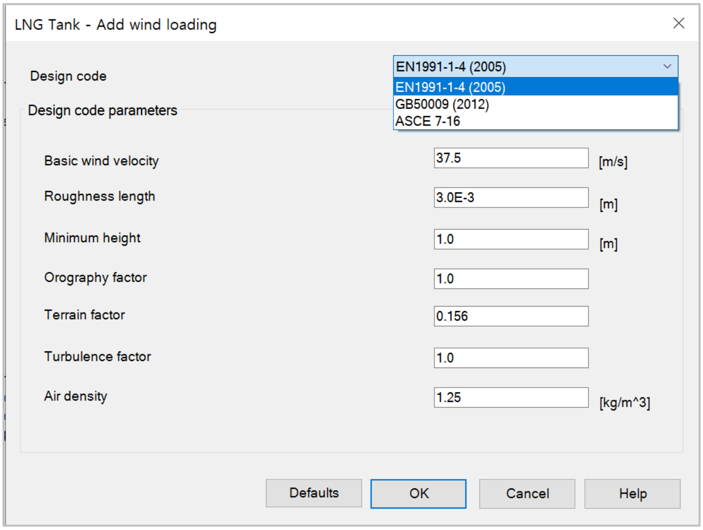


Fig 161 Test Load (Pneumatic) in a 3D Shell Static Analysis Model

Wind Load

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



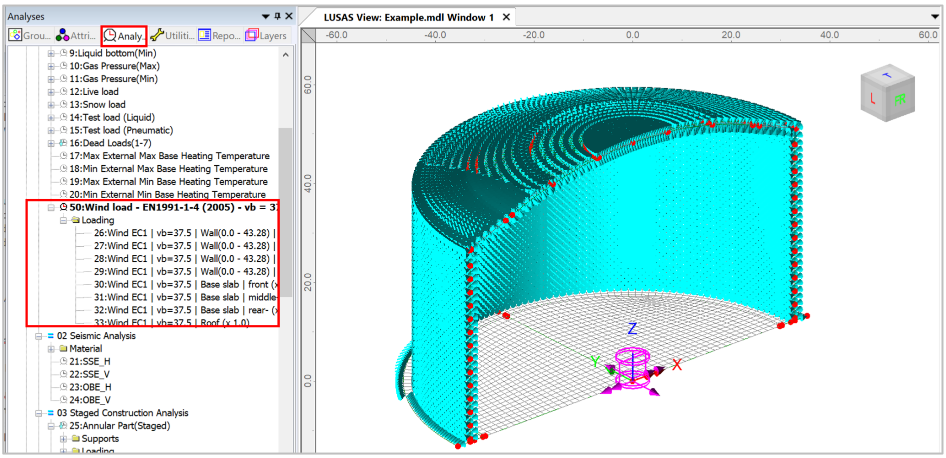


Fig 162 Wind Load in a 3D Shell Static Analysis Model

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.

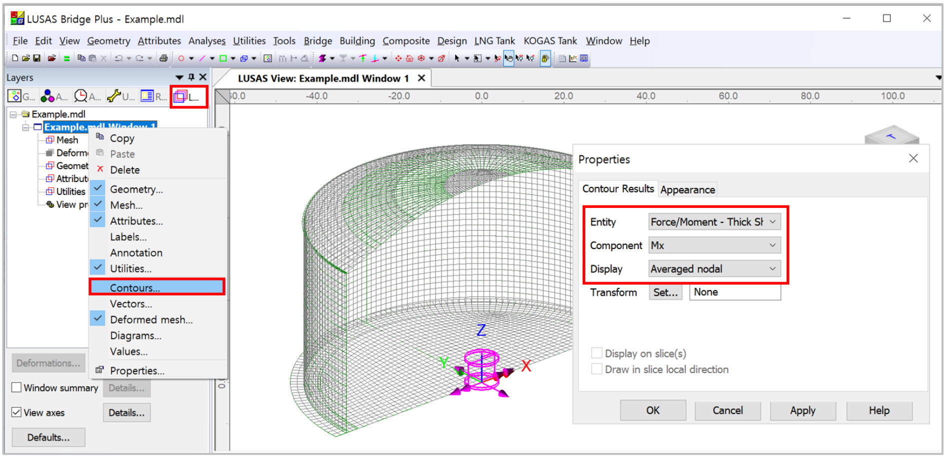


Fig 163 Selection for Contour Display in 3D Shell Model

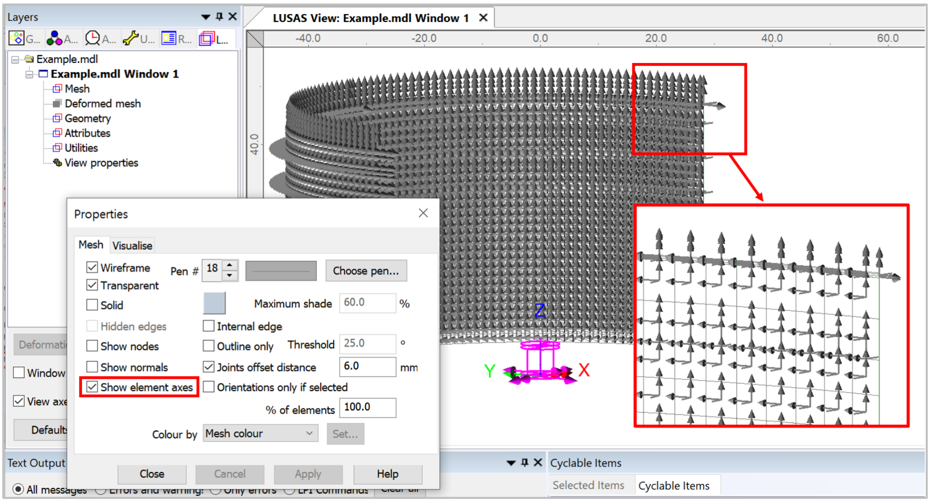


Fig 164 Element Local Axis in a 3D Shell Model

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

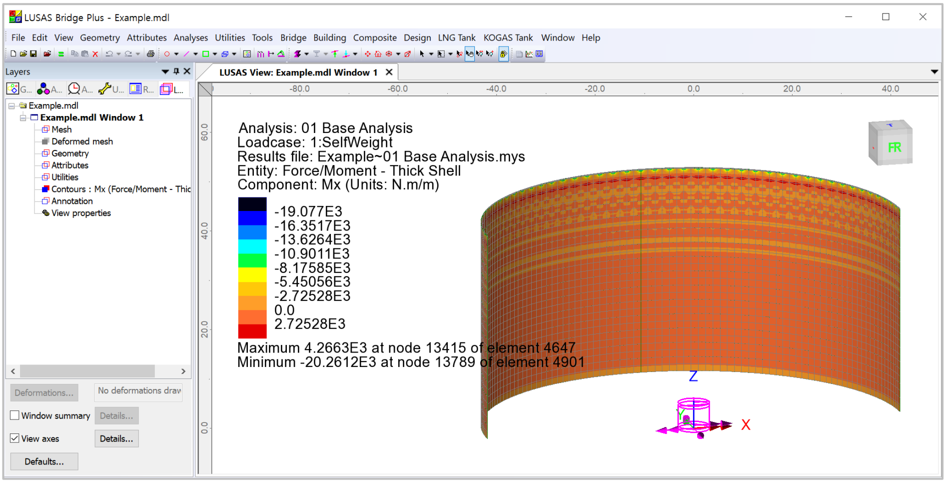


Fig 165 Mx Contour for Self Weight in a 3D Shell Model

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

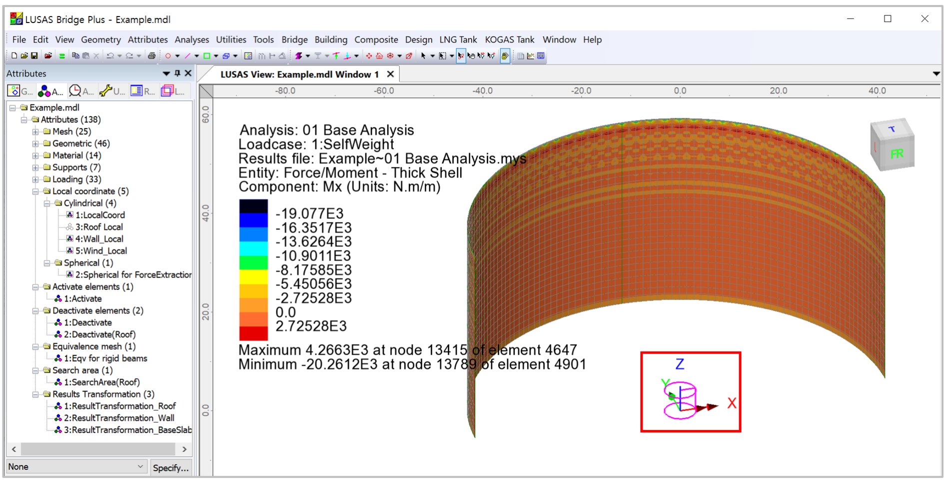


Fig 166 Local Coordinate in a 3D Shell Model

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

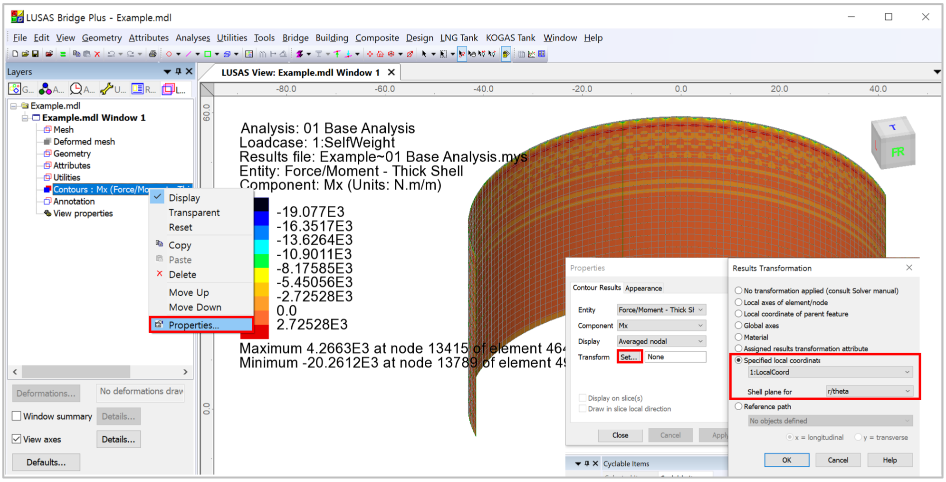


Fig 167 Contour Display using Local Coordinate in a 3D Shell Model

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

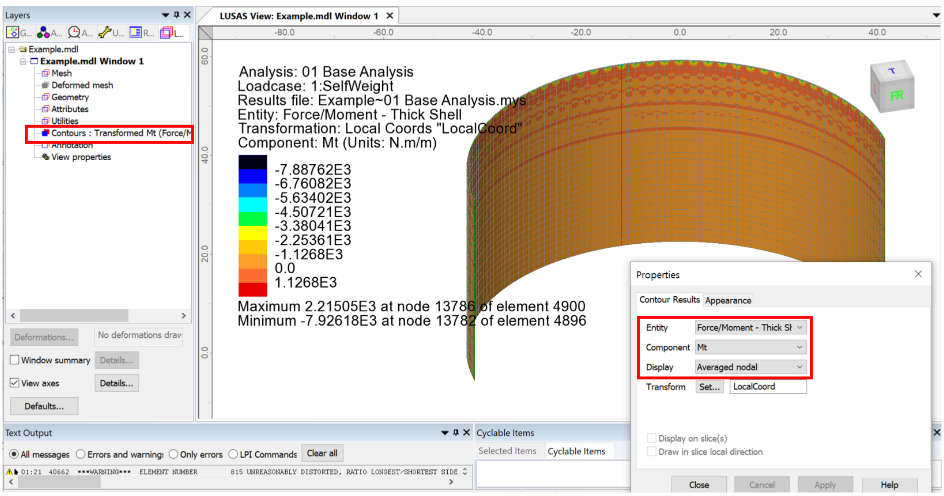
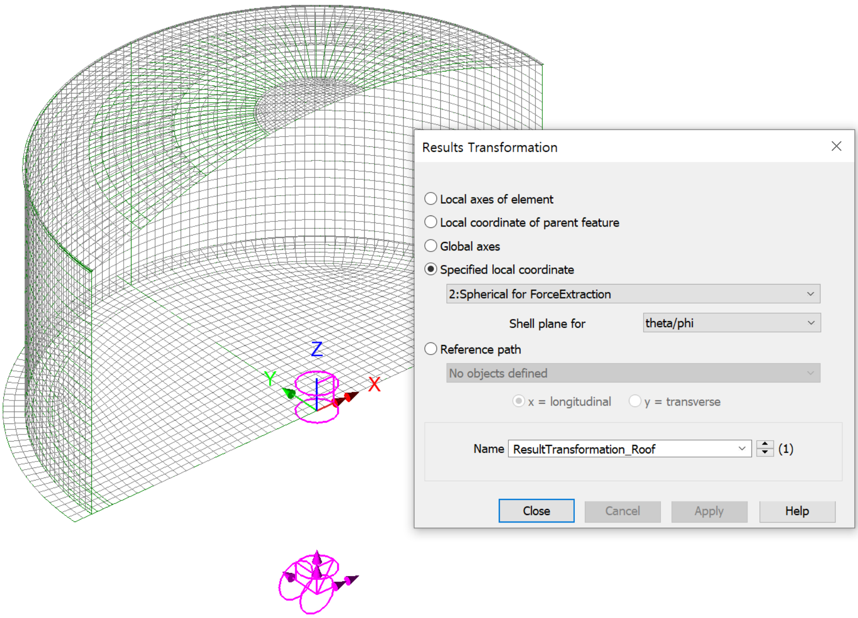
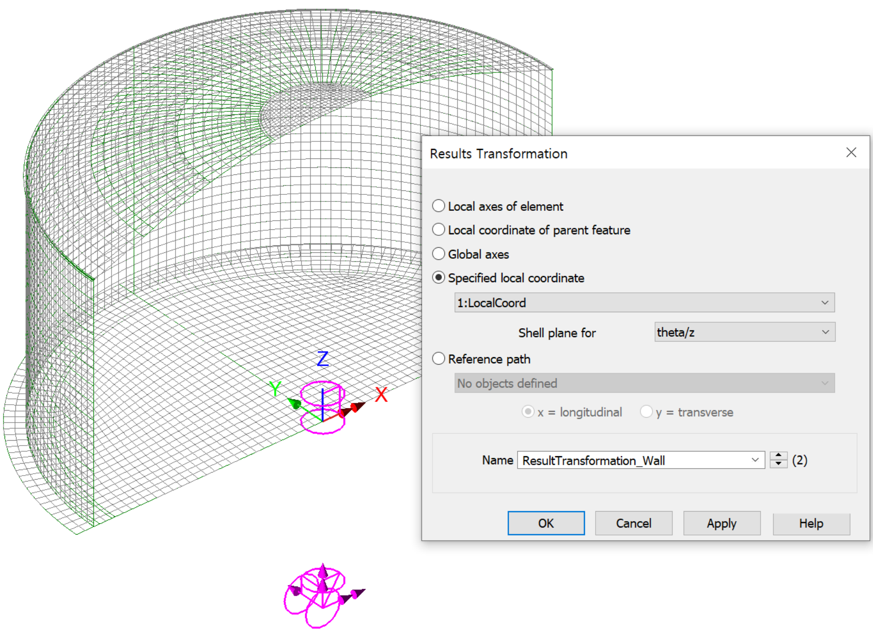


Fig 168 Mt Contour in a 3D Shell Model

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


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)
OK
Cancel
Apply
Help
Z
Y
X

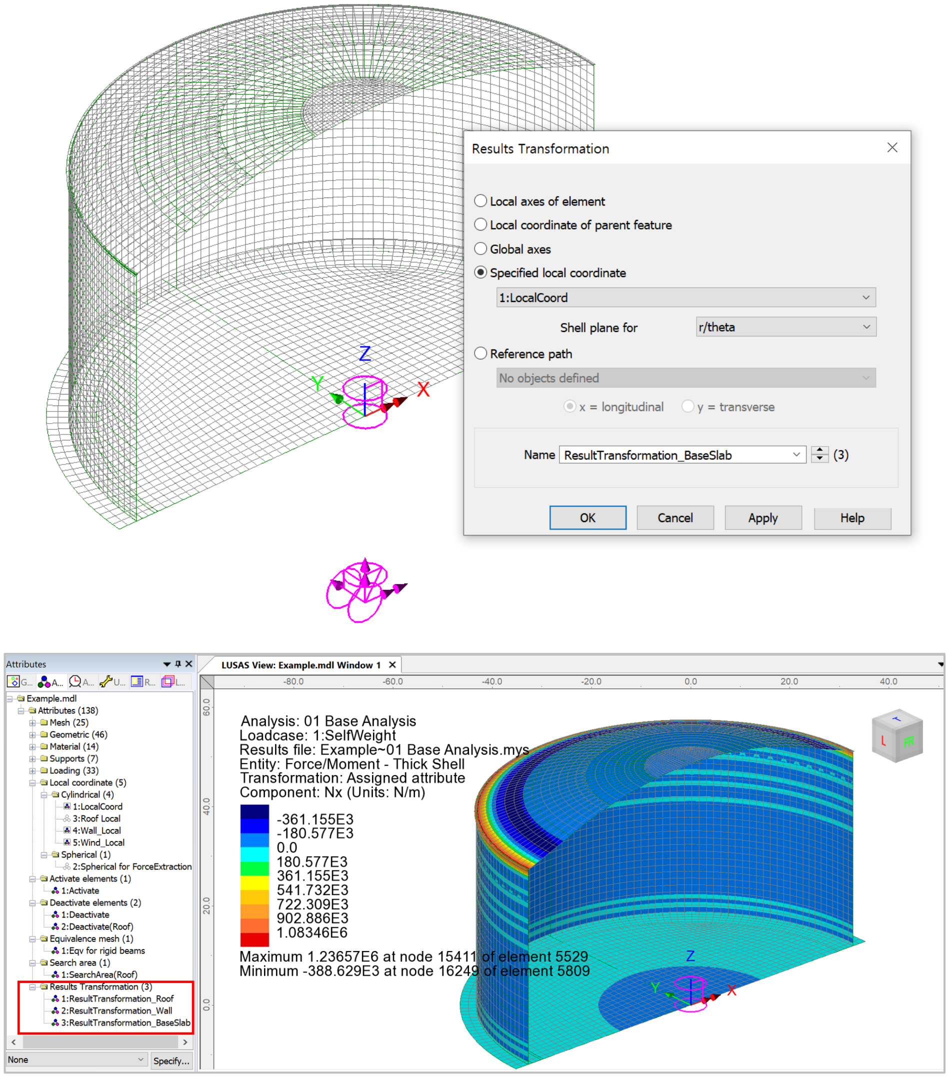


Fig 169 Results Transformation in a 3D Shell Model

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

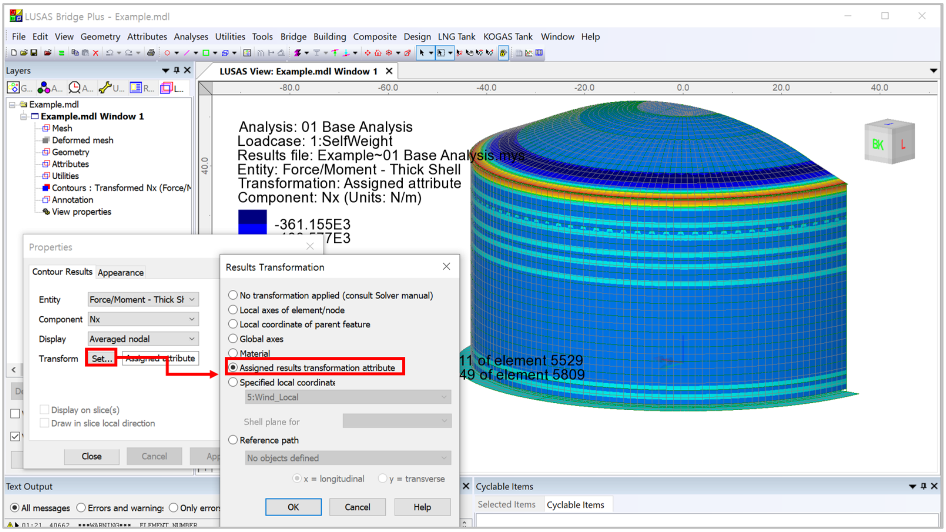


Fig 170 Contour Display using Results Transformation in a 3D Shell Model

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

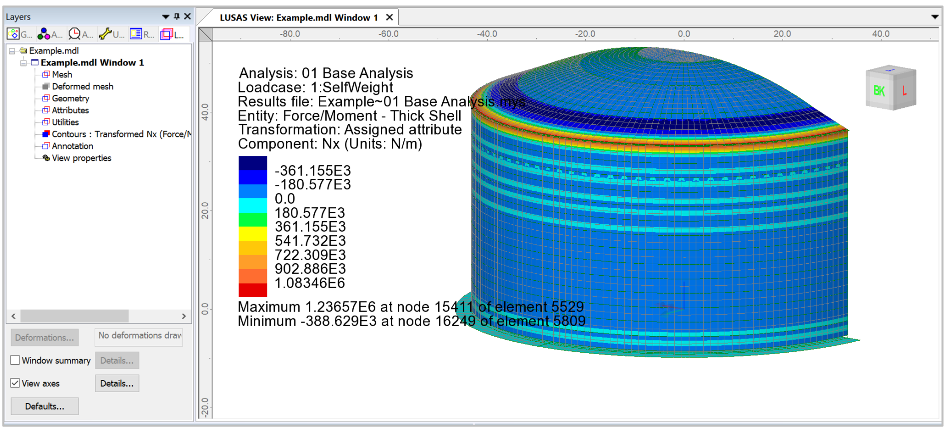


Fig 171 Nx Contours in a 3D Shell Model

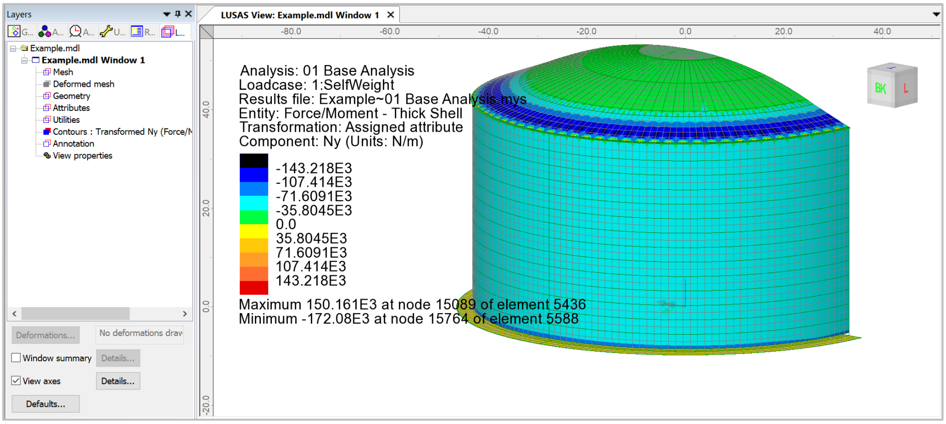


Fig 172 Ny Contours in a 3D Shell Model

Values

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

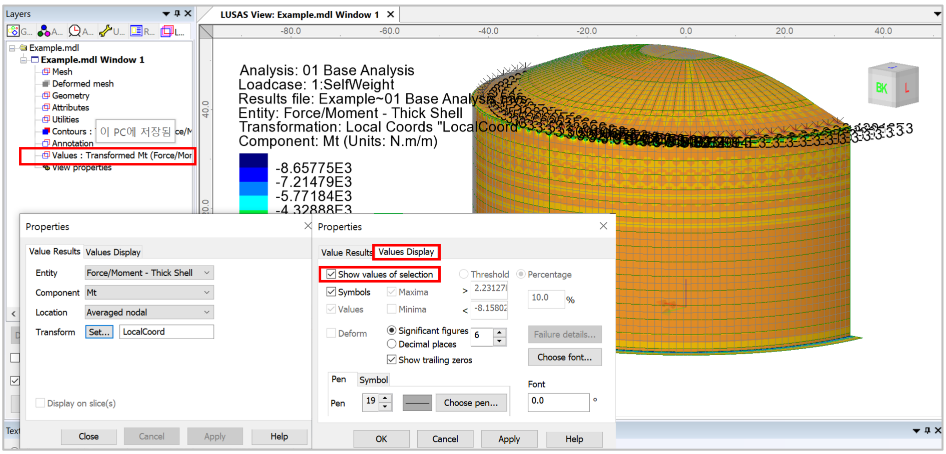


Fig 173 Value Display in a 3D Shell Model

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

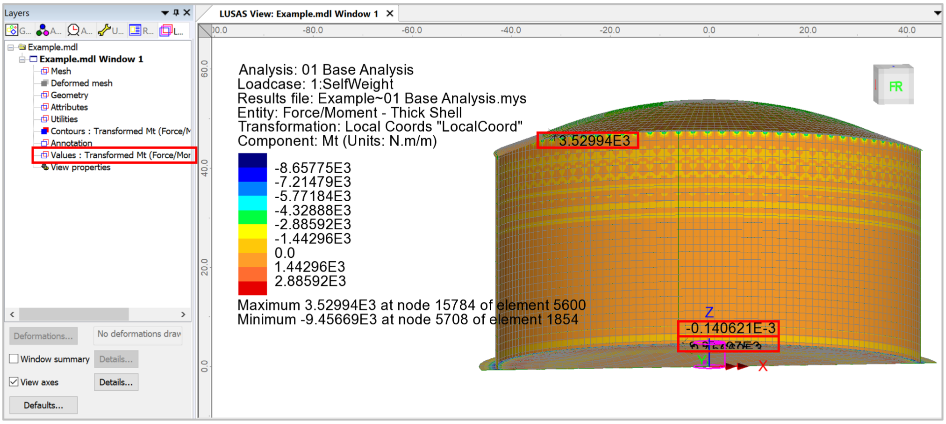


Fig 174 Values Displayed for Selected Nodes in a 3D Shell Model

Graph through 2D

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

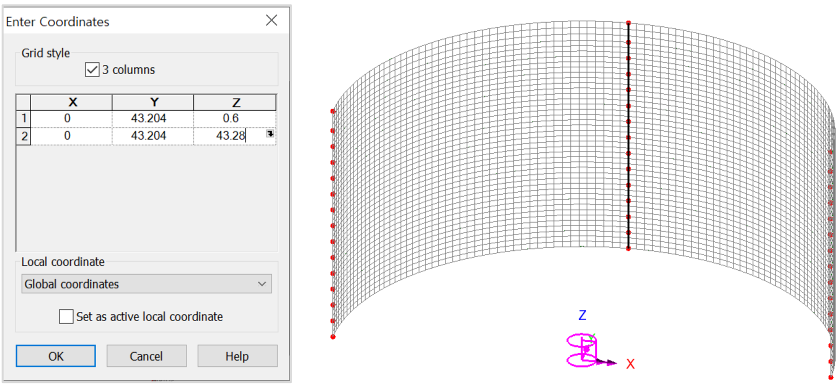


Fig 175 Line for Slicing Results in a 3D Shell Model

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

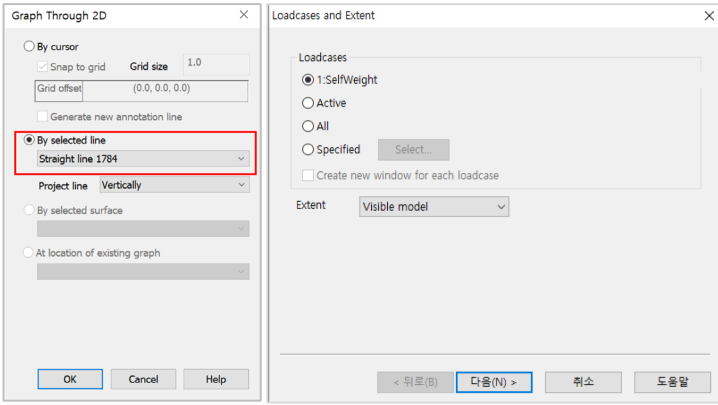


Fig 176 Graph Through 2D in a 3D Shell Model (1)

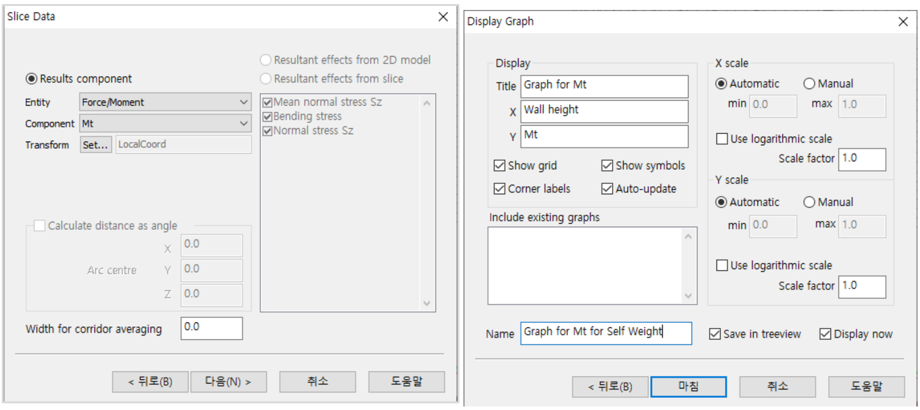


Fig 177 Graph Through 2D in a 3D Shell Model (2)

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

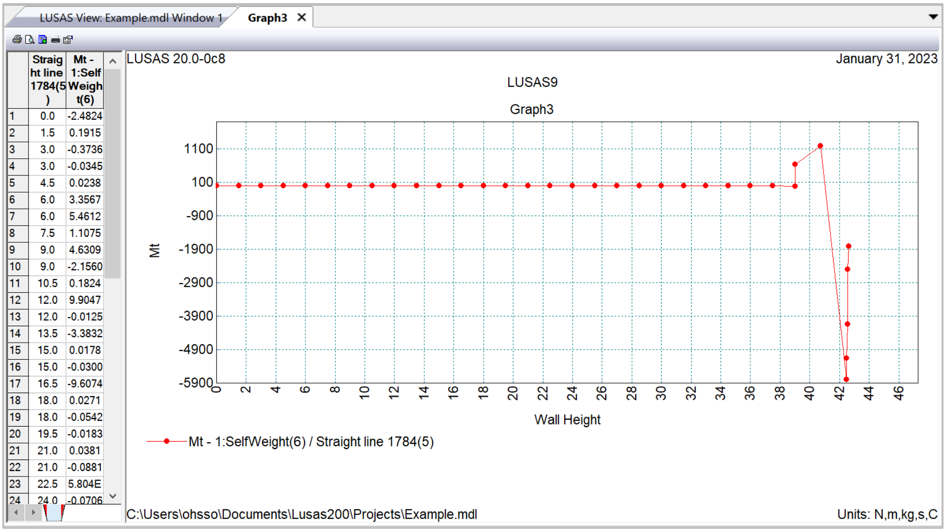


Fig 178 Mt Graph for Sliced Line in a 3D Shell Model

Export Forces to Excel (3D)

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

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

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

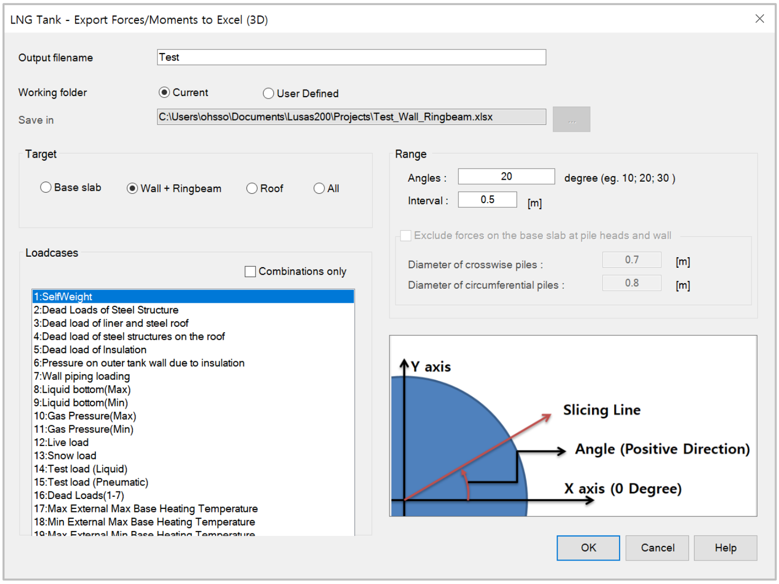


Fig 179 Export Forces for 3D Shell Model (1)

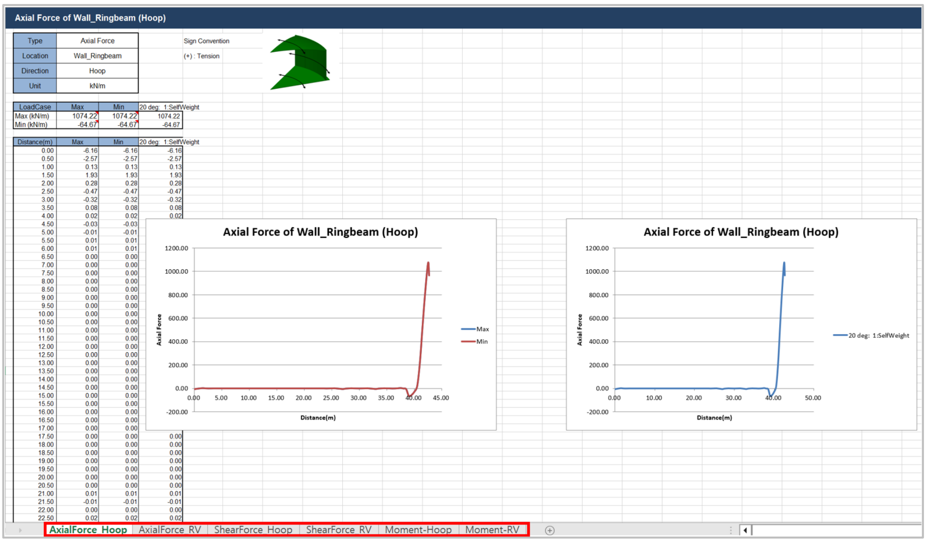


Fig 180 Section Force Spreadsheet for Self Weight

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

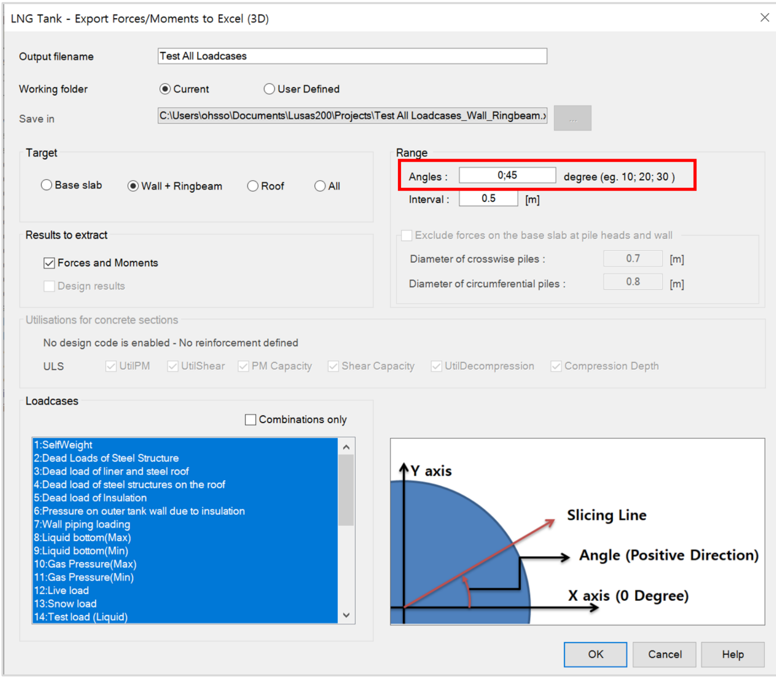


Fig 181 Export Forces for a 3D Shell Model (2)

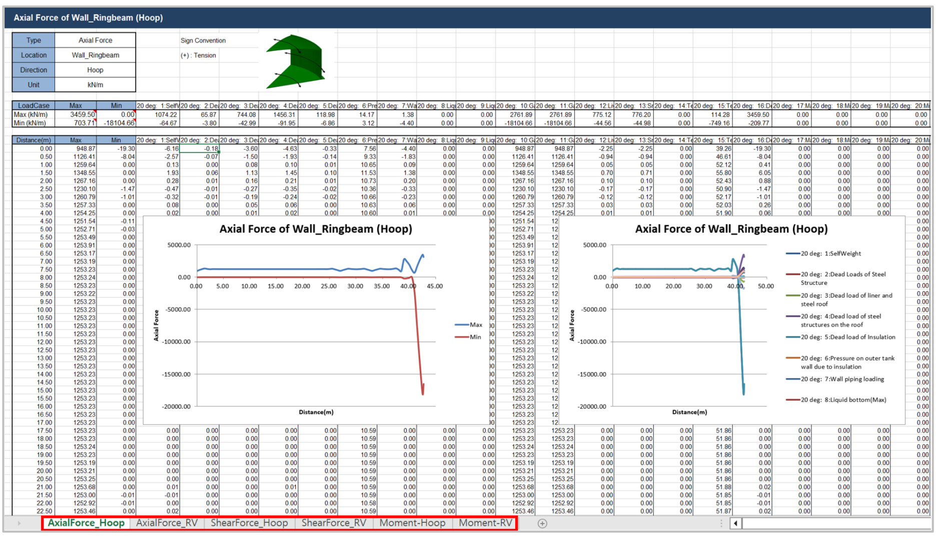


Fig 182 Section Force Spreadsheet for All Loadcases

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

Sign convention

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

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

LUSAS

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