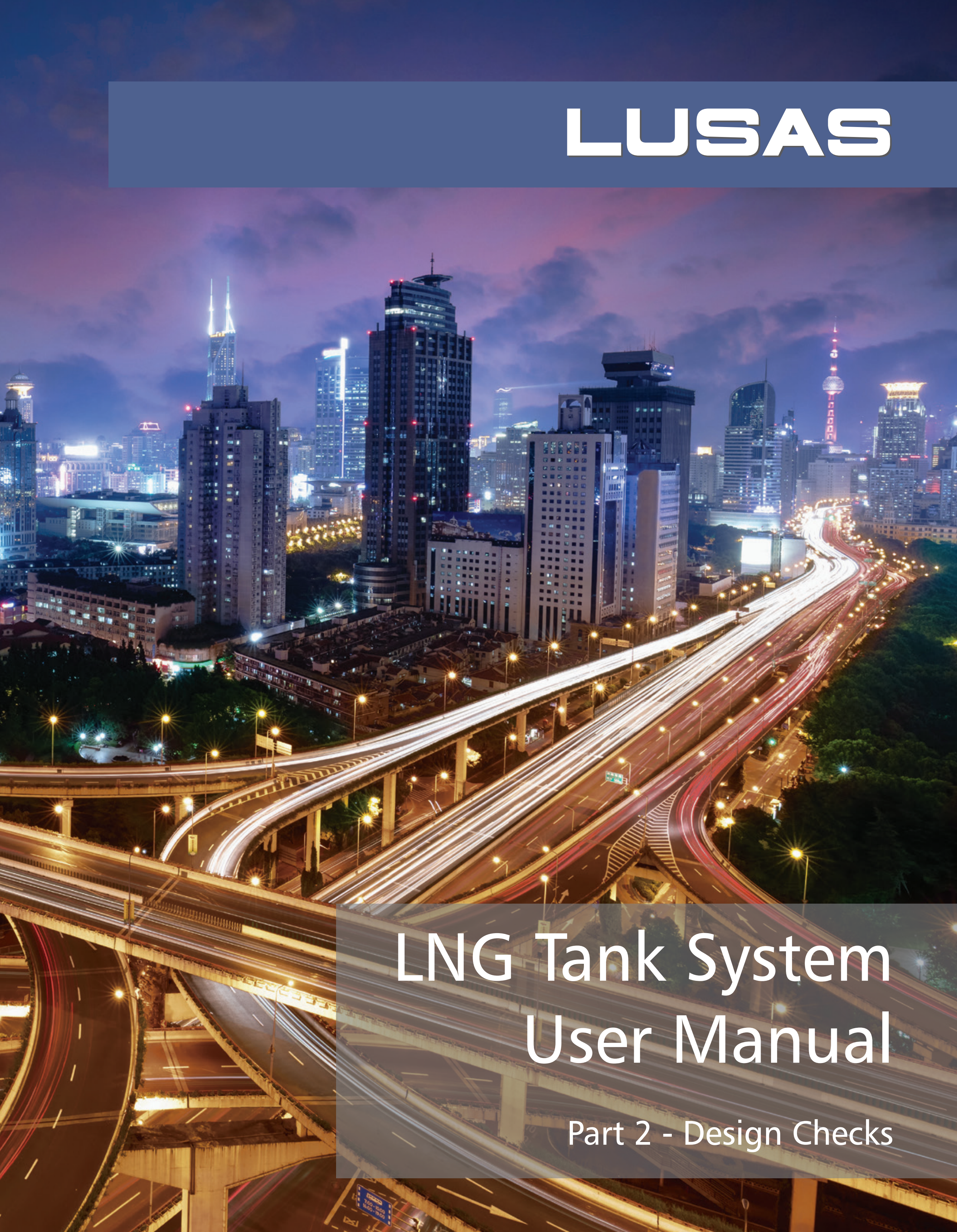


LUSAS

LNG Tank System User Manual

Part 2 - Design Checks

LNG Tank System

User Manual (Online) : Part 2 - Design Checks

LUSAS Version 19.1 : Issue 1

LUSAS

Forge House, 66 High Street, Kingston upon Thames,

Surrey, KT1 1HN, United Kingdom

Tel: +44 (0)20 8541 1999

Fax +44 (0)20 8549 9399

Email: info@lusas.com

http://www.lusas.com

Distributors Worldwide

Table of Contents

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: 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: 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**…

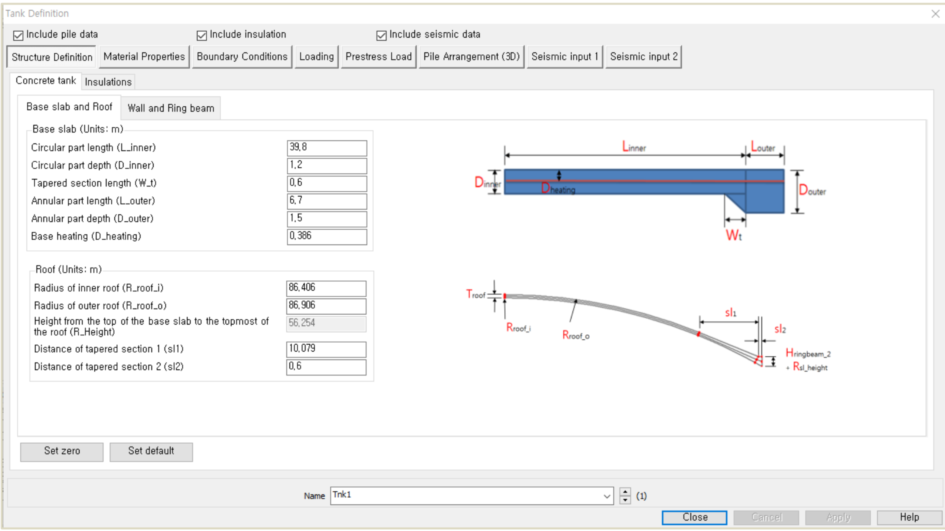


Fig 1 User Input for Tank Dimension and Loadings

Complete all the inputs, and click **OK** to save the data with the name ‘ **Tnk1**’.

Reinforcement and Prestress tendon

A Reinforcement and Prestress tendon arrangement is required to build the corresponding model.

The template is available from the folder where LUSAS is installed. For example:

C:\Program Files (x86)\LUSAS190\Programs\scripts\Tank**Tank_Template_Reinforcement.xlsx**.

Copy this file to the current working folder or download the file from the ‘Build model for design check’ dialog.

Wall reinforcement

The worksheet named ‘**Wall**’ is used to define the inner and outer reinforcement arrangement.

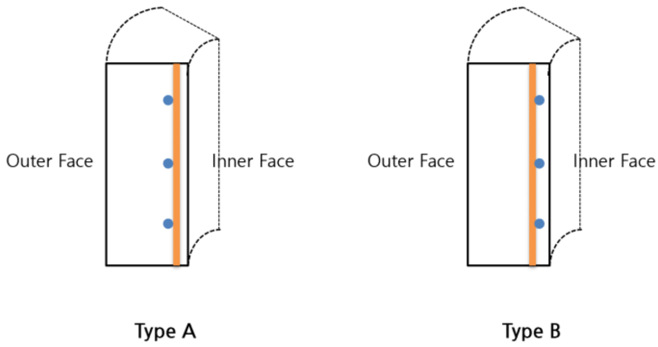
A maximum of 15 sets of wall reinforcement (excluding any in Ringbeam) can be defined.

If not all of the permitted sets are required, the remaining rows can be left empty.

Inner Face										
Location	Height (m)	Separate Stage (Y/N)	Cover Thickness (mm)	Rebar Arrangement Type	Hoop Direction			Vertical Direction		
					Diameter (mm)	Number of Rebar	Spacing (mm)	Diameter (mm)	Number of Rebar	spacing (mm)
Lot 0	0.30	N	60	A	28.6	1	175	25.4	1	175
Lot 1	4.2	Y	60	A	25.4	1	175	25.4	1	175
Lot 2	4.2	Y	60	A	31.8	1	175	25.4	1	175
Lot 3	4.2	Y	60	A	25.4	1	175	25.4	1	175
Lot 4	4.2	Y	60	A	25.4	1	175	25.4	1	175
Lot 5	4.2	Y	60	A	25.4	1	175	25.4	1	175
Lot 6	4.2	Y	60	A	25.4	1	175	25.4	1	175
Lot 7	4.2	Y	60	A	25.4	1	175	25.4	1	175
Lot 8	4.2	Y	60	A	25.4	1	175	25.4	1	175
Lot 9	4.2	Y	60	A	25.4	1	175	25.4	1	175
Lot 10	4.58	Y	60	A	25.4	1	175	25.4	1	175
Lot 11		Y		A						
Lot 12		Y		A						
Lot 13		Y		A						
Lot 14		Y		A						
RingBeam		Y	60	A	25.4	1	175	25.4	1	175

Fig 2 Reinforcement Arrangement for Inner Face of Wall

- ❑ **Location** This will be used as the rebar arrangement dataset name in Modeller.
- ❑ **Height :** The height of each section where the same rebar arrangement is used. The sum of ‘Height’ should be equal to the **Height of Wall** as defined in the Tank Definition.
- ❑ **Separate Stage(Y/N)** if set to ‘N’, the construction stage for the current section will not be created and the section is activated at the previous stage.
- ❑ **Cover thickness** The distance from the rebar surface to the nearest concrete face (mm).
- ❑ **Rebar Arrangement Type**
  - **A -** Vertical reinforcement is in the inner face of the wall
  - **B** - Horizontal reinforcement is in the inner face of the wall



- ❑ **Diameter** Diameter of reinforcement (mm).
- ❑ **Number of Rebar** Number of bundled reinforcement bars. If bars are not bundled, the value should be **1**.
- ❑ **Spacing** Centre to centre distance between reinforcement. (mm)

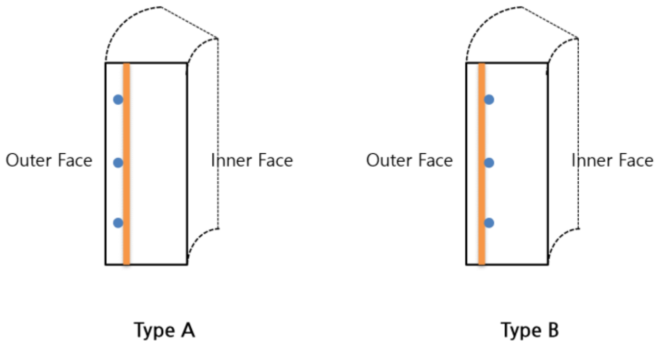
Outer Face

Location	Height (m)	Separate Stage (Y/N)	Cover Thickness (mm)	Rebar Arrangement Type	Hoop Direction			Vertical Direction		
					Diameter (mm)	Number of Rebar	Spacing (mm)	Diameter (mm)	Number of Rebar	spacing (mm)
Lot 0	Same with Inner face	Same with Inner face	60	A	34.8	1	175	31.8	2	175
Lot 1			60	A	25.4	1	175	31.8	1	175
Lot 2			60	A	25.4	1	175	25.4	1	175
Lot 3			60	A	25.4	1	175	25.4	1	175
Lot 4			60	A	25.4	1	175	25.4	1	175
Lot 5			60	A	25.4	1	175	25.4	1	175
Lot 6			60	A	25.4	1	175	25.4	1	175
Lot 7			60	A	25.4	1	175	25.4	1	175
Lot 8			60	A	25.4	1	175	25.4	1	175
Lot 9			60	A	25.4	1	175	25.4	1	175
Lot 10			60	A	25.4	1	175	25.4	1	175
Lot 11				A						
Lot 12				A						
Lot 13				A						
Lot 14				A						
RingBeam			60	A	25.4	1	175	25.4	1	175

Fig 3 Reinforcement Arrangement for Outer Face of Wall

❑ **Rebar Arrangement Type**

- **A** – Horizontal reinforcement is in the outer face of the wall
- **B** – Vertical reinforcement is in the outer face of the wall.



Slab reinforcement

The worksheet named '**Slab**' is used to define the inner and outer reinforcement arrangement.

- A maximum of two sets of slab reinforcement can be defined: one for the Annular part, the other for the Central Part. Only reinforcement in the radial and circumferential directions is allowed (i.e. no crosswise reinforcement is permitted)
- The Annular Part relates to the outer part of the Slab as illustrated below. (=Louter)

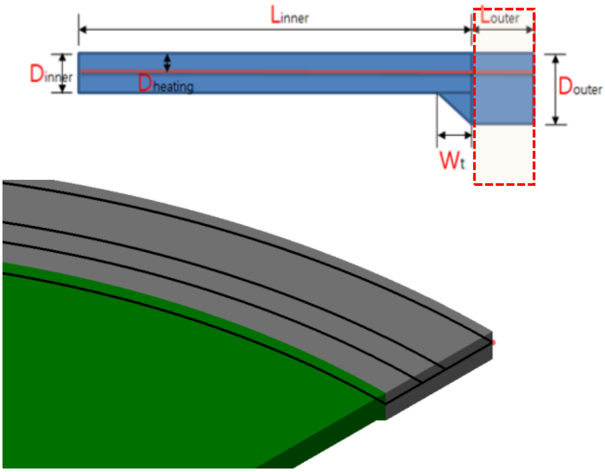


Fig 4 Definition of the Base Slab Annular Part

Inner Face (Top Face)

Location	Cover Thickness (mm)	Rebar Arrangement Type	Circumferential Direction			Radial Direction		
			Diameter (mm)	Number of Rebar	Spacing (mm)	Diameter (mm)	Number of Rebar	spacing (mm)
Annular Part	60	A	28.6	2	175	28.6	2	175
Central Part	60	A	31.8	1	175	31.8	1	175

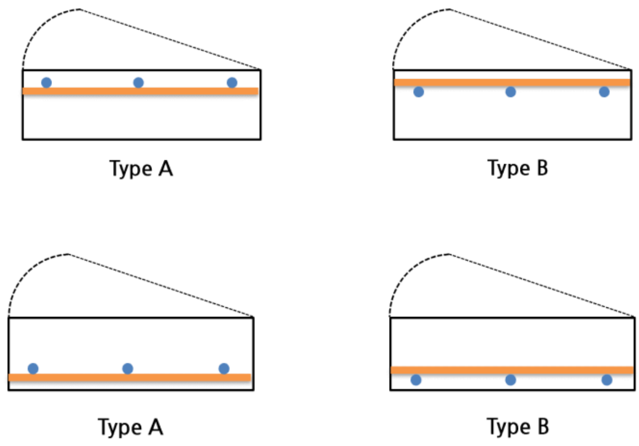
Outer Face (Bottom Face)

Location	Cover Thickness (mm)	Rebar Arrangement Type	Circumferential Direction			Radial Direction		
			Diameter (mm)	Number of Rebar	Spacing (mm)	Diameter (mm)	Number of Rebar	spacing (mm)
Annular part	150	A	31.8	1	175	31.8	2	175
Central part	150	A	28.6	1	175	28.6	1	175

Fig 5 Reinforcement Arrangement for Slab

Rebar Arrangement Type

- **A** – Hoop reinforcement is in the upper face of the slab
- **B** – Radial reinforcement is in the upper face of the slab



Roof reinforcement

The worksheet named ‘**Roof**’ is used to define the inner and outer reinforcement arrangement.

- A maximum of two sets of slab reinforcement can be defined..
- The Varying Part relates to the outer part of the Roof as illustrated below. (=sl_1 + sl_2)

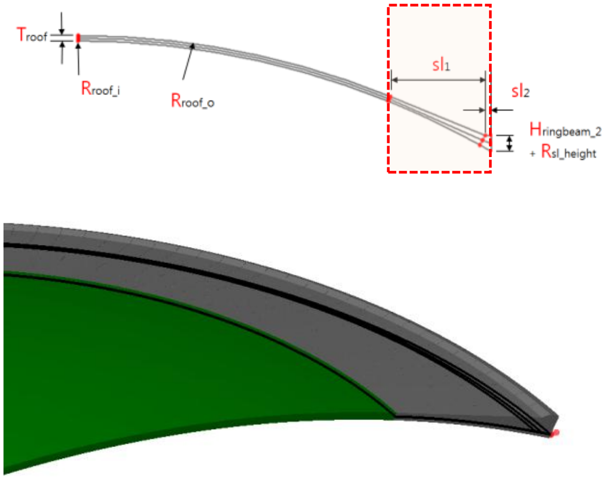


Fig 6 Definition of Roof Varying Part

Inner Face

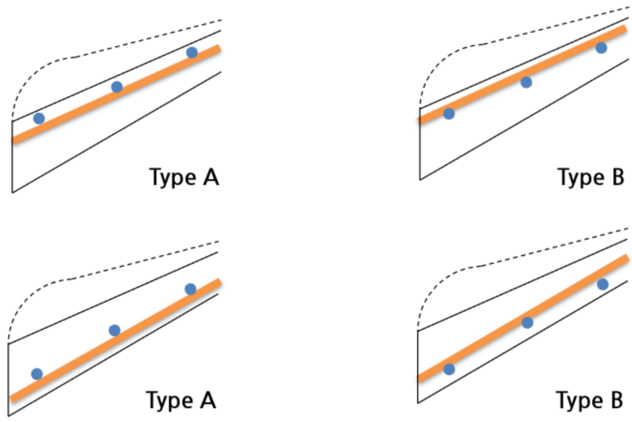
Location	Cover Thickness (mm)	Rebar Arrangement Type	Circumferential Direction			Radial Direction		
			Diameter (mm)	Number of Rebar	Spacing (mm)	Diameter (mm)	Number of Rebar	spacing (mm)
Varying Part	50	A	28.6	1	175	31.8	1	175
Central Part	50	A	25.4	1	175	25.4	1	175

Outer Face

Location	Cover Thickness (mm)	Rebar Arrangement Type	Circumferential Direction			Radial Direction		
			Diameter (mm)	Number of Rebar	Spacing (mm)	Diameter (mm)	Number of Rebar	spacing (mm)
Varying Part	60	A	28.6	1	175	31.8	1	175
Central Part	60	A	28.6	1	175	28.6	1	175

Fig 7 Reinforcement Arrangement for Roof

- ❑ **Rebar Arrangement Type**
  - **A** – Hoop reinforcement is in the upper face of the roof
  - **B** – Radial reinforcement is in the upper face of the roof



Prestress Tendon

The worksheet named '**PreStress**' is used to define the Tendon arrangement for the wall and slab.

- A maximum of 15 sets of tendon arrangements (excluding any in Ringbeam) can be defined.
- If not all of the permitted sets are required, the remaining rows can be left empty.
- The cover thickness to be stated is the distance from the <u>outer surface to the tendon surface</u>.
- Fse, the effective stress in prestressing reinforcement for the base slab, applies to the annular part only.

Wall									
Location	Hoop Direction				Vertical Direction				Hoop
	Cover Thickness (mm)	Diameter (mm)	Spacing (mm)	fse (MPa)	Cover Thickness (mm)	Diameter (mm)	spacing (mm)	fse (MPa)	No tendons
Lot 0	600	1.00E-03	1000	1	600	69.05	1262	1005	
Lot 1	545	69.05	600	1005	545	69.05	1262	1005	7
Lot 2	421	69.05	525	1005	421	69.05	1262	1005	8
Lot 3	375	69.05	525	1005	375	69.05	1262	1005	8
Lot 4	375	69.05	600	1005	375	69.05	1262	1005	7
Lot 5	375	69.05	840	1005	375	69.05	1262	1005	5
Lot 6	375	69.05	840	1005	375	69.05	1262	1005	5
Lot 7	375	69.05	1050	1005	375	69.05	1262	1005	4
Lot 8	375	69.05	1050	1005	375	69.05	1262	1005	4
Lot 9	375	69.05	1050	1005	375	69.05	1262	1005	4
Lot 10	375	69.05	1145	1005	375	69.05	1262	1005	4
Lot 11									
Lot 12									
Lot 13									
Lot 14									
RingBeam	700	69.05	680	1005	700	69.05	1262	1005	5

Base Slab Annular Part				
Location	Hoop Direction			
	Cover Thickness from Bottom (mm)	Diameter (mm)	Spacing (mm)	fse (MPa)
Annular Part				

Fig 8 Prestress Tendon arrangement for Wall and Base Slab

Build Base Model

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

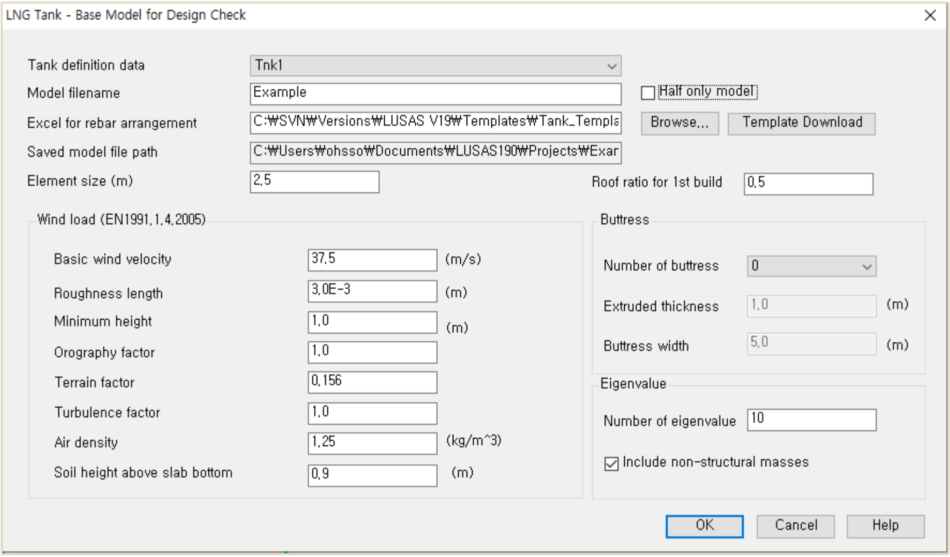


Fig 9 LNG Tank for Base Model for Design Check

- ❑ The spreadsheet for the reinforcement arrangement should be selected.
- ❑ Number of buttresses is set to 0.
- ❑ Roof ratio for 1st build is set to 0.5. The roof thickness for 1st build will vary according to this value in the staged construction analysis model. The 1st Prestress amount for Ringbeam will also vary at the 'Ringbeam 1st PS stage' in staged construction analysis model.

The model built is shown below.

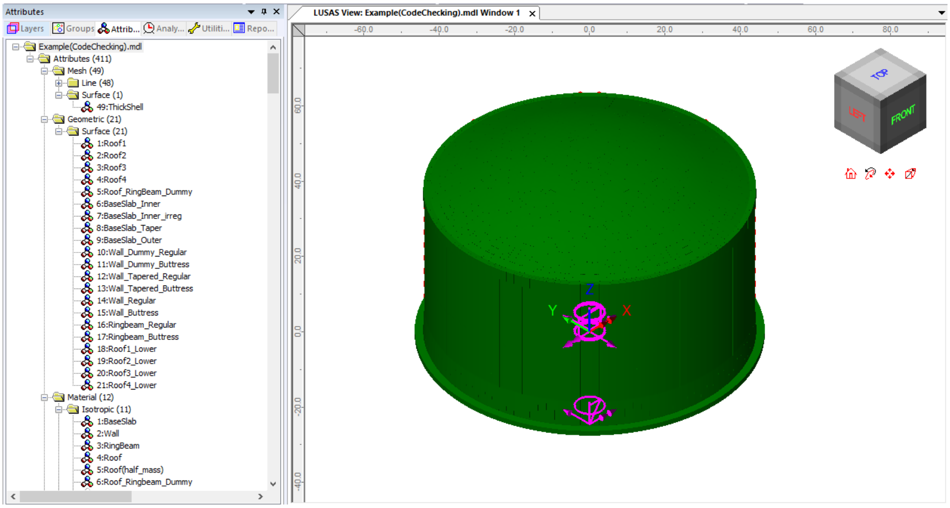


Fig 10 Base Model for Design Check

Reinforcement arrangement

The Reinforcement arrangement is saved as a Wood-Armer attribute in Modeller.

- **Hoop Roof** reinforcement is defined as **Bar in X**, and ***radial reinforcement*** as ***Bar in Y***.
- **Horizontal Wall** reinforcement is defined as **Bar in X**, and ***vertical reinforcemen****t* as ***Bar in Y***.
- **Radial Slab** reinforcement is defined as **Bar in X**, and ***hoop reinforcement*** as ***Bar in Y*.**
- **Inner face** reinforcement is defined at Top Face, and *outer face* at *Bottom Face*. The local z axis of the shell elements is orientated toward the inner direction in the model, hence the bottom face is equivalent to outer surface of wall.

For **bundled reinforcement**, the spacing is defined by **Spacing / Number of rebar.** Note that this may affect the crack width, therefore care should be taken. Alternatively, an equivalent diameter of bundled reinforcement with the Number of rebar set to 1 could be appropriate.

The table that follows shows the reinforcement for Wall Lot 1. Note that all references to ‘Lot1’, ‘Lot 2’ etc. relate to ‘segments or regions of a wall with a specific reinforcement arrangement’.

- **Rebar Arrangement Type** of **A** for the Wall means that the vertical reinforcement is on the inner face, therefore the Bar in Y is located in Layer 1.

Inner Face

Location	Height (m)	Separate Stage (Y/N)	Cover Thickness (mm)	Rebar Arrangement Type	Hoop Direction			Vertical Direction		
					Diameter (mm)	Number of Rebar	Spacing (mm)	Diameter (mm)	Number of Rebar	spacing (mm)
Lot 0	0.30	N	60	A	28.6	1	175	25.4	1	175
Lot 1	4.2	Y	60	A	25.4	1	175	25.4	1	175

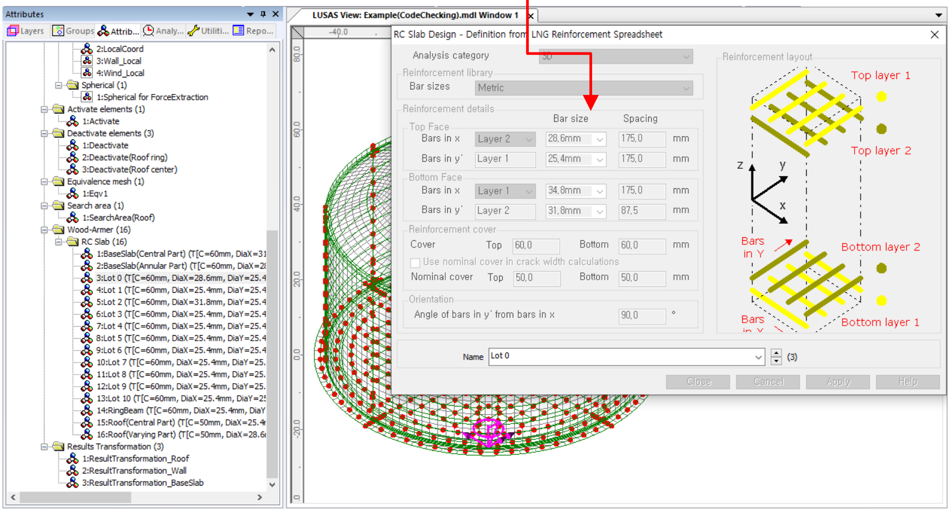


Fig 11 Reinforcement Arrangement for Wall Lot 1

Tendon Arrangement

The Tendon arrangement is also saved as a Wood-Armer/RC Slab attribute. However this cannot be seen in Modeller. *Note that because of this, if the reinforcement information is updated from Modeller directly, tendon information will be lost.*

Analyses

Six analyses having a total of 95 loadcases are set-up in the model.

- ❑ **Base Analysis** A static analysis. A total of 30 loadcases are provided including 5 for user-defined loading.
- ❑ **Thermal Analysis** 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** 29 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

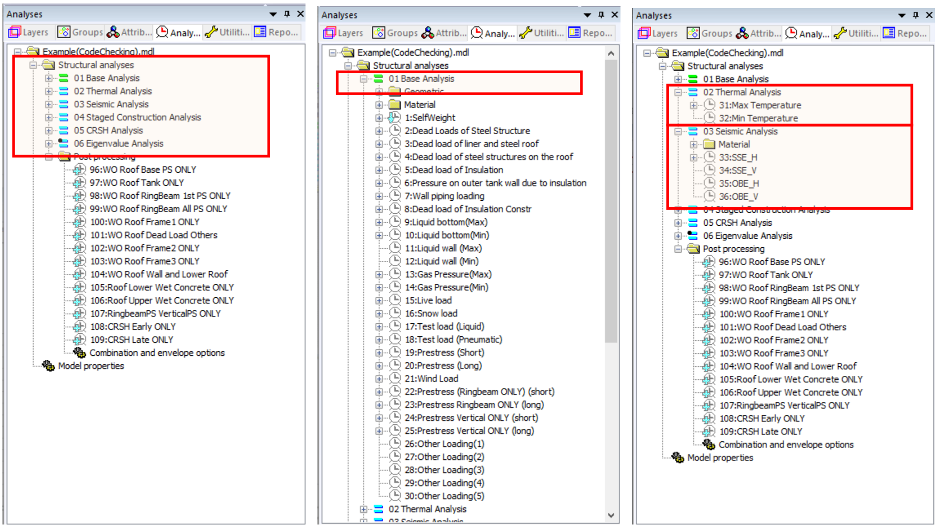


Fig 12 Analyses Available in Base Model

Roof element layers

The roof is built in two layers. Hence, two sets of overlapping surfaces have been created to consider the staged construction. As only the final layer is required for the other analyses, one set of surfaces is deactivated for the 1st loadcase of Base Analysis, Thermal Analysis, Seismic Analysis, and Eigenvalue analysis.

Loadings

Self weight is assigned in all analyses.

- In the Base Analysis, 30 loadcases are provided, including 5 for user-defined loading.
- In the other analyses (with the exception of the self weight) loading is not yet assigned.

Base Load Combinations

As seen in [Fig 12], 10 load combinations are pre-defined. These combinations are used to obtain the isolated effect of adding loading from a Staged Construction Analysis, so that they can be used as a single loadcase in the design load combination.

A staged construction analysis is nonlinear by definition and hence the principle of superposition is not generally applicable. However, this simplified approach provides a systematic and efficient way of verifying hundreds of load combinations in the context of linear design according to a code of practice.

For example, the load combination of '**WO Roof 1st PS ONLY**' is for the subtraction of 'Wall Ringbeam[Staged]' from 'Ringbeam 1st PS[Staged]'.

1) Wall Ringbeam[Staged] : Wall & Ringbeam is built. (i.e. no roof)
2) RingBeam 1st PS[Staged] : Wall & Ringbeam is built. (i.e. no roof). 1st PS is added.

Both loadcases are included in the Staged Construction Analysis, and the combination of 2)-1) is used to obtain the effect of the 1st PS loading during the construction stage.

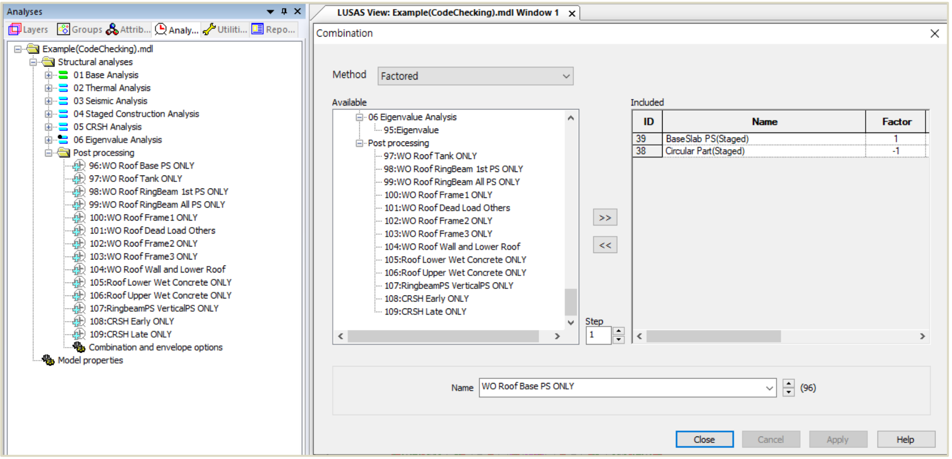


Fig 13 Pre-defined Load Combinations - WO Roof 1st PS only

Template for load combinations

A template for use in defining design load combinations is saved in the current working folder with name of the form: **<Model name>_ComboTemplate.xlsx**.

Update Reinforcement and Prestress

If the reinforcement or prestress tendon arrangement needs to be changed after the model is built, it can be updated by selecting the menu item **LNG Tank > Update Reinforcement…**

This only updates the existing properties. If the attribute is to be deassigned or deleted, it should be done manually.

If one-side only rebar is required, use a negligible value for Es. (e.g. a very small value

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** > **Thermal Analysis…** 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**.

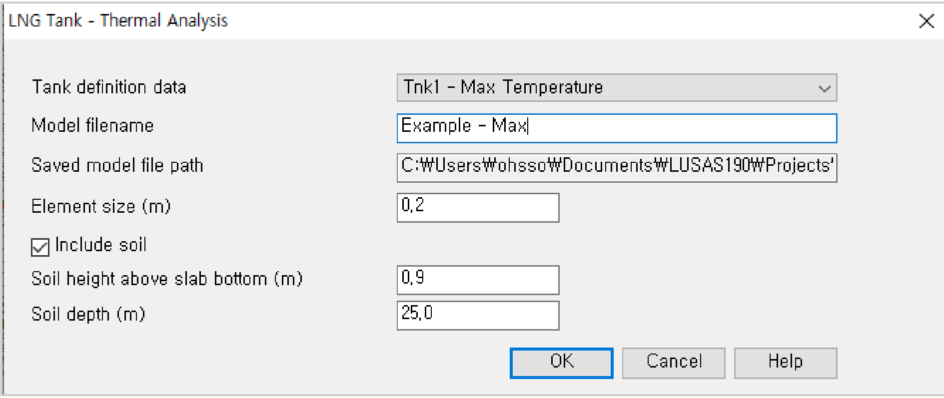


Fig 14 Dialog for Thermal Analysis (Max)

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

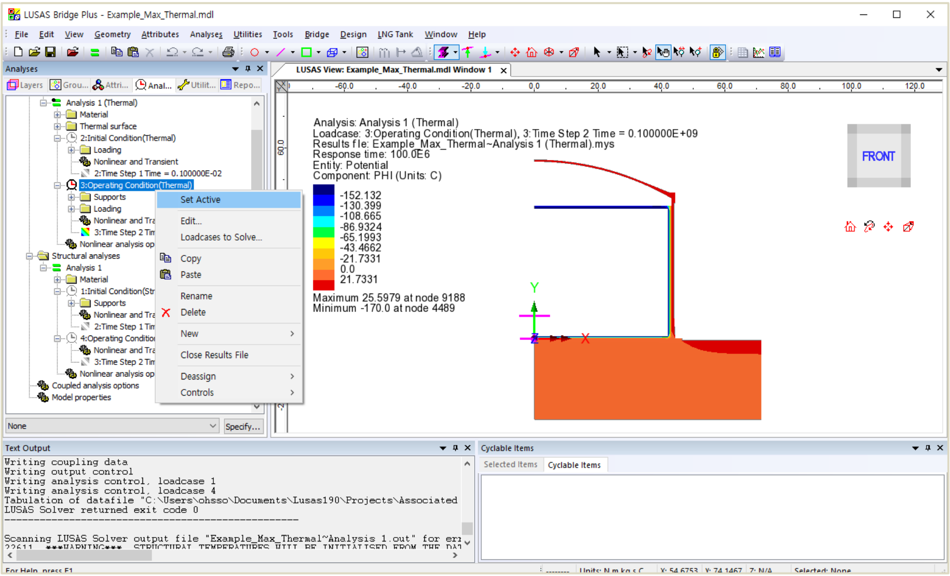


Fig 15 Thermal Analysis for Max Environmental Temperature

- 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 > Export Temperatures to Excel (2D)…**

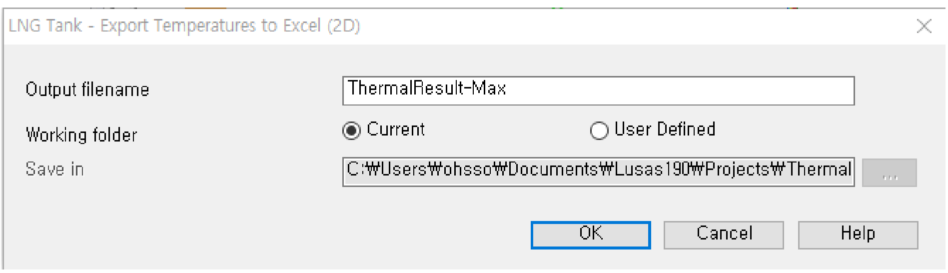
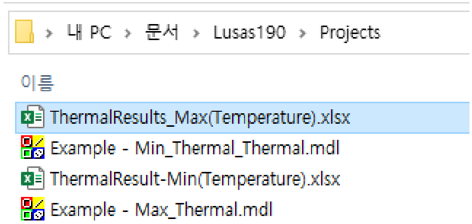


Fig 16 Extraction Thermal Max Results

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)
90.00	0.50	86.91	-30.65	0.50

Distance (m)	Inner Temp (°C)	Outer Temp (°C)	Thickness (m)	Average Temp (°C)
0.00	23.89	25.33	0.50	24.52
0.50	23.61	25.33	0.50	24.47
1.00	23.62	25.33	0.50	24.47
1.50	23.60	25.32	0.50	24.47
2.00	23.61	25.33	0.50	24.47

Fig 17 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…**

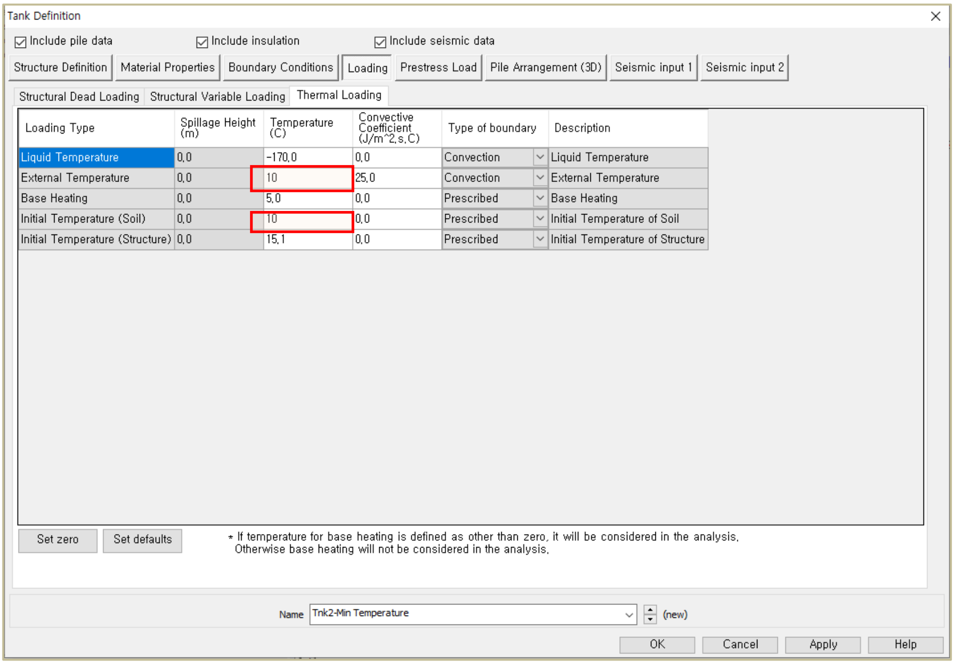


Fig 18 User Input for Min Environmental Temperature

- Enter **10 (**Celsius) for both the **External Temperature** and the **Initial Temperature (Soil)**. The **Initial Temperature (Structure)** is the temperature at the time when concrete is poured, hence there is no need to change at the moment.
- Save these inputs with name '**Tnk2-Min Temperature**'.
- Select the menu item **LNG Tank > Thermal Analysis…** and enter **Example - Min** for the model file name, then click **OK**.

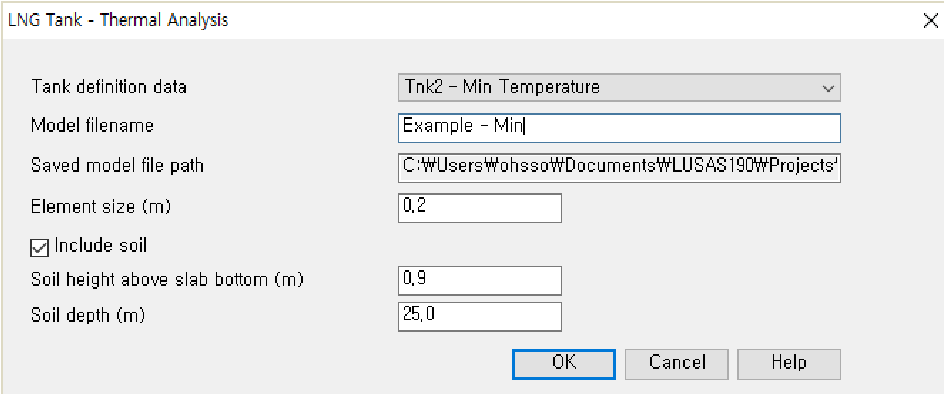


Fig 19 Dialog for Thermal Analysis (Min)

- 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 > Export Temperatures to Excel (2D)…**

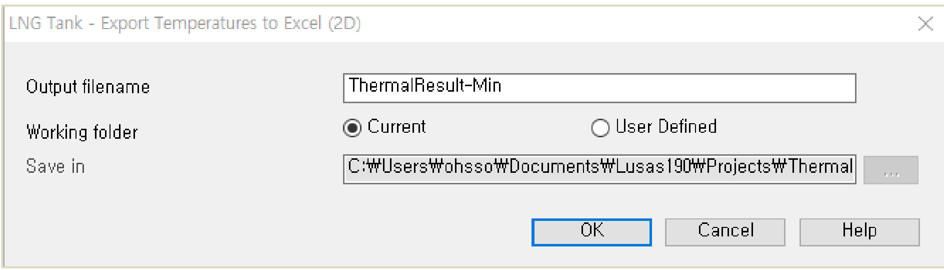


Fig 20 Extraction Thermal Min Results

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 '**Example(CodeChecking).mdl**'.
- Select the menu item **LNG Tank** > **Add Thermal Loading…** then select the spreadsheets for thermal analysis results and click **OK**.

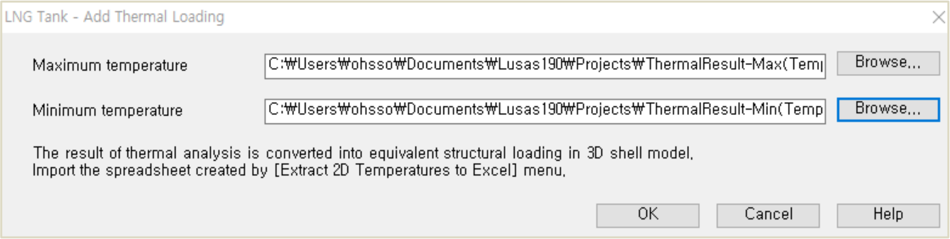


Fig 21 Dialog for Adding Thermal Loading

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

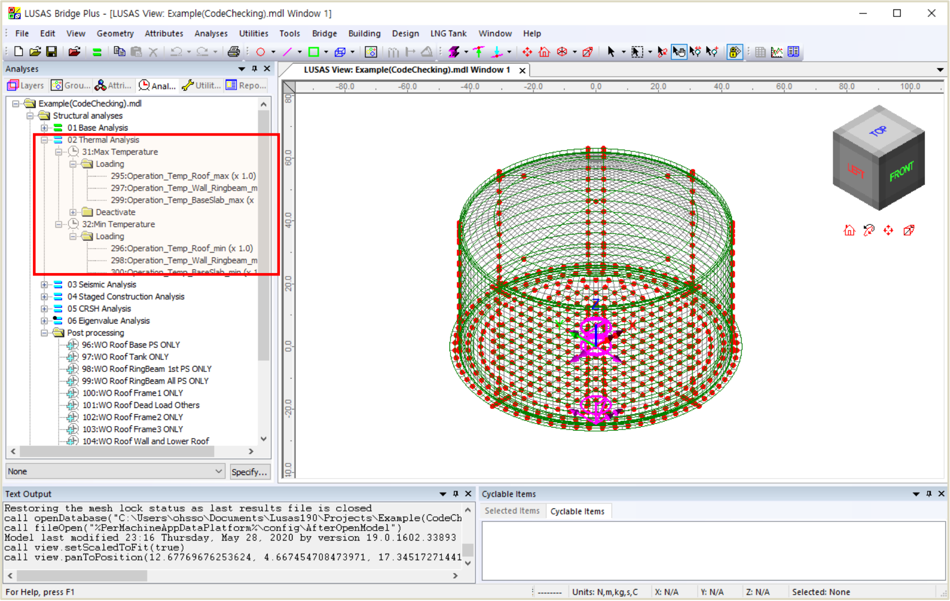


Fig 22 Temperature Loading from Thermal Analyses

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 Seismic Loading…**

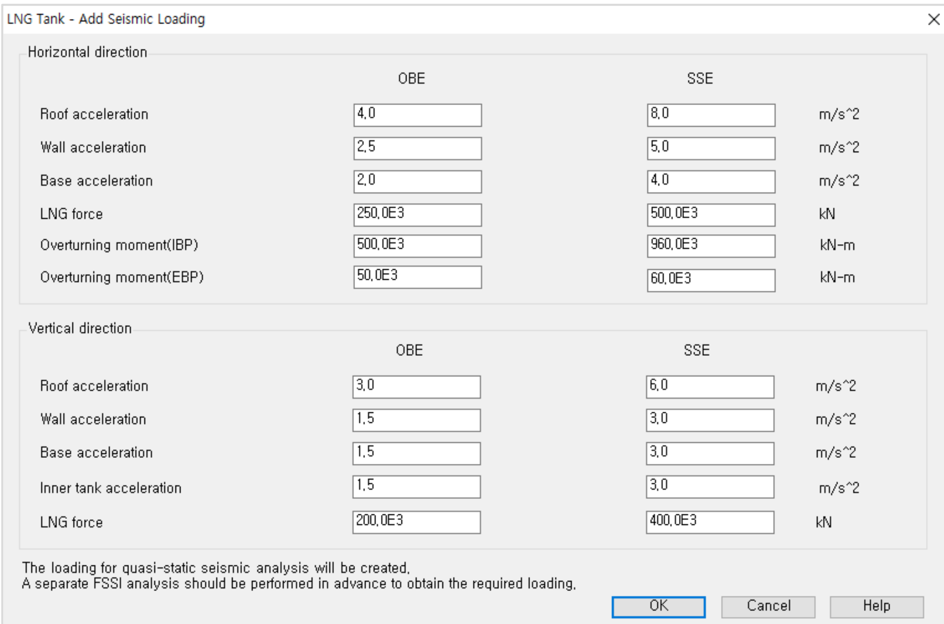


Fig 23 Dialog for Adding Seismic Loadings

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.

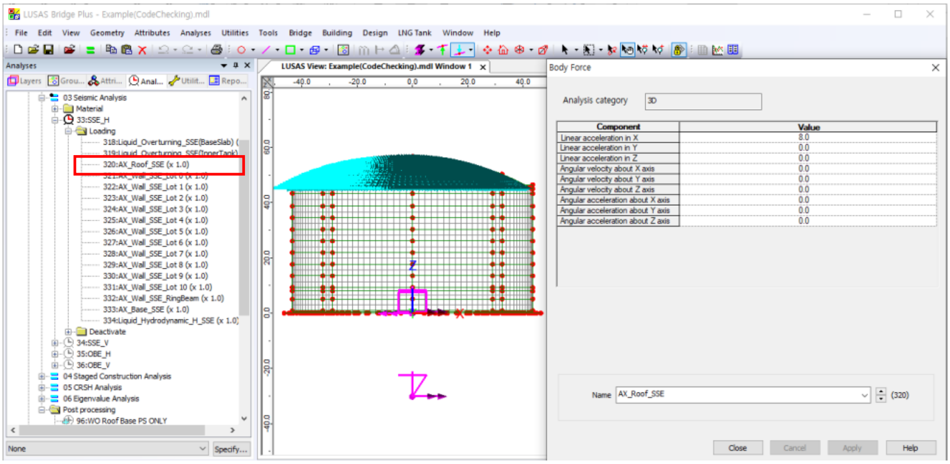


Fig 24 Horizontal Seismic Loading for Roof (SSE)

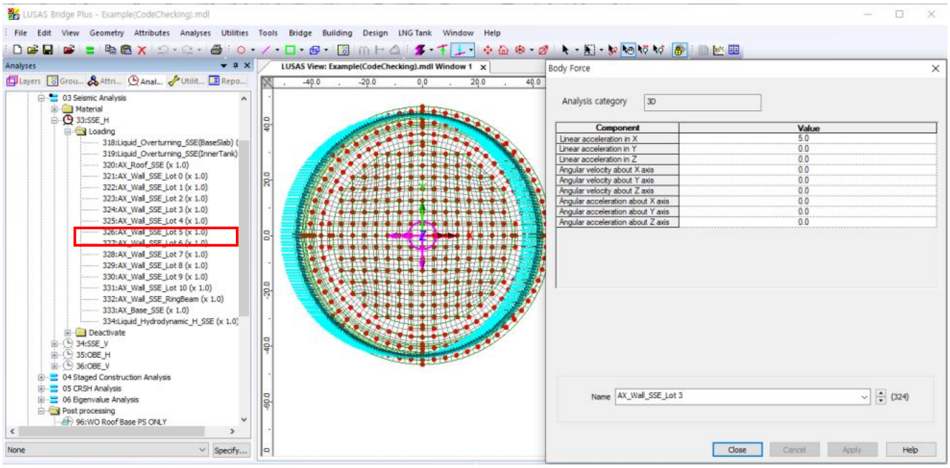


Fig 25 Horizontal Seismic Loading for Wall (SSE)

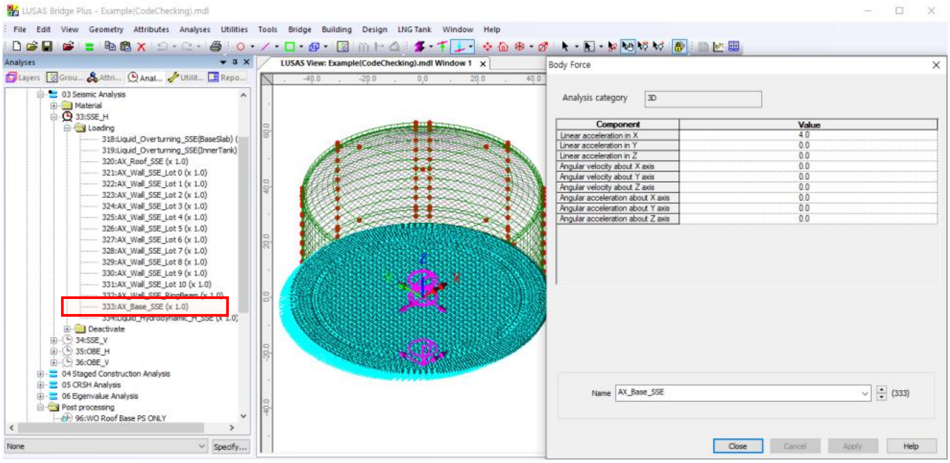


Fig 26 Horizontal Seismic Loading for Base Slab (SSE)

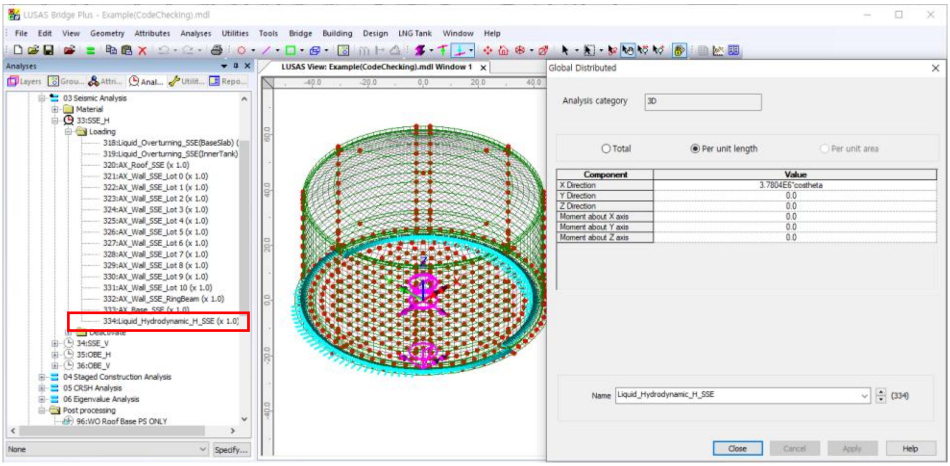


Fig 27 Horizontal Seismic Loading of Liquid Force (SSE)

- ❑ 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.

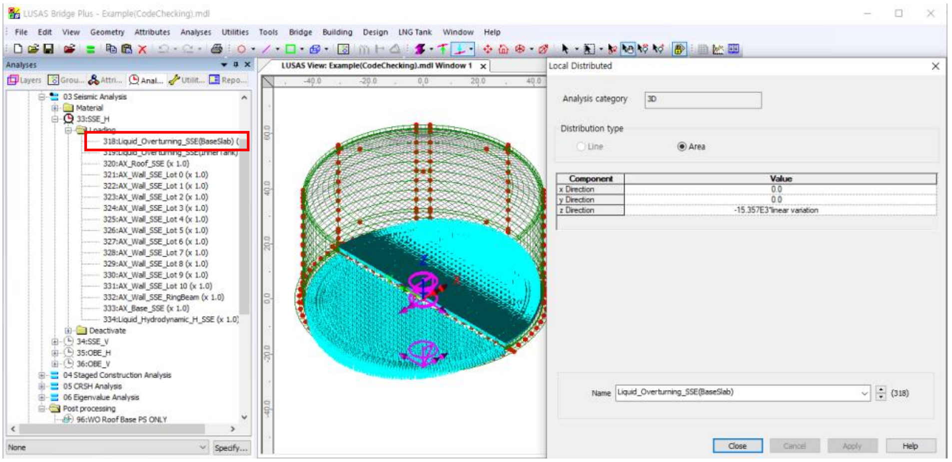


Fig 28 Horizontal Seismic Loading of Overturning Moment from Base Slab (SSE)

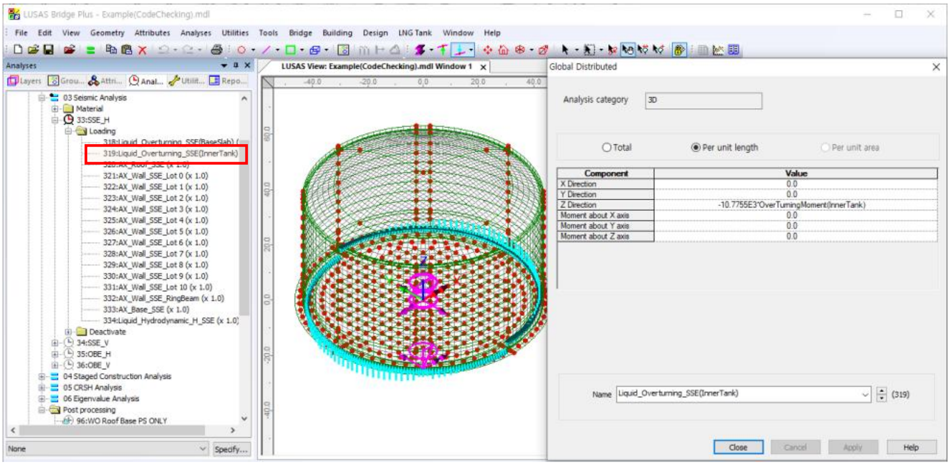


Fig 29 Horizontal Seismic Loading of Overturning Moment from Inner Tank (SSE)

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.

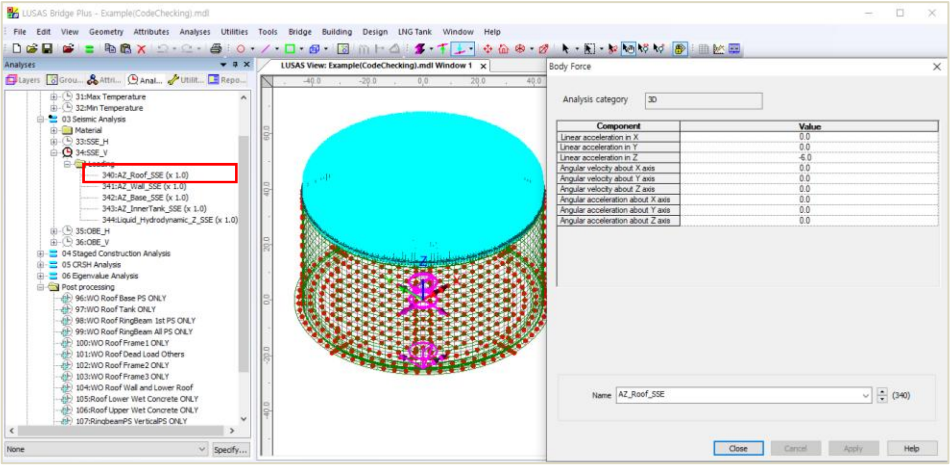


Fig 30 Vertical Seismic Loading for Roof (SSE)

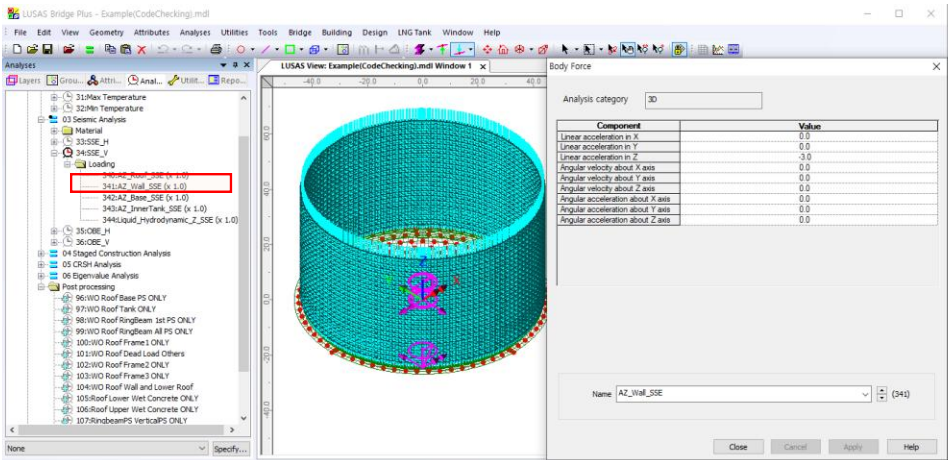


Fig 31 Vertical Seismic Loading for Wall (SSE)

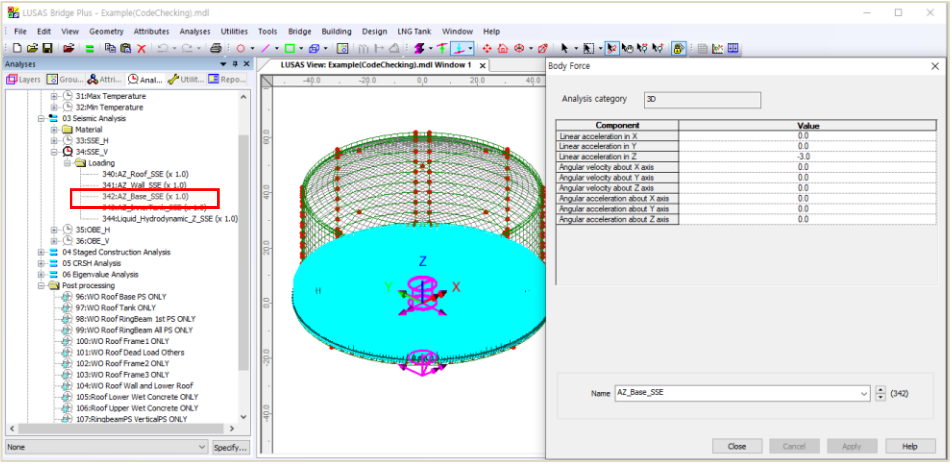


Fig 32 Vertical Seismic Loading for Base Slab (SSE)

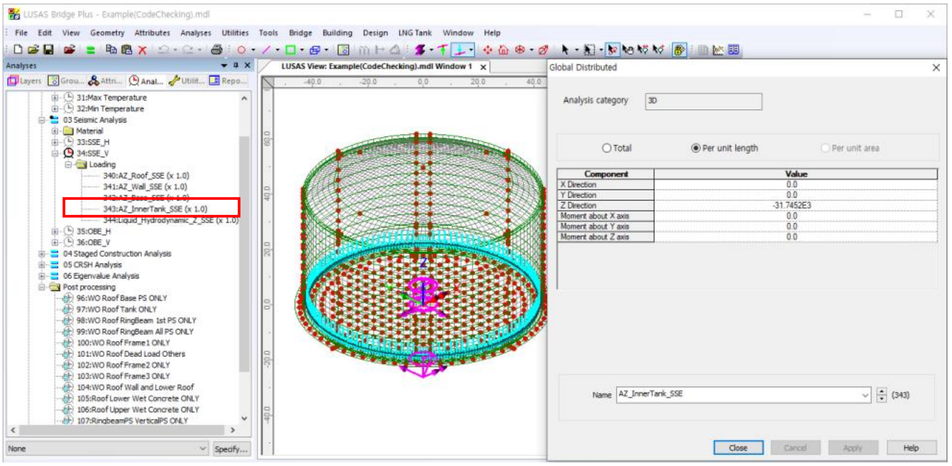


Fig 33 Vertical Seismic Loading for Inner Tank (SSE)

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

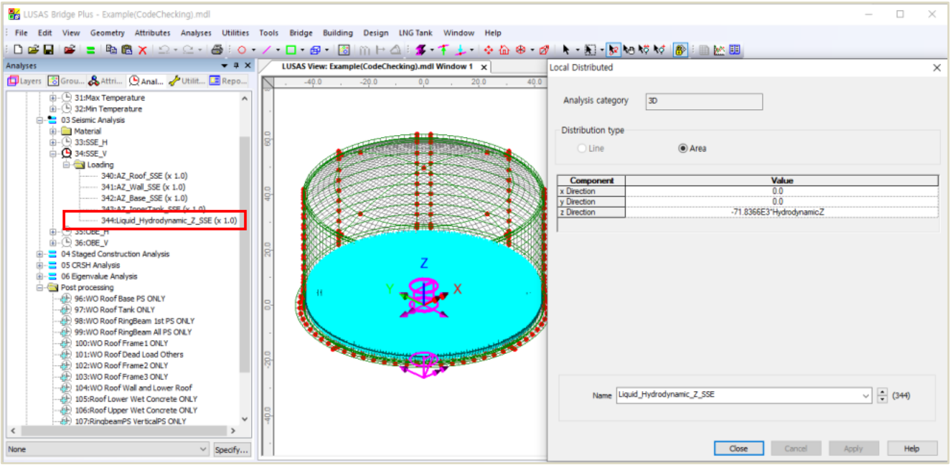


Fig 34 Vertical Seismic Loading for Liquid (Hydrostatic, SSE)

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

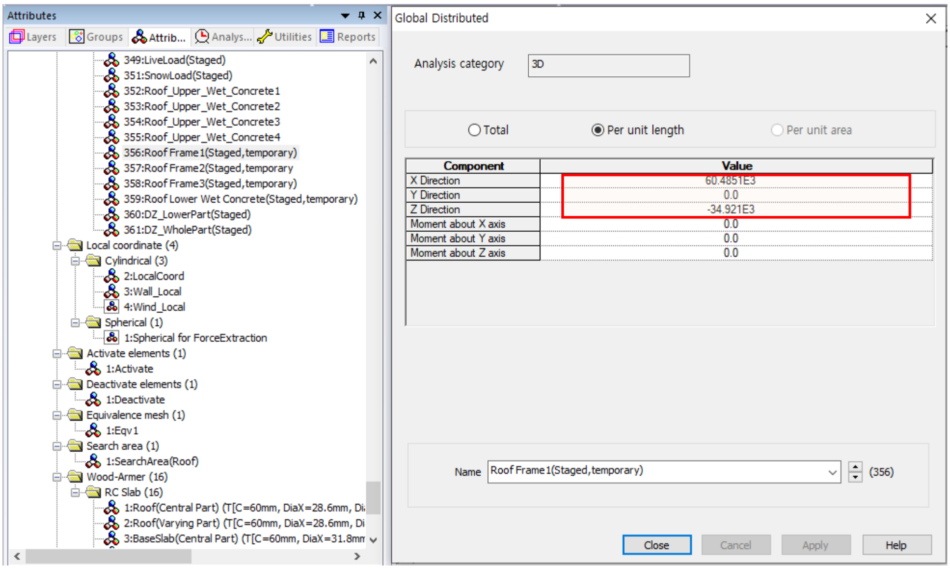


Fig 35 Roof Frame1 Loading

As these loadings are not permanent loading, they are not inherited by the subsequent stages and are marked as '**temporary**' in the loadcase name.

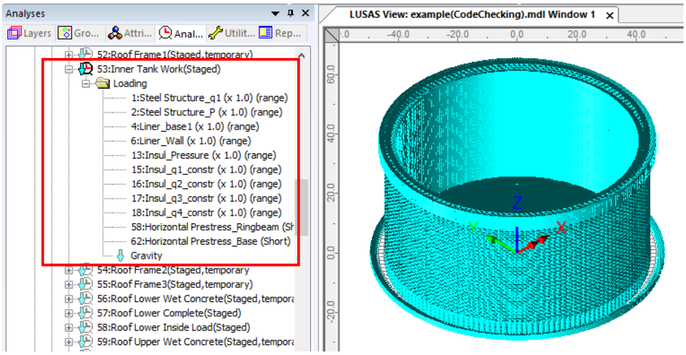


Fig 36 Stage 18 of Staged Construction Analysis

Stage 18 assumes that the inner tank has been built. All insulation loading except for ‘Roof Liner’ will be defined and assigned at this stage.

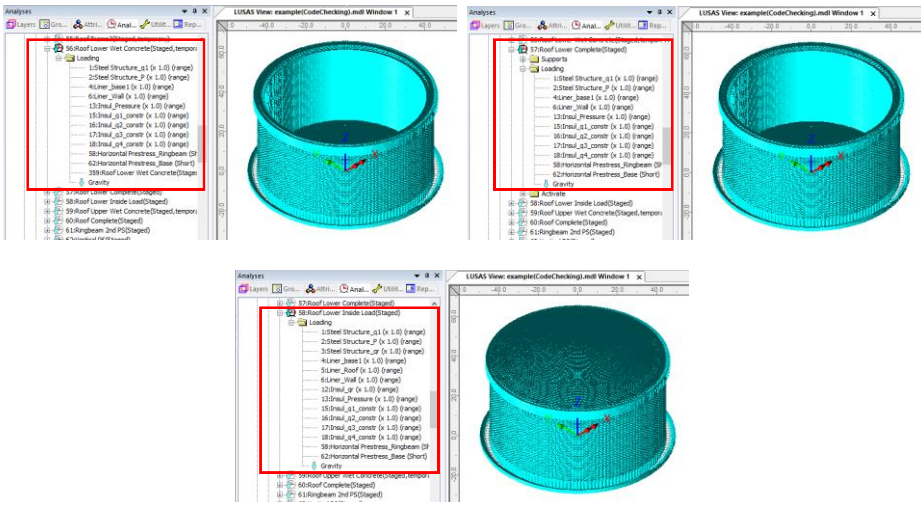


Fig 37 Stage 20-22 of Staged Construction Analysis

Stage 20 assumes that the lower half of the roof is being built and the poured concrete is acting as a loading on the ringbeam.

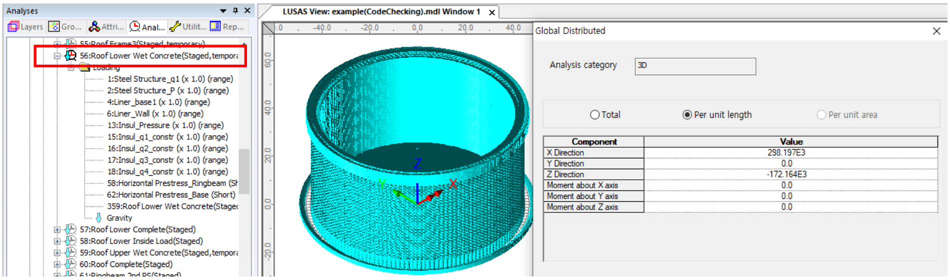


Fig 38 Stage 20 of Staged Construction Analysis

Stage 21 assumes that the lower half of the roof is built (if Roof ratio for 1st built = 0.5 from **Tank Definition**), and the lower part of the roof is newly activated. As shown below, the geometric properties used represent those for only half of the Roof at stage 21, only becoming geometric properties for the whole roof at stage 24.

Stage 22 assumes that the lower part of roof is completed. At this stage the wet concrete loading assigned at Stage 21 is removed and replaced with the body force of the lower part of Roof.

Fig 39 Roof Shape at Stage 21, 24.

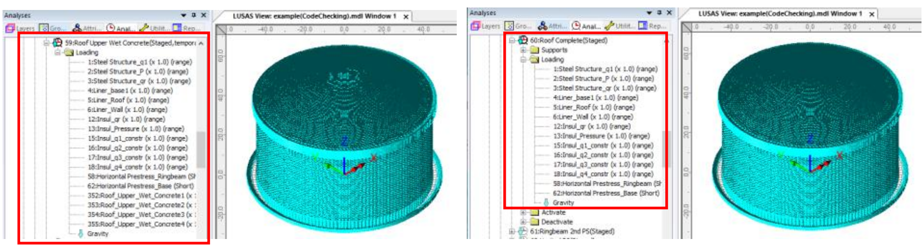


Fig 40 Stage 23 ~ 24 of Staged Construction Analysis

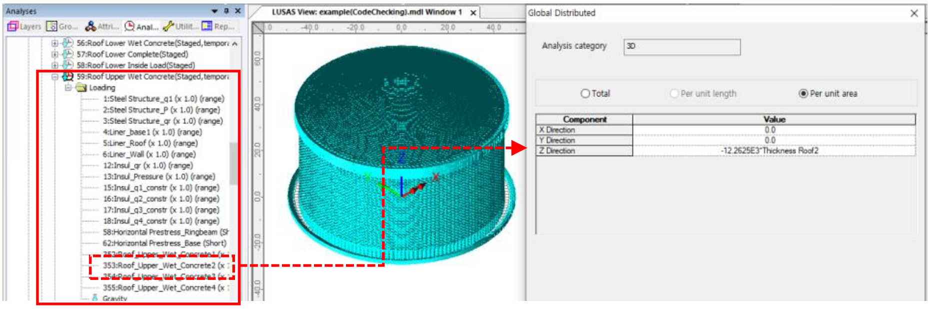


Fig 41 Stage 23 of Staged Construction Analysis

Stage 23 models the upper half of the Roof being built with the poured concrete acting as a load on the already cast lower half of the Roof.

Stage 24 assumes that the upper part of the Roof is now built. The wet concrete loading assigned at Stage 23 is removed and replaced with the body force of the upper part of the Roof.

At Stage 25, 50% of additional RingBeam Prestress is added. (The Load Factor is updated from 0.5 to 1.0 for Horizontal Prestress_Ringbeam load.)

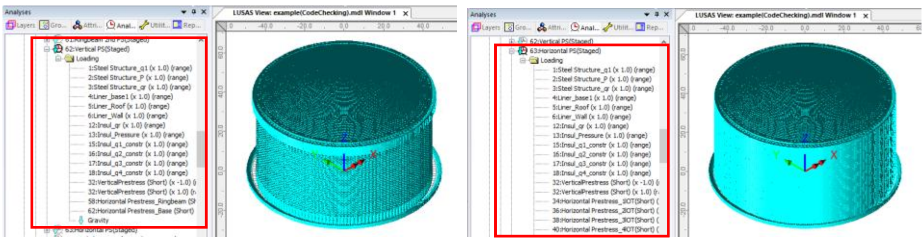


Fig 42 Stage 26 ~ 27 of Staged Construction Analysis

At **Stage 26,** the vertical prestress is added.

At **Stage 27**, all the horizontal prestress for the Wall is added.

Stage 28 is the final stage. The structures is complete as built, and all loadings for the operating condition are added.

Stage 29 models long-term effects. The prestress values are updated to those for long-term PS.

If 'Roof ratio for 1st built' is set to 1, the following sequence for the staged construction analysis will be applied.

Stage	Description	Note
No. 1	Annular part	

Stage	Description	Note
No. 2	1) + Circular part	
No. 3	2) + Base PS	
No. 4~14	3) + Wall & Ringbeam is added up in stages	
No. 15	4) + Ringbeam 1st PS	
No. 16	15) + Roof Frame 1	
No. 17	15) + Inner Tank Work	
No. 18	17) + Roof Frame 2	
No. 19	17) + Roof Frame 3	
No. 20	17) + Roof Wet Concrete	
No. 21	17) + Roof Complete	
No. 22	21) + Roof Lower Inside Load	
No. 23	22) + Ringbeam 2nd PS	
No. 24	23) + Vertical PS	
No. 25	24) + Horizontal PS	
No. 26	25) + Final Short term	
No. 27	25) + Final Long term (Long term PS applied)	

Table 1 Sequence of construction stages (Roof ratio for 1st built = 1)

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.

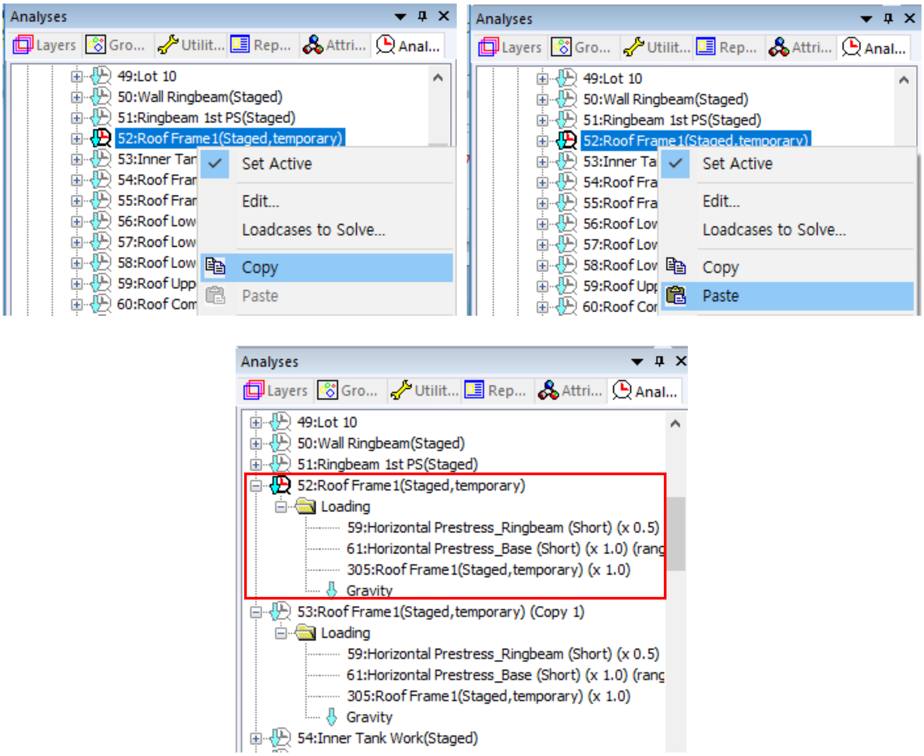


Fig 43 Copy and Paste of Stages (Loadcases)

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.

User Updates

Loadings

As discussed at in the section titled **Error! Reference source not found.**, some loadings may need 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 5 and Stage 6 needs to be changed to 15 days, Nonlinear Control for Stage 5 should be updated.

Default settings for Stage 5 are shown below.

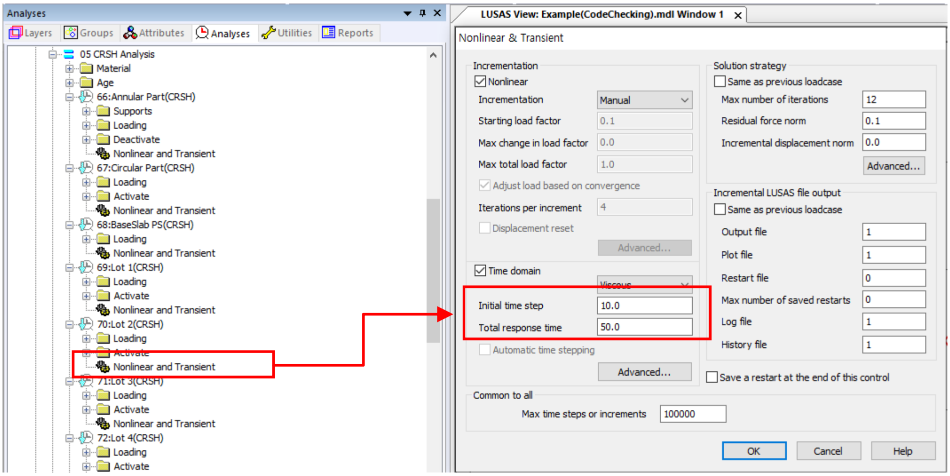


Fig 44 Nonlinear Control for Stage of Lot 2 (Default)

- ❑ **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 <u>Total Response Time of current stage</u> minus <u>Total Response Time of the previous stage</u>.

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.

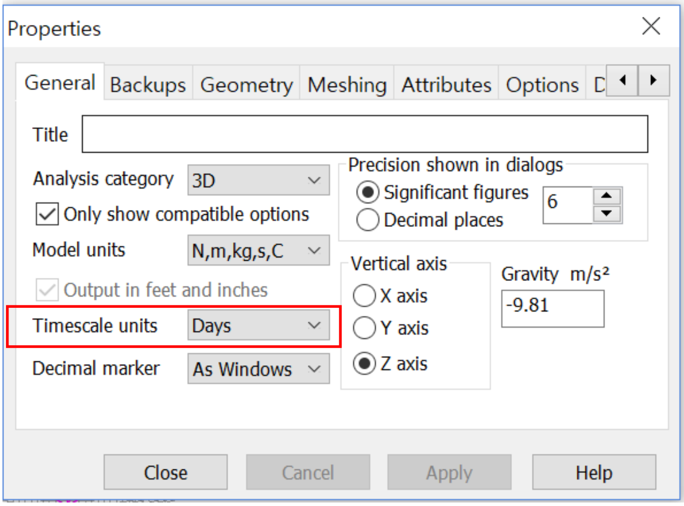


Fig 45 Model Properties

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

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.

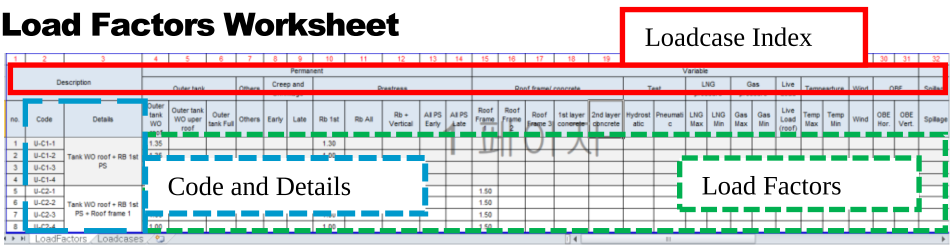
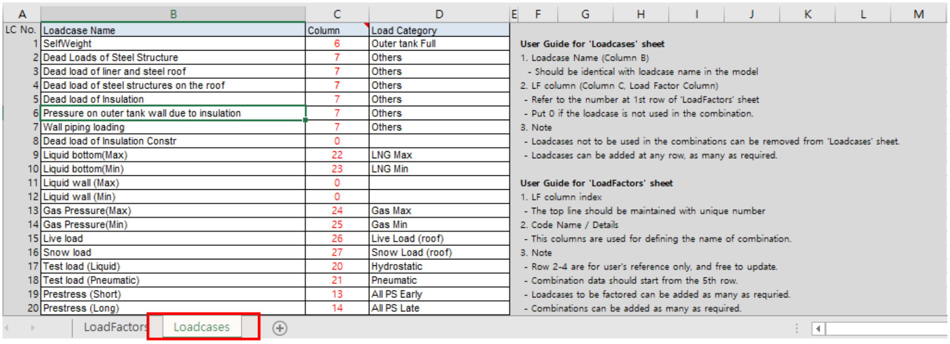


Fig 46 Template for Design Load Combinations, Load Factors

- **Loadcase Index** The numbers in this row are used to match a Modeller loadcase number with a loadcase and its associated details in this template. The row should contain a series of numbers with no duplication.
- **Code** and **Details** Code and Details are used for naming the combination data in Modeller.
- **Load Factors** Load factors for each loadcase is defined here.
- **Others** All other data are for users reference, and not used for processing.
- Loadcase to consider can be added. (more columns as necessary can be added.)
- Load combinations can be added. (more rows as necessary can be added)

Loadcases Worksheer



LC No.	Loadcase Name	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	Dead load of Insulation Constr	0	
9	Liquid bottom(Max)	22	LNG Max
10	Liquid bottom(Min)	23	LNG Min
11	Liquid wall (Max)	0	
12	Liquid wall (Min)	0	
13	Gas Pressure(Max)	24	Gas Max
14	Gas Pressure(Min)	25	Gas Min
15	Live load	26	Live Load (roof)
16	Snow load	27	Snow Load (roof)
17	Test load (Liquid)	20	Hydrostatic
18	Test load (Pneumatic)	21	Pneumatic
19	Prestress (Short)	13	All PS Early
20	Prestress (Long)	14	All PS Late

User Guide for 'Loadcases' sheet
1. Loadcase Name (Column B)
- Should be identical with loadcase name in the model
2. LF column (Column C, Load Factor Column)
- Refer to the number at 1st row of 'LoadFactors' sheet
- Put 0 if the loadcase is not used in the combination.
3. Note
- Loadcases not to be used in the combinations can be removed from 'Loadcases' sheet.
- Loadcases can be added at any row, as many as required.

User Guide for 'LoadFactors' sheet
1. LF column index
- The top line should be maintained with unique number
2. Code Name / Details
- This columns are used for defining the name of combination.
3. Note
- Row 2-4 are for user's reference only, and free to update.
- Combination data should start from the 5th row.
- Loadcases to be factored can be added as many as requried.
- Combinations can be added as many as required.

LoadFactors | Loadcases

Fig 47 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. <u>Note that the loadcase names used must be the same as the loadcase names defined in Modeller.</u>
- ❑ **(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 <u>Self Weight</u> loadcase in Modeller is used in the <u>'Outer Tank Full'</u> combination defined in LoadFactors sheet. By entering 0 (zero), the loadcase is ignored and will not be used in the combination.
- ❑ **Others** 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 Load Combination…**

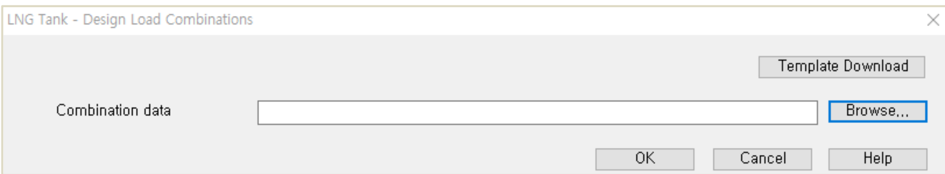


Fig 48 Dialog for Design Load Combination

- ❑ **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.

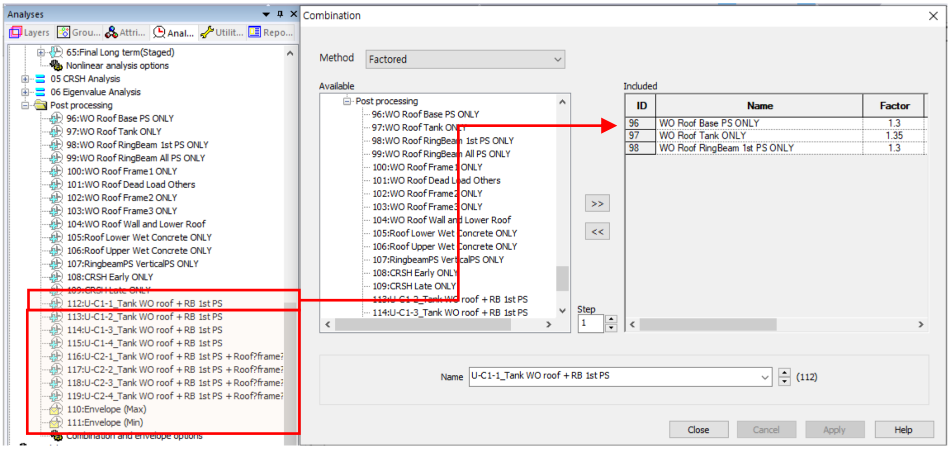


Fig 49 Load Combinations Created in Modeller

Design Check

COP Parameters

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

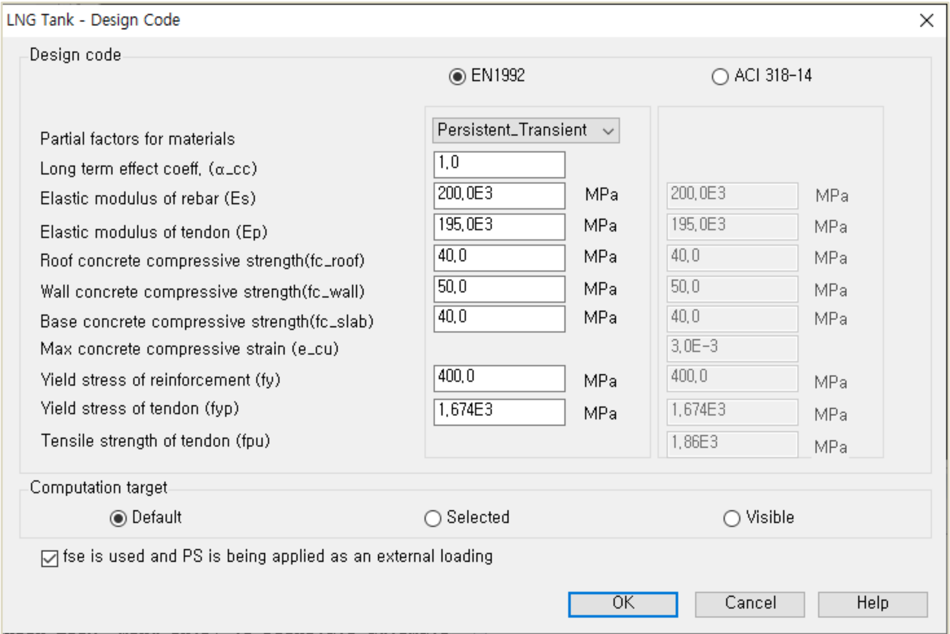
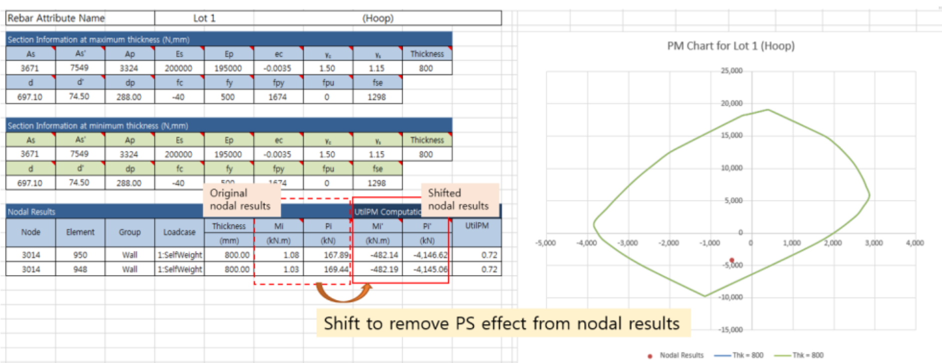


Fig 50 Dialog for Design Parameters

- ❑ **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.
- ❑ **Long term effect coeff. (α_{cc})** : Long term effect coefficient (EN1991 only)
- ❑ **Elastic Modulus of rebar (Es) :** Elastic Modulus of reinforcing steel in MPa.
- ❑ **Elastic Modulus of Tendon (Ep) :** Elastic Modulus of prestressing steel in MPa
- ❑ **Roof Concrete Compressive Strength (fc_roof) :** Concrete strength in MPa.
- ❑ **Wall Concrete Compressive Strength (fc_wall) :** Concrete strength in MPa.
- ❑ **Base Concrete Compressive Strength (fc_slab):** Concrete strength in MPa.

- ❑ **Max Concrete Compressive Strain :** Ultimate strain at concrete failure (ACI 318-14 only)
- ❑ **Yield Stress of Reinforcement (fy)** : Yield strength of reinforcing steel in MPa.
- ❑ **Yield Stress of Tendon (fyp)** : Yield strength of prestressing steel in MPa.
- ❑ **Tensile strength of tendon (fpu)** : Tensile strength of prestressing steel in MPa. (ACI 318-14 only)
- ❑ **Computation Target :** 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.
- ❑ 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 <u>double count</u> 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.

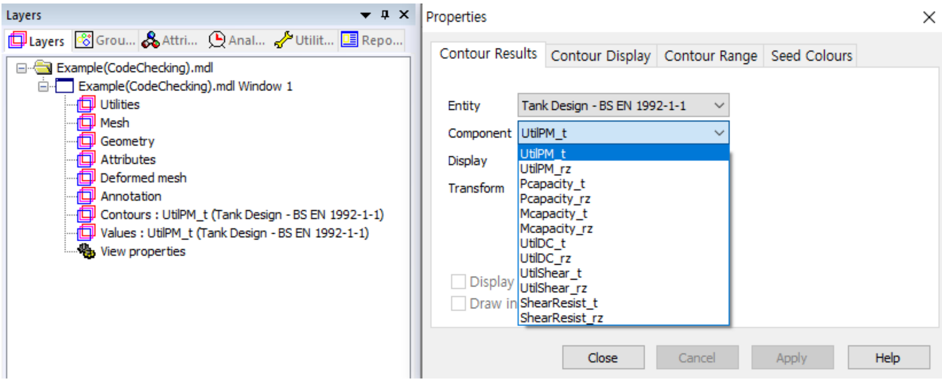


Fig 51 Components for Design Checks

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.

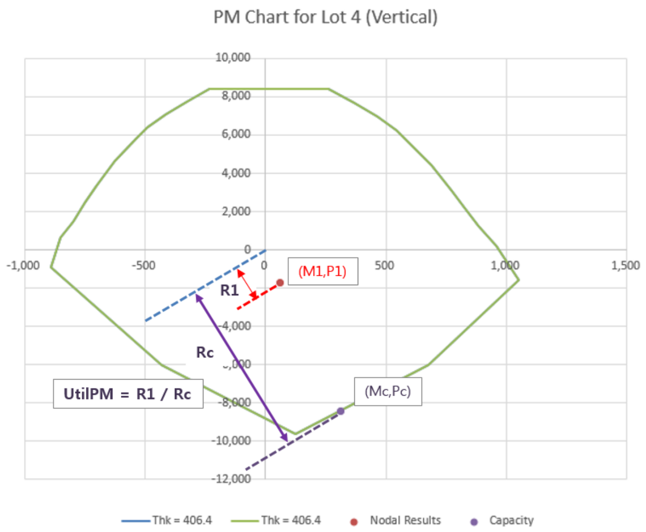


Fig 52 Definition of UtilPM

- **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.

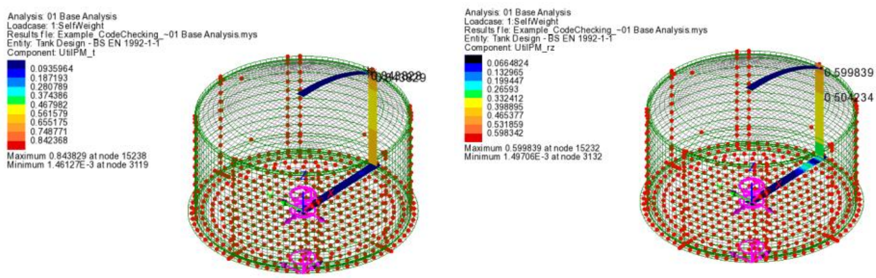


Fig 53 Contour for UtilPM_t, UtilPM_rz

UtilDC (Decompression Utilization)

UtilDC aims to check if 25mm of concrete around a tendon is in compression. UtilDC is only available if **EN1992** is chosen, and a value of less than 1 means it satisfies the design code.

The UtilDC calculation assumes the stress distribution through the thickness is linear.

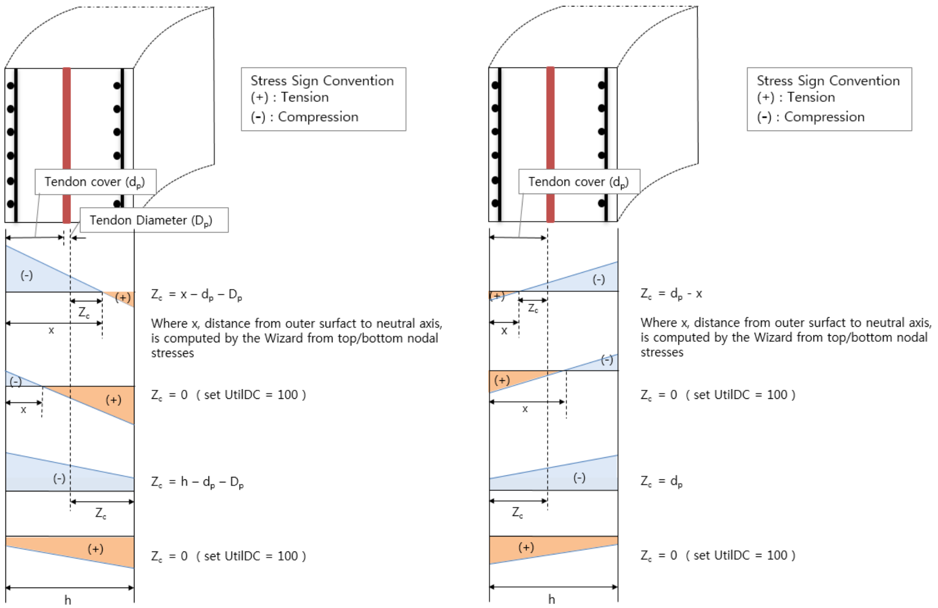


Fig 54 Zc for Decompression Check

The tendon cover (d_p) is the distance from outer surface to the tendon surface.

When the section is fully in compression the maximum value is set to 100. (ie. if Zc<=0.25mm, UtilDC becomes 100.)

UtilDC = Zdec / Zc = 25 / Zc.

- ❑ **UtilDC_t** is the tendon and stress utilization in the hoop direction, and **UtilDC_rz** is the utilization in the radial direction for the Roof and Base Slab, and in the vertical direction for the Wall.

UtilDC is only available for sections that contain prestress tendons. In the hoop direction, **UtilDC_t** is available for Wall and Slab. In the radial and vertical direction, **UtilDC_rz** is available only for Wall.

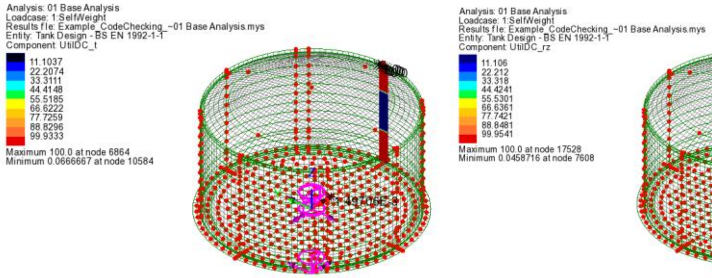


Fig 55 Contours for UtilDC_t, UtilDC_rz

If concrete at the prestress tendon location is in tension UtilDC is set to 100.

ShearResist (Shear Resistances)

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

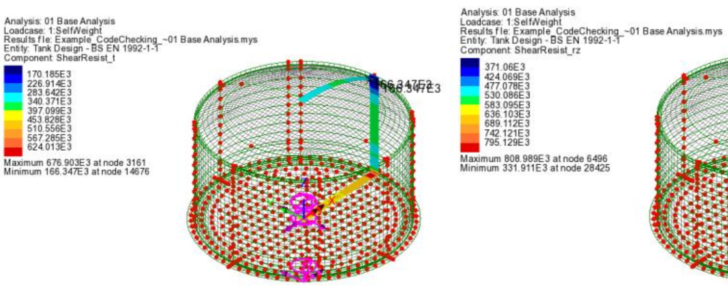


Fig 56 Contour for ShearResist_t, ShearResist_rv

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.

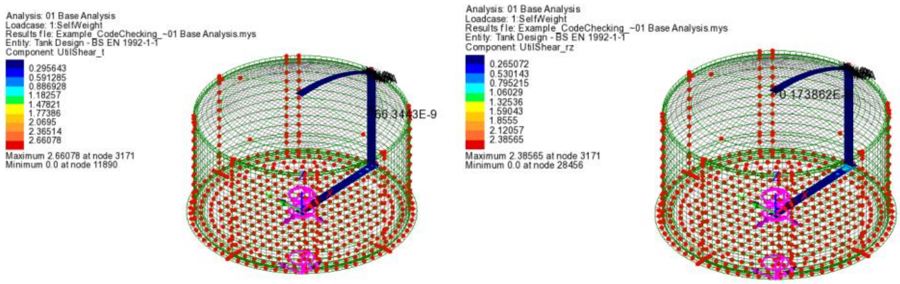


Fig 57 Contour for UtilShear_t, UtilShear_rv

PM Chart Report

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

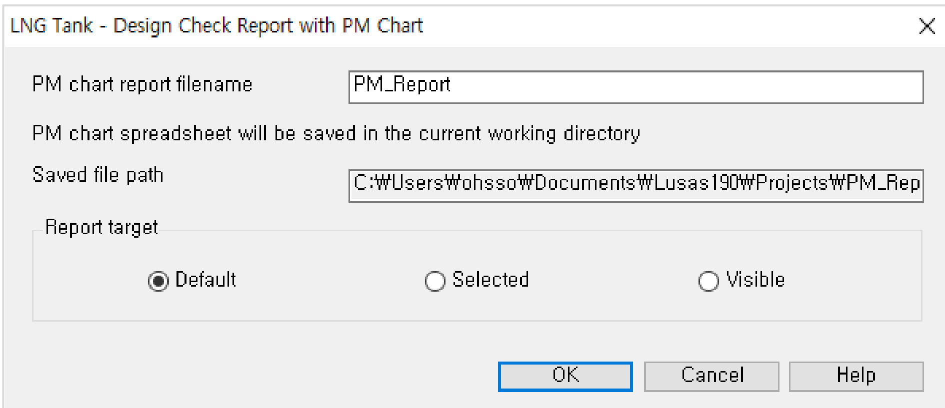
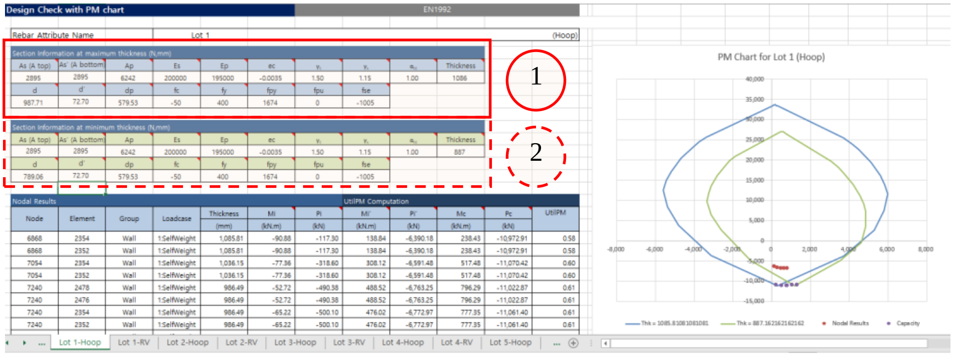


Fig 58 Dialog for Design Check Report with PM Chart

- ❑ **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.

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.



Design Check with PM chart — EN1992

Rebar Attribute Name: Lot 1 (Hoop)

Section Information at maximum thickness (N,mm)

As (A top)	As' (A bottom)	Ap	Es	Ep	ec	γ_c	γ_s	α_{cc}	Thickness
2895	2895	6242	200000	195000	-0.0035	1.50	1.15	1.00	1086
d	**d'**	**dp**	**fc**	**fy**	**fpy**	**fpu**	**fse**		
987.71	72.70	579.53	-50	400	1674	0	-1005		

Section Information at minimum thickness (N,mm)

As (A top)	As' (A bottom)	Ap	Es	Ep	ec	γ_c	γ_s	α_{cc}	Thickness
2895	2895	6242	200000	195000	-0.0035	1.50	1.15	1.00	887
d	**d'**	**dp**	**fc**	**fy**	**fpy**	**fpu**	**fse**		
789.06	72.70	579.53	-50	400	1674	0	-1005		

Nodal Results / UtilPM Computation

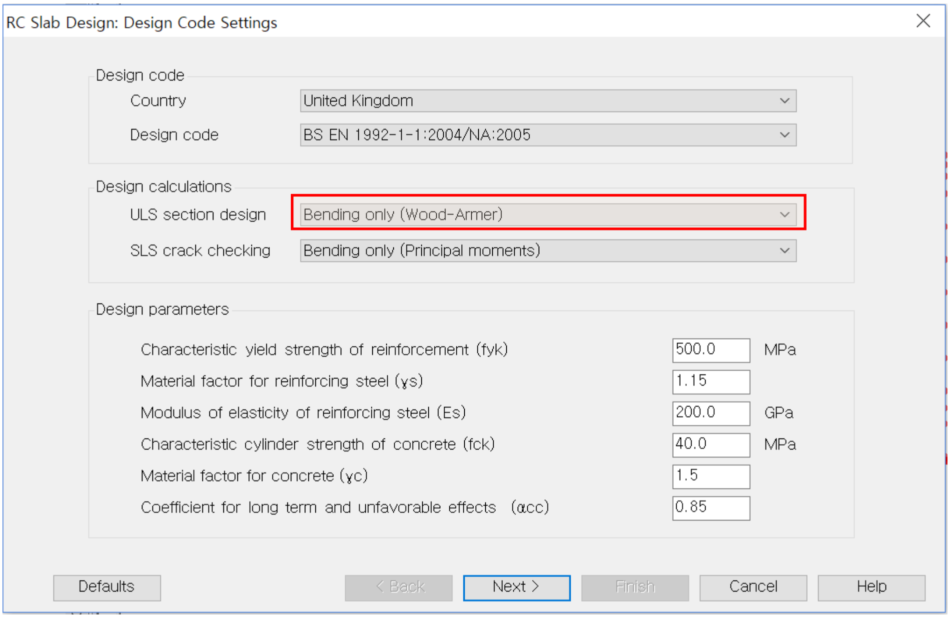
Node	Element	Group	Loadcase	Thickness (mm)	Mi (kN.m)	Pi (kN)	Mi' (kN.m)	Pi' (kN)	Mc (kN.m)	Pc (kN)	UtilPM
6868	2354	Wall	1:SelfWeight	1,085.81	-90.88	-117.30	138.84	-6,390.18	238.43	-10,972.91	0.58
6868	2352	Wall	1:SelfWeight	1,085.81	-90.88	-117.30	138.84	-6,390.18	238.43	-10,972.91	0.58
7054	2354	Wall	1:SelfWeight	1,036.15	-77.36	-318.60	308.12	-6,591.48	517.48	-11,070.42	0.60
7054	2352	Wall	1:SelfWeight	1,036.15	-77.36	-318.60	308.12	-6,591.48	517.48	-11,070.42	0.60
7240	2478	Wall	1:SelfWeight	986.49	-52.72	-490.38	488.52	-6,763.25	796.29	-11,022.87	0.61
7240	2476	Wall	1:SelfWeight	986.49	-52.72	-490.38	488.52	-6,763.25	796.29	-11,022.87	0.61
7240	2354	Wall	1:SelfWeight	986.49	-65.22	-500.10	476.02	-6,772.97	777.35	-11,061.40	0.61
7240	2352	Wall	1:SelfWeight	986.49	-65.22	-500.10	476.02	-6,772.97	777.35	-11,061.40	0.61

Fig 59 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

Design Checks for RC Slab

By selecting the **Design>RC Slab Design** menu item, followed by an appropriate design code, a number of other design checks are available.



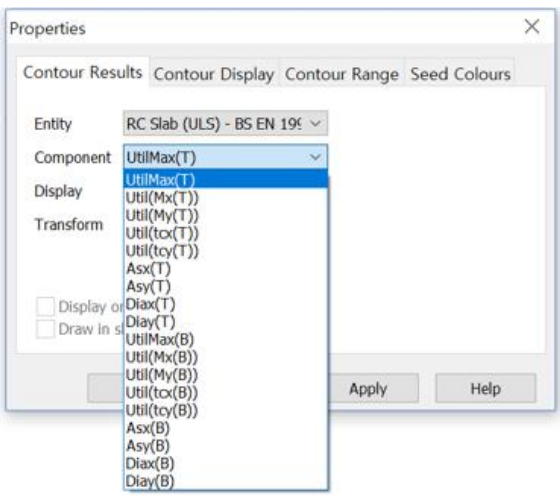
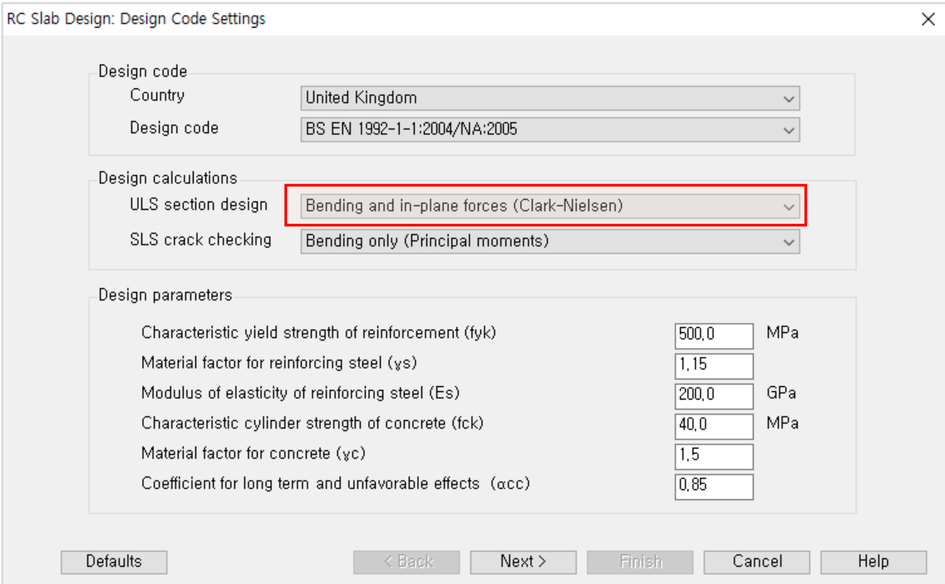


Fig 60 Dialog for RC Slab Design and Result Components (1)



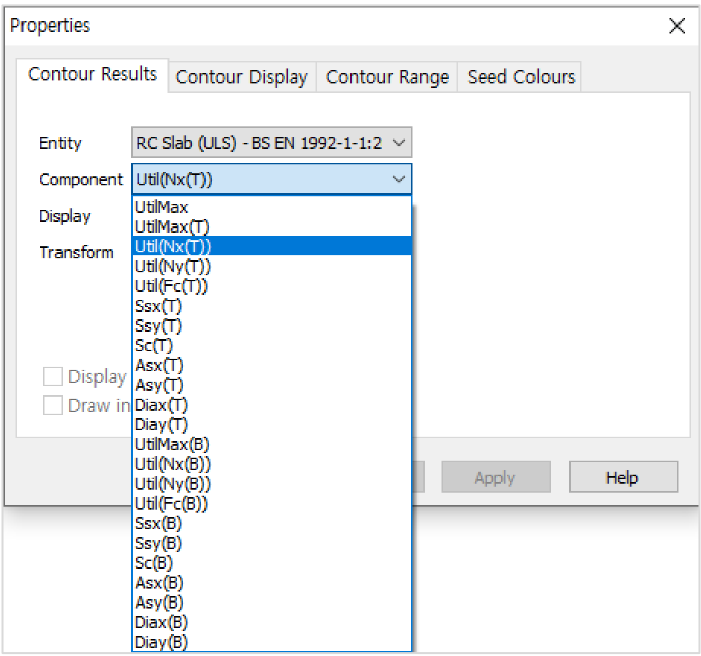


Fig 61 Dialog for RC Slab Design and Result Components (2)

The target group for which design check results are to be obtained must be set in the 'Groups' treeview as shown below.

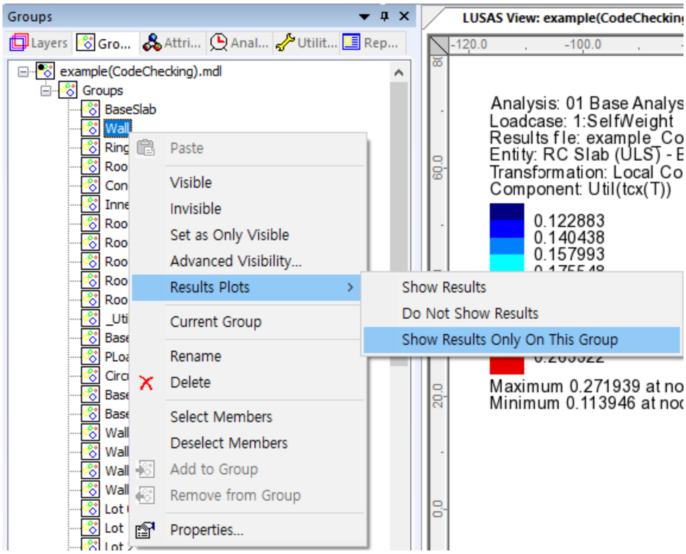


Fig 62 Show Results for a Selected Group

ULS - Stress Check

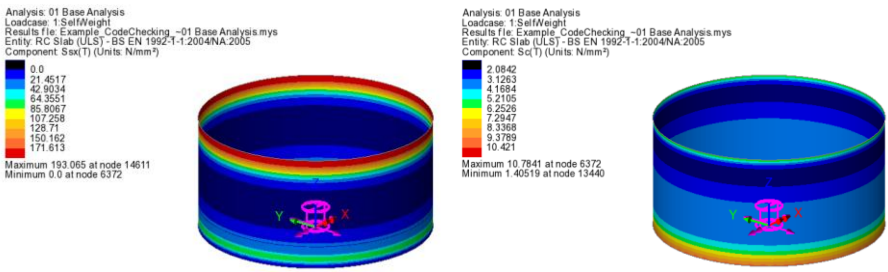


Fig 63 RC Slab Design - Stress Checks

- **Ssx(T)/Ssy(T) :** Inner rebar stress
- **Ssx(B)/Ssy(B) :** Outer rebar stress
- **Sc(T) :** Inner surface concrete stress
- **Sc(B)** : Outer surface concrete stress

ULS – Utilization

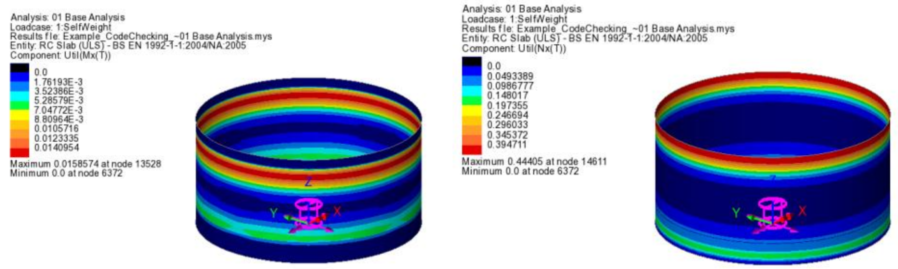


Fig 64 RC Slab Design – Utilization for moment and axial stress

- ❑ **Util(Mx(T)), Util(My(T)) :** Inner surface moment ratio (moment / moment resistance)
- ❑ **Util (Mx(B)), Util(My(B)) :** Outer surface moment ratio (moment / moment resistance)
- ❑ **Util(Nx(T)), Util(Ny(T)) :** Inner rebar stress ratio (Design rebar stress / rebar Yield stress)
- ❑ **Util(Nx(B)), Util(Ny(B)) :** Outer rebar stress ratio (Design rebar stress / rebar Yield stress)

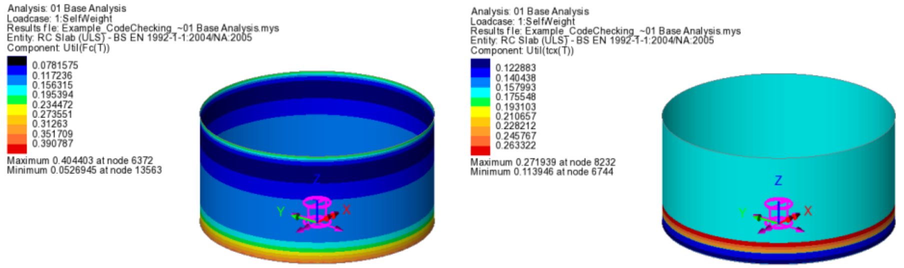


Fig 65 RC Slab Design – Utilization for Concrete Stress

- ❑ **Util(Fc(T)) :** Ratio (Concrete stress / Concrete compressive strength)
- ❑ **Util(Tcx(T)),Util(Tcy(T)) :** Ratio (Concrete tensile stress / Tensile limit) for inner surface
- ❑ **Util(Tcx(B)),Util(Tcy(B)) :** Ratio (Concrete tensile stress / Tensile limit) for outer surface

ULS – Design

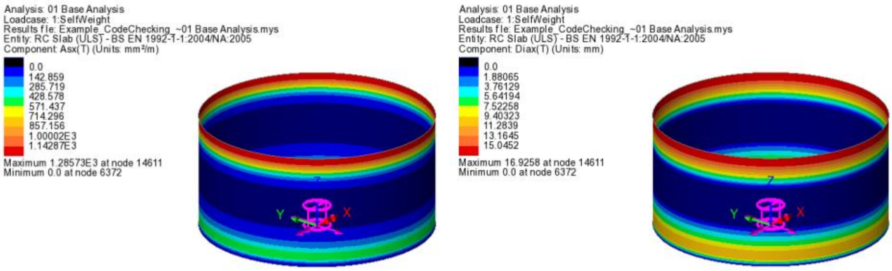


Fig 66 RC Slab Design – Reinforcement Design

- ❑ **Asx(T), Asy(T)** Minimum reinforcement area required for inner side
- ❑ **Asx(B), Asy(B)** Minimum reinforcement area required for outer side
- ❑ **Diax(T), Diay(T)** Minimum reinforcement diameter required for inner side based on given rebar spacing.
- ❑ **Diax(B), Diay(B)** Minimum reinforcement diameter required for outer side based on given rebar spacing.

SLS– Crack Width

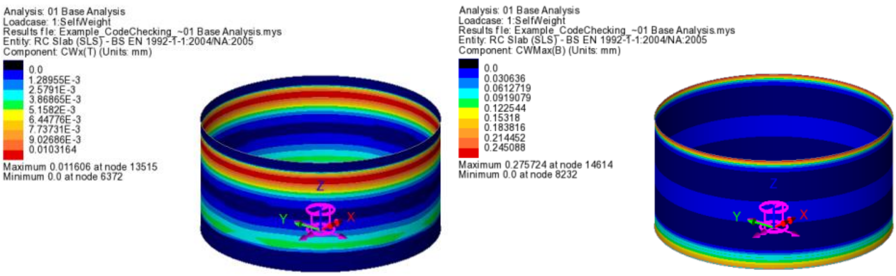


Fig 67 RC Slab Design – Crack Width

- ❑ **CWx(T), CWy(T)** Maximum crack width for local direction at inner side surface
- ❑ **CWx(B), CWy(B)** Maximum crack width for local direction at outer side surface
- ❑ **CWMax(T), CWMax(T)** Maximum crack width orthogonal to principle stress at inner side surface
- ❑ **CWMax(B), CWMax(B)** Maximum crack width orthogonal to principle stress at outer side surface

SLS– Utilization

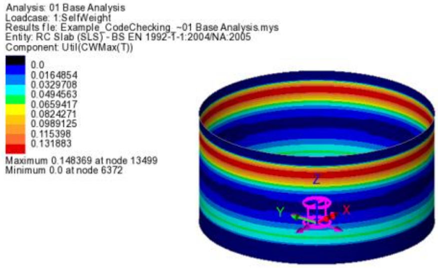


Fig 68 RC Slab Design – Crack Width Utilization

- **Util(CWMax(T)), Util(CWMax(T)) :** Ratio (max crack width / allowable limit) for inner surface
- **Util(CWMax(B)), Util(CWMax(B)) :** Ratio (max crack width / allowable limit) for outer surface


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