Chapter 7

3D Abutment Foundation Pile

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	Define Material of Ground and Structure	4
3.2	Define Properties	6
Section 4	**Modeling**	**7**
4.1	Modeling Geometry	7
4.2	Generate Mesh	12
Section 5	**Analysis**	**16**
5.1	Setting Load Condition	16
5.2	Setting Boundary Conditions	17
5.3	Define Construction Stages	19
5.4	Setting Analysis Case	23
5.5	Perform Analysis	24
Section 6	**Results**	**25**
6.1	Verify Displacement	25
6.2	Verifying Member Forces & Stresses	31
6.3	Verify Frictional Force and Displacement of Pile/Ground	33

3D Abutment Foundation Pile

Section 1

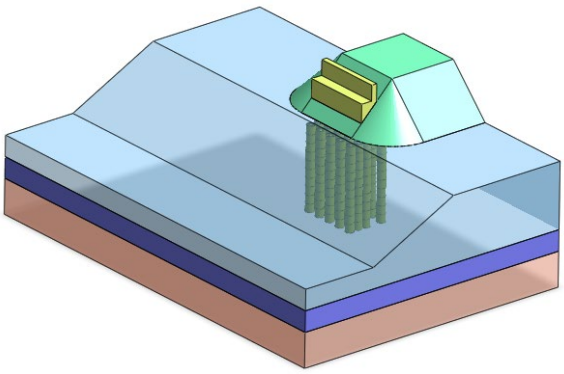
Overview

1.1 Learning Purpose

This tutorial will cover the evaluation of horizontal displacement of bridge abutment foundation and pile stability. The assessment of soil-structure behavior utilizes pile elements rather than simple beam elements. Pile elements, resembling embedded beam elements, do not necessitate node sharing, facilitating their incorporation into 3D modeling. Additionally, pile elements are well-suited for assessing soil-structure behavior as they can account for the interface influence between piles and adjacent ground.

The tutorial introduces a 3D model of the bridge abutment structure. Step by step, it verifies displacements, foundation integrity, and member forces of pile elements based on the abutment and applied loads. The results will be analyzed under two distinct scenarios: initially, only the abutment's influence will be considered, followed by the addition of piles to further mitigate ground displacement beneath the abutment. The model can accommodate two construction stage analysis cases. Furthermore, a gauging shell will be applied to the upper abutment section to assess structural member forces affecting the foundation plate due to applied loads.

Analysis Model Overview



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

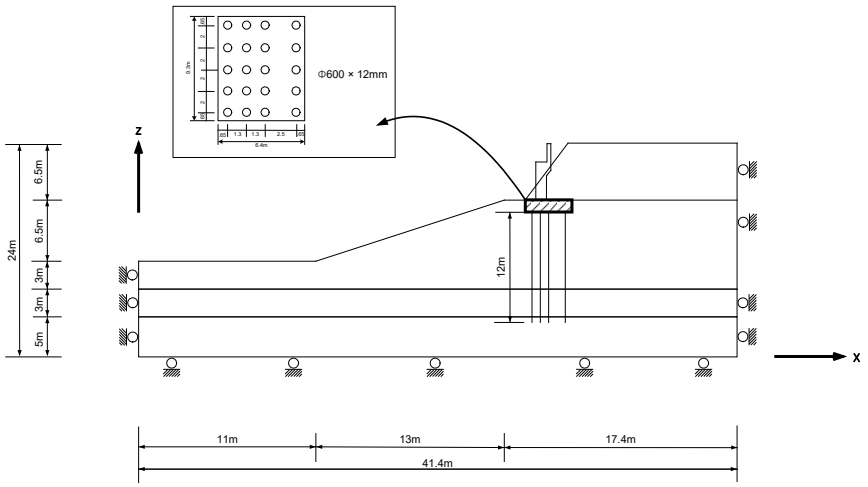
- 3D geological stratum modeling (Mohr-Coulomb Law)
- Modeling of pile elements (Checking the friction at the surface of piles caused by the load application)
- Setting the load steps
- Evaluating horizontal deformation of the abutment at each construction step
- Drawing the result graph of Load-Settlement for each construction stage
- Application of Gauging shell elements (Evaluating the structural forces around solid elements)

1.2 Modeling and Analysis summary

This tutorial involves a model comprising three layers of stratum and an abutment foundation pile situated adjacent to a sloped site. Subsequently, a construction analysis is conducted in accordance with the abutment construction and loading. The abutment is constructed on a foundation slab measuring 6.4x10m², with a load of 100kN/m² applied on the embankment. This load will be distributed and applied in 5 construction stages. For each stage, horizontal displacement and settlement influence will be examined. Concurrently, 20 steel pipe piles of 600x12mm dimensions will be introduced, and their impact on settlement will be assessed. By segmenting the analysis into 2 construction stages representing pre- and post-pile application, it becomes feasible to evaluate the influence of the piles.

The model of the geological stratum is depicted below:

Cross-Section



Section 2

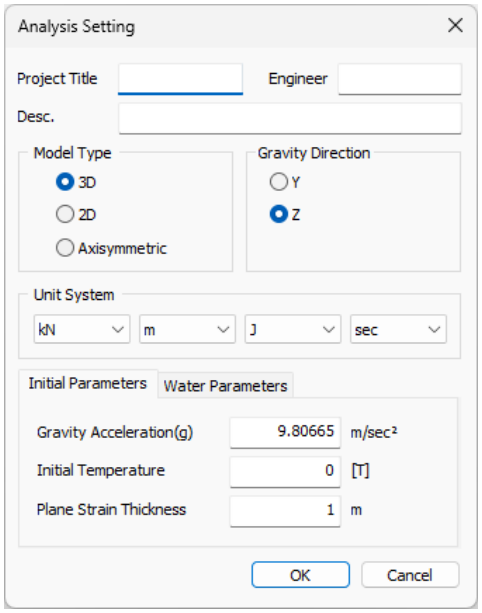
Analysis Setting

2.1 Starting with Analysis Setting

Configure the model type, gravity direction, initial parameters, and unit system for the analysis. The unit system is flexible and can be adjusted at any point during the modeling process or after conducting the analysis. Input parameters will be automatically converted based on the selected unit system.

This tutorial employs a 3D model with gravity acting in the Z direction and utilizes the SI unit system.

Analysis Setting Window



Section 3

Material and Property

3.1 Define Material of Ground and Structure

Define the material properties for both the ground and structure, and generate a mesh with properties assigned to each mesh.

For the ground material, the 'Mohr-Coulomb' model, commonly employed for analyzing elastic behavior, has been utilized. For the structure material, both the 'Elastic' model, which does not account for material nonlinearity, and the 'Pile Element' model, for assessing skin friction force, are employed.

The materials for the ground and structure are outlined in the following table:

Ground Material

Name	Weathering soil	Weathered rock	Soft rock	Embankment	Abutment
Material	Isotropic	Isotropic	Isotropic	Isotropic	Isotropic
Model Type	Mohr Coulomb	Mohr Coulomb	Mohr Coulomb	Mohr Coulomb	Elastic
Elastic Modulus	1.2E+04	1.2E+05	1.2E+06	3.0E+04	2.1E+07
Poisson's Ratio	0.33	0.3	0.25	0.3	0.18
Unit Weight	19	20	24	18	25
Ko	0.5	0.5	0.74	0.5	1
Cohesion	2	34	200	15	-
Frictional Angle	28	33	37	25	-

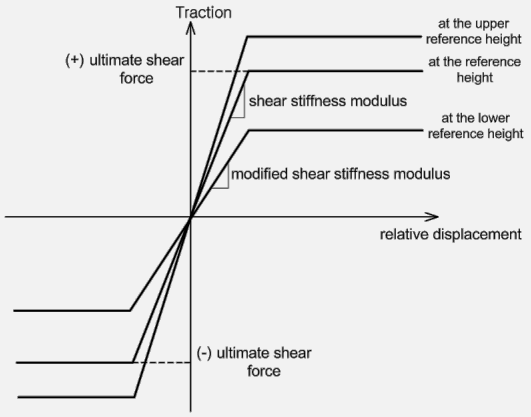
Structure Material

Name	Pile	Pile(Interface)
Material	Isotropic	Interface and Pile
Model Type	Elastic	Pile
Elastic Modulus	2.10E+08	-
Poisson's Ratio	0.3	-
Unit Weight	74	-
Ultimate Shear Force	-	650
Shear Stiffness Modulus	-	50000
Normal Stiffness Modulus	-	500000

Tip

The behavior of pile elements can be categorized into normal and tangential behaviors. Initially, the normal behavior between the pile and the surrounding ground is considered fixed and rigid, whereas the tangential behavior exhibits nonlinear elastic characteristics. This nonlinear elastic behavior is further delineated into the yield force and the assigned yield function.

The graph below illustrates the relative displacement between the two bodies and the friction when the yield force is defined. If the relationship is governed by a function, the correlation between relative displacement and friction can be defined with greater precision.



The pile tip element functions as a solid-point interface, representing the relative behavior between the ground elements and the pile node. In the element coordinate system of the pile tip element, the normal direction behavior towards the element is treated as rigid behavior, resembling pile behavior. Conversely, the tangent direction behavior is characterized as nonlinear elastic behavior.

To define the behavior, the material and properties of a pile element can be inputted based on test data, such as Load Test results.

For more detailed information regarding the entry of parameters for pile elements, refer to [User Manual] Chapter 4 (General Material) or access the [Online Manual] by selecting F1.

3.2 Define Property

Set the material properties for the ground and define the type of structure along with the cross-section shape (cross-section rigidity) for the structure property.

Additionally, if pile elements are utilized, the bearing capacity and stiffness coefficient at the pile end can be further specified. By employing beam elements to simulate the pile structure's resistance to axial, shear, and bending forces, it becomes feasible to assess relative displacement and friction behavior with the adjacent ground by defining rigidity around the pile end. Although pile elements do not share nodes with the adjacent ground like interface elements, they can be seamlessly integrated into 3D modeling.

To verify the member forces of the structure at the solid boundary face, gauging shell elements can be incorporated. These elements are generated on the upper side of the base slab to analyze member forces such as axial force, shear force, and moment loaded onto the base slab.

The materials for each ground layer are outlined in the following table:

Ground Property

Name	Weathering soil	Weathered rock	Soft rock	Embankment	Abutment
Property	3D	3D	3D	3D	3D
Material	Weathering soil	Weathered rock	Soft rock	Embankment	Abutment

The property of each structure is as shown in the table below. If you specify the cross-section shape, the rigidity of the cross-section will be automatically calculated.

Structure Property

Name	Pile	Pile (Interface)	Pile tip	Gauging shell
Type	1D	1D	Other	2D
Model Type	Beam	-	Pile tip	Gauging shell
Material	Pile	Pile	-	-
Section	Pipe	-	-	-
Section Size	600x12	-	-	-
Thickness	-	1	-	-
Tip Bearing Capacity	-	-	4000	-
Tip Spring Stiffness	-	-	160000	-
Stiffness Scale Factor	-	-	-	1e-06

Section 4

Modeling

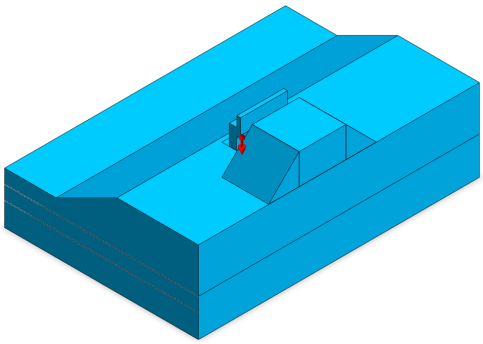
4.1 Modeling Geometry

[: Geometry > Protrude > Extrude]

This process involves creating lines, faces, and solids by extruding from lower-dimensional geometries such as points, edges, and faces. With lines that form a closed domain, it's possible to directly generate a solid. Follow these steps to generate solids for the 3D soil, base slab, and embankment, and create faces to classify each stratum:

① Select the face of the soil as the target object.
② Choose the Y axis for the extrude direction and enter a length of 60m.
③ Click the [Apply] button to execute the extrusion. Verify the generated solid on the screen.
④ Select the two faces of the abutment.
⑤ Choose the Y axis for the extrude direction and enter 10m as the extrude length.
⑥ Click the [Apply] button to apply the extrusion.
⑦ Select the three faces of the embankment.
⑧ Set the direction to the X axis. Ensure to check the [Reverse Direction] option.
⑨ Enter 10m as the length and press [Apply].

Ground and Structure Geometry



Tip

There are two methods for selecting the target object and direction. One is to select using the [Work Tree], and the other is to select directly on the screen.

⑩ Create two faces to divide the soil in the same process.
⑪ Change the [Selection Filter] to 'Edge' and select the two surface lines.
⑫ Choose the Y axis as the extrude direction. Enter a length of '70m' to ensure the generated soil is divided by the surfaces.
⑬ Uncheck the [Reverse Direction] option and click [OK].

Tip

It's recommended that the face used as a tool object for dividing a solid should be larger than the solid it divides. If the size of the face is smaller than the solid, even if the difference is minor, the solid will not be divided.

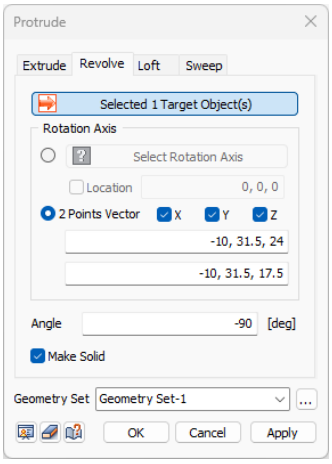
[: Geometry > Protrude > Revolve]

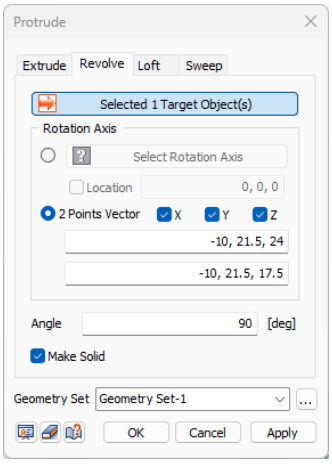
This is the process of creating lines, faces, and solids by rotating the upper shape of a point, line, or face to a specific angle.

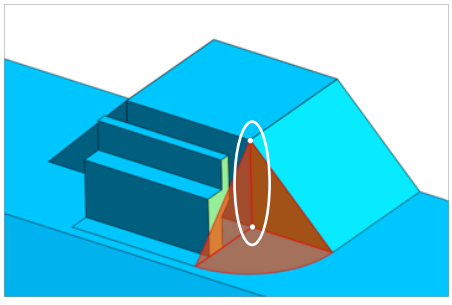
Rotate the face of the embankment to create a solid of the curved corner of the embankment.

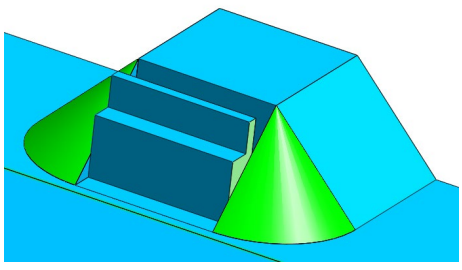
① Select the slope face of the embankment as shown in the image.
② Choose the rotation axis using [2 Points Vector] and select two points in order as shown in the image.
③ Enter 90 in the [Angle] field and preview the shape by clicking the preview icon.
④ Similarly, generate the opposite corner in the same process.

Create Embankment Solid









Tip

Select two points and define the rotation axis as the directional vector passing through those two points. The direction of the vector is determined by the selection order. A positive (+) value indicates a clockwise direction for the rotation angle.

[: Geometry > Protrude > Extrude]

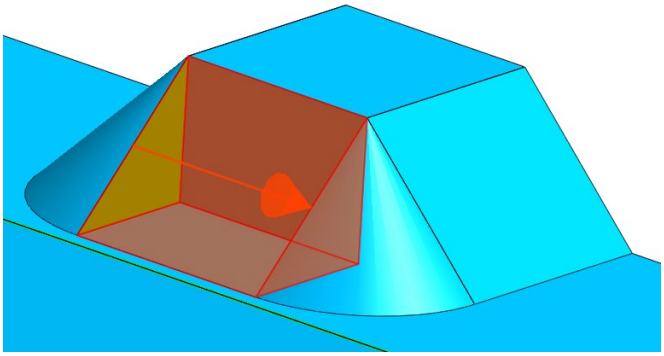
Complete the generation of the solid for the embankment and abutment connection.

① Choose the inner face of the corner solid as shown in the following image.

② Ensure to check the [Normal to Profile(s)] option for the direction.

③ Enter a length of 10m and click [OK]. Verify the generated solid.

Complete Embankment Solid



[: Geometry > Divide > Solid]

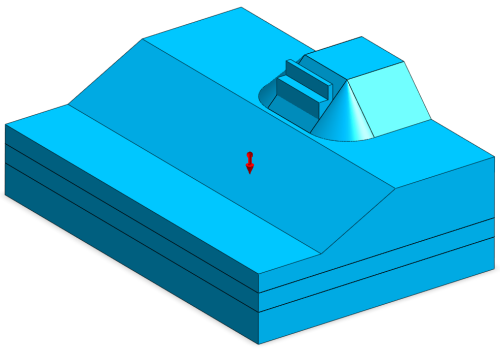
Separate the soil to classify the stratum. Divide the solid using the generated surfaces.

① Select the 'Soil' solid as the target object.

② Choose the two generated surfaces as the tool surfaces.

③ Click [OK] and verify the divided solids.

Complete Dividing Ground Solid



[: Geometry > Boolean > Solid]

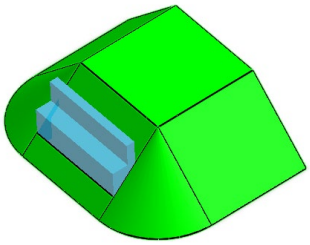
Utilize the Boolean functions to merge the solids and remove duplicated portions of the embankment solid. Combine all the embankment solids that were divided into several pieces for convenience.

① Navigate to the [Fuse] tab.

② Select the 6 embankment solids generated in the previous step as the target objects.

③ Ensure to check [Merge Faces] and then click [OK].

Combine Embankment Solids

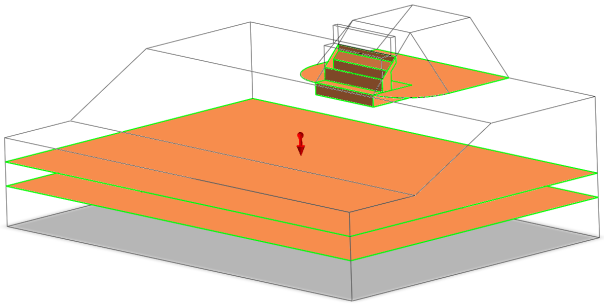


[: Geometry > Surface & Solid > Auto Connect]

This is the process of automatically generating shared faces and deleting duplicate parts between all the generated solids. When creating a mesh, it's necessary to generate shared faces so that nodes can be generated on every boundary of the domain.

① Select all the solids on the work plane, and then click [OK].

Check Duplicates (Auto Connect)



Tip

To prevent analysis errors caused by unconnected nodes between meshes, it's recommended to verify whether shared faces have been generated before creating the mesh. The [Auto Connect] function is used to create shared faces for all the solids at once. You can check if the shared faces have been correctly generated by using the tool available in the menu Geometry > Tools > Check Geometry > Check Duplicates. All surfaces appearing in yellow indicate faces where the shared face has been successfully created. If the yellow surface doesn't appear between two surfaces that should be connected, the connection is not correctly established and needs to be created using either "Auto-connect" or other Boolean operations.

In situations where the sharing face cannot be created due to complex geometrical shapes, "contacts" provide an option to remedy this situation by establishing connections between mesh sets.

4.2 Generate Mesh

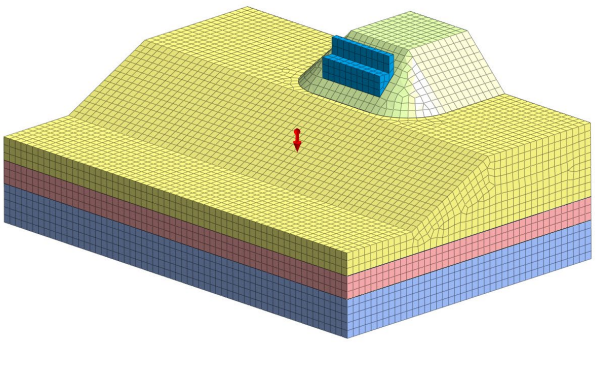
Indeed, mesh shape and mesh quality play crucial roles in finite element analysis (FEA). Typically, smaller mesh sizes result in better mesh shape (quality). However, reducing mesh size can also lead to longer analysis times. Therefore, it's advisable to determine the mesh size by considering both the accuracy and efficiency of the analysis.

[: Mesh > Generate > 3D]
This is the process of generating elements for 3D geometry. Select the 3D geometry of the ground/structure to generate the mesh. Properties for each solid can be assigned during the mesh generation. The property of each mesh can be assigned either during the mesh generation process or later using Mesh > Elements > [Parameters] after generating all the mesh sets for all the solids.

Generate a hybrid mesh for 'Weathering soil' and 'Embankment', including the concrete structure, using the [Auto-Solid] tab. Generate a mesh for 'Weathered rock' and 'Soft rock' in hexahedral shapes using the [Map-Solid] tab because the shapes of these two layers are regular and can be meshed using Map-meshing.

② Navigate to the [Auto-Solid] tab.
③ Select the 4 solids (Abutment, Base slab, Embankment, 1st layer of soil).
④ Enter a mesh size of 1m.
⑤ Choose the [Hybrid Mesher] from the dropdown menu.
⑥ Click the preview icon to check the mesh size before generation.
⑦ Click [Apply] and verify the generated mesh.
⑧ Switch to the [Map-Solid] tab.
⑨ Select the solids, 2nd and 3rd layer of soil.
⑩ Enter a mesh size of 1m and click [OK].

Generating Solid Mesh



[: Mesh > Generate > 1D]

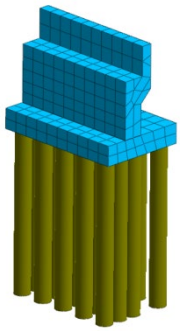
This is the process of generating Beam elements for the Pile elements.

① Select all the Pile lines (20) and enter '1m' in the [Division] field.

② Set the property for the pile type and enter 'Pile-beam elements' as the mesh set name.

③ Click the [OK] button.

Generating Pile-Beam Elements



Tip

The size of the beam elements and division doesn't have significant impact before generating the pile elements. This is because the nodes of the beam elements and the ground are automatically connected when the 'Pile elements' are generated. However, even though the beam element nodes are automatically connected to the nodes of the ground, it's still possible to define separate Pile interface elements to consider interactions with the ground in more detail.

[: Mesh > Element > Pile/Pile Tip]

This is the process of adding Pile and Pile Tip elements. Pile Tip elements are created from the Pile-Beam elements generated in the previous step.

① Navigate to the [Pile] tab.

② Select all the 20 generated Pile-beam elements.

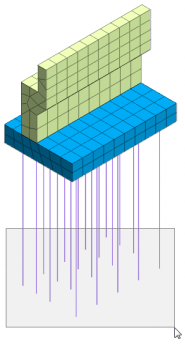
③ Set the property to 'Pile(Interface)' and click [Apply].

④ Switch to the [Pile Tip] tab.

⑤ Select the 20 pile tip nodes as shown in the following image.

⑥ Choose the property as 'Pile tip' and click the [Apply] button.

Generating Pile Tip Elements



[: Mesh > Element > Create]

To check the structural member forces of the base slab, add a gauging shell. The gauging shell is generated by selecting the solid elements (to check the member forces) and the element boundary (to generate the gauging shell).

① Navigate to the [Other] tab.

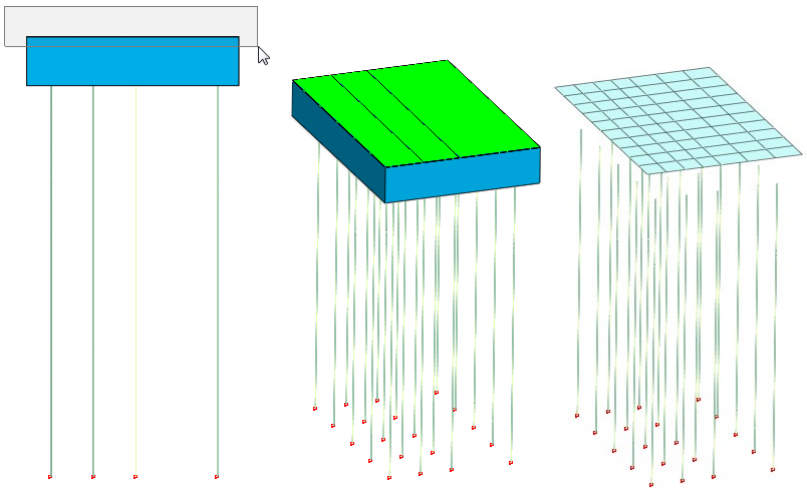
② Select 'Gauging shell' from the dropdown menu.

③ Choose the 'Base slab' solid in the part selection. If the geometry is hidden, ensure to check the geometry checkbox to display it on the screen in the work tree.

④ Select the faces where the gauging shell elements will be generated. Choose the 3 faces on the top of the base slab as shown in the following image.

⑤ Set the property to [Gauging shell] and click [OK].

Generating Pile / Pile Tip / Gauging Shell



Tip

The ground elements should be generated before generating pile elements. Pile elements, which simulate connected surface behavior, cannot be created from a beam element without a ground element because connections with nodes of the ground need to be considered during the creation of the pile elements.

[: Mesh > Element > Parameters]

① In this step, we verify if the correct properties are assigned to each mesh set. If the same property was used for all the mesh sets during the meshing phase, different properties corresponding to each mesh set have to be assigned during this step using the [parameter] option.

② Go to the [3D] tab.

③ Assign the properties of each stratum according to the model cross-section (please refer to the picture below). As the property of the base slab changes during the analysis, its property has to be set to 'Weathering Soil' in this step and will be changed to 'Concrete' in the next step using the function Static/Slope Analysis > Boundary > Change property.

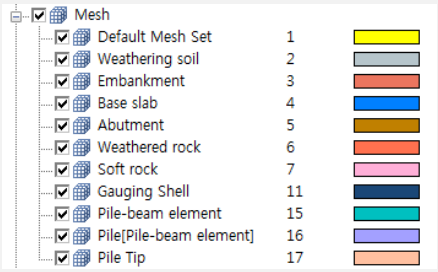
④ Assign the suitable properties to the selected mesh set.

⑤ Apply the changes by clicking the [Apply] button.

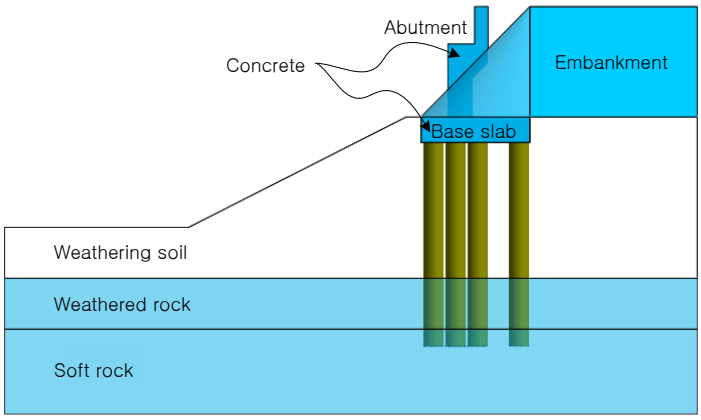
⑥ If you select a mesh set in the work tree, you can view its material/property in the property window.

Tip

Grouped mesh sets corresponding to each solid are automatically generated during the meshing process, and they can be selected in the work tree. The property of each mesh set can be changed according to the associated material. It's recommended to rename the mesh sets using the [F2] key on the keyboard to facilitate defining the construction stages of the analysis more easily.



Overview (Cross-Section)



Section 5

Analysis

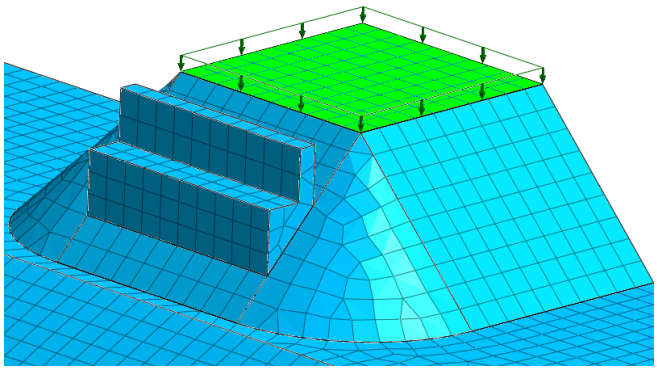
5.1 Setting Load Condition

[: Static/Slope Analysis > Load > Pressure]

In this step, let's apply a surface load at the top of the embankment. You can define a uniform or linear/nonlinear load.

① Navigate to the [Face] tab.
② Choose 'Face' as the [Type]. (Ensure to check the Geometry checkbox of the solid in the work tree to display it on the work window)
③ Select the face at the top of the embankment.
④ Set the load direction type as the normal direction.
⑤ Check the [Uniformly Distributed Load] option and enter 100(kN/m^2) in the [P or P1] field.
⑥ Click the preview icon to verify the loading.
⑦ Enter 'Surface load' in the [Load set] field, and click [OK].

Surface Load



Tip

For applying loads or boundary conditions, geometric parts can be directly selected (Edge, Face) as well as element boundaries (Edge, Face) in the [Select object] option. Since all the loads and boundary conditions are automatically converted to nodal loads for analysis, the type of the target object does not affect the analysis results. Selecting geometry instead of nodes/elements makes the selection process more convenient. Additionally, it's useful in cases of mesh deletion and re-meshing because loads bound to geometry will remain associated with the model, whereas loads associated with nodes/elements will be deleted along with the mesh sets.

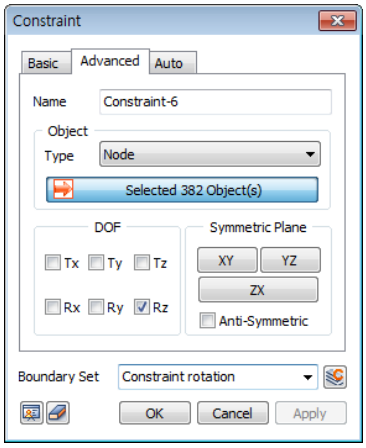
5.2 Setting Boundary Condition

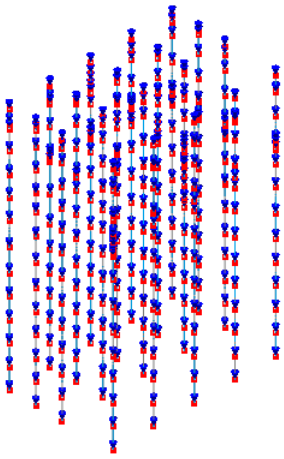
* : Static/Slope Analysis > Boundary > Constraint

In this step, we will set the boundary conditions to constrain displacements and rotations of the model in the global coordinate system (GCS). First, boundary conditions of the ground will be automatically set using the 'Auto' tab, then the rotation Rz of the piles will be fixed to prevent additional degrees of freedom in the model that could cause errors during the solving phase of the problem.

① Go to the [Auto] tab.
② Check [Consider All Mesh Sets], and enter 'Ground boundary condition' in [Boundary Set].
③ Click [Apply].
④ Display all the meshes in the work tree and verify the generated boundary condition.
⑤ Switch to the [Advanced] tab.
⑥ Set the [Type] to 'Node'. Select the nodes of the Pile-beam elements, and check [Rz].
⑦ Name the [Boundary Set] as 'Constraint rotation', and click [OK].

Pile Constraint





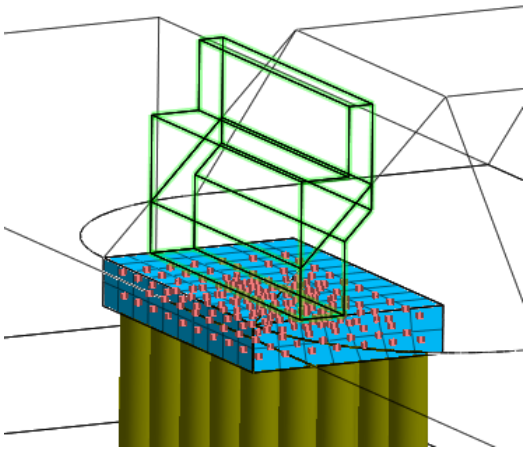
[: Static/Slope Analysis > Boundary > Change Property]

In this process, we will learn how to change the property and material assigned to a mesh set during the construction stage analysis. Even though only one property can be assigned to one mesh set at a time, the [Change property] function can be used to alter the property assigned to a mesh set during one of the stages of the construction analysis when this boundary condition becomes activated.

Change the property of the 'Base slab' from 'Weathering soil' to 'Concrete'.

① Go to the [General] tab.
② Select the elements of the 'Base slab'.
③ Change the property to 'Concrete'.
④ Set the boundary condition name as 'Structure' and click [Ok].

Change Property



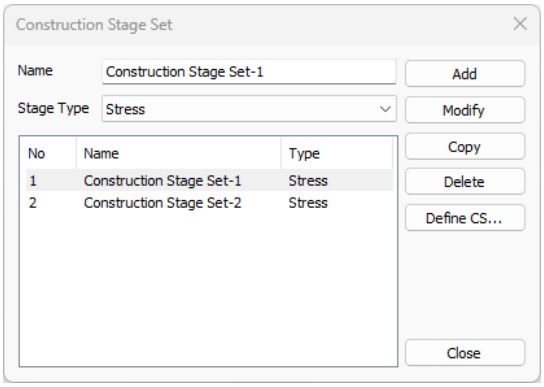
5.3 Defining Construction Stage

[: Static/Slope Analysis > Construction Stage > Stage Set]

In this step, we will learn how to set up the construction stage analysis for each stage. Two construction stage sets will be generated to consider whether the embankment is supported by piles or not. The final loading of the embankment will be divided into several increments. Construction stages are defined according to the mesh set names that you previously assigned for the mesh sets, so it's recommended to adjust the mesh set names accordingly to ensure accurate construction stage definition.

① Set the analysis type as [Stress].
② Click [Add] twice to create the two construction stages.
③ Select [Define Construction Stage] to define each stage.
④ The construction stage for each set is defined as follows:

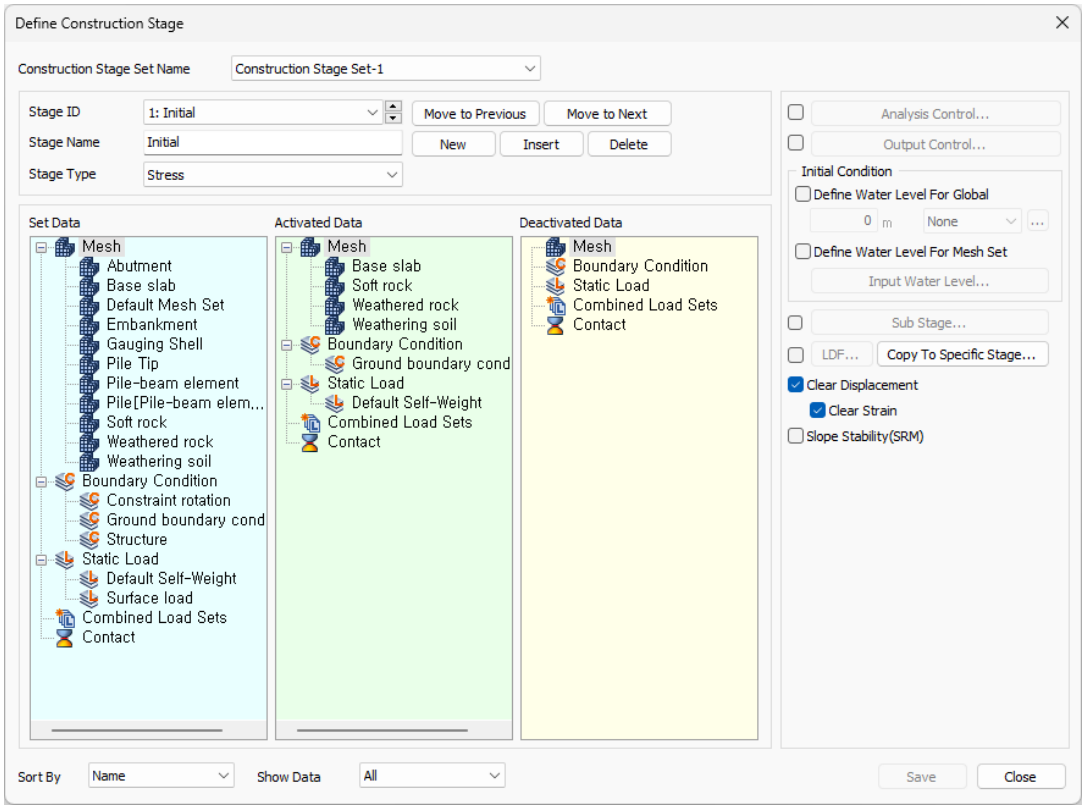
Construction Stage Set



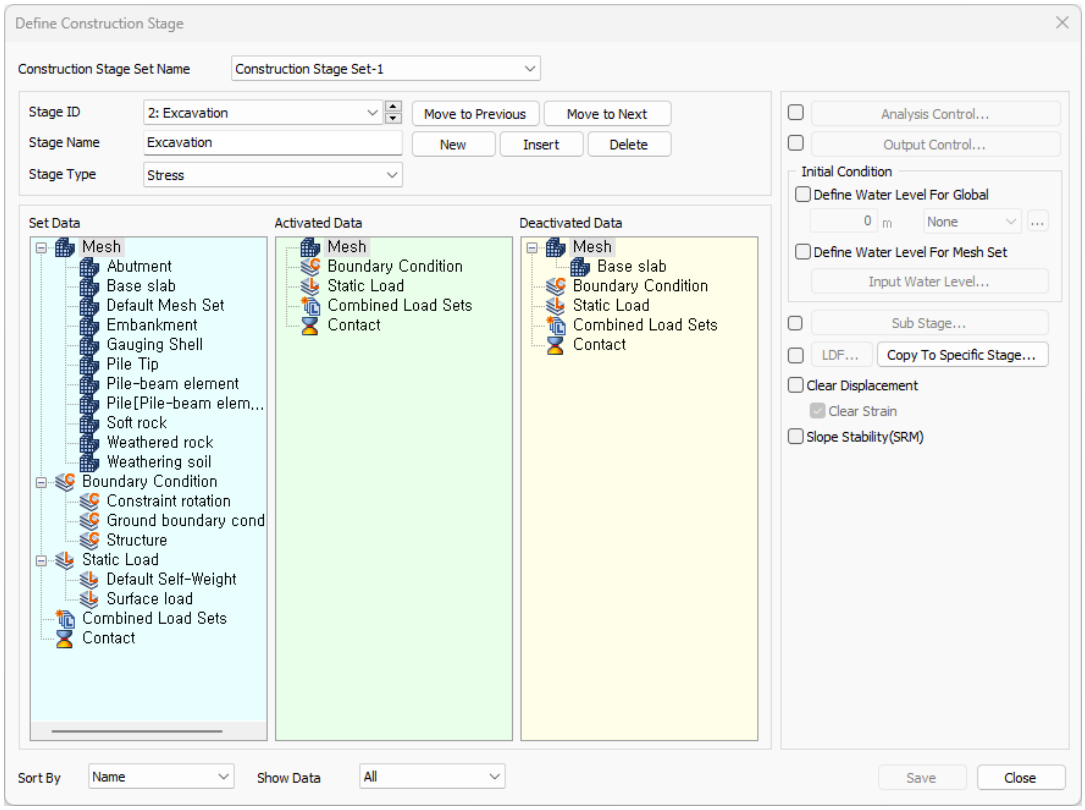
Tip

When defining multiple construction stage sets with only a few specific conditions changed, it's convenient to define one construction stage and then create another by using the [Copy] button. This allows you to duplicate the construction stage and then modify only the specific conditions that need to be changed for the copied construction stage.

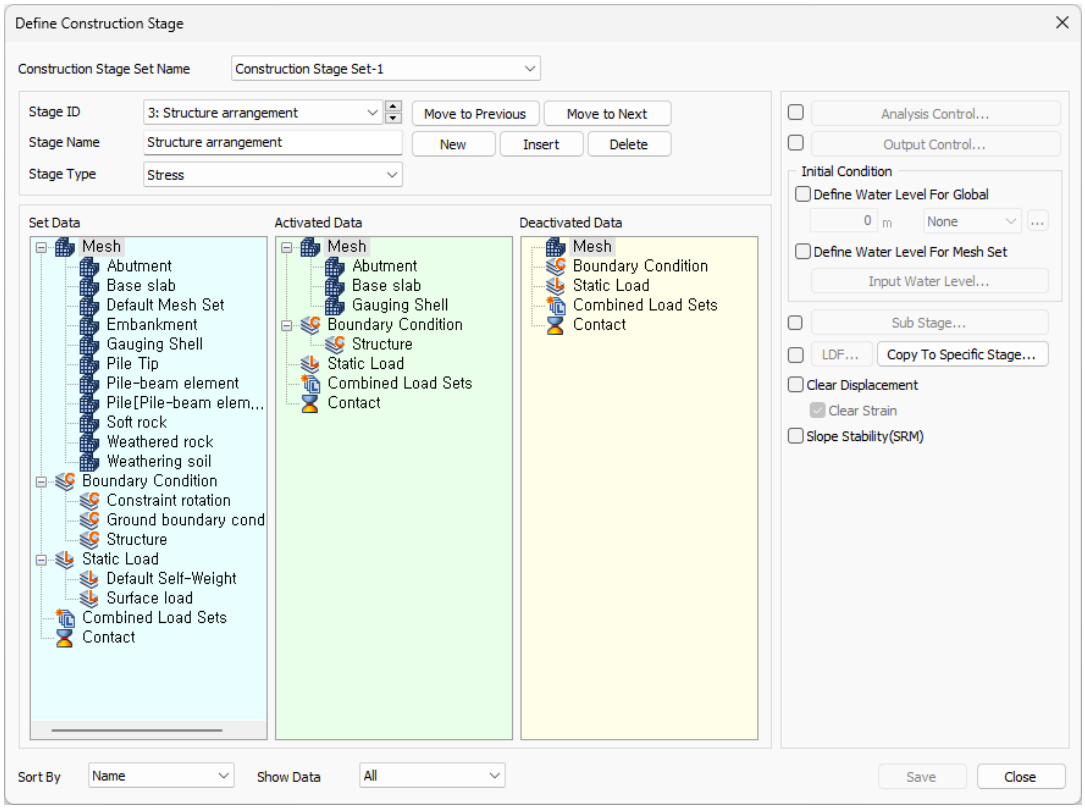
Stage 1.
Initial



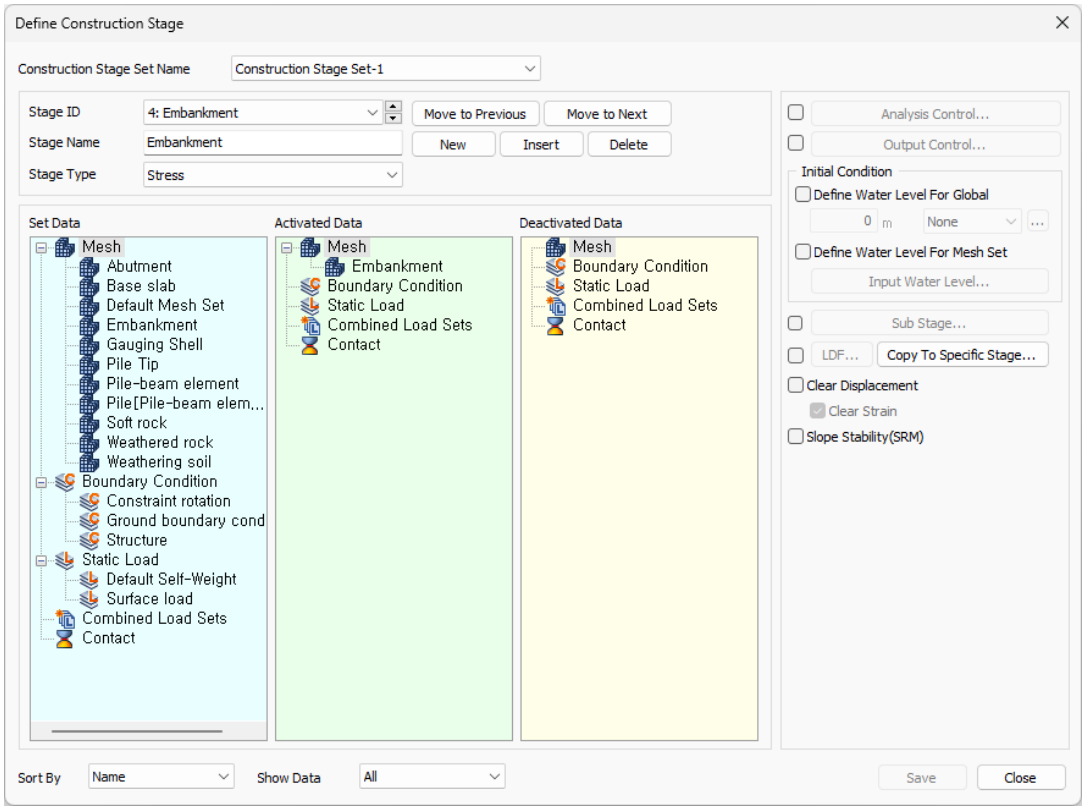
Stage 2.
Excavation



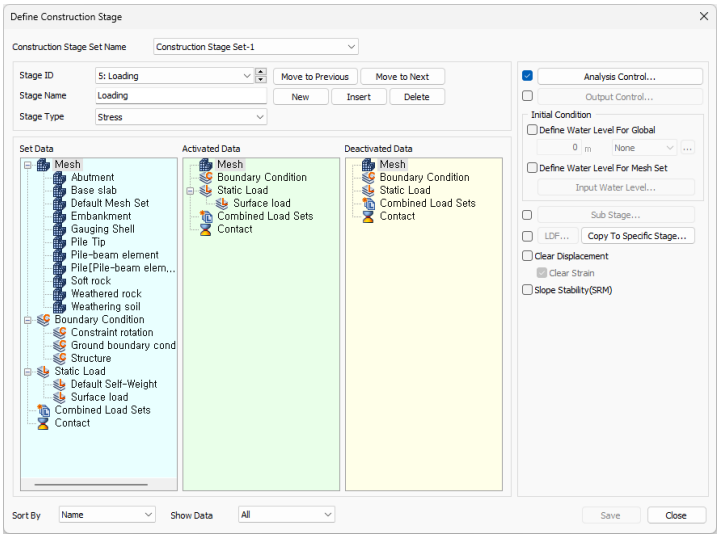
Stage 3.
Structure Arrangement

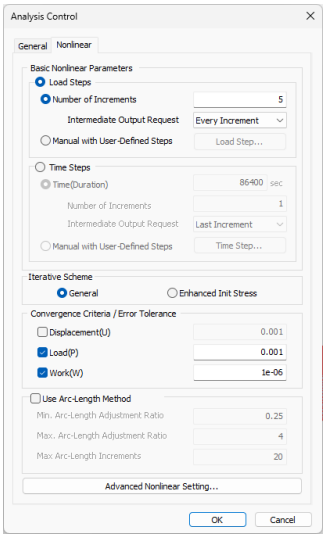


Stage 4.
Embankment



Stage 5. Loading





For Construction Stage Set-2 (After applying Pile Foundation),

Stages 1, 2, 4, 5 are identical to the 'Construction Stage set-1'. The only changes will appear in stage 3: 'Structure Construction' where pile elements will be activated additionally.

Stage 3 - Name: Structure Arrangement

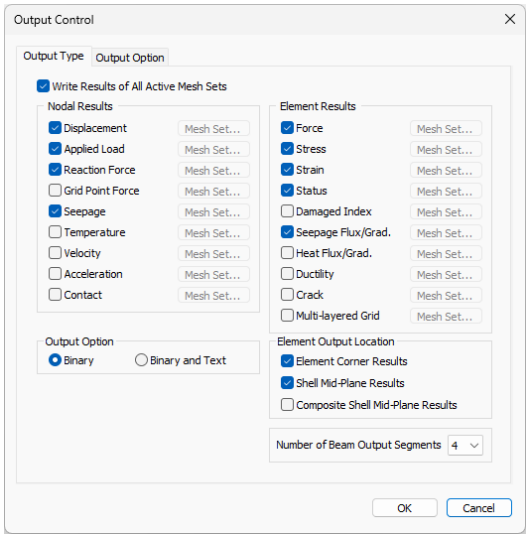
- Activate Data-Mesh: [Abutment], [Base slab], [Gauging shell], [Pile], [Pile Tip], [Pile-beam elements]

5.4 Setting Analysis Case

In this step, we will set the type of analysis, the data to be considered during the analysis, along with some output options. In the Analysis Case window, the 'Analysis control' options will govern the solving process, along with advanced options used to control the nonlinear solver. The 'Output Control' options will determine the type of results that will be calculated by the solver and displayed in the results. These options can be defined separately for each Subcase as well during the construction stage definition by activating the Subcase control option checkbox in the stages.

In [Output Control], to plot the relative displacement of elements such as pile elements among the ground when interfacial behavior occurs, you need to check Element Results > Strain before analyzing.

Output Control



[: Analysis > Analysis Case > General]

① Enter the name of the Analysis case and select 'Construction Stage' as the solution type.

② Set Analysis > General > Initial Stage > Initial Stage for Stress Analysis to '1:Foundation'. (Since the foundation line is not horizontal, do not check the k0 condition.)

③ Click [OK].

④ Generate the analysis case for each of the 2 construction stage sets.

5.5 Perform Analysis

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

[: Analysis > Analysis > Perform]

① Perform the analysis.

During the analysis, you can monitor the calculation process in the [Output Window]. Check for convergence and any warnings or errors that may occur during the analysis. They will be displayed in the Output window as well.

Section 6

Results

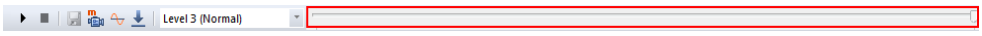
You can examine the displacement of ground and structural elements, the member forces of the base slab, and the behavior of the piles based on construction stage steps in the work tree. All the results are displayed as contours, tables, and graphs. In this tutorial, the main result items that need to be checked are:

- The horizontal displacement of the abutment for each step of the construction stage analysis (Check lateral ground movement).
- The difference in displacement of the abutment with and without piles.
- The member forces of the base slab and pile foundation at each step (axial force, shear force, moment).
- The surrounding friction and relative displacement of the pile foundation.

6.1 Verify Displacement

Verify the 'Displacement' in the work tree after the analysis. TX, TY, TZ represent the displacements based on the X, Y, Z axes respectively. Each of the horizontal displacement and settlement tendency occurring due to banking and surface loading can be verified by TX, TY, TZ. '(V)' indicates 'vector' and 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 for displacements and principal stresses.

- By moving the sliding bar at the bottom of the work window, you can simulate the changes in results for each construction stage and each load step.

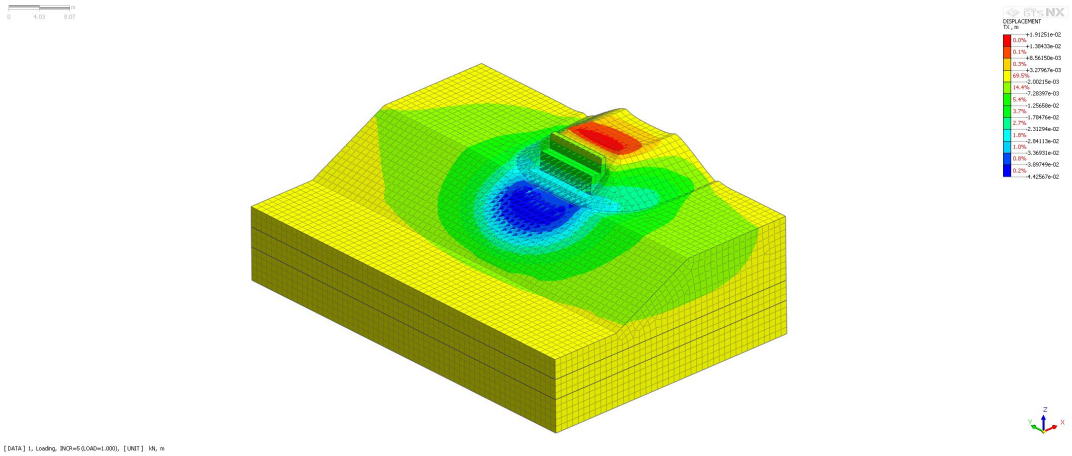


Check the results at the last loading step.

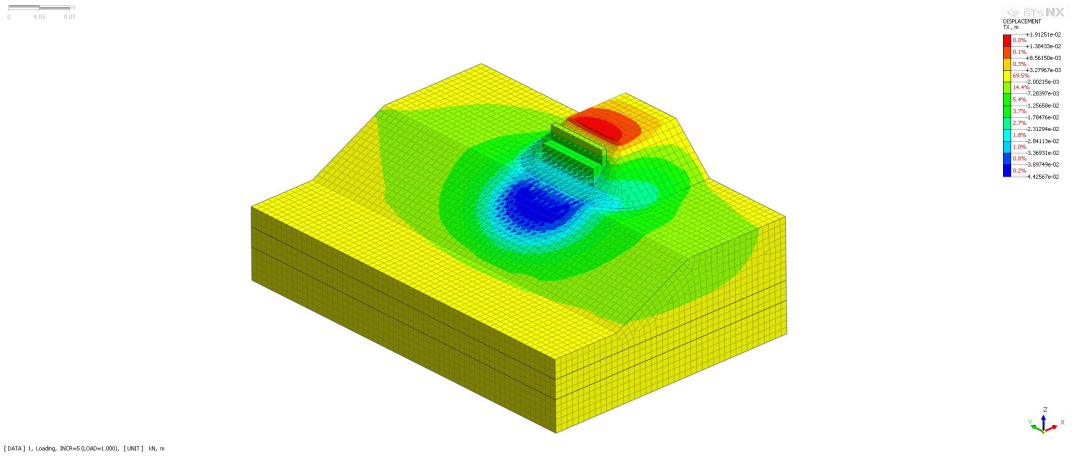
Select the last construction stage/last load step of the result tree, then choose Displacement > TX TRANSLATION(V).

- You can opt to view either the deformed or undeformed model according to the X direction in Result > General > Deform. (The degree of deformation of the model can be adjusted by the scale factor through the property window. It can be displayed in the work window by enabling Results > Show/Hide > [Actual Deformation])
- Use Results > Advanced > Probe to examine the result value of the selected node/element. Additionally, you can locate and identify the value of the Max/Min/Abs Max of the result.

Horizontal Displacement (Deformed)



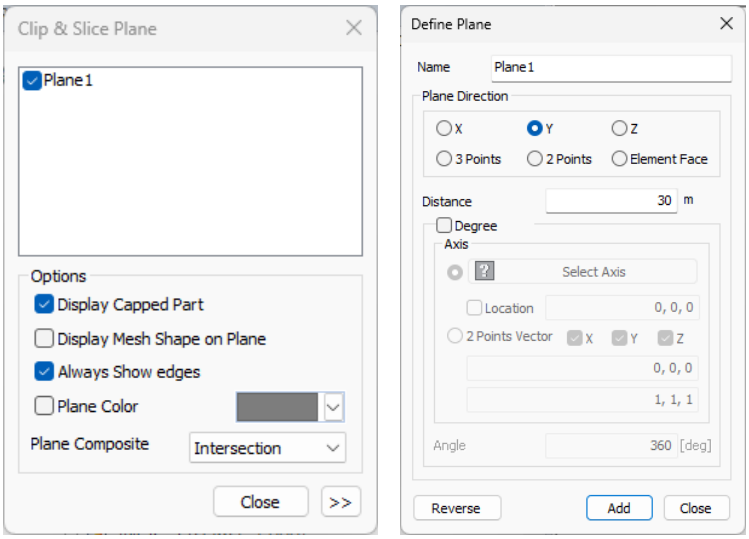
Horizontal Displacement (Undeformed)



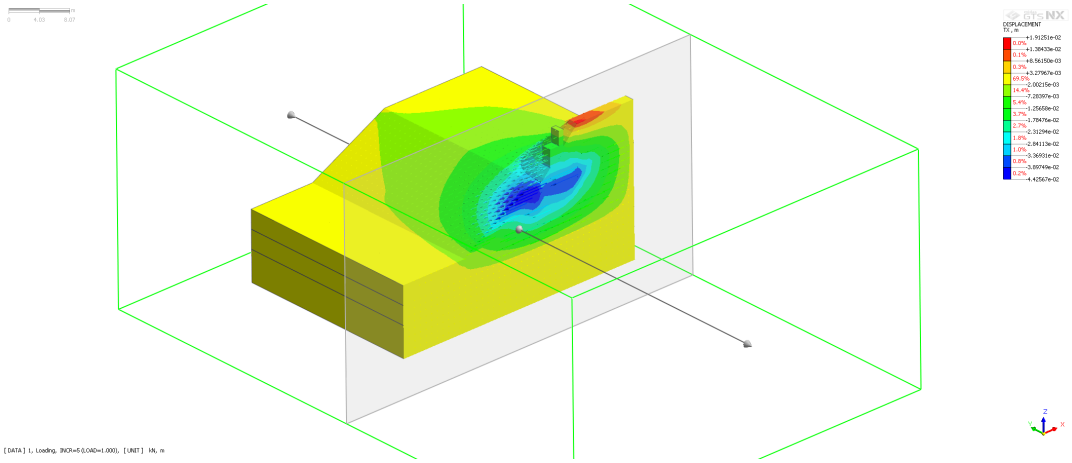
It is possible to divide the 3D model by a specific surface and view the value of the results on this face. GTS NX offers the 3D-2D Wizard function, which is used for checking specific point results inside the model by dividing the model with a defined plane.

- First, select Clipping Plane to define a plane for checking results in the Advanced view Control bar. The [Clipping Plane( )] function can be applied by the axial direction based on GCS or by setting a specific plane. You can apply several planes at the same time. For the [Plane Composite] Method (Intersection/Union), reverse direction can also be verified.
- Select the [Plane Composite] to 'Intersection'.

Clipping Plane

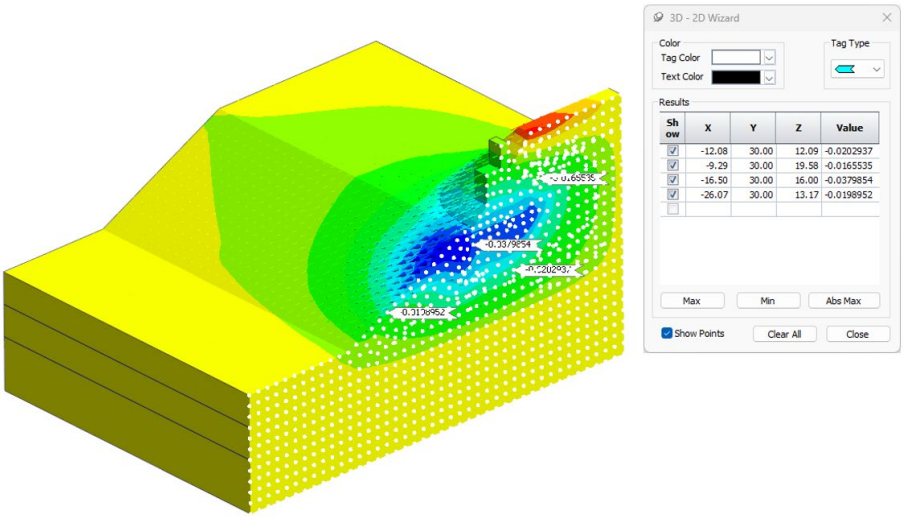


Sliced Plane-1



- Select Result > Advanced > Others > 3D->2D Wizard. The 3D->2D Wizard is a function used for tagging the result value of a point at the cutting plane of the 3D model. If you check [Show Points], the points of the cutting plane will be displayed in the work window.

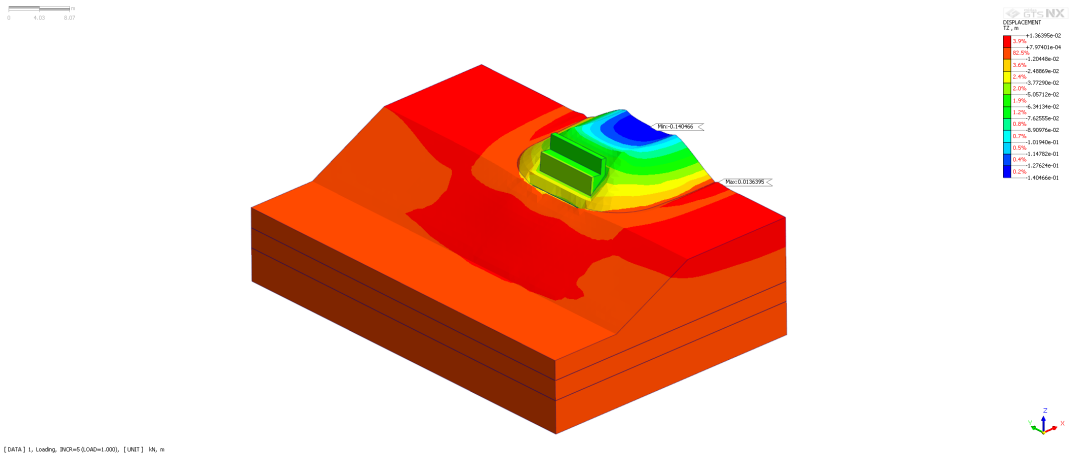
3D-2D Wizard



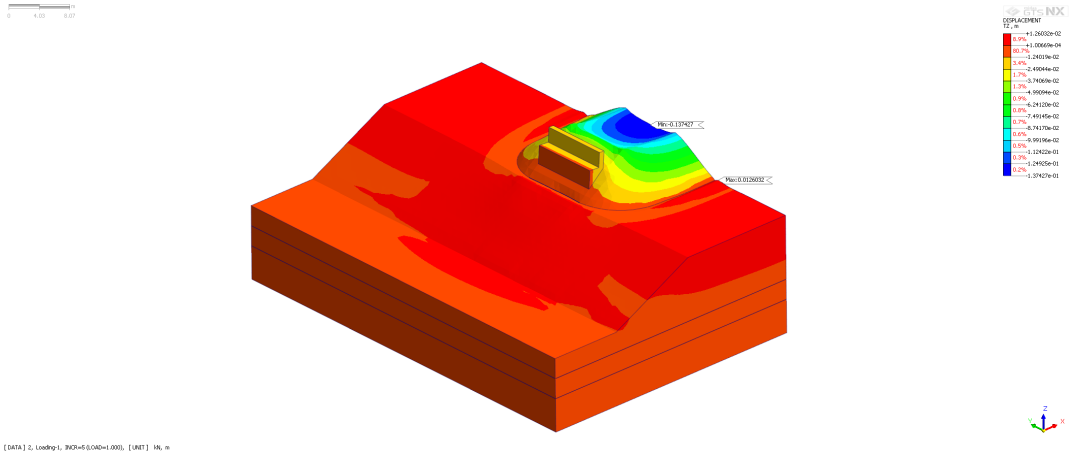
Let's check and compare the abutment settlement with and without piles.

- Select the last construction stage/load step in the results tree, then choose Displacement > TZ TRANSLATION (V).

Settlement (without Group Pile)

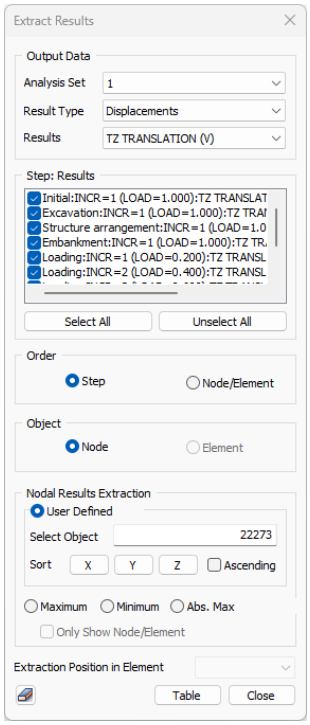


Settlement (with Group Pile)



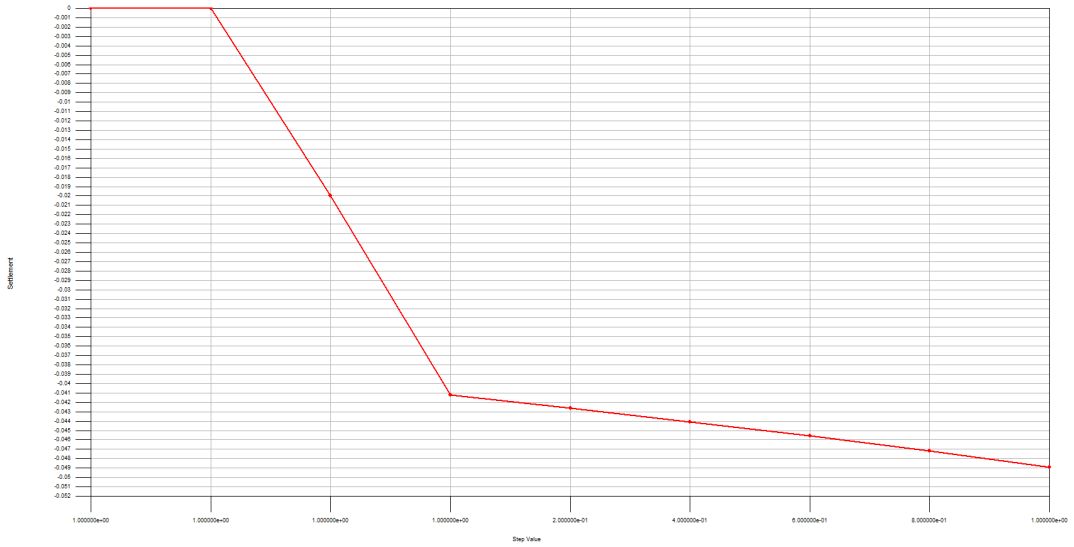
- Extract the results of each step at the end of the abutment and compare its settlement after and before the application of pile foundation.
- Select Results > Displacement > TZ. Choose the node (22273), or type '22273' and select the [Table] button. In the plotted table, the graph can be plotted by right-clicking the mouse. Similarly, select the other analysis case and compare the settlement graph at the same node. The maximum settlement before applying pile elements is about 60mm, and the settlement after applying pile elements decreased to 17mm.

Extract Results and Table



No	Step	Step Value	Node: 22273 TZ TRANSLATION (V) (m)
1	Initial:INCR=1 (LOAD=1.000)	1.000000e+00	-
2	Excavation:INCR=1 (LOAD=1.000)	1.000000e+00	-
3	Structure arrangement:INCR=1 (LO	1.000000e+00	-1.993627e-02
4	Embankment:INCR=1 (LOAD=1.000)	1.000000e+00	-4.123580e-02
5	Loading:INCR=1 (LOAD=0.200)	2.000000e-01	-4.263076e-02
6	Loading:INCR=2 (LOAD=0.400)	4.000000e-01	-4.406971e-02
7	Loading:INCR=3 (LOAD=0.600)	6.000000e-01	-4.557993e-02
8	Loading:INCR=4 (LOAD=0.800)	8.000000e-01	-4.718588e-02
9	Loading:INCR=5 (LOAD=1.000)	1.000000e+00	-4.893493e-02

Show the Graph



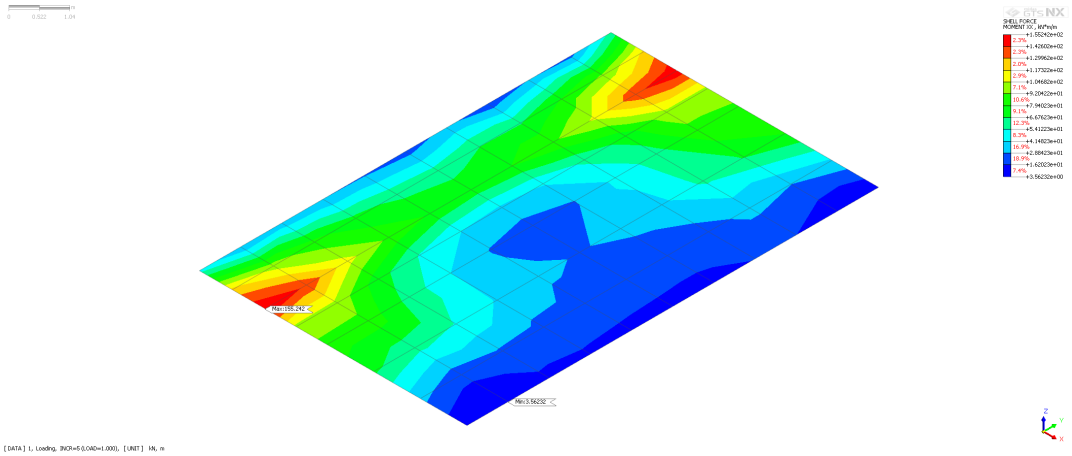
6.2 Verifying Member Force & Stresses

You can verify the member force of the solid element through the gauging shell generated on the top of the abutment. If the model consists of a solid element concrete structure, design member force can be checked using the gauging shell.
Verify the gauging shell and the member force of the pile foundation. Also, compare the member force of the base slab based on whether the pile foundation is applied or not.
Check the member force of the base slab in 'Shell Element Forces/Stresses', and check the pile foundation in 'Beam Element Forces/Stresses'. The result of each structure member will be plotted based on the element coordinate system set in the default settings. If you need to consider another coordinate system, it is possible to change it when defining the material coordinate system or when generating the analysis case in the analysis control. Then, after analysis, you need to switch the coordinate system used for result output in the property window > general > Output CSys from default to user-defined or material CSys.

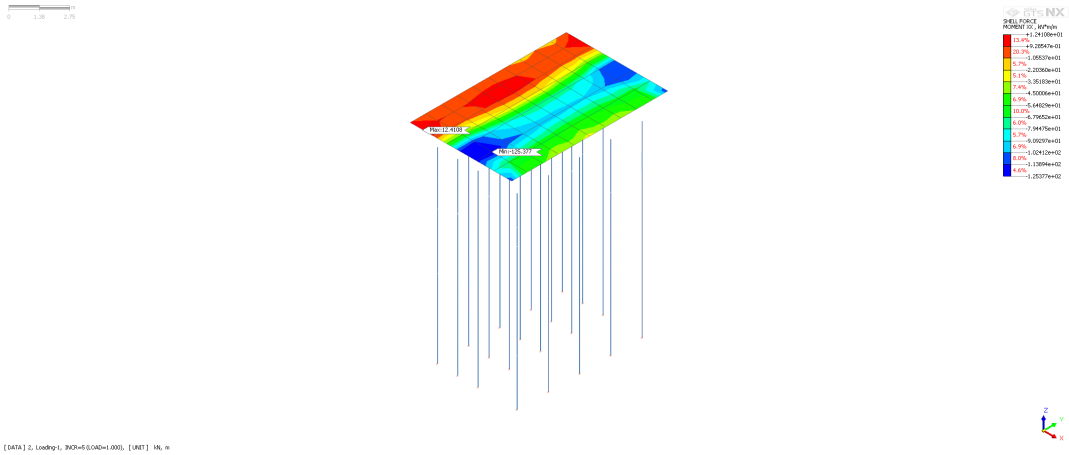
Verify the results after the final loading.

- In the result tree, choose Shell Element Forces > BENDING MOMENT XX in the last load step. Check the maximum moment and distribution of the Base slab.
- To see only the structure member force, select Result > General > No Result > Exclude.

Slab Bending Moment (without Group Pile)

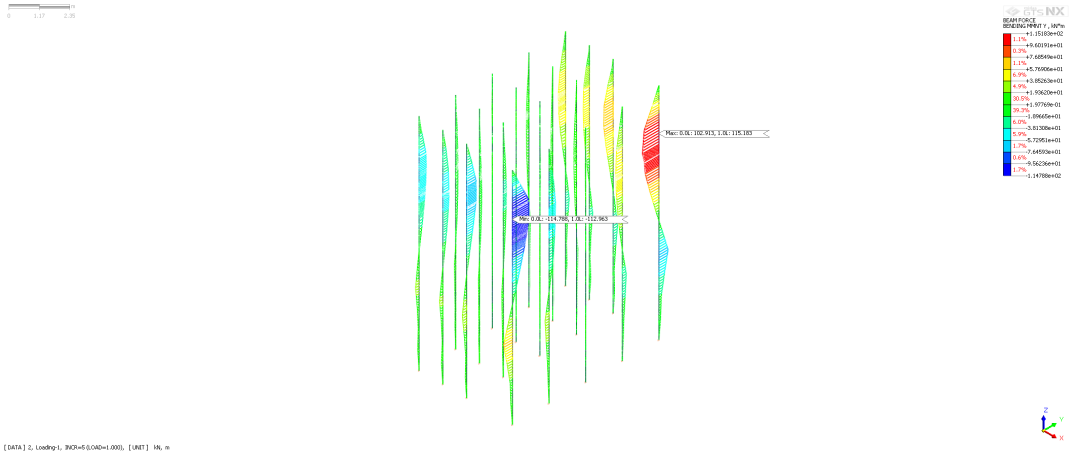


Slab Bending Moment (with Group Pile)



- In the Result tree, choose Beam Element Forces > BENDING MOMENT Z of the last load step after applying pile foundation. Check the maximum moment of the pile foundation in the result tree for the last load step after applying the pile foundation. In the model tree, it is possible to select a member by checking show/hide to plot the result that you want to see.

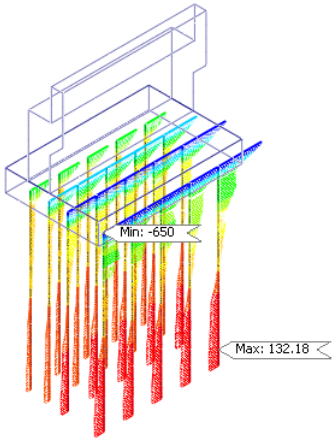
▶Pile moment



6.3 Verify Frictional Force and Displacement of Pile/Ground

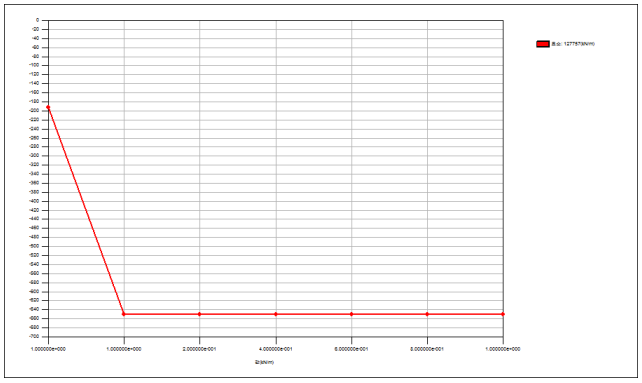
You can verify the displacement and friction between the ground and the structure in about two normal directions and the tangent direction through Pile element results. By displaying the displacement-friction diagram of each stage, you can check whether the ultimate bearing power of piles is reached or not.

- Verify the tangential friction force between the pile and the ground by checking Pile Force > TANGENTIAL X of the last load step in the result tree. By plotting this in relation to relative displacement in a diagram, you can observe the applied 'T-Z CURVE'. The results indicate that the ultimate shear force is generated right after the banking. Plastic behavior occurs when loads exceed the ultimate shear force.
- Select Pile Relative Displacements > TANGENTIAL X to observe the relative displacement of the pile and the ground. It appears to have a similar trend to the friction. However, even after reaching the ultimate bearing power, the displacement continues to increase.



▶Pile tangential friction force

▶▶change of pile head friction force



▶Tangent direction displacement of each step