Chapter 1
3D Abutment Foundation Pile

Workflow

3D Abutment Foundation Pile

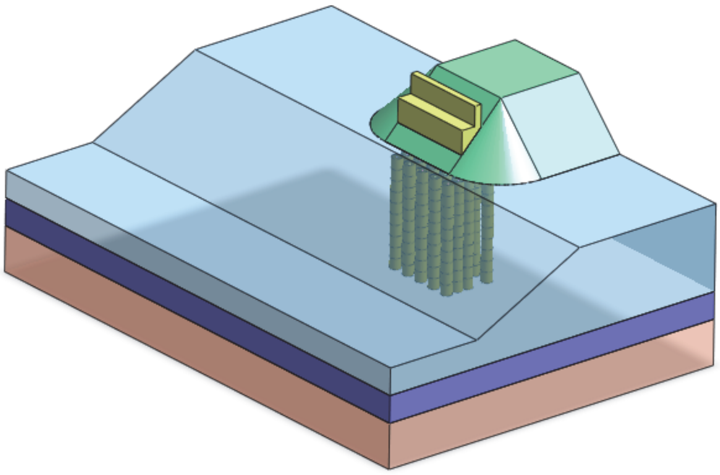
Section 1
Overview

1.1 Learning Purpose

This tutorial will talk about the evaluation of the horizontal displacement of bridge abutment foundation and pile stability. The evaluation of the soil-structure behavior is done by using pile elements, not by simple beam elements. Pile elements are like embedded beam elements which don't require node sharing, so it can be easily used in 3D modeling. Also pile elements are perfect for evaluating the soil-structure behavior because it can consider the influence of interface between pile and adjacent ground.

This Tutorial presents 3D Model of the bridge abutment structure. Stage by stage, this tutorial checks the displacements, the foundation and the member force of pile elements according to the abutment and load. The results will be analyzed in two distinct cases, first, only the influence of the abutment will be considered and secondly, the piles will be added in order to decrease even more the displacement of the ground below the abutment. Two analysis case of construction stage can be generated in a model. In addition, gauging shell will be applied to the upper abutment part in order to evaluate the structural member forces which affect foundation plate generated from loads.

▶Overview analysis model



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

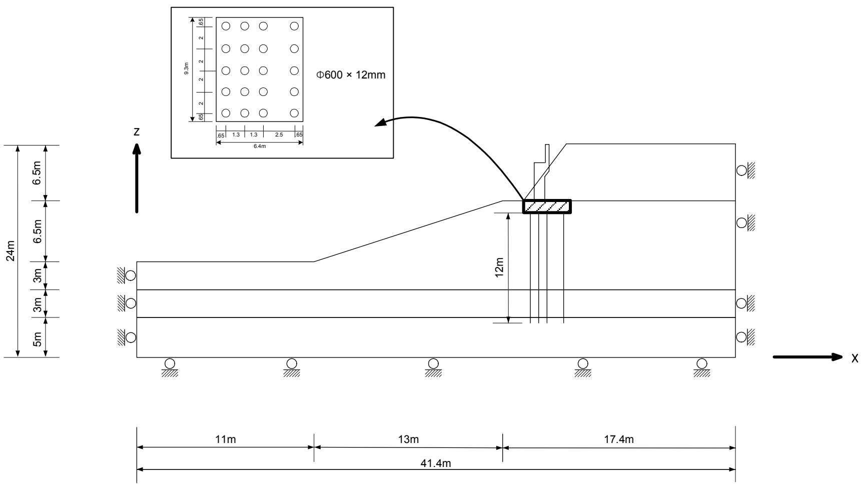
- 3D geological stratum modeling (Mohr-Coulomb Law)
- Modeling of pile elements (Checking the friction at the surface of piles caused by the load application)
- Setting the load steps
- Evaluate horizontal deformation of the abutment of each construction steps
- Draw the result graph of Load-Settlement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

1.2 Modeling and Analysis summary

In this tutorial, the model is composed of three layers of stratum and an abutment foundation pile adjacent to a slope site. And then perform the construction analysis according to the abutment construction and loading. The abutment is constructed on foundation slab of 6.4x10m^2 and a load of 100kN/m^2 is applied on the embankment. This load will be divided and applied in 5 steps of construction. For each step, horizontal displacement and influence on settlement will be investigated. At the same time, 20 steel pipe piles of 600x12mm will be added and their influence on the settlement will be studied. By dividing the analysis into 2 construction stage representing the analysis before and after application of the piles, it will become possible to assess the influence of the piles.

The model of the geological stratum is shown below:

▶Cross-section of the analysis model



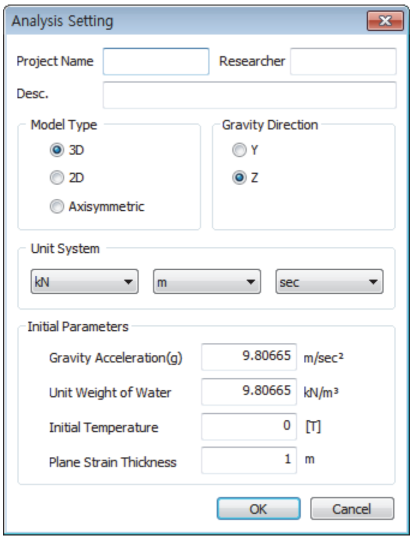
Section 2
Analysis Setting

[Open the attached start file (01_Pile element_start)]

* : Analysis > Analysis Case > General

Set model type, gravity direction and initial parameters, and check the unit system which will apply to the analysis. The unit system can be changed during the modeling process and when checking analysis results. The input parameters will be automatically converted in the chosen unit system.

This tutorial is a 3D model with Z gravity direction and using SI unit system (kN, m).



Section 3
Define Material and Property

Define material of ground and structure. Generate mesh and assign property to each mesh.

3.1 Define Material of Ground and Structure

For ground material, 'Mohr-Coulomb' model, which is generally used for analyzing elastic behavior, has been used. For structure material, both 'Elastic 'model which doesn't consider material nonlinearity and 'Pile Element' for checking skin friction force are used.

Materials for ground/structure are as following table.

▶Table. Ground material

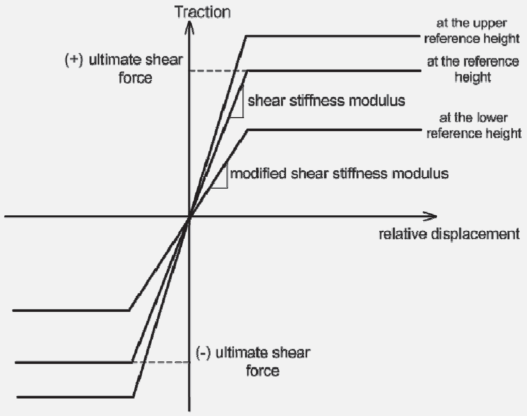
Name	Weathering soil	Weathered rock	Soft rock	Embankment	Abutment
Material	Isotropic	Isotropic	Isotropic	Isotropic	Isotropic
Model Type	Mohr Coulomb	Mohr Coulomb	Mohr Coulomb	Mohr Coulomb	Elastic
General					
Elastic Modulus (E)	1.2E+04	1.2E+05	1.2E+06	3.0E+04	2.1E+07
Poisson's Ratio (v)	0.33	0.3	0.25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

▶Table. Structure material

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

Tip

The behavior of pile elements can be divided into a normal behavior and a tangential behavior. First, the normal behavior between the pile and the surrounding ground is considered as fixed and rigid, whereas the tangential behavior is a nonlinear elastic behavior. The nonlinear elastic behavior is divided into the yield force and the yield function assigned.
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, the relation between relative displacement and friction can be defined more precisely.



The Pile tip element works as solid-point interface that presents the relative behavior between the ground elements and pile node. In the element coordinate system of the pile tip element regard the normal direction behavior toward the element as rigid behavior just like a pile behavior. And, regard the tangent direction behavior as nonlinear elastic behavior.

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

For more information about entering parameters of pile element, look up [User Manual] Ch4 (General Material) or Select F1 for [Online Manual].

3.2 Define Property

Set the material for the ground property. Set type of structure and cross-section shape (cross-section rigidity) for a structure property.

If you use pile element, the bearing power and the coefficient of stiffness at the end of the pile can be set additionally.
By using beam element which resists for axial/shear/bending for the pile structure, it is possible to check the relative displacement and friction behavior with adjacent ground by defining rigidity around the end of the pile. Pile element doesn't share nodes with the adjacent ground like interface element, but it can be easily used in 3D modeling.

IN order to verify the member force of the structure at the solid boundary face, gauging shell element can be added additionally. The gauging shell element is generated on the upper side of the base slab to check the member force such as axial force/shear force/ moment loaded to the base slab.

Materials for each ground layer are as in the following table:

▶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

Property of each structure is as following table below. If you set the cross-section shape, the rigidity of the cross section will be automatically calculated.

▶Table. 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

[Start modeling]

This tutorial is focused on modeling process of creating 3D geometry and mesh. Also it is focused on analyzing settlement tendency based on foundation type. Start the tutorial by opening the start file in which material property of ground/structure and basic geometry is inputted.

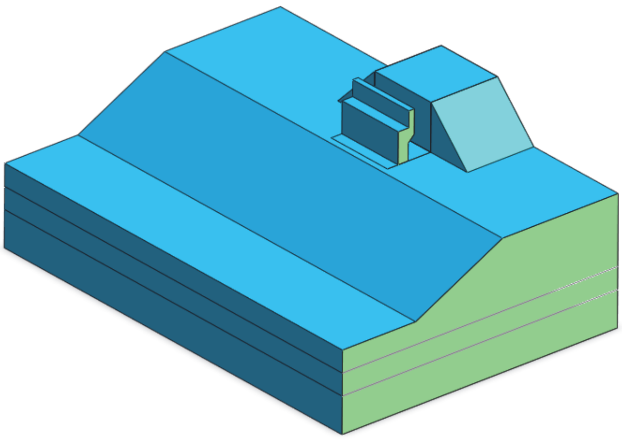
4.1 Modeling Geometry

*  : Geometry > Protrude > Extrude

This process makes line/face/solid by extruding from geometries of lower dimensions: point/edge/face. With lines which constitute a closed domain, it is possible to directly make a solid.
Generate solid for 3D Soil, Base slab and Embankment. Create faces to classify each stratum.

- Select the face of [Soil] as the target object.
- Choose Y axis for the extrude direction, and enter extrude length as 60m.
- Select [Apply] button. Check the generated solid on the screen.
- Select the 2 faces of the Abutment.
- Choose Y axis for the extrude direction, and enter 10 m as the extrude length. Select [apply] button.
- Choose the 3 faces of the embankment.
- The direction is X axis. Check the [Reverse Direction] option.
- Enter 10m in the Length and press [Apply].

▶Generating ground/structure solids



Tip

There is two ways of selecting target object and direction. One is to select using the [Work tree], another is to select directly on the screen.

- Create 2 faces to divide the soil in the same way.
- Change the [Selection Filter] into 'Edge', and select the 2 surface lines.
- Select the Y axis as the extrude direction. Enter length '70m' so that generated soil can be divided by surfaces.
- Uncheck the [Reverse Direction]. Select [OK].

Tip

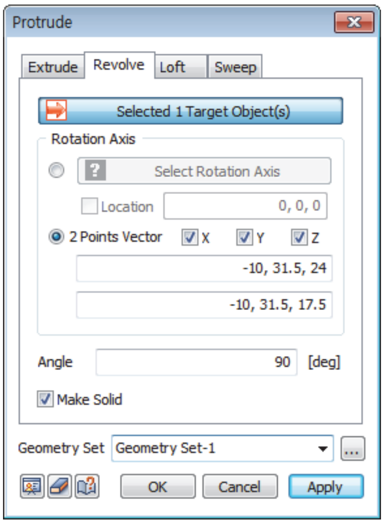
It is recommended that the face used as a tool object for dividing solid should be larger than the solid it divides. If the size of the face is smaller than the solid, even if the difference is minor, the solid is not going to be divided.

* : Geometry > Protrude > Revolve

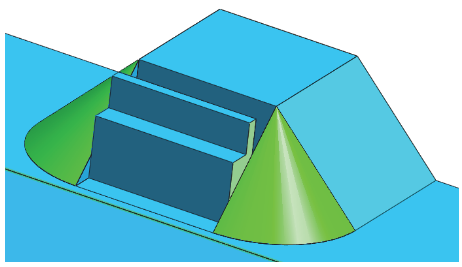
This is the process of creating line/face/solid the upper shape of point/line/face by rotating the shape a into specific angle.

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

- Select the slope face of the embankment as following image.
- Select rotation axis by [2 Points Vector] and select two dots in order as following image.
- Enter 90 in [Angle], check the shape by clicking preview icon.
- In the same way, generate the opposite corner.



▶Generate embankment solid



Tip

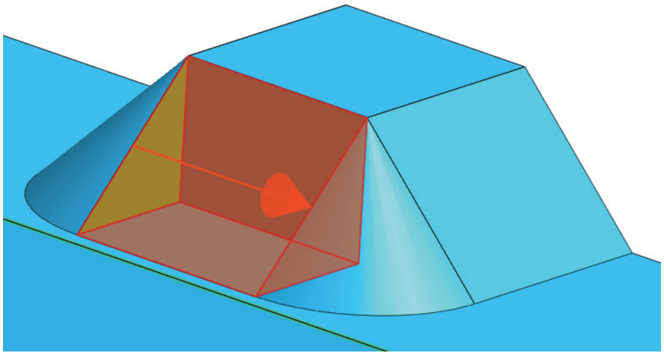
Select 2 points and set the rotation axis as the directional vector which passes through those 2 points. The direction of the vector is determined by the selection order. (+) value refers to the clockwise direction of the rotation angle.

* : Geometry > Protrude > Extrude

Complete generating solid of the embankment and abutment connection.

- Select the inner face of the corner solid as the following image.
- Check the [Normal to Profile(s)] for the direction.
- Enter 10m in length. Select [OK] and check the generated solid.

▶Complete generating embankment solid

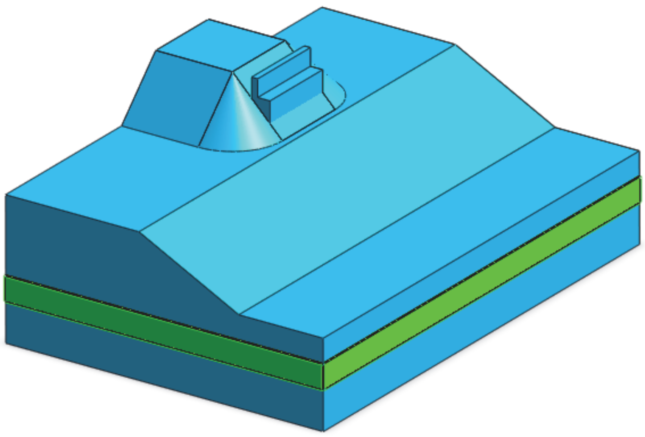


* : Geometry > Divide > Solid

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

- Choose the 'Soil' solid as a target object.
- Select the 2 generated surfaces as tool surfaces.
- Select [Ok], and check the divided solids

▶Complete dividing 'Soil' solid

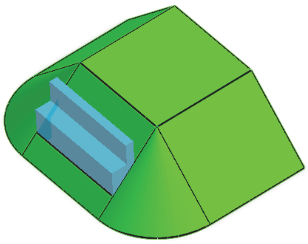


* : Geometry > Boolean > Solid

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

- Choose the [Fuse] tab.
- As a target object, select the 6 embankment solids generated in the previous step.
- Check [Merge Faces] and Select [OK].

▶combine embankment solids.



* : Geometry > Surface & Solid > Auto Connect

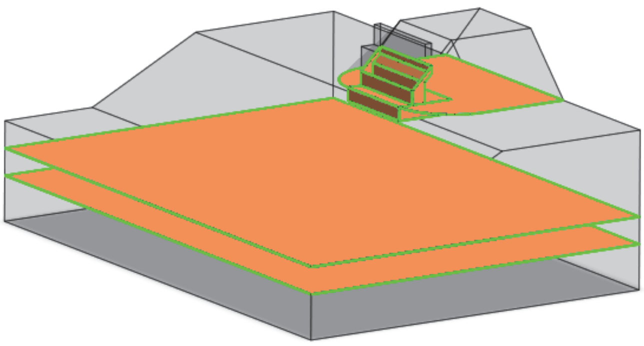
This is the process of automatically generating shared face and deleting the duplicate part between all the generated solids. When creating a mesh, it is necessary to generate a shared face so that nodes can be generated on the every boundary of the domain.

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

Tip

To prevent analysis error caused by unconnected nodes between meshes, it is recommended to check whether shared face has been generated or not before creating mesh. The [Auto Connect] function is for creating shared face of all the solids at once. Using the tool available in the menu Geometry > Tools > Check Geometry > Check Duplicates, it is possible to verify if the shared has been correctly generated. All the surfaces appearing in yellow represent the faces where the shared face is successfully created. If the yellow surface doesn't appear between 2 surfaces that should be connected, the connection is not correctly done and has to be created using either "Auto-connect" or other Boolean operations.
In situation where the sharing face cannot be created because of complex geometrical shapes, "contacts" are an option that can remedy to this situation by providing the connection between the mesh sets.

▶Check duplicates – Auto connect



4.2 Generate Mesh

Mesh shape and mesh quality are very important in finite element analysis. Generally speaking, small mesh size makes good mesh shape (quality). However small mesh sizes will also extend analysis time. So it is recommended to determine the mesh size by considering both accuracy and efficiency of the analysis.

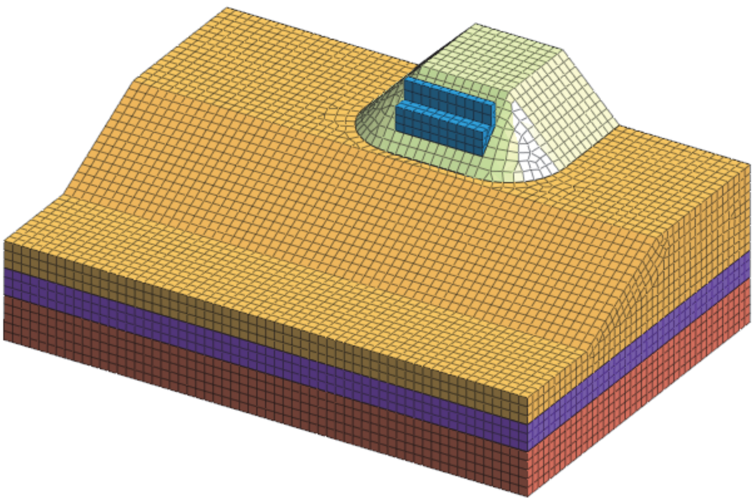
*  : Mesh > Generate > 3D

This is the process of generating element for 3D geometry. Select 3D geometry of ground/structure to generate the mesh. Property of each solid can be assigned during the mesh generation. The property of each mesh can be assigned one by one like that during the mesh generation, or it can simply be assigned later using Mesh>elements>[Parameters] after generating all the mesh set for all the solids.

Generate hybrid mesh, using [Auto-Solid] tap, for 'Weathering soil' and 'Embankment', including the concrete structure. Generate mesh for 'Weathered rock' and Soft rock' in hexahedron shaped using [Map-Solid] tab because the shape of these 2 layers is regular and Map-meshing can be performed.

- Select [Auto-Solid] tab...
- Select the 4 solids (Abutment, Base slab, Embankment, Weathering soil).
- Enter 1m for the [Size].
- Choose the [Hybrid Mesher] in the dropdown menu.
- Select preview icon for checking the mesh before you create it.
- Select [Apply]. Check the generated mesh.
- Select the [Map-Solid] tab.
- Select 'Weathered rock' and 'Soft rock'.
- Enter a 1m mesh size and Select [OK] button.

▶Generating solid mesh

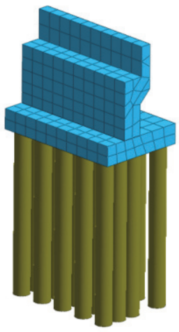


* : Mesh > Generate > 1D

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

- Select all the Pile lines (20) and type 1m in [Division].
- Set property for the pile type and enter 'Pile-beam elements' as the mesh set name.
- Select [OK] button.

▶Generating pile-beam elements



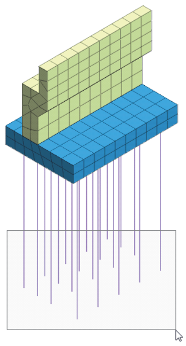
Tip

The size of the beam elements and division does not have a great significant before the generation of the pile elements. It is because the beam element nodes and the ground nodes are automatically connected together when the 'Pile elements' are generated. Even if the beam element nodes are connected to the nodes of the ground automatically, it is possible to define separately Pile interface elements to consider interactions with the ground more in details.

* : Mesh > Element > Pile/Pile Tip

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

- Choose the [Pile] tab.
- Choose all the 20 generated Pile-beam elements.
- Set the property to 'Pile(Interface)' and Select [Apply]
- Choose the [Pile Tip] tab.
- Select the 20 pile tip nodes as in the following image.
- Choose the property as 'Pile tip' and Select the [Apply] button.

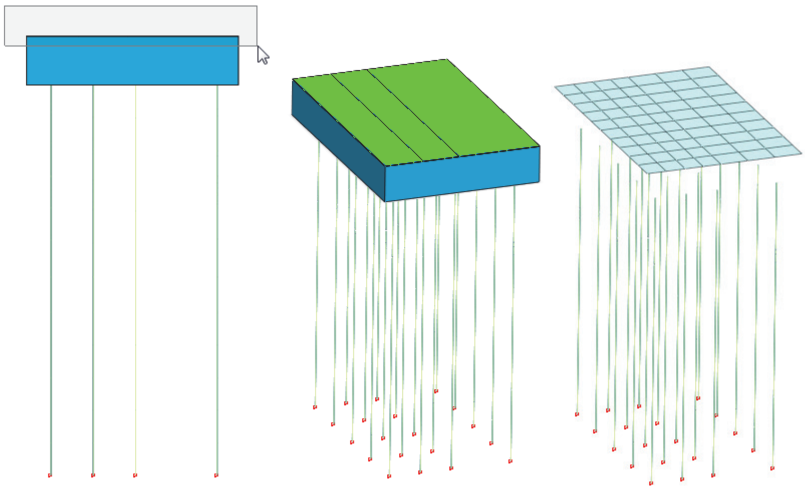


* : Mesh > Element > Create

Add a gauging shell to check the structural member force of the base slab. The Gauging shell is generated by selecting the solid elements (to check the member force) and element boundary (to generate the gauging shell).

- Select [Other] tab.
- Select the gauging shell in the drop down menu.
- Select 'Base slab' solid in the part selection. (If the geometry is Hidden, check the geometry check box to show it on the screen in the work tree)
- Select the face where the gauging shell element will be generated. Select the 3 faces of the top of the base slab as in the following image.
- Set the property to [Gauging shell] and Select [OK].

▶Generate Pile/Pile tip/Gauging shell



Tip

The ground element should be generated before generating pile elements. Pile element which simulates 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.

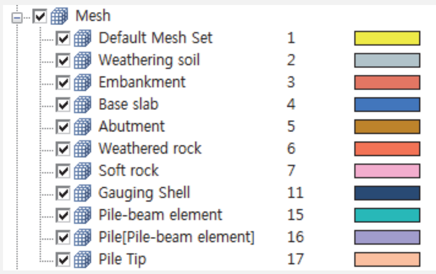
*  : Mesh 〉 Element 〉 Parameters

In this step, we check if the right 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 at each mesh set have to be assigned during this step using the [parameter] option.

- Select the [3D] tab.
- 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 is changing during the analysis, its property have to be set to 'Weathering Soil' in this step and will be changed to 'Weathering soil' to 'Concrete' in the next step using the function Static/Slope Analysis>Boundary>Change property.
- Assign the suitable properties to the selected mesh set.
- Change the properties by clicking the [Apply] button.
- If you select a mesh set in the work tree, you can see its material/property in the property window.

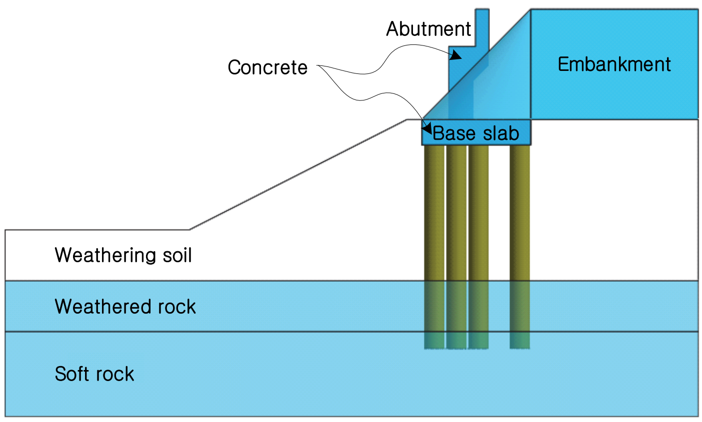
Tip

Grouped mesh sets corresponding to each solid are generated automatically during the meshing and they can be selected in the work tree. The property of each mesh set can be changed according the associated material.
It is recommended to change the name of the mesh sets using the [F2] key of the keyboard in order to be able to define more easily the construction stages of the analysis.



<Change mesh set names>

▶Overview(cross-section)



Section 5

Analysis Setting

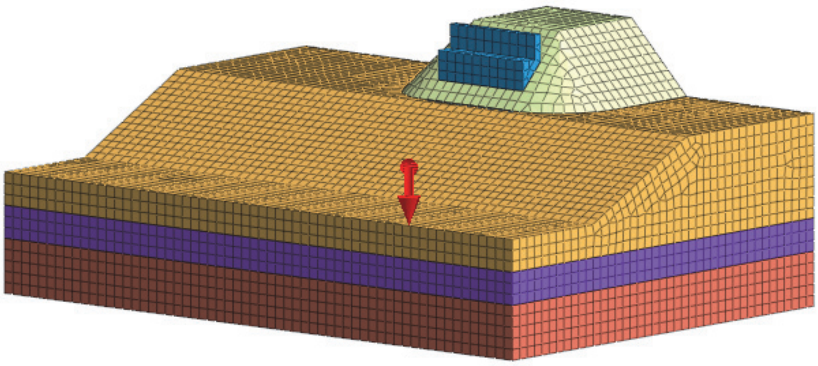
5.1 Setting load condition

* : Static/Slope Analysis > Load > Self Weight

Gravity applied on the model is calculated automatically by the multiplication of the ground and structure's unit weight (in the material data) with the acceleration of the gravity. The setting of gravity is done easily by selecting the scale factor corresponding to the direction and scale of the gravity. Default value is 1 (normal gravity), applied in Z direction.

- Put -1 for Gz value, and type 'Self weight' at the [Load set] name. Select [OK] button.

▶Set a self weight

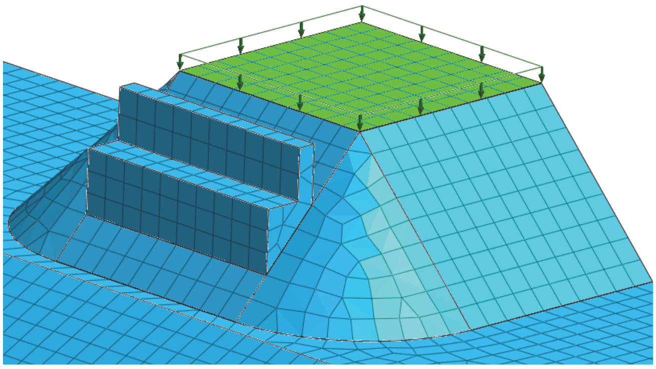


* : Static/Slope Analysis > Load > Pressure

In this step, let's apply a surface load at the top of the embankment. Uniform or linear/nonlinear load can be defined.

- Select the [Face] tab.
- Select 'Face' as the [Type].(Check the Geometry checkbox of the solid in the work tree to show it on the work window)
- Select the face of the top of the embankment.
- Choose the load direction type as the normal direction.
- Check the [Uniformly Distributed Load] option and enter 100(kN/m2) in the [P or P1].
- Click the preview icon to check the loading.
- Enter 'Surface load' in the [Load set], and Select [OK].

▶Set surface load



Tip

For load or boundary condition application, geometric parts can be directly selected (Edge, Face) as well as element boundaries (Edge, Face) in [Select object]. Since all the loads and BC are automatically converted to nodal loads for analysis, type of the target object does not make any difference in analysis results. Selecting geometry instead of nodes/elements makes the selection process more convenient. It is also useful in case of mesh deletion and re-meshing because loads bounded to geometry will remain associated to the model, whereas loads associated to nodes/elements will be deleted along with the mesh sets.

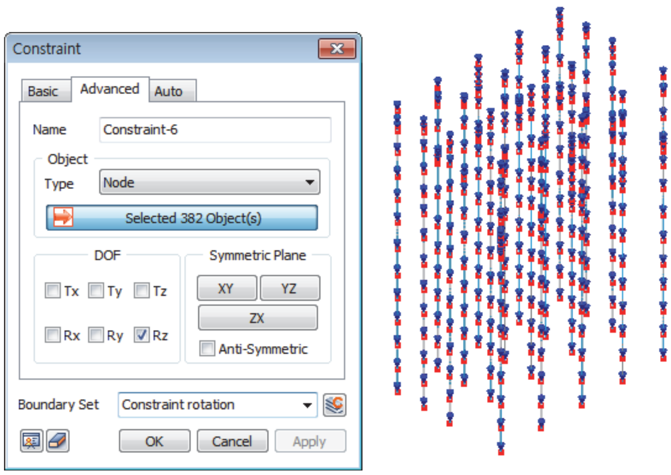
5.2 Setting Boundary Condition

* : Static/Slope Analysis > Boundary > Constraint

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

- Select [Auto] tab.
- Check [Consider All Mesh Sets], and enter 'Ground boundary condition' in [Boundary Set].
- Select [Apply].
- Set to show all the mesh in the work tree and check the generated boundary condition.
- Select [Advanced] tab.
- Set the [Type] to 'Node'. Select the node of the Pile-beam elements, and check [Rz].
- Name the [Boundary Set] as 'Constraint rotation', and click [OK].

▶Set pile constraint



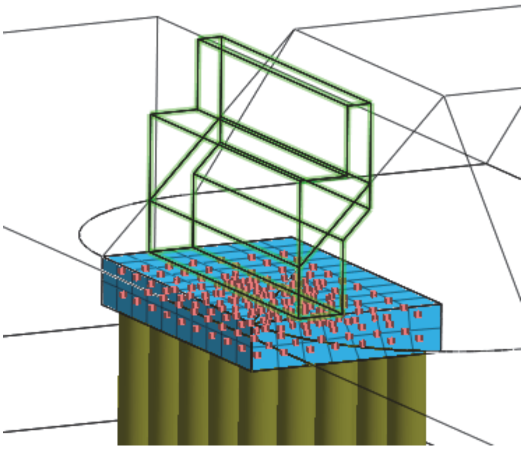
* : Static/Slope Analysis > Boundary > Change Property

In this process, we will learn how to change the property and material assign to a mesh set during the construction stage analysis. Even if only one property can be assigned to one mesh set at the same time, [Change property] function can be used to change 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'.

- Select [General] tab.
- Select the elements of the 'Base slab'.
- Change the property to 'Concrete'.
- Set the boundary condition name as 'Structure' and Select [Ok].

▶Change property of the base slab



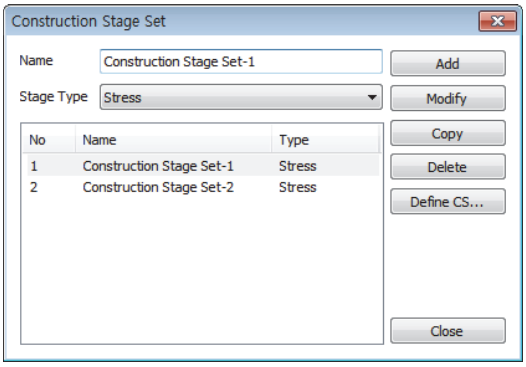
5.3 Defining Construction Stage

* : Static/Slope Analysis > Construction Stage > Stage Set

In this step, we will learn how to set the construction stage analysis for each stage. Two construction stage sets will be generated to consider the embankment which is supported by piles or not. The final loading of the embankment will be divided in several increments. Construction stage is defined according to mesh set names that you defined previously for the mesh sets, so it is recommended to change the mesh set names accordingly to be sure to follow correctly this construction stage definition procedure.

- Set the analysis type as [Stress].
- Click 2 times on [Add] to create the two construction stages.
- Select [Define Construction Stage] to define each stage.
- Construction stage of the each set is defined as following below.

▶Define construction stage set



Tip

When you want to define multiple construction stage sets by changing only few specific conditions, construction stage sets can be easily defined, copied and then modified. Define a construction stage and then make another construction stage by a [Copy] button, then you can modify specific stage that needs to be changed for the copied construction stage.

Construction Stage Set-1 (Before applying Pile Foundation)

Stage 1 - Name: Foundation

- Activated Data Mesh : [Base slab], [Weathering soil], [Weathered rock], [Soft rock]
- Activated Data Boundary Condition : [Ground boundary condition]
- Activated Data Static Load : [Self weight]
- Check the option [Clear Displacement].
- Save and define next stage by clicking [New] button.

Tip

When setting construction stage, stage 1 is an initial stage for calculating stress distribution of the foundation status. If there is no additional displacement, the foundation will be assumed to be in a stable status. So, to exclude the displacement by the self weight in the total ground displacement, check the [Clear Displacement].

Stage 2 - Name: Excavation

- Deactivated Data : [Base slab]
- Save, and define next stage by clicking [New] button.

Stage 3 - Name: Structure Construction

- Activated Data-Mesh: [Abutment], [Base slab], [Gauging shell]
- Activated Data-Boundary Condition : [Structure]
- Save, and define next stage by clicking [New] button.

Stage 4 - Name: Embankment

- Activated Data Mesh : [Embankment]
- Save, and define next stage by clicking [New] button.

Stage 5 – Name ; Loading

- Activated Data-Static Load : [Surface load]
- In [Analysis Control], enter 5 as the 'number of increments' and select 'Every Increment' so the results of each increment will be plotted in the result work tress.
- By doing so, we divide the total load 100 kn/m2 into 5 steps and the results of the each step will be saved.
- Save and close.

Construction Stage Set-2 (After applying Pile Foundation)

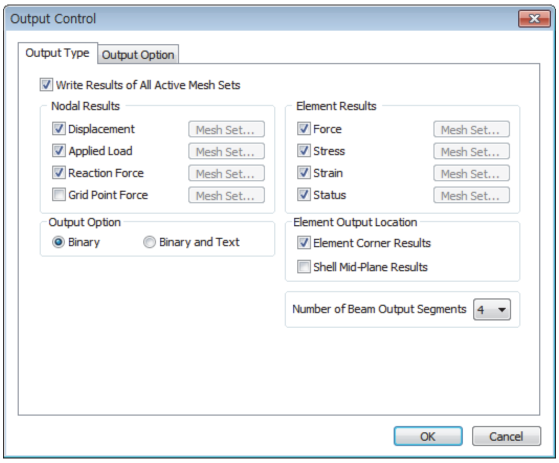
Stages 1,2,4,5 are identical to the 'Construction Stage set-1'. The only changes will appear in the stage 3: 'Structure Construction' where pile elements will be activated additionally.

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 control the solving process along with the advanced options used to control the nonlinear solver and the 'Output Control' options will control the type of results that will be calculated by the solver and then 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 element such as pile element among the ground when interfacial behavior occurs, you have to check **Element Results > Strain** before analyze.



* : Analysis > Analysis Case > General

- Type in the name of the Analysis case and select 'Construction Stage' as the solution type.
- Set Analysis > General > Initial Stage > Initial Stage for Stress Analysis to '1:Foundation'. (Since foundation line is not horizontal, do not check the k0 condition.)
- Select [OK].
- Generate the analysis case for each of the 2 construction stage sets.

5.5 Perform Analysis

Perform analysis and analyze the results. After the analysis is done, the software automatically switches to [Post-Mode](checking results). To do some modification of the model and options after the analysis, you have to switch back to the [Pre-Mode].

* : Analysis > Analysis > Perform

- Perform analysis

Tip

During the analysis, you can follow the calculation process in the [Output Window], to check whether the analysis is converging or not, warnings and errors that may occur during the analysis will appear in the Output window as well.

Section 6
Results

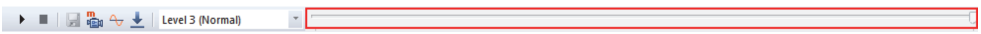
You can check the displacement of ground and structural elements, the member force of the base slab and the pile behavior based on construction stage steps in the work tree. All the results are shown as contours, tables, and graphs. In this tutorial, the main result items which need to be checked are:

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

6.1 Verify Displacement

Verify the 'Displacement' in the work tree after the analysis. TX, TY, TZ are the displacements based on X, Y, Z. Each of the horizontal displacement and settlement tendency occur according to banking and surface loading can be verified by TX, TY,TZ. '(V)' means '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 at the same time of displacements and principal stresses.

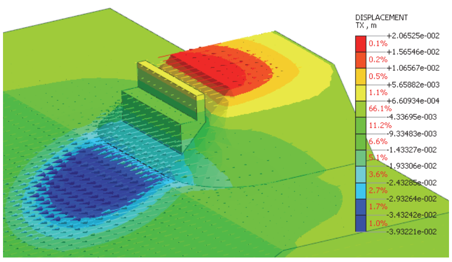
- By moving the sliding bar at the bottom of the work window, it is possible to simulate the results changing in each construction stage and each load step.

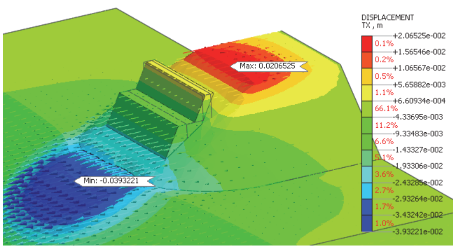


Check the results at the last loading step.

- Select the last construction stage/last load step of the result tree, select Displacement > TX TRANSLATION(V).
- You can see select to view either the deformed or undeformed model according to X direction in Result > General > Deform. (The degree of deformation of the model can be modified by the scale factor through the property window. It can be shown in the work window by checking Results > Show/Hide > [Actual Deformation])
- Select Results > Advanced > Probe to see the result value of the selected node/element, and also it is possible to find location and value of the Max/Min/Abs Max of the result.

▶Horizontal displacement(Undeformed)
▶▶Horizontal displacement(Deformed)

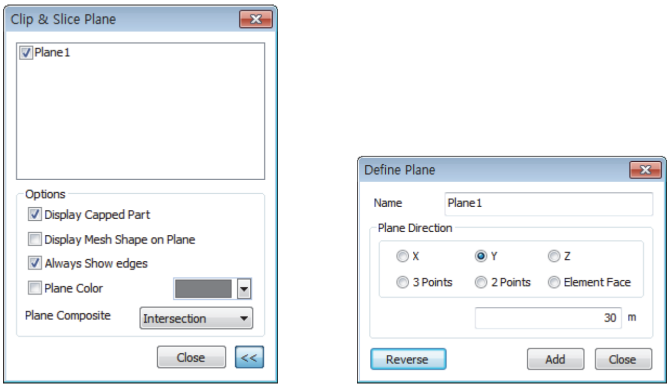




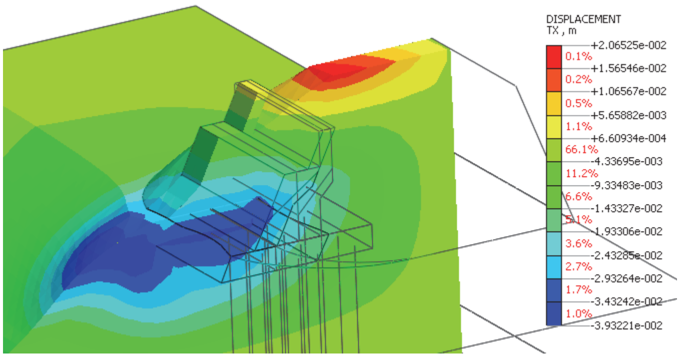
It is possible to divide 3D model by a specific surface and see the value of the results on this face. GTS NX offers 3D-2D Wizard function which is used for checking some specific point results of the inside of the model by dividing model with a defined plane.

- 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 axial direction based on GCS or by setting 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'.

▶Clip & Slice Plane option
▶▶Define Plane

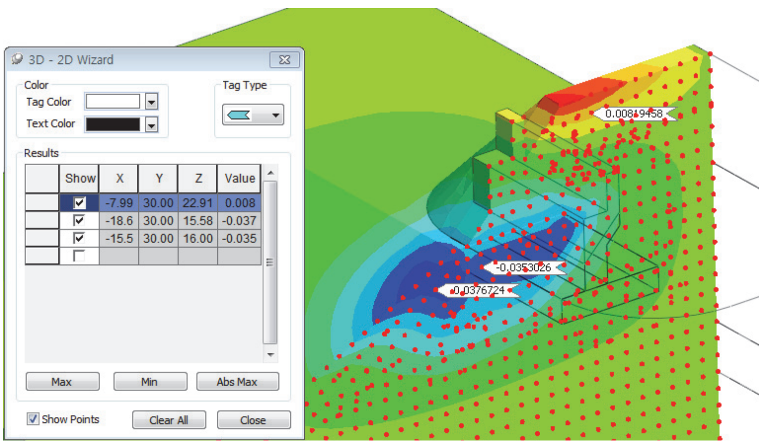


▶Sliced by plane 1



- Select Result > Advanced > Others > 3D->2D Wizard. 3D->2D Wizard is a function of 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 shown in the work window.

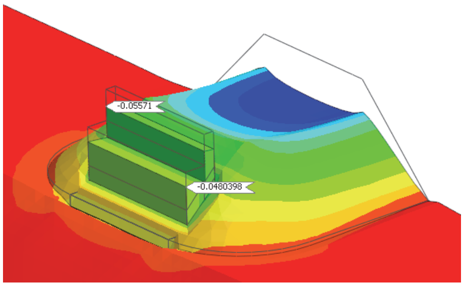
▶3D-2D Wizard
▶▶Result tag of the cutting plane

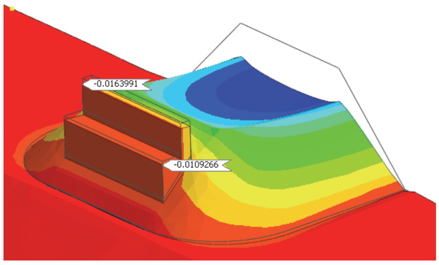


Let's check and compare the abutment settlement with and without piles.

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

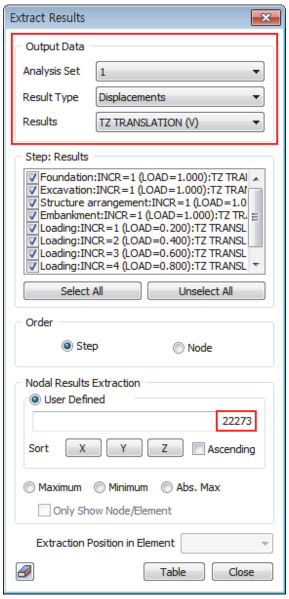
▶Vertical displacement (Before applying pile foundation)
▶▶Vertical displacement (After applying pile foundation)

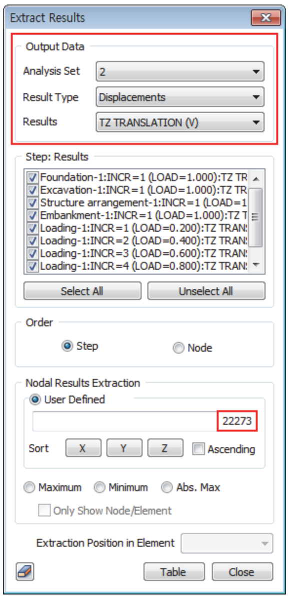




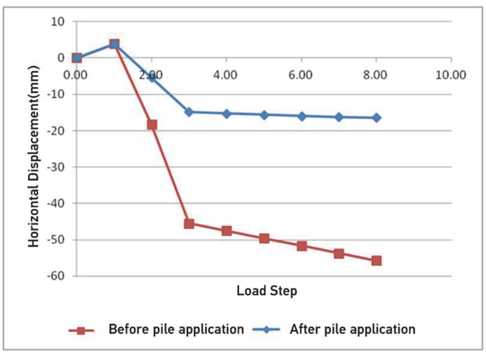
- Extract the results of each step at the end of the abutment. And compare its settlement after/before the application of pile foundation.
- Select Results > Displacement > TZ. Select the node (22273), or Type '22273' and select [Table] button. In the plotted table, the graph can be plotted a right click of the mouse. In the same way, select the other analysis case and compare the settlement graph at the same node. The max settlement before applying pile element is about 60mm and the settlement after applying pile element decreased to 17mm.

▶Extract Results
▶▶Extract table
▶▶▶show graph





▶Comparing settlement result graph



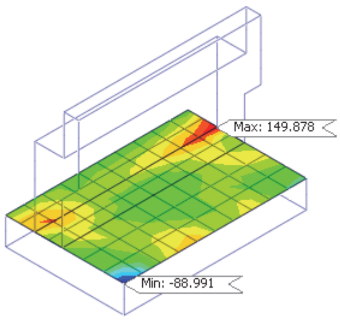
6.2 Verifying Member Force & Stresses

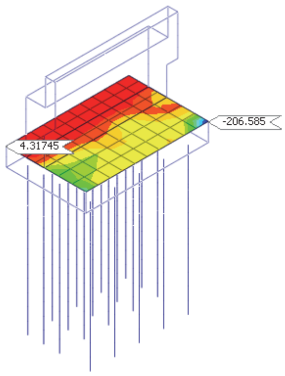
You can check the member force of the solid element through the gauging shell generated on the top of the abutment. If the model consists of solid element concrete structure, design member force can be checked using gauging shell.
Verify the gauging shell and the member force of the pile foundation. 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 material coordinate system or when generating 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 structure member force, select Result > General > No Result > Exclude.

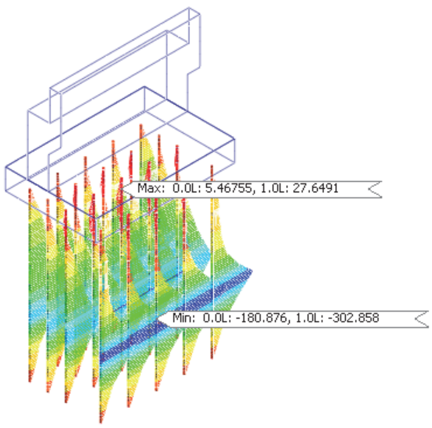
▶Base slab moment (Before applying Pile)
▶▶Base slab moment (After applying Pile)





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

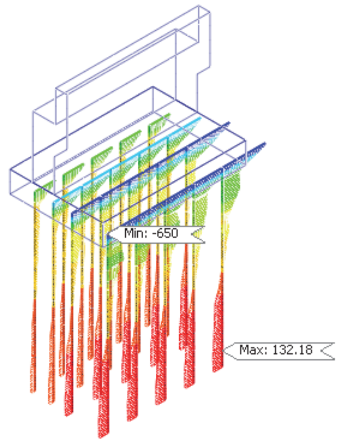
▶Pile moment



6.3 Verify Frictional Force and Displacement of Pile/Ground

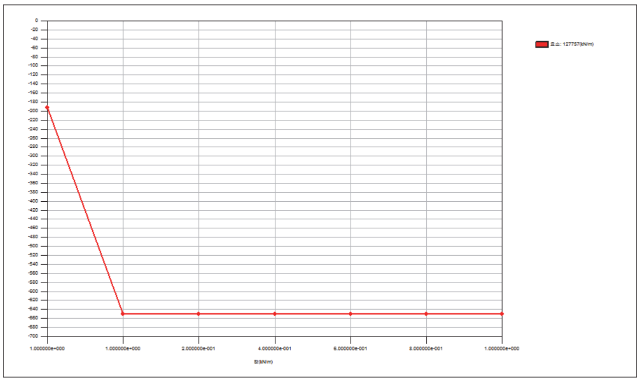
You can check the displacement and friction between the ground and the structure in about two normal directions and tangent direction through Pile element results. By showing the displacement-friction diagram of the each stage, you can check whether the ultimate bearing power of piles is reached or not.

- Verify tangential friction force between pile and ground by checking Pile Force > TANGENTIAL X of the last load step in the result tree. If you put this in relation of relative displacement by a diagram you can see the applied 'T-Z CURVE'. The results show that ultimate shear force is generated right after the banking. And plastic behavior occurs when loads are over the ultimate shear force.
- Select Pile Relative Displacements > TANGENTIAL X to see the relative displacement of the pile and the ground. This looks like it has similar tendency with the friction. But you can see that even after the appearance of the ultimate bearing power, the displacement still increases.



▶Pile tangential friction force

▶▶change of pile head friction force



▶Tangent direction displacement of each step