Chapter 6
3D Embankment Consolidation

Workflow

	: Geometry > Protrude > Extrude	8
	: Geometry > Protrude > Loft	9
	: Geometry > Divide > Solid	10
	: Geometry > Surface & Solid > Auto Connect	10
	: Mesh > Generate > 3D	11
	: Mesh > Element > Parameters	12
	: Mesh > Generate > 1D	13
	: Mesh > Element > Pile/Pile Tip	13
	: Seepage/Consolidation Analysis > Load > Self Weight	15
	: Seepage/Consolidation Analysis > Boundary > Constraint	15
	: Seepage/Consolidation Analysis > Boundary > Draining Condition	16
	: Seepage/Consolidation Analysis > Boundary > Non Consolidation)	17
	: Seepage/Consolidation Analysis > Boundary > Change Property	17
	: Seepage/Consolidation Analysis > Construction > Stage Set	18
	: Analysis > Analysis Case > General	20
	: Perform Analysis	20

3D Embankment Consolidation

Section 1
Overview

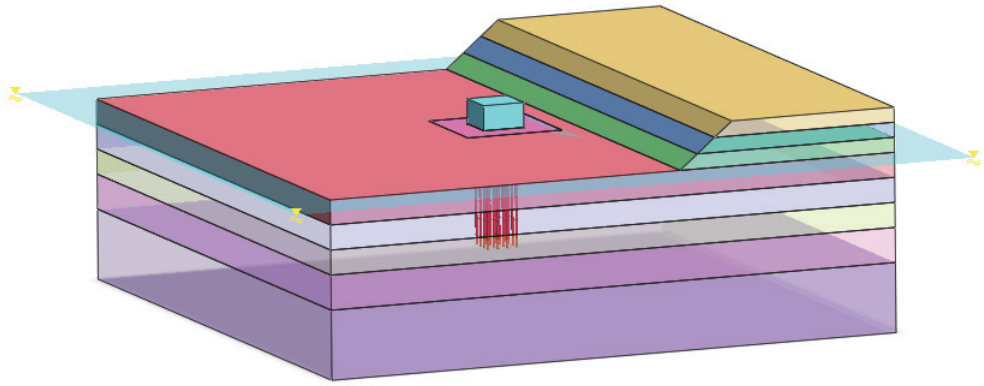
1.1 Learning Purpose

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

In Consolidation, generally excess pore water pressure inside the ground decreases with time, and this makes the effective stress increase. Water level locates at the top of the weak stratum which has relatively lower permeability.
When banking is loaded, the water which bears load because of initial undrained condition will slowly leak out and pass the load to soil particles. During this process settlement happens in gravity direction and this settlement will influence the stability of adjacent structures.

Consolidation analysis is related to Drained/Undarined analysis. According to drainage condition, the density, rigidity, and strength of the ground increase during the consolidation process.

▶Model overview



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

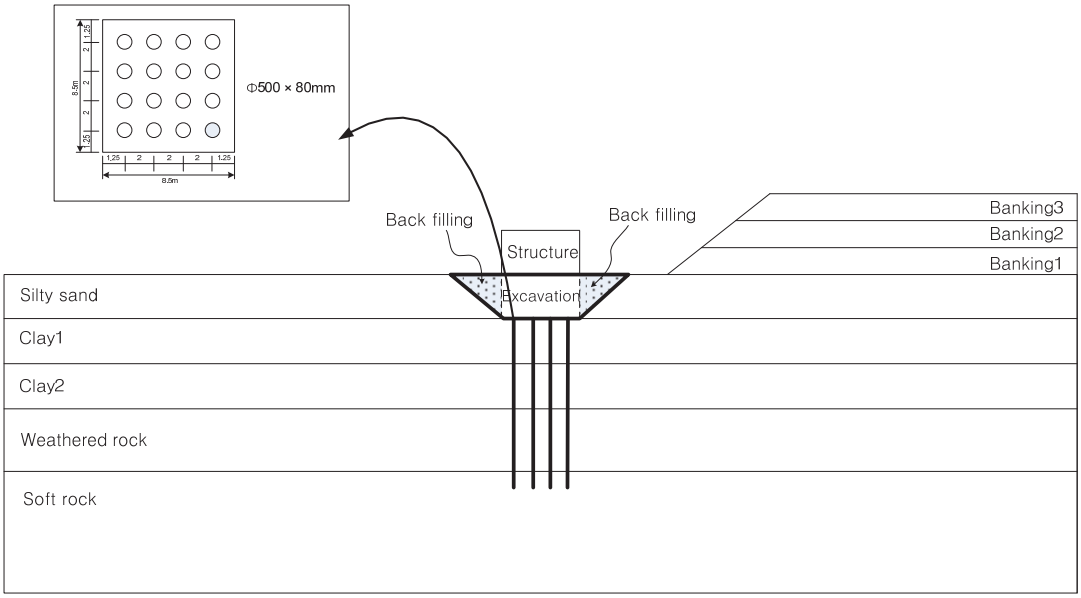
- Weak stratum - Apply Modified Cam Clay model to simulate the limit state behavior
- Modeling Pile elements (verify settlement of weak stratum according to pile skin friction)
- Set construction stages (time step) for consolidation analysis
- Analyze excess pore water pressure distribution/consolidation settlement according to time
- Verify Structure's settlement and the pile skin friction according to time

1.2 Model and Analysis Summary

In this tutorial, pile foundation and structure are located at the top of soft rock layer and the 9m banking is divided into 3 layers to carry out construction stage consolidation analysis. When setting construction stages, set time step of each stage by considering banking height (3cm) per day. Also leave a negligence period after banking at each construction stage to ensure the degree of consolidation. To minimize the influence of boundary conditions, set ground area over twice larger than excavation area.

Strata composition and model are illustrated in the following image.

▶Cross-section diagram



Section 2

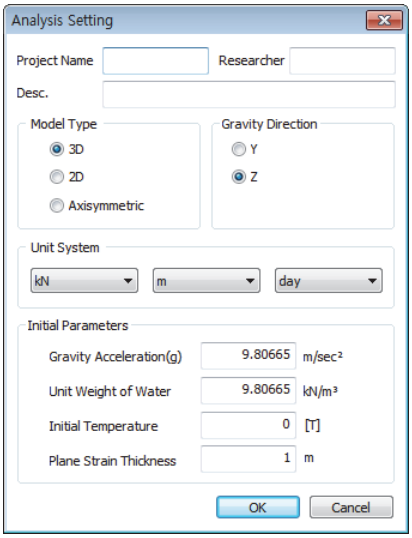
Analysis Setting

[Open the attached start file (06_Consolidation_start)]

* : Analysis > Analysis Case > General

Set Model type, Gravity direction and Initial parameters. Check the Unit system which will apply to the analysis. The unit system can be changed both during the modeling process and after performing analysis. The input parameters will automatically convert to the right unit system.

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



Section 3

Define Material and Property

Define ground materials and properties, and assign them to mesh sets during mesh generation.

3.1 Definition of Ground and Structural Materials

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

Ground and structure materials are defined in the following table

▶Table. Ground material

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
General							
Elastic modulus(E)	1.0E+04	1.0E+04	5.0E+04	5.0E+04	1.5E+05	3.0E+05	2.0E+07
Poisson's Ratio(v)	0.45	0.4	0.25	0.3	0.2	0.15	0.2
Unit Weight(r)	17	17	19	18	20	23	25
Ko	0.5	0.5	0.74	0.6	1	1	1
Prous							
Unit Weight (Saturated)	20	20	21	20	21	23	-
Initial Void Ratio(eo)	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(ck)	0.5	0.5	0.5	0.5	0.5	0.5	-
Non-Linear							
Over Consolidation Ratio(OCR)	1	1	-	-	-	-	-
Slope of Consol Line(λ)	0.221	0.32	-	-	-	-	-
Slope of Over Consol Line(k)	0.015	0.028	-	-	-	-	-
Slope of Critical State Line(M)	0.18	0.18	-	-	-	-	-
Pc	-	-	-	-	-	-	-
Allowable Tensile Stress	0	0	-	-	-	-	-
Cohesion	-	-	19	20	100	200	-
Frictional Angle	-	-	25	25	33	38	-

Tip

Coefficient of infiltration according to void ratio
- When there is a pressure on the ground, both the ability of infiltration (Coefficient of infiltration) and the void ratio decrease. To simulate this behavior, define 'Void ratio dependency of permeability (ck)'. According to 'Saturated coefficient water permeability (ksat)' and void ratio, the relation is described in the following formula.

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

In [Consolidation] and [Fully Coupled Stress Seepage] analysis, it is recommended to consider not only 'ck' but also 'partially saturated effects for Stress analysis' to get a more realistic approximate solution. If you are not considering instauration and partial saturation, self weight is considered depend on whether pore water pressure is above or below '0(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

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

The effect of partial saturation reflects to calculating self weight, undrained rigidity and inner force of element. By using the above options, it is possible to get exact value which considered water level change according to drainage and void ration (permeability) change of ground deformation.

Tip

[Allowable tension stress]
Generally MCC material model does not allow tensile stress in fracture criterion (relation of Stress-strain ratio). But during the consolidation process, tensile stress such as heaving by load from banking and arching by excavation can be generated. To overcome the limitation of material model and to extend the application to reality, in GTS NX is programmed to perform analysis about tensile stress, if they are generated in range of 'Allowable tensile stress'.
Value of allowable tensile stress is not a fixed value, so to input bigger value than tensile stress which generated from overburden load (Banking) and fracture behavior, repetitive analysis is necessary. If the analysis is interrupted because the results diverge according to the tensile fracture, you can set 'Allowable tension stress'.
If 'pc (Preconsolidation load)' is directly inputted, the value of 'Allowable tension stress' cannot go over 'pc' value. If 'pc' value is defined through 'OCR', the software automatically calculates 'pc' value according to the inputted 'allowable tension stress' value.

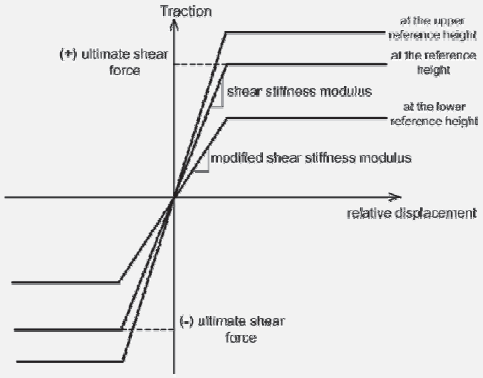
▶Table. Structure material

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

Normal Stiffness Modulus(kn)	-	115789

Tip

Behavior of pile elements can be divided to behavior on two normal directions and behavior on one tangent direction. The normal behavior considers piles and the surrounding ground as a whole rigid body, whereas the tangent behavior is a nonlinear elastic behavior. The nonlinear elastic behavior can be defined by yield force or yield function.
The graph bellow represents the relative displacement between the 2 bodies and the friction when yield force is defined. If the relation is defined by a function, more precise results can be obtained.



The pile tip element work as solid-point interface that presents the interactive behavior between the ground elements and the pile nodes. In the element coordinate system of the pile tip element, behavior in normal direction is considered as rigid behavior. And behavior in tangent direction is considered as nonlinear elastic behavior, just like for a pile element.

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

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

3.2 Definition of Properties

Properties represent 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 just like in this tutorial, pile tip bearing capacity and rigidity modulus can be defined.
Use beam element for structure member of PHC pile which resists to axial, shearing and bending force. It is possible to verify friction behavior and relative displacement with adjacent ground by defining rigidity around and at the tip of the pile. Since pile element, same as interface element, does not share node with adjacent ground, you can easily apply it in 3D model.

Properties for ground mesh sets are defined as below:

▶Table. 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 below (stiffness will be automatically calculated when the section shape is defined)

▶Table. 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

[Start modeling]

This tutorial focus on 3D geometry and elements generation, consolidation behavior analysis and results output. Start 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

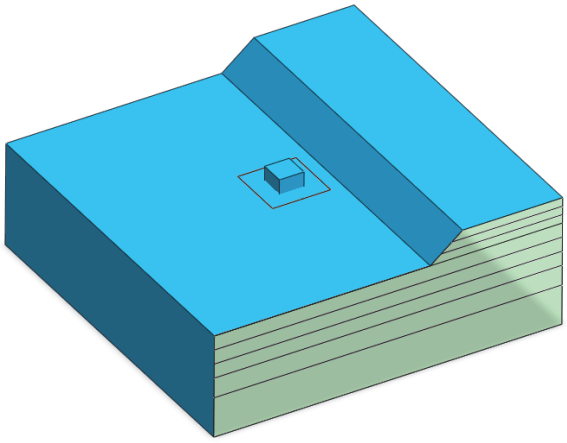
∗ : **Geometry > Protrude > Extrude**

This process creates line/face/solid shapes from lower shapes such as point/edge/face. With lines which constitute a closed domain, it is also possible to make a solid at once.

Extrude 3D solid of the ground and structures and create dividing surface from the dividing lines to classify each stratum.

- Select ground faces (1).
- Choose the direction to 'Y' and input 120(m) in length.
- Click [Apply]. Check generated solid.
- Select structure faces (1).
- Choose direction to 'Z', and input 10(m) in length. Click [Apply].

▶Generate ground/structure solid



Tip

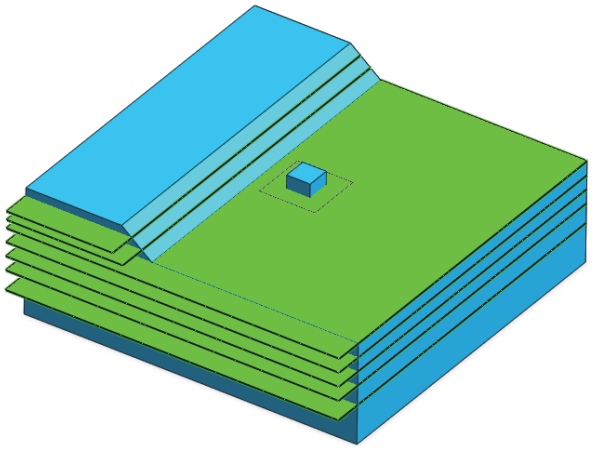
When selecting object or direction, you can both use Work Tree and select directly in the Work Window.

- Change [Selection Filter] to 'Line', Select dividing lines (7).
- Select the direction to 'Y' and input 130(m) to length which is big enough to divide solid.

Tip

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

▶Generate dividing surface



* : **Geometry > Protrude > Loft**

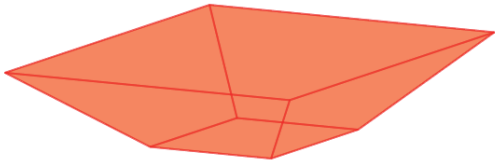
[Loft] is used to create a solid by linearly connecting two separated faces. Generate excavation area for installing structure with [Loft] function.

Tip

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

- Select faces (2) of structure and excavation area. Click [OK].
- With [Preview] function you can check generated shape.

▶Generate excavation area

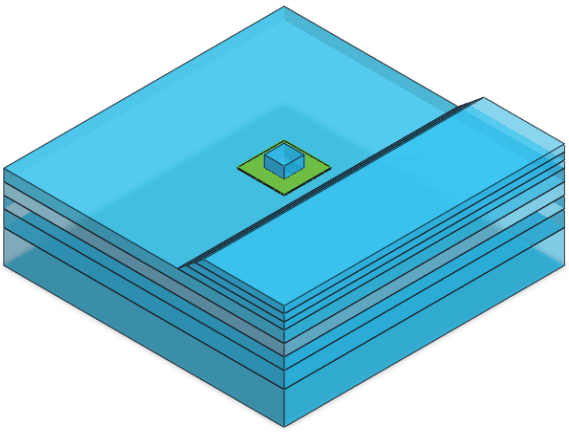


* : Geometry > Divide > Solid

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

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

▶Separate ground



* : Geometry > Surface & Solid > Auto Connect

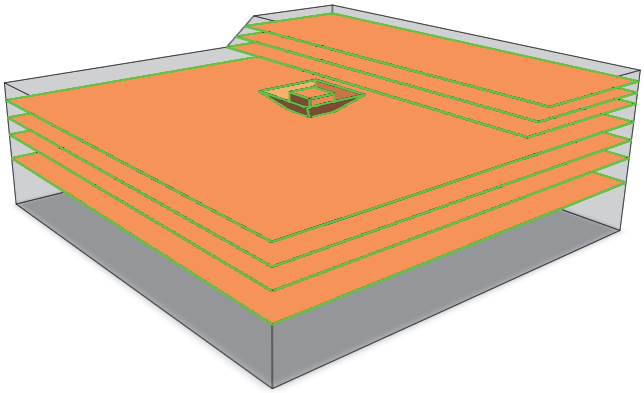
This process generates shared face automatically after generating 3D geometries. Shared faces must be created before mesh generation to connect nodes.

- Select all the generated solid (11), Click [OK].

Tip

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

▶Check share face



4.2 Generate Meshes

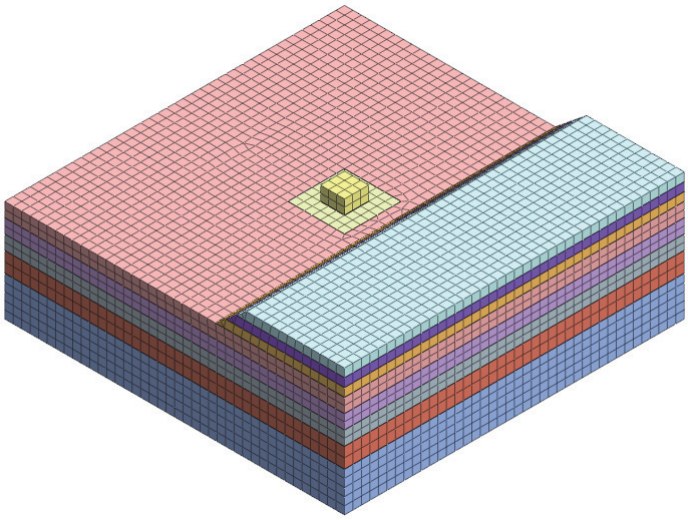
Mesh shape and mesh quality are very important in finite element analysis. For stable analysis, it is recommended to use hexahedron centered meshes for 3D and tetrahedron centered meshes for 2D to the ground where plastic failure occurs according to compressive and shearing deformation. Use Map mesher or hexahedron centered Hybrid Mesher to create meshes.

∗ : **Mesh > Generate > 3D**

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

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

▶Generate ground element



* : Mesh > Element > Parameters

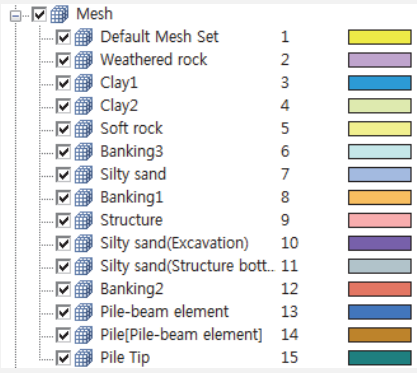
Change properties for each mesh set and Check. You can generate all the mesh sets in one property first and change the properties later in [Parameter].

- Select [3D] tab.
- Change the properties as shown in the Model Overview (cross-section diagram).
- Select each mesh set, and assign the relevant property.
- 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.

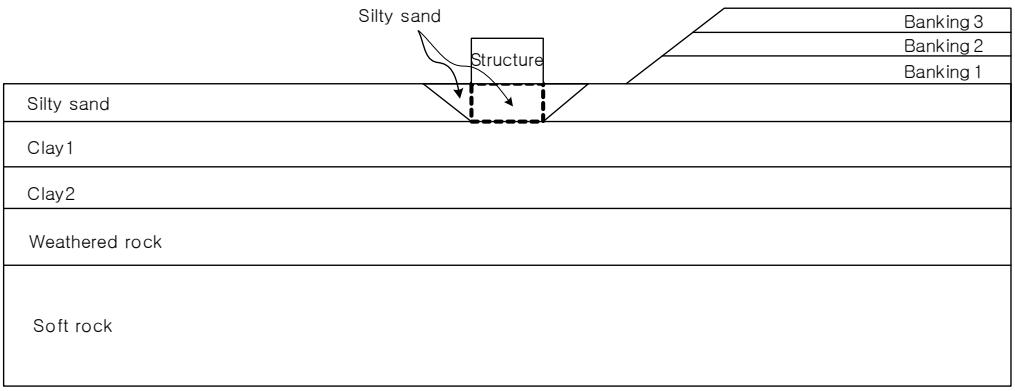
Tip

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



<Change mesh set name>

▶Overview(Property of each area)

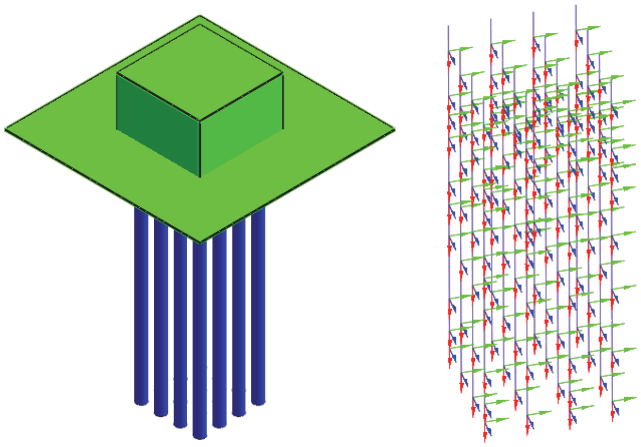


* : Mesh > Generate > 1D

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

- Select entire pile lines (16) and input '1' in [Division].
- Choose [PHC pile] property.
- Name mesh set to 'Pile-beam element'. Select [OK].

▶Generate beam element
▶▶Align beam element coordinate system



Tip

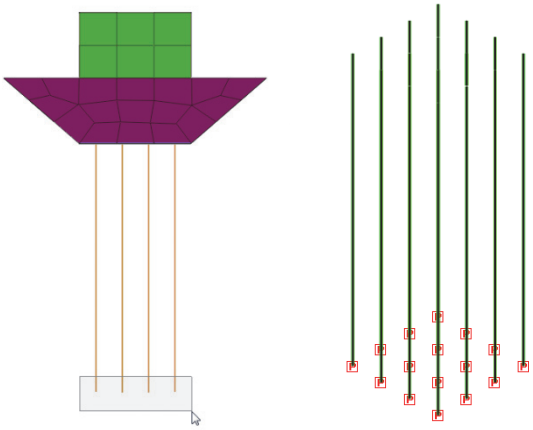
The size and division of the beam elements before generation of pile elements are not important. It is because the beam element nodes and the ground nodes are automatically connected when the 'Pile elements' are generated.
To output result in the element coordinate system, coordinate system of each structure member element should be aligned in one direction. Otherwise the result on this element will have an opposite. Element coordinate system can be change in Mesh > element > Parameter. For further information refer to [User Manual] or [Online Manual (F1)].

* : Mesh > Element > Pile/Pile Tip

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

- Choose [Pile] tab.
- Select all the generated pile-beam elements (16).
- Select 'Pile' property. Click [Apply].
- Choose [Pile-tip] tab.
- Select nodes (16) of pile tip as following image.
- Select [Pile-tip] property. Select [OK].

▶Generate pile/pile tip element



Tip

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

Section 5
Analysis Setting

5.1 Setting Loading Condition

∗ : **Seepage/Consolidation Analysis > Load > Self Weight**

Gravity is calculated automatically by multiplying the inputted unit weight of the ground, the structure geometry and the acceleration of gravity. It can be easily set by inputting a scale factor of direction. The default value of the gravity direction is set.

- Put -1 for Gz value. Enter 'Self weight' as the [Load set] name.

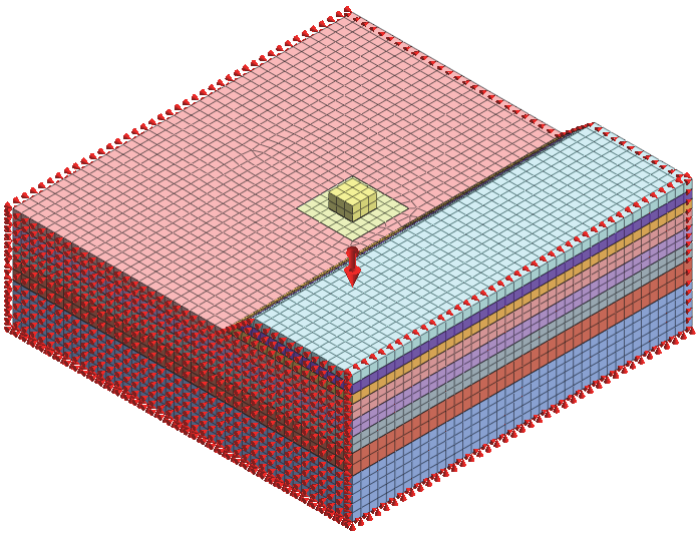
5.2 Setting Boundary Conditions

∗ : **Seepage/Consolidation Analysis > Boundary > Constraint**

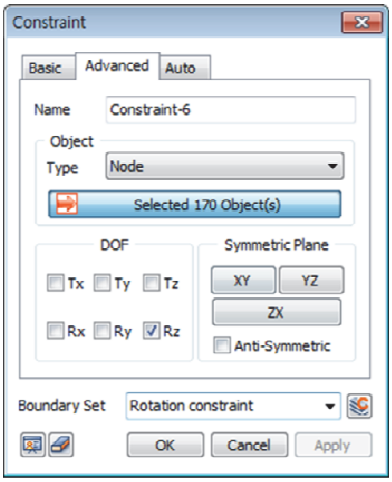
This process sets boundary conditions against internal deformation or rotation based on GCS.
For boundaries of the entire model, automatically set constraints of left/right/bottom displacements according to GCS. Constrain rotation in Rz direction to prevent degree of freedom error of pile elements according to torsion

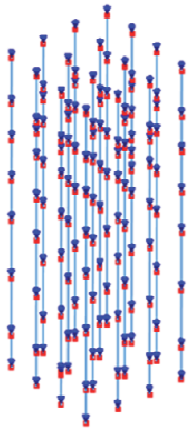
- Select [Auto] tab.
- Check [Consider All Mesh Sets] and name boundary set to 'Ground boundary'.
- Click [Apply].
- Select [Advanced] tab.
- Select the type to 'Node' and select entire node of generated pile element. Check 'Rz'.
- Name boundary set to 'Rotation constraint'. Select [OK].

▶Generate ground boundary condition



▶Setting pile rotation constraint





∗ : Seepage/Consolidation Analysis > Boundary > Draining Condition

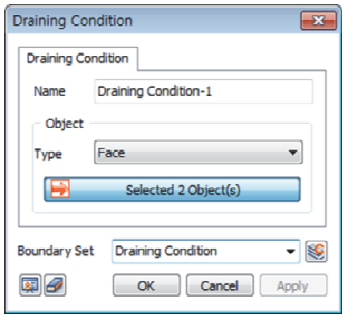
This process sets draining condition for excess pore water pressure. In case no extra drainer is needed, generally set 'both-side drainage' or 'single-side drainage' at the clay layer, which has relatively lower permeability. But in this way 'complete drainage' assumption is used, which neglects permeability of other ground materials around the clay layer. So if you want to consider permeability of the surrounding ground too, set 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 drainage condition at the left/right/bottom with ground constraint boundary condition. However in this tutorial we will simply set draining conditions at the surface and bottom sides of the clay layer.

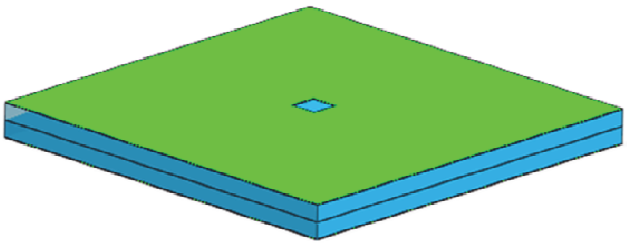
- Select Clay 1, 2 to show only.
- Set the type to 'Face'.
- Select top of 'Clay1' and bottom of 'Clay2' as following image.
- Name the boundary set to 'Draining condition'.
- Click [OK]. Check generated draining condition.

Tip

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

▶Generate draining condition





* : Seepage/Consolidation Analysis > Boundary > Non Consolidation

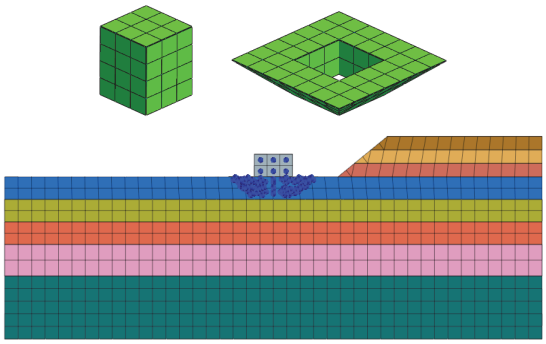
When performing consolidation analysis, all the solid elements get DOF of the pore water pressure. Therefore, for elements which ignore consolidation behavior such as concrete structure or elements which are not directly affected by consolidation, they should be defined as non-consolidation elements.

Although non-consolidation is sometime set to bank materials above water level as loading condition, however in case of a precise analysis which considers water level change and partial saturation, non-consolidation should not be set to the lower part of the bank to simulate the water level change. In this tutorial, set non-consolidation is only used on concrete structure.

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

- Select meshes: [Structure, Silty sand (Structure), Silty sand (Backfilling)].
- Name boundary set as'Non consolidation'. Click [OK].

▶Generate non-consolidation



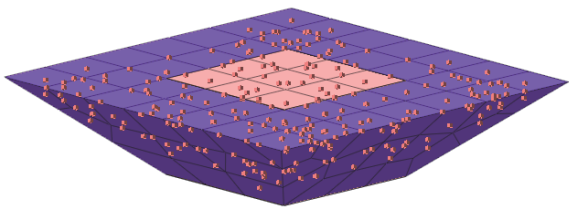
* : Seepage/Consolidation Analysis > Boundary > Change Property

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

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

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

▶Generate boundary condition for changing property



5.3 Define Construction Stages

* : Seepage/Consolidation Analysis > Construction > Stage Set

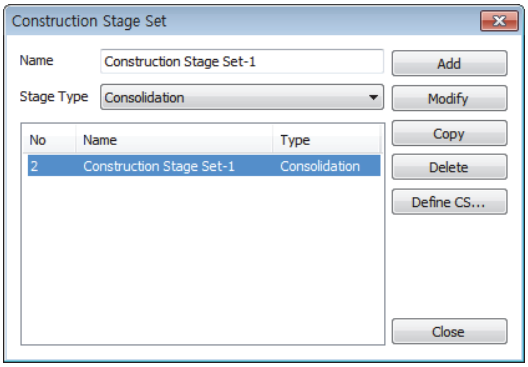
Set construction stage to verify results of each stage. For consolidation analysis, you can set 'time step' for each step and verify the result according to time.

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

In construction stage setting, data once activated will sustain the same status until you choose to deactivate it. Set water level condition at Stage1. If the water level located at the specific level, you can simply define it by inputting height.

- Name construction stage set.
- Set stage type to [Consolidation]. Click [OK].
- Click [Define constriction stage] to create construction stage.
- Construction stages are defined below

▶Setting construction stage set



Stage 1 - Name: Original Ground

- Activated data-Mesh : [Clay1], [Silty sand(Structure)], [Silty sand(Excavation)], [Silty sand], [Clay2], [Weathered rock], [Soft rock]
- Activated data-Boundary Condition : [Ground boundary]
- Activated data-Static load : [Self weight]
- Check [Define Water Level for Global], and input '36(m)'.
- Check [Clear Displacement].
- Click [Time Step]. Set both duration and step number to '1'.
- Check [Save Results]. Press [Generate step] button and click [OK].
- Click [Save] and press [New] button to generate next construction stage.

Tip

In consolidation analysis linear analysis about initial self weight load is needed. Generally the stress status of the original ground is within the linear range, so we apply the load at one step. Therefore define 'Duration' and 'Step number' as '1'.

Stage 2 - Name : Excavation

- Deactivated data-Mesh : [Silty sand(Excavation)], [Silty sand(Structure)]
- Click [Time Step]. Set both duration and step number to '1'.
- Check [Save Results]. Press [Generate step] button and click [OK].
- Click [Save] and press [New] button to generate next construction stage.

Stage 3 - Name: Install structure and pile (Backfilling)

- Activated data-Mesh : [Silty sand(Excavation)], [Silty sand(Structure)], [Structure], [Pile-beam element], [Pile tip], [Pile[Pile-beam element]]
- Activated data-Boundary Condition : [Non consolidation], [Change property], [Draining condition], [Rotation constraint]
- Click [Time Step]. Set duration and step number to '10(day)'and'5'.
- Check [Save Results]. Press [Generate step] button and click [OK].
- Click [Save] and press [New] button to generate next construction stage.

Stage 4 - Name : Banking1

- Activated data-Mesh : [Banking1]
- Click [Time Step]. Set duration and step number to '100(day)'and'10'.
- Check [Save Results]. Press [Generate step] button and click [OK].
- Click [Save] and press [New] button to generate next construction stage.

Tip

Set banking quantity to 3cm per day and set duration of each construction stage (100 days). Also assign same duration of negligence stage

Stage 5 - Name: Negligence 1

- Click [Time Step]. Set duration and step number to '100(day)'and'10'.
- Check [Save Results]. Press [Generate step] button and click [OK].
- Click [Save] and press [New] button to generate next construction stage.

Stage 6 - Name : Banking 2

- Activated data-Mesh : [Banking2]
- Click [Time Step]. Set duration and step number to '100(day)'and'10'.
- Check [Save Results]. Press [Generate step] button and click [OK].
- Click [Save] and press [New] button to generate next construction stage.

Stage 7 - Name: Negligence 2

- Click [Time Step]. Set duration and step number to '100(day)'and'10'.
- Check [Save Results]. Press [Generate step] button and click [OK].
- Click [Save] and press [New] button to generate next construction stage.

Stage 8 - Name : Banking 3

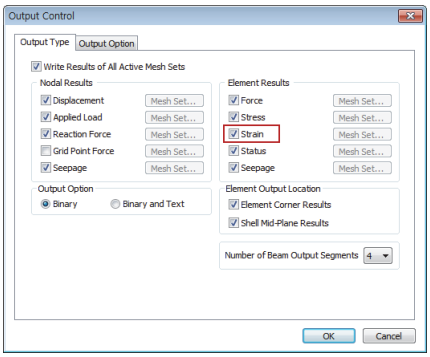
- Activated data-Mesh : [Banking3]
- Click [Time Step]. Set duration and step number to '100(day)'and'10'.
- Check [Save Results]. Press [Generate step] button and click [OK].
- Click [Save] and press [New] button to generate next construction stage.

Stage 9 - Name: Final (Complete consolidation)

- Click [Time Step]. Set duration and step number to '2 year (730 days)'and'10'.
- Check [Save Results]. Press [Generate step] button and click [OK].
- Click [Save] and [Close]

5.4 Setting Analysis Case

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



* : Analysis > Analysis Case > General

- Input name and choose the solution type to 'Construction stage'.
- In Analysis Control > General tab > Initial stage for stress analysis, set the initial stage to '1: Original Ground'. Do not check [Apply K0 Condition].
- Check [Automatically Consider Water Pressure] to consider water pressure below water lever when excavating.
- Click [OK] to add analysis case.

Tip

If you use Ko condition, vertical stress and horizontal stress of initial foundation are calculated by theoretical static coefficient of earth pressure (K0). When considering nonlinear (Undrained) behavior characteristic about excavation and load at weakness layer below water level, because it is reasonable to set initial stress by Poisson's ratio of each stratum, generally K0 condition is not used in consolidation analysis.

5.5 Perform Analysis

After the analysis is done, the software automatically switches to [Post-Mode] (checking results). To modify model and option after the analysis, you have to switch back to the [Pre-Mode].

* : **Analysis > Analysis > Perform**

- Perform analysis.

Tip

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

Section 6
Results

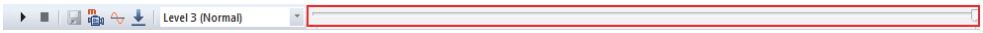
You can verify displacement of each construction stage and each time interval. Results of ground deformation, excess pore water pressure, degree of consolidation, deformation of adjacent structure, and member force of piles can be verified from the [Result Tree]. All the results can be output as contour, table, and graph. In this tutorial, the main result items which need to be checked are as following below.

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

6.1 Verify Displacement

Verify by 'Displacement' from the work tree after the analysis. TX, TY, TZ signify displacements in 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.

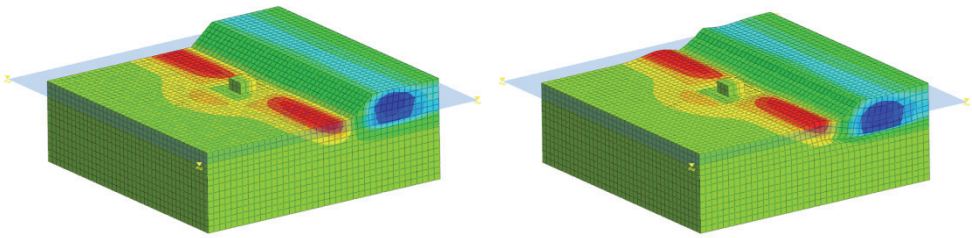
- In the consolidation analysis which is applied by time step, it is important to verify as time passes whether the settlement, pore water pressure dissipation converges or not. By moving the simulation bar at the bottom of the work window, it is possible to see the change of settlement at each construction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



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

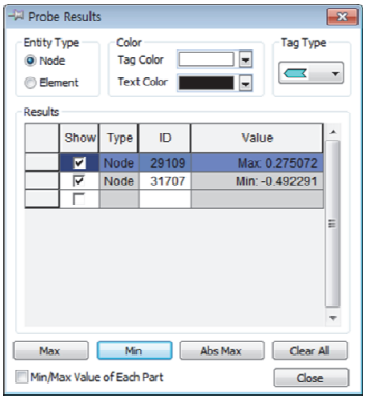
- Select the last stage/last time step in the Work Tree, and select Displacement > TZ TRANSLATION (V).
- In Result > General > Deform, it is possible to see unreformed/deformed shape directly in Z direction. (It is possible to modify degree of deformation by scale at the property window. And you can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

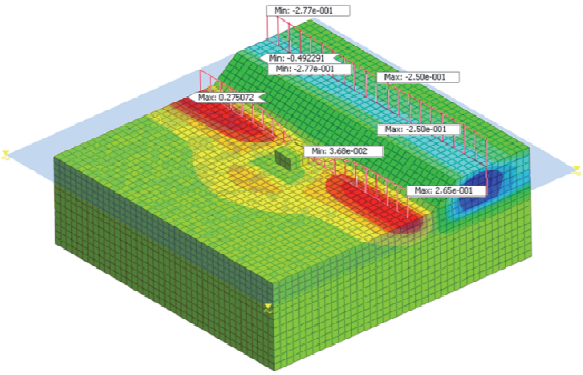
▶Vertical displacement (Unreformed)
▶▶Vertical displacement (Deformed)



- 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 specific line (face).

▶Probe



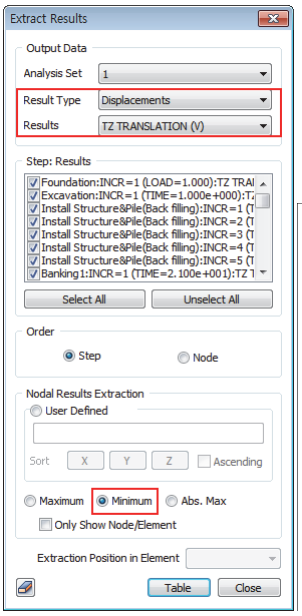


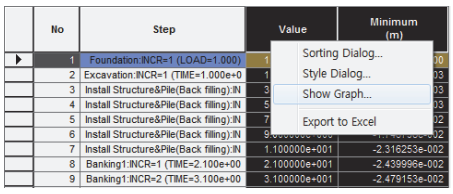
Verify time history result of settlement in each stage and each time step.

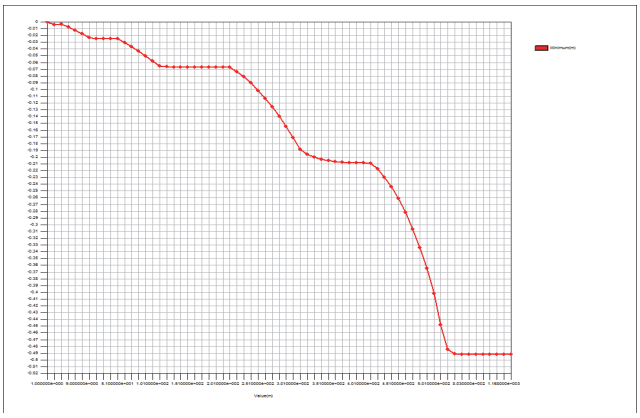
- By using Result > Advance > Extract, you can extract tables and draw graphs of results at selected. In consolidation analysis, since the final settlement is a convergent result after complete dissipation of excess pore pressure, verification of 'time - settlement' graph is needed. Only checking max/min value in the legend after the analysis is not enough.

- Select Result > Displacement > TZ, check 'Minimum' to see the maximum settlement (-). Select [Table]. By mouse right click in the plotted table, you can plot graph. In Banking stage and negligence stage, it is possible to verify the increase and convergence of settlement. Also in 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 negligence duration if the settlement still shows increase at the final stage.

▶Extract
▶▶Extract table
▶▶▶Show graph





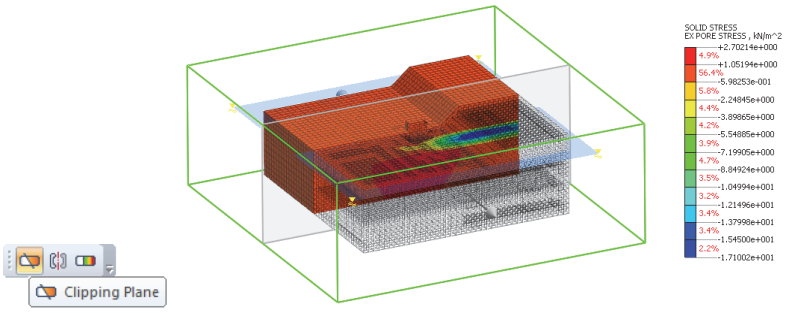


6.2 Verify Stress

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

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

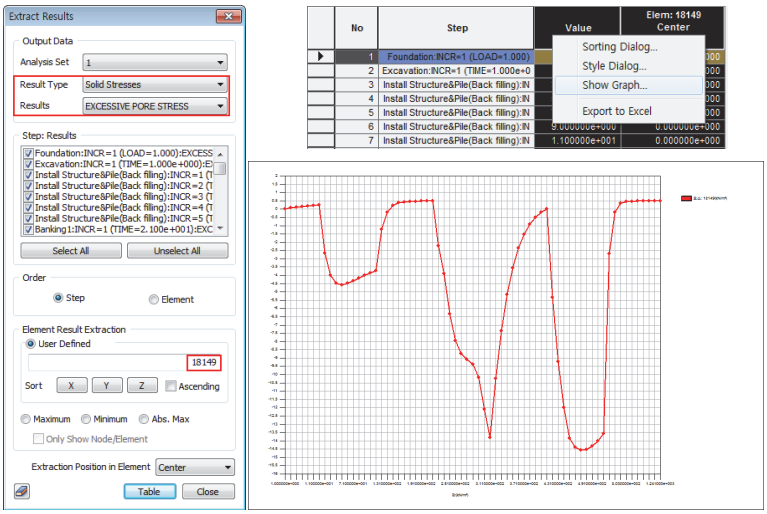
▶Horizontal Stress of the ground SYY
▶▶Diagram of SYY of each depth



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

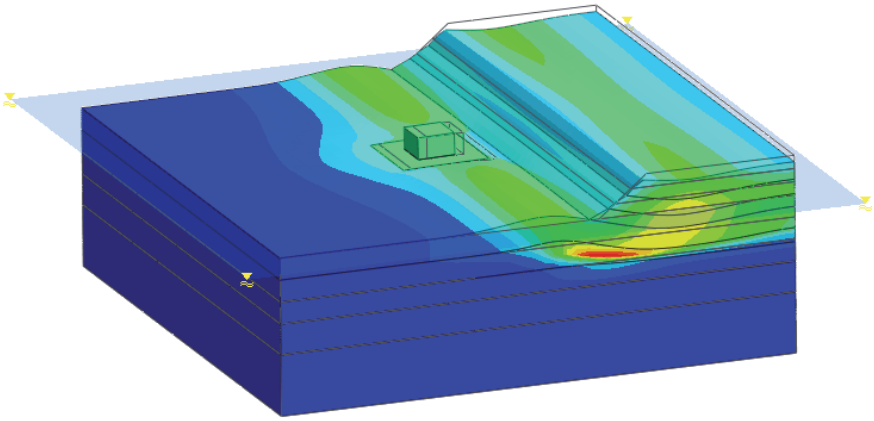
- In consolidation, settlement and excess pore pressure dissipation are important items for calculating construction period. To figure out time where consolidation is completed, verify distribution of excess pore pressure according to time.
- Select Results Tree > Solid Stresses > EXCESSIVE PORE STRESS to verify history of excess pore pressure dissipation. Choose element (18149 in the tutorial) where excessive pore pressure concentration occurs because of surface load. Input the element ID and press [Table] button. By mouse right click in the plotted table, you can plot graph. After the banking, excess pore pressure increases. In negligence stage, you can figure out dissipation tendency according to time. And from the graph, you can reset 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 duration of negligence stage or review whether accelerating consolidation method is applied or not.

▶Extract result
▶▶Extract graph
▶▶▶Show graph



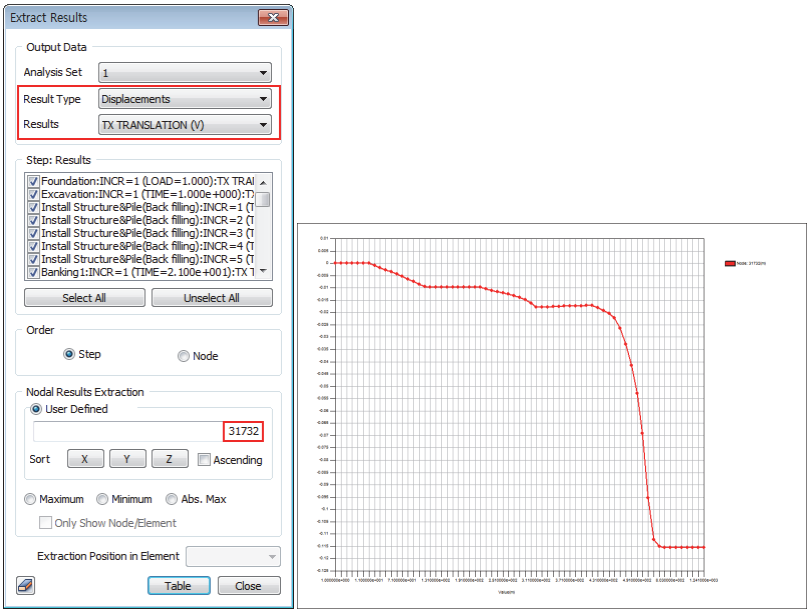
6.3 Verify Effect of Adjacent structure

As the 'Total translation (V)' at the final stage shows as below, there is concentration of displacement at weak layer because of banking. Therefore there is also horizontal displacement at adjacent structure.



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

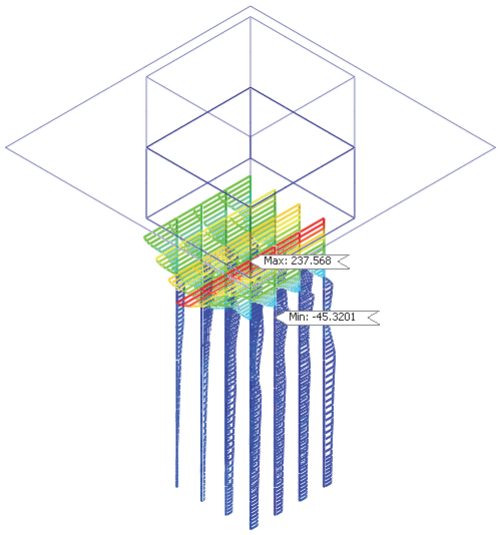
▶ Extract result
▶▶ Show graph

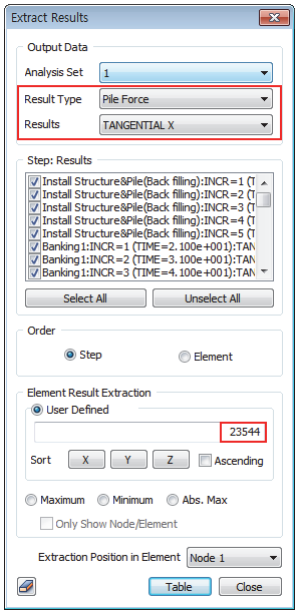
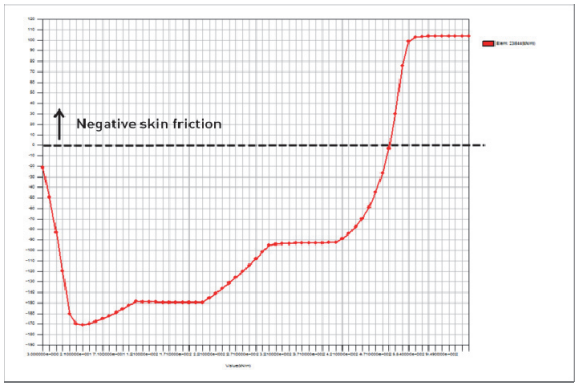


Verify friction and displacement of pile element.
From pile element result, it is possible to verify friction with ground and displacement in the tangent direction and two normal directions. By showing relation of 'Displacement of each stage-friction' by diagram, you can review ultimate bearing capacity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

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

▶Tangent line friction of pile



▶Extract result

▶▶Show graph