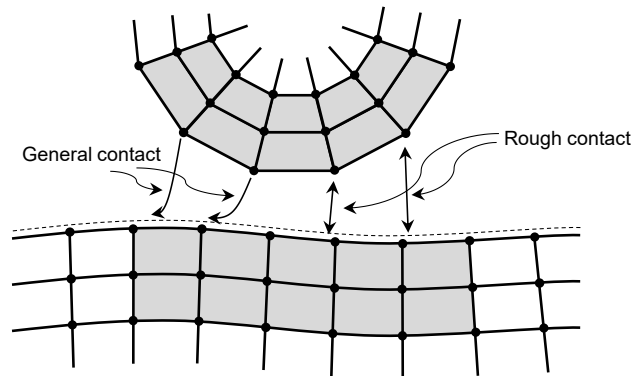




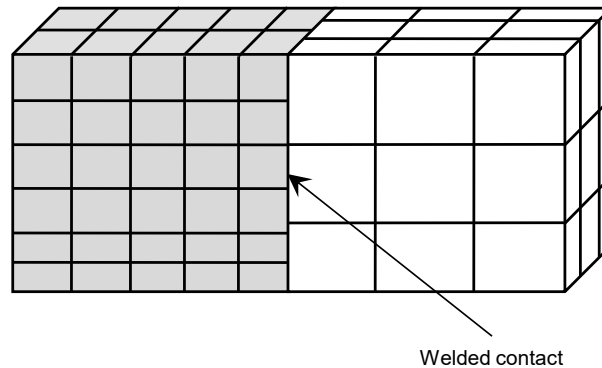
Section 1 Contact

Contact analysis fundamentally assumes that two objects in a space can be in contact, but cannot penetrate each other (non-penetration condition), and are nonlinear in behavior or condition from a physical point of view. The type of contacts are general contact (considers the impact and impact friction between two objects in analysis), rough contact (does not consider sliding), welded contact (two objects are welded from the start of analysis) and sliding contact (only considers the sliding in the tangent direction). In the example below, general contact and rough contact are assigned depending on the position of two objects at the start of analysis and can be seen as linear. FEA NX supports the welded and general contact feature.

►Concept of General contact and Rough contact



►Concept of Welded contact





1.1

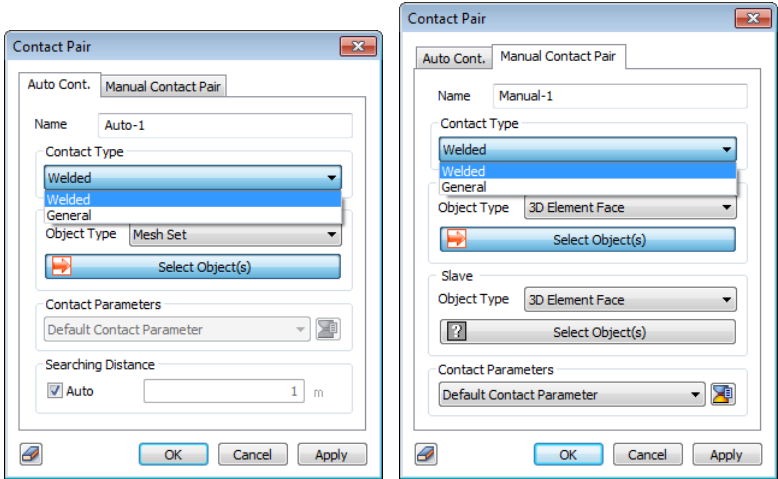
Define Contact

Overview

Use welded contact elements where element faces meet, but the nodes are not shared, to induce the same behavior. It can be used as the initial contact conditions between adjacent objects in structural analysis, consolidation analysis or seepage analysis. It is often used when node sharing on very complex geometry needs to be ignored to create an element. This function prevents analysis error and checks the analysis results that are similar to node sharing.

General contact considers the impact and impact friction between two objects in analysis, otherwise two objects are bonded like rigid link each other by welded contact. General contact can be used in nonlinear (static, dynamic) and fully coupled analysis. With Geometric Nonlinearity option, solver will take into account all possible contact area automatically regardless of defined Contact tolerance between two objects. It is also possible to consider Frictional behavior by Friction coefficient between two objects and the penetration at initial stage can be ignored by adjusting slave nodes automatically.

- Define auto contact
- Create manual contact pair



Methodology

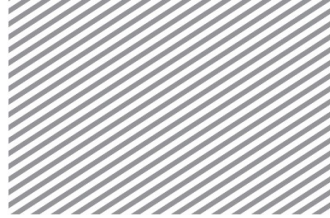
The contact can be defined through the Automatic Contact and the Create Manual Contact Pair functions.

Auto contact

This function automatically searches for areas where the selected meshes meet without node sharing and creates a contact surface.
[Searching Distance] : Input the distance between the main contact surface and the sub contact surface. The function searches for contact surfaces within this range.

Manual contact pair

The user can directly specify the main contact surface and the sub contact surface to create a contact surface. Face, 2D element, 3D element, 2D element free face, and 3D element free face geometries can be selected.
When creating the contact surface manually, the node-to-surface contact or surface-to surface-contact can be selected to create the surface. Node-to-surface contact takes less time, but the solution accuracy is relatively low because the nodes of the main object tend to penetrate through the sub object. On the other hand, surface-to surface-contact takes longer but the non-penetrating conditions are satisfied relatively accurately, allowing for more accurate simulations of structural behavior.



[Contact Parameters] : Input the coefficient value to calculate the initial contact search distance. The initial contact search distance can be found by multiplying the coefficient value by the longest length on the element face. If the main and sub contact faces are within this distance, contact is considered to have occurred.

Contact Parameters

Define the normal and tangential stiffness of contact element. Based on the material properties of adjacent elements, the stiffness will be updated automatically according to the generated strain. Highly recommend to use Default setting for Scaling Factor.

Following is the summary of parameters for the Contact Elements.

Parameter	Reference value (kN, m)
Contact stiffness (Recommend to use default setting)	
Normal stiffness scaling	1 (The smaller value, the larger penetration)
Tangential stiffness scaling	0.1 (Normal stiffness / 10)
Advanced options (parameters)	
Contact Tolerance	Auto (Uncheck) : Find contact area within the tolerance
Friction Coefficient (Optional)	0.3 ~ 0.6 (Depending on material types)
Conduction for Seepage Flow	Impermeable (Uncheck) : Possible to allow seepage flow through the contact elements
Thermal Conductance	Property used to describe ability to conduct heat between two bodies in contact.



Section 2 Construction Stage

Most numerical analysis for ground over the entire construction process is done through construction stage analysis. Ground analysis is normally material non-linear analysis and the material non-linear properties can be obtained from the initial conditions within the ground. Here, the initial conditions are the in-situ conditions of the site before construction.

After the initial stresses have been obtained from the initial conditions, the excavation loading and Shear strength from applying material properties can also be obtained. Hence, construction stage analysis includes the sequential construction process, starting from the initial ground conditions. Because on-site construction stages are very complex and subject to change, the analysis simplifies the process and focuses on the important construction stages.

For example, the construction stages for a tunnel are as follows.

- 1 Stage: Initial ground stress
- 2 Stage: 1st face excavation
- 3 Stage: 1st reinforcement + 2nd face excavation
- 4 Stage: 2nd reinforcement + 3rd face excavation
- 5 Stage: 3rd reinforcement + 4th face excavation
- (Repeat)

FEA NX does not create an independent analysis model for each construction stage. Rather, it uses a cumulative model concept where only the structural or loading changes are input for each construction stage and the analysis results are accumulated from the previous stage analysis results. Therefore in construction stage analysis, the structural changes and loading history from the previous stage affects the next stage analysis results. For example, when applying a load in an arbitrary construction stage, the loading will continue to exist in future construction stages until it is removed.

Instead of generating all the needed elements for an arbitrary construction step, only the elements needed for that construction stage are generated.

The definition of the construction stage is done by dragging & dropping the mesh set, boundary set, load set and contact set into the activated data column or deactivated data column. FEA NX provides the [Stage Definition Wizard], [Define Construction Stage] and [Simulate Construction Stage] functions to ease the construction stage setup process.



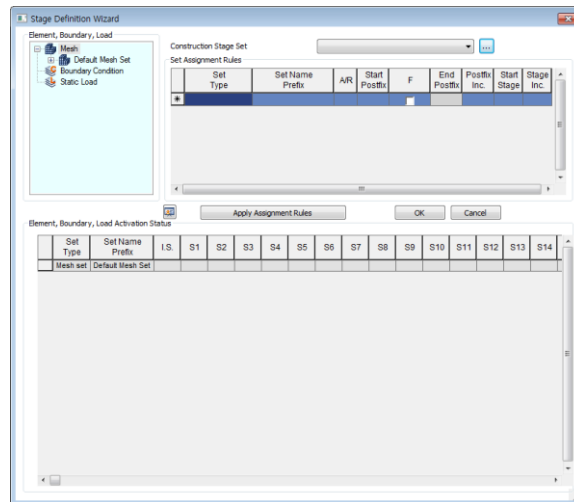
2.1 Stage Definition Wizard

Overview

A wizard to efficiently define the construction stages. A regular number (postfix) needs to be assigned to each set to define the construction stage using wizard. This number can be assigned using the [Rename] function for mesh sets. Sets that are only used once in the entire construction stage process does not need to be assigned a number.

It can be used in construction stage-using analysis (Static/Slope analysis, Seepage/Consolidation analysis).

►Stage definition wizard



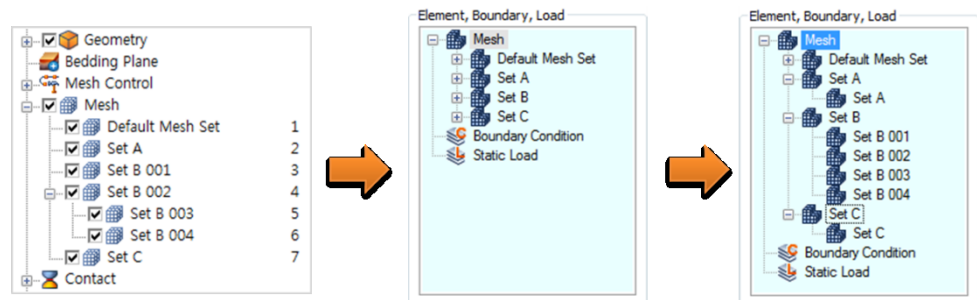
Methodology

Display the usable Mesh sets, Boundary sets and Load sets. Select the desired data and drag it to the Set assignment rule, or drag the Mesh set, Boundary set and Load set to the activation state.

Be aware that the display format of each set is completely different from the worktree. Here, all mesh sets are displayed as individual mesh sets, ignoring the relationship between mesh sets and sub-mesh sets in the worktree. Also, the upper most display name is the mesh set name without the postfix. Expanding one step displays the mesh set name with the postfix.

Refer to the following example.

►Element, Boundary,
Load display format





Set Assignment Rule

Specify the construction stage set used to define the construction stage and specify the assignment rule.

Set Assignment Rules									
	Set Type	Set Name Prefix	A/R	Start Postfix	F	End Postfix	Postfix Inc.	Start Stage	Stage Inc.
*	▼				<input type="checkbox"/>				

[Set Type]

The type of set on which the construction stage will be defined. The user can select between Mesh set, Boundary set or Load set.

[Set Name Prefix]

Specify the name without the serial number of the set on which the construction stage will be defined. For example, if a Mesh set is specified from 'Final Core #001' to 'Final Core #012', select the Set name prefix as "Final Core #".

[A/R]

Select whether to Add or Remove a selected set. A is displayed in green and R is displayed in yellow.

[Start Postfix]

Specify the set number of the selected set that is used first in the construction stage.

For example, if a Mesh set specified from 'Final Core #001' to 'Final Core #012' is selected and the Final Core #001' is removed at the second construction stage with the other Cores removed in sequence after each step, the first used set number is 001 and so the postfix can be set as '1'.

[F]

Check this option on when the selected set is not used until the final number. The user can input the end suffix when checked. If it is not checked, the set is used sequentially until the final number.

[End Postfix]

Specify the set number of the selected set that is used last in the construction stage.

For example, if a Mesh set specified from 'Final Core #001' to 'Final Core #012' is selected and the Final Core #001' is removed at the second construction stage with sequential removal until the 'Final Core #006', check F and input the postfix 6.

[Postfix Increment]

Input the postfix number increment used as the construction stage progresses.

For a Mesh set specified from 'Final Core #001' to 'Final Core #012', if the mesh is removed in order of 'Final Core #001', 'Final Core #003' and 'Final Core #005' for each construction stage, the postfix increment is 2 and hence, input a postfix spacing of 2.

[Start Stage]

Input the stage number first used in the construction stage.

For example, if a Mesh set specified from 'Final Core #001' to 'Final Core #012' is selected and the Final Core #001' is removed at the second construction stage with the other Cores removed in sequence after each step, the second stage is first used and so the start stage is set as 2.

[Stage Increment]



Select the stage increment for a selected set that is used every few stages.



For a Mesh set specified from 'Final Core #001' to 'Final Core #012', if the 'Final Core #001' is removed at the second construction stage and the Final Core #002' is removed at the fourth construction stage, the set is used every 2 stages and so the stage spacing is 2.

[Apply Assignment Rule]

Press this button to display the specified data according to the assignment rule on the Mesh, Boundary, Load set activated state. Press OK to create the construction stage.

The activated Mesh, Boundary, and Load set can be checked for each stage using the Preview construction stage() option. This function has the same function as Construction stage simulation ().

Mesh, Boundary, Load Activation Status

Display the currently specified construction stage on a table.

The construction stage progresses as it moves to the right. The I.S. and S1 on the top of the columns are abbreviations for initial stage and Stage1, respectively. The added data is displayed in green and the removed data is displayed in orange. Data defined by postfixes are expressed in each construction stage as numbers. Data that does not use a prefix (eg. Ground) are expressed as a line. If the mesh, boundary condition and load data are dragged onto the menu [Mesh, Boundary, Load Activation Status], the delete setting cannot be conducted and only additional settings can be conducted.

►Mesh, Boundary, Load display format

Element, Boundary, Load Activation Status		I.S.	S1	S2	S3	S4	S5	S6	S7	S8	S9	S10	S11	S12	S13	S14	S15	S16	S17
Boundary Set	Boundary Set-																		
Mesh set	Default Mesh Set																		
Mesh set	Evacuation R/B																		
Mesh set	Evacuation S/C																		
Mesh set	Evacuation Tunnel																		
Load Set	Load Set-																		
Mesh set	Main R/B																		
Mesh set	Main S/C																		
Mesh set	Main Tunnel-	A: 1to30	R: 1	R: 2	R: 3	R: 4	R: 5	R: 6	R: 7	R: 8	R: 9	R: 10	R: 11	R: 12	R: 13	R: 14	R: 15	R: 16	R: 17

Click the [Apply Assignment Rule] button to display the construction stage on the Mesh, Boundary, Load activation status. To delete the displayed construction stage, select that data cell and press the Delete key on the keyboard.

This method deletes the created construction stage data, but not the construction stage. Hence, a construction stage with no content is created. Select the whole target column and press the Ctrl and Delete key together to delete the construction stage for the whole column.

The stage where all the mesh sets are activated initially (in-situ state) is when all the mesh sets in the I.S. (initial stage) are activated. This stage can be specified by entering 0 for start stage (the 0 stage is the initial stage) and 0 for stage spacing (the stage increment number is 0 and thus all elements are activated in one stage).

The advanced options (LDF etc.) used in the construction stage can be set in the Define Construction Stage menu. Hence for complex models, it is convenient to use the Construction Stage Wizard to create the framework of the overall construction stages. It is also convenient to use the [Define Construction Stage] menu to specify the individual options used in each construction stage.

Example

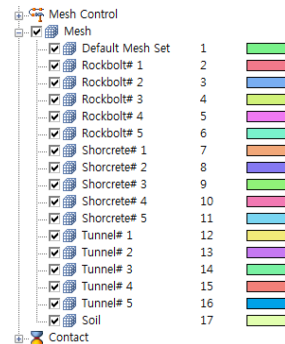
Let us examine a simple example of Construction stage definition.

The construction stage is defined for a tunnel modeled on a homogeneous ground. The entire tunnel shape is excavated at once and the rock bolts and shotcrete are created in the following stages. The excavation is



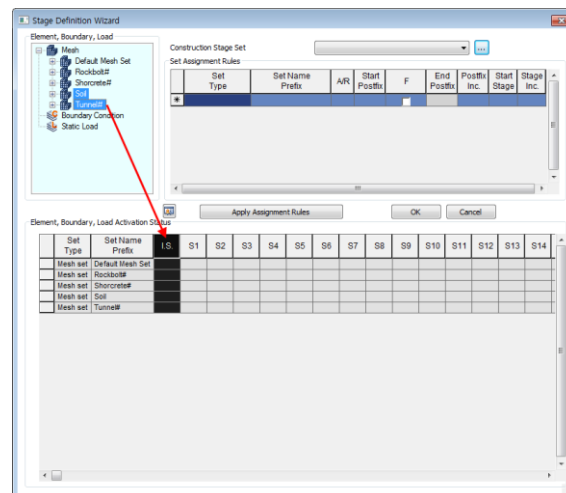
done in 5 stages. The excavation is assumed to start from the smaller postfix number and progresses to the larger postfix.

The mesh set is created as follows.



Run the Stage definition wizard.

'Soil' and 'Tunnel#' need to be included in the in-situ state. Select using the Ctrl key and drag & drop into the I.S. column of the Mesh, Boundary, Load activation status.



Start the tunnel excavation.

In this example, the exaction is done stage by stage, starting from the first stage as follows: 1st face tunnel excavation -> 1st face rock bolt/shotcrete installment -> 2nd tunnel excavation -> 2nd face rock bolt/shotcrete installment ->

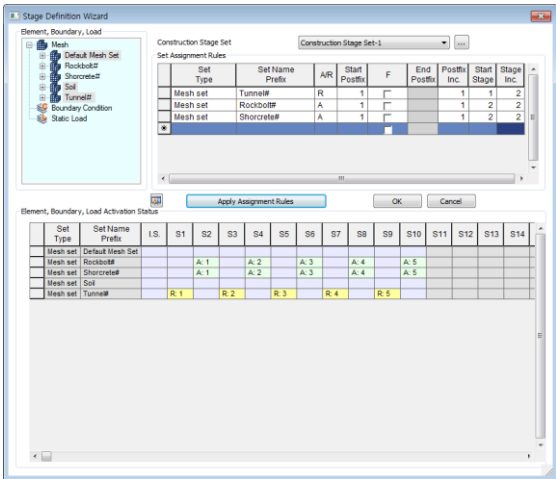
Select R to select and delete the 'Tunnel#' of the element type, the Start postfix as 1 and do not check F to process until the last number. The Start stage is 1 and the input is 2 for the Stage spacing to allow for the installation of rock bolts and shotcrete between excavation stages. Afterwards, click the [Apply Assignment Rules] button to specify the construction stage of the Mesh, Boundary, Load activation status.

The shotcrete and rock bolts are also specified on the construction stage.

Input 2 for the Start stage and set as A to create both elements in the second construction stage. Also, input Start postfix 1, do not check F and input Postfix spacing to use all the numbers from 1 to the final number. Finally, input 2 for the Start stage and Stage spacing to create the first elements at the second construction stage with two stage spacing between element creation.



Press the [Assign Assignment Rules] button to automatically create the Mesh, Boundary, Load activation status and click the [OK] button to create the construction stage.

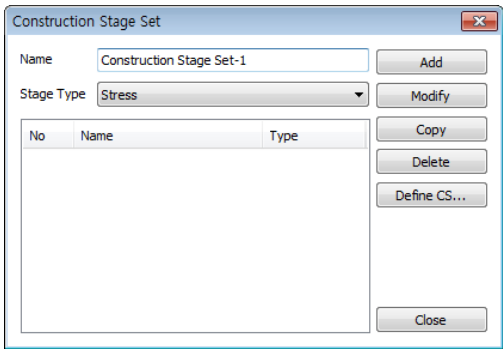


2.2 Construction Stage Set

►Construction stage set

Overview

Define the Construction stage set for analysis.

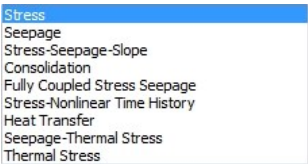


Used for analyses that incorporate construction stages (Static/Slope analysis, Seepage/Consolidation analysis).

Methodology

Define the Construction Stage Set and then define the Construction Stage.
A single file can be composed of multiple Construction stage sets.
The construction stage types are [Stress], [Seepage], [Stress-Seepage-Slope], [Consolidation], [Fully Coupled stress], [Heat Transfer], [Seepage-Thermal Stress], [Thermal Stress].

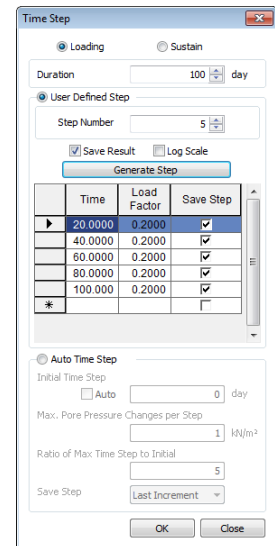
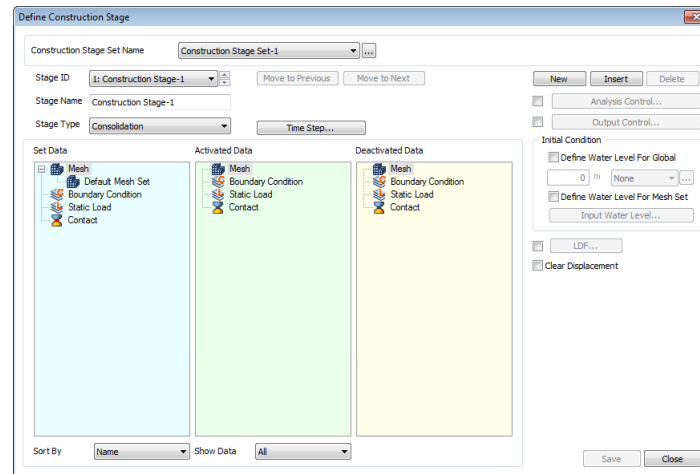
►Analysis methods supported for construction stage formation






Click the Define Construction Stage button to form the construction stage. Advanced options that are not available on the [Stage Definition Wizard] can be set.

► Define construction stage



Stage name

Define the construction stage name. Use [New] to create a new construction stage and use [Insert] to add a new construction stage in between existing stages.

For example, clicking the Insert button at Stage 2 moves the current stage to Stage 3, and the new stage becomes Stage2. Click the  button to move to the previous or next stage.

Stage type

Specify the construction stage type. Be aware that the designated [Analysis Control], [Output Control] options are different and the boundary conditions/loading conditions for each stage type are different.

Refer to the Analysis > Analysis case > General > Analysis/Output Control for more information on control options.

Move to Previous/Next

The construction stage order may need modification when many construction stages are created. Use the Move to Previous or Next button to change the order of created construction stages.

Time Step

Define time steps used in the analysis.

[Duration]

Insert the duration to be analyzed. 'User Defined Step' generates steps by dividing with Step Number. 'Auto Time Step' automatically divides defined period with time step.

[Auto Time Step]

It will automatically choose appropriate time steps for a consolidation or fully coupled stress seepage analysis.



When the calculation runs smoothly, resulting in very few iterations per step, then the program will choose a larger time step. When the calculation uses many iterations due to an increasing amount of plasticity, then the program will take smaller time steps.

This function reduces the pore water pressure result errors when loading is applied in short period of time.

[Initial Time Step]

Initial Time Step can be either manually defined by user or calculated automatically within solver. The automatic calculation formula is as follows:

$$\Delta t_{initial} = \frac{\gamma_w \Delta h^2}{\alpha k} \left(\frac{1}{K_{bulk}} + A \right), \quad A = n \left(\frac{s}{K_{water}} - \frac{\partial s}{\partial p} \right)$$

Where, γ_w : Unit weight of water

Δh : Length of element

α : Shape factor (1/3)

k : Permeability

K_{bulk} : Bulk modulus

n : Porosity

s : Degree of saturation

K_{water} : Bulk modulus of water

p : Pore water pressure

[Max. Pore Pressure Changes per Step]

Input Max. Pore Pressure Changes per step. When pore pressure changes exceeds the maximum value, step size is automatically reduced and analyzed.

[Ratio of Max Time Step to Initial]

Input the maximum value of time step ratio compared to 'Initial Critical Time Step'.

[Max. Temperature Changes per Step]

Input Max. Temperature Changes per step. When temperature changes exceeds the maximum value, step size is automatically reduced and analyzed.

[Save Step]

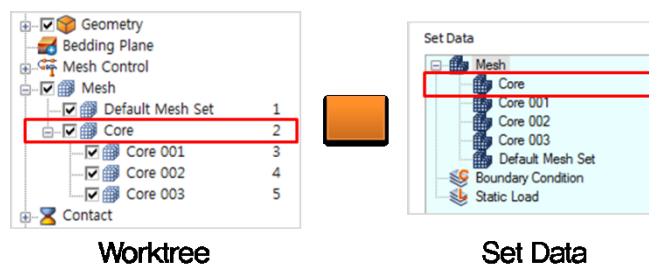
Select the output method of results. 'Last Increment': Only output results from last step, 'Every Increment': Output results from all steps.



Set Data

Display the usable Mesh sets, Boundary sets, and Load sets in a worktree. Be aware that the sub-sets are also displayed independently, so take caution when selecting the mesh sets.

For example, the set data for the created Core mesh set with registered mesh sub-sets (Core 001, Core 002, Core 003) are shown in the right figure. In this case, activating Core does not activate the mesh sub-sets Core 001, Core 002, Core 003. Hence, mesh sets that are not registered directly on the set data are useless.




Activated Data

Register the activated sets for each construction stage. The activated sets remain active for future construction stages without needing re-activation until it is deactivated. The sets that need to be activated for the construction stage can be selected using the left mouse button and dragged & dropped into the activated data. Another method is to select the sets using the right mouse button on the Set data and select activate on the Context menu.

Deactivated data

Register the deactivated set for each construction stage. The deactivated sets remain active for future construction stages until they are re-activated. The sets that need to be deactivated for the construction stage can be selected using the left mouse button and dragged & dropped into [Deactivated data]. Another method is to select the sets using the right mouse button on [Set data] and select deactivate on the Context menu.

Define Water Level For Global

Input the groundwater level that changes according to the construction stage with respect to the GCS. Click  to set the ground water level function. If the water level and function are both specified, the input water level is multiplied onto the function and applied on the analysis.

Define Water Level for Mesh Set

Define the groundwater level that changes according to the construction stage for each mesh set.

If the groundwater layer is surrounded by rocks or an impermeable clay layer (confined aquifer), the presence/absence of the groundwater level for each ground layer can be set for analysis.

If the total groundwater level is input and a mesh set has a defined groundwater level, the mesh set groundwater level has priority and the total groundwater level is applied to mesh sets that do not have a defined level.

If the water level and function are both specified, the input water level is multiplied onto the function and applied on the analysis.



LDF

Set the Load Distribution Factor. The sum of all distribution factors need to be 1, and the keyboard Enter key needs to be pressed after the input to apply the value properly.

For the example case shown below, a LDF of 0.4 is applied to the current stage and a LDF of 0.3 is applied to the next stage and the subsequent stage. Here, the LDF does not need to be checked for the latter two stages and the LDFs need to be set such that they do not overlap in the construction stages.

	After Current Stage	Distribution Factor
1	0	0.4000
2	1	0.3000
3	2	0.3000
4	3	0.3000

Total: 1

Buttons: Add, Modify..., Delete, OK, Close

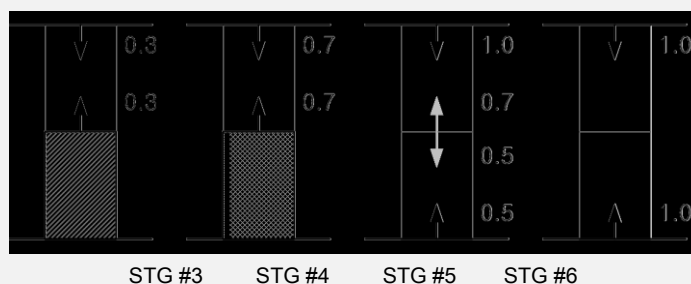
Tip

The application method of the Load Distribution Factor is as follows:

The LDF keeps track of the Internal Forces of a deleted element and loads it in stages according to the factor assigned for each construction stage, rather than loading it at once.

When applying the LDF on the top and bottom simultaneously, the factors need to be set such that they do not overlap in the construction stages. If the LDFs overlap as shown below, when calculating the internal forces of this stage, the 0.5 on the bottom element released in STG #5 releases the stress using the internal force of the bottom element, created by the 0.7 on the top element defined in STG #4. Hence, the 1.0 on the top element of STG #4 is not released and so, the internal force for the 0.3 on top is not taken into account for analysis.

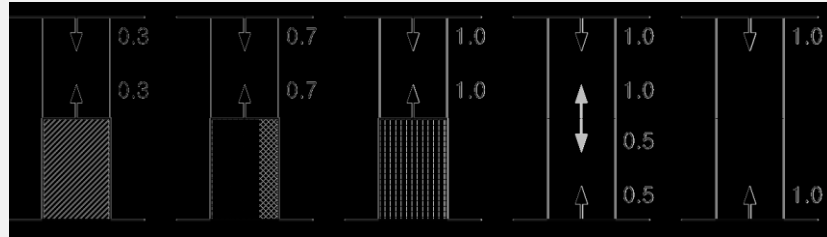
Part	STG #3	STG #4	STG #5	STG #6
Top	0.3	0.4	0.3	-
Bottom	-	-	0.5	0.5



Hence, be careful not to overlap the LDFs in the construction stage, as shown below.



Part	STG #3	STG #4	STG #5	STG #6	STG #7
Top	0.3	0.4	0.3	-	-
Bottom	-	-	-	0.5	0.5



Clear Displacement

Set the displacement of an analysis result in the current stage as 0. It is used to set the initial conditions of the in-situ state. The stress is not reset to 0.

Slope Stability (SRM/SAM)

Decide whether to conduct the slope stability analysis (SRM) in the current construction stage. If this option is checked, it is automatically registered as an analysis case and analysis is conducted. In other words, the ground stress from the non-linear analysis results in the previous stage is coupled and slope stability analysis is conducted. (However for SAM analysis, it is only applicable for 2D analysis and the boundary conditions of the virtual slip surface needs to be set.)

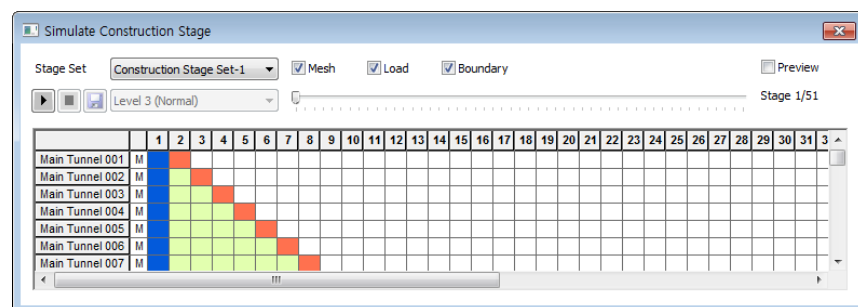
Overview

Check the defined construction stage as a video.


It can be used in analyses that incorporate construction stages (Static/Slope analysis, Seepage/Consolidation analysis).

2.3 Simulate Construction Stage

► Simulate Construction Stage



Methodology

Specify the defined construction stage set and click the  button to play the construction stage video. The video is created by capturing the whole work screen. Unwanted frames can be inserted if a different dialog box is open above the model.

Check the [Mesh], [Load] and [Boundary] to check the activated/deactivated data of the construction stage.



2.4

Auto Set

Overview

Automatically generate construction stages using the mesh, boundary condition, load that are viewed on the current model screen.

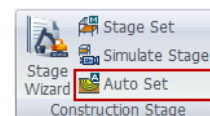
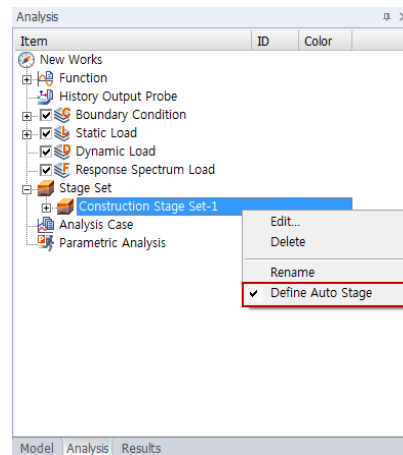
It can be used in construction stage-using analysis (Static/Slope analysis, Seepage/Consolidation analysis). The data used for each stage can be checked on the model, allowing intuitive composition of stages.

Methodology

The construction stage auto set function can be activated by using the following steps:

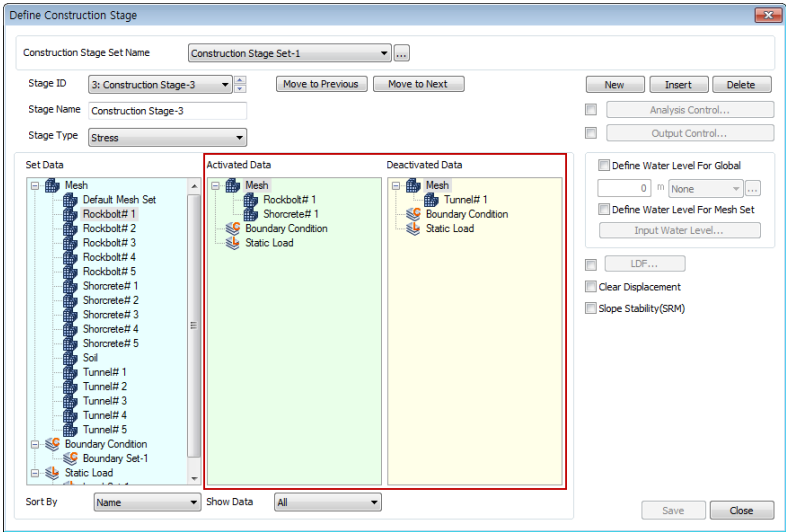
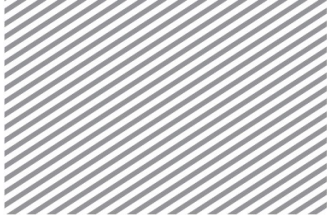
1. Adding the construction stage set after finishing modeling registers the added construction stage set on the worktree, as shown below.
2. Check the [Define Auto Stage] option by right mouse clicking on the registered construction stage set. This option activates the Construction stage > Auto set on the ribbon menu that allows for the specification of construction stages on the selected construction stage set.

►Define auto stage



3. Click the [Auto Set] icon to automatically register the displayed meshes, boundary conditions, load conditions on the activated data column and the un-displayed (not checked) meshes, boundary conditions, load conditions on the deactivated data column. In other words, set the show/hide model information for each composing stage and select [Auto Set] to automatically set the construction stage with reference to the currently shown information. The created stages can be checked on the worktree.

Because the concept of construction stages is cumulative, the program compares the activated/deactivated model information in the previous stage and only adds/deletes the changed information. Hence, it is recommended that the creation be done in stages, after the displayed model information is returned to its initial in-situ state.



Tip

The auto set function generates construction stages using the work environment displayed on the screen. Hence, to specify the individual options (LDF setting, clear displacement etc.) for each stage, use the Construction stage set specification menu to check the options for each stage.

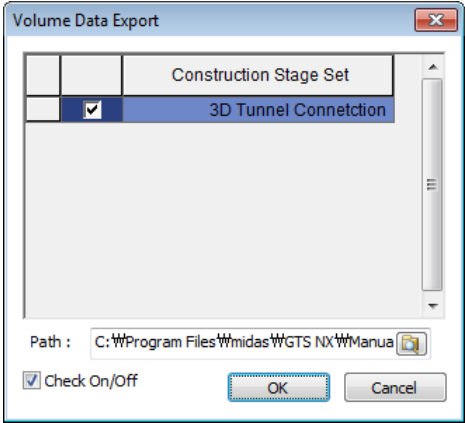
2.5 Volume Data Export

Overview

The volume data of 1D/2D/3D elements defined to the construction stage is exported to excel file. This shows the length/area/volume of activated/deactivated set for stages. This doesn't apply for the other element types (point spring, matrix spring, free field, interface, shell interface, pile tip, elastic link, rigid link, user supplied behavior for shell interface, mass).

Methodology

► Volume Data Export





	A	B	C	D	E	F	G	H	I	J	K	L
1												
2		Stage Name	I.S.		S1		S2		S3		S4	
3			Activated Data	Deactivated Data	Activated Data	Deactivated Data	Activated Data	Deactivated Data	Activated Data	Deactivated Data	Activated Data	Deactivated Data
4		Mesh Set Name	(m³)	(m³)	(m³)	(m³)	(m³)	(m³)	(m³)	(m³)	(m³)	(m³)
5		Main tunnel#-003	257.693	-	-	-	-	-	-	-	-	-
6		Main tunnel#-005	257.693	-	-	-	-	-	-	-	-	-
7		Main tunnel#-002	257.839	-	-	-	-	-	-	-	-	-
8		Main tunnel#-012	257.749	-	-	-	-	-	-	-	-	-
9		Main tunnel#-014	257.693	-	-	-	-	-	-	-	-	-
10		Main tunnel#-001	256.399	-	-	-	-	-	-	-	-	-
11		Main tunnel#-015	257.406	-	-	-	-	-	-	-	-	-
12		Main tunnel#-013	257.693	-	-	-	-	-	-	-	-	-
13		Main tunnel#-010	257.749	-	-	-	-	-	-	-	-	-
14		Main tunnel#-011	257.693	-	-	-	-	-	-	-	-	-
15		Main tunnel#-006	257.749	-	-	-	-	-	-	-	-	-
16		Main tunnel#-004	257.749	-	-	-	-	-	-	-	-	-
17		Main tunnel#-007	257.693	-	-	-	-	-	-	-	-	-
18		Main tunnel#-008	257.749	-	-	-	-	-	-	-	-	-
19		Main tunnel#-009	257.693	-	-	-	-	-	-	-	-	-
20		Passageway#-001	85.164	-	-	-	-	-	-	-	-	-
21		Passageway#-003	89.86	-	-	-	-	-	-	-	-	-
22		Passageway#-002	71.073	-	-	-	-	-	-	-	-	-
23		Vertical shaft#-001	183.824	-	-	183.824	-	-	-	-	-	-
24		Vertical shaft#-003	229.61	-	-	-	-	-	-	229.61	-	-
25		Vertical shaft#-006	228.693	-	-	-	-	-	-	-	-	-
26		Vertical shaft#-004	229.61	-	-	-	-	-	-	-	-	-
27		Vertical shaft#-005	267.701	-	-	-	-	-	-	-	-	-
28		Vertical shaft#-002	191.342	-	-	-	-	191.342	-	-	-	-
29		Weathering soil	21442.444	-	-	-	-	-	-	-	-	-
30		Weathered rock	49065.679	-	-	-	-	-	-	-	-	-



Section 3 Boundary Condition

The boundary conditions on FEA NX can be divided into the following three categories:

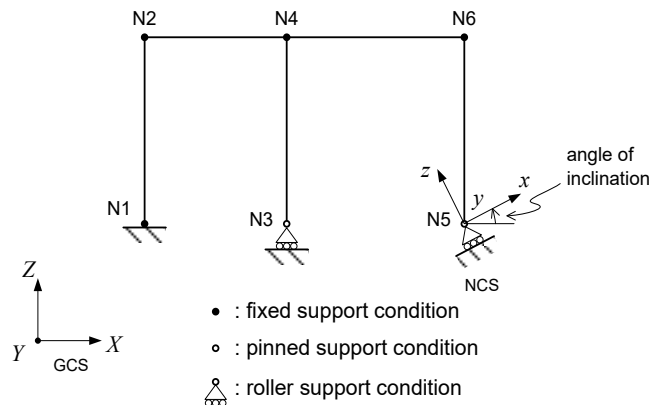
- Degree of freedom (DOF) constraint for displacement for Stress analysis
- Seepage boundary for Seepage analysis
- Boundary element for Equivalent linear analysis

The DOF constraint for the displacement function is used to constrain the displacement of an arbitrary node or the DOF component when merging elements (Beam element with Plane stress element, Plate element with Solid element etc.) with different DOF's for each node.

The DOF constraint for an arbitrary node is input as 6, with reference to the GCS.
If the node coordinate system exists, the node coordinate system becomes the reference.

For example, the method of assigning the DOF constraint conditions of a planar frame model below is as follows. The movement of this 2D model is only allowed in the GCS X-Z plane and hence, the displacement DOF in the GCS Y direction and the rotational DOF in the X and Z direction needs to be constrained for all nodes.

►Planar frame model that considers the DOF constraint condition



Also, for the N1 node with the fixed support condition, the displacement DOF in the GCS X,Z direction and the rotational DOF in the Y direction is additionally constrained using the [Constraint] function. For the pin supported N3 node with the roller support condition, the displacement DOF in the Z direction is constrained.

For the roller supported N5 node, the node coordinate system is rotated by the angle of inclination with reference to the GCS X axis, and then the displacement DOF in the Z direction is constrained with reference to the node coordinate system.

This DOF constraint for node displacement is often used for support conditions where the displacement can be ignored, and assigning the constraint condition on an arbitrary node creates reaction forces on that node. The reaction forces are output with reference to the GCS and if a node coordinate system is assigned, the forces can be output with reference to the node coordinate system.

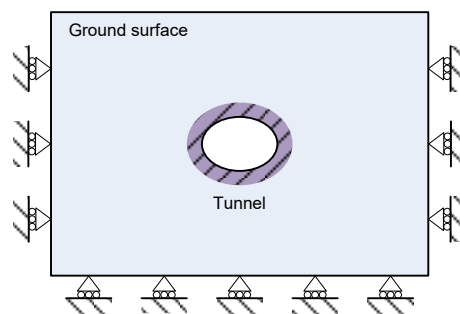
The method of assigning the DOF constraint conditions of a planar ground model below is as follows.



Because the ground model restricts the normally semi-infinite analysis area to the tunnel surroundings, the analysis boundary is defined at a position where there is nearly no change in stress or displacement due to the tunnel construction. The nodes at the left and right boundary are constrained for the X DOF and the nodes at the bottom boundary are constrained for the Z DOF. An additional X DOF constraint can be assigned to the bottom boundary if the horizontal displacement at the bottom is not symmetric.

If the top boundary is in contact with the atmosphere like the ground surface, no degrees of freedom are constrained. However, if the top ground is omitted from the analysis and the effects of gravity are ignored when defining the interior initial stress of the model evenly for analysis efficiency, such as for deep road tunnels, the Z direction DOF for the top boundary is also constrained.

► Planar ground model that considers the DOF constraint condition

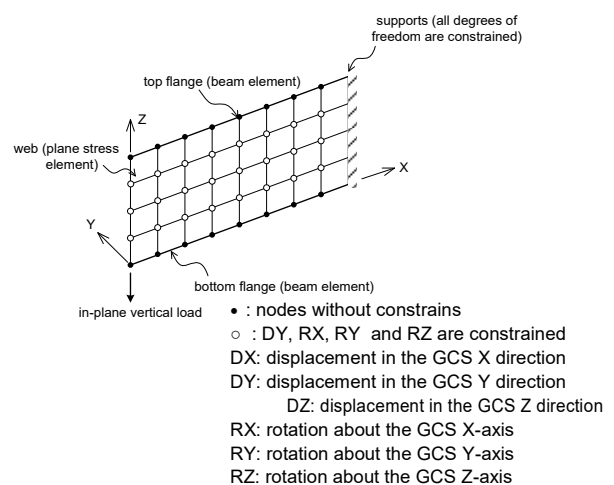
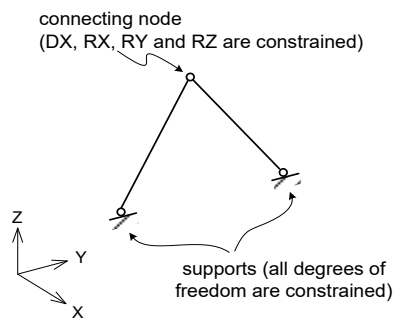


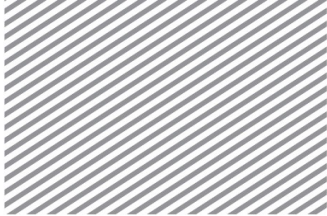
The analysis boundary surface constraint conditions of a 3D ground model also follows the 2D model method, where the DOF is constrained in the direction perpendicular to the boundary surface.

The figure below shows examples of 3D degrees of freedom constraint conditions. For a truss connection, only the axial direction of the truss element has a DOF for displacement and so, the X direction displacement and all rotation direction displacement components at the connection node are constrained. A beam element with replaced top, bottom flanges has 6 degrees of freedom for each node and so, nodes that are connected to the beam element do not need separate constraint conditions. For the nodes where the plane stress elements meet, the plane stress element only has a DOF for in-plane behavior and so, the degrees of freedom for the out-plane Y direction displacement and all rotational degrees of freedom need to be constrained.

► When truss elements are joined together

►► When the top, bottom flanges of a H shape cantilever beam is modeled as a beam element and the web is modeled as a plane stress element



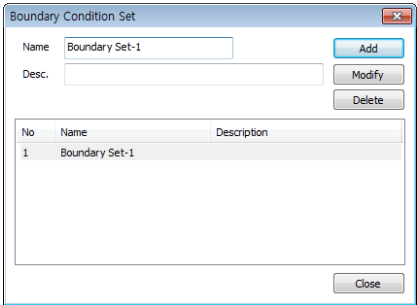


3.1 Define Set

Overview

Define the boundary condition set.

►Define set

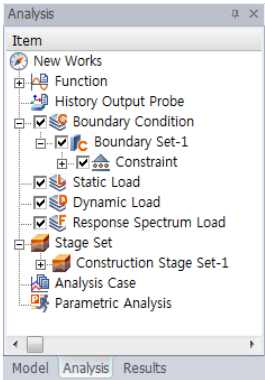


Methodology

Input the name and specification and click [Add] to define the boundary condition set. The boundary condition set can be input beforehand, and the name of each boundary condition can be entered when it is generated.

The registered boundary condition set is automatically registered under Workstree > Analysis > Boundary Condition and the checkbox is used to show or hide the set.

►Workstree – Boundary



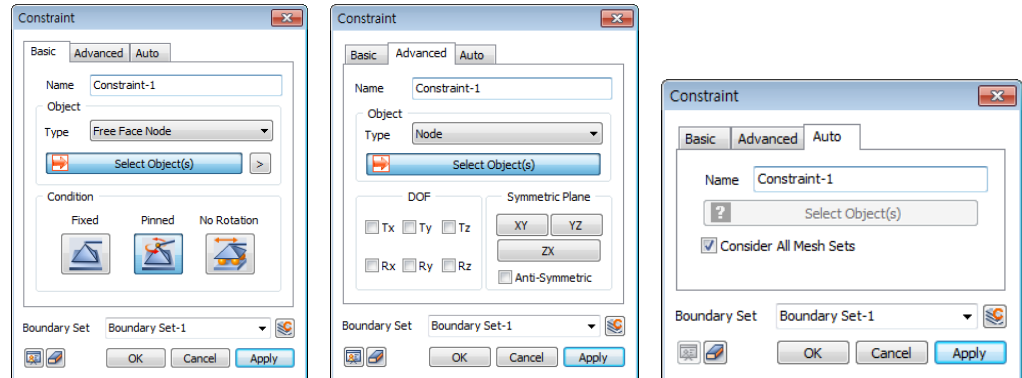


3.2 Constraint

- Basic constraint
- Advanced constraint
- Auto constraint

Overview

Set the constraint conditions of a model.



Methodology

The methods for setting a constraint condition of a model are [Basic], [Advanced] and [Auto].

[Basic]

Select the target and assign a [Fixed], [Pinned] or [No Rotation] that fits the behavior of the analysis model.

[Advanced]

The 6 degrees of freedom of a node can be fully or partially constrained.

Tx, Ty, Tz are the displacement constraints in the x, y, z direction and Rx, Ry, Rz are the rotational constraints in the x, y, z direction.

The constraint conditions can be input for a desired boundary condition (Point, Edge, Face, Node, Free face node).

Point
Edge
Face
Node
Free Face Node

The constraint conditions are assigned to the element node and reflected in the analysis. Setting the constraint conditions on a point, edge, face etc. is a convenient method of selecting element nodes included in the selected geometry shape

[Auto]

Select the target mesh set to automatically create constraint conditions. The ground conditions for general stress analysis are set automatically. The x direction displacement is constrained for the left/right side, the y direction displacement is constrained for the front/back side and the x,y direction displacement is constrained for the bottom of a model.

Boundary set

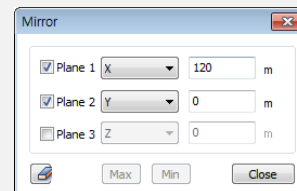
Register the set constraint conditions on the desired boundary set. The user can specify the name of the boundary set.

**Tip***** Symmetric and inverse symmetric constraints**

Boundary conditions can be largely divided into two conditions;

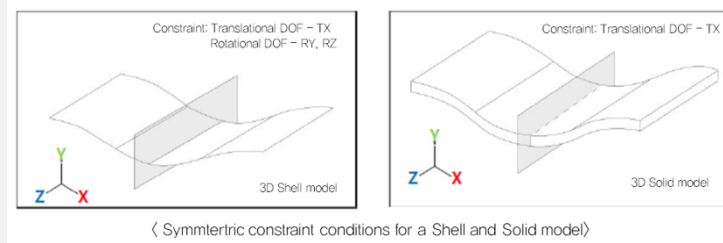
1. Constraint conditions of the analysis target are specified.
2. Symmetry of the structure is used to analyze the symmetric area only, not the entire model.

Applying symmetry is a very effective way to increase the convenience of modeling and decrease the analysis time. If the geometry of the structure and loading is symmetrical, a 1/2 model or 1/4 model can be used to decrease the number of elements and create an economic model that reduces analysis time. However, constraints exist when checking the deformed shape or stress distribution for the entire model because the analysis results from the symmetric model cannot be shown on the entire model. Here, use the View symmetric model function on the Additional view control toolset to expand the analysis results of the 1/2 or 1/4 model onto the entire model.

**How to apply symmetric boundary conditions**

If one or more of the geometry shapes, materials, loads or boundary conditions is symmetrical about a plane or axis, the symmetric boundary condition can be used. The View symmetric model function can output, expand a 1/2, 1/4, 1/8 symmetric model onto the entire model. To assign a symmetric boundary condition, a boundary condition needs to be set such that the structure does not invade the symmetry plane.

The figure below is an example of symmetric constraint conditions applied to a solid model. To apply the symmetric constraint conditions in the YZ plane, the Translation degree of freedom Tx is constrained. For the XY plane, the Tz needs to be constrained and for the ZX plane, the Ty needs to be constrained.



Because Solid elements do not have a rotational degree of freedom, only the displacement boundary condition of the symmetric model is constrained. However, the rotational degree of freedom needs to be constrained for a Shell model such that the symmetry plane is not invaded. In other words, Tx,Ry,Rz need to be constrained for the XY plane, Tz,Rx,Ry for the YZ plane and Ty,Rx,Rz for the ZX plane.

When applying a symmetric boundary condition, the loading size needs to be converted to fit the symmetric condition. Also, the symmetric condition cannot be applied to the model shape and buckling shape because of the asymmetric vibration mode or the possibility of asymmetric buckling.



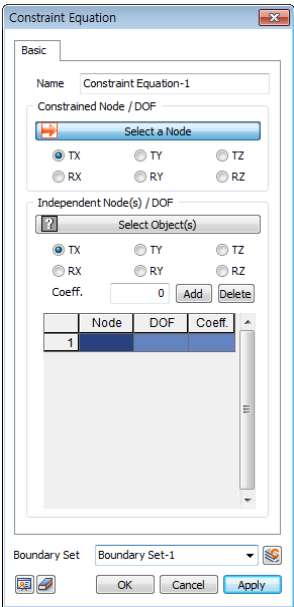
3.3

Constraint Equation

►Constraint equation

Overview

Constrain the degree of freedom such that a particular node is dependent on the behavior of another node.



Methodology

Constrain the behavior of one node to the behavior of another node. Define the main node that affects the deformation of another node and degree of freedom [Constrained Node/DOF]. TX, TY, TZ are the degrees of freedom for displacement and RX, RY, RZ are the degrees of freedom for rotation. [Independent Node/DOF] inputs the coefficient applied on the displacement of the independent node, to define the degree of freedom and interrelationship between the dependent node.

Boundary Set

Register the set constraint conditions on the desired boundary set. The user can specify the name of the boundary set.

Tip

The behavioral characteristics are similar to Mesh > Element > Create > Other > Rigid link. For example, when the main node moves by a certain distance, the dependent node can be constrained such that it moves by twice that length. When creating a complex interrelationship between two or more nodes, the constraint equation can be used.



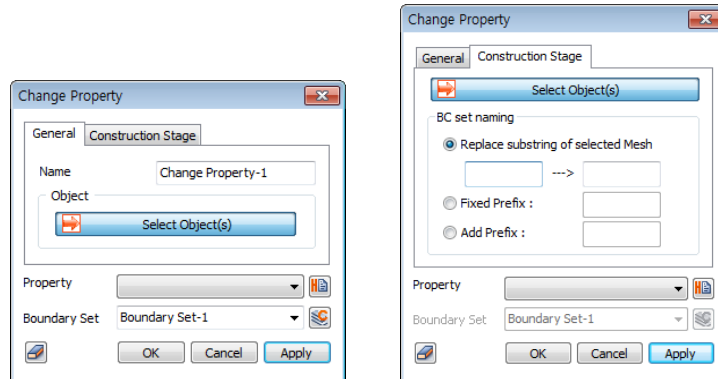
3.4 Change Property

Overview

Apply a new attribute data or substitute an existing attribute data for an element that changes with the construction stage.

It can be used in construction stage-using analysis (Static/Slope analysis, Seepage/Consolidation analysis).

- ▶ Change property–
General
- ▶▶ Change property
– Construction stage



General

Methodology

Select the target element and specify the element property subject to change.

Construction Stage

Methodology

If the construction stage is specified, change the element property collectively.

For example, apply a stage by stage change in the material property of shotcrete, from ductile to hardening, of a 3D model. If the mesh set is already specified by shotcrete 001 ~ shotcrete 010, the boundary conditions for element property change need to be specified separately for each element when using Change property > General. However, for Change property > Construction stage, selecting shotcrete 001 ~ shotcrete 010 can create 10 boundary condition sets with just one click.

The name of the boundary condition set can be specified by [Replace substring of selected Mesh], [Fixed Prefix] and [Add Prefix] to collectively change the name of the created mesh sets. If nothing is input, the boundary condition set is created with the same name as the mesh set.

[Replace substring of selected Mesh] : Create a boundary condition set by using a different postfix on the selected mesh set name.

[Fixed Prefix] : Create a boundary condition set by adding a different prefix, instead of using the selected mesh set name.

[Add Prefix] : Create a boundary condition set by adding a prefix to the selected mesh set name.

Boundary set

Register the set constraint conditions on the desired boundary set. The user can specify the name of the boundary set.

**Tip**

[Change Property] can be used when the element properties change with the construction stage. If the ground has a property A in Stage 1 and that property changes to B in Stage 2, which then changes to property C in Stage 3, the following two boundary conditions are created.

Boundary condition 1 : A -> B

Boundary condition 2 : A -> C

Using Define construction stage, activate Boundary condition 1 in Stage 2 and for Stage 3, activate Boundary condition 2 and deactivate Boundary condition 1.

As a result, an element property that changes by A->B->C can be defined.



3.5 Slip Circular Surface

Overview

Define an circular virtual slip surface.

The virtual slip surface is a boundary condition used in slope stability analysis. The virtual slip surface can only be set when the analysis setting is for 2D models.

►Slip circular surface

Methodology

Define the slip surface of the virtual fracture arc using the grid point of the arc center and radius where the tangent line to the arc is located.

Specify the rectangular grid area where the arc center can be located using 3 points (reference point x,y). The center count x, y is the number of divisions in the grid area. For example, entering a center count of 5,4 creates 5 x 4 = 20 arc center points.

The radius where the tangent line of the arc is located can be set using [Method using Tangent Line of Circle] or [Method using Length and Range of Radius].

[Method using Tangent Line of Circle]

Directly specify the rectangular area where the tangent line is located on the work screen using [Draw Range]. The number of radius divisions divides the rectangular area by that number and the tangent direction of the arc radius domain can be changed using [Change Tangent Direction].

[Method using Length and Range of Radius]

Directly input the length of initial circular radius, increment for circle radius and number of increments for circle radius.

Boundary Set

Register the set constraint conditions of the desired boundary set. The user can specify the name of the boundary set.



3.6

Slip Polygonal Surface

Overview

Define a non circular virtual slip surface.

The virtual slip surface is a boundary condition used in slope stability analysis. The virtual slip surface can only be set when the analysis setting is for 2D models.

►Slip polygonal surface

	X	Y
▶	-180.0000	120.0000
	-80.0000	20.0000
*	60.0000	-20.0000

Methodology

The non circular virtual slip surface can be defined by:

- Directly input the coordinate values of the non-circular slip surface onto a table
- Mouse click the non circular slip surface domain on the work-plane

When defining the virtual slip surface on the work-plane, use the right mouse click to stop the input.

Boundary Set

Register the set constraint conditions on the desired boundary set. The user can specify the name of the boundary set.

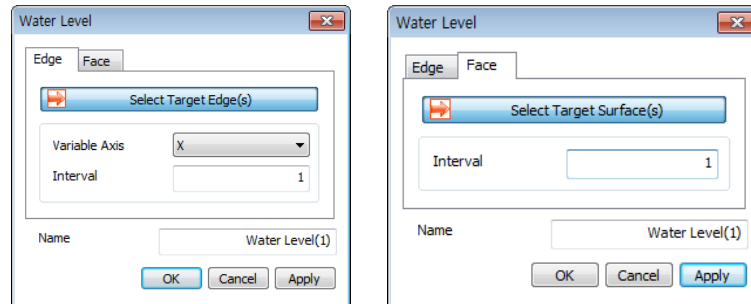


3.7 Water level

Overview

Create a changing groundwater level by selecting a geometry shape on the work screen.

- ▶Water level – edge
- ▶▶Water level – face



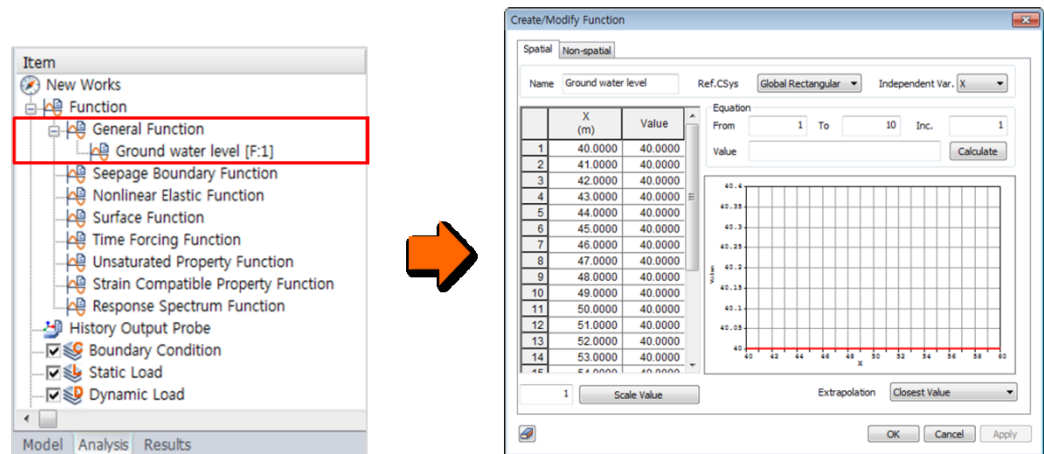
Edge

Methodology

Create a changing groundwater level by selecting edges.

Specify the axis direction of the changing variable. For example, if the groundwater level changes in the x direction of the model, select the variable axis as x. Then input the spacing value. For example, if the spacing is 1m, the groundwater level lines are created in 1m intervals.

The created water level is registered under Worktree > Analysis > Function > General function and can be edited as a table using right mouse click > Edit.



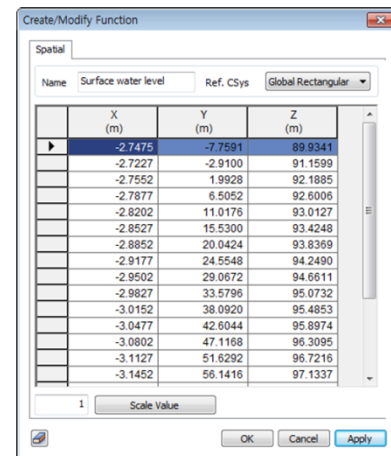
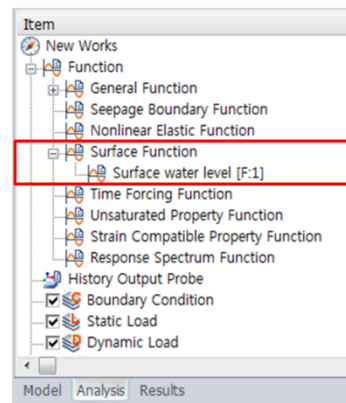


Face

Methodology

Select a face and input the spacing value to create a changing groundwater level.

The created water level is registered under Worktree > Analysis > Function > General function and can be edited as a table using right mouse click > Edit.





3.8

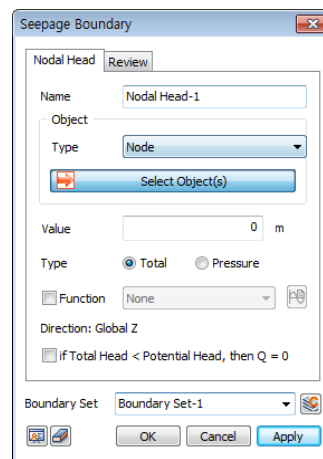
Nodal Head

Overview

Input the head of the model. Both the constant head value for steady state analysis and the changing head value for Transient analysis can be entered by applying the Seepage boundary condition function.

The Nodal head is used as a boundary condition for Seepage/Consolidation analysis (Fully-Coupled).

►Nodal head

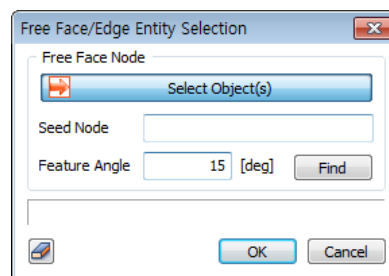


Methodology

Directly input the head value of a specific point. The target can be selected from [Node], [Edge], [Face], [Free Face Node].

For [Node], the node is directly selected to define the head condition. Selecting [Edge] or [Face] defines the head condition at all nodes in the selected line/face.

For [Free Face Node], select a free face node and all points that make contact with the node-containing element at an angle smaller than the specified angle. Press to open the Free face/line entity selection window to select the target.



There are two input methods:

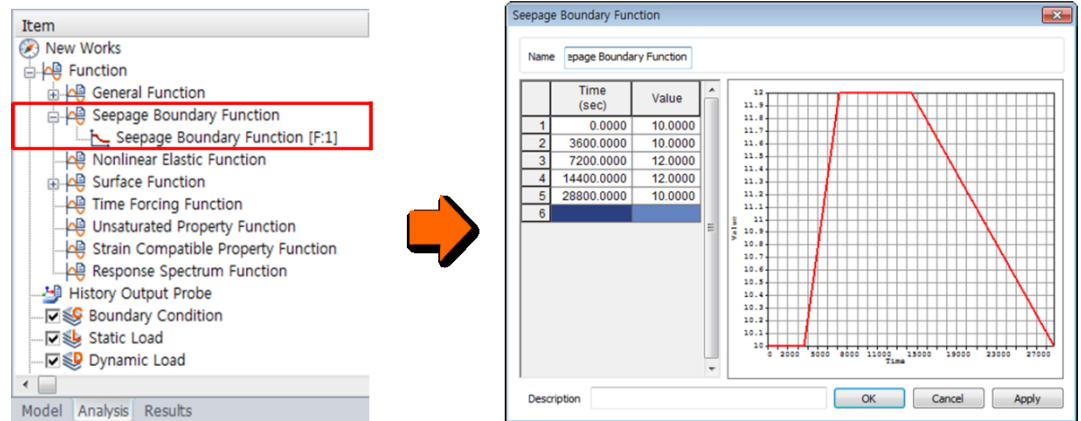
1. [Total] : Input the head value calculated from the origin, regardless of the model position.
2. [Pressure] : Set the groundwater level condition by entering '0' for nodes that are on the groundwater surface.

Transient analysis, in which the water level changes with time, can be defined as a function.

When using a function, the input value and function value are multiplied and reflected in the analysis.



The defined function is registered under Function > Seepage Boundary Function, and can be edited as a table using right mouse click > Edit.

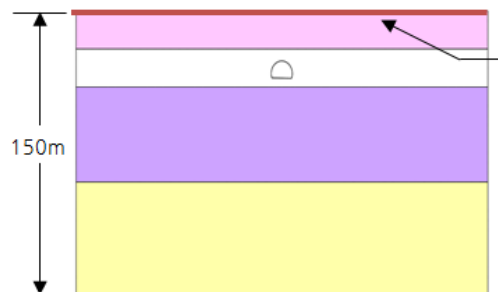


If Total Head < Potential Head, then $Q = 0$

A head-flow rate conversion boundary condition for water level variation analysis.

As the water level changes with time, such as for rapid drawdown, suction can occur and the seepage flow can be reversed. If the water level falls suddenly for embankments or dams, the descending water level speed is generally faster than the seepage speed within the body. To simulate these real conditions, the head boundary conditions need to change automatically according to the water level. In other words, when the node boundary at the bottom of the water level is exposed to the top, the total nodal head is not the total head of the descended water level; it is the total head value before the descent, which is maintained for a certain period, after which it falls gradually with time.

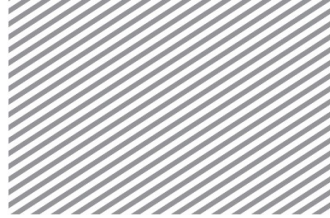
This option can be applied where the water level changes periodically and can be applied simultaneously with a time variant function. However, if this option is checked when the input (total head) height of the water level is above the selected node position, the boundary condition is automatically eliminated and so, the option must be unchecked for this case.



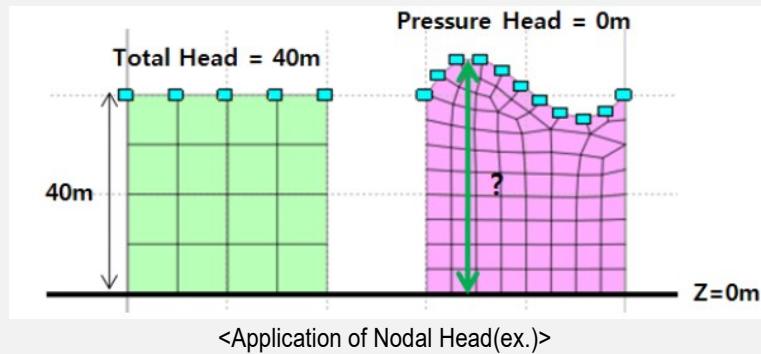
If you put the total head as 140m, the height of the water level is below the selected territory. therefore,
→ **Uncheck [If Total Head < Potential Head, then $Q=0$]**

Tip

The nodal head is applied then the water level position is known in advance. It is used to simulate confined flow that does not form a phreatic surface. It is also used to simulate unconfined flow that creates a seepage face.

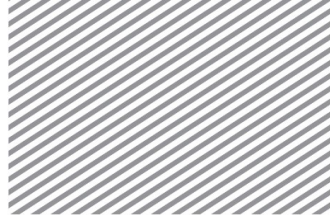


The head boundary conditions can be selectively input between the Total Head and Pressure Head, depending on the analysis condition. As shown in the figure below, it is convenient to input the Total Head directly when the node position for boundary condition specification is easy to find from $Z=0$ on the GCS. However, if the exact height is hard to find or if the height changes, it is convenient to input the Pressure head as 0.



Boundary condition set

Register the set constraint conditions on the desired boundary condition set. The user can specify the name of the boundary condition set.



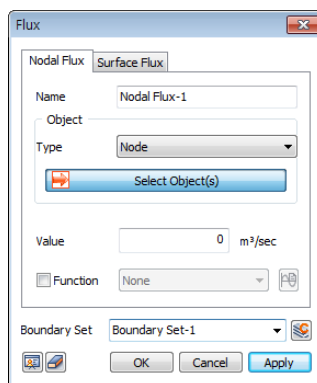
3.9 Nodal Flux

Overview

Input the flux at an arbitrary node.

The nodal flux is a boundary condition used in Seepage/Consolidation analysis (Fully-Coupled).


► Nodal flux

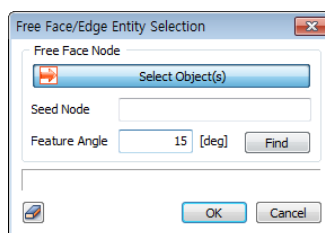


Methodology

Input the inflow/outflow per unit time on a particular position in terms of volumetric units. The target can be selected from [Node], [Edge], [Face], [Free Face Node].

For [Node], the node is directly selected to define the nodal flux. Selecting [Edge] or [Face] defines the flow rate conditions at all nodes in the selected edge/face.

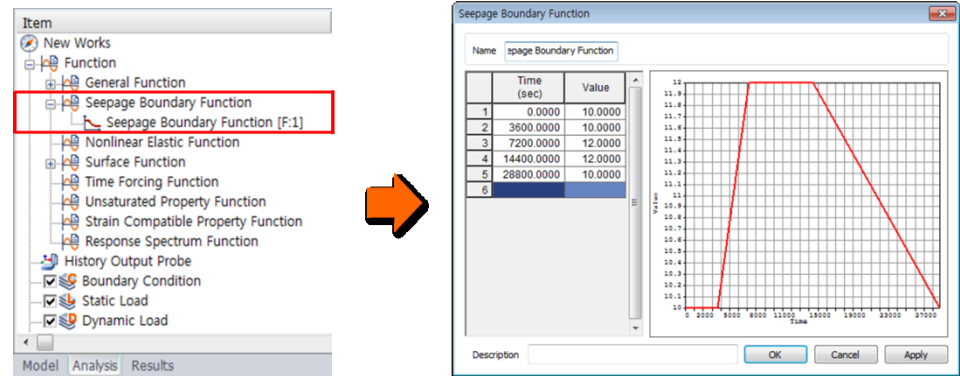
For [Free Face Node], select a free face node and all points that make contact with the node-containing element at an angle smaller than the specified angle selected. Press the  button to select the reference node, target element and feature angle.



Transient analysis (where the water level changes with time) can be defined as a function.

When using a function, the input value and function value are multiplied and reflected in the analysis.

The defined function is registered under Function > Seepage Boundary Function, and can be edited as a table using right mouse click > Edit.

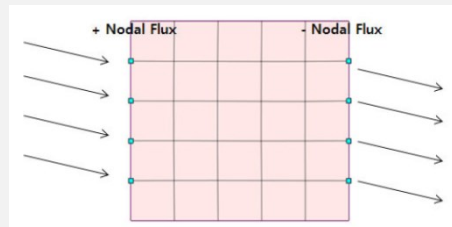


Boundary Set

Register the set constraint conditions on the desired boundary set. The user can specify the name of the boundary set.

Tip

The nodal flux boundary condition is used to simulate the inflow and outflow that happens at a node. The (+) represents water flow into the node and (-) represents water flow out of the node. The time variant - flux boundary condition can be input by coupling with a seepage function.



<Application of Nodal Flux>



3.10 Surface Flux

Overview

Input the surface flux boundary condition in terms of flow rate per unit area.
The surface flux is a boundary condition used in Seepage/Consolidation analysis (Fully-Coupled).

► Surface Flux

The Flux dialog box, Surface Flux tab, contains the following fields and options:

- Name: Surface Flux-1
- Type: Face Flux (dropdown)
- Object Type: Surface (dropdown)
- Select Object(s) button
- Value: 0 m³/sec/m²
- Function: None (dropdown)
- Checkbox: If $q > K_{sat}$, then Total Head = Potential Head
- Boundary Set: Boundary Set-1 (dropdown)
- Buttons: OK, Cancel, Apply

Methodology

Input the surface inflow/outflow rate of a specific point in terms of flow rate per unit area. It can be defined as either an [Edge Flux] or [Face Flux]. Generally, the [Edge Flux] is input for 2D models and [Face Flux] is input for 3D models.

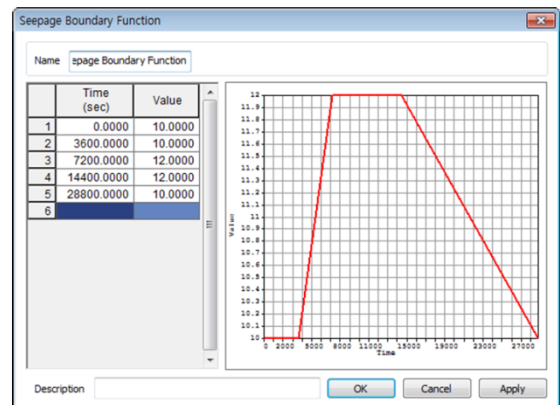
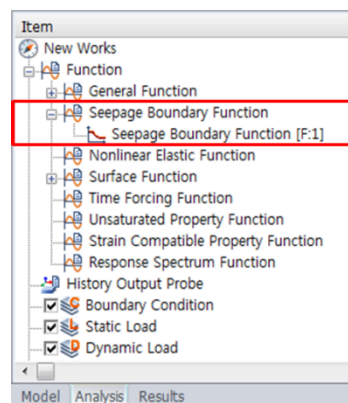
The Face flux can be defined by entering the value on the edge or surface geometry, or directly entering it on the selected element edge.

Define the inflow input from rainfall etc. as a (+) value and define the outflow input from excavation or pumping etc. as a (-) value.

Transient analysis (where the water level changes with time) can be defined as a [Function].

When using a function, the input value and function value are multiplied and reflected in the analysis.

The defined function is registered under Function > Seepage Boundary Function, and can be edited as a table using the right mouse click > Edit.



If $q > K_{sat}$, then Total Head = Potential Head



The flux-head boundary conversion condition for rainfall analysis.

For example, the Surface flux can be used to define the ground surface boundary conditions when the rainfall intensity on the ground surface is inputted.

This function applies a forced inflow rate, as large as the rainfall intensity, onto the ground surface. If the absorption capability of the soil stratum surface is larger than the rainfall intensity, the soil stratum absorbs all the rain water. However if the absorption capability is smaller, rain is absorbed into the ground surface by only the absorption capability amount, and the rest of the rain flows across the ground surface.

If the rainfall intensity is larger than the absorption capability, the ground surface is in a saturated state during rainfall, as if the groundwater level existed above the surface. Hence, the area of rainfall needs to be changed to a water level line.

Use the [If $q > K_{sat}$, then Total Head = Pressure Head] option to automatically change the ground surface boundary from the existing rainfall intensity inflow condition to a water level condition for analysis. This option is only available when the rainfall intensity acting on the surface is larger than the absorption capability of the ground surface.

Boundary Set

Register the set constraint conditions on the desired boundary set. The user can specify the name of the boundary set.

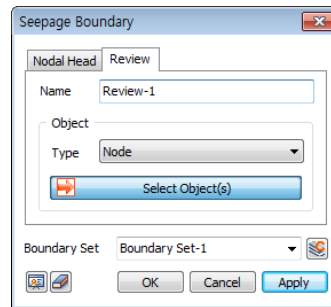


3.11 Review

Overview

Conduct iterative calculations using the Review function when the exact seepage line is hard to find. Review is used as a boundary condition for Seepage/Consolidation analysis(Fully-Coupled).

►Review boundary

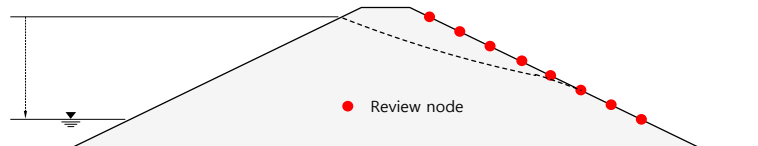


Methodology

Select the points upon which to place the Review boundary.


For example, assuming that seepage occurs at the downstream face of a homogeneous dam, the seepage line intersecting the downstream face cannot be found. In this case, set the review boundary and conduct iterative calculations.

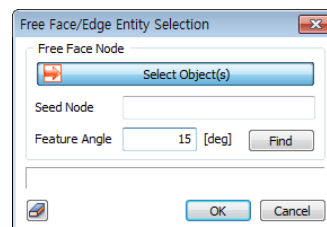
►Specify Review boundary node (ex.)



The target can be selected from [Node], [Edge], [Face], [Free Face Node].

For [Node], the node is directly selected to define the node on which iterative calculation will be conducted. Selecting [Edge] or [Face] conducts the review for all nodes in the geometry shape.

For [Free Face Node], select a free face node and all points that make contact with the node-containing element at an angle smaller than the specified angle selected. Press the  button to select the reference node, target element and feature angle.



The pore pressure P measured at the re-examined node is considered as the following condition, and these two roles can be used to automatically search for the seepage surface.

- 1) When $P > 0$, Consider as $P=0$
- 2) Delete when $P < 0$

Boundary Set



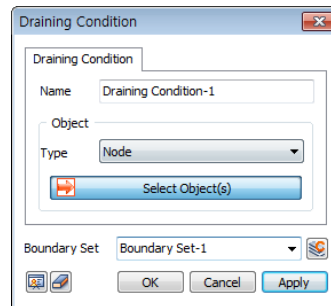
Register the set constraint conditions on the desired boundary set. The user can specify the name of the boundary set.

3.12 Draining Condition

► Draining condition

Overview

Used to simulate the domain where the excess pore pressure is 0 (drain).
The Drainage condition is used as a boundary condition for Consolidation analysis.




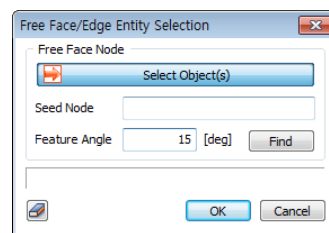
Methodology

Select the point to assign Drainage conditions.

The target can be selected from [Node], [Edge], [Face], [Free Face Node].

For [Node], the node is directly selected to define the node on which the Drainage conditions will be assigned. Selecting [Edge] or [Face] applies Drainage conditions for all nodes in the geometry shape.

For [Free Face Node], select a free face node and all points that make contact with the node-containing element at an angle smaller than the specified angle are selected. Press the  button to select the reference node, target element and feature angle.



The excess pore pressure in an area with assigned drainage conditions is maintained as 0 and this implies that water can escape due to loading applied to the ground. The Drainage condition is mainly used when the permeability coefficient is large, or if the loading change is small.

Boundary Set

Register the set constraint conditions on the desired boundary set. The user can specify the name of the boundary set.



3.13

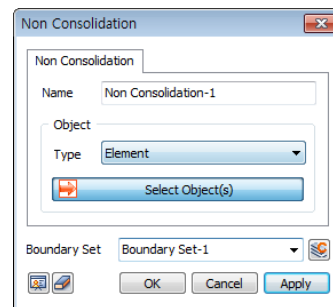
Non-Consolidation

► Non-consolidation

Overview

Used to model non-consolidation layers.

Non-consolidation elements are used as boundary conditions for Consolidation analysis.



Methodology

Select the point to assign the Non-consolidation boundary conditions.

The target can be selected from [Element], [2D Element], [3D Element], [Face], [Part].

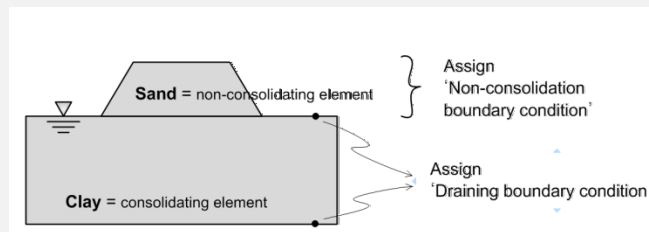
Selecting [Face] or [Part] applies the unconsolidated boundary conditions for all elements in the geometry shape.

Boundary Set

Register the set constraint conditions on the desired boundary set. The user can specify the name of the boundary set.

Tip

In Consolidation analysis, the elements have an additional pore pressure degree of freedom, as well as displacement degree of freedom, at the nodes. FEA NX assumes all elements have a degree of freedom for pore pressure, unless the two boundary conditions (Non-consolidating condition, Drainage condition) are specified for consolidation analysis. Hence, for embankment materials that do not express consolidated behavior directly, the non-consolidating element conditions need to be defined to apply it as a general structural element. Also, the drainage conditions need to be defined for drainage boundaries in consolidating elements. If the boundary conditions are properly defined and consolidation analysis is conducted, the excess pore pressure is 0 (zero) where the non-consolidation conditions and drainage conditions are applied.





3.14

Create Boundary from Results

► Create Boundary from Results

Overview

The 'Nodal Seepage' is created to the boundary condition from the results which analysis has been completed, and this is available in another analysis case as the boundary condition type.

Node	Total Head (m)	Pore Pressure Head (m)
230	1.7757e+000	7.7569e-001
231	1.7766e+000	5.7657e-001
232	1.7776e+000	3.7761e-001
233	1.7788e+000	1.7877e-001
234	1.7799e+000	-2.0086e-002
235	1.7814e+000	-2.1861e-001
246	1.7587e+000	7.5867e-001
247	1.7596e+000	5.5960e-001
248	1.7607e+000	3.6072e-001

3.15

Boundary Table Import / Export

Overview

- Import the information of boundary conditions from excel file or export them to excel. The sample of table for boundary conditions (BoundaryTable Sample.xlsx) can be found in the installation folder. (ex. C:\Program Files\MIDAS\FEA NX\Sample)

Methodology

There are limitations in this function as follows.

- Constraint : The 'Advanced' type is only supported. The DOFs of 'Tx-Rz' are displayed '1' (check on) or '0' (check off).
- Nodal Head : The 'Total' and 'Pressure' type are separated.
- Nodal Flux : This is exported the same way as the 'Nodal Head' type.
- Review : This is exported only the node information.

3.16

Transmitting

Overview

Approximately express the semi-infinite ground layer by setting a virtual slip surface perpendicular to the horizontally layered ground. This is done to consider the surface wave propagation into the far-field ground.

The transmit boundary condition is only used for Dynamic analysis > 2D Equivalent linear analysis.



The boundary conditions in ground modeling can be largely divided into:

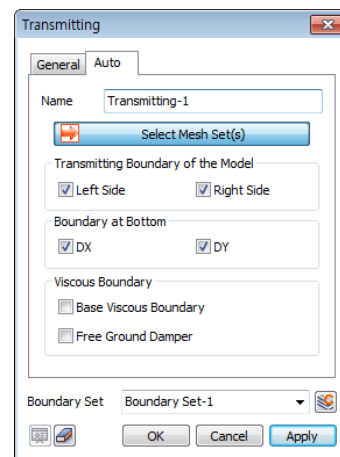
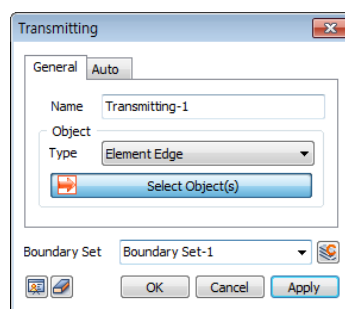
1. [Element Boundary Condition]
2. [Viscous Boundary Condition]
3. [Transmitting Boundary Condition].

1. The [Element Boundary Condition] can be divided into: 1) the free end, where the force of the earthquake response load is input, and 2) the fixed end, where the displacement is input at the free field boundary position. The [Element Boundary Condition] can consider the effects of earthquake waves in the free field, but it does not consider the effects of waves reflecting off the foundation slabs of an existing structure. This effect gets larger as the boundary position moves closer to the foundation slab.

2. The [Viscous Boundary Condition] was developed as a solution to the flaws of the [Element Boundary Condition] using a boundary condition that absorbs material waves having a certain angle to the boundary, developed by Lysmer & Kuhlemeyer, Ang & Newmark, etc. However, because the [Viscous Boundary Condition] cannot fully process the effects of complex surface waves, the boundary needs to be set at a certain distance from the foundation slab, as with the element boundary.

3. The [Transmitting Boundary Condition] supplements the flaws of the [Viscous Boundary Condition] and considers the effects of nearly all types of material and surface waves. The horizontal soil layer can be expressed as a spring and damper using a function of frequency. The [Transmitting Boundary Condition] generally assumes that the horizontal properties of each ground layer are equal and so, satisfactory results can be obtained even when the boundary condition exists at the structure itself. However, to accurately consider the property changes due to horizontal deformation, it is effective to maintain a certain distance between the boundary and foundation slab.

- Transmitting – General
- Transmitting – Auto



General

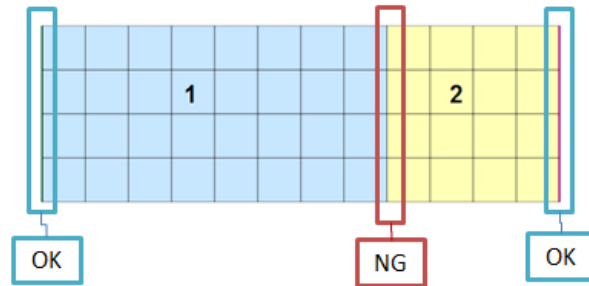
Methodology

Select the element edge and line to set the transmit boundary on that element edge. The ground information assigned to the element can be used to create the transmit boundary.

When the line between 2 different elements is selected, the transmit boundary is not created.



►Transmitting – General



Auto

Methodology

Selecting a mesh set automatically creates boundary conditions and elements at the left/right/floor of the mesh, according to the user specified options. The spring constant value is calculated depending on the ground material characteristics defined for the mesh set.

The right/left transmit boundaries of the analysis model can be set, and the viscous boundary can be created at the floor surface and free face.

It is difficult to accurately simulate the ground, which exists almost infinitely, using a 2D model used in Ground-Structure Analysis. Hence, the model boundary needs to be set at an engineering-appropriate position and processed to simulate in-situ conditions.

Boundary Set

Register the set constraint conditions on the desired boundary set. The user can specify the name of the boundary set.

Tip

* Auto create viscous boundary

The Viscous boundary condition can be created as follows.

1. Compute Cp and Cs

The Cp and Cs are calculated using the equations below:

$$C_p = \rho \cdot A \cdot \sqrt{\frac{\lambda + 2G}{\rho}} = W \cdot A \cdot \sqrt{\frac{\lambda + 2G}{W \cdot 9.81}} = c_p \cdot A$$

$$C_s = \rho \cdot A \cdot \sqrt{\frac{G}{\rho}} = W \cdot A \cdot \sqrt{\frac{G}{W \cdot 9.81}} = c_s \cdot A$$

Here, $\lambda = \frac{v \cdot E}{(1+v)(1-2v)}$, $G = \frac{E}{2(1+v)}$

λ : Volume modulus, G : Shear modulus, E : Elastic modulus, v = Poisson's ratio,

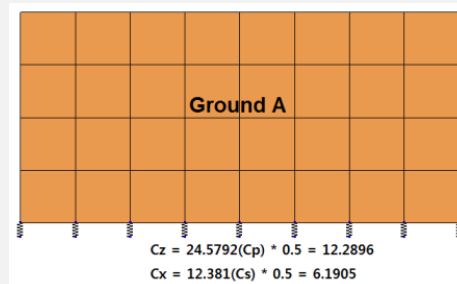
A : Cross-section area

2. The cross-section area is automatically considered until the surface spring is created, so only Cp and Cs need to be computed.



Class	Elastic modulus	Volume modulus	Shear modulus	Unit weight	Poisson's ratio	P wave	S wave
	E	λ	G	W	ν	Cp	Cs
	(tonf/m ²)	(tonf/m ²)	(tonf/m ²)	(tonf/m ³)		(tonf·sec/m ³)	(tonf·sec/m ³)
Ground A	2000	1459.53	751.88	2	0.33	24.579	12.381

- Multiplying the Cp, Cs (tonf·sec/m³ units) by the cross-section area eventually leads to the spring stiffness of the viscous boundary element in tonf·sec/m units.
- The shaded cell parameters are the physical properties of the ground that the user inputs during modeling. The volume modulus and Shear modulus are calculated using the Elastic modulus and Poisson's ratio. Hence, there is no need to input additional values when creating a viscous boundary element.
- When creating the bottom viscous boundary element, the spring is automatically created by considering the element area (effective length*unit width) as shown below.
- Input the Cp value for the normal direction coefficient at the point of spring creation and input the Cs value for the parallel direction. Hence, the bottom spring coefficient Cz becomes the Cp value and the Cs value is applied to the Cx.



<Auto-create viscous boundary>

Section 4 Load

Load (Static)

Loading that can be applied as an external force for linear/non-linear static analysis (Static/Slope/Consolidation analysis) can be specified as Self weight, Force/Moment, Displacement, Pressure, Water pressure, Line/Element beam load, Temperature, Prestress, Initial equilibrium force or Combined load.

►Table. Static load

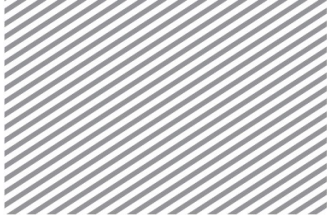
Type of Static load	Definition
Self weight	Input the self-weight of an element as a load
Force	Input the Force (as X, Y, Z axis direction component) on a desired model node
Moment	Input the moment (as X, Y, Z axis direction component) on a desired model node
Displacement	Input the displacement on a desired model node The displacement acts in the node coordinate system direction
Pressure	Input the pressure on a surface or line
Water pressure	Automatically input the water pressure load according to the position on the input water level line
Line beam load	When multiple beam elements are connected continuously, specify both ends of the beam element and input a distributed force
Element beam load	Input a distributed force on a beam element
Temperature	Input the node temperature on an arbitrary node for thermal stress analysis. Input the initial temperature value used on a model node into the analysis condition
Prestress	Input the pre-loading on a Structure/Ground element
Initial equilibrium force	Input the initial in-situ stress
Combined load	Set the load combination using load sets and the factor for each set

Load (Dynamic)

Loads that are applied as external forces for linear/non-linear dynamic analysis can be specified as Response Spectrum, Ground Acceleration, Time Varying Static Load, Dynamic Nodal, Dynamic Surface, Mass, Dynamic Result Function or Table form. It generally implies that the time variant load function form and the static load can be converted to a mass form to be used in dynamic analysis.

►Table. Dynamic load

Type of dynamic load	Definition
Response spectrum	Input the spectrum data and load conditions needed for Response Spectrum Analysis
Ground acceleration	Input the ground acceleration in time load function form (DB can be used)
Time varying static	Multiply a time function by a static load to define a dynamic load
Dynamic nodal	Input the time load function (load component x time function) on an arbitrary node
Dynamic surface	Input the time load function(load component x time function) on a surface in pressure form
Load to Mass	Convert a static load to mass for application
Train Dynamic Load Table	Input or modify the train moving load using a table



►Table. Thermal load

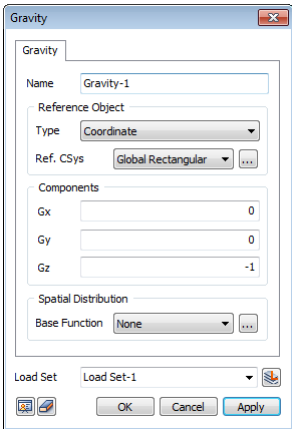
Type of thermal load	Definition
Heat Flux	Input the heat flux load. It can be applied to nodal points / edges / faces.
Heat Source	Defines the heat generated in a solid body.
Prescribed Temperature	Sets the temperature to have fixed value during analysis.
Convection	Input the convection condition. It can be applied to nodal points / edges / faces.

Self Weight

Overview

Enter the self-weight of elements included in the model as applied loads, or modify or delete previously entered self-weight.

The computed self-weight can be applied in each GCS X, Y and Z direction as a body force for static analysis. When considering the effects of self-weight in dynamic analysis computations, the option can be considered through the Project Setting function.



Methodology

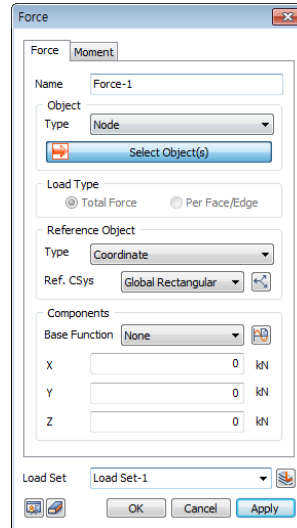
Input the factor for the self-weight application direction in Project settings depending on the work environment (2D/3D). The volume, density and gravitational acceleration of the input element is used to automatically compute the self-weight included in the analysis model. The self-weight direction is defined by a unit vector. The default value for the gravitational direction is set as -1.

[Spatial Distribution]
The 'Generalized Space Function' can be applied in the 'Self Weight'. The input of 'Generalized Space Function' is applied by scaling according to the location.

4.2 Force / Moment

Overview

Apply the Force or Moment load on an element node. Force is one of the most fundamental loads and it can be specified by 3 force components and a moment for each node. The direction can be defined with reference to an arbitrary coordinate system.

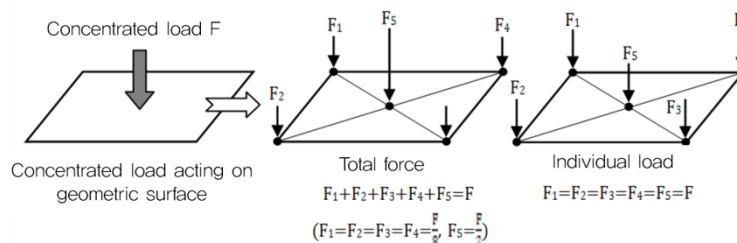


Methodology

Select the node where the load will be applied and set the size and direction. The load direction can be set using Default Load or Reference Object Load methods. For Default Load, the x, y, z, components are input with reference to the coordinate system in the bottom right corner. For Reference Object Load, select the target and reference shape to set the load direction. If the reference shape is a line, the component direction is set in the line creation direction. If the reference shape is a surface, the normal direction to the surface becomes the z component direction and the left, right directions become the x, y component directions.

If the selected target shape is geometric, the Force can be selected as either a total force or individual loads in Load Type as shown below. For total force, the input load size is considered as the total force acting on the selected line/surface and is distributed evenly to all nodes. For individual loads, the input load size is applied to all nodes of the selected line/surface.

►Total force / Individual load processing



The total force is distributed according to the length or area ratio when two or more targets are selected and the individual load is applied to each selected target.

[Object]

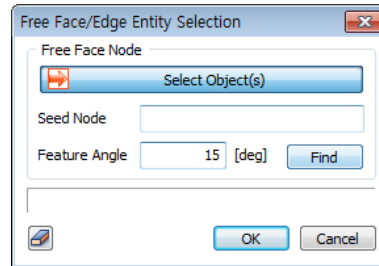
Point
Edge
Face
Node
Free Face Node

The applied load is defined on the node, but the selection target can also be set as a geometry shape (edge, surface etc.) or using auto-select free face nodes. For an edge or surface, the selected shape must



have been used for element creation and the force is applied to all nodes in the specified direction/size. For free face nodes, select a free face node and all points that make contact with the node-containing element at an angle smaller than the specified angle (feature angle) will automatically be selected.

►Free face node



[Reference Object]

Coordinate
Edge
Face
Vector
Normal

The load direction can be set using different methods. The reference coordinate system is the global rectangular(cylindrical) coordinate axis. Geometry shapes (edge and surface) can be selected as a reference direction. Selecting edge or surface displays the coordinate system of the selected shape and the load is set with reference to that system. The vector is used to specify the load direction using X,Y,Z vector components. The tangent direction can only be selected for surface objects and automatically sets the direction in the tangent direction of the selected surface.

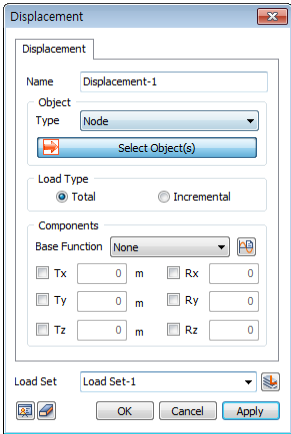
[Components]

Input the load size according to the set direction. A positive (+) value applies the load in the set direction and a negative (-) value applies the load in the direction opposite of the set. The load size changes with respect to the coordinate value increase in the GCS can be defined using a reference function. Here, the input value is multiplied by the function value for application.

4.3 Displacement

Overview

Input the displacement at an element node. The displacement is used to assign a displacement to a particular node. It is classified as a load because it causes structural deformation, but it has similar characteristics as the boundary condition. For example, constraint forces occur on the node where the displacement is input. The input displacement acts in the node coordinate system direction, which is defined by the GCS by default. The displacement is useful when applying the measured displacement in analysis or when understanding the plastic (limit) state of the element.



Methodology

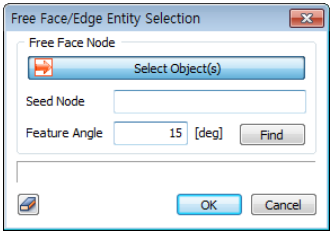
Select the node where the displacement will be applied and set the size and direction. The target selection method and size/direction settings are as follows:

[Object]

- Point
- Edge
- Face
- Node
- Free Face Node

The applied displacement is defined on the node, but the selection target can also be set as a geometry shape (edge, surface etc.) or using auto-select free face nodes. For an edge or surface, the selected shape must have been used for element creation and the displacement is applied to all nodes in the specified direction/size. For free face nodes, select a free face node and all points that make contact with the node-containing element at an angle smaller than the specified angle (feature angle) are automatically selected.

►Free face node



[Components]

Input the displacement size according to the set direction. A positive (+) value applies the displacement in the set direction and a negative (-) value applies the displacement in the opposite direction to the set direction. The size change with respect to the coordinate value increase in the GCS can be defined using a reference function. Here, the input value is multiplied by the function value for application.

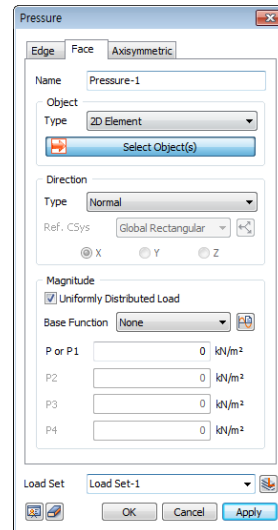
4.4
Pressure

Overview

Input the pressure loads on the face or edge of a plate element, plane stress element or solid element. Uniform load and linearly/non-linearly distributed loads can all be defined. The pressure load acts on the target geometric surface in a linearly/non-linearly distributed form per unit area and hence, the units are [Force/Area]. Because it is the force per unit area, it is applied to all selected target surfaces equally. The pressure is differ with the Force because the Force is using unit [N] and the force is loaded as nodal loads at sub-nodes generated at the selected geometric surface.



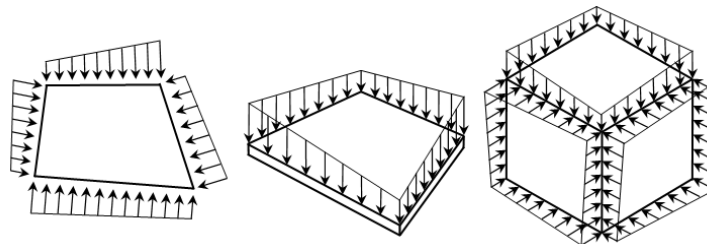
Because the Pressure load considers the area of the target surface (element surface) and is automatically converted as a nodal load for analysis, the two load conditions do not have a difference in analysis results. The more convenient load, depending on the applied direction, can be selected and used from the concentrated/distributed load option given in Analysis Condition.



Methodology

The pressure is input in distributed force form for an element face or edge. It can be used on 2D or 3D elements and the input direction can be specified as an arbitrary coordinate axis direction, arbitrary vector direction or normal direction. The direction setting is the same as for [Force]. Uniform load size or linearly/non-linearly distributed load size can all be specified as shown below, and the load change can be defined using a function of coordinate direction/distance. When applying a function, the input value is multiplied by the function value for application as a total load.

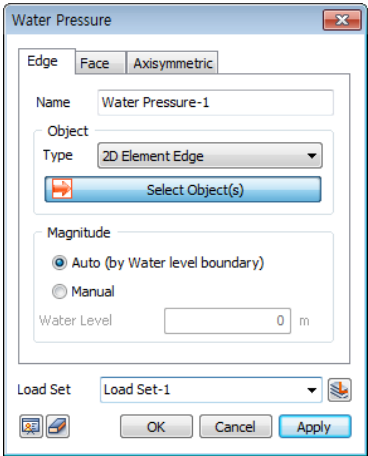
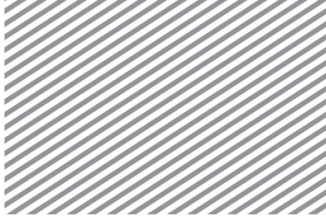
►Pressure acting on each element (target)



4.5 Water Pressure

Overview

A function that automatically calculates and applies the water pressure on an element boundary line or surface.



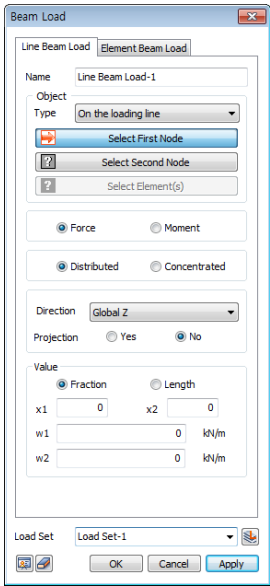
Methodology

Select the element boundary edge or surface where the water pressure will be applied. The [Auto] option can be used when the water level is pre-set, and it is used to automatically calculate the hydrostatic pressure based on the height difference between the water level and element boundary edge/surface. Use the [Manual] option to directly specify the water pressure size applied on the selected element edge/surface.

4.6
Beam Load

Overview

Concentrated and distributed force(moment) can be applied with reference to the GCS or the beam element coordinate system. Linear beam load can be used when multiple beam elements are connected continuously. Both ends of the beam element can be specified and a continuous beam load can be applied as either a distributed load or force load. The load can also be applied to a continuous beam placed in a curve on the same plane as the loading direction. For a beam element load, the beam load is applied as a distributed or force on the singular beam element.





Methodology

[Object]

For beam elements, select the beam element directly or select the line used to create the element to apply a load. For continuous beam loads, the On Load Application Line Method or Select Element Method can be used.

The [On the Loading Line] command applies the load to elements placed on the line between the two points used to specify the continuous beam load. In this case, selecting two nodes sequentially inputs the load for elements existing on the line between the two points. For the [Selected Element] command, the load is applied to the selected element. It can be used to apply the continuous beam load on elements that are not placed on the line. Select the start and end points of the beam element on which the load will be applied.

[Direction]

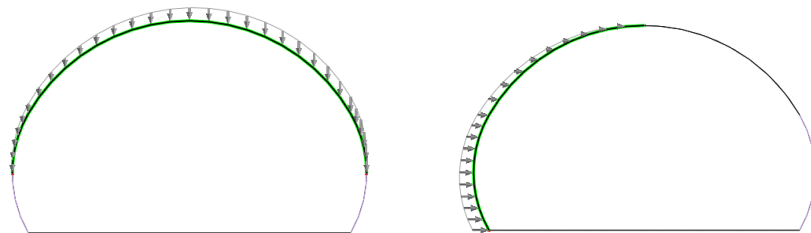
The direction can be set with reference to the global coordinate system (X,Y,Z) or element coordinate system (x,y,z). The projected area can be additionally set, which specifies whether to apply the load on the total beam element within the load application section, or to apply the load by the projected length, perpendicular to the load application direction. This option is only valid for [Distributed] loads in directions with respect to the global coordinate system (GCS).

[Value (Fraction/Length)]

- Fraction: Input the continuous beam load position using a relative length ratio of the load application section.
- Length: Input the actual length as the reference for the continuous beam load position.

x1 and x2 are the start and end points of the beam load respectively and w1,w2 are the load size at points x1,x2. Entering a negative load size applies the load in the opposite direction to the set direction, and a linearly distributed (increment, decrement) load can be set using the size difference.

►Apply linear beam load

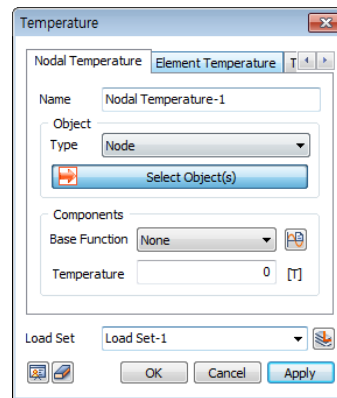


4.7

Temperature Load

Overview [Nodal/Element Temperature]

Input the final temperature on a node or element for thermal stress analysis. The temperature load induces deformation by temperature difference and all nodes are assigned an initial temperature in Analysis Setting. Entering a node temperature calculates the load caused by the temperature difference with the initial element temperature, and thermal stress does not occur if there is no displacement constraint. The element temperature load is similar to the nodal temperature load. However, specifying an element sets a uniform temperature for that selected element so that it has the same effect as entering the nodal temperature load for all nodes connected to that element.

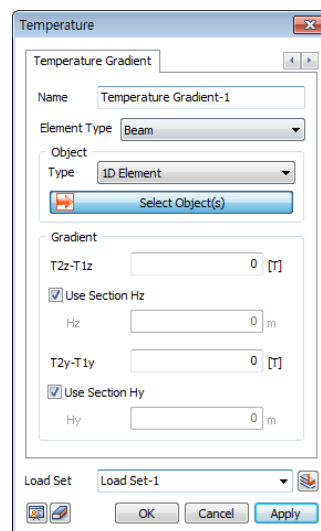


Methodology

Select a node or element to input the temperature. If the target shape is a geometry shape, the shape must have been used to create the element. All nodes in the selected shape are defined by the input initial temperature. The load is calculated by the temperature difference with that initial temperature. The temperature difference can be simulated by directly entering the temperature or applying a function with reference to the global coordinate system (GCS). When applying a function, the input value is multiplied to the function value for application.

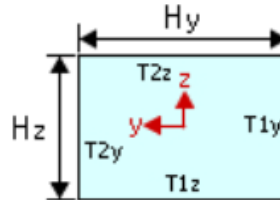
Overview [Temperature Gradient]

The temperature difference between the top and bottom of a beam element or plate element can be defined. Temperature gradient analysis can only be conducted for beam or plate elements, which can consider flexural rigidity. For beam elements, the temperature difference and distance of the outermost portion is input with reference to the y axis and z axis of the element coordinate system to consider temperature gradient loading. For plate elements, the temperature difference between the top and bottom faces and the plate thickness is input to consider temperature gradient loading.





Methodology

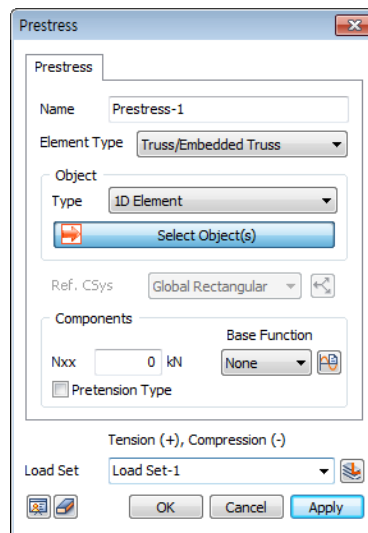


Select the beam (plate) element. The element can be selected directly, or the Beam/Shell that was used to create that element can be selected. Selecting a shape inputs the load for all element nodes in the shape. For a beam element, the temperature difference and distance in the z,y direction of the element coordinate system can be input. Checking the [Used Section] option uses the structural section information assigned to the element. For a plate element, the temperature difference in the thickness direction is input, and the plate element thickness is used. The distance can also be input directly.

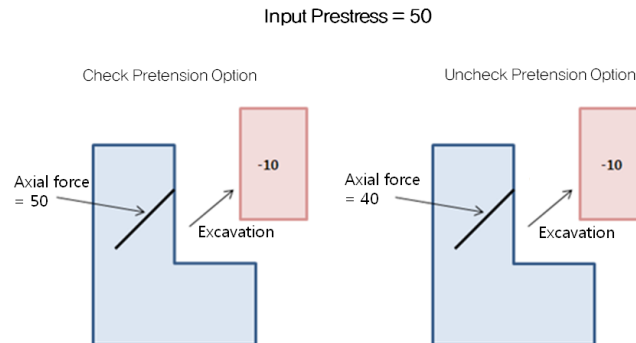
4.8 Prestress

Overview

Used to input the prestress load. For truss/beam elements, the initial load for axial direction force and moment can be applied. For plane strain elements, axis symmetric elements and solid elements, the initial stress can be defined.



For truss/beam elements, the [Pretension Type] option can be applied. Checking this option maintains the input pretension regardless of the stress state changes that occur in the construction stage. If this option is unchecked, the prestress changes with the input stress state. If a prestress of 50 is input as shown in the figure below, checking [Pretension Type] retains the prestress as the axial force, regardless of the stress change due to excavation. If the pretension is not checked, the axial force is affected by the stress change of 10 due excavation and so, the output axial force is 40. Pretension can only be applied if the prestress is input for a truss element type.



Methodology

[Element Type]

Truss/Embedded Truss
Beam
Plane Strain/Plane Stress
Axisymmetric
Solid

The applicable element types are as follows and the input load component is different for each type. The target shape can be selected by directly selecting the element or geometry shape (edge, face, solid). For shapes, the selected shape must have been used for element creation and the input load is applied to all element nodes within the shape.

[Components]

Define the load or stress for each axial direction and use a function to simulate the linear increment/decrement of the load size with reference to the global coordinate system(GCS). The load components for different element types are as follows:

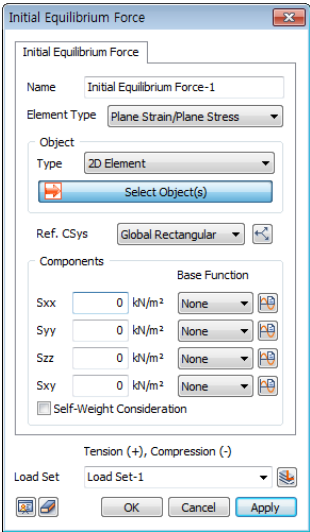
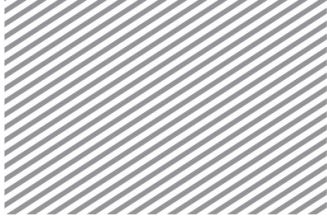
- N_{xx} : Initial axial force acting on 1D element
- M_x, M_y, M_z : Bending force (moment load) with reference to each element coordinate system
- S_{xx}, S_{yy}, S_{zz} : Axial stress in each axial direction
- S_{xy}, S_{yz}, S_{zx} : Shear stress in each plane direction

The load application direction for the initial stress of Plane/Axis Symmetric/Solid elements can be set with reference to the GCS or element coordinate system.

4.9 Initial Equilibrium Force

Overview

Use the prestress function to apply the resultant force or stress as initial conditions, depending on the element type. If the initial stress state is given as such, a force corresponding to the initial stress occurs. This initial equilibrium force uses the force created by the initial stress from the prestress function as an external force. If additional external forces do not exist, the initial equilibrium force is in equilibrium with the initial stress, hence the initial state is maintained. Also, use the [Self-Weight Consideration] option to assume the initial stress to be caused by gravity. If this assumption is applied, the load distribution factor considering the self weight of the element is applied when the element is removed from the construction stage.



Methodology

[Element Type]

- Truss/Embedded Truss
- Beam
- Plane Strain/Plane Stress
- Axisymmetric
- Solid

The applicable element types are as follows and the input load component is different for each type. The target shape can be selected by directly selecting the element or geometry shape (edge, face, solid), depending on the selected element type. For shapes, the selected shape must have been used for element creation and the input load is applied to all element nodes within the shape.

[Components]

Define the load or stress for each axial direction and use a function to simulate the linear increment/decrement of the load size with reference to the global coordinate system (GCS). The load components for different element types are as follows:

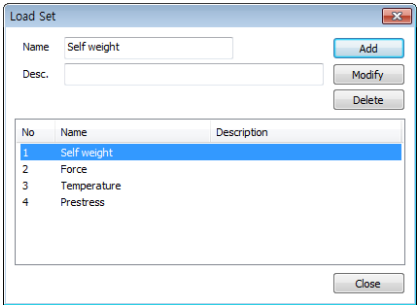
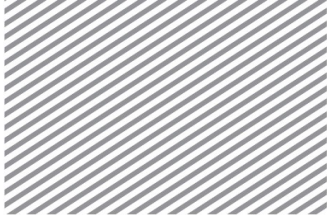
- Nxx : Initial axial force acting on 1D element
- Mx, My, Mz : Bending force (moment load) with reference to each element coordinate system
- Sxx, Syy, Szz : Axial stress in each axial direction
- Sxy, Syz, Sxz : Shear stress in each plane direction

The load application direction for the initial stress of Plane/Axis symmetric/Solid elements can be set with reference to the GCS or element coordinate system.

4.10
Load Set

Overview

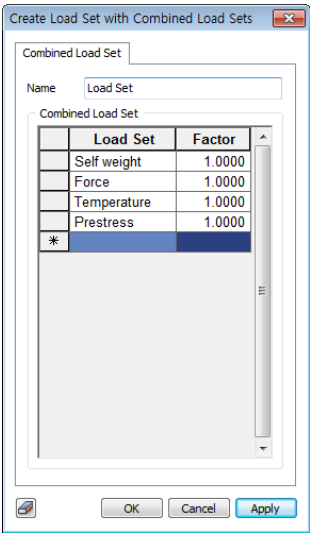
The load conditions can be classified as sets. It is useful to create the load set in advance for easy classification of data such as mesh sets, boundary sets, etc, during analysis. When entering the actual loads, the loads can be inputted individually by entering their names. If the load set is already classified, a particular factor can be assigned to each load set to create a combined load.



4.11 Combined Load Set

Overview

Create a combined load set by combining existing load sets. A factor can be assigned to each load set and the combined load is created with respect to these load factors. It is generally used when creating a load combination according to the design code.



4.12 Create Load from Results

Overview

The 'Nodal Force', 'Nodal Moment', 'Nodal Translational Displacement' and 'Nodal Rotational Displacement' are created to the loads from the results which analysis has been completed, and these are available in another analysis case as the load type.



► Create Load from Results

Node	V1 (m)	V2 (m)	V3 (m)
443	5.3879e-006	1.9681e-006	0.0000e+000
447	6.5462e-006	5.7651e-006	0.0000e+000
451	-2.7671e-006	-1.1108e-006	0.0000e+000
458	4.2829e-006	-5.3521e-007	0.0000e+000
463	-2.6624e-006	-1.9585e-006	0.0000e+000
464	3.3082e-006	-2.2490e-006	0.0000e+000
465	-2.0845e-006	-2.7927e-006	0.0000e+000
468	-1.1034e-006	-3.4815e-006	0.0000e+000
471	2.3852e-006	-3.3726e-006	0.0000e+000

Load Set: Self weight

4.13 Contraction

Overview

Consider shrinkage or simulate a volume loss around a lining of TBM tunnel. It can be applied by selecting beam/shell elements in 2D/3D model.

► Contraction – Shell
►► Contraction – Beam

Methodology

The 'Contraction' is for the shrinkage in the circumferential direction of tunnel and the 'Contraction Inc.' is for the shrinkage in the excavation direction of 3D tunnel. The 'Rep. Depth' is for the reference depth to calculate the shrinkage in the excavation direction of 3D tunnel. To specify this contraction, a contraction value is define as a strain value in percentage.

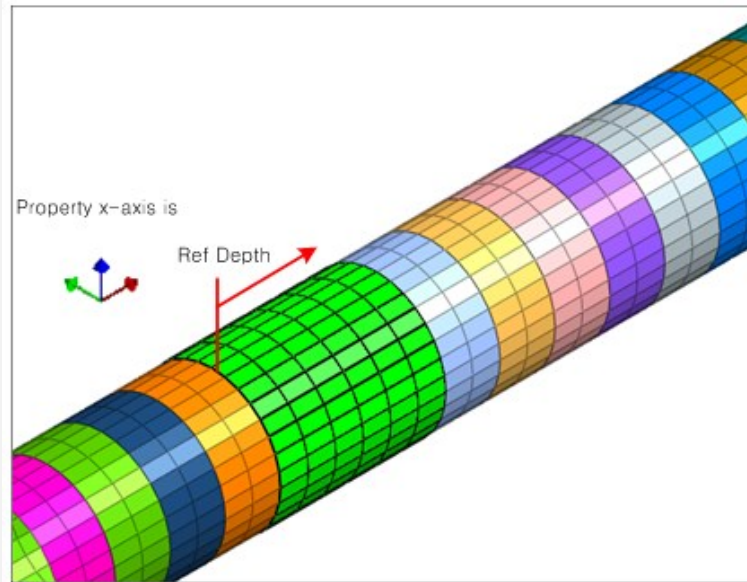
Tip

* Modeling Precautions

1. Contraction can only be applied to circular tunnels (bores tunnels) with an active continuous homogeneous lining. (If the selected elements are closure, contraction can be applied even though non-circular tunnels, but it is unable to get the correct results.)
2. In case of shell elements in 3D model, the coordinate system needs to be aligned that the excavation



direction is the Element CSys-X to calculate the excavation length automatically.

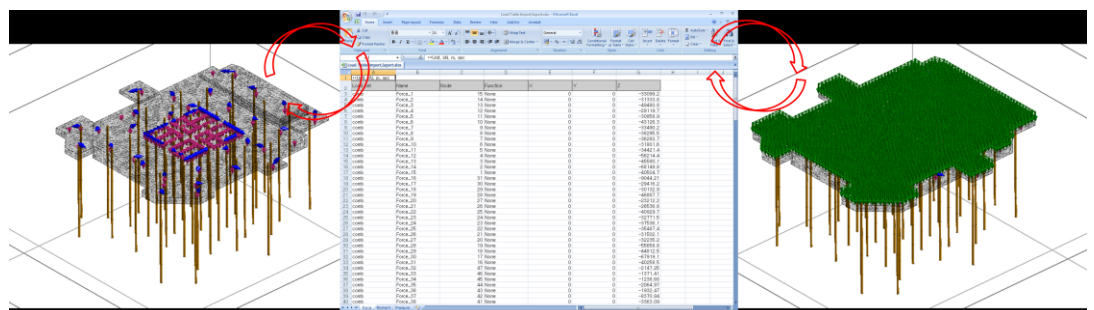


4.14

Load Table Import / Export

Overview

- Define or modify load sets through excel file like the usage of Load Table.
- Users can import the amounts of load sets from excel file and export defined load sets (node/element number, magnitude, and direction) to excel. The sample of table for load sets (LoadTable Sample.xlsx) can be found in the installation folder. (ex. C:\Program Files\MIDAS\FEA NX\Sample)
- Force, Moment, Pressure, Prescribed Displacement and Element Beam Load are available to use this function.
- Can be useful when users have to manage (input and modify) large numbers of load sets at once.
- In case that users select Geometries (Edge, Face) to define load sets, these load sets cannot be detected when exporting load sets to excel, meaning that users must select element nodes or faces to define pressure load.



Methodology

There are limitations in this function as follows.

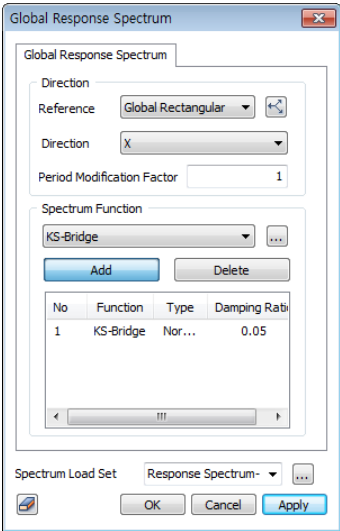
- Force / Moment : GCS (Global Coordinate System) is only applicable to import and export.
- Pressure : Axisymmetric type is not available. In terms of direction, "Normal" and "Direction" are available.
- Only one excel file can communicate with FEA NX at the same time.



4.15 Response Spectrum

Overview

Input the response spectrum function (spectrum data) and response spectrum load direction for response spectrum analysis. Response spectrum analysis expresses the natural period, natural angular frequency or natural frequency at the maximum physical quantity response as a function when a dynamic load is applied to the structure. The analysis can be expressed as a displacement response spectrum, pseudo rapidity response spectrum or pseudo acceleration response spectrum. The load and boundary conditions required for response analysis are similar to that of static analysis, but the load is defined as a function of time and the internal force and damping force are included for response analysis. The important results obtained from the transient response analysis are the node displacement, velocity, acceleration, and the force and stress on the element. Detailed explanations on mode combination methods and damping setting methods for maximum physical quantity (displacement, stress, member forces etc.) judgment of each mode can be found in the Analysis Control Settings.




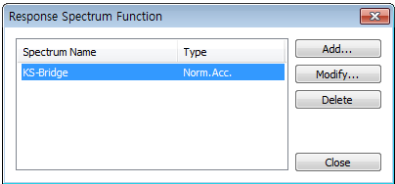
Methodology

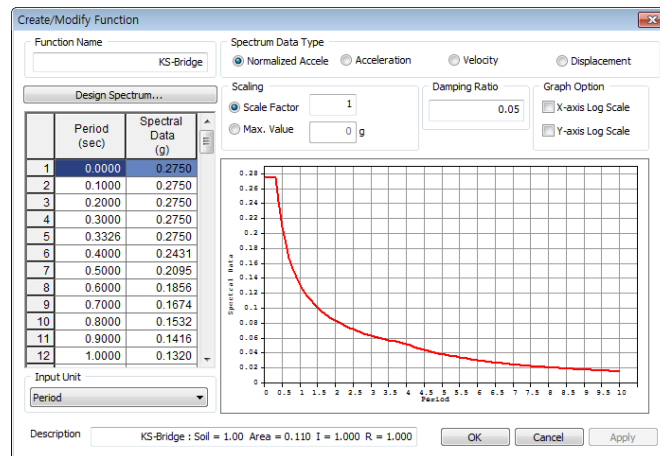
[Direction]

Set the application direction of the response spectrum load with respect to the global coordinate system (GSC) and input the [Period Modification Factor] that increases all applicable natural periods when applying the natural frequency from the eigenvalue analysis.

[Spectrum Function]

Set the spectrum data for analysis. Select the  button to define the spectrum function.





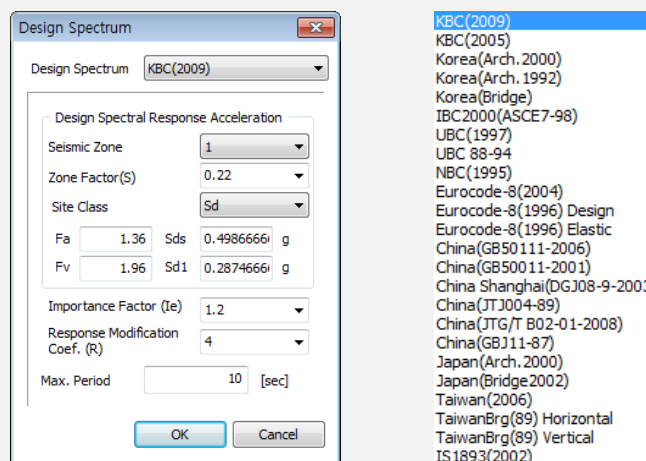
Directly input the period and spectrum value in the left input column on the dialog box. The spectrum function is expressed as a graph by plotting the spectrum value against the period. The spectrum function value for the natural period is linearly interpolated and used in response spectrum analysis. Hence, for regions where the curvature of the spectrum curve changes rapidly, dividing the region into multiple segments for compact spectrum values is recommended. The period range of the spectrum function must contain all natural periods of the structure.

The spectrum data types are normalized acceleration (acceleration spectrum, gravitational acceleration), acceleration, velocity and displacement spectrum. Changing the time function data format only changes the application format, not the data format. The scale factor is a gradient modulus for the entered data. The entire data can be scaled to fit the specified maximum value.

In the [Damping Ratio] space, input the damping ratio applied on the response spectrum. But if the damping ratio of the target structure is different, the input spectrum data is processed to fit the structural damping ratio.

Tip*** Apply Design spectrum**

Use the built-in design spectrum. The default built-in design spectrum types are as follows.



- ▶ Korea(Bridge) : Korea, Traffic Specification
- ▶ Japan(Bridge02) : Japan, Load instruction and Dynamic Analysis of Building
- ▶ China(JTJ004-89) : China, Specifications of Earthquake Resistant Design for Highway Engineering
- ▶ KBC 2009 : Korea, Korea Building Code-Structural 2009
- ▶ KBC 2005 : Korea, Korea Building Code-Structural 2005
- ▶ IBC2000(ASCE7-98) : USA, International Building Code 2000
- ▶ UBC(1997) : USA, Measure of Uniform Building Code 97
- ▶ EURO(2004H-ELASTIC) : Europe, Sesimic Design Measure for Structures

4.16

Ground Acceleration

Overview


Load the time load function for time history analysis using the ground acceleration. It is mostly used to check the seismic design of structures or liquefaction due to earthquakes. Time history analysis uses the dynamic structural properties and the applied load to calculate the structural behavior (displacement, member forces etc.) at an arbitrary time under dynamic loading. The Mode Superposition Method and Direct Integration Method are available for time history analysis, and detailed information on the damping setting for each analysis method can be found in [Analysis Control].

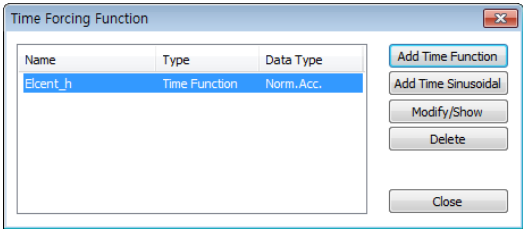
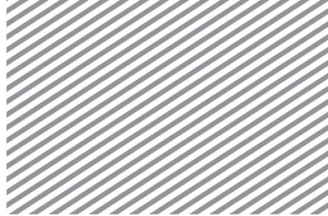
Methodology

[Direction]

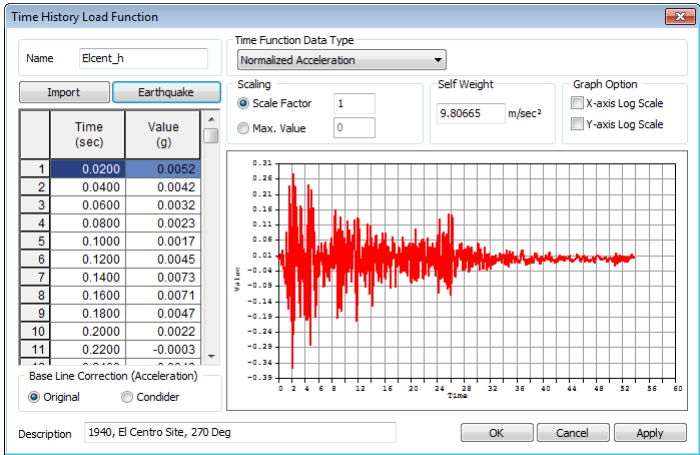
Define the input (transmission) direction of ground acceleration. It is set with reference to the global coordinate system (GCS) and multiple directions (X,Y,Z) can be combined to set a single ground acceleration load set. The increment coefficient for the ground acceleration can be defined using a scale factor and the arrival time can be controlled to set the ground acceleration delay time.

[Function]

Select the  button to define the ground acceleration function.



Add time function



Construct the time varying load by directly entering the time and the corresponding time varying load value in the left input column on the dialog box. The time function data types are classified by Acceleration, Force(load), Moments, Normalized Acceleration (Time history acceleration / Gravitational acceleration) or Normal(generalized). Changing the data format only changes the application format, not the data format. The scale factor is a gradient modulus for the entered data. The entire data can be scaled to fit the specified maximum value.

Tip

* Time function load application

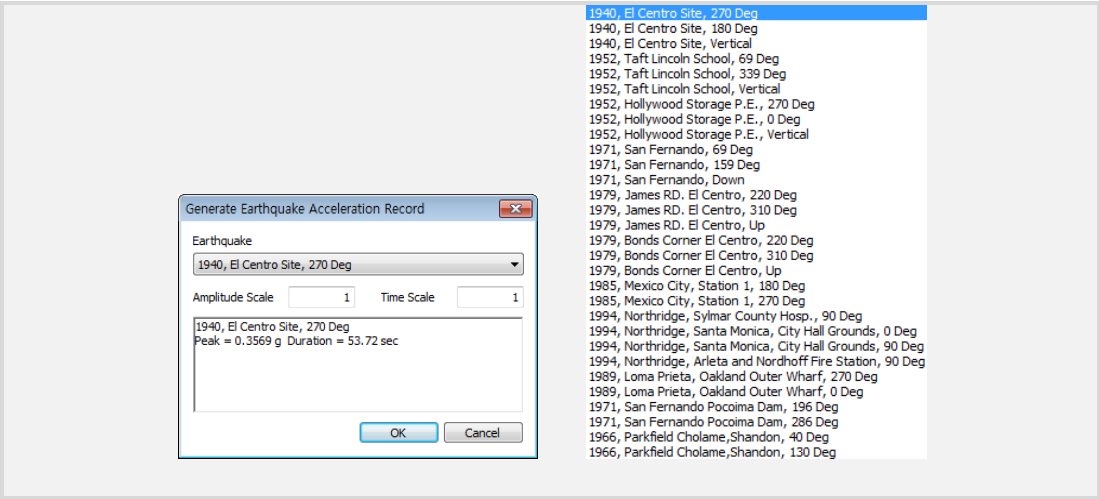
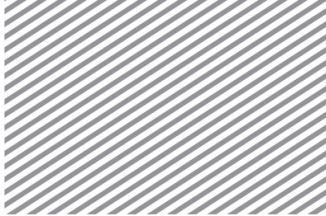
The defined function is also applicable for [Dynamic nodal (surface)] and [Time varying static]. Specifying 'force' or 'moments' uses the time varying load as a [Dynamic Nodal(surface)] input and specifying 'Normalized acceleration' or 'Acceleration' uses it to input the [Ground Acceleration]. Specifying 'Normal' uses the time varying load as the change in static load with time for [Time Varying Static] or [Dynamic surface].

Tip

* Import/Earthquake waves

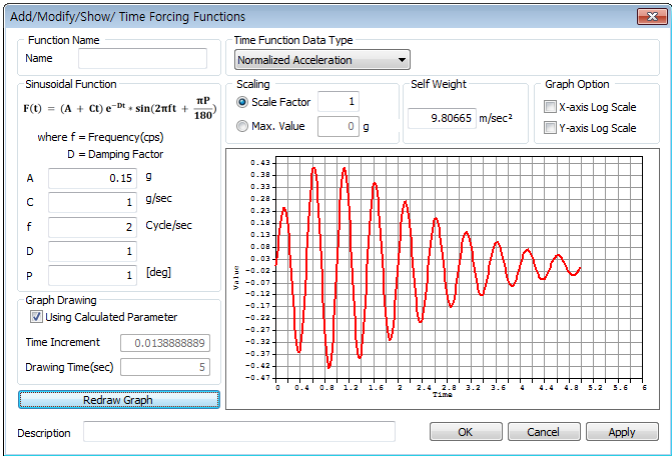
Save and import frequently used time varying load or select earthquake acceleration data from the program DB. There are a total of 32 types of earthquake accelerations.

[SGS, DGS, THD or WVE File (*.sgs, *.dgs, *.thd, *.wve)]



Add Time sinusoidal

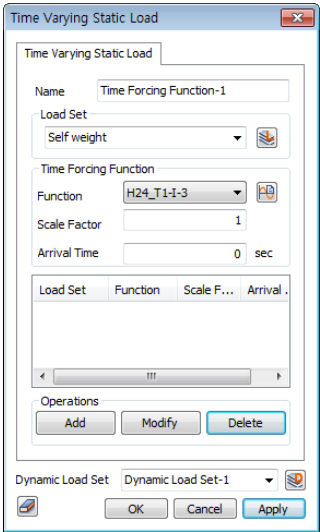
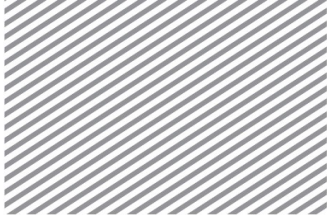
A sine function can be used to define the time varying load. A and C are constants, f is the frequency of the input load, D is the damping factor and P is the phase angle. If the time varying load is entered as a harmonic function, input the necessary sine function variables and click [Redraw Graph] to view the loading on the right hand side.



4.17 Time Varying Static


Overview

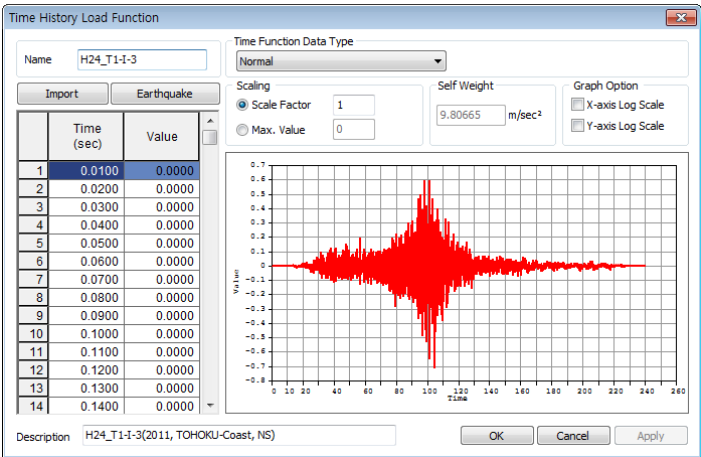
Use the pre-entered static load to create a time load function for time history analysis. The dynamic load is defined by multiplying the static load by the time load function. The time forcing function used here is dimensionless and only the 'Normal' type time history data can be used. This function is a replacement function that includes the [Dynamic nodal (surface)] created by the static load-time forcing function combination.



Methodology

Select the static load to be applied on the load set. The load application position, direction and size are already determined when entering the static load, and the to-be applied time forcing function is selected. The gradient modulus of the load can be defined by a scale factor and the arrival time can be set to simulate the delay time.

Select the  button to add (select) a time forcing function. It can only be applied when the time forcing function data type is 'Normal' (dimensionless function)



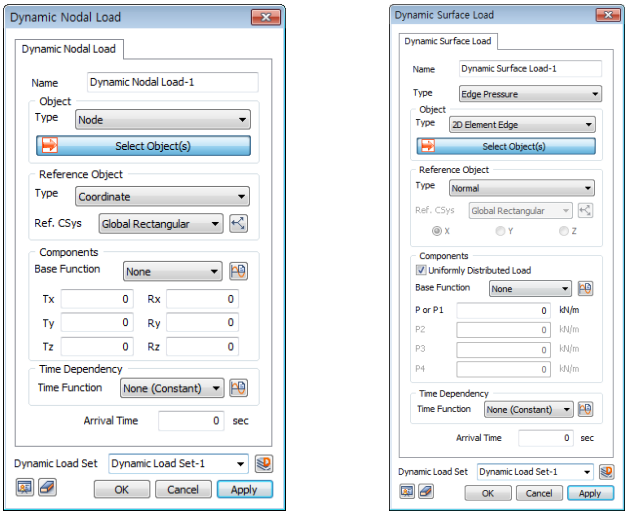
Construct the time varying load by directly entering the time and the corresponding load change ratio in the left input column on the dialog box. The scale factor is a gradient modulus for the entered data. The entire data can be scaled to fit the specified maximum value.



4.18 Dynamic Nodal (Surface)

Overview

Used to create a time load function for time history analysis directly. The static load and force (pressure load) is defined by applying a time forcing function. The applicable time function data type for dynamic nodal loads are 'Force', 'Moments' and, for dynamic surface loads, 'Normal'. In addition, the reference function can be applied to define a linear/non-linear distributed dynamic load that changes with position. Generally it is used to define vibration, driving, blast and railway movement loads. The arrival time can be set to simulate the delay time.



Methodology (Dynamic Nodal)

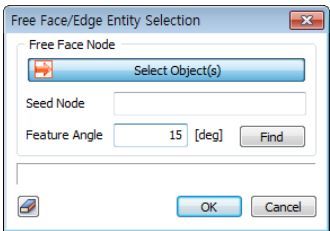
Select the node where the load will be applied and specify the direction. The load size is applied by multiplying the time function (load size by time) and each load component (scale factor).

[Object]

- Point
- Edge
- Face
- Node
- Free Face Node

The applied load is defined on the node , but the selection target can also be set as a geometry shape (edge, surface etc.) or using auto-select free face nodes. For a line or surface, the selected shape must have been used for element creation and the force is applied to all nodes in the specified direction/size. For free face nodes, select a free face node and all points that make contact with the node-containing element at an angle smaller than the specified angle (feature angle) will be automatically selected.

►Free face node



[Reference Object]



Coordinate
Edge
Face
Vector


The load direction can be set using different methods. The default input reference is the Global rectangular(cylindrical) coordinate axis. Geometry shapes (Edge and Surface) can be selected as a reference direction. Selecting 'Line' or 'Surface' displays the coordinate system of the selected shape and the load is set with reference to that system. 'Vector' is used to specify the load direction using X,Y,Z vector components.

[Components]

Input the load scale factor according to the set direction. Generally, the load size is pre-defined as a time variant value in the time function, and if the maximum ratio value is defined as 1, the actual load size is input in the time function. A positive (+) value applies the load in the set direction and a negative (-) value applies the load in the opposite direction to the set direction. The load size change, depending on the coordinate value increase in the global coordinate system (GCS), can be defined using a reference function. Here, the input value is multiplied by the function value for application.

[Time Dependency (Time function)]

Define the load size change with respect to time.

Select the  button to add (select) a time forcing function. It can only be applied when the time forcing function data type is 'Acceleration', 'Velocity', 'Displacement', 'Force' and 'Moments'.

Methodology (Dynamic Surface)


Dynamic surfaces are input as the distributed load change with respect to time on the element face or edge. It can be used on 2D or 3D elements and the input direction can be specified as an arbitrary coordinate axis direction, arbitrary vector direction or normal direction.

[Components]

Input the load scale factor according to the set direction. The uniform or linear/non-linear distributed load can be defined. A positive (+) value applies the load in the set direction, while a negative (-) value applies the load in the direction opposite of the set direction. The load size change, depending on the coordinate value increase in the global coordinate system (GCS), can be defined using a reference function. Here, the input load component is multiplied by the function value for application.

[Time Dependency (Time function)]

Define the load size change with respect to time.

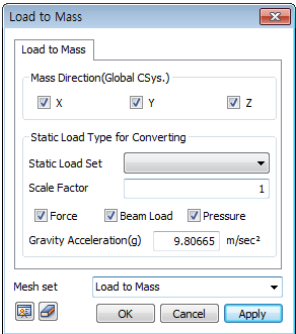
Select the  button to add (select) a time forcing function. It can only be applied when the time forcing function data type is 'Normal'. 'Normal' functions are dimensionless and have no units. Hence, if the pressure load size is directly entered, only the scale factor is input for the load component. If the maximum ratio value is defined as 1, the actual load size is input.

4.19

Load to Mass

Overview

Convert the already defined static load into a mass for application in dynamic analysis, such as Response Spectrum or Time History Analysis.



Methodology

[Mass direction(Global CSys.)]
Specify the directional component for consideration of the converted mass. When conducting analysis that considers earthquake effects (Response spectrum analysis, Time history analysis using earthquake data), the lateral behavior is mainly considered and the selected X,Y generally ignores the perpendicular direction mass component. However, for driving or floor slab vibration analysis, the gravitational direction dictates the dominant vibration mode and the Z direction is selected. Thus, the mass consideration direction is set to fit the purpose of the analysis type.

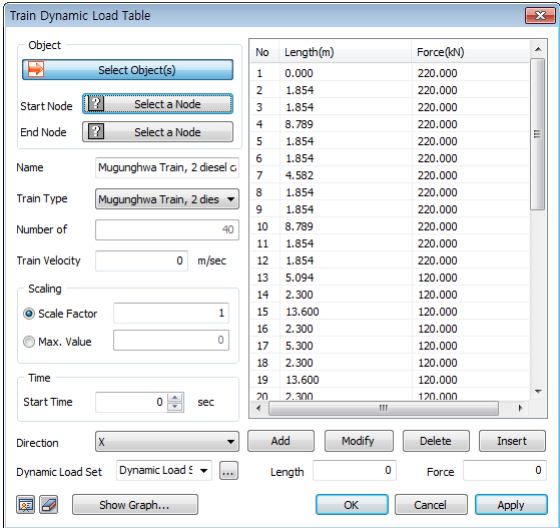
[Static Load Type for Converting]
Select the static load set to convert and select the force, beam load and pressure load information in the selected static load set for conversion. The scale factor during load-mass conversion can be considered and the gravitational acceleration value applied in the conversion is specified.

4.20
Train Dynamic
Load Table

Overview

Create a train dynamic load and apply it to the analysis model.
The train dynamic load can be created in the same was as Dynamic Analysis > Tools > Create Dynamic Load Data Generator > Train Dynamic Load.
The created load judges the train velocity and spacing between nodes and automatically applies it to the analysis model as a dynamic nodal.

►Train dynamic load table





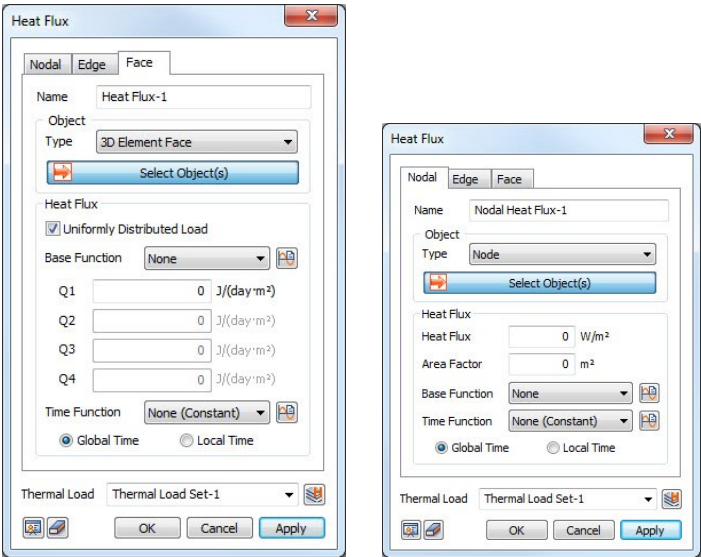
Methodology

[Object] : Select the edge or node the train dynamic load passes through, and select the start and end nodes.
[Name]: Define the train type name.
[Train Type] : Select the type of train. 6 basic DB (Mugunghwa Train, 2 diesel cars, Korea), [Saemaeul Train, 8 cars, Korea), [KTX, 20 cars, Korea), [EL-18 Standard, 6 cars, Korea), [EL-18 Standard, 8 cars, Korea), [EL-18 Standard, 10 cars, Korea) are provided and the user can directly input the length/force depending on the number of wheels.
[Number of Wheels] : Stands for the number of wheels on the train. It is the same as the added number on the table.
[Train Velocity] : Input the velocity of the train.
[Time] : Input the time the train dynamic load is applied.
[Direction] : Input the direction in which the train dynamic load is applied. It is generally applied in the gravitational direction and thus the user can specify it in the same direction as the gravitational direction of the 3D model.
[Dynamic Load Set] : Input the name that will be registered on the load set.

4.21
Heat Flux

Overview

Heat flux is used to model heat input through the surface of an object.
It is the function to input the heat flux load. It can be applied to nodal points / edges / faces.



Methodology

It is used as a load vector in heat transfer analysis. The heat flux is entered as the unit time energy per time x unit area (Watt/Area). A time function can also be applied.
[Name] : Enter the name of the heat flux to be defined.
[Type] : Select the type of target objects to which heat flux will be applied. Only the edges of 2D elements can be selected.

[Heat Flux] : Enter the heat flux value to the selected target objects.

[Uniformly Distributed Load] : Load the heat flux uniformly. If this is not selected, the heat flux will be loaded in a linear varying form.

[Area Factor] : Enter Area factor when Nodal method is used.

[Base Function] : Set a spatial function or a non-spatial function as a base function to be applied to the heat flux.

[Q1] : Enter the heat flux value when Uniformly Distributed Load is selected.

[Q2 - Q4] : For a linearly varying distribution, enter the heat flux values sequentially.

[Time Function] : Select the function to be applied to time.

[Global Time] : The time applied to the time function is based on the total analysis time.

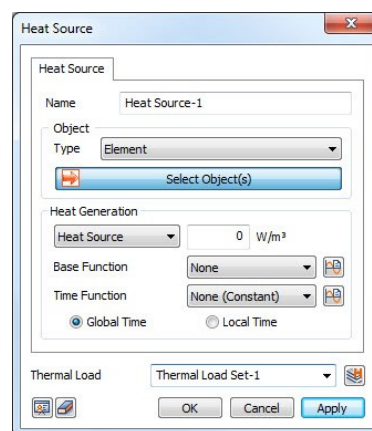
[Local Time] : The time applied to the time function is based on the subcase time.

[Thermal Load] : Register the specified heat flux to a desired load set. The user may assign any name to the load

4.22 Heat Source

Overview

An exothermic load is a modeling of the heat generated in a solid body, and is defined as the energy per unit time of a unit volume. The larger the volume of the structure, the greater the total calorific value, and it plays a similar role to the self weight in structural analysis. A time function can also be applied.



Methodology

An exothermic load is a modeling of the heat generated in a solid body, and is defined as the energy per unit time of a unit volume. The larger the volume of the structure, the greater the total calorific value, and it plays a similar role to the self weight in structural analysis. A time function can also be applied.

[Name Enter] the name of the condition of heat generation (source).

[Type Select] the type of target objects to which the transient heat generation conditions will be applied. Element, 1D Element, 2D Element, 3D Element, Edge, Face and Part can be selected.

[Heat Generation] Enter the magnitude of heat generation per unit volume.

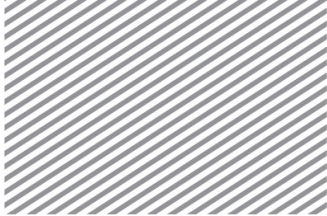
[Base Function] Set a spatial function or a non-spatial function as a base function to be applied to the magnitude of heat generation.

[Time Function] : Select the function to be applied to time.

[Global Time] : The time applied to the time function is based on the total analysis time.

[Local Time] : The time applied to the time function is based on the subcase time.

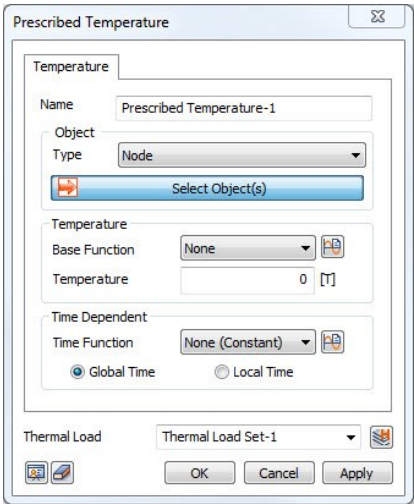
[Thermal Load] : Register the specified heat generation conditions to a desired thermal load set.



4.23 Prescribed Temperature

Overview

The prescribed temperature (fixed) serves as the boundary condition of the heat transfer analysis, makes it always maintained during analysis. It has a similar role to the constraint in structural analysis. A time function can also be added.



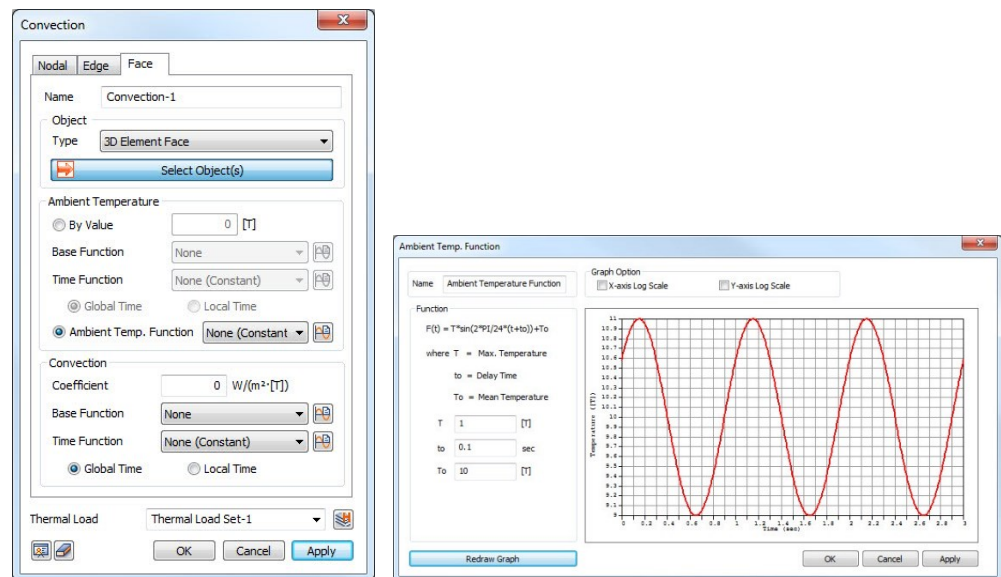
Methodology

- Name] :Enter the name of the prescribed temperature condition.
- [Type] : The applied temperature is defined on the node, but the selection target can also be set as a geometry shape (edge, surface etc.) or using auto-select free face nodes.
- [Temperature] : Enter the temperature value for the selected target objects.
- [Base Function] Set a spatial function or a non-spatial function as a base function to be applied to the magnitude of temperature.
- [Time Function] : Select the function to be applied to time.
- [Global Time] : The time applied to the time function is based on the total analysis time.
- [Local Time] : The time applied to the time function is based on the subcase time.
- [Thermal Load] : Register the specified fixed temperature conditions to a desired thermal load set.

4.24 Convection

Overview

The prescribed temperature (fixed) serves as the boundary condition of the heat transfer analysis, makes it always maintained during analysis. It has a similar role to the constraint in structural analysis. A time function can also be added.



Methodology

The phenomenon of heat transfer in a liquid or gas as the hot and cold parts move together is called convection. Convection conditions can be entered in the desired part of the model (nodes, lines, faces). Convection generated by buoyancy due to density change caused by temperature difference in fluid is called free convection.

Convection heat transfer rate is generally proportional to temperature difference. Enter the atmospheric temperature at which the structure is placed at ambient temperature, and the convection coefficient is the atmospheric convection coefficient. A time function can also be applied.

[Name] : Enter the name of the convection condition.

[Type] : Select the type of target objects to which convective heat transfer conditions will be applied. Face, 2D Element, 3D Element Face and 3D Element Free Face can be selected.

[Ambient Temperature] : Enter the ambient temperature value for the selected target objects.

[By Value] : Enter the ambient temperature directly.

[Ambient Temp. Function] : In the ambient temperature function, you can define the outside temperature using Sine function, where:

T: Maximum outside temperature,

to: Delay time, and

To: Minimum outside temperature.

[Convection Coefficient]: Enter the convection coefficient.

[Base Function] Set a spatial function or a non-spatial function as a base function to be applied to the magnitude of convection or temperature..

[Time Function] : Select the function to be applied to time.

[Global Time] : The time applied to the time function is based on the total analysis time.

[Local Time] : The time applied to the time function is based on the subcase time.

[Thermal Load] : Register the specified convection conditions to a desired thermal load set.



Section 5 Wizard and Tools

The 'Wizard and Tools' option is a tool that streamlines the modeling and design for analysis. The Wizard consists of the [Tunnel Modeling] function, which creates tunnel shapes and construction stages, and the [Anchor Modeling] function, which creates anchor shapes easily. The Tools consists of [Seismic Data Generator] for Response Spectrum Analysis, [Dynamic Load Data Generator] to compute the blast and train vibration load, and [Free Field Analysis].

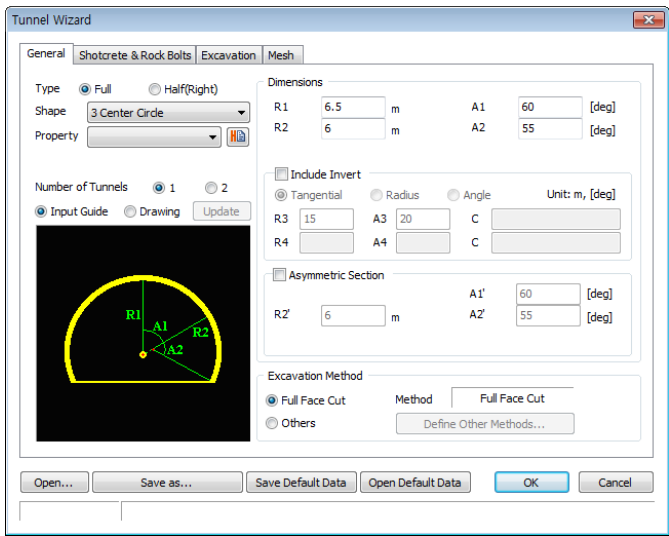
5.1 Tunnel Modeling

Overview

The Wizard creates a simple 3D tunnel model easily. It can be used to model a 3D tunnel model that considers the undisturbed stratum and ground surface. The Wizard is made up of 4 tabs: 1)General, 2)Shotcrete & Rock Bolts, 3)Excavation and 4)Mesh. The data in all 4 tabs need to be properly input to create the tunnel model. Because everything up to the material selection can be conducted at once, the Tunnel Wizard menu can be used immediately without any separate processes after running FEA NX. The data can be set as the default wizard data, or a separate Tunnel Wizard save file can be created. This allows for fast modeling when analyzing similar tunnels; only certain variables need to be modified for the existing tunnel model.

The Tunnel Model Wizard cannot be used if the modeling is not done. If a model exists, the Tunnel Model Wizard cannot be executed.

►Tunnel Modeling Wizard





General

Methodology

Input the number of tunnels and tunnel shapes, and set the excavation method.

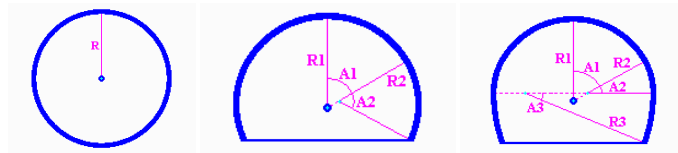
First, decide whether to model the full tunnel face or the half(right) of the full tunnel face. Be aware that the specified construction stages and result data etc. cannot be used if the model, created using the Tunnel Modeling Wizard, is modified later.

[Shape]

Determine the tunnel shape. Circular, 3 Center circle, 5 Center circle shapes are supported. The dimensions specify the tunnel changes, depending on the specified tunnel shape. Set the input guide to display the relationship between the tunnel shape and input values on the Tunnel Wizard dialog box. The guide can be used as a reference when entering the variables. The input values for the tunnel shape are the same as for Geometry > Point&Curve > Tunnel (Wire).

►Circular

►►3 Center circle



[Property]

Input the ground material around the tunnel. If the tunnel model is created using the Tunnel Wizard, a basic rectangular ground shape can be created around the tunnel and the upper stratum and index shape can be added to the model optionally. A basic model is the rectangular ground shape on the periphery of the tunnel. The basic shape can be composed of a homogeneous material. The material and properties are the same as the specified Material property in Property/Coordinate and System/Function.

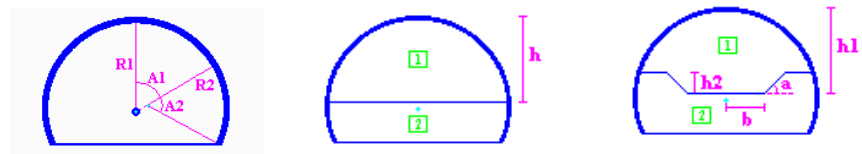
[Excavation Method]

Determine the excavation method for the tunnel section. FEA NX supports the section shapes for [Full Face Cut], [Bench cut 1], [Bench cut 2], [Ring cut 1], [Ring cut 2] and [3D cut]

►Full face cut

►►Bench cut 1

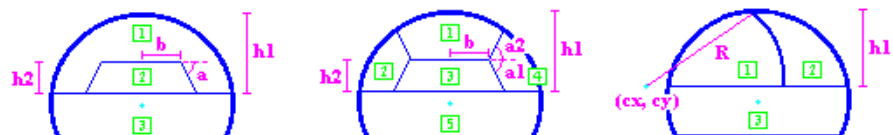
►►►Bench cut 2



►Ring cut 1

►►Ring cut 2

►►►CD cut

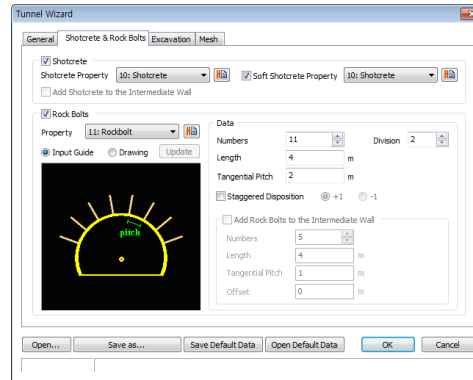




Shotcrete & Rock Bolts

Methodology

►Shotcrete & Rock bolts



The Shotcrete & Rock Bolts tab determines the generation of shotcrete and rock bolts, type and material, arrangement, shape, etc.

The generation of shotcrete, soft shotcrete and rock bolts can be set using the checkbox. Shotcrete and soft shotcrete are specified as plate elements and rock bolts are specified as embedded truss elements.

The material and properties are the same as the specified Material Property in Property/Coordinate System/Function. The properties of shotcrete & rock bolts must be specified as a structural property.

[Add Shotcrete to the Intermediate Wall]

If the 3D cut tunnel excavation method is activated, the installation of shotcrete on the intermediate wall can be decided.

[Rock Bolts]

Input the number, division length and spacing between rock bolts (Tangential pitch).

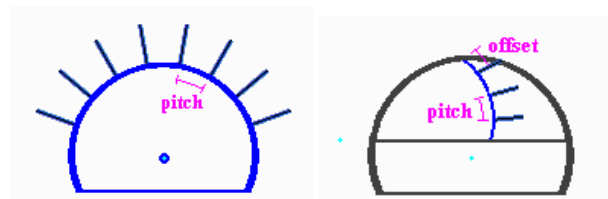
- Staggered Disposition

This option determines whether to place the rock bolts in an intersecting array for each construction stage. Setting +1 creates the set number of rock bolts in the first construction stage, an extra rock bolt is added in the second stage, and the number returns back to the set number in the third stage. Setting -1 creates the set number of rock bolts in the first construction stage, a rock bolt is removed in the second stage, and the number returns back to the set number in the third stage.

The [Add Rock Bolt to the Intermediate Wall] option is activated when the tunnel excavation method is set as 3D cut, and determines whether to install a rock bolt on the center wall.

►Rock bolt

►►Add rock bolt to center wall



[Input Guide] or [Drawing] can be used to check the drawn section shape in real time.



Excavation

Methodology

►Excavation

Tunnel Wizard

General | Shotcrete & Rock Bolts | **Excavation** | Mesh

Excavation Type
☒ One Dir. ☐ Both Dir.

1st Excavation Tunnel
☐ Left ☒ Right

Define Stages after 1st Excavation

Core [1]	Shotcrete Set	Rock Bolt Set
1	1	1

2nd Excavation Tunnel 1

Advancing Length
 Advances 30@2
 Total Length of Tunnel 60

	Adv.	Dist	Div	LD
1	2.0	2.0	1	...
2	2.0	4.0	1	...
3	2.0	6.0	1	...
4	2.0	8.0	1	...
5	2.0	10.0	1	...
6	2.0	12.0	1	...
7	2.0	14.0	1	...
8	2.0	16.0	1	...
9	2.0	18.0	1	...

Rock Bolt Location
☒ Auto(at Mid. Adv.) ☐ User

Pitches 1, 29@2

	Pit.	Dist.	Ang.
1	1.0	1.0	90.0
2	2.0	3.0	90.0
3	2.0	5.0	90.0
4	2.0	7.0	90.0
5	2.0	9.0	90.0
6	2.0	11.0	90.0
7	2.0	13.0	90.0
8	2.0	15.0	90.0
9	2.0	17.0	90.0
10	2.0	19.0	90.0

Open... Save as... Save Default Data Open Default Data OK Cancel

Determine the tunnel excavation for each construction stage.

Select the excavation type, whether to excavate in one direction or in both directions. For both directions, the program creates the construction stages until the tunnel is perforated. After this point, the user needs to create the construction stages directly.

When modeling 2 tunnels, specify the tunnel to be excavated first.

[1st Excavation Tunnel]

Input the stage spacing for shotcrete and rock bolts after excavation. For example, if 1 is inputted, the shotcrete or rock bolts are created in the subsequent stage after excavation. When creating soft shotcrete, the soft shotcrete is created in the specified stage and then stiffens in the next stage.

[2nd Excavation Tunnel]

Input the excavation stage for the second tunnel in the case of two tunnel construction. For example, if 2 is inputted, one tunnel is excavated and the next tunnel is excavated 2 stages later. The shotcrete and rock bolt creation point is determined by the specified value in Define Stages after the 1st Excavation.

[Advancing Length]

Specify the excavation length for each construction stage. Entering advancing length automatically calculates and displays the total tunnel length. The excavation length for each construction stage is input using commas or spaces. Repeating lengths can be input using number@length. For example, if the excavation is done for lengths of 2,2,2,2,3,4, input '2,2,2,2,3,4' or 4@2,3,4.

Divisions are the number of created elements in the excavation direction for each excavation stage. If loading is needed, click the button to input the load distribution factor for each stage.

[Rock Bolt Location]



Input the rock bolt creation location for each construction stage. Rock bolts can be automatically created at the center of the excavation length of each stage and the user can directly input the pitch and angle to adjust the creation location.

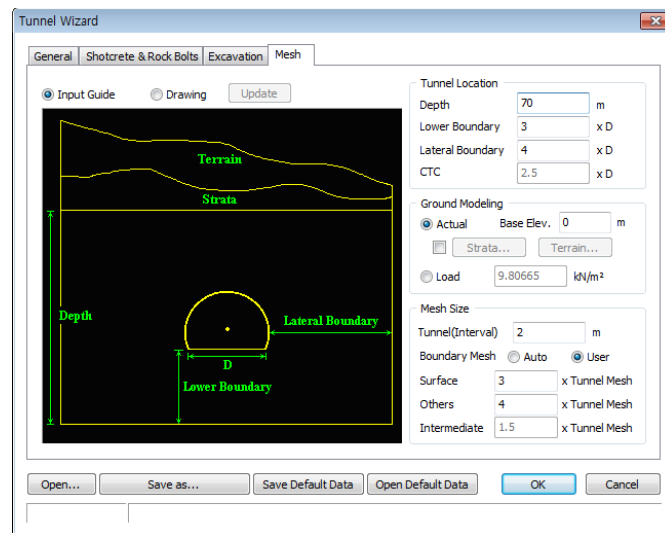
[Pitches] expresses the rock bolt creation location using the start portion of the tunnel and the rock bolt creation location of the previous stage.

[Angle] is the angle between the tunnel length direction and the rock bolt.

Mesh

Methodology

►Mesh



Input the mesh shape, strata and ground surface shape.

For the tunnel location, the model depth, the distance from tunnel floor to the lower boundary of the model, the distance from the tunnel to the left/right boundaries of the model, and the distance between tunnels for 2 tunnel construction can be input.

Each boundary distance is input as a multiple of the tunnel floor width.

[Ground Modeling]

Model the upper stratum. The upper stratum can be modeled using the [Actual] or [Load] methods.

The [Actual] method creates real meshes to conduct modeling. The [Load] method does not model the ground surface shape directly, but processes it as a pressure load.

The reference height is the [Base Elevation] on which the input values from the created stratum or terrain will be added.

[Strata]

The stratum exists above the upper part of the tunnel, and multiple strata can be created using the [New] button. The material and properties are the same as the specified material property in Property/Coordinate System/Function.

- x

Input the stratum shape following the model width direction. Input the width coordinates in the x direction. The origin is located at the bottom left corner of the tunnel as viewed from the front.

- Value



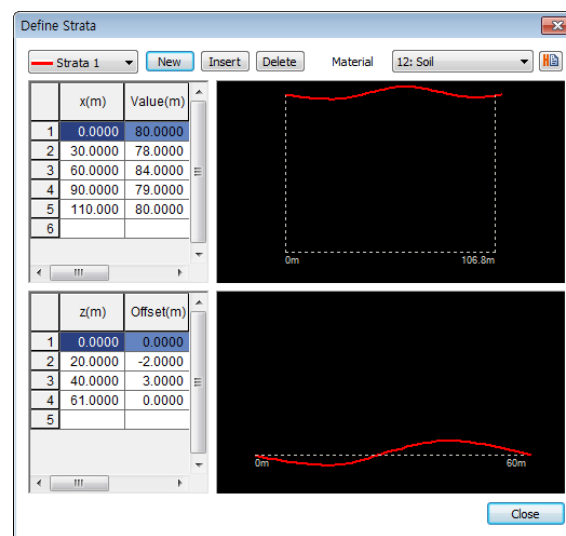
Input the height value at each x coordinate position. Input the x coordinate and value to draw the input shape on the right.

- Z

Input the stratum shape following the tunnel direction. Input the tunnel length direction coordinates in the z direction. The origin is located at the bottom left corner of the tunnel as viewed from the front.

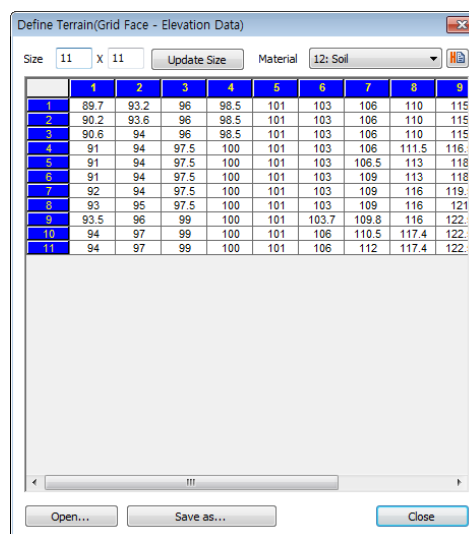
- Offset

Input the height value at each z coordinate position. The height value is the change in height respective to the previous value.



[Terrain]

Model the terrain. The terrain is generated using virtual grids identical to the grid face. The height is input at the grid intersections. The terrain height on a text file can be imported or the height can be input directly. The material and properties are the same as the specified material property in Property/Coordinate System/Function.



[Mesh Size]



Input the size of the mesh created in the tunnel. For the tunnel (Interval), the user can input the mesh size directly. The mesh boundary is input as a multiple of the created mesh size. Using the automatic setting sets the boundary element size automatically.

Open

Open a saved Tunnel Wizard file (*.wzd).

Save as

Save the entered values in current Tunnel Wizard as a Tunnel Wizard file (*.wzd).

Save Default Data

Set the current input values as the default values of the Tunnel Wizard.

Open Default Data

Delete all current input values and reset to the default values of the Tunnel Wizard.

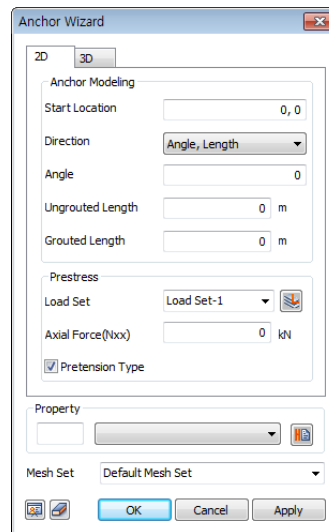
5.2

Anchor Modeling

►Anchor modeling Wizard

Overview

The Anchor Modeling Wizard creates a simple anchor model.



Methodology

Directly input the start position(Anchor head node position) or click from the work screen.

The anchor installation direction can be specified using [Angle,Length], [Relative dx, dy] or [Absolute x, y].

[Angle,Length] : Input the length and angle from the point entered in the previous stage. The angle is the rotation angle in the counterclockwise direction relative to the x axis of the work-plane.

[Relative (dx, dy)] : Input the relative distance from the point entered in the previous stage as 2D coordinates on the work-plane.

[Absolute (x,y)] : Input the absolute 2D coordinate value in the work-plane.

The anchor length can be classified as an ungrouted length or a grouted length. The ungrouted length always creates 1 element using the seed method and setting the 'number of divisions' as 1. The grouted length creates elements with a unit length using the seed method and setting the 'length spacing' as 1.

Prestress is the initial prestressing force of the anchor. A (+) value represents tension and A (-) value represents compression.

Checking the [Pretension Type] option does not create axial force loss in the a load activated construction stage. The created prestress can be registered by specifying the load set name.

Mesh set

Register the created anchor on the mesh set. The user can specify the name of the mesh set.

Overview

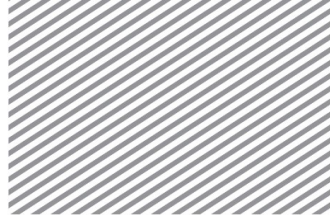
In engineering practice, soil parameters are obtained from one or more laboratory tests. In order to perform the best calculation, these soil parameters have to be translated into input parameters for the constitutive model used, taking into account the possibilities and limitations of the constitutive model. Most parameters for the constitutive models can be determined directly from standard laboratory tests such as triaxial test and oedometer test. However, due to the complexity of the models, it is recommended to not simply accept the parameters determined from those tests, but to actually model the tests and see if the parameters found actually give a proper representation of the real laboratory test results within the limits of the constitutive models. For this purpose, the Soil Test wizard is available with which in a simple manner laboratory tests can be simulated without the need for making a finite element model.

Methodology

5.3 Soil Test

► Soil Test Wizard

[illegible]



[Soil Test Set Name]

Define the set name of soil test to simulate.

[Method]

Select the test that will be simulated. The test options available are Tri-axial, Oedometer, CRS, DSS and General.

- **Tri-axial Test**

In a triaxial test, stress is applied to a sample of the material being tested in a way which results in stresses along one axis being different from the stresses in perpendicular directions. This is typically achieved by placing the sample between two parallel platens which apply stress in one direction, and applying fluid pressure to the specimen to apply stress in the perpendicular directions. With the triaxial test, it is possible to test soil properties while controlling the stresses applied in the vertical and horizontal directions relative to the specimen.

- **Oedometer Test**

Oedometer test is designed to simulate the one-dimensional deformation and drainage conditions that soils experience in the field. To simulate these conditions, rigid confining rings are used to prevent lateral displacement of the soil sample. Porous stones are placed on the top and bottom of the sample to allow drainage in the vertical direction.

With the oedometer test, it is possible to define a one-dimensional compression test for soil models.

- **CRS (Constant Rate of Strain) Test**

In a Constant Rate of Strain test (CRS test), it is possible to gradually apply a load to a soil model by increasing the displacement of a pressure cylinder at a constant rate.

In a CRS test, it is possible to gradually apply a load to a soil model by increasing the displacement of a pressure cylinder at a constant rate.

- **DSS (Direct Simple Shear) Test**

The DSS test can be used to determine the shear strength of a soil model.

- **General Test**

The general soil test can be used as a customisable soil test mode. The user can define arbitrary stress and strain conditions.

[Material]

Define the soil material properties.

[Initial Stress]

Define the initial stress. In case of 'General' type, it is possible to define the stresses of each direction.

[Boundary Condition]

The boundary condition is decided by the test method. In case of 'General' type, it is possible to restrain the stress / strain conditions of each direction. This is an initial condition which cannot be changed to the construction stages (applies to all stages).

[Stage Name]

Define the different stages of the test.

[Advanced Setting]

Define the details for analysis.

[Analysis]

Start the calculation for the selected soil tests.

[Show Graph]



The results of the test are displayed in the predefined diagrams in the graph window.

Stage

Methodology

► Stage

The 'Advanced Setting' dialog box has three tabs: 'Stage', 'Analysis', and 'Output'. The 'Stage' tab is active, showing a table with the following data:

	Stage Name	Inc.	Time(day)	EZZ
1	Stage-1	100	0	0

At the bottom of the dialog are 'OK' and 'Cancel' buttons.

Each stage is defined by a number of steps (Inc.), a duration (in units of time) and a applied stress or strain increments. The given stress or strain increment will be reached at the end of the given duration in the given number of steps. A negative stress or strain increment means additional compression, whereas a positive stress or strain increment means unloading or tension.

Analysis

Methodology

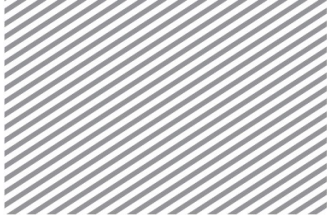
► Analysis

The 'Advanced Setting' dialog box has three tabs: 'Stage', 'Analysis', and 'Output'. The 'Analysis' tab is active, showing 'Convergence Criteria / Error Tolerance' settings:

- ☒ Stress: 0.001
- ☒ Strain: 0.001

At the bottom of the dialog are 'OK' and 'Cancel' buttons.

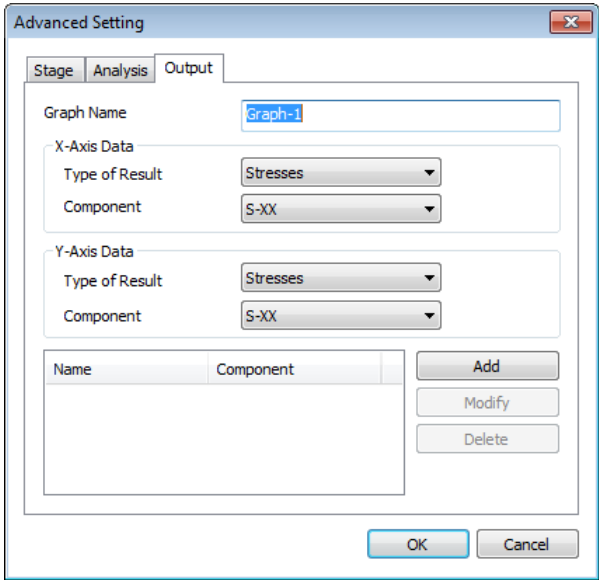
Define the convergence criteria of analysis.



Output

Methodology

► Output



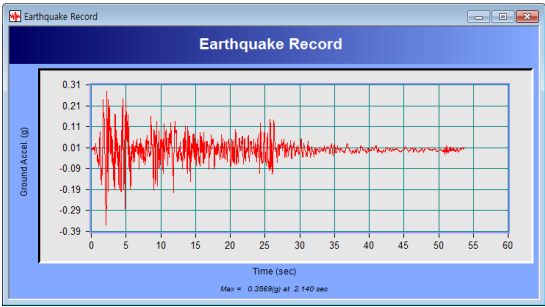
Define the graph setting. You can select the configuration of diagrams to display.

5.4
Seismic Data
Generator

Overview

Construct the earthquake acceleration, earthquake response spectrum and design response spectrum using the earthquake wave database built into FEA NX.

► Seismic data generator



Methodology

[File]

Save the data created on the SGS in various forms, output or import existing data.

[Generate]

Construct the earthquake acceleration, earthquake response spectrum and design response spectrum using the earthquake database built into the SGS.

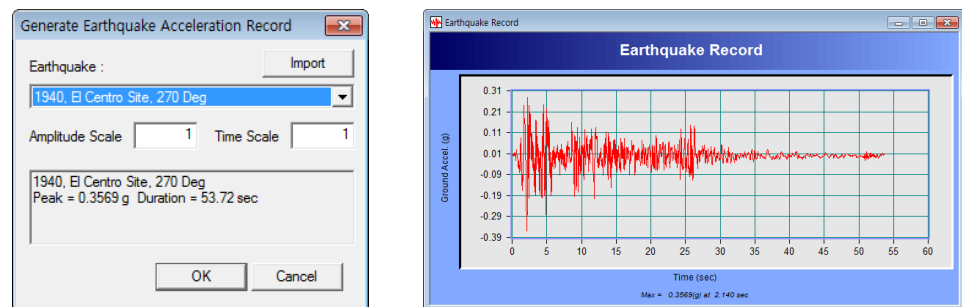
[Earthquake Record]

Display the earthquake wave data on the graph.

Select the earthquake wave, input the amplitude and time scale and click the [OK] button.

The created data is the normalized acceleration. The data time spacing is 0.02 seconds for North American earthquake waves and 0.01 seconds for Japanese earthquake waves. Other earthquake wave data on a '*.dbs (SGS dbase)' file can be used by clicking the [Import] button.

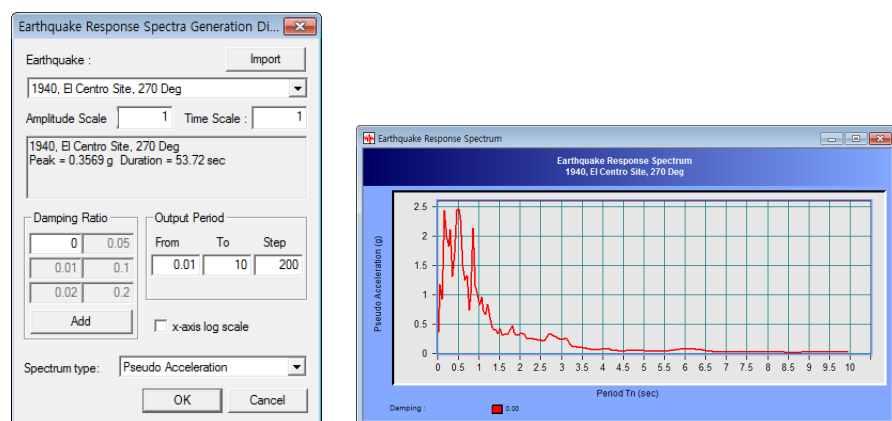
►Example



Earthquake Response Spectra

Display the earthquake response spectrum as a graph, using the earthquake wave data in the database or from the user defined earthquake wave data.

1. Select the earthquake wave and input the amplitude and time scale. Import other earthquake wave data on a '*.dbs (SGS database)' file by clicking the [Import] button.
2. Input the damping ratio. To output multiple damping ratio graphs simultaneously, click the [Add] button.
3. Input the period range and spacing of graph output on the output period.
4. Select the spectrum type.
5. Check the [X-axis log scale] to set the x axis of the spectrum in log scale.



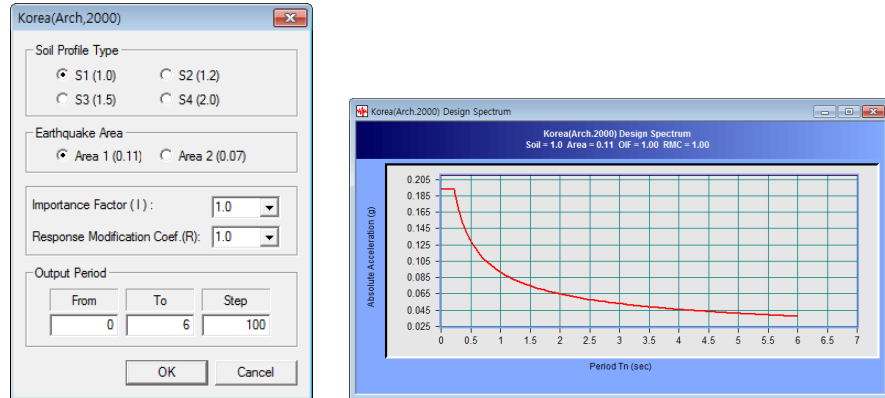
Design Response Spectra

Create a Design Response Spectra and display as a graph.

Input the data necessary for spectra creation on the dialog box, input the spectrum period range and click the [OK] button to display the graph.



►Example



Option

Set various options for the graph. The graph x axis and y axis can be displayed in log scale. The scale marks can be displayed. The thickness of the graph line can be modified and the graph title can also be modified.

View

Change the show/hide status of the tool bar or status bar. Also, the Zoom function for the graph and the Time Domain and Frequency Domain of the graph can be modified.

Zoom Out All

Return from the zoomed state to the original state.

To magnify a section of the graph, drag and select the desired section using the left mouse button. The right mouse button can be used to return the view state to the original state.

[Time<->Frequency]

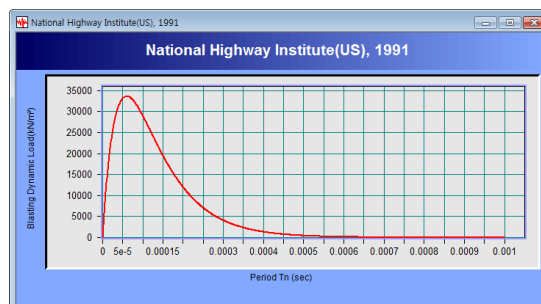
Convert the graph x axis from the time domain to the frequency domain and vice versa.

5.5 Dynamic Load Data Generator

►Dynamic load data
generator

Overview

Create a blast load function from the suggested blast load formula, and create a railway load function from the railway load database.



Methodology

File

Save the data created using the DGS in various forms, or import existing data.

[Unit System]

Specify the force and length units of the dynamic load data. The unit system must be specified before creating the dynamic load data.

Blasting Dynamic Load

Select a suggested blasting load equation type, suggested by various institutions. The institution, suggested equation and corresponding units are as follows:

Suggested by	Suggested equation	Used units
National Highway Institute(US), 1991	$P_d = \frac{4.18 \times 10^{-7} \times S \times V^2}{1 + 0.8 \times S}$	PD : kbar V : ft/sec S : g/cm3
International Society of Explosive Engineers, 2000	$P_d = 2.5 \times S \times V^2 \times 10^{-6}$	PD : kbar V : m/sec S : g/cm3
Johannson & Persson,. 1973	$P_d = 2.1 \times (0.36 + S) \times V^2$	PD : kbar V : km/sec S : g/cm3
Jones & Hino, 1974	$P_d = 0.000424 \times V^2 \times S \times (1 - 0.543 \times S + 0.193 \times S^2)$	PD : g/cm2 V : cm/sec S : g/cm3
Liu & Tidman, 1995	$P_d = 1.62 \times (S \times V^2) \times \left(\frac{D \times C}{S \times V}\right)^{0.25}$	PD : kbar V : km/sec S : g/cm3
Atlas powder company, 1987	$P_d = 2.325 \times 10^{-7} \times S \times V^2$	PD : kbar V : ft/sec S : g/cm3

When calculating the blast pressure, use the $P_B = \left(\frac{dc}{dh}\right)^2 \times P_D$ equation to consider the effects of decoupling. Depending on whether you are considering the effects of the blast periphery length, use $P' = P_B \times W \times 2\pi \times \frac{dh}{L}$ or $P' = P_B \times W$ to calculate the value.

The final computed blast function can be calculated using the equation suggested by Starfield and Pugliese (1968) that considers the Window Function:

$$P(t) = 4 \times P' \times \left\{ \exp\left(\frac{-B \times t}{\sqrt{2}}\right) - \exp(-\sqrt{2} \times B \times t) \right\}$$

Parameter

[Blasting Velocity(V)] : Input the blasting velocity. The fixed unit can change depending on the suggested equation.



[Explosive Density(S)] : Input the density of explosives. The units are fixed as [g/cm³].
[Charge Diameter(dc)] : Input the explosive diameter. The units are fixed as [mm].
[Borehole Diameter(dh)] : Input the diameter of the borehole. The units are fixed as [mm].
[Maximum Charge Amount (W)] : Input the maximum charge per delay. The units are fixed as [kg].
[Load Factor(B)] : Input the load coefficient.
[Sound Velocity in Rock(C)] : Input the ground sound velocity. It is only considered in the 'Liu & Tidman, 1995' equation and the units are fixed as [km/sec].
[Rock Density(D)] : Input the ground density. It is only considered in the 'Liu & Tidman, 1995' equation and the units are fixed as [g/cm³].

Time

[End time] : Input the final time for blast load creation. The units are fixed as [sec].
[Time Increment] : Input the time increment for blast load creation. The units are fixed as [sec].

Graph option

[X-axis log scale] : Output the X axis in log scale.
[Y-axis log scale] : Output the Y axis in log scale.

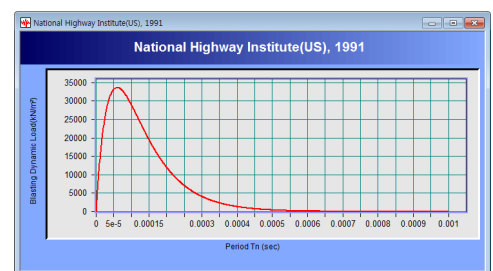
Equivalent Transform Blasting Load

Consider the equivalent blasting load. (When considering, refer to the content above)

Blasting Hole Perimeter(L)

Input the blast periphery length. The units are fixed as [mm].

►Example



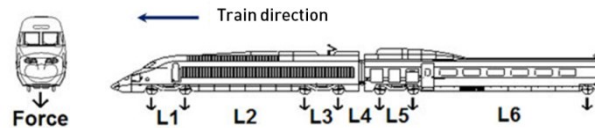
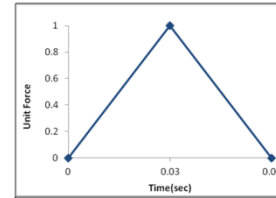
Train Dynamic Load

[Name]: Define name.
[Train Type] : User can directly input the distance/force depending on the number of wheels.
[Number of Wheels] : Stands for the number of wheels on the train. It is the same as the added number on the table.
[Train Velocity] : Input the velocity of the train. The units are fixed as [km/hr].
[Element Size] : Input the size of the element which the train passes through. The units are fixed as [m].
The input element size and train velocity can be used to compute the arrival time when constructing the influence line function. For railway moving load, the load is applied gradually to the node as the train arrives and decreases as the load is removed, hence creating a triangle shaped load function form.



**ex] Train velocity
Element size**

nb.	Time	Load Factor
t1	0.03	1



[Table] : Input the dynamic load of the train. Typical domestic train loads are provided in the database and the load can also be input by the user defined load (Number of wheels/Length/Force).

- Number : Input the wheel number. The number starts from 1 and the wheel spacing and force are input sequentially.
- Length : Input the wheelbase. For the first starting wheel, the length is input as 0.
- Force : Input the axial load acting on the wheels. The axial load can be divided into 2 and used as a working load.

[Scaling] : Input the gradient modulus of the railway dynamic load.

- Scale factor : Input the gradient modulus.
- Maximum value : Modify the railway dynamic load data such that the maximum load value becomes the specified desired value.

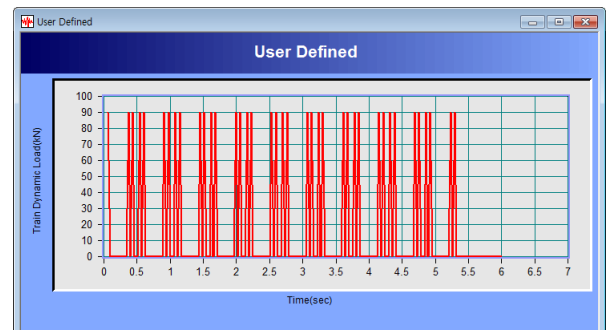
[Start time] : Input the start time of the railway load. The default value is 0 sec.

[Import] : Import a railway dynamic load in MS-EXCEL format. It is only activated when the train type is set to User Defined.

►Train dynamic load
example

No	Length(m)	Force(KN)
1	0	220
2	1.854	220
3	1.854	220
4	8.789	220
5	1.854	220

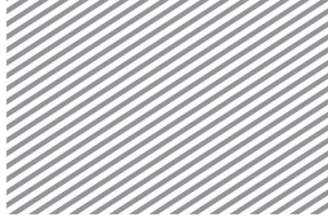
►Example



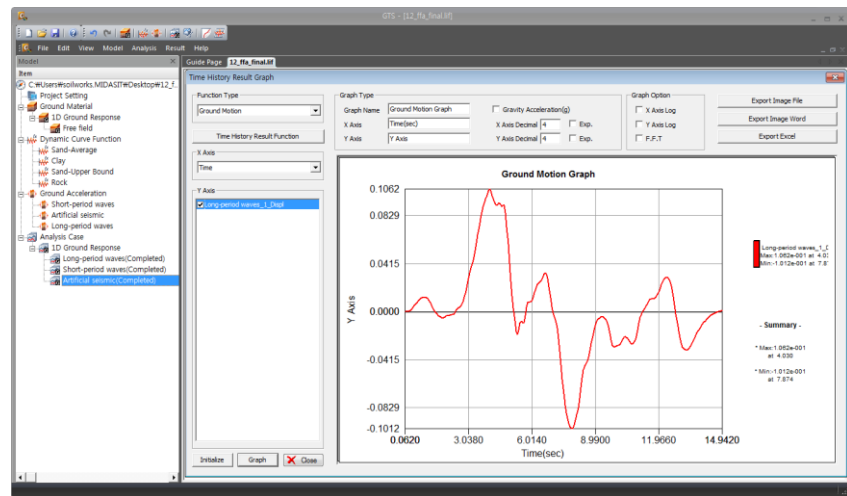
5.6 Free Field Analysis

Overview

1D Ground Response Analysis, also called Site Response Analysis, is a program used to analyze the in-situ response of the ground for earthquake inputs before excavation or construction.



►1D Ground response analysis



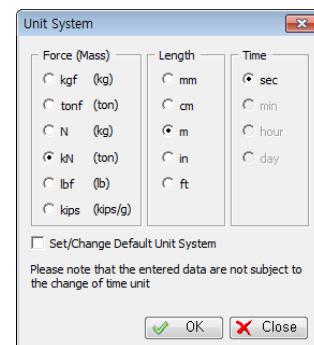
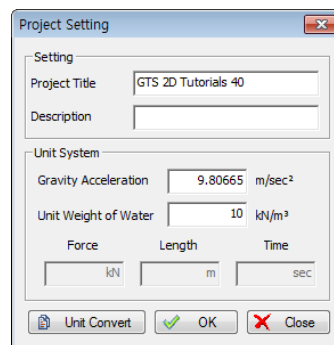
Methodology

First set the project and then input the strata information and material properties, dynamic material property function and ground acceleration function onto the model. Later, create an analysis case and conduct the analysis for result analysis.

File

Set the basic information of the project. Because the analysis results are affected by the values set in Project Setting, the settings need to be accurately set before analysis. It is best if the Project settings are pre-set when starting a new project, but it is fine if the settings are changed during modeling.

- Project setting
- Convert unit system



Set the unit system applied in the project. Click the [Unit Convert] button to set a different unit system. The time unit settings are not changed.

Edit

Return to the immediate previous state using undo, or use redo to return to the state before the command execution.

Model : Strata modeling

Overview



Create a ground model for the free field analysis. Input the ground material property data for each stratum to create a strata property.

►Strata modeling

No.	Depth (m)	Thickness (m)	Layer No.	Unit Weight (kN/m³)	Vs (m/sec)	G0 (kN/m²)	H0	Dynamic Curve Type	Output Motion Type
1	2.0000	2.00000	1	18.00000	155.000	44097.6	0.01000	1:Sand-Average	Within
2	4.0000	2.00000	2	18.00000	155.000	44097.6	0.01000	1:Sand-Average	Within
3	5.0000	1.00000	3	18.00000	155.000	44097.6	0.01000	2:Clay	Within
4	8.0000	3.00000	4	19.00000	390.000	294688.	0.01000	1:Sand-Average	Within
5	11.000	3.00000	5	19.00000	390.000	294688.	0.01000	1:Sand-Average	Within
6	14.000	3.00000	6	19.00000	390.000	294688.	0.01000	1:Sand-Average	Within
7	17.000	3.00000	7	19.00000	390.000	294688.	0.01000	1:Sand-Average	Within
8	20.000	3.00000	8	20.00000	640.000	835352.	0.01000	3:Sand-Upper B	Within
9	22.000	2.00000	9	20.00000	640.000	835352.	0.01000	3:Sand-Upper B	Within
10	24.000	2.00000	10	20.00000	640.000	835352.	0.01000	3:Sand-Upper B	Within
11	27.000	3.00000	11	24.00000	980.000	2350410	0.01000	4:Rock	Within
12	30.000	3.00000	12	24.00000	980.000	2350410	0.01000	4:Rock	Within

Methodology

Click the Input ground material property data button to input the strata material properties. (The number of stratum is limited to a maximum of 50.)

[Number] : Automatically assigned identification number that starts from 1.

[Depth] : Input the depth of a stratum. The depth value needs to be larger than 0 and is applied cumulatively. Entering the depth automatically calculates the stratum thickness.

[Unit weight] : Input the unit weight of the stratum.

[Vs(m/sec)] : Input the shear wave velocity of the ground.


[G0(kN/m²)] : Input the maximum shear modulus.

[H0] : Input the initial damping ratio of the ground.

[Output Motion type]

- **Outcropping**
Applied to strata that outputs the analysis results in outcrop form.
- **Within**
Applied to strata that outputs the analysis results as a ground response.

[Dynamic Curve Function]

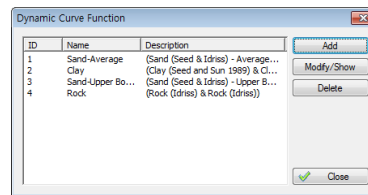
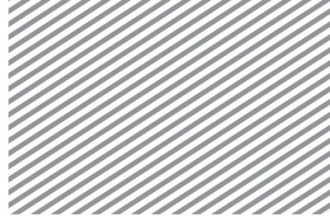
Click  to select or create the shear modulus and damping ratio function to consider the non-linear, non-elastic behavior of the ground. (For shear strain, refer to the dynamic material property function.)

Model : Dynamic Curve Function

Overview

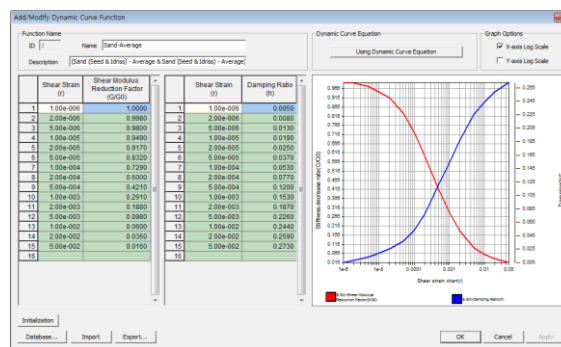
Define the shear modulus and damping ratio function depending on the shear strain to consider the non-linear, non-elastic behavior of the ground.

►Dynamic curve function



Add/Delete/Modify

Define a new dynamic material property function, or check, modify or delete the existing input data.



Function Name

[ID] : Define the function ID number.

[Name] : Input the function name.

[Description] : Display a simple description that explains the shear strain function.

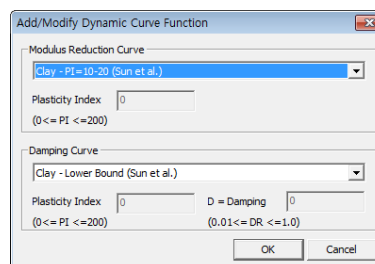
Initialization

Conduct the reset function on the entered data.

Database

Import the database suggested by previous researchers.

►Add/Modify dynamic curve function



Add/Modify Dynamic Curve Function

The database supported on the FFA (Free Field Analysis) is shown on the following table.

Classification	Database
Shear modulus damping ratio curve	Clay - PI=10-20 (Sun et al.) Clay - PI=20-40 (Sun et al.) Clay - PI=40-80 (Sun et al.) Clay - PI=5-10 (Sun et al.) Clay - PI=80+ (Sun et al.) Clay (Seed and Sun 1989)



	Gravel (Seed et al.) Linear Rock Rock (Idriss) Sand (Seed & Idriss) - Lower Bound Sand (Seed & Idriss) - Average Sand (Seed & Idriss) - Upper Bound Sand (Seed and Idriss 1970) Vucetic - Dobry
Damping ratio curve	Clay - Lower Bound (Sun et al.) Clay - Average (Sun et al.) Clay - Upper Bound (Sun et al.) Clay (Idriss 1990) Gravel (Seed et al.) Linear Rock Rock (Idriss) Sand (Idriss 1990) Sand (Seed & Idriss) - Lower Bound Sand (Seed & Idriss) - Average Sand (Seed & Idriss) - Upper Bound Vucetic - Dobry

Import

Import the saved data file.

Export

Save the entered data as a file.

Using Dynamic Curve Equation

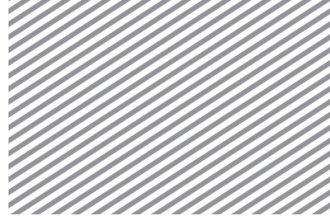
Define the material property values from the various existing databases. The dialog boxes below represent the Japan Public Works Research Institute (JPWRI) equation, Port facility technical standard, Yasda's method, Database according to the liquefaction manual, Japan Building Standard Law no.1457 and Yamade's equation.

►PWRI equation

►►Method of the technical standards of port facilities

The dialog box titled "Using Dynamic Curve Equation" has a dropdown menu set to "Public Works Research Institute". Under the "Stratum" section, there are three radio button options: "Alluvial Cohesive Soil" (selected), "Diluvial Cohesive Soil", and "Alluvial Sandy Soil, Diluvial Sandy Soil, Sandy Gravelly Soil". Below this, there are two input fields: "Mean Principal Effective Stress (σ'_m)" with a value of 0 and unit "Kgff/cm2", and "Maximum Damping Ratio (η_{max})" with a value of 0. At the bottom are "OK" and "Cancel" buttons.

The dialog box titled "Using Dynamic Curve Equation" has a dropdown menu set to "Method of the technical standards of port facilities". Under the "Plasticity Index(I_p)" section, there are three radio button options: " $I_p < 9.4$ " (selected), " $9.4 \leq I_p < 30$ ", and " $30 \leq I_p$ ". Below this, there is one input field: "Mean Principal Effective Stress (σ'_m)" with a value of 0 and unit "Kgff/cm2". At the bottom are "OK" and "Cancel" buttons.



- ▶ Asuda's method
- ▶▶ Liquidal manual

Using Dynamic Curve Equation

Dynamic Curve Equation: Asuda's Method

Mean Principal Effective Stress (σ'_m) Kg/cm²
(0.2 ≤ σ'_m ≤ 3.0 Kg/cm²)

Particle (D₅₀) [mm]
(0.02 ≤ D₅₀ ≤ 1.0 mm)

OK Cancel

Using Dynamic Curve Equation

Dynamic Curve Equation: Liquidal Manual

Stratum:
☒ Clay
☐ Sandy Soil

OK Cancel

- ▶ The building standard law of Japan, no.1457
- ▶▶ Yamade's equation

Using Dynamic Curve Equation

Dynamic Curve Equation: The Building Standard Law of Japan, No. 1457

Stratum:
☒ Clay
☐ Sandy Soil

OK Cancel

Using Dynamic Curve Equation

Dynamic Curve Equation: Yamade's Method

Stratum:
☒ Clay Plasticity Index(I_p) 0 ~ 30 %
☐ Sandy Soil Confining Pressure(σ_0) 0 ~ 5

OK Cancel

Model : Ground acceleration function

Overview

Input the time varying load for earthquake analysis on the input ground.

- ▶ Ground acceleration function

Ground Accel. Function

Function Name	Function Type	Data Type
Short-period waves	Time	Norm.Accel.
Artificial seismic	Time	Norm.Accel.
Long-period waves	Time	Norm.Accel.

Add Acceleration
Modify/Show
Delete
Close

Add/Modify/Delete Acceleration

Input a new acceleration data, or check, modify or delete the existing input data.

Add/Modify/Show Ground Acceleration Function

Function Name: Long-period waves

Import Earthquake

	Time (sec)	Function
1	0.0620	0.0037
2	0.1240	0.0047
3	0.1860	0.0066
4	0.2480	0.0085
5	0.3100	0.0077
6	0.3720	0.0050
7	0.4340	0.0033
8	0.4960	0.0006
9	0.5580	-0.0030
10	0.6200	-0.0029
11	0.6820	-0.0028
12	0.7440	-0.0055
13	0.8060	-0.0165
14	0.8680	-0.0035
15	0.9300	-0.0071
16	0.9920	0.0000

Scaling:
☒ Normalized Accel. ☐ Acceleration
☒ Scale Factor 1
☐ Maximum Value 0 g

Gravity: 9.80665 m/sec²

Graph Options:
☐ X-axis log scale
☐ Y-axis log scale
☐ F.F.T

Desc: Generate Earthquake Response Spectrum... Export... OK Close Apply

Function Name

Selected the input time history analysis condition. If the condition is not input beforehand, press the button to call up this function.

Time function data type

[Normalized acceleration]: This spectrum is found by dividing the acceleration spectrum with the gravitational acceleration, and it cannot be set as another type.

[Acceleration]: The acceleration spectrum with time.

Scaling

Input the gradient modulus of the ground acceleration function.

[Scale factor] : Input the gradient modulus.

[Maximum value] : Modify the spectrum data such that the maximum acceleration becomes the specified desired value.

Gravity

Input the gravitational acceleration.

Graph Options

Specify whether to express the direction of each axis of a graph in log scale.

[x-axis log scale]

Specify whether to express the x-axis direction of a graph in log scale.

[y-axis log scale]

Specify whether to express the y-axis direction of a graph in log scale.

[F.F.T]

Specify whether to convert the graph based on Fourier Transformation.

Description

Display a simple description that explains the ground acceleration function. If an [Earthquake] is created, the maximum acceleration, time etc. used to create the wave is displayed.

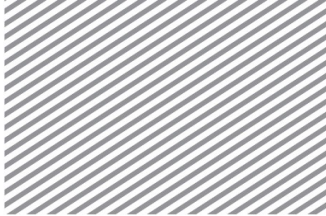
Export

Export the ground acceleration function as a text(*.txt) file.

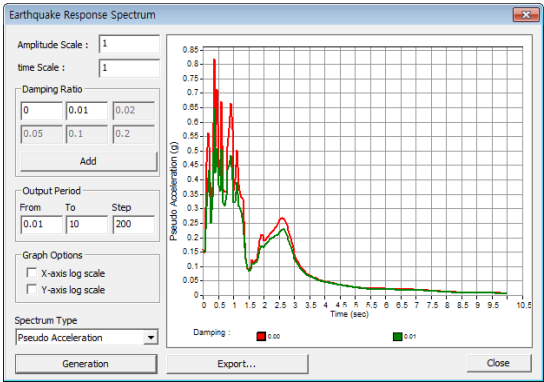
Earthquake Response Spectrum**Overview**

Calculate and display the graph of the earthquake response spectrum using the earthquake wave data in the database or from the user defined earthquake wave data. Input the damping ratio. To output multiple damping ratio graphs simultaneously, click the [Add] button. Input the period range and spacing of graph output for the result stage.

Select the spectrum type. Check the X-axis log scale and Y-axis log scale item to change the X-axis, Y-axis of the spectrum to a log scale.



►Table. 'fn.sgs' file format



The FFA provides the following 3 types of time varying load input methods for the convenience of the user:

- 1) Method of saving the frequently used time varying load as a file and importing
- 2) Method of calling up the time varying load from the database
- 3) Direct user input method

The earthquake load input is only supported for the value obtained by dividing the time history acceleration by the gravitational acceleration.

1. Method of saving the frequently used time varying load as a file and importing
[Import]

Used to import a time varying load from an existing data. The data format is '*.sgs' or '*.thd' and the file is created in the following format:

*SGSw	States that this file is in the data format of the FEA NX automatic earthquake data extraction module "Seismic Data Generator".
*TITLE, Elcentro 1940, N-S	-
Component	
*X-AXIS, Second	-
*Y-AXIS, Normalized Acceleration	-
*UNIT&TYPE, GRAV, ACCEL	-
*FLAGS, 0, 0	-
*DATA	-
1.00000E-010, 3.50102E-001	-
5.00000E-002, 3.82861E-001	-
1.00000E-001, 5.08226E-001	-
1.50000E-001, 5.17459E-001	-
:	-

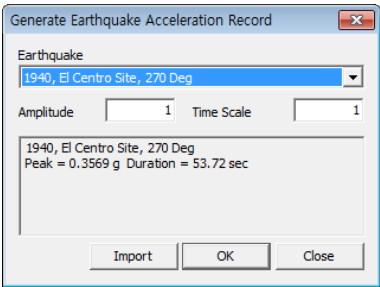
►Table. 'fn.thd' file format
-: User input

Selective item	** Annotation– Can be entered anywhere
-	*UNIT, M , N - Length : Available for MM, CM, M, INCH, FEET, GRAV
	*Load : KG, TON, KN, LBF, KIP
-	*TYPE, ACCEL – Available for ACCEL, FORCE, MOMENT
Essential item	*Data
-	X1 , Y1 (X : Time, Y : Time Function)
-	X2 , Y2
-	X3 , Y3
-	:



- 2. Method of calling up the time varying load from the FEA NX database [Earthquake wave]
Create a time varying load by importing various earthquake data from the database. There are a total of 32 types of earthquake acceleration in the DB.

►Generate earthquake acceleration record



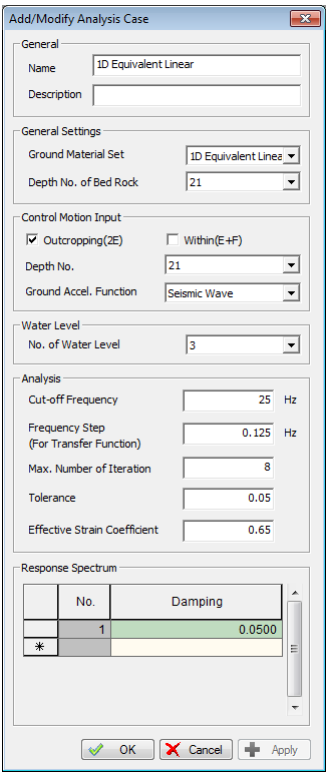
- 3. Direct user input method
Construct the time varying load by directly entering the time and the corresponding time varying load value in the left input column on the dialog box.

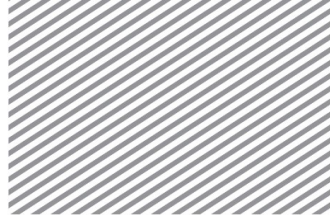
Analysis

Overview

Create an analysis case. Add a new analysis case, or modify, copy, delete an existing analysis case. The input window for analysis case creation is as follows:

►Create analysis case



**General**

[Name] : Input the analysis model name.

[Description] : Input the description for the analysis model.

General Setting

[Ground Material Set] : Select the modeled stratum to be used for analysis.

[Depth No. of Bed Rock] : Select the stratum number corresponding to the bedrock.

Control Motion Input

[Outcropping(2E)] : Set the ground acceleration as an outcrop state.

[Within(E+F)] : Set the ground acceleration as the ground response.

[Depth No.] : Select the control point for the earthquake wave input stratum.

[Ground Acceleration Function] : Select the input earthquake acceleration time varying load.

Water Level

[No. of Water Level] : Stratum ID inputted in Ground Material can be selected, and it calculates assuming that the water level exists from the selected stratum.

Analysis

[Cut-off Frequency] : Set the maximum frequency range for frequency analysis.

[Frequency Step (for Transfer function)] : Set the calculation frequency spacing for transfer function analysis.

[Maximum Number of Iteration] : Input the maximum iterative calculations to find the equivalent linear material property value.

[Tolerance] : Input the tolerance for the shear modulus and damping ratio used to find the equivalent linear material property value through iterative calculations.

[Effective strain coefficient] : Input the coefficient needed to calculate the effective shear strain from the maximum shear strain.

Response spectrum

[Damping] : Input the damping ratio to calculate the response spectrum.

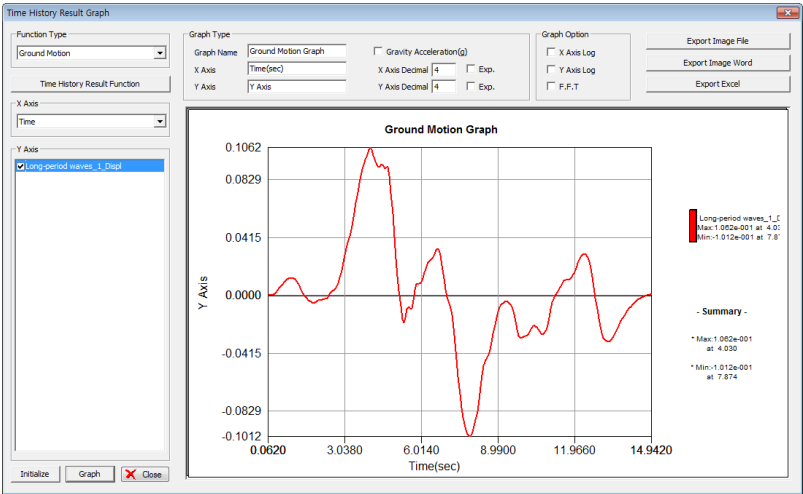
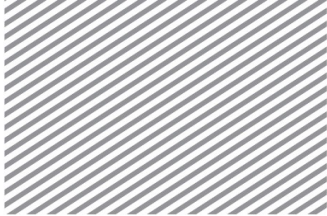
Result

The analysis results are output in a graph or table for each depth.

- Converging results and table (Absolute) (Maximum acceleration, Maximum velocity, Maximum displacement)
- Converging results and table (Relation) (Maximum acceleration, Maximum velocity, Maximum displacement)
- Strain/Stress result table (Uniform strain, Maximum strain, Maximum shear strain)
- Soil profile result table (Converging damping ratio, Converging shear modulus, Shear modulus ratio)
- Soil profile table (Shear wave velocity, Shear modulus, Damping ratio)

Result : Time History Result

Select the function type of the output graph. The function types are ground movement, response spectrum, stress/strain, and transfer function.



[Time History Result Function]
Select the function data, strata, result type of the output graph. Each result type can be selected depending on the ground movement result function type.

Ground Motion Result Function Type	Result type
Ground Motion	Displacement, Velocity, Acceleration, Relative displacement, Relative Velocity, Relative acceleration
Response Spectrum	Relative displacement, Relative pseudo-velocity, Relative velocity, Absolute pseudo-acceleration, Absolute acceleration
Stress/Strain	Stress, Strain
Transfer Function	Transfer function

The selected result function is registered on the y axis.
The graph name, x axis name and y axis name can be defined for the time history graph in Define Graph, and the values can also be expressed in exponential form.
The x axis, y axis can be represented in log scale or the F.F.T can be set in Graph Options.
The generated graph is displayed in the dialog window and can be exported in many forms such as image file, image word file or excel file.

5.7 Artificial Earthquake Generator

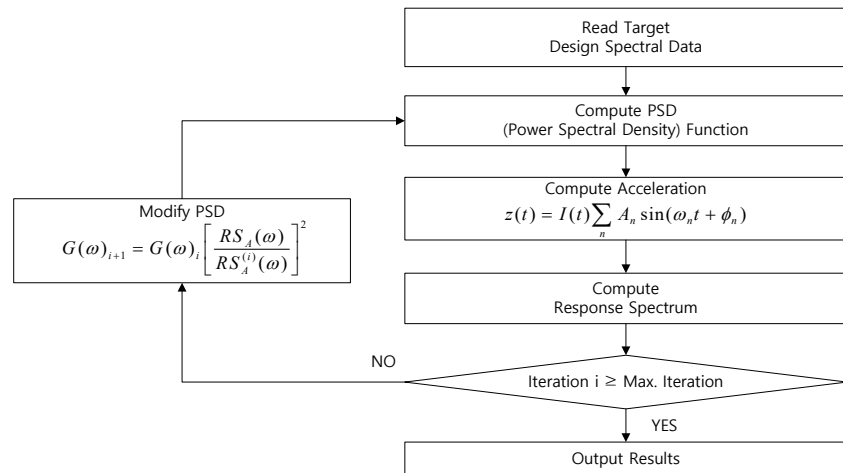
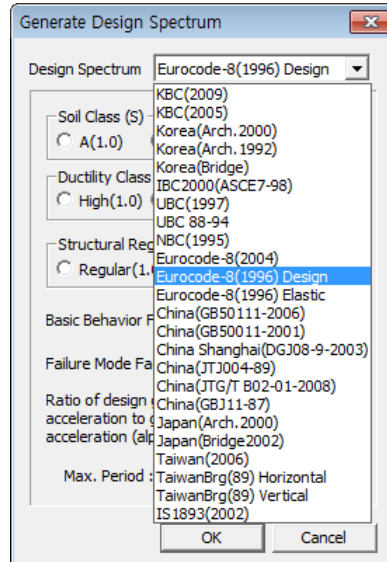
Overview

Generate artificial earthquake data from the embedded design spectral data. Envelope Function enables to generate transient earthquake data. There are three types of envelope functions : Trapezoidal, Compound and Exponential. FEA NX supports Trapezoidal Type.



Methodology

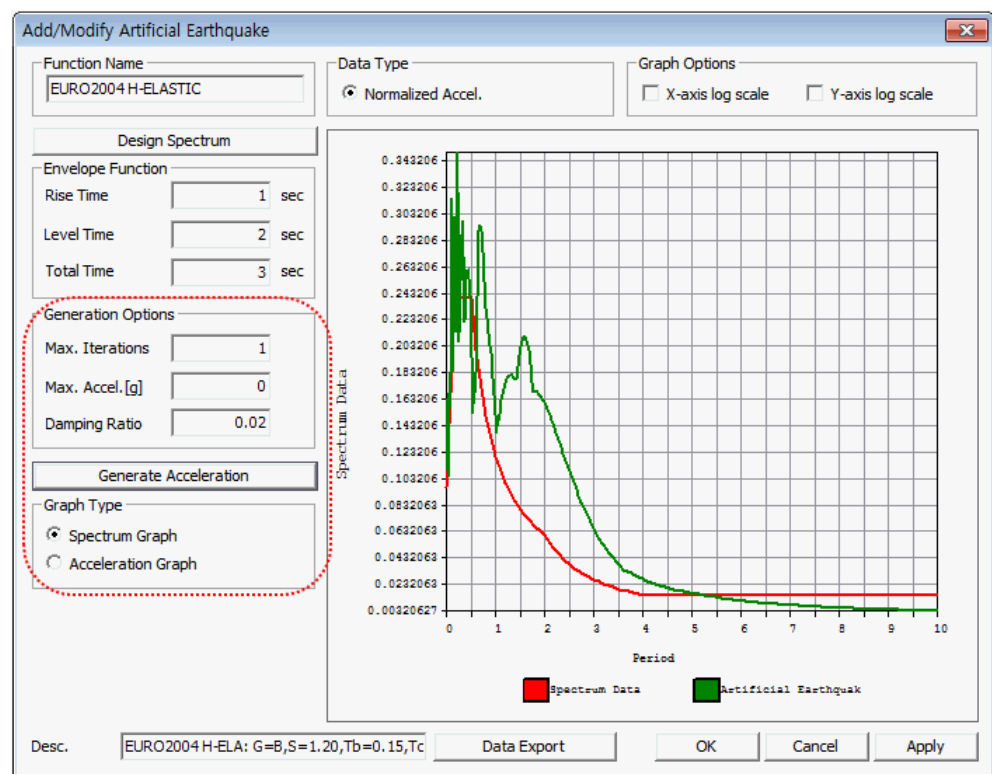
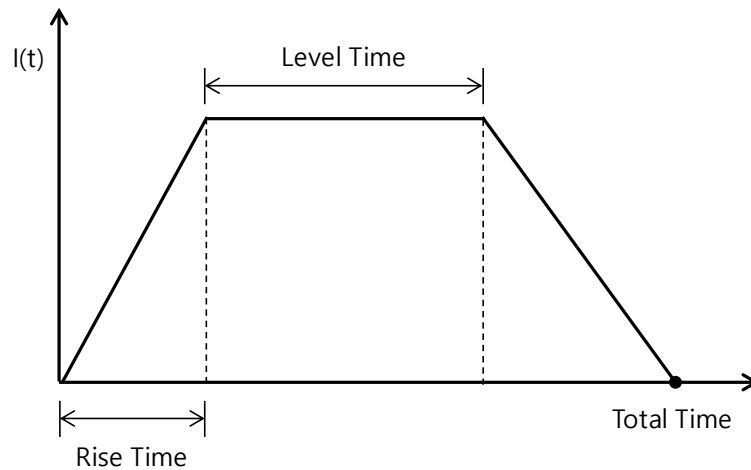
►Design Spectral Data



$$z(t) = I(t) \sum_{n=1}^N A_n \cos(\omega_n t + \phi_n)$$

where, ω_n = Frequency, A_n = Amplitude, Φ_n = Phase Angle, and $I(t)$ = Envelope Function

►Envelope Function



[Generate Options]

- Max Iterations : Maximum number of iterations to fit computed spectral data to target one.
- Max. Acceleration : Maximum acceleration of artificial earthquake data
- Damping Ratio : Damping ratio to calculate spectral data

[Generate Acceleration]: Convert from response spectrum to acceleration data

- Spectrum Graph : Check results based on spectral data
- Acceleration Graph : Check results based on acceleration data