



Chapter 8

3D Embankment Consolidation

Workflow

Section 1	Overview	1
1.1	Learning Purpose	1
1.2	Model and Analysis Summary	2
Section 2	Analysis Setting	3
2.1	Starting with Analysis Setting	3
Section 3	Material and Property	4
3.1	Definition of Ground and Structural Materials	4
3.2	Definition of Property	7
Section 4	Modeling	8
4.1	Modeling Geometry	8
4.2	Generate Meshes	11
Section 5	Analysis	15
5.1	Setting Boundary Conditions	15
5.2	Define Construction Stages	19
5.3	Setting Analysis Case	25
5.4	Perform Analysis	26
Section 6	Results	27
6.1	Verify Displacement	27
6.2	Verify Stress	30
6.3	Verify Effect of Adjacent Structure	32



3D Embankment Consolidation

Section 1

Overview

1.1 Learning Purpose

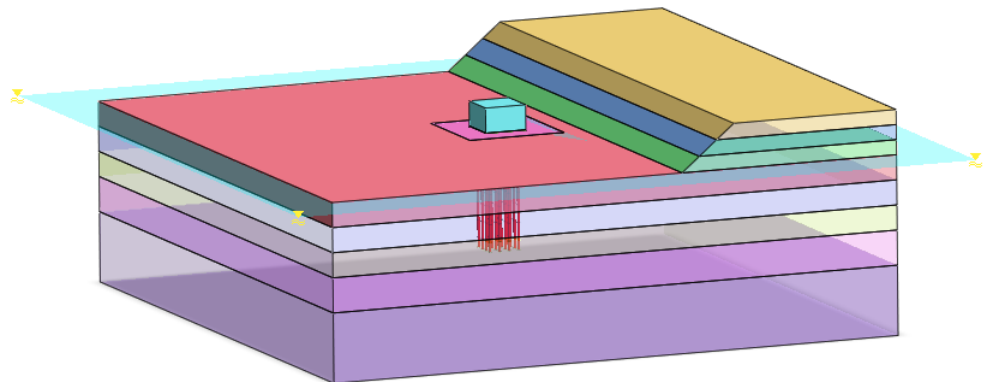
Through the consolidation analysis of a 3D construction stage, verify the ground deformation (settlement) and stress changes over time. Additionally, apply adjacent structures and bottom pile elements. Verify the deformation of the structure influenced by ground settlement.

In consolidation, the excess pore water pressure inside the ground generally decreases over time, causing the effective stress to increase. The water level is located at the top of the weak stratum, which has relatively lower permeability.

When embankment loading is applied, the water, which initially bears the load due to undrained conditions, will slowly seep out and transfer the load to the soil particles. During this process, settlement occurs in the direction of gravity, and this settlement will affect the stability of adjacent structures.

Consolidation analysis is related to Drained/Undrained analysis. Depending on the drainage conditions, the density, rigidity, and strength of the ground increase during the consolidation process.

Analysis Model Overview



In this tutorial, the following main concepts will be explained:

- Weak stratum - Apply the Modified Cam Clay model to simulate the limit state behavior.
- Modeling pile elements - Verify the settlement of the weak stratum according to pile skin friction.
- Set construction stages - Establish time steps for consolidation analysis.
- Analyze excess pore water pressure distribution and consolidation settlement - Evaluate these factors over time.
- Verify structure's settlement - Assess the pile skin friction and settlement of the structure over time.

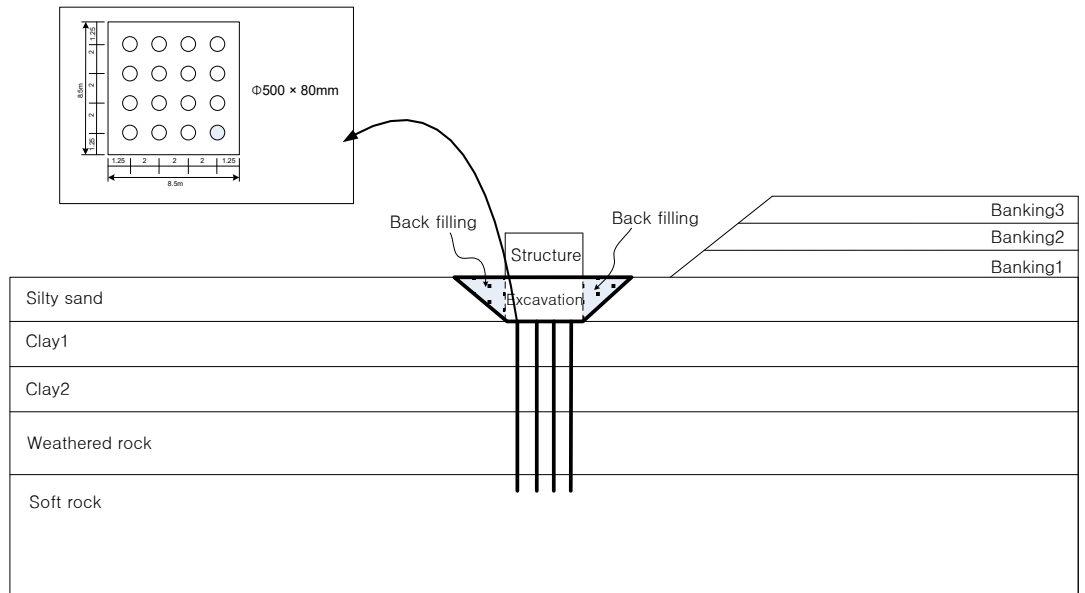


1.2 Model and Analysis Summary

In this tutorial, the pile foundation and structure are located at the top of a soft rock layer, and the 9-meter embankment is divided into three layers to carry out a construction stage consolidation analysis. When setting the construction stages, set the time step of each stage by considering the embankment height increase of 3 cm per day. Also, allow a waiting period after each embankment stage to ensure adequate consolidation. To minimize the influence of boundary conditions, set the ground area to be more than twice the size of the excavation area.

The strata composition and model are illustrated in the following image.

Cross-Section





Section 2

2.1 Starting with Analysis Setting

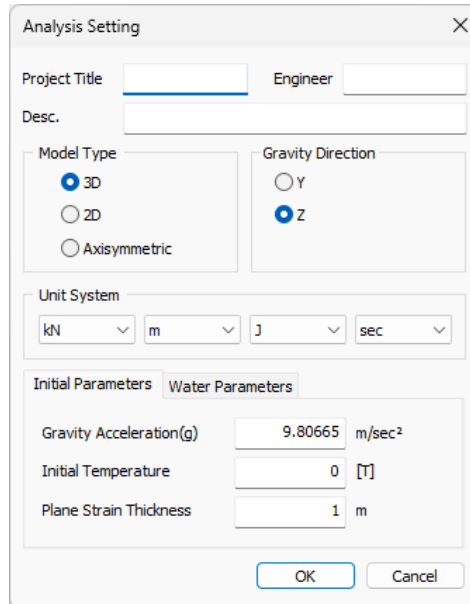
Analysis Setting

[ : Analysis > Analysis Case > General]

Set the model type, gravity direction, and initial parameters. Check the unit system that will be applied to the analysis. The unit system can be changed both during the modeling process and after performing the analysis. The input parameters will automatically convert to the appropriate unit system.

The model used in this tutorial is a 3D model with gravity in the Z direction. The model uses the SI unit system (kN, m) and a time unit of 'day' to define periods of consolidation and waiting.

Analysis Setting Window





Section 3

Material and Property

3.1 Definition of Ground and Structural Materials

To simulate the limit state behavior of the soft clay layer, define the model type as 'Modified Cam Clay'. For the other ground layers, use the general elastic model 'Mohr-Coulomb'. For the structures, use both the 'Elastic' model, which does not consider material nonlinearity, and the 'Pile' model, which is used to verify pile skin friction.

The ground and structure materials are defined in the following table:

Ground Materials

Name	Clay1	Clay2	Banking	Silty sand	Weathered rock	Soft rock	Structure
Material	Isotropic	Isotropic	Isotropic	Isotropic	Isotropic	Isotropic	Isotropic
Model type	Modified Cam Clay	Modified Cam Clay	Mohr Coulomb	Mohr Coulomb	Mohr Coulomb	Mohr Coulomb	Elastic
Elastic modulus	1.0E+04	1.0E+04	5.0E+04	5.0E+04	1.5E+05	3.0E+05	2.0E+07
Poisson's Ratio	0.45	0.4	0.25	0.3	0.2	0.15	0.2
Unit Weight	17	17	19	18	20	23	25
Ko	0.5	0.5	0.74	0.6	1	1	1
Unit Weight (Saturated)	20	20	21	20	21	23	-
Initial Void Ratio	2	1.5	0.5	0.5	0.5	0.5	-
Drainage Parameter	Drained	Drained	Drained	Drained	Drained		
kx	1.73E-03	1.73E-03	1.728	1.728	0.1728	0.01728	-
ky	1.73E-03	1.73E-03	1.728	1.728	0.1728	0.01728	-
kz	8.64E-04	8.64E-04	0.864	0.864	0.0864	0.00864	-
Void ratio dependency of permeability	0.5	0.5	0.5	0.5	0.5	0.5	-
Over Consolidation Ratio	1	1	-	-	-	-	-
Slope of Consol Line	0.221	0.32	-	-	-	-	-
Slope of Over Consol Line	0.015	0.028	-	-	-	-	-
Slope of Critical State Line	0.18	0.18	-	-	-	-	-
Pc	-	-	-	-	-	-	-
Allowable Tensile Stress	0	0	-	-	-	-	-
Cohesion	-	-	19	20	100	200	-



Tip

Coefficient of Infiltration According to Void Ratio

When pressure is applied to the ground, both the infiltration capacity (coefficient of infiltration) and the void ratio decrease. To simulate this behavior, define 'Void Ratio Dependency of Permeability (ck)'. According to the 'Saturated Coefficient of Water Permeability (ksat)' and the void ratio, the relationship is described by the following formula:

$$k = 10^{(\Delta e / c_k)} k_{sat}$$

In [Consolidation] and [Fully Coupled Stress Seepage] analyses, it is recommended to consider not only 'ck' but also 'Partially Saturated Effects for Stress Analysis' to obtain a more realistic approximate solution. If you do not consider saturation and partial saturation, self-weight is considered depending on whether the pore water pressure is above or below zero.

→ Ground density when partial saturation is used (Se: Saturation):

$$\rho = (1 - S_e) \rho_{unsat} + S_e \rho_{sat}$$

→ Ground density when partial saturation is not used:

$$\rho = \begin{cases} \rho_{unsat} & (p \leq 0) \\ \rho_{sat} & (p > 0) \end{cases}$$

The effect of partial saturation is reflected in the calculation of self-weight, undrained rigidity, and the internal force of the elements. By using the above options, it is possible to obtain accurate values that consider water level changes according to drainage and void ratio (permeability) changes due to ground deformation.

Tip

Allowable Tensile Stress

Generally, the MCC material model does not allow tensile stress in its fracture criterion (relation of stress-strain ratio). However, during the consolidation process, tensile stresses such as heaving from embankment loading and arching from excavation can be generated. To overcome the limitations of the material model and extend its applicability to real-world conditions, GTS NX is programmed to perform analyses on tensile stress if it is generated within the range of 'Allowable Tensile Stress'.

The value of allowable tensile stress is not fixed. To input a value greater than the tensile stress generated from overburden load (embankment) and fracture behavior, repetitive analysis is necessary. If the analysis is interrupted because the results diverge due to tensile fracture, you can set the 'Allowable Tensile Stress'.

If 'pc (Pre-consolidation load)' is directly inputted, the value of 'Allowable Tensile Stress' cannot exceed the 'pc' value. If the 'pc' value is defined through 'OCR', the software automatically calculates the 'pc' value according to the inputted 'Allowable Tensile Stress' value.

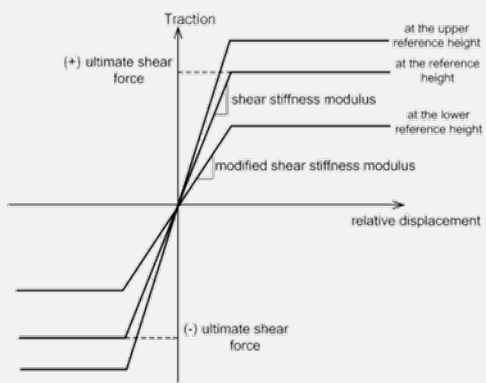


Structure Materials	Name	PHC pile	Pile
	Material	Isotropic	Interface/Pile
	Model type	Elastic	Pile
	Elastic modulus(E)	2.10E+07	-
	Poisson's Ratio(ν)	0.2	-
	Unit weight(r)	23	-
	Ultimate Shear Force	-	526.3
	Shear Stiffness Modulus(kt)	-	10526
	Normal Stiffness Modulus(kn)	-	115789



The behavior of pile elements can be divided into two normal directions and one tangential direction. The normal behavior considers the piles and the surrounding ground as a single rigid body, whereas the tangential behavior is nonlinear elastic. This nonlinear elastic behavior can be defined by a yield force or yield function.

The graph below represents the relative displacement between the two bodies and the friction when the yield force is defined. If the relationship is defined by a function, more precise results can be obtained.



The pile tip element acts as a solid-point interface that represents the interactive behavior between the ground elements and the pile nodes. In the element coordinate system of the pile tip element, the normal direction behavior is considered rigid, while the tangential direction behavior is considered nonlinear elastic, similar to the behavior of a pile element.

To define the behavior, the material and properties of pile elements can be entered based on experimental data, such as from a Load Test.

For more information about entering parameters of pile elements, refer to [User Manual] Ch4 (General Material) or press F1 for the [Online Manual].



3.2 Definition of Properties

Properties represent the physical attributes of the meshes and will be assigned to mesh sets during mesh generation. While defining ground and structure properties, you should choose the material to be used.

When pile elements are used, as demonstrated in this tutorial, you can define the pile tip bearing capacity and rigidity modulus. Use a beam element for the structural member of the PHC pile, which resists axial, shearing, and bending forces. It is possible to verify friction behavior and relative displacement with the adjacent ground by defining rigidity around and at the tip of the pile. Since the pile element, like the interface element, does not share nodes with the adjacent ground, it can be easily applied in a 3D model.

Properties for ground mesh sets are defined as below:

Ground Property

Name	Clay1	Clay2	Banking	Silty sand	Weathered rock	Soft rock	Structure
Type	3D	3D	3D	3D	3D	3D	3D
Material	Clay1	Clay2	Banking	Silty sand	Weathered rock	Soft rock	Structure

Properties for structure mesh sets are defined as follows. The stiffness will be automatically calculated when the section shape is defined.

Structure Property


Name	PHC pile	Pile	Pile tip
Type	1D	1D	Others
Model type	Beam	-	Pile tip
Material	PHC pile	Pile	-
Section shape	PIPE	-	-
Section	500x80	-	-
Thickness	-	1	-
Tip bearing Capacity	-	-	400
Tip Spring Stiffness	-	-	16000



Section 4 Modeling

The tutorial focuses on 3D geometry and element generation, consolidation behavior analysis, and results output. Begin the tutorial by opening the start file, in which basic materials, properties, and lines for generating 3D geometry shapes have already been predefined.

4.1 Modeling Geometry

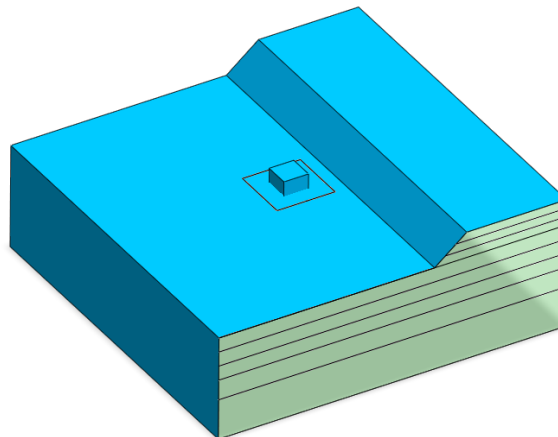
[ : Geometry > Protrude > Extrude]

This process involves creating line/face/solid shapes from lower shapes such as points/edges/faces. With lines that form a closed domain, it's also possible to create a solid at once.

Extrude 3D solids for the ground and structures, and create dividing surfaces from the dividing lines to classify each stratum.

- ① Select ground faces (1).
- ② Choose the direction as 'Y' and input a length of 120 meters.
- ③ Click [Apply] and check the generated solid.
- ④ Select structure faces (1).
- ⑤ Choose the direction as 'Z' and input a length of 10 meters.
- ⑥ Click [Apply].

Solid for Ground and
Structure



When selecting objects or directions, you can use both the Work Tree and select directly in the Work Window.

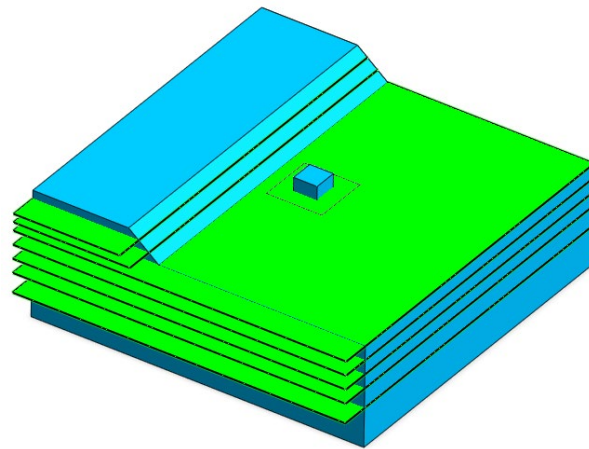



- ① Change the [Selection Filter] to 'Line'. Select the dividing lines (7).
- ② Choose the direction as 'Y' and input a length of 130 meters, which is large enough to divide the solid.



When dividing a solid with a face, it is recommended to make the face larger than the solid. If the face is smaller than the solid, even if the difference is very small, the separation may not be performed.

Divide Surface



[] : Geometry > Protrude > Loft

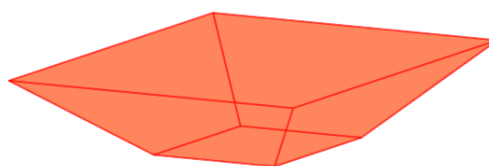
[Loft] function is used to create a solid by linearly connecting two separated faces. Use the [Loft] function to generate the excavation area for installing the structure.




Geometries can be shown or hidden from the Work Window by checking or unchecking the checkbox in the Work Tree. To create a solid at the inner part of the ground, hide unnecessary geometries.

- ① Select faces (2) of the structure and excavation area.
- ② Click [OK].
- ③ Use the [Preview] function to check the generated shape.

Excavation Area



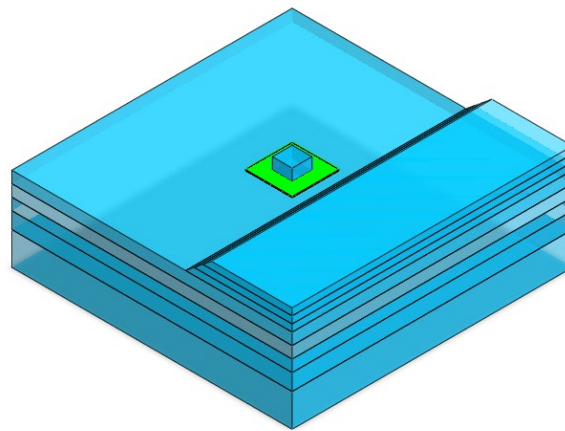



[ : Geometry > Divide > Solid]

This process separates areas to classify ground strata and banking stages. Use the dividing faces created in the first step to separate the ground solid.

- ① Select entire solids (3) for the object.
- ② For the tool object, select the dividing faces (7).
- ③ Click [OK].
- ④ Check the separated solids.

Separated Ground



[ : Geometry > Surface & Solid > Auto Connect]

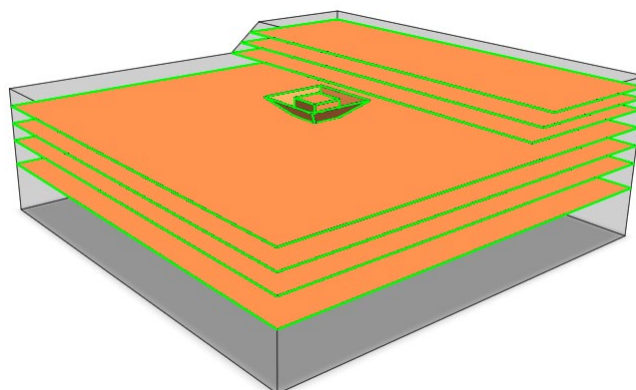
This process generates shared faces automatically after generating 3D geometries. Shared faces must be created before mesh generation to connect nodes.

- ① Select all the generated solids (11).
- ② Click [OK].



To prevent analysis errors caused by unconnected nodes between elements, it is recommended to verify the generated shared faces. Shared faces can be checked by navigating to Geometry > Tools > Check Shape > Check Geometry > Check Duplicates.


Checking the Shared Face





4.2 Generate Meshes

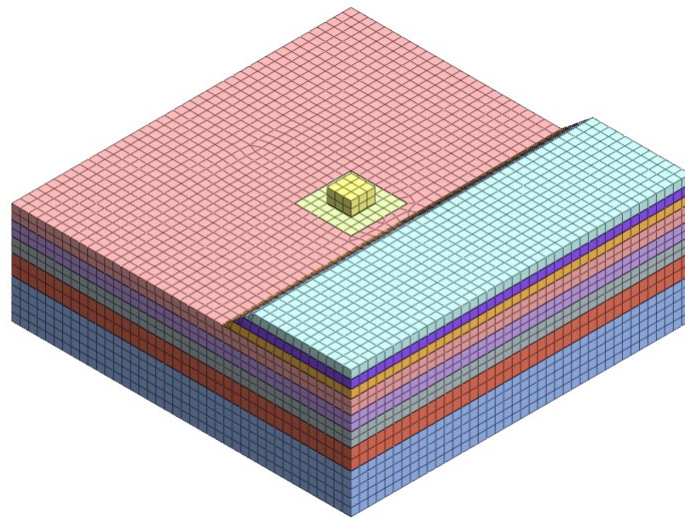
Mesh shape and quality are crucial in finite element analysis. For stable analysis, it is recommended to use hexahedral-centered meshes for 3D models and tetrahedral-centered meshes for 2D models, especially in the ground where plastic failure occurs due to compressive and shearing deformation. Use Map Mesher or hexahedral-centered Hybrid Mesher to create meshes.

[ : Mesh > Generate > 3D]

Generate mesh for the 3D ground area. Select the entire solid and generate the mesh at once.

- ① Select [Auto-Solid].
- ② Choose the entire solid (11).
- ③ Input '3' for the mesh size.
- ④ Select 'Hybrid Mesher' from the dropdown menu.
- ⑤ Press the Preview button to check the generated element nodes.
- ⑥ Click [OK] and check the generated mesh.
- ⑦ Press the [F2] key on the Work Tree to change names for mesh sets.

Ground Mesh





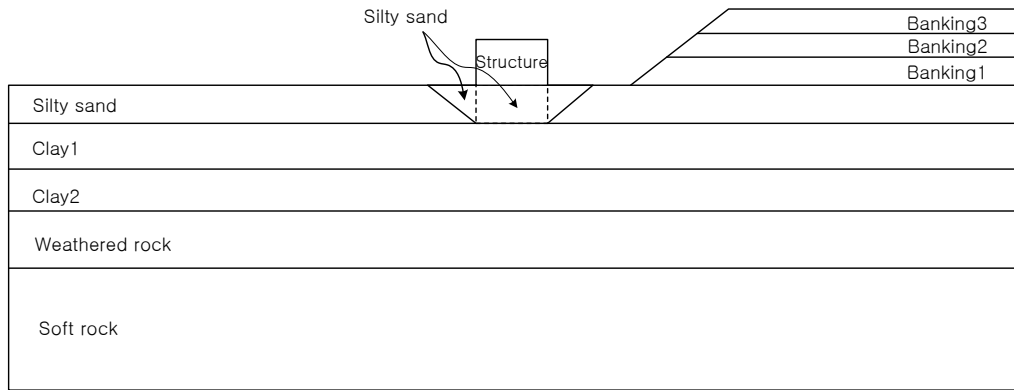
[ : Mesh > Element > Parameters]

Change properties for each mesh set and check. You can initially generate all the mesh sets with one property and then change the properties later in [Parameter].

- ① Select the [3D] tab.
- ② Change the properties as shown in the Model Overview (cross-section diagram).
- ③ Select each mesh set and assign the relevant property.
- ④ Click [Apply] to change the property.

If you select a mesh set in the Work Tree, you can see the material/property of the mesh set in the property window.


Property Information



The mesh set will be separated automatically by each solid. Select the mesh set in the model tree and change the parameters for each mesh set. Also, for the construction stage setup, change the names of the mesh set. You can change the mesh set name in the work tree by pressing the [F2] key.

Mesh Set	Number	Color
Default Mesh Set	1	Yellow
Weathered rock	2	Purple
Clay1	3	Blue
Clay2	4	Light Green
Soft rock	5	Yellow
Banking3	6	Light Blue
Silty sand	7	Blue
Banking1	8	Orange
Structure	9	Red
Silty sand(Excavation)	10	Purple
Silty sand(Structure bott..	11	Grey
Banking2	12	Red
Pile-beam element	13	Blue
Pile[Pile-beam element]	14	Brown
Pile Tip	15	Dark Green

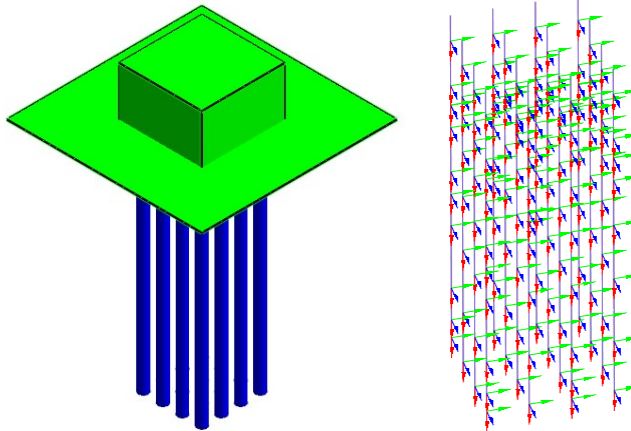


[ : Mesh > Generate > 1D]

Create beam elements for piles. Beam elements are connected with nodes of adjacent ground. However, if you use pile (interface) elements, the connection needs to be created independently.

- ① Select all pile lines (16) and input '1' in the [Division] field.
- ② Choose the [PHC pile] property.
- ③ Name the mesh set 'Pile-beam element' and click [OK].

Generate Beam Element



The size and division of the beam elements before generating pile elements are not important. This is because the beam element nodes and the ground nodes are automatically connected when the 'Pile elements' are generated.

To output results in the element coordinate system, the coordinate system of each structure member element should be aligned in one direction. Otherwise, the result on this element will be opposite. The element coordinate system can be changed in Mesh > Element > Parameter. For further information, refer to the [User Manual] or [Online Manual (F1)].

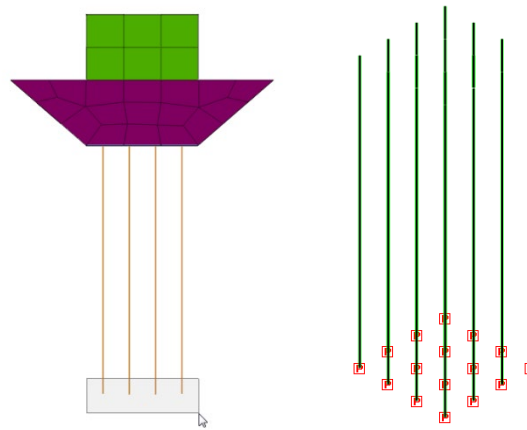


[ : Mesh > Element > Pile/Pile Tip]

To simulate the interaction between the pile and ground, add pile elements. Using the Pile-beam elements generated in the former step, generate pile elements and pile tip elements.

- ① Choose the [Pile] tab.
- ② Select all the generated pile-beam elements (16).
- ③ Choose the 'Pile' property and click [Apply].
- ④ Switch to the [Pile-tip] tab.
- ⑤ Select nodes (16) of the pile tip as shown in the image.
- ⑥ Choose the [Pile-tip] property and select [OK].

Generate Pile Interface /
Pile Tip



 Tip


Adjacent ground elements must be created before generating pile elements. For beam elements that are not connected to the ground, it is impossible to generate pile elements to simulate contact surface behavior. Therefore, ensure that the ground elements are created first before generating pile elements.



Section 5

5.1 Setting Boundary Conditions

Analysis

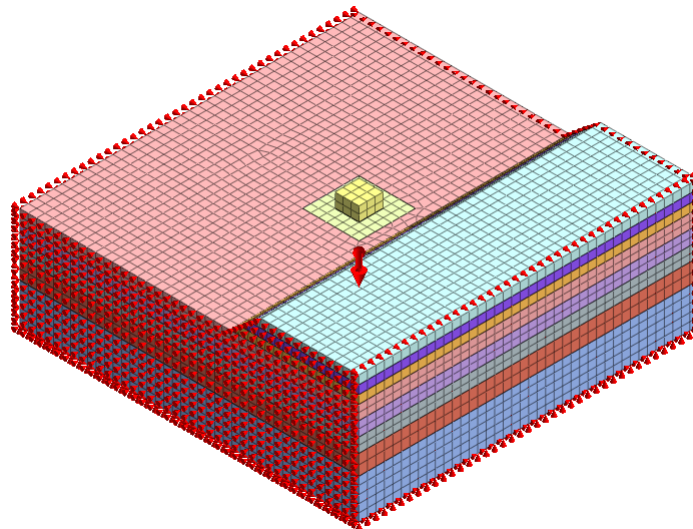
[ : Seepage/Consolidation Analysis > Boundary > Constraint]

This process sets boundary conditions against internal deformation or rotation based on the Global Coordinate System (GCS).

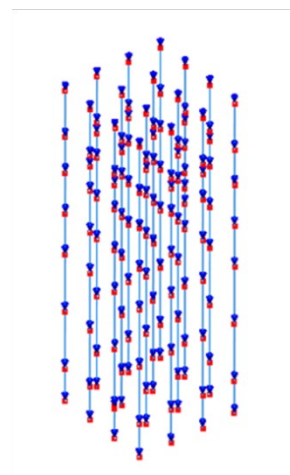
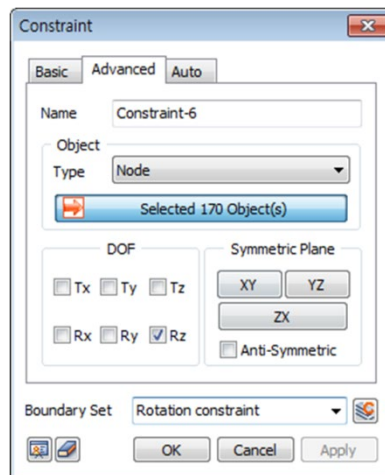
For boundaries of the entire model, automatically set constraints of left/right/bottom displacements according to GCS. Constrain rotation in the Rz direction to prevent the degree of freedom error of pile elements due to torsion.

- ① Select the [Auto] tab.
- ② Check [Consider All Mesh Sets] and name the boundary set 'Ground boundary'.
- ③ Click [Apply].
- ④ Select the [Advanced] tab.
- ⑤ Choose the type as 'Node' and select the entire node of the generated pile element. Check 'Rz'.
- ⑥ Name the boundary set 'Rotation constraint' and select [OK].


Boundary Condition for Ground



Boundary Condition for Pile





[ : Seepage/Consolidation Analysis > Boundary > Draining Condition]

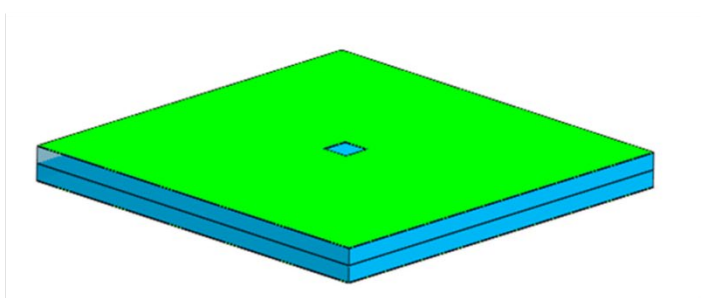
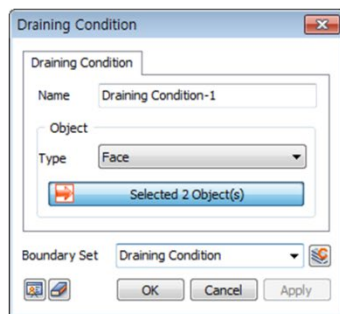
This process sets the drainage condition for excess pore water pressure. When no extra drainer is needed, typically 'both-side drainage' or 'single-side drainage' is set at the clay layer, which has relatively lower permeability. However, this 'complete drainage' assumption neglects the permeability of other ground materials around the clay layer. If you want to consider the permeability of the surrounding ground as well, set the drainage condition at the surface and at the bottom parts of the model. If the model size (ground area) is relatively bigger than the interested area, it is possible to set the drainage condition at the left/right/bottom with ground constraint boundary conditions. However, in this tutorial, we will simply set draining conditions at the surface and bottom sides of the clay layer.

- ① Select Clay 1 and Clay 2 to show only.
- ② Set the type to 'Face'.
- ③ Select the top of 'Clay1' and bottom of 'Clay2' as shown in the image.
- ④ Name the boundary set 'Draining condition'.
- ⑤ Click [OK] and check the generated draining condition.




Geometry (Line, Face) or element node can be directly selected as object shape. If you select geometry, boundary conditions will be automatically applied to all element nodes within the range of the selected shape. In other words, the type of object is distinguished only for the convenience of selection. The conditions reflected in the model are all the same. If you set the solid to be shown only, you can easily select the bedding plane.

Draining Condition





[ : Seepage/Consolidation Analysis > Boundary > Non Consolidation]

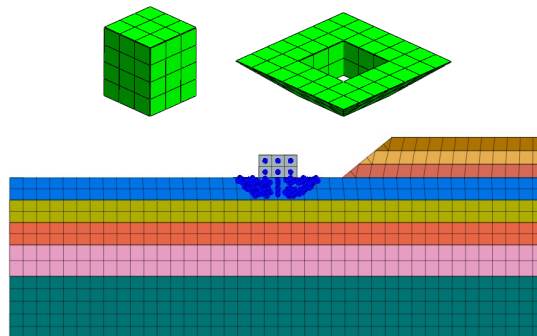
When performing consolidation analysis, all solid elements acquire degrees of freedom (DOF) for pore water pressure. Therefore, for elements that ignore consolidation behavior, such as concrete structures or elements that are not directly affected by consolidation, they should be defined as non-consolidation elements.

Although non-consolidation is sometimes set for bank materials above the water level as a loading condition, in cases of precise analysis that consider water level changes and partial saturation, non-consolidation should not be set for the lower part of the bank to simulate water level changes. In this tutorial, non-consolidation is only used for concrete structures.


After excavation, set the 'backfilling' area and 'structure' to non-consolidation elements so that the area does not bear excess pore pressure.

- ① Select meshes: [Structure, Silty sand (Structure), Silty sand (Backfilling)].
- ② Name the boundary set as 'Non consolidation' and click [OK].

Condition for
Non-Consolidation





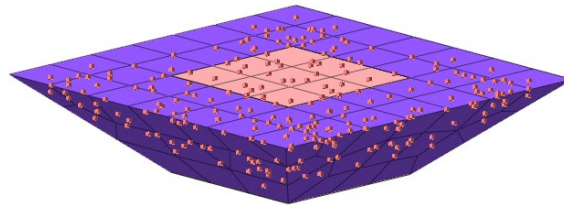
[ : Seepage/Consolidation Analysis > Boundary > Change Property]

Set boundary conditions for mesh sets, whose properties change according to the construction stage. When assigning properties, only one property can be assigned to each mesh set. However, the [Change property] boundary condition can be set to simulate the property change of mesh sets during the construction.

At the [Original Ground] step, the property of the excavation part is silty sand, but in the [Install structure & pile (backfilling)] step, the property of this area is changed, and this [Change property] boundary needs to be activated.


- ① Select mesh sets [Silty sand (Structure), Silty sand (Excavation)].
- ② Change the property to 'Structure' and name the boundary set 'Change property'.

Change Property





5.2 Define Construction Stages

[ : Seepage/Consolidation Analysis > Construction > Stage Set]

Set the construction stage to verify the results of each stage. For consolidation analysis, you can set a 'time step' for each step and verify the results according to time.

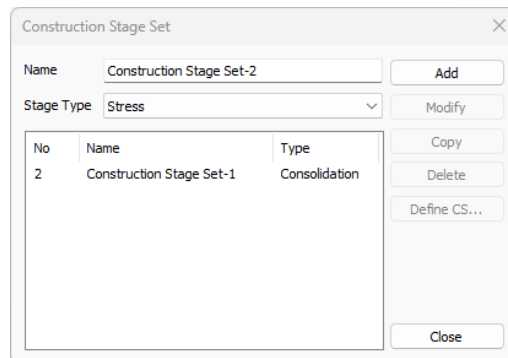
Loading time and distribution have a significant influence on the analysis results. For banking, consider the banking quantity per day to decide the number of banking steps and the duration of each step. In the [Negligence] period after the banking, set only the duration and step number without changing any conditions.

In the construction stage setting, data once activated will sustain the same status until you choose to deactivate it. Set the water level condition at Stage 1. If the water level is located at a specific level, you can simply define it by inputting the height.

Name the construction stage set.

- ① Set the stage type to [Consolidation] and click [OK].
- ② Click [Define construction stage] to create the construction stage.
- ③ Construction stages are defined below:

Construction Stage Set





Stage 1. Initial

The dialog box is titled "Define Construction Stage" and shows the configuration for "Construction Stage Set-1".

- Stage ID:** 1: Initial
- Stage Name:** Initial
- Stage Type:** Consolidation

The data is organized into three columns:

- Set Data:** Mesh, Banking1-3, Clay1-2, Default Mesh Set, Pile Tip, Pile-beam element, Silty sand, Soft rock, Structure, Weathered rock, Boundary Condition, Change property, Draining Condition, Ground constraint, Non Consolidation, Rotation constraint, Static Load, Default Self-Weight, Combined Load Sets.
- Activated Data:** Mesh, Clay1-2, Silty sand, Silty sand(Excavati...), Silty sand(Structure bo), Soft rock, Weathered rock, Boundary Condition, Static Load, Default Self-Weight, Combined Load Sets, Contact.
- Deactivated Data:** Mesh, Boundary Condition, Static Load, Combined Load Sets, Contact.

Initial Condition settings:

- Define Water Level For Global (36 m)
- Define Water Level For Mesh Set
- Sub Stage...
- LDF...
- Clear Displacement
- Clear Strain
- Slope Stability(SRM)

Buttons: Move to Previous, Move to Next, New, Insert, Delete, Time Step..., Save, Close.

Stage 2. Excavation

The dialog box is titled "Define Construction Stage" and shows the configuration for "Construction Stage Set-1".

- Stage ID:** 2: Excavation
- Stage Name:** Excavation
- Stage Type:** Consolidation

The data is organized into three columns:

- Set Data:** Mesh, Banking1-3, Clay1-2, Default Mesh Set, Pile Tip, Pile-beam element, Silty sand, Silty sand(Excavati...), Silty sand(Structure bo), Soft rock, Structure, Weathered rock, Boundary Condition, Change property, Draining Condition, Ground constraint, Non Consolidation, Rotation constraint, Static Load, Default Self-Weight, Combined Load Sets.
- Activated Data:** Mesh, Boundary Condition, Static Load, Combined Load Sets, Contact.
- Deactivated Data:** Mesh, Silty sand(Excavati...), Silty sand(Structure bo), Boundary Condition, Static Load, Combined Load Sets, Contact.

Initial Condition settings:

- Define Water Level For Global (0 m)
- Define Water Level For Mesh Set
- Sub Stage...
- LDF...
- Clear Displacement
- Clear Strain
- Slope Stability(SRM)

Buttons: Move to Previous, Move to Next, New, Insert, Delete, Time Step..., Save, Close.



Stage 3. Install Structure & Pile

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 3: Install Structure&Pile(Back filling) | Move to Previous | Move to Next

Stage Name: Install Structure&Pile(Back filling) | New | Insert | Delete

Stage Type: Consolidation | Time Step...

Set Data: Mesh, Banking1, Banking2, Banking3, Clay1, Clay2, Default Mesh Set, Pile Tip, Pile-beam element, Pile[Pile-beam elerr, Silty sand, Silty sand(Excavati..., Silty sand(Structure, Soft rock, Structure, Weathered rock, Boundary Condition, Change property, Draining Condition, Ground constraint, Non Consolidation, Rotation constraint, Static Load, Default Self-Weight, Combined Load Sets

Activated Data: Mesh, Pile Tip, Pile-beam element, Pile[Pile-beam elem..., Silty sand(Excavati..., Structure, Boundary Condition, Change property, Draining Condition, Non Consolidation, Rotation constraint, Static Load, Combined Load Sets, Contact

Deactivated Data: Mesh, Boundary Condition, Static Load, Combined Load Sets, Contact

Initial Condition: Define Water Level For Global, Define Water Level For Mesh Set, Sub Stage..., LDF..., Clear Displacement, Clear Strain, Slope Stability(SRM)

Sort By: Name | Show Data: All | Save | Close

Stage 4. Banking 1

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 4: Banking 1 | Move to Previous | Move to Next

Stage Name: Banking 1 | New | Insert | Delete

Stage Type: Consolidation | Time Step...

Set Data: Mesh, Banking1, Banking2, Banking3, Clay1, Clay2, Default Mesh Set, Pile Tip, Pile-beam element, Pile[Pile-beam elerr, Silty sand, Silty sand(Excavati..., Silty sand(Structure, Soft rock, Structure, Weathered rock, Boundary Condition, Change property, Draining Condition, Ground constraint, Non Consolidation, Rotation constraint, Static Load, Default Self-Weight, Combined Load Sets

Activated Data: Mesh, Banking 1, Boundary Condition, Static Load, Combined Load Sets, Contact

Deactivated Data: Mesh, Boundary Condition, Static Load, Combined Load Sets, Contact

Initial Condition: Define Water Level For Global, Define Water Level For Mesh Set, Sub Stage..., LDF..., Copy To Specific Stage..., Clear Displacement, Clear Strain, Slope Stability(SRM)

Sort By: Name | Show Data: All | Save | Close



Stage 5. Negligence 1

The dialog box is titled "Define Construction Stage" and has a close button (X) in the top right corner. The "Construction Stage Set Name" is "Construction Stage Set-1".

Stage ID: 5: Negligence1 (dropdown menu). Buttons: Move to Previous, Move to Next.

Stage Name: Negligence1 (text field). Buttons: New, Insert, Delete.

Stage Type: Consolidation (dropdown menu). Button: Time Step...

Three columns of data:

- Set Data:** Mesh (expanded), Banking1, Banking2, Banking3, Clay1, Clay2, Default Mesh Set, Pile Tip, Pile-beam element, Pile[Pile-beam elerr], Silty sand, Silty sand(Excavati..), Silty sand(Structure), Soft rock, Structure, Weathered rock, Boundary Condition, Change property, Draining Condition, Ground constraint, Non Consolidation, Rotation constraint, Static Load, Default Self-Weight, Combined Load Sets.
- Activated Data:** Mesh, Boundary Condition, Static Load, Combined Load Sets, Contact.
- Deactivated Data:** Mesh, Boundary Condition, Static Load, Combined Load Sets, Contact.

Right-hand side controls:

- Analysis Control... (checkbox)
- Output Control... (checkbox)
- Initial Condition:
 - Define Water Level For Global (checkbox), 0 m, None (dropdown), ...
 - Define Water Level For Mesh Set (checkbox), Input Water Level...
- Sub Stage... (checkbox)
- LDF... (checkbox), Copy To Specific Stage... (button)
- Clear Displacement (checkbox)
- Clear Strain (checkbox, checked)
- Slope Stability(SRM) (checkbox)

Bottom: Sort By: Name (dropdown), Show Data: All (dropdown), Save, Close.

Stage 6. Banking 2

The dialog box is titled "Define Construction Stage" and has a close button (X) in the top right corner. The "Construction Stage Set Name" is "Construction Stage Set-1".

Stage ID: 6: Banking2 (dropdown menu). Buttons: Move to Previous, Move to Next.

Stage Name: Banking2 (text field). Buttons: New, Insert, Delete.

Stage Type: Consolidation (dropdown menu). Button: Time Step...

Three columns of data:

- Set Data:** Mesh (expanded), Banking1, Banking2, Banking3, Clay1, Clay2, Default Mesh Set, Pile Tip, Pile-beam element, Pile[Pile-beam elerr], Silty sand, Silty sand(Excavati..), Silty sand(Structure), Soft rock, Structure, Weathered rock, Boundary Condition, Change property, Draining Condition, Ground constraint, Non Consolidation, Rotation constraint, Static Load, Default Self-Weight, Combined Load Sets.
- Activated Data:** Mesh, Banking2, Boundary Condition, Static Load, Combined Load Sets, Contact.
- Deactivated Data:** Mesh, Boundary Condition, Static Load, Combined Load Sets, Contact.

Right-hand side controls:

- Analysis Control... (checkbox)
- Output Control... (checkbox)
- Initial Condition:
 - Define Water Level For Global (checkbox), 0 m, None (dropdown), ...
 - Define Water Level For Mesh Set (checkbox), Input Water Level...
- Sub Stage... (checkbox)
- LDF... (checkbox), Copy To Specific Stage... (button)
- Clear Displacement (checkbox)
- Clear Strain (checkbox, checked)
- Slope Stability(SRM) (checkbox)

Bottom: Sort By: Name (dropdown), Show Data: All (dropdown), Save, Close.



Stage 7. Negligence 2

The dialog box is titled "Define Construction Stage" and has a close button (X) in the top right corner. It contains the following elements:

- Construction Stage Set Name:** A dropdown menu set to "Construction Stage Set-1".
- Stage ID:** A dropdown menu set to "7: Negligence2". Buttons for "Move to Previous" and "Move to Next" are to its right.
- Stage Name:** A text input field containing "Negligence2". Buttons for "New", "Insert", and "Delete" are to its right.
- Stage Type:** A dropdown menu set to "Consolidation". A "Time Step..." button is to its right.
- Initial Condition:** A section with several checkboxes:
 - Analysis Control...
 - Output Control...
 - Define Water Level For Global (with a value of "0 m" and a "None" dropdown).
 - Define Water Level For Mesh Set (with an "Input Water Level..." button).
 - Sub Stage...
 - LDF... (with a "Copy To Specific Stage..." button).
 - Clear Displacement (with a checked Clear Strain).
 - Slope Stability (SRM).
- Data Columns:** Three columns: "Set Data" (light blue background), "Activated Data" (light green background), and "Deactivated Data" (light yellow background).
 - Set Data:** A tree view containing a "Mesh" folder with sub-items like Banking1-3, Clay1-2, and various material types (Silty sand, Soft rock, etc.), and a "Boundary Condition" folder with items like Static Load, Combined Load Sets, and Contact.
 - Activated Data:** Contains "Mesh", "Boundary Condition", "Static Load", "Combined Load Sets", and "Contact".
 - Deactivated Data:** Contains "Mesh", "Boundary Condition", "Static Load", "Combined Load Sets", and "Contact".
- Bottom:** "Sort By" dropdown set to "Name", "Show Data" dropdown set to "All", and "Save" and "Close" buttons.

Stage 8. Banking 3

The dialog box is titled "Define Construction Stage" and has a close button (X) in the top right corner. It contains the following elements:

- Construction Stage Set Name:** A dropdown menu set to "Construction Stage Set-1".
- Stage ID:** A dropdown menu set to "8: Banking3". Buttons for "Move to Previous" and "Move to Next" are to its right.
- Stage Name:** A text input field containing "Banking3". Buttons for "New", "Insert", and "Delete" are to its right.
- Stage Type:** A dropdown menu set to "Consolidation". A "Time Step..." button is to its right.
- Initial Condition:** A section with several checkboxes:
 - Analysis Control...
 - Output Control...
 - Define Water Level For Global (with a value of "0 m" and a "None" dropdown).
 - Define Water Level For Mesh Set (with an "Input Water Level..." button).
 - Sub Stage...
 - LDF... (with a "Copy To Specific Stage..." button).
 - Clear Displacement (with a checked Clear Strain).
 - Slope Stability (SRM).
- Data Columns:** Three columns: "Set Data" (light blue background), "Activated Data" (light green background), and "Deactivated Data" (light yellow background).
 - Set Data:** A tree view containing a "Mesh" folder with sub-items like Banking1-3, Clay1-2, and various material types (Silty sand, Soft rock, etc.), and a "Boundary Condition" folder with items like Static Load, Combined Load Sets, and Contact.
 - Activated Data:** Contains "Mesh", "Banking3", "Boundary Condition", "Static Load", "Combined Load Sets", and "Contact".
 - Deactivated Data:** Contains "Mesh", "Boundary Condition", "Static Load", "Combined Load Sets", and "Contact".
- Bottom:** "Sort By" dropdown set to "Name", "Show Data" dropdown set to "All", and "Save" and "Close" buttons.



Stage 9.
Final
(Complete Consolidation)

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 9: Final(Complete consolidation) | Move to Previous | Move to Next

Stage Name: Final(Complete consolidation) | New | Insert | Delete

Stage Type: Consolidation | Time Step...

Set Data	Activated Data	Deactivated Data
Mesh	Mesh	Mesh
Banking1	Boundary Condition	Boundary Condition
Banking2	Static Load	Static Load
Banking3	Combined Load Sets	Combined Load Sets
Clay1	Contact	Contact
Clay2		
Default Mesh Set		
Pile Tip		
Pile-beam element		
Pile[Pile-beam elern		
Silty sand		
Silty sand(Excavati..		
Silty sand(Structure		
Soft rock		
Structure		
Weathered rock		
Boundary Condition		
Change property		
Draining Condition		
Ground constraint		
Non Consolidation		
Rotation constraint		
Static Load		
Default Self-Weight		
Combined Load Sets		

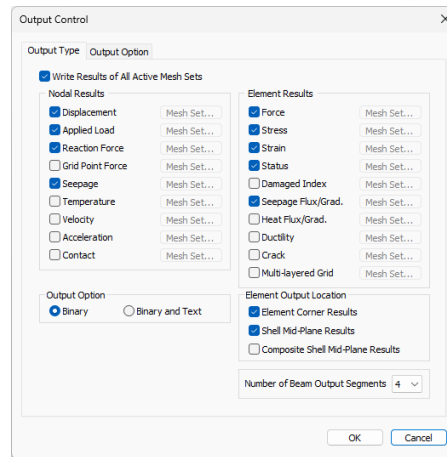
Sort By: Name | Show Data: All | Save | Close



5.3 Setting Analysis Case

This process sets the model data and analysis method. You can control the analysis type and output type in the advanced options. Through result control, it is possible to manage the analysis time and file size by plotting the result item you need. In construction analysis, the data for each stage is set, so [Analysis Case Model] is deactivated. In [Output Control], check Element Results > Strain to plot relative displacement occurring from interface behavior between pile elements and ground.

Output Control



[ : Analysis > Analysis Case > General]

- ① Input the name and choose the solution type as 'Construction stage'.
- ② In Analysis Control > General tab > Initial stage for stress analysis, set the initial stage to '1: Original Ground'. Do not check [Apply KO Condition].
- ③ Check [Automatically Consider Water Pressure] to consider water pressure below the water level when excavating.
- ④ Click [OK] to add the analysis case.


Tip

If you use the K_0 condition, the vertical stress and horizontal stress of the initial foundation are calculated by the theoretical static coefficient of earth pressure (K_0). However, when considering the nonlinear (Undrained) behavior characteristics of excavation and load at a weakness layer below the water level, it is more reasonable to set the initial stress using the Poisson's ratio of each stratum. Therefore, generally, the K_0 condition is not used in consolidation analysis.



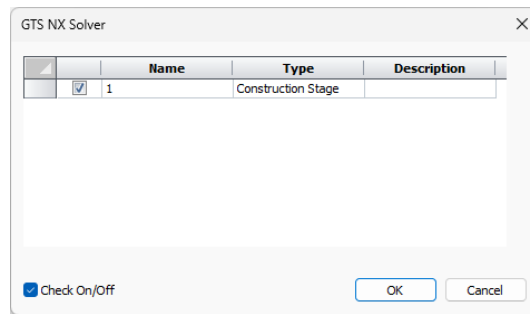
5.4 Perform Analysis

After the analysis is completed, the software automatically switches to [Post-Mode] for checking results. To modify the model and options after the analysis, you have to switch back to the [Pre-Mode].

[ : Analysis > Analysis > Perform]

① Perform Analysis

Perform Window



During the analysis, you can monitor the calculation process, check whether the results are converged or not, and view any warnings and errors through the [Output Window].



Section 6 Results

You can verify the displacement of each construction stage and each time interval. Results such as ground deformation, excess pore water pressure, degree of consolidation, deformation of adjacent structures, and member forces of piles can be examined from the [Result Tree]. All results can be output as contour plots, tables, and graphs. In this tutorial, the main result items that need to be checked are as follows:

- Surface settlement according to time (Time-Settlement Graph)
- Consolidation according to time (Time-Excess pore water pressure graph)
- Deformation of adjacent structures over time
- Pile surrounding friction over time

6.1 Verify Displacement

Verify by 'Displacement' from the work tree after the analysis. TX, TY, TZ signify displacements in the X, Y, Z directions. Settlement according to bank load and consolidation can be checked in 'TZ'.

'(V)' refers to the result item which can represent both contour and vector at the same time. In GTS NX, it is possible to show contour/vector simultaneously about displacement and principal stress.

- In the consolidation analysis which is applied by time step, it is important to verify as time passes whether the settlement, pore water pressure dissipation converges or not. By moving the stage bar at the bottom of the work window, it is possible to see the change of settlement at each construction stage.

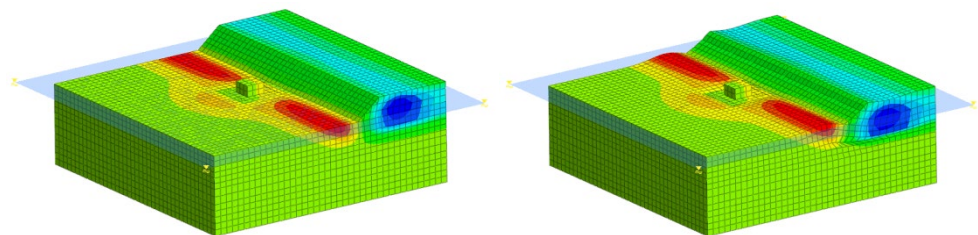
Stage Bar



Verify the result at the final (Complete consolidation) stage.

- Select the last stage/last time step in the Work Tree, and select Displacement > TZ TRANSLATION (V).
- In Result > General > Deform, it is possible to see the unreformed/deformed shape directly in the Z direction. (It is possible to modify the degree of deformation by scale in the property window. You can view the actual deformation by checking [Actual Deformation] at Result > Show/Hide.)

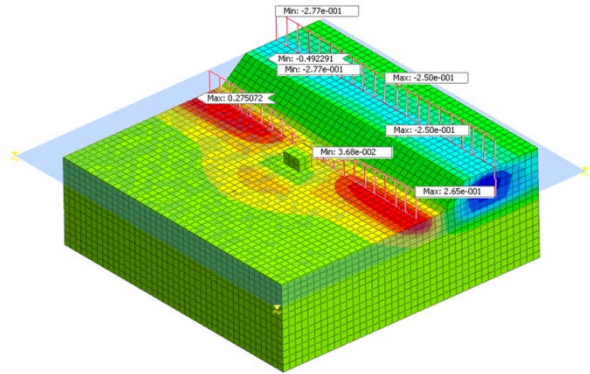
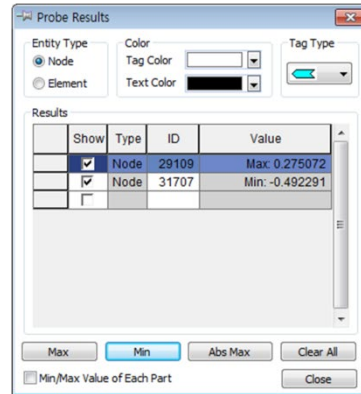
Settlement of the Ground





- By Result > Advance > Probe, it is possible to verify results on specific nodes and elements. Select the nodes which you want to see the result by mouse click. It is also used to verify values and locations of Maximum/Minimum/Absolute Maximum points.
- By [Cutting Diag], it is possible to plot settlement diagram of a specific line (face).

Probe Function

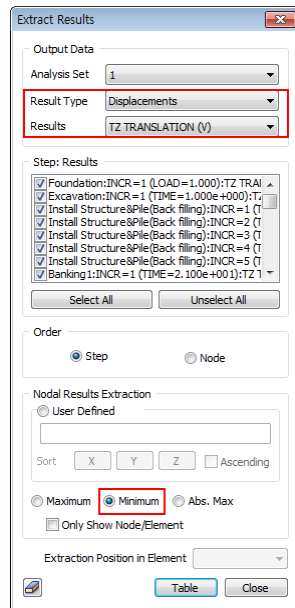




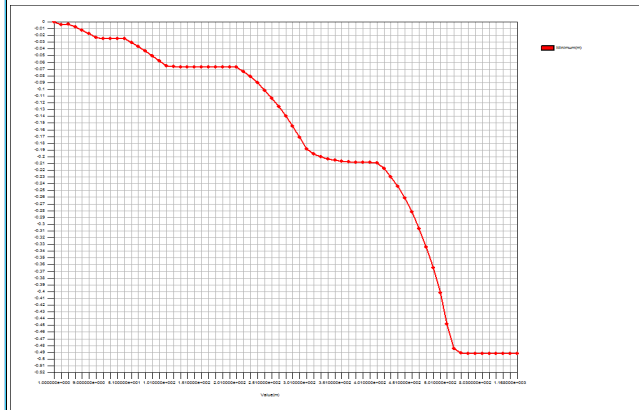
Verify the time history result of settlement in each stage and each time step.

- By using Result > Advance > Extract, you can extract tables and draw graphs of results at selected stages. In consolidation analysis, since the final settlement is a convergent result after complete dissipation of excess pore pressure, verification of the 'time - settlement' graph is needed. Only checking the max/min value in the legend after the analysis is not enough.
- Select Result > Displacement > TZ, check 'Minimum' to see the maximum settlement (-). Select [Table]. By right-clicking in the plotted table, you can plot a graph. In the banking stage and negligence stage, it is possible to verify the increase and convergence of settlement. Also, in the final negligence stage, you can see that the settlement increase will converge to '0(zero)' as time passes. Because there is a consolidation time difference according to permeability, you have to perform repeated analysis by extending the negligence duration if the settlement still shows an increase at the final stage.

Extract the Results



No	Step	Value	Minimum (m)
1	Foundation:INCR=1 (LOAD=1.000)		
2	Excavation:INCR=1 (TIME=1.000e+0)		
3	Install Structure&Pile(Back filling):IN		
4	Install Structure&Pile(Back filling):IN		
5	Install Structure&Pile(Back filling):IN		
6	Install Structure&Pile(Back filling):IN		
7	Install Structure&Pile(Back filling):IN	1.100000e+001	-2.316253e-002
8	Banking1:INCR=1 (TIME=2.100e+00)	2.100000e+001	-2.439996e-002
9	Banking1:INCR=2 (TIME=3.100e+00)	3.100000e+001	-2.479153e-002



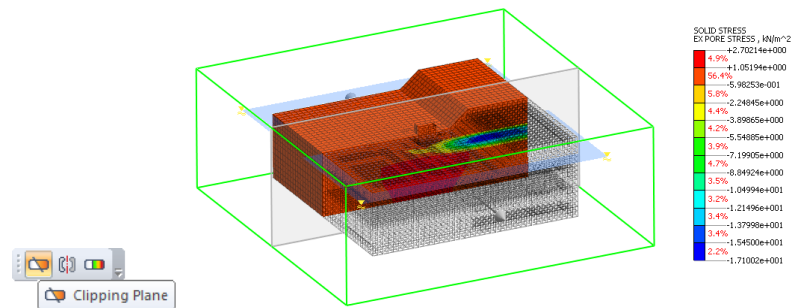


6.2 Verify Stress

Stress generated in the ground can be verified at 'Solid Stresses' in the Work Tree. S-XX, S-YY, S-ZZ represent stress in each direction, and it is also possible to verify excess pore pressure dissipation according to the banking/negligence stage.

- Select Solid Stresses > EXCESSIVE PORE STRESS at the stage right after the 3 layer banking in the Work Tree.
- You can verify the inner distribution chart of the ground by selecting View > Advanced Toolbar > Clipping Plane.

Clipping Function





Verify the history of excess pore pressure dissipation of each construction stage and each time step.

- In consolidation, settlement and excess pore pressure dissipation are important items for calculating the construction period. To figure out the time when consolidation is completed, verify the distribution of excess pore pressure according to time.
- Select Results Tree > Solid Stresses > EXCESSIVE PORE STRESS to verify the history of excess pore pressure dissipation. Choose the element (e.g., 18149 in the tutorial) where excessive pore pressure concentration occurs because of surface load. Input the element ID and press the [Table] button. By right-clicking in the plotted table, you can plot a graph. After the banking, excess pore pressure increases. In the negligence stage, you can figure out the dissipation tendency according to time. From the graph, you can reset the construction/negligence duration to obtain the target degree of consolidation. Consolidation time is different when ground permeability changes. Therefore, if at the last stage the excess pore pressure does not converge to '0(zero)', you have to either reset the duration of the negligence stage or review whether accelerating consolidation method is applied or not.

Extract the Results

The screenshot shows the 'Extract Results' dialog box on the left, the 'Table' view for element 18149 in the middle, and a line graph of the excess pore pressure history on the right.

Table Data:

No	Step	Value	Elem: 18149 Center
1	Foundation:INCR=1 (LOAD=1.000)		Sorting Dialog...
2	Excavation:INCR=1 (TIME=1.000e+0)		Style Dialog...
3	Install Structure&Pile(Back filling):N		Show Graph...
4	Install Structure&Pile(Back filling):N		Export to Excel
5	Install Structure&Pile(Back filling):N	5.000000e+000	0.000000e+000
6	Install Structure&Pile(Back filling):N	1.100000e+001	0.000000e+000
7	Install Structure&Pile(Back filling):N		

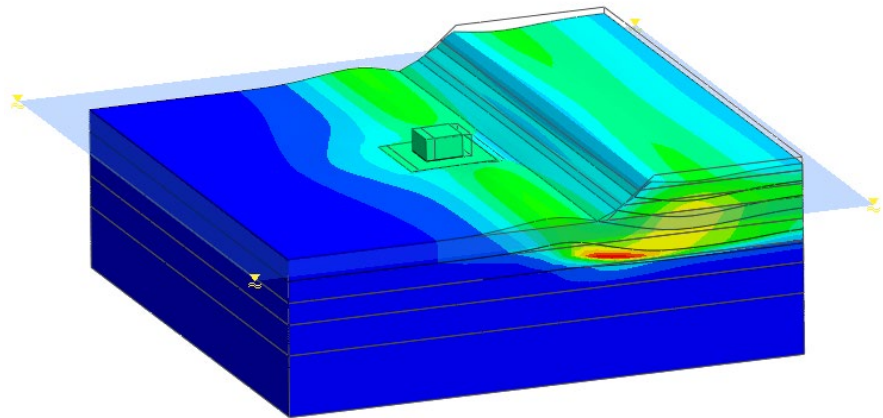
Graph Description: The graph plots the excess pore pressure (y-axis, ranging from -10 to 1) against time (x-axis, ranging from 0 to 100). The curve shows a series of peaks and troughs, indicating the fluctuation of excess pore pressure over time during different construction stages.



6.3 Verify Effect of Adjacent Structure

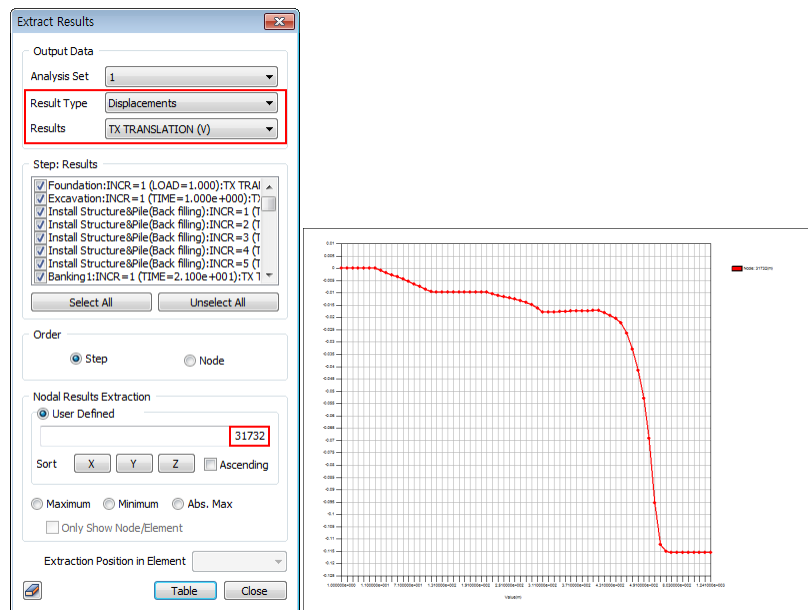
As the 'Total translation (V)' at the final stage shows, there is a concentration of displacement at the weak layer due to banking. Therefore, there is also horizontal displacement at adjacent structures.

Total Displacement



Verify the horizontal displacement generated at the adjacent structure as the consolidation progresses and the member force on bottom pile elements. Use the [Extract] function to plot the displacement in the X direction over time at the node (32962) at the top of the structure. You will observe that it has a similar tendency as the settlement graph. The maximum horizontal displacement is 20cm.

Extract Result & Graph



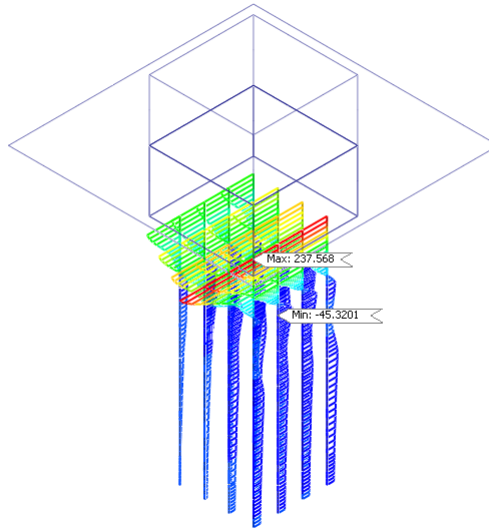


Verify the friction and displacement of the pile element.

From the pile element result, it is possible to verify the friction with the ground and the displacement in the tangent direction and two normal directions. By showing the relation of 'Displacement of each stage-friction' in a diagram, you can review whether the ultimate bearing capacity of the pile is generated or not.

- Verify the tangent direction friction between the pile and the ground by selecting Pile Forces > TANGENTIAL X of the final stage in the Work Tree. As a result of banking loading, negligence step, and adjacent structure up/down displacement during consolidation, friction is generated.

Tangent Lin Friction of Pile



Extract Result & Graph

