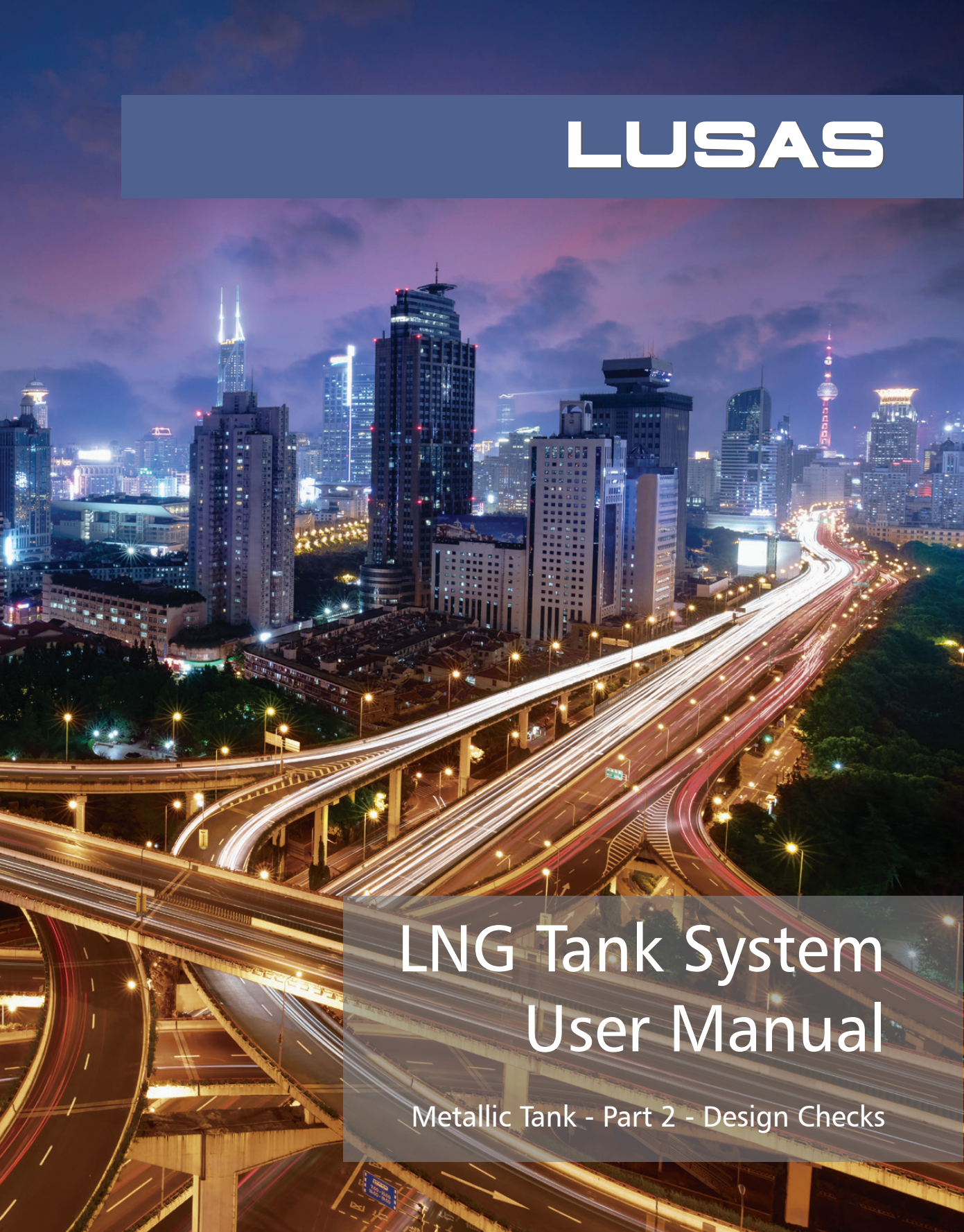
LUSAS



LNG Tank System User Manual

Metallic Tank - Part 2 - Design Checks

LNG Tank System

User Manual (MetallicTank)

Part 2 - Design Checks

LUSAS Version 20 : Issue 1

LUSAS

Forge House, 66 High Street, Kingston upon Thames,

Surrey, KT1 1HN, United Kingdom

Tel: +44 (0)20 8541 1999

Fax +44 (0)20 8549 9399

Email: info@lusas.com

http://www.lusas.com

Distributors Worldwide

Copyright ©1982-2023 LUSAS

All Rights Reserved.

Table of Contents

LNG Tank Design Checks 1

Base Model 3

Preparations ... 3

Build Base Model ... 4

Thermal Analysis 9

Thermal Analysis for Max Environmental Temperature ... 9

Thermal Analysis for Min Environmental Temperature ... 12

Update Base Model ... 15

Seismic Analysis 17

Preparation ... 17

Update Base Model ... 17

Staged Construction Analysis 25

Update Base Model ... 25

Stage Definition and Loadings ... 26

User Updates ... 28

CRSH Analysis 31

Preparation ... 31

Update Base Model ... 31

Stage Definition and Loadings ... 32

User Updates ... 32

Design Load Combinations 35

Template for Design Load Combinations ... 35

Update Base Model ... 36

Design Check 38

COP Parameters ... 38

Design Checks for Tank ... 40

Design Check Report ... 42

LNG Tank Design Checks

The LNG Tank System carries out design checks to supported design codes for specified load combinations.

Its use requires the **Microsoft Excel** spreadsheet application to be installed in advance for full functionality as certain applications may use it during the design or reporting process. For example, the Wizard imports the spreadsheet template for design load combinations.

Due to the low temperature of LNG, a thermal analysis requiring solid elements has to be performed and this often causes difficulties for load combinations that are based on structural results when the structural analysis is performed using shell elements. The LNG Tank System imports the temperatures and temperature gradients that are obtained from thermal analysis using solid elements into shell elements and allows all required load combinations to be assembled in a single model, to enable design check results to be obtained in an efficient way.

This manual focuses on the procedures involved in performing design checks using the LNG Tank System. A separate manual titled ‘LNG Tank System (Metallic Tank): Part 1 – Tank Wizard’ provides details of modelling concepts used to build the range of models supported.

User inputs shown in the manual ‘LNG Tank System (Metallic Tank): Part 1 – Tank Wizard’ are used in this manual unless otherwise stated.

Base Model

Preparations

User Inputs

Select the menu item **LNG Tank >Tank Definition**…

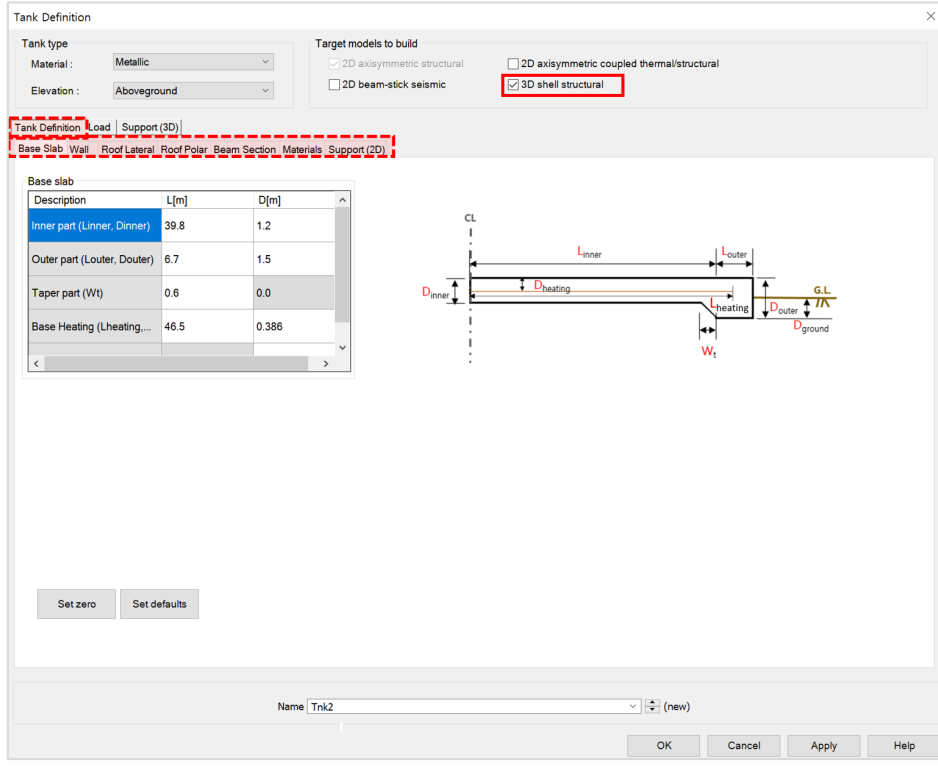


Fig 1 User Input for Tank Dimension and Load

'**3D Shell Structural**' should be checked from 'Target models to build'. Complete all the inputs and click **OK** to save the data with the name '**Tnk1**'.

Build Base Model

The Base Model for a design check can be built by selecting the menu item **LNG Tank> Create 3D Shell Model…**

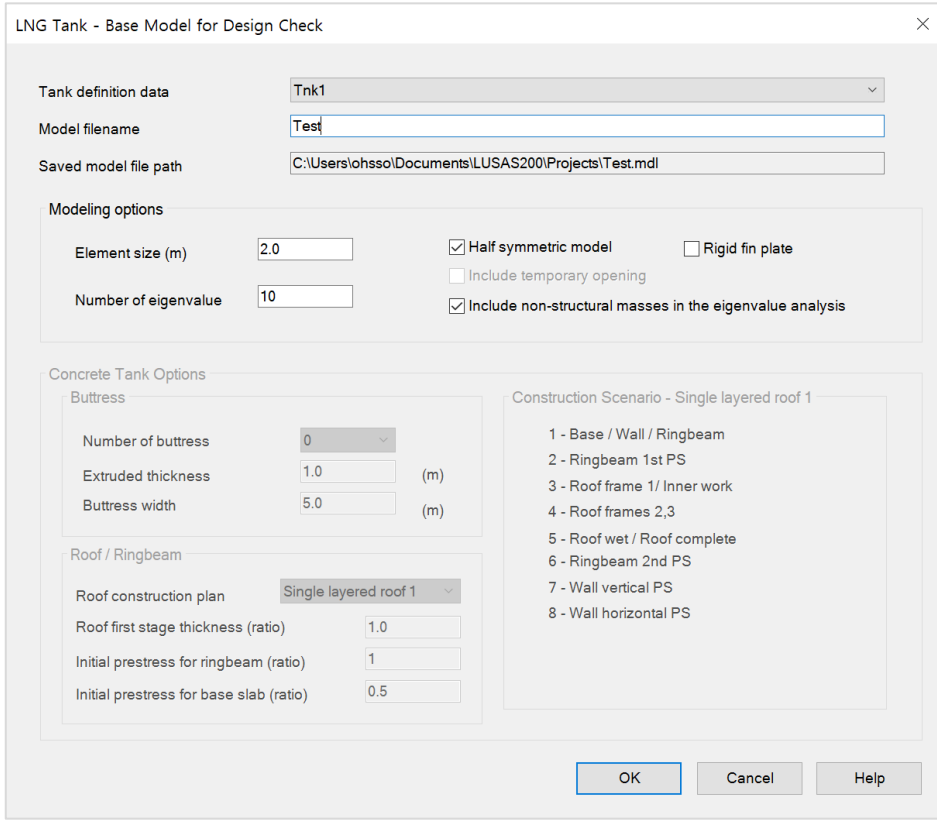


Fig 2 LNG Tank for Base Model for Design Check

- Depending on the selected 'Tank Type' in Tank Definition, relevant user inputs are enabled and not required user inputs are disabled.

The model built is shown below.

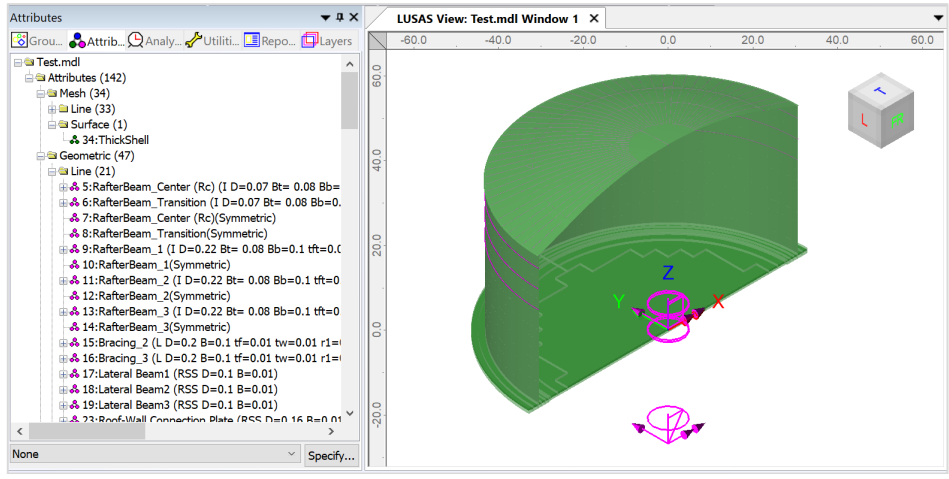


Fig 3 Base Model for Design Check

Analyses

A number of analyses having a total of 49 loadcases are set-up in the model.

- **Base Analysis** A static analysis. A total of 28 loadcases are provided including 4 thermal loading. Thermal loading can be added to the Base Analysis using '**LNG Tank > Add Loading> Thermal…**' The result of a separate thermal analysis is to be imported here.
- **Seismic Analysis** Equivalent peak seismic acceleration and hydrodynamic loading are to be added.
- **Staged Construction Analysis** 19 construction stages are defined with self-weight only.
- **CRSH Analysis** As for Staged Construction Analysis, combinations have been added to consider the pure effect from Creep and Shrinkage alone. This is obtained by subtracting the results in Analysis 5 (including CRSH) and the results in Analysis 4 (not considering CRSH).
- **Eigenvalue Analysis** To obtain Eigenvalue results.

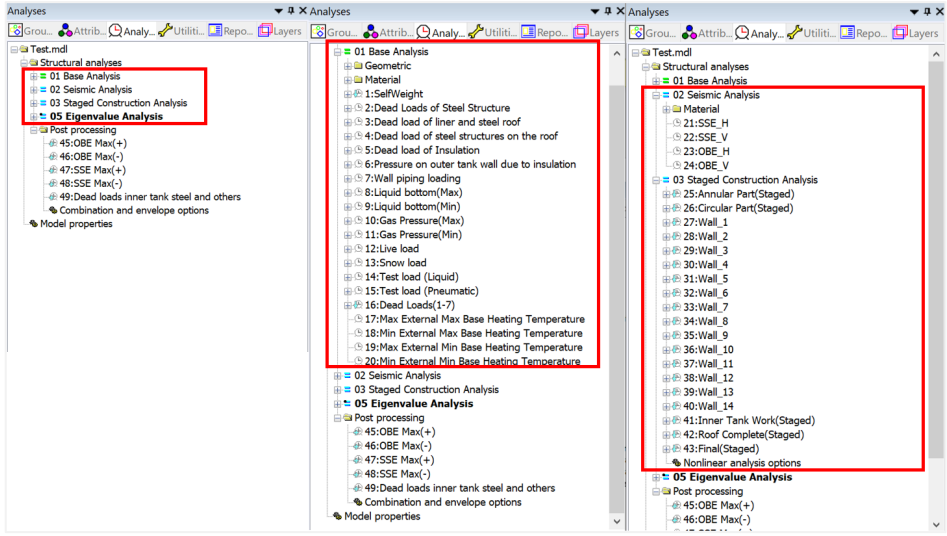


Fig 4 Analyses Available in Base Model

Loadings

Self weight is assigned in all analyses.

- In the Base Analysis, 20 loadcases are provided, including 4 thermal loading.
- In the other analyses (with the exception of the self weight) loading is not yet assigned.

Base Load Combinations

As seen in [Fig 4], 5 load combinations are pre-defined.

Template for load combinations

A template for use in defining design load combinations is saved in the LUSAS installation folder **LUSAS200\Programs (x86)\scripts\Tank** with the name of **Tank_Template_Combinations.xlsx**

Alternatively, a template can be downloaded by clicking the 'Template Download' button in the 'Design Load Combinations' dialog (**LNG Tank > Design Checks > Design Load Combinations**). The template for design load combinations will be saved in the current working folder with the name of **Tank_Template_Combinations.xlsx.**

Fig 5 Design Load Combinations

Thermal Analysis

Thermal analysis that requires the evaluation of heat transfer through thickness cannot be performed with shell elements, so a thermal analysis using 2D axisymmetric solid elements is required instead.

Thermal Analysis for Max Environmental Temperature

Select the **LNG Tank > Create 2D Model > Coupled Thermal/ Structural…** menu item, and assuming the default inputs are for maximum temperature, perform a 2D Thermal Analysis with the input data of '**Tnk1 – Max Temperature**'.

- Enter **Example - Max** for the model file name and click **OK**.

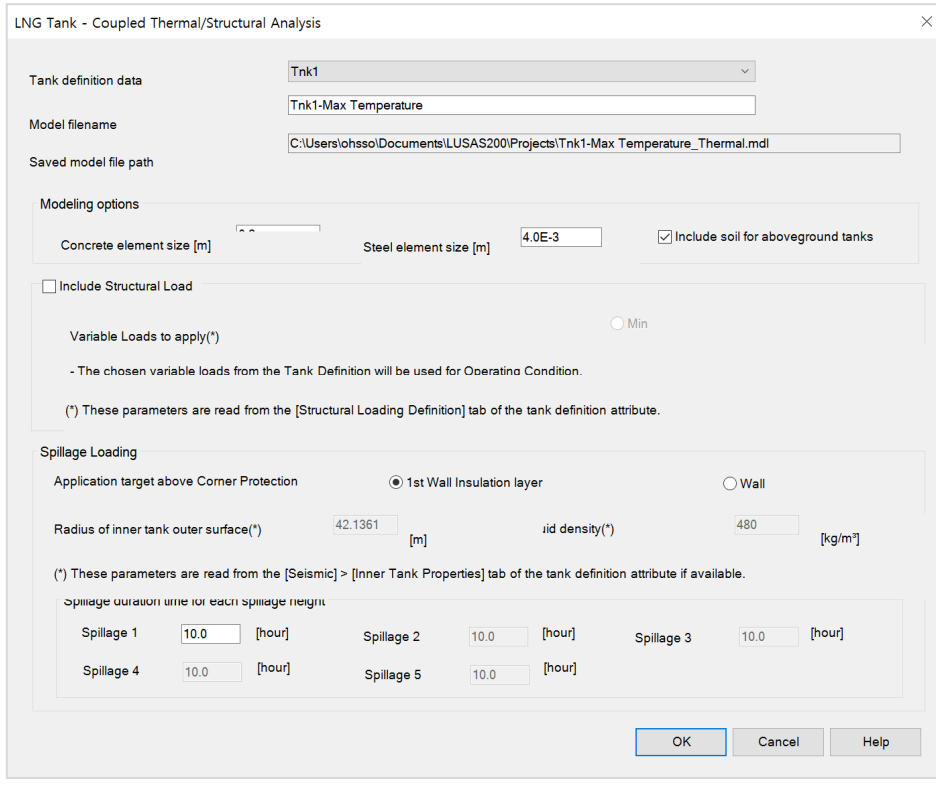


Fig 6 Dialog for Thermal Analysis (Max)

The current shell model will be closed and a new 2D axisymmetric solid model suitable for thermal analysis will be built.

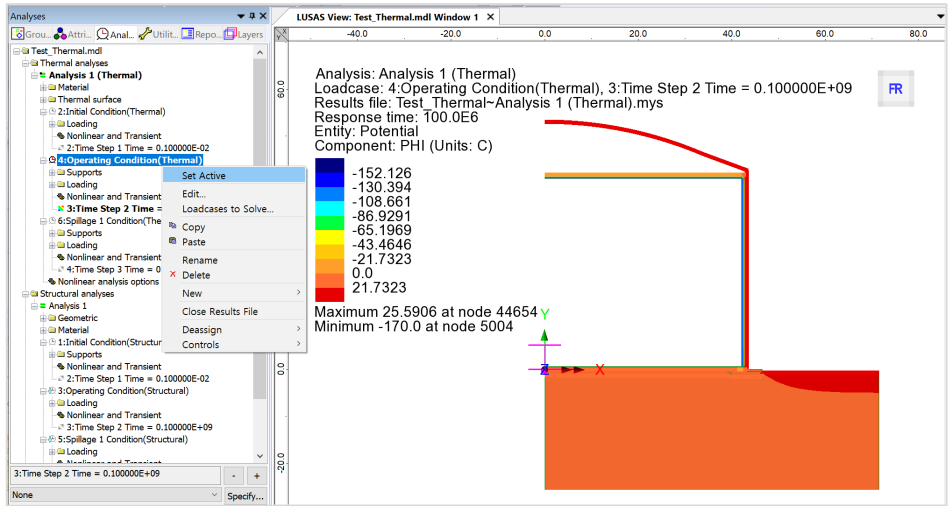


Fig 7 Thermal Analysis for Max Environmental Temperature

- Solve the model, and in the Analyses treeview, set the loadcase **Operating Condition (Thermal)** active.

- The Equivalent Uniform Temperature and Linear Temperature Gradient through thickness can be computed and saved in a spreadsheet by selecting the menu item **LNG Tank >Excel Tools> Export Temperatures…**

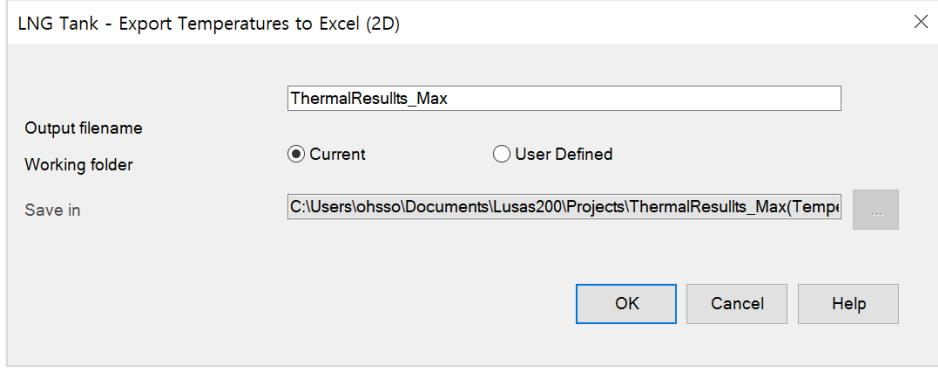
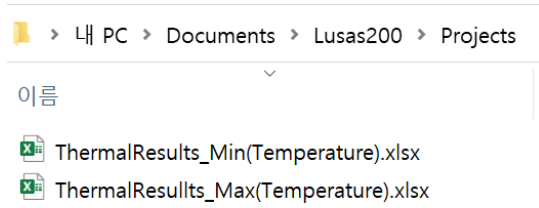


Fig 8 Extraction Thermal Max Results

In the current working folder, the file **ThermalResult-Max.xlsx** containing the extracted results for Roof/Slab/Wall is created.



Temperature of Roof

Type	Temperature
Location	Roof
Unit	Celsius

No of Slices	Gap (m)	Outer Radius (m)	Rad. Origin Y (m)	Min. Thickness (m)		
92.00	0.50	86.41	-33.36	0.01		

Distance (m)	Inner Temp (°C)	Outer Temp (°C)	Thickness (m)	Average Temp (°C)	Linear Gradient (°C/m)	Vertical Distance (m)
0.00	25.27	25.27	0.01	25.27	-0.15	0.00
0.51	25.32	25.32	0.01	25.32	-0.34	0.00
1.01	25.30	25.30	0.01	25.30	-0.33	0.01
1.52	25.32	25.33	0.01	25.32	-0.34	0.01
2.03	25.30	25.30	0.01	25.30	-0.31	0.02
2.53	25.32	25.32	0.01	25.32	-0.34	0.04

Fig 9 Extracted Thermal Results in a Spreadsheet

Thermal Analysis for Min Environmental Temperature

To perform additional thermal analyses for a minimum environment temperature, re-select the menu item **LNG Tank >Tank Definition…**

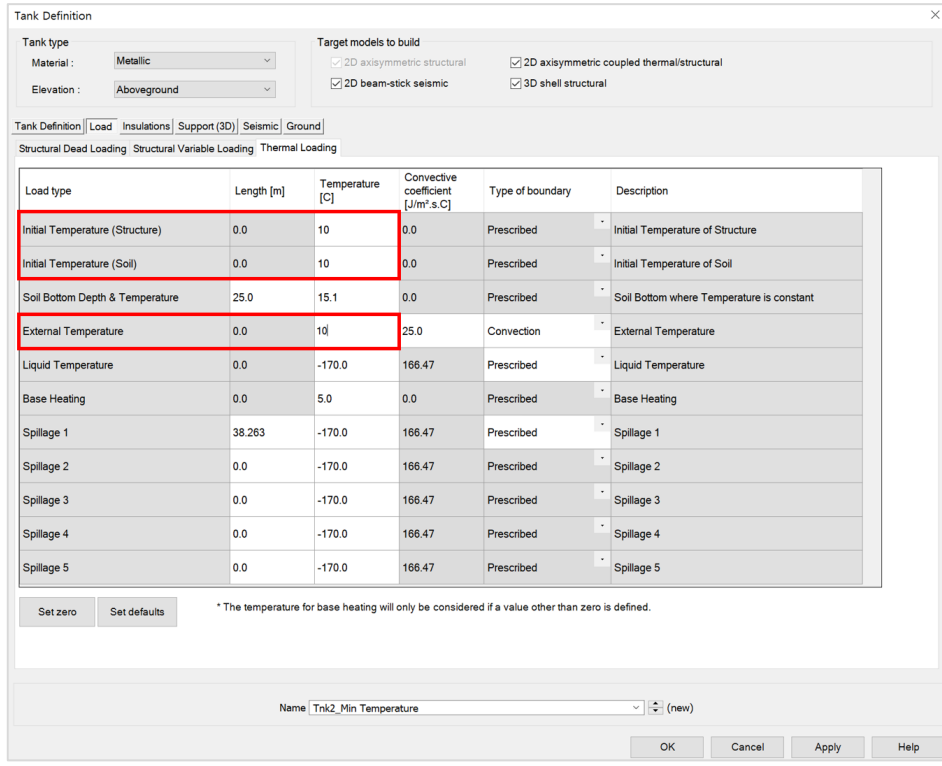


Fig 10 User Input for Min Environmental Temperature

- Enter **10 (**Celsius) for both the **Initial Temperature (Structure), Initial Temperature (Soil)** and the **External Temperature**. The **Initial Temperature (Structure)** is the temperature at the time when concrete is poured, hence zero stress will be occurred at this temperature.
- Save these inputs with name '**Tnk2-Min Temperature**'.
- Select the menu item **LNG Tank > Create 2D Model> Coupled Thermal/Structural…** and enter **Example - Min** for the model file name, then click **OK**.

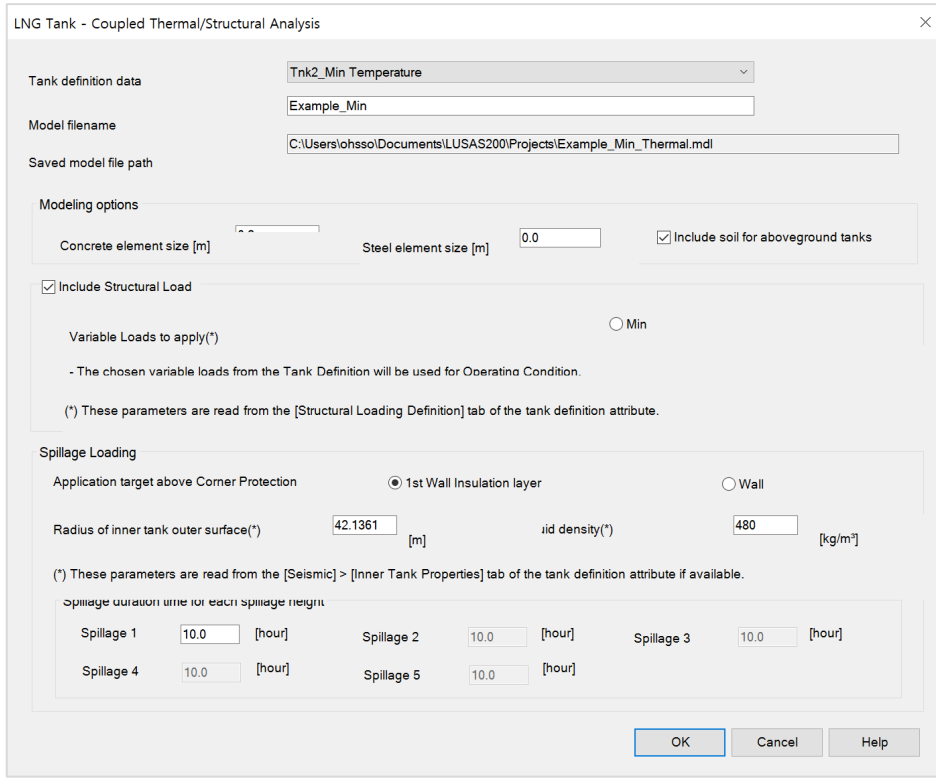


Fig 11 Dialog for Thermal Analysis (Min)

- Solve the model, and in the Analyses treeview, set the loadcase **Operating Condition (Thermal)** active.
- Export results into a spreadsheet by selecting the menu item **LNG Tank > Excel Tools> Export Temperatures…**

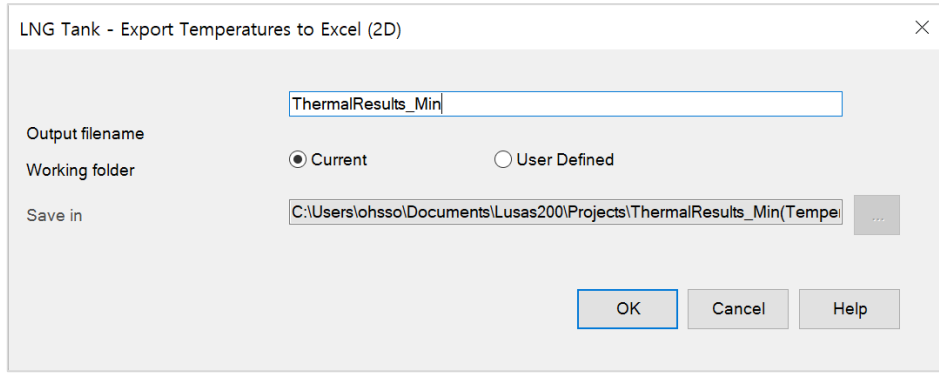


Fig 12 Extraction Thermal Min Results

Update Base Model

The thermal analysis results can be converted to 3D loading for load combinations.

- Close the thermal model and open the shell model of '**Test.mdl**'.
- Select the menu item **LNG Tank > Add loading> Thermal…** then select the spreadsheets for thermal analysis results and click **OK**.

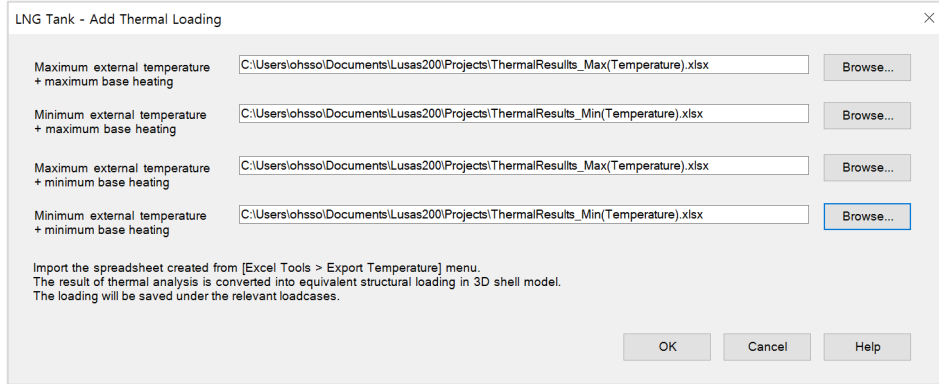


Fig 13 Dialog for Adding Thermal Loading

The thermal analyses results are converted into equivalent structural temperature loadings in the 3D shell model.

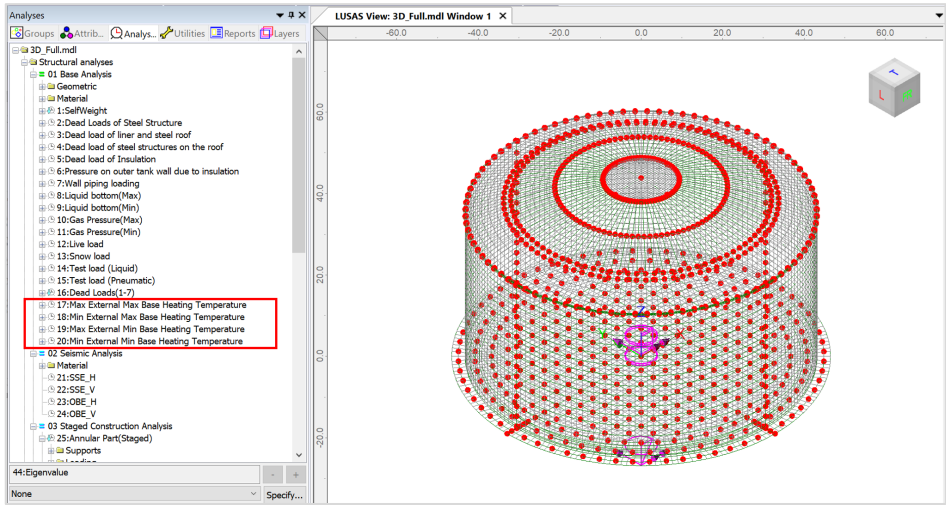


Fig 14 Temperature Loading from Thermal Analyse

Seismic Analysis

Preparation

A seismic analysis considering fluid-soil-structure interaction under seismic action should be carried out prior to the 3D shell model investigation. The inertial and hydrodynamic peak effects obtained from a seismic analysis can be transformed to equivalent static loading for a 3D model in the form of accelerations that will act on the structural masses and other structural loadings.

Update Base Model

Both OBE and SSE loadings can be defined by selecting the menu item **LNG Tank > Add loading> Seismic…**

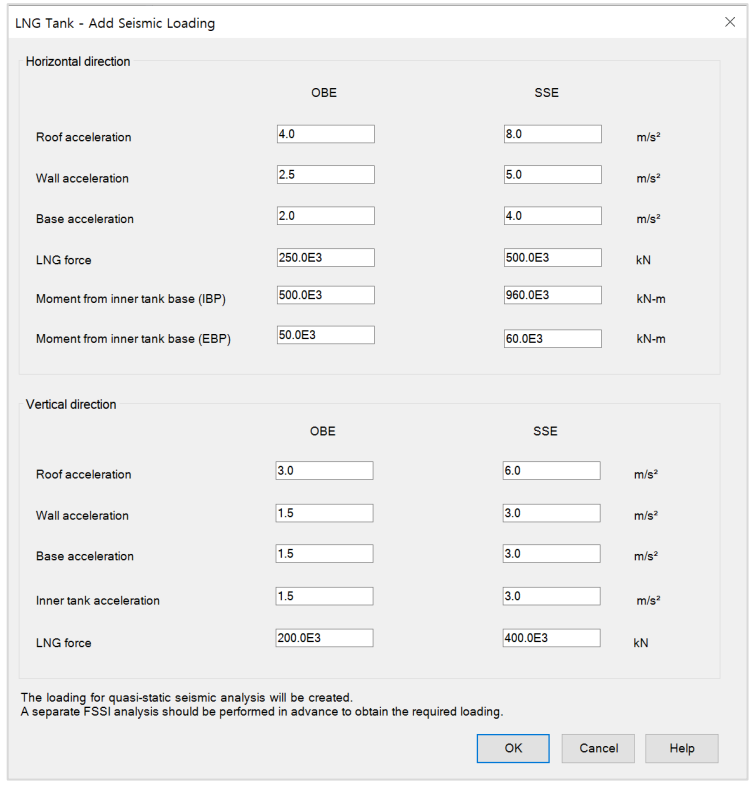


Fig 15 Dialog for Adding Seismic Loadings

Horizontal Loadings

Based on the given inputs, the loadings are defined as shown below. The acceleration loadings are directly used for defining Body Force loading in Modeller, and the other loadings are converted to equivalent structural loadings.

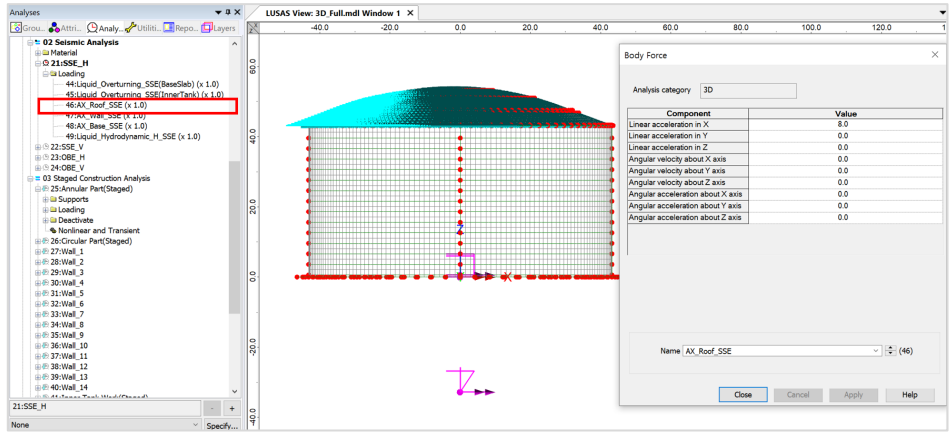


Fig 16 Horizontal Seismic Loading for Roof (SSE)

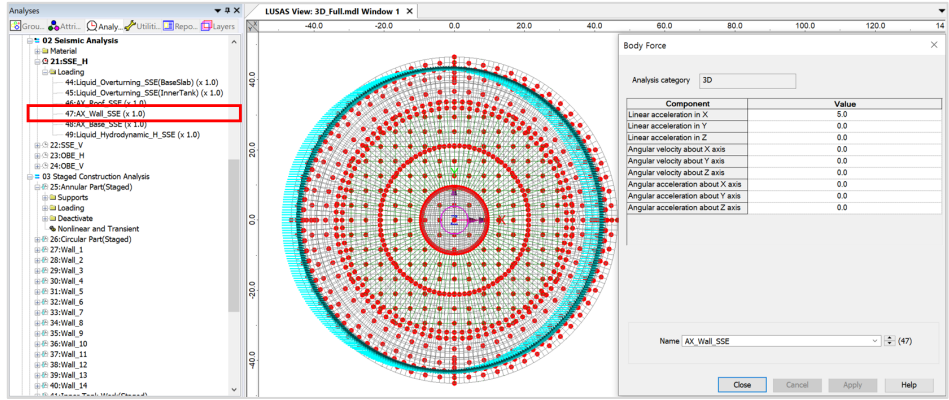


Fig 17 Horizontal Seismic Loading for Wall (SSE)

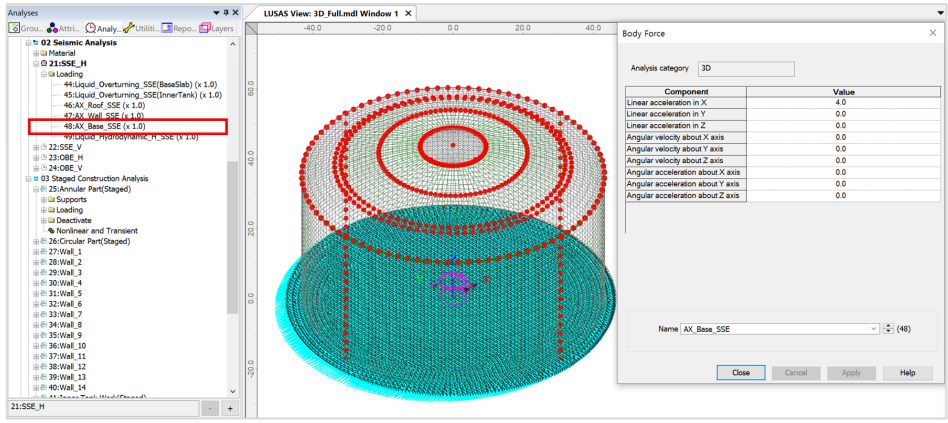


Fig 18 Horizontal Seismic Loading for Base Slab (SSE)

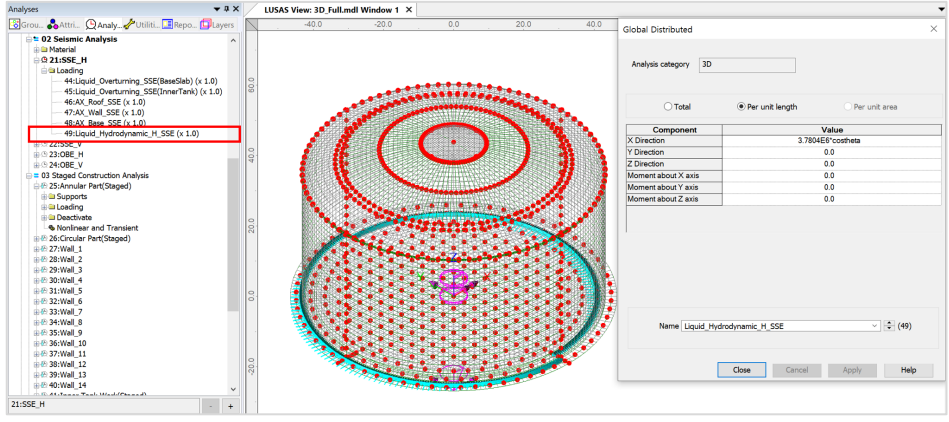


Fig 19 Horizontal Seismic Loading of Liquid Force (SSE)

- ❑ The force of the liquid is transferred to the Base Slab through the inner tank, so the loading is applied at the location of the inner tank wall.
- ❑ The total force defined from user input is 500E3 kN in the global X direction, however the pressure of the liquid acts perpendicular to the inner tank wall surface with an intensity following a cosine variation.
- ❑ A cylindrical local coordinate system is applied to the lines to ensure loading is in a radial direction.

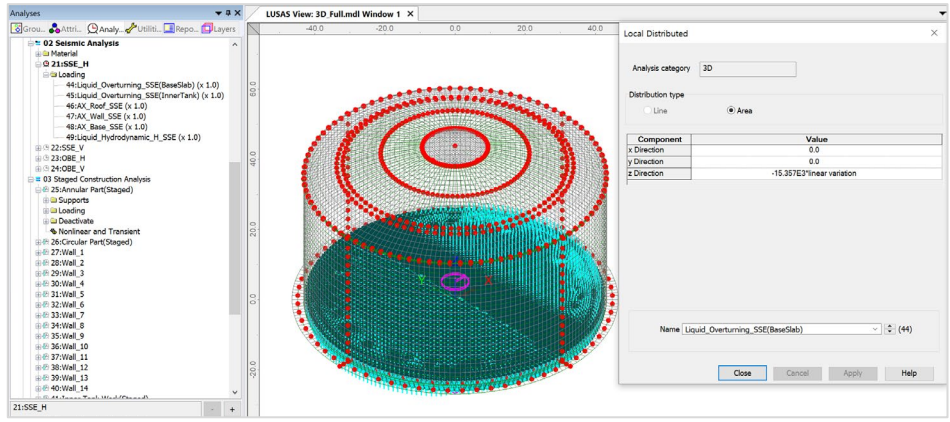


Fig 20 Horizontal Seismic Loading of Overturning Moment from Base Slab (SSE)

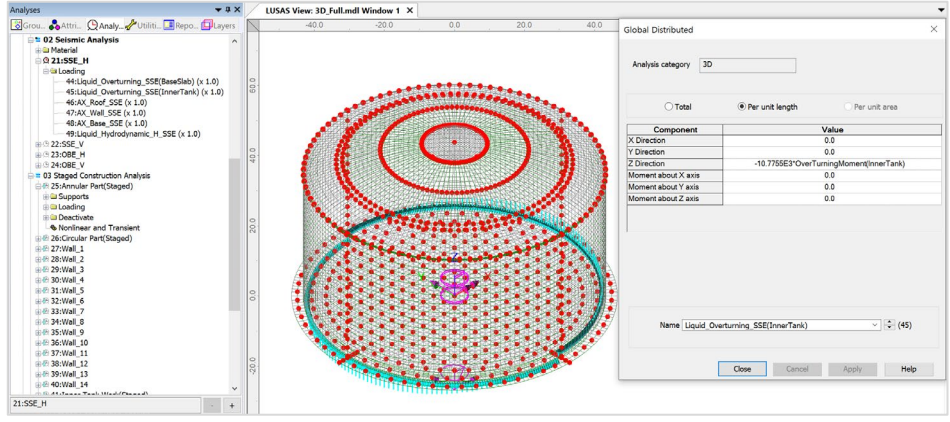


Fig 21 Horizontal Seismic Loading of Overturning Moment from Inner Tank (SSE)

Vertical Loadings

The acceleration loading is directly used to define Body Force loading in Modeller except for the acceleration for the Inner Tank. As the Inner Tank is not included in the meshed model, the loading is converted to equivalent structural loading.

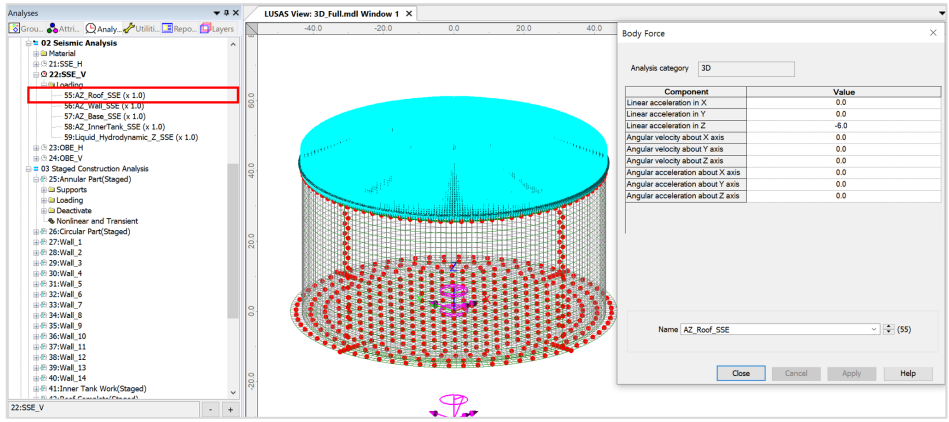


Fig 22 Vertical Seismic Loading for Roof (SSE)

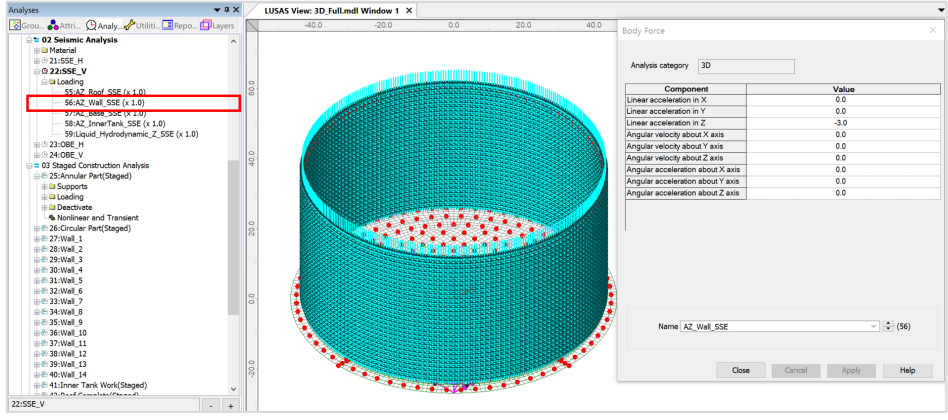


Fig 23 Vertical Seismic Loading for Wall (SSE)

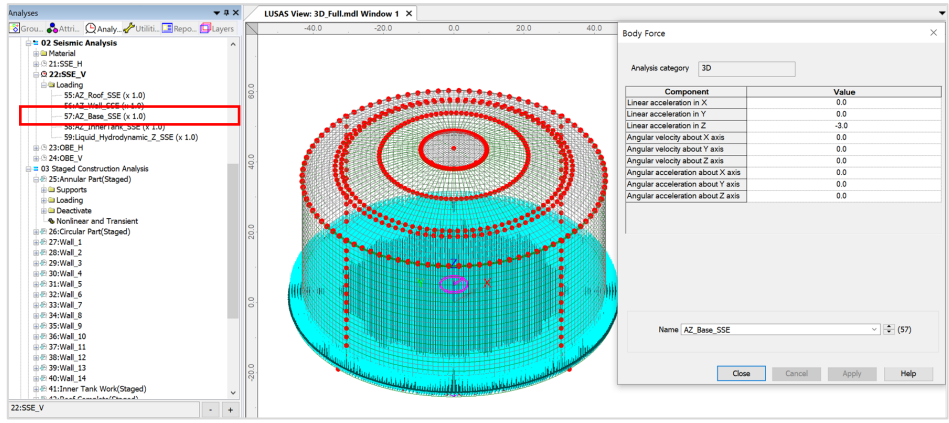


Fig 24 Vertical Seismic Loading for Base Slab (SSE)

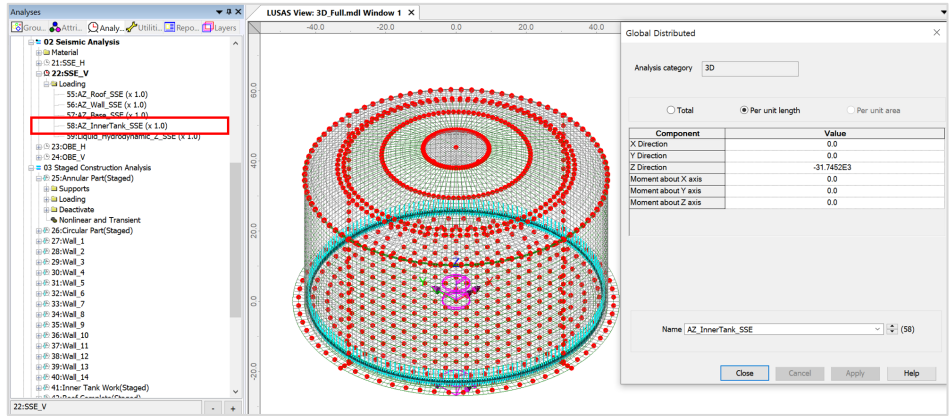


Fig 25 Vertical Seismic Loading for Inner Tank (SSE)

- ❑ The inner tank is not included as structural elements, so the vertical loading from the inner tank should be converted for structural loading.

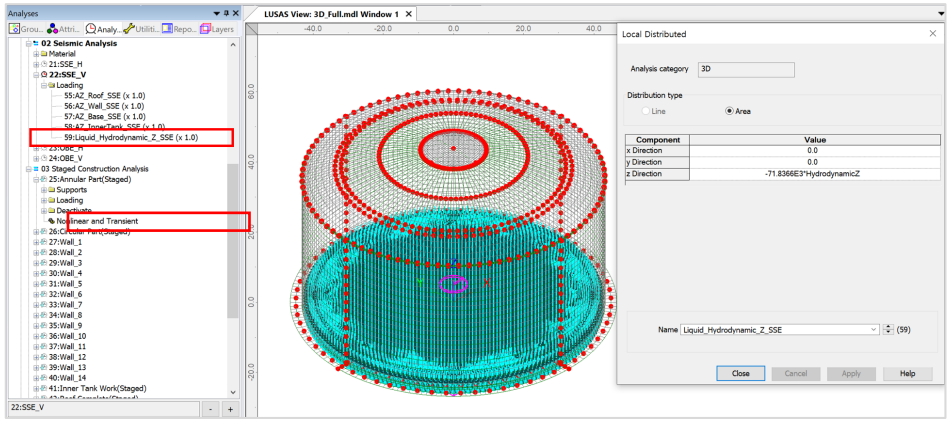


Fig 26 Vertical Seismic Loading for Liquid (Hydrostatic, SSE)

- The given liquid pressure is 400E3 kN is converted as a distributed load for 3D model.

Staged Construction Analysis

Update Base Model

By selecting the menu item **LNG Tank > Add Loading > Staged Construction Loads…** the loadings for Staged Construction Analysis will be created and assigned to the appropriate loadcases. Dead loads will reference the data defined at the previous step of 'Base Model', hence no further input for loading is required.

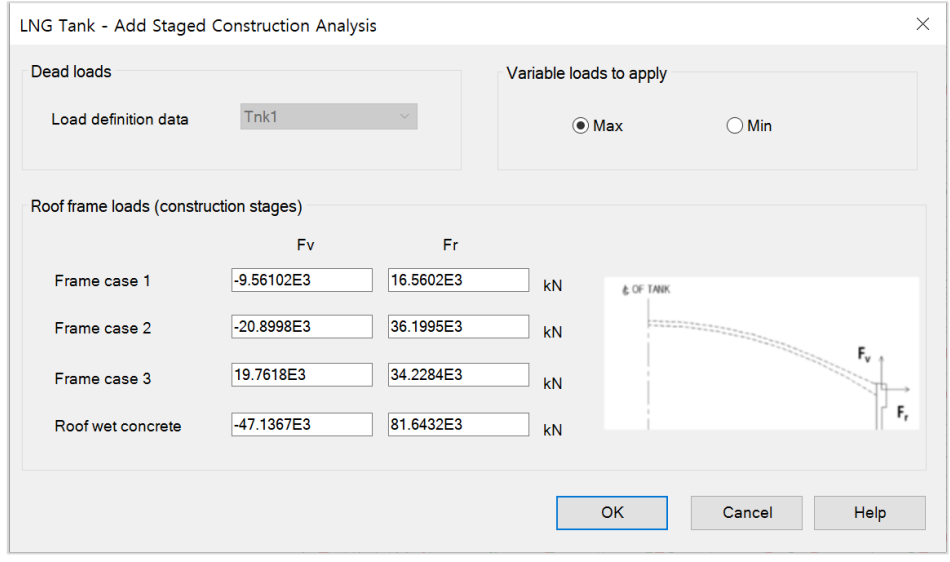


Fig 27 Dialog for Adding Staged Construction Analysis

Stage Definition and Loadings

The following stages and loadings are ready.

Stage	Description	Note
No. 1	Annular part	
No. 2	1) + Circular part	
No. 3~16	2) + Wall is added up in stages	
No. 17	16) + Inner Tank Work	
No. 18	17) + Roof Complete	
No. 19	18) + Final	

Table 1 Sequence of Construction Stages

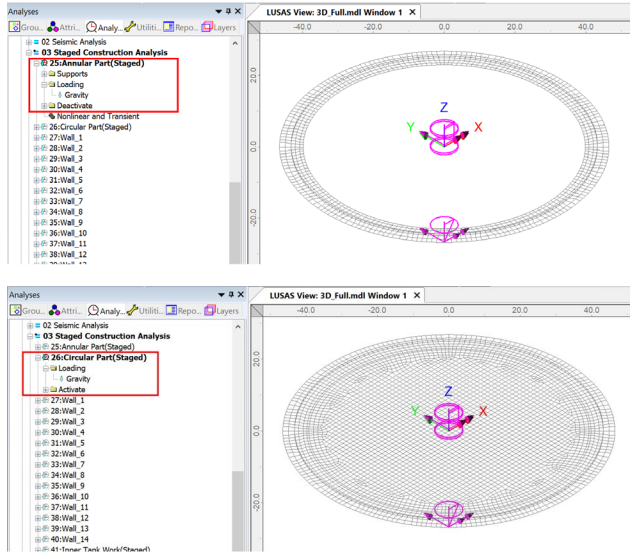


Fig 28 Stage 1 ~ 2 of Staged Construction Analysis

Stage 1 builds the annular part of the slab.

Stage 2 adds the central part of slab.

Note that self weight is always assigned when a new part of the structure is added. Loading defined and assigned at a stage is inherited by the subsequent stages.

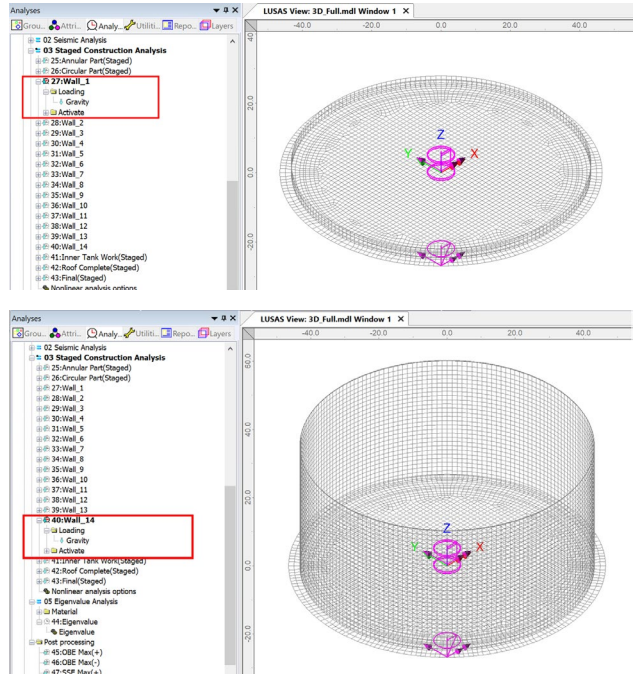
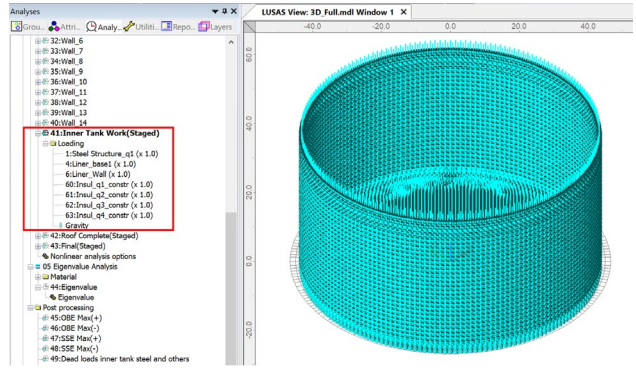


Fig 29 Stage 3 ~ 16 of Staged Construction Analysis

At **Stage 16,** the Wall construction is complete.



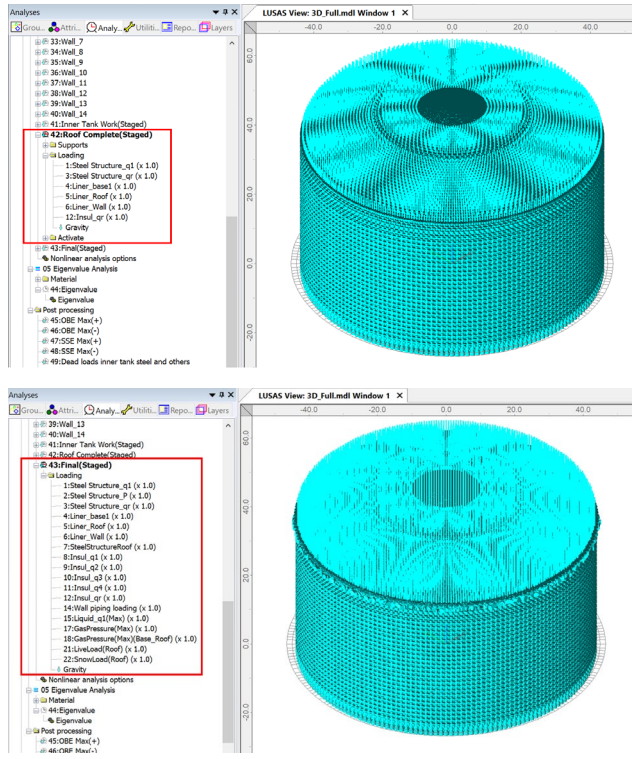


Fig 30 Stage 17 ~ 19 of Staged Construction Analysis

In a double steel tank, it is assumed that the roof is built in one go and no prestress is considered.

Stages 16 assumes that the Inner tank work is completed. At this stage, dead load for insulations acting on slab and wall will be assigned.

Stages 17 assumes that the roof is completely built. At this stage, the rest of dead load for insulation acting on Roof will be assigned.

Stage 19 is the final stage. The structures are complete as built, and all loadings for the operating condition are added.

User Updates

Construction Sequence

If required, additional loadings or stages can be added.

To duplicate loading types within the Analyses treeview, the Copy and Paste options can be used as shown below. This will create additional stages (loadcases) and include all loadings previously assigned to the copied loadcase.

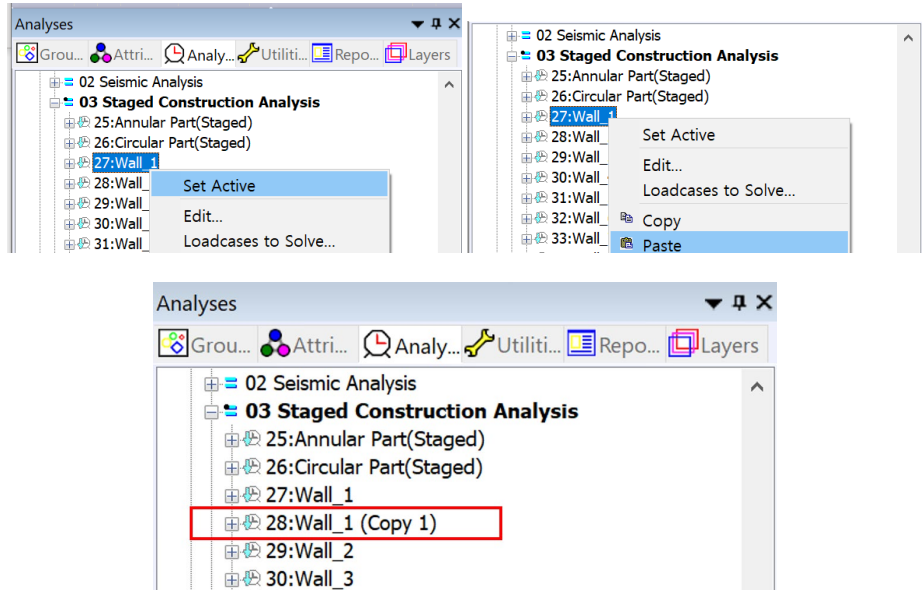


Fig 31 Copy and Paste of Stages (Loadcases)

The required activation/deactivation/loadings/support can now be assigned or removed for this stage. The loadings at other loadcases can be also copied and pasted in the same way if required.

CRSH Analysis

Preparation

CRSH Analysis considers the effect of creep and shrinkage changing with time and requires a nonlinear creep and shrinkage material model to be used. This is defined by selecting the **Attributes > Material > Concrete (Nonlinear)** menu item.

- Select '**EN 1992 (2004-2014)'**, accept the default inputs, and save with name of '**CRSH EN1992'**.

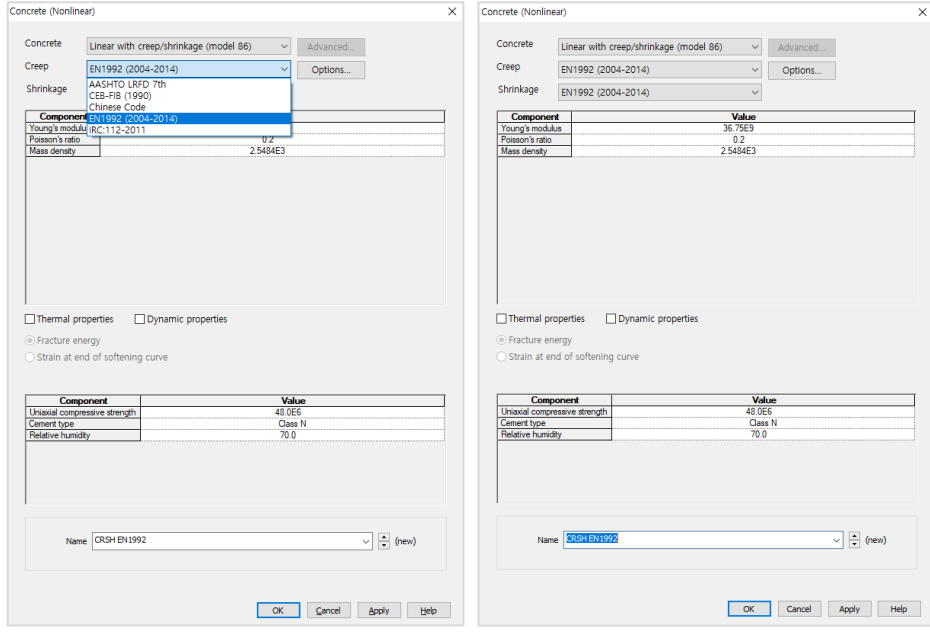


Fig 32 Creep & Shrinkage Material for Base Slab

Update Base Model

By selecting the **LNG Tank > Add CRSH Loading…** menu item, the loadings for the Creep and Shrinkage Analysis will be created and assigned to the appropriate loadcases. Dead loads will reference the data defined at the previous step of 'Base Model', hence no further input for loading is required.

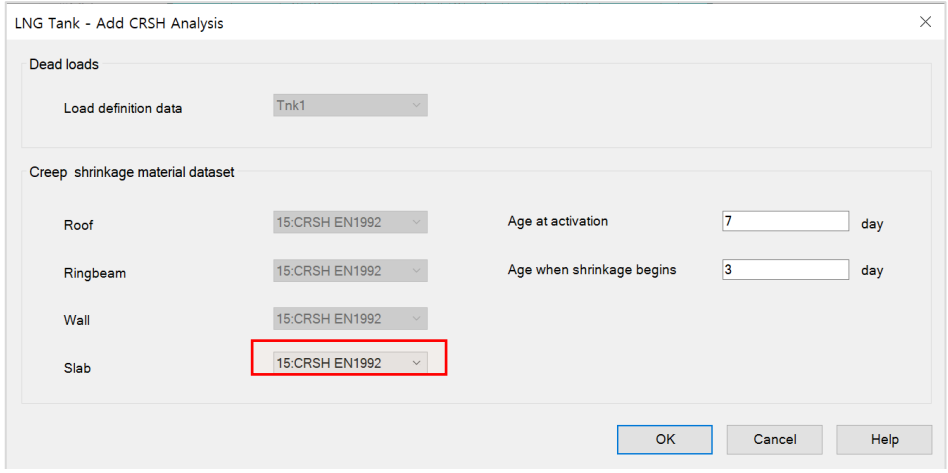


Fig 33 Dialog for Adding CRSH Analysis

- **Age at Activation:** The concrete age at the time of activation (i.e. cast removal)
- **Age when shrinkage begins:** The number of days/hours etc. between the concrete being cast and shrinkage being considered to have started. For Codes of Practice where autogenous shrinkage and drying shrinkage are handled separately, such as EN1992-1-1, this input refers to the age at which drying shrinkage begins; autogenous shrinkage is always taken as occurring from when the component is cast. For Codes of Practice where there is no such distinction, this value is used to define the commencement age for all shrinkage.

Make sure the Creep and Shrinkage Material dataset for base slab is selected.

Stage Definition and Loadings

The stages defined in this analysis are the same as for Staged Construction Analysis.

Additionally, the **'Age'** property is assigned to Base Slab, and the **time gap** between each stage and the **Total Response Time** are defined for each stage.

User Updates

Loadings

As discussed in the section titled ***Staged Construction Analysis***, some loadings may need to be updated.

Construction Schedule

The duration (length of time) of each stage is set to 10 days by default for all stages. This should be updated to follow the actual construction schedule.

Each stage uses a Nonlinear Control, and both the time and Total Response Time should be updated together.

For example, if the time gap between Stage 4 and Stage 5 needs to be changed to 15 days, Nonlinear Control for Stage 4 should be updated.

Default settings for Stage 5 are shown below.

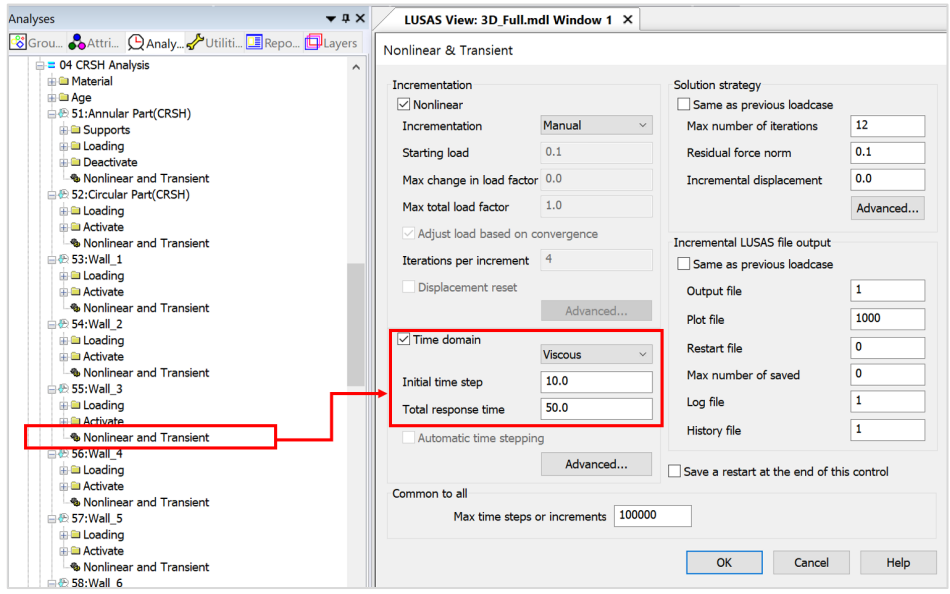


Fig 34 Nonlinear Control for Stage of Wall 2 (Default)

- **Initial Time Step** : Analysis is performed at every 10 days.
- **Total Response Time** : This stage lasts up to 50 days from the start of 1st Stage.

The number of days that the current stage lasts for is the Total Response Time of current stage minus Total Response Time of the previous stage.

By modifying **Total Response Time** to **55**, this stage lasts for 5 more days. (e.g. a total of 15 days)

Note that the Total Response Time for the subsequent stages should be also updated. Otherwise the 6th Stage will last only 5 days.

Modifying **Initial Time Step** is optional and depends on the accuracy required. With a smaller time step, the creep and shrinkage material properties are updated frequently (e.g. using smaller time gaps) hence the accuracy would increase, however the solution time will increase accordingly.

Tip

The unit of time is set on the Model Properties dialog by selecting the **File > Model Properties** menu item.

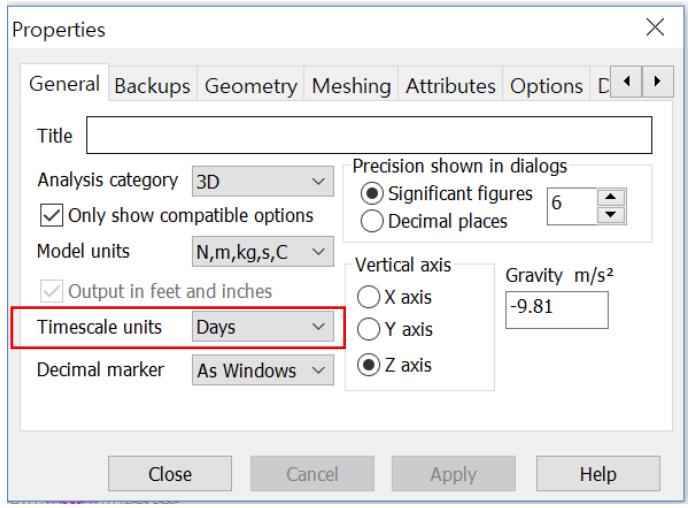


Fig 35 Model Properties

Age

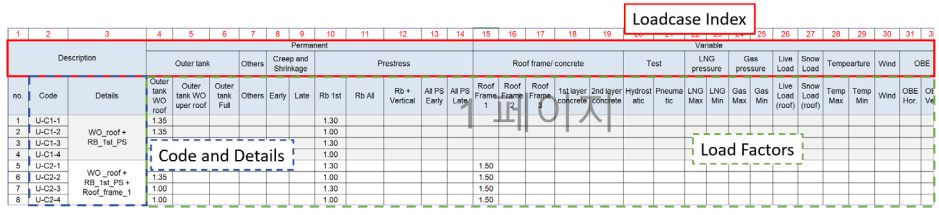
The Wizard built model assumes that each member is activated at the same concrete Age. The Age property represents the concrete age at the time of activation. (i.e. the time gap between pouring the concrete and removal of formwork.) If a different age is required for some members, another Age attribute should be defined and assigned manually for those members.

Design Load Combinations

Template for Design Load Combinations

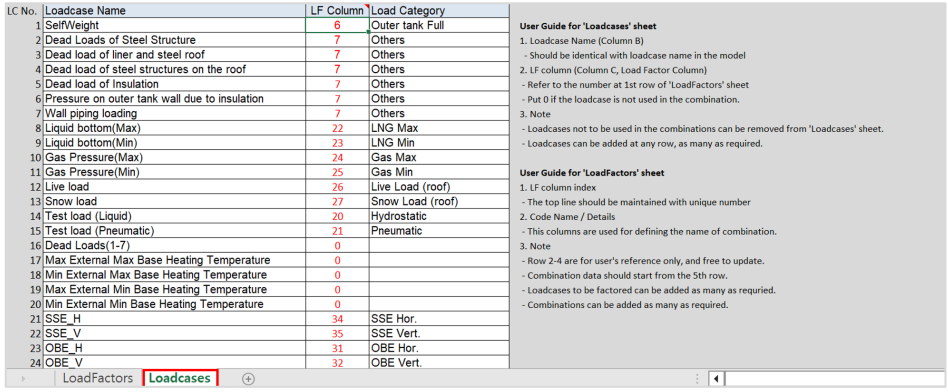
The template for Design Load Combination is saved in the current working folder where the Base Model was built, with the name **[Model name]_ComboTemplate.xlsx**, as shown below. The template can also be downloaded from the Design Load Combination dialog.

Load Factors Worksheet



| | | |

Loadcases Worksheet



LC No.	Loadcase Name	LF Column	Load Category
1	SelfWeight	6	Outer tank Full
2	Dead Loads of Steel Structure	7	Others
3	Dead load of liner and steel roof	7	Others
4	Dead load of steel structures on the roof	7	Others
5	Dead load of Insulation	7	Others
6	Pressure on outer tank wall due to insulation	7	Others
7	Wall piping loading	7	Others
8	Liquid bottom(Max)	22	LNG Max
9	Liquid bottom(Min)	23	LNG Min
10	Gas Pressure(Max)	24	Gas Max
11	Gas Pressure(Min)	25	Gas Min
12	Live load	26	Live Load (roof)
13	Snow load	27	Snow Load (roof)
14	Test load (Liquid)	20	Hydrostatic
15	Test load (Pneumatic)	21	Pneumatic
16	Dead Loads(1-7)	0	
17	Max External Max Base Heating Temperature	0	
18	Min External Max Base Heating Temperature	0	
19	Max External Min Base Heating Temperature	0	
20	Min External Min Base Heating Temperature	0	
21	SSE_H	34	SSE Hor.
22	SSE_V	35	SSE Vert.
23	OBE_H	31	OBE Hor.
24	OBE_V	32	OBE Vert.

Fig 37 Template for Design Load Combinations, Loadcases

- **Loadcase Name** The loadcase names defined in Modeller. The loadcase number may change during the process of updating the model, so the loadcase name is used in the definition of the Load Combination. Note that the loadcase names used must be the same as the loadcase names defined in Modeller.
- **(LF) Column** This column is used to match a Modeller loadcase to a corresponding loadcase on the LoadFactors worksheet by entering **Loadcase Index** on the LoadFactors worksheet. For example, the Self Weight loadcase in Modeller is used in the 'Outer Tank Full' combination defined in LoadFactors sheet. By entering 0 (zero), the loadcase is ignored and will not be used in the combination.
- **Load Category** All other data are for users' reference only, and not used for processing.

Update Base Model

The template is imported into Modeller by selecting the **LNG Tank > Design checks> Design Load Combination…**

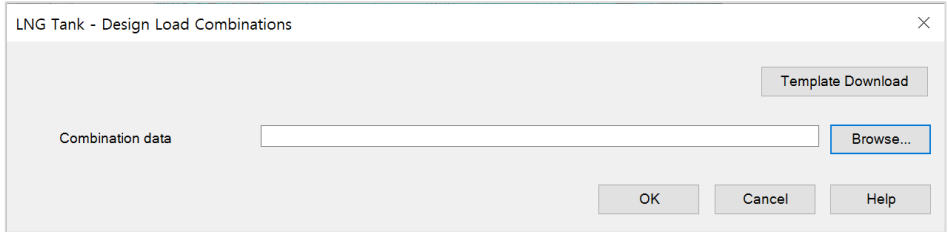


Fig 38 Dialog for Design Load Combination

- **Template Download** Downloads the template to be used for creating load combinations to the current working folder.
- **Combination data** Select the load combination template.

On re-loading this template, the **Code** and **Details** columns in the LoadFactors worksheet will be compared with combination names in Modeller and the load factors will be updated. If any new combinations are present, they will be added. However, any existing combinations will not be deleted.

The design load combinations are created as shown below.

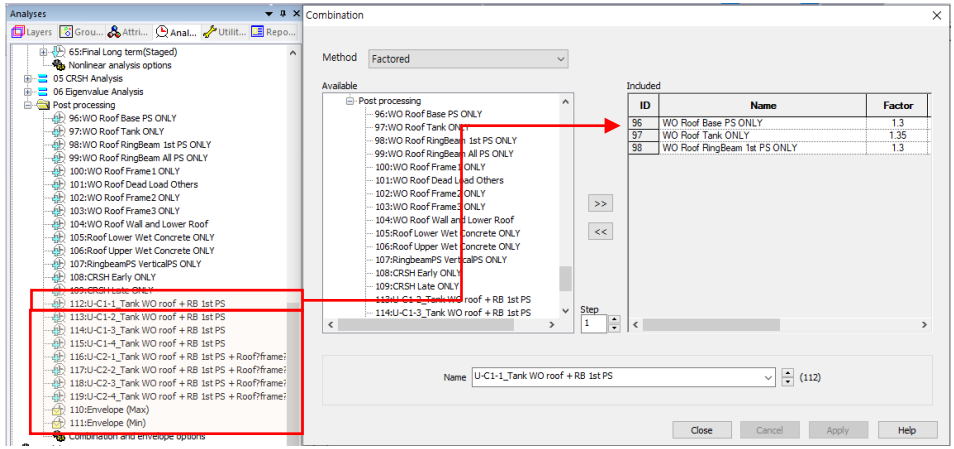


Fig 39 Load Combinations Created in Modeller

Design Check

COP Parameters

Design code parameters can be defined by the selecting the menu item **LNG Tank > Design Checks > Enable…**

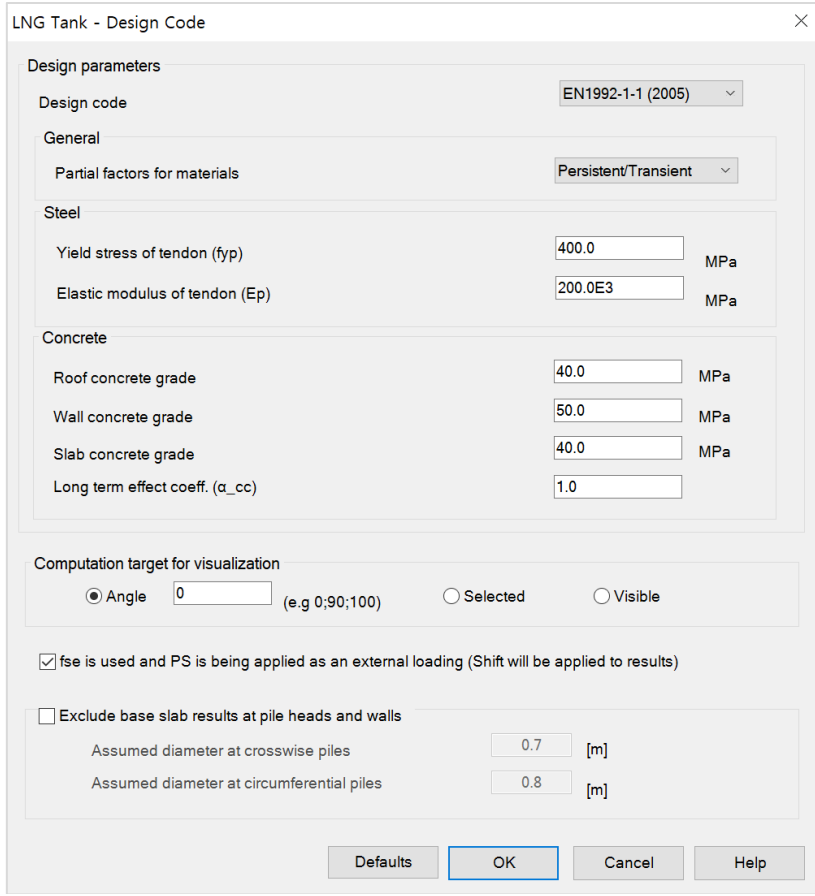
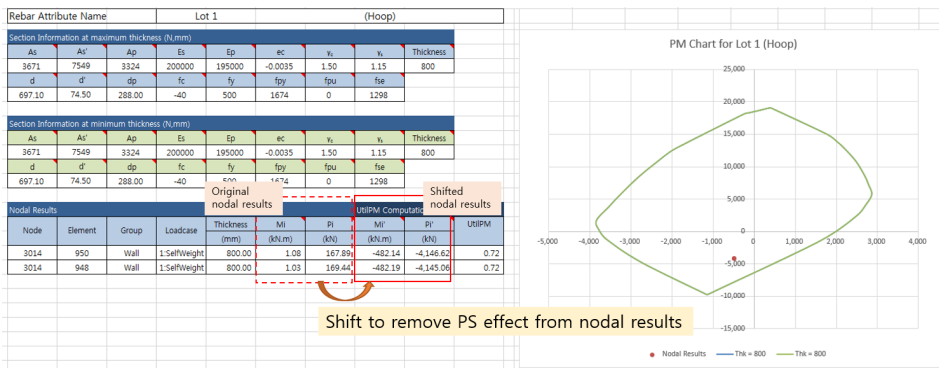


Fig 40 Dialog for Design Parameters

- **Partial Factors for Materials :** The partial factors for materials are defined based on Table 2.1N on EN1992. For Persistent & Transient, 1.5, 1.15, 1.15 are given to γ_c for concrete, γ_s for reinforcing steel, γ_s for prestressing steel respectively. For Accidental, 1.2, 1.0, 1.0 are given.
- **Yield Stress of Tendon (fyp)** : Yield strength of prestressing steel in MPa.
- **Elastic Modulus of Tendon (Ep) :** Elastic Modulus of prestressing steel in MPa
- **Tensile strength of tendon (fpu)** : Tensile strength of prestressing steel in MPa. (ACI 318-14 only)
- **Stability coefficient (φ)** : Stability coefficient (GB50010-2010 only)
- **Roof concrete grade :** Concrete strength in MPa.
- **Wall concrete grade :** Concrete strength in MPa.
- **Base concrete grade :** Concrete strength in MPa.
- **Max Concrete Compressive Strain :** Ultimate strain at concrete failure (ACI 318-14 only)
- **Long term effect coeff. (α_{cc})** : Long term effect coefficient (EN1991 only)
- **Computation Target for visualization:** The design check computations will be performed for the targets of
  - **Default**: The node at Y=0 and X>=0.
  - **Selected**: The nodes that user selected before opening this dialog
  - **Visible**: All visible nodes in Modeller.

- **Elastic Modulus of rebar (Es) :** Elastic Modulus of reinforcing steel in MPa.
- fse is used and PS is being applied as an external loading:
  - When '**fse**' is specified in the **reinforcement template** and the PS being applied as external loading, the code-checking that creates the PM chart will double count the PS effect.
  - Ticking this option shifts the results to avoid double counting the PS effect.
  - When a loadcase (or load combination) does not have PS as external loading, this option should be disabled (unticked).
  - It makes no difference if '**fse**' is not specified in the **reinforcement template.**



Shifted force, Pi' = Pi – fse * Apd

Shifted moment, Mi' = Mi – fse * Apd * ecc_ps

Design Checks for Tank

Once design code parameters have been defined and the OK button is pressed a **Tank Design** Entity will then be available for selection in relevant result processing dialogs. The associated available components are shown below.

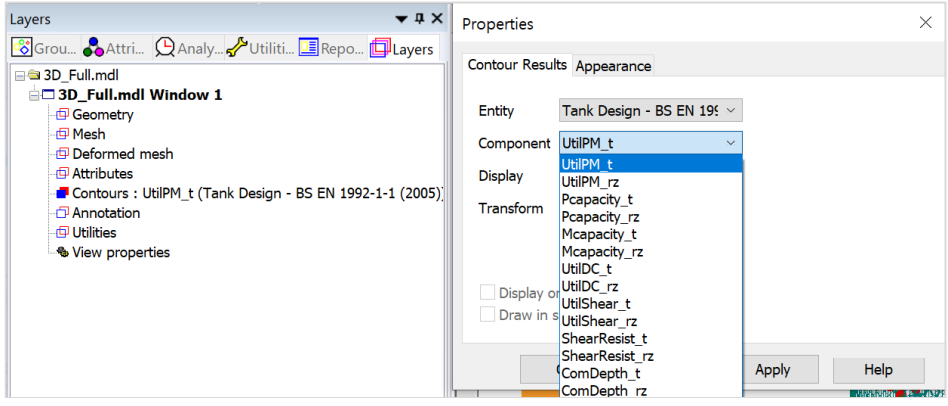


Fig 41 Components for Design Checks

Whenever a results component is chosen, or a loadcase is set active, the design check for a selected component will take place. Design checks are carried out on a node-by-node basis using analysis results and relevant design code formulae.

UtilPM (PM Utilization)

UtilPM can be checked with reference to a PM chart, as illustrated below. A value of less than 1 means it satisfies the design code.

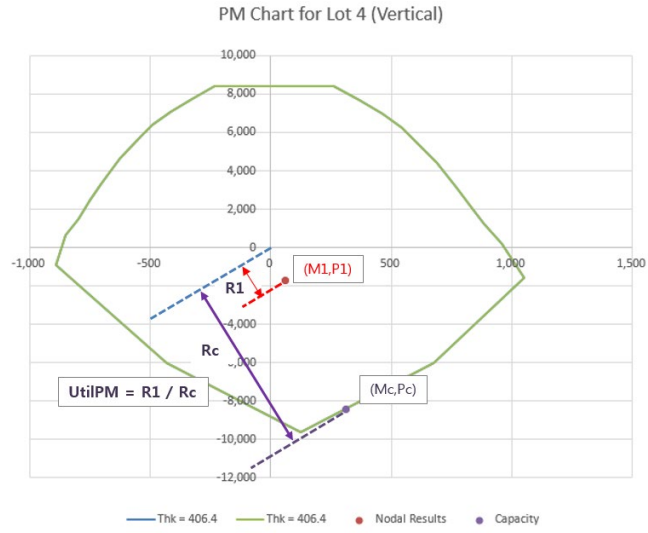


Fig 42 Definition of UtilPM

- **UtilPM_t** is the force and moment utilization in the hoop direction, and **UtilPM_rz** is the utilization in the radial direction for the Roof and Base Slab and for the vertical direction for the Wall.
- **Pcapacity_t** and **Pcapacity_rz** is for the computed Pc at the given P1/M1 slope, **Mcapacity_t, Mcapacity_rz** is for the Mc at the given P1/M1 slope.

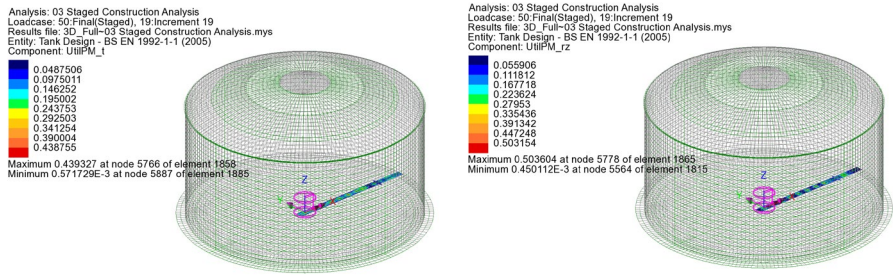


Fig 43 Contour for UtilPM_t, UtilPM_rz

UtilDC (Decompression Utilization)

UitilDC is available only for the section contains prestress tendons.

Please refer to LNG_Tank_System_Manual (Concrete Tank) for more information.

ShearResist (Shear Resistances)

Shear capacity for the tank components is based on concrete shear resistance as per COP specifications.

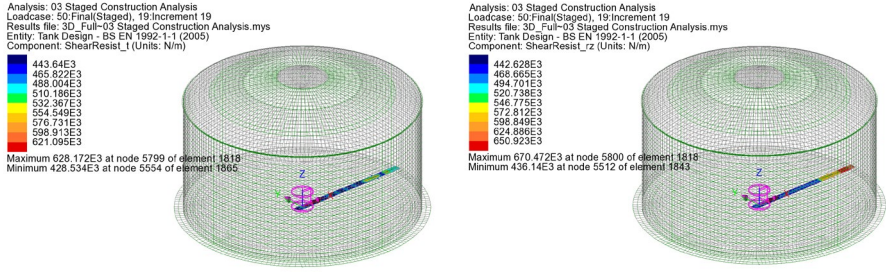


Fig 44 Contour for ShearResist_t, ShearResist_rv

UtilShear (Shear Utilization)

Shear utilisation factors are given as the absolute ratio between the shear forces (Sp, St, Sz, Sr) and the relevant shear resistances (ShearResist_t and ShearResist_rz). A value for UtilShear > 1 denotes failure in shear.

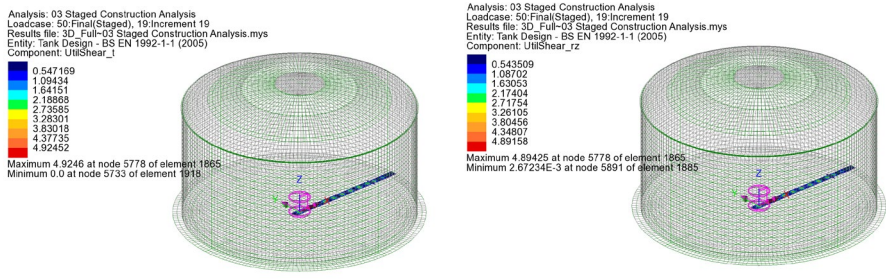


Fig 45 Contour for UtilShear_t, UtilShear_rv

Design Check Report

A spreadsheet report that includes a PM chart can be produced by selecting the menu item **LNG Tank > Design Checks> Design Check Report...**

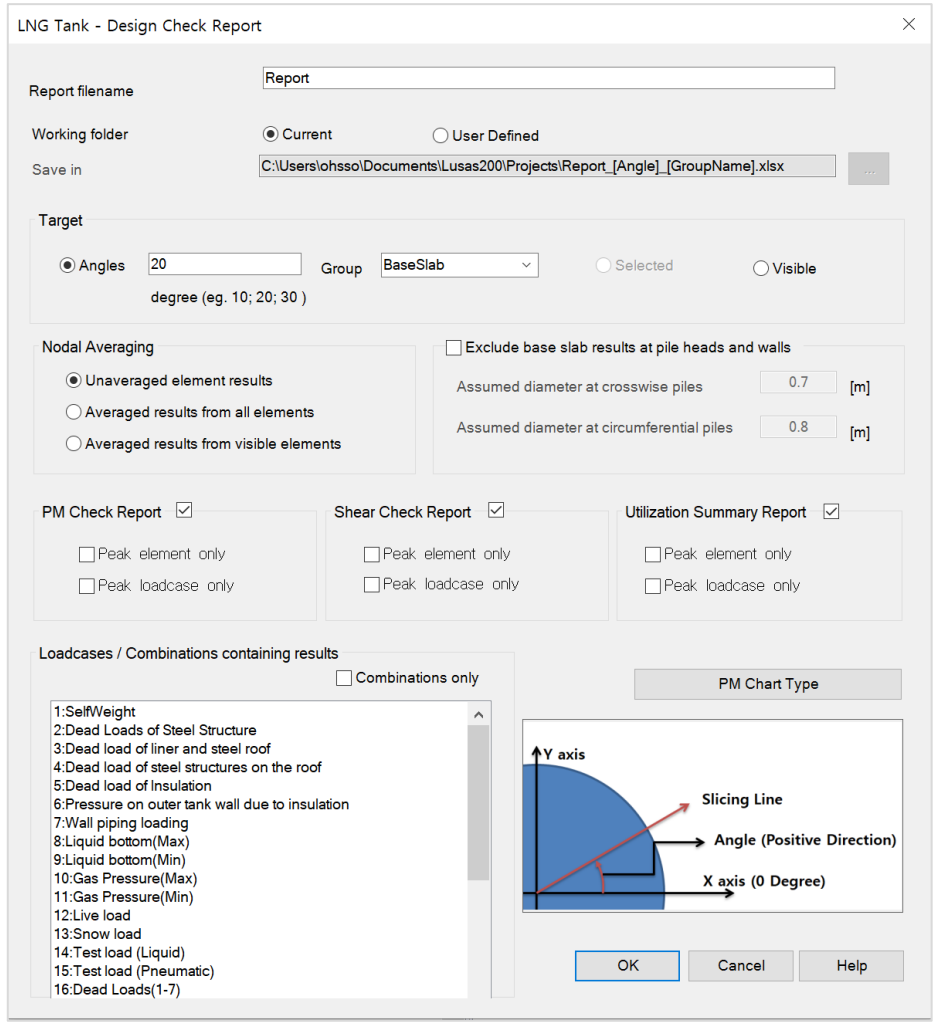


Fig 46 Dialog for Design Check Report

- **Target** The design check computations will be performed for the targets of
  - **Default**: The node at Y=0 and X>=0.
  - **Selected** : Any nodes that were selected before opening this dialog
  - **Visible**: All visible nodes in Modeller.

- **Nodal Averaging**

  - **Unaveraged element results:** Unaveraged element results will be output and used to compute Utilizations. 'Peak element only' option is available only with 'Unaveraged element results' option for Nodal averaging.

- **Averaged result from active elements**: Averaged element results for active elements will be output and used to compute Utilizations. 'Peak element only' option will be disabled.

- **Averaged results from visible elements**: Averaged element results only for visible elements will be output and used to compute Utilizations. 'Peak element only' option will be disabled.

- ❑ **Avoid pile/Slab, wall/slab intersection:** If this checkbox is ticked, the results of the base slab nodes located in the pile heads or in the wall and slab intersection area where there could be unrealistic high stress concentration will not be printed in the report.

  - **Crosswise piles diameter**: Diameter of the crosswise piles.

  - **Circumferential piles diameter**: Diameter of circumferential piles.

- ❑ **PM Check Report:** 'PM Check Report' has been updated to have 'Tendon(%)' column for 'Nodal results' to indicate how much the % of tendons are functioning at that stage/ in that loadcase. Moment shift and force shift % have been added to the PM check report as well. Depending on the term of prestress, either 'fse_short' or 'fse_long' can be selectively considered to compute PM capacity.

- ❑ **PM chart type**

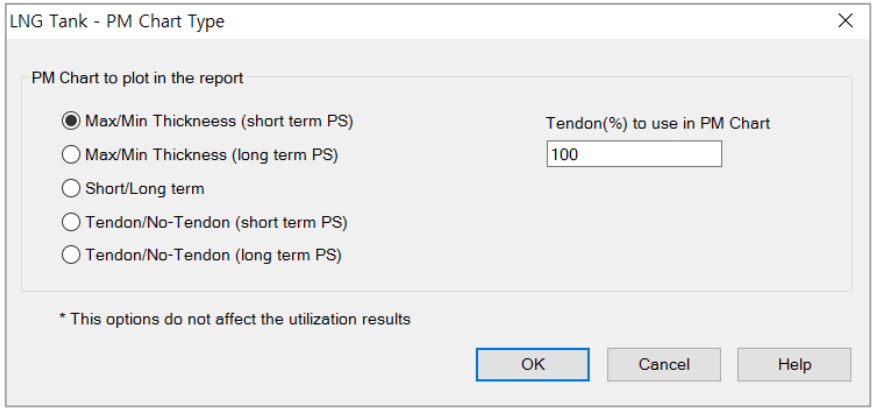


Fig 47 PM chart type dialog

PM Check Report: 'PM Check Report' has been updated to have 'Tendon(%)'

Following 5 types of PM chart are available. The following options do not affect the Utilization results.

- **Max/Min Thickness (short term PS):** PM Chart will be created for Max and Min thickness of each part (Base slab, Wall, Roof and Ringbeam) for short term Prestress
- **Max/Min Thickness (long term PS):** PM Chart will be created for Max and Min thickness of each part (Base slab, Wall, Roof and Ringbeam) for long term Prestress
- **Short/Long term:** PM Chart will be created for Max and Min thickness of each part (Base slab, Wall, Roof and Ringbeam) for long term Prestress
- **Tendon/No-Tendon (short term PS):** PM Chart will be created including tendon and excluding tendon of each part (Base slab, Wall, Roof and Ringbeam) for short term Prestress
- **Tendon/No-Tendon (long term PS):** PM Chart will be created including tendon and excluding tendon of each part (Base slab, Wall, Roof and Ringbeam) for long term Prestress
- **Tendon (%) to use in PM Chart:** Percentage of tendon (%) to use PM Chart.

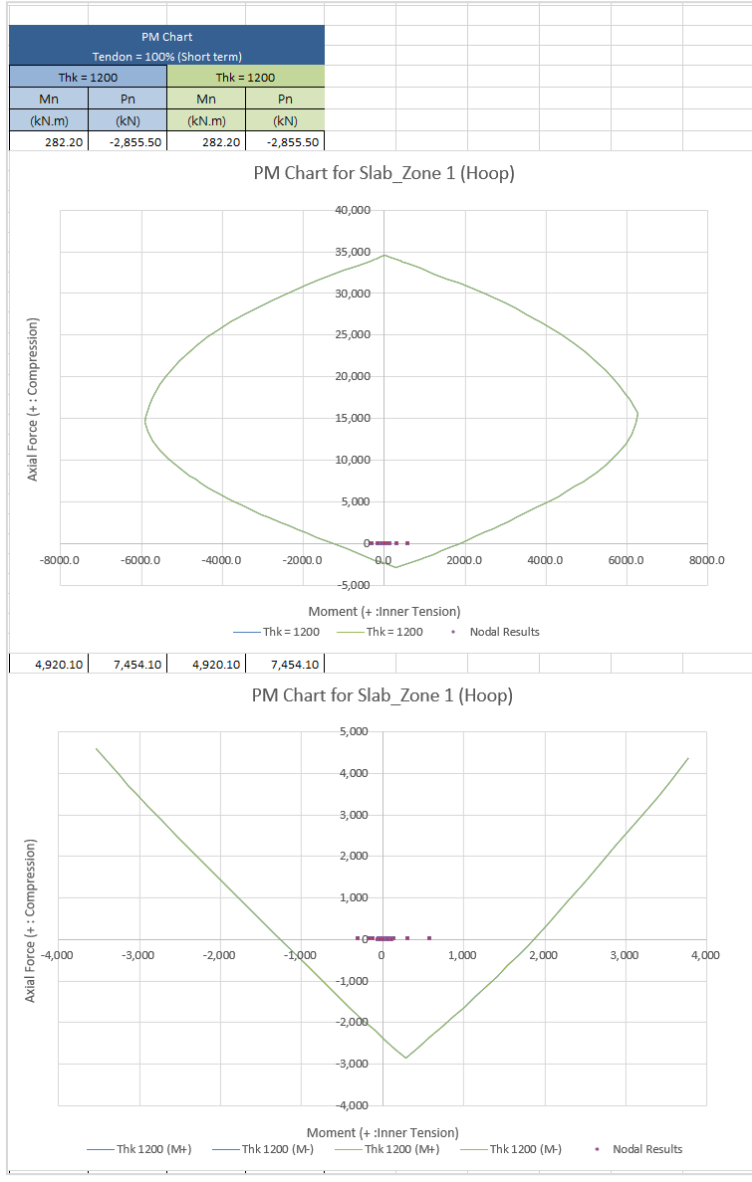


Fig 48 PM chart

- **Shear Check Report:** It has been updated to include 'Tendon area' to compute shear capacity based on ACI when generate shear check report.

If ‘Default’ is selected for Report Target, pre-defined target nodes will be used for each of the 15 types of different reinforcement arrangements in the current model, and the report will contain 30 worksheets for producing PM charts for two directions (hoop/vertical or hoop/radial) for all 15 rebar arrangements.

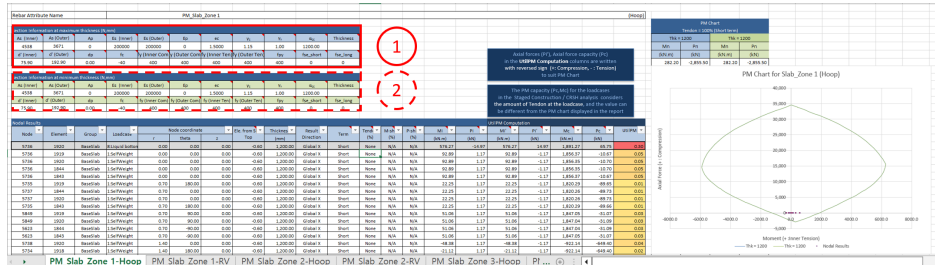


Fig 49 Design Check Report with PM Chart for Lot 1

- **Section Information at maximum thickness (1)** The maximum thickness from the selected nodes is printed, for which the PM chart is displayed.
- **Section Information at minimum thickness (2)** The minimum thickness from the selected nodes is printed, for which the PM chart is displayed.
- **Node** Node number for UtilPM computation
- **Element** Element numbers sharing the node
- **Group** Group name where the node is included in.
- **Thickness** Section thickness at the node location
- **Mi** Bending moment at the node
- **Pi** Axial force at the node
- **Mi’, Pi’** The shifted Mi, Pi when ‘fse is used and PS is being applied as an external loading’ option is ticked from the Design Code dialog.
- **UtilPM** PM Utilization
- **PM Chart** Two PM Charts are presented; one for the maximum thickness section, the other for minimum thickness section of the selected nodes. The value for Pcapacity in Modeller has different sign from the value of Pcapacity stated in the PM report

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