Chapter 2
3D Retaining Wall

Workflow

	: Geometry > Protrude > Extrude	7
	: Geometry > Divide > Solid	8
	: Geometry > Surface & Solid > Imprint	9
	: Geometry > Surface & Solid > Auto Connect	10
	: Mesh > Generate > 3D	11
	: Mesh > Element > Extract	12
	: Mesh > Generate > 1D	13
	: Mesh > Element > Parameters	14
	: Mesh > Element > Interface	15
	: Static/Slope Analysis > Load > Self Weight	17
	: Static/Slope Analysis > Load > Prestress	17
	: Static/Slope Analysis > Boundary > Constraint	18
	: Static/Slope Analysis > Construction Stage > Stage Set	19
	: Analysis > Analysis Case > General	20
	: Analysis > Analysis > Perform	20

3D Retaining Wall

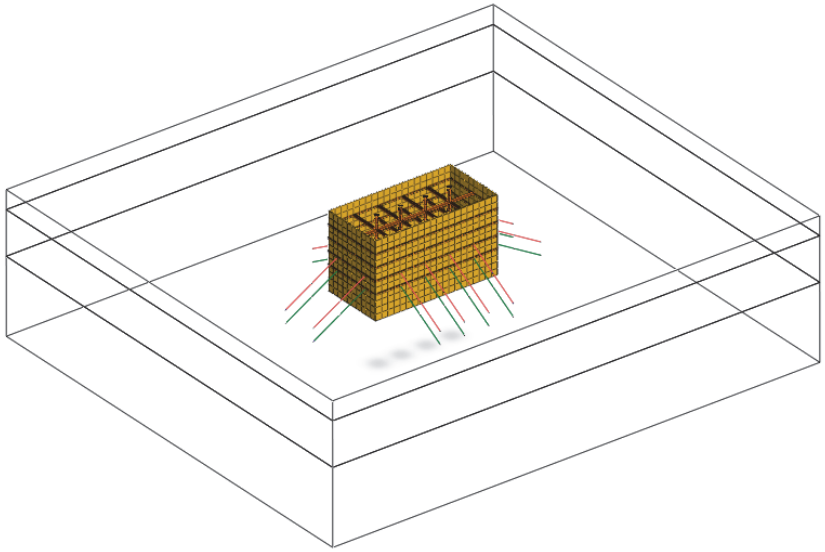
Section 1
Overview

1.1 Learning Purpose

This tutorial identifies the ground – structure interaction by analyzing construction stage of 3D retaining wall excavation. The objective of reviewing retaining wall stability through finite element method is to calculate displacement and stress according to ground elastic – plastic characteristics and the interaction between retaining wall and structure members. Through this method, you can verify not only stress and displacement of the retaining wall but also the influence on the surrounding ground and adjacent structures at the same time.

Unlike 2D analysis, the retaining wall installed in the 3D excavation model is highly affected by structure's direction and boundary conditions. Especially, it is possible to review in detail the stress distributions on cross-sections, which is not possible in 2D models. Also interface is added between ground and retaining wall to simulate the ground-structure interaction more realistically.

▶Analysis model overview



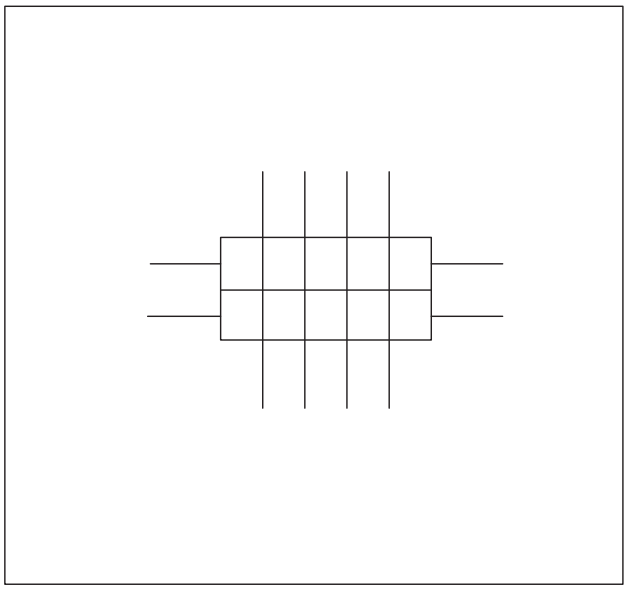
In this tutorial, the following main concepts will be explained:

- Apply Modified Mohr Coulomb model (simulate ground hardening)
- Modeling of sheet piles, beams, trusses and anchors
- Apply interface element (simulate separation behavior between wall and ground)
- Apply anchor pretension
- Verify the ground stress, member force, and deformation results

1.2 Modeling and Analysis Summary

The model represents 10mX20m excavation area with temporary facilities which are sheet piles, walling, struts, plus pegs and anchors. Maximum excavation depth is 10m. The excavation process consists of 5 stages(3,5,7,9,10m) and total supports(walling and Stiffener) are installed at 4 levels(2,4,6,8m) with interval distance of 2m. Depending on the ground properties, set struts at level 1, 2 and set anchors at level 3, 4 to reinforce each level. The height of the retaining wall (Sheet pile) is 12m, and the distance from bottom of the excavation is 2m. The ground is consist of 3 layers and depths are respectively: Buried layer 3m, Colluviums 10m, Weathering soil 22m from the surface.

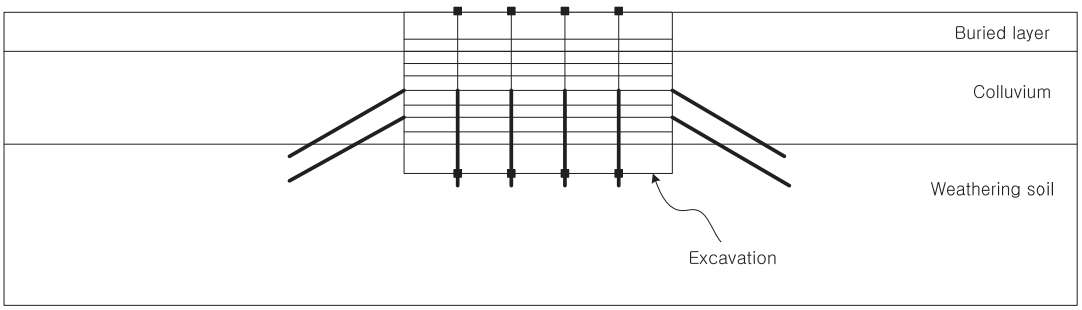
▶Plane



Create the width of total ground area 1.5~3 times wider than excavation width to minimize the influence of the boundary conditions.

The ground composition and excavation stages are illustrated below.

▶Cross-section



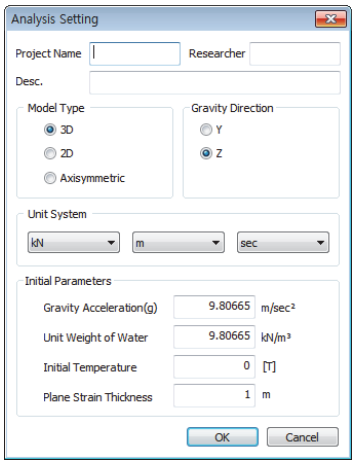
Section 2
Analysis Setting

[Open the attached start file (02_3D Retaining wall_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 the analysis. The input parameters will automatically be converted to the right unit system.

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



Section 3
Defined Material and Property

3.1 Material Definition for Ground and Structures

For material modal type, apply 'Mohr-Coulomb' for the ground and 'Elastic' for the structure. 'Modified Mohr-Coulomb' model is a material model which follows 'Power-law'. It can be used to simulate the combined behavior of nonlinear elastic models and elasto-plastic models. Especially by defining elastic modulus duringloading and unloading processes, we can minimize the uplift of the excavated surface caused by excavation (unloading process). 'Elastic' model does not consider material nonlinearity.

It is necessary to consider nonlinearity of the interface elements which simulate the separation behavior of ground and sheathing wall.

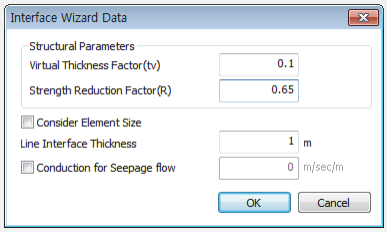
The material for each ground and structural member is listed in the following table. For the interface material, use the parameters calculated automatically by Wizard.

▶Table. Ground material

Name	Buried layer	Colluvium	Weathering soil
Material	Isotropic	Isotropic	Isotropic
Model Type	Modified Mohr-Coulomb	Modified Mohr-Coulomb	Modified Mohr-Coulomb
General			
Poisson's Ratio(v)	0.2	0.2	0.2
Unit Weight(r)	16	17	20
Ko	0.5	0.44	0.74
Porous			
Unit Weight(Saturated)	20	20	22
Initial Void Ratio(eo)	0.5	0.5	0.5
Unsaturated Property	Drained	Drained	Drained
Non-Linear			
Elastic Modulus at Loading(Eref)	22000	43000	60000
Elastic Modulus at Unloading(Eurref)	66000	129000	180000
Reference Pressure	100	100	100
Power Law	0.5	0.5	0.5
Porosity	0.6	0.6	0.6
KNC(>0)	0.9	0.9	0.9
Friction Angle at shear	30	34	38
Ultimate Dilatancy Angle	0	4	5
Cohesion	1	1	1
Preconsolidation	-	-	-
Cap Shap Factor	-	-	-
Material	Interface/Pile	Interface/Pile	Interface/Pile
Model Type	Interface	Interface	Interface
Normal Stiffness Modulus(Kn)	655417	1281042	1787500
Shear Stiffness Modulus(Kt)	59583	116458	162500
Interface Nonlinearities	Coulomb Friction	Coulomb Friction	Coulomb Friction
Cohesion(C)	0.65	0.65	0.65
Frictional Angle	30	34	38
Dilatancy Angle	0	0	0

Tip

When generating interface element using interface wizard, input the 2 parameters (tv, R) as below so that the material properties will be automatically calculated according to the properties of the adjacent ground elements.



- The wizard will calculate material properties through the following method. Apply [Virtual Thickness Factor (tv)] and [Strength Reduction Factor(R)] by using stiffness and nonlinear parameters of the adjacent elements. According to the stiffness of the surrounding or structural members, the parameters and stiffness of the interface material are applied differently.

Ex) $K_n = E_{oed,i} / t_v$

$K_t = G_i/t_v$

$C_i = R \times C_{soil}$

Here, $E_{oed,i} = 2 \times G_i \times (1-\nu_i)/(1-2 \times \nu_i)$

(ν_i =Interface Poisson's ratio =0.45, Interface is for simulating the incompressibility friction behavior. To prevent the numerical error, use 0.45 to calculate Interface Poisson's ratio.)

t_v = Virtual thickness factor (Generally use the value in the range of 0.01~0.1. if the stiffness is big, use smaller value.)

$G_i = R \times G_{soil}$ ($G_{soil} = E/(2(1+\nu_{soil})$), R = Strength reduction factor

General strength reduction factor according to structural members and adjacent ground properties are listed as below.

Sandy soil/Steel material = R : 0.6~0.7

Clay/Steel material = R : 0.5

Sandy soil/Concrete = R : 1.0~0.8

Clay/Concrete = R : 1.0~0.7

▶Table. Structure material

Name	Structure material 1	Structure material2
Material	Isotropic	Isotropic
Model Type	Elastic	Elastic
Elastic Modulus(E)	2.1E+08	2.0E+08
Poisson's Ratio(v)	0.3	0.3
Unit Weight(r)	76.98	76.98

3.2 Define Properties

Properties represent physical attributes of the meshes and will be assigned to mesh sets during mesh generation. While defining ground and structure properties, firstly choose the material to be used. And for structure properties, structure types and cross-section shapes (cross-section stiffness) should be further defined.

Use 'beam element' for sheet piles since they are continuous walls with thickness. 'Beam element' is also used for walling, plus pegs and struts since they need to resist to axial/shearing/bending forces. Use 'embedded truss element' for anchors which only resists to axial force. Struts are usually assumed as 'truss elements' which only resist to axial force. But in case of a model with plus peg just like the model of this tutorial, it is reasonable to assume that they also resists to the shearing and bending forces. 'Embedded truss element' is only for buried structural members. Even though It behaves like 'truss element', it does not need to be connected to adjacent elements with nodes in 3D analysis. So it can be applied very easily in 3D analysis.

The ground properties are shown in the following table. For interface material properties, use the parameters which are calculated automatically through the Wizard.

▶Table. Ground property

Name	Interface (Buried layer)	Interface (Colluvium)	Interface (Weathering soil)	Buried layer	Colluvium	Weathering soil
Type	Other	Other	Other	3D	3D	3D
Model Type	Interface	Interface	Interface	-	-	-
Interface Type	Face	Face	Face	-	-	-
Material	Buried layer	Colluvium	Weathering soil	Buried layer	Colluvium	Weathering soil

The structure properties are shown in the following table. The rigidity of the cross-section will be automatically calculated once the cross-section shape is defined.

▶Table. Structure property

Name	Sheet Pile	Walling, Plus peg, Strut	Anchor
Type	2D	1D	1D
Model Type	Shell	Beam	Embedded Truss (linear elasticity)
Material	Structure material 1	Structure material 1	Structure material 2
Section	-	H-Section	Solid Round
Section Size	Uniform Thickness : 0.1	300x300x10/15	0.025

Section 4
Modeling

[Start Modeling]
Since the purpose of this tutorial is to study 3D geometry / mesh generation, analysis workflow and results checking, you can start the tutorial by opening the start file in which basic materials and properties have already been predefined.

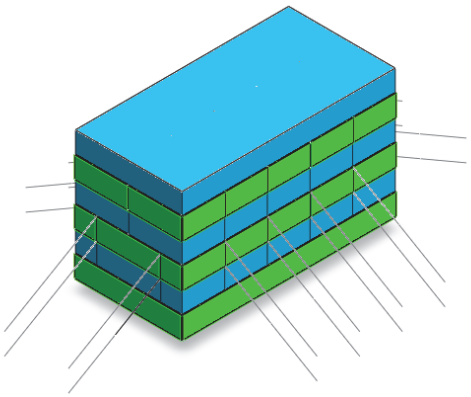
4.1 Geometry Modeling

∗ : **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 extrude solid directly too.
Create 3D ground and excavation area.

- Change the selection filter to edge, and select lines (68) of the 'Excavation and wall' as a target object.
- Choose axis Z as direction and check the [Reverse Direction] option.
- Enter the length as 2(m) which signifies the height for struts.
- Click [Apply] and check the generated solids on the work window.

▶Generate excavation solid



Tip

For beam elements in 3D model, the ground elements and nodes need to be connected. Beam elements have to be generated by using [Extrude] function after ground meshing. Therefore spaces are needed in the solid surfaces to extrude beam elements. Also anchor elements must be connected to the nodes of wailing/walls. Therefore the solid surfaces must be divided so that the elements can generated on these locations.
When creating solid by extruding from closed lines, the intersections are locations for the beam elements extrusion later.

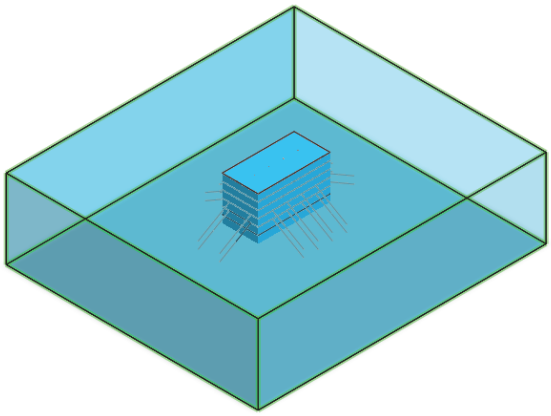
Tip

It is possible to select target objects both in the work tree and directly on the screen.

- Generate solid of 'Ground area'.
- Select face(1) of the ground area..

- Set the direction to Y axis. Since extrude direction is the same as GCS, uncheck [Reverse Direction].
- Enter 80(m) for whole length of the ground area. Click on [OK].

▶Complete generating whole solid

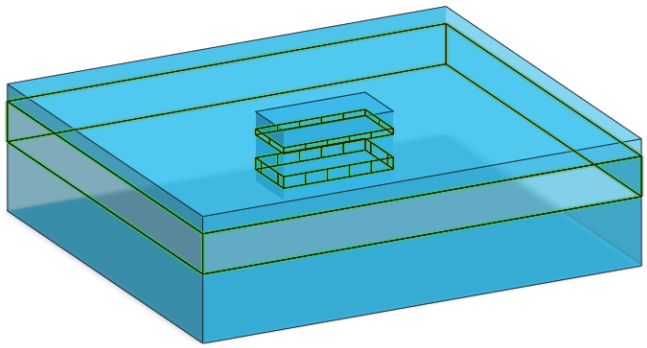


* **: Geometry > Divide > Solid**

This process divides ground solid and excavation area to using bedding planes and cutting surfaces of construction stages.

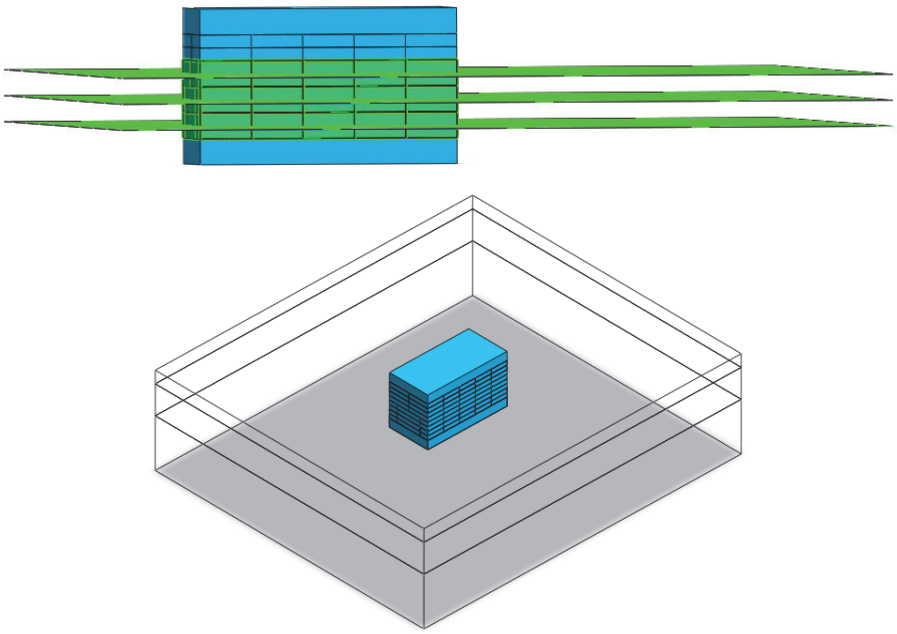
- First, divide ground and excavation solids by bedding planes
- Select the entire solid (7) as a target object.
- For the tool object, select 'Bedding plane'(2)..
- Click on [Apply].

▶Complete dividing bedding plane



- Divide the excavation solid by cutting planes.
- Select the solids (3) which will divide by cutting planes.
  (Since the other solids cannot be divided even if they are selected in target objects, you can just select all the solids)
- Select cutting plane (3) as a tool object.
- Click [OK].

▶Dividing excavation

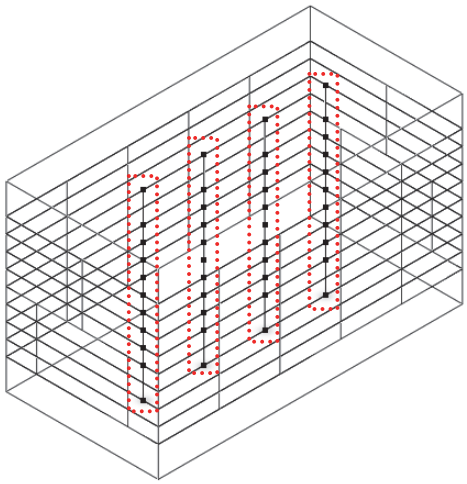


* : Geometry > Surface & Solid > Imprint

When your generate beam elements in a 3D model, if the elements are located on the surface of the solid, connecting nodes are automatically generated by extruding function. However, in case of plus pegs which pass through the solid, [Imprint] function need to be used to recognize the locations of beam elements on the solid surface. Use [Imprint Auto] to generate nodes on the all the excavation surfaces which the plus pegs go through.

- Select [Imprint Auto] tab.
- Select entire excavation solids (10) as target object.
- Select plus pegs (4) as a tool object.
- Click [Ok].

▶Auto imprint plus peg



* : Geometry > Surface & Solid > Auto Connect

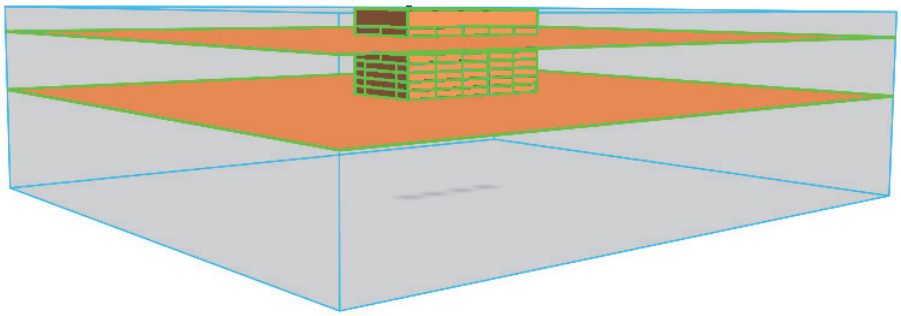
This process generates shared faces automatically after deleting the duplicate parts in the entire solid. It is a necessary step before mesh generation, since nodes need to be connected to transfer forces.

- Select all the solids (13) and click on [OK].

Tip

To prevent the analysis errors from unconnected nodes between elements, it is recommended to verify the generated shared face. You can generate the share faced through the [Auto Connect] function, and the generated share face can be checked by Geometry > Tools > Check Shape > Check Geometry > Check Duplicates.

▶Auto connect



4.2 Generate Meshes

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.

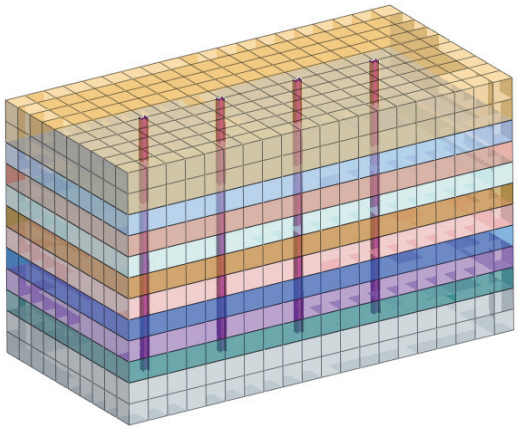
When generating mesh, you can assign properties to the each solid first and mesh them respectively. Or you can mesh the entire model first and then assign properties to each mesh using [Parameter].

* : Mesh > Generate > 3D

This process generates 3D meshes of Ground area. When you generate meshes for edges which pass through a solid, you can check the 'interior Edge' option to generate the meshes simultaneously with the solid. In this way the elements and nodes will be connected to the ground.
Select excavation/ground solids to generate mesh.

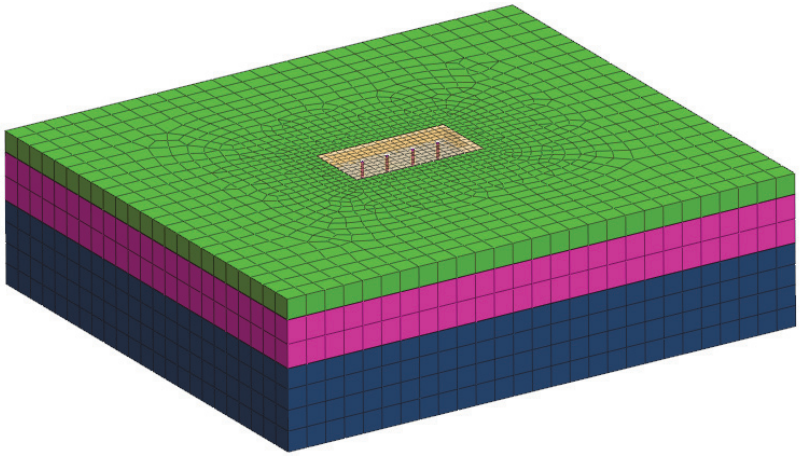
- Generate excavation solid mesh.
- Choose 'Auto-Solid' tab.
- Select the excavation solids (10).
- Enter '1' in the mesh size.
- Select 'Hybrid Mesher (Hexahedron centered)' at the dropdown menu.
- Set advanced option by selecting >> button.
- Check [Interior Edge/Point] and select lines (40) of the imprints as Interior edge.
- Check [Consider Imprinting Shape on Face] option to generate meshes with nodes connected to interior edges.
- Select the property 'Plus peg' to assign to the interior edge.
- Click on [OK] to close the advanced option window. Click [Apply] to generate meshes.

▶Generate element of excavation and plus peg



- Generate mesh of ground area solid.
- Select ground area solid (3).
- Enter '3' in the mesh size.
- Click on [OK] and generate meshes.

▶Generate element of ground area



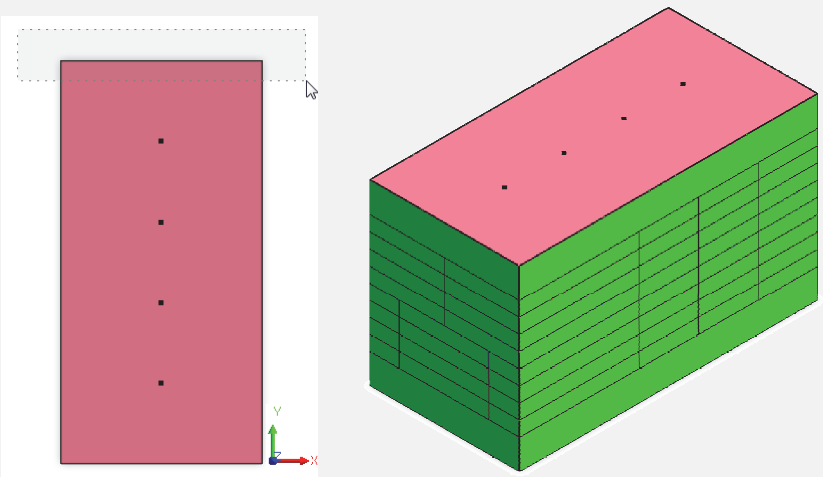
* : Mesh > Element > Extract

This is a process of extruding structure elements. Instead of generating structure meshes separately, they are extruded from faces and lines after the ground mesh generation to connect the nodes with ground elements. The locations (information) of nodes on the generated ground elements can be brought by the extrude function.

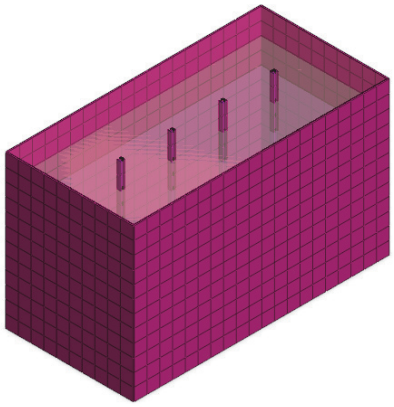
- Generate 'Sheet Pile' element.
- Under the Geometry tab, select the 'Face' type.
- Select entire faces around the excavation solid in the direction of X,Y axis.
- Name the mesh set as 'Sheet Pile', click on [Apply].

Tip

The geometry need to be shown on the work window to be extruded from. A part of the geometry shape can be selected in the following way. From the view toolbar, select the top view. Drag and select all the faces around the excavation solid in X,Y axis direction.

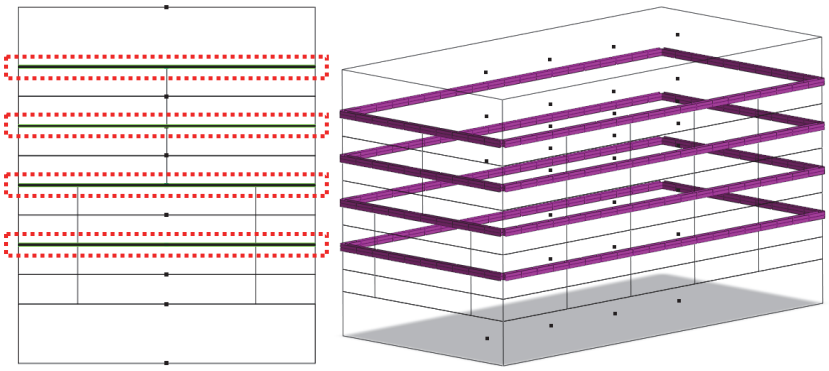


▶Extrude sheet pile elements



- Change to 'Edge' type..
- Select the boundary lines of the Walling as following image, the location of the Walling (depth: 2, 4, 6,8m) are presented as solid boundaries. To distinguish the construction stage, extrude elements separately for each boundary line.
- First, select boundary line located at the Stage1 Walling.
- Select the property to 'Walling' and name 'Stage1 Walling'
- Generate meshes for˙stage2~4 walling' by selecting each of walling in order.
- Each walling mesh set can be later assigned to relevant construction stage referring to the name.

▶Extrude walling element



* : Mesh > Generate > 1D

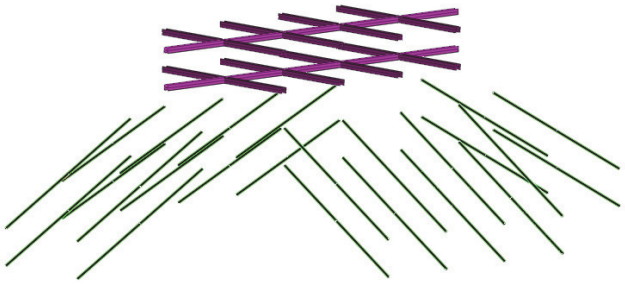
This process generates structure elements which don't need to be connected to adjacent ground element, such as embedded trusses and piles.

Generate strut elements which do not connect to ground and install the anchors after the excavation. Generate mesh sets from 'stage1 strut' to 'stage4 anchor' separately, to install support structures in each construction stage.

- Generate meshes for 'Stage1 strut'.
- Select 'Stage1 strut'(13).
- Enter '1' for division and select the property 'Strut'.
- Name mesh set name as 'Stage1 strut' and click on [Apply].

- In the same way, generate 'Stage2 strut', 'Stage3 anchor', 'Stage4 anchor' by separating the names and properties.

▶Generate structure element



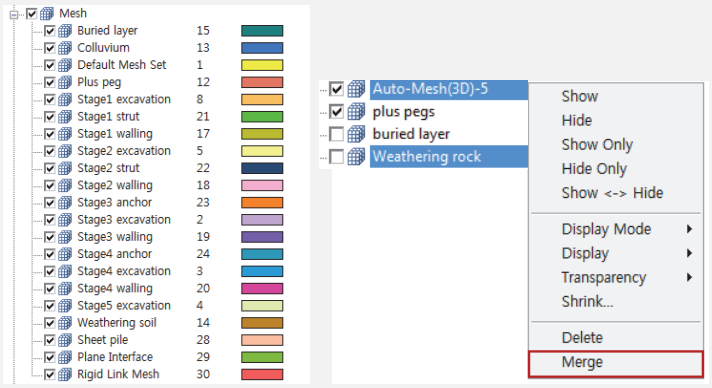
* : Mesh > Element > Parameters

This process checks properties and assigns the right properties to mesh sets. During the automatic mesh generation, all the elements are assigned into one property. You can change the material properties of the each mesh set using [Parameter]. Change properties for the each stratum.

- Select [3D] tab.
- Refer to the image below, select 3D mesh set one by one and assign the proper properties.
- Click on [Apply] button.
- Click on each mesh set in the work tree to verify its properties in the property window.

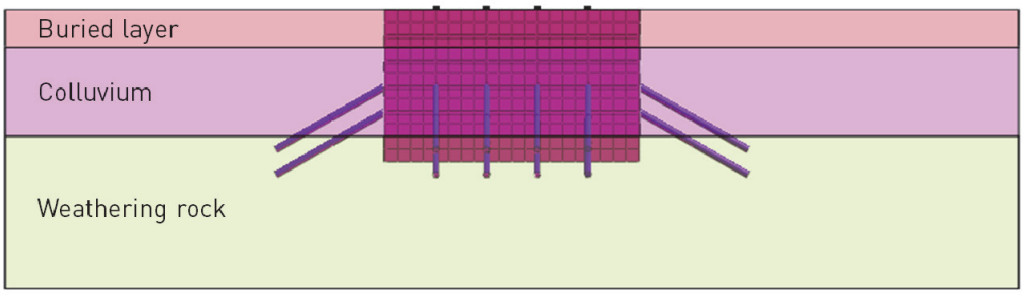
Tip

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 the name of the mesh set. You can change the mesh set name in the work tree by using [F2] key. If meshes which are activated/deactivated at the same time in a construction stage are divided into several mesh set, you can merge those mesh sets by [Merge] function of the context box by mouse right click.



<Change mesh set name/ Merge mesh set>

▶Stratum distribution outline

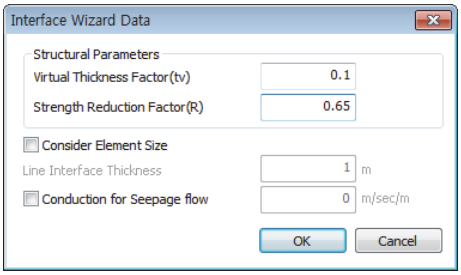


* : Mesh > Element > Interface

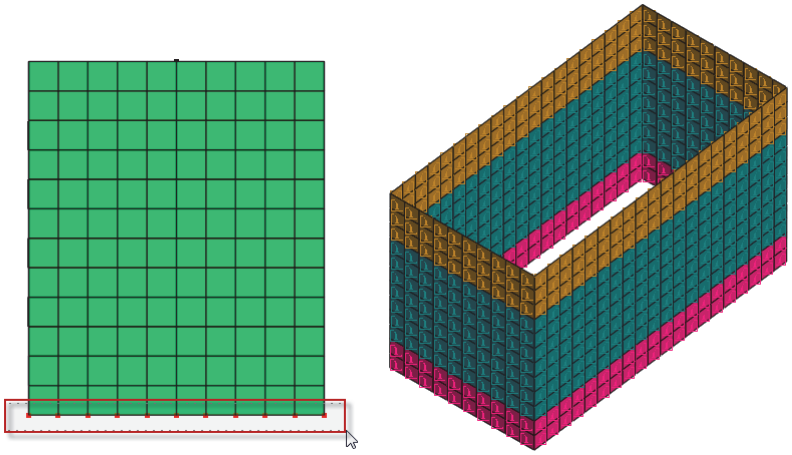
This is process generates 3D interface elements to simulate the separation behavior between ground and wall. Use the generated Sheet piles (shell element) to create interface elements at the side and rear end of the excavation part.

[Interface] function works in the following way. Right after the interface elements are generated, connected nodes are automatically detached at the spots of the interface. And, in between of the detached nodes, it creates a kind of elements which have specific rigidity in normal and tangent directions. For stages in which the interface elements are not activated yet (ex: foundation), to prevent the error, rigid links must be applied to connect the nodes. ON the other hand, for stages iin which the interfaces elements are activated, rigid links should be excluded. In the tutorial, the material properties of the interface elements are automatically set in the wizard by calculating from the surrounding material properties.

- Select [Plane] tab.
- Select the 'From shell' type.
- Select the 'Sheet Pile' elements (720) and choose 'Both' direction.
- Check [Merge Nodes] option and select the nodes (60) of the bottom part of the sheet pile as following image.
- Select 'Wizard' and enter parameters (tv:0.1, R:0.65) like following below.
- Check [Create Rigid Link Element] option.
- Press [Ok] button and see the generated interface elements.
- Tree of Interface material/property for the each stratum is generated automatically.



▶Generating interface element



Tip

If you check [Consider Element Size] option, the wizard will calculate the interface material properties by considering average length (line) and average area (face) of the adjacent ground elements. In other words, it calculates the interface rigidity in normal and tangent direction of interface by multiplying average length (l) and average area (A) to virtual thickness with the following relation.

$$K_n = E_{oed,i} / (l \text{ or } \sqrt{A} \times t_v) \text{ , } K_t = G_i / (l \text{ or } \sqrt{A} \times t_v)$$

If you uncheck the option, it applies unit length (area).

In case of line interface, define the thickness separately. The thickness is an important factor when using the interface for ground material which shows hardening (Modified Mohr-Coulomb). Generally it is determined by considering the diameter of the adjacent ground. But, in case there is no exact value, use the default value set in the program. You do not have to manually type in the thickness for the plane interface in 3D model like this tutorial.

When you define seepage rigidity for the interface elements, 'Seepage Flow' can be defined just like the ground permeability coefficient. But if you uncheck [Conduction for Seepage flow], it is assumed as impermeable layer.

Section 5
Analysis Setting

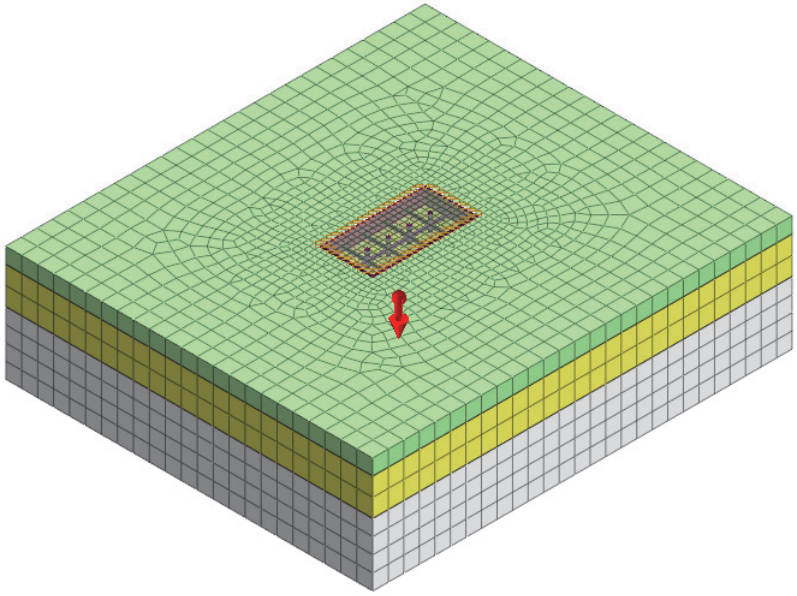
5.1 Setting Load Condition

* : Static/Slope 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.
- Type 'Self weight' at the [Load set] name. Click on [OK] button.

▶Setting self weight

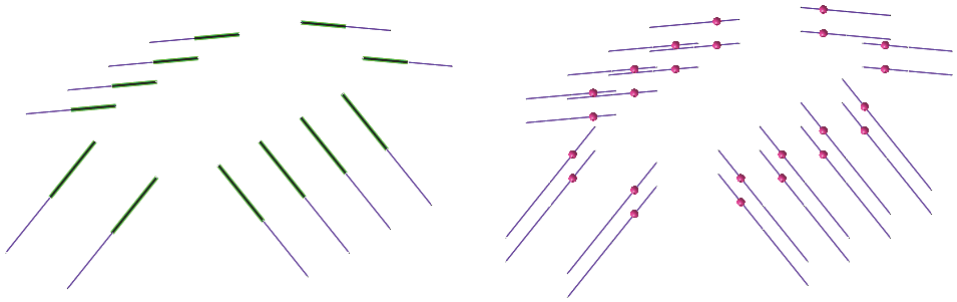


* : Static/Slope Analysis > Load > Prestress

This process sets pretension for anchor free face. It is possible to control the ground displacement by applying pretension (prestress) to Truss/Embedded Truss elements.

- Select element type 'Truss/Embedded Truss'.
- Select free face (12) of 'Stage3 anchor' as image below.
- Enter 200(KN) in load components.
- Change the Load set name to 'Stage3 anchor tension'.
- In the same way, set 200(kN) of prestress for the 'Stage4 anchor' free face element.

▶Setting preestress for anchor



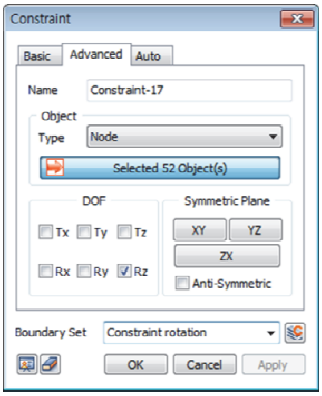
5.2 Setting Boundary Conditions

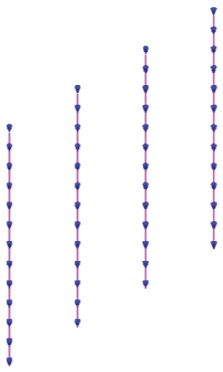
∗ : Static/Slope Analysis > Boundary > Constraint

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 of Rz direction in plus pegs to prevent the degree of freedom errors.

- Select [Auto] tab.
- Check [Consider All Mesh Sets] option. And enter boundary set name to 'Ground boundary'.
- Click on [Apply].
- Set all the mesh to show on the work window, and verify the generated boundaries on the screen.
- Select [Advanced] tab.
- Select the 'Node' type. Select all the nodes of generated plus peg elements and check 'Rz'.
- Name the boundary set to 'Constraint rotation'. Click on [OK] button.

▶Setting constraint pile rotation





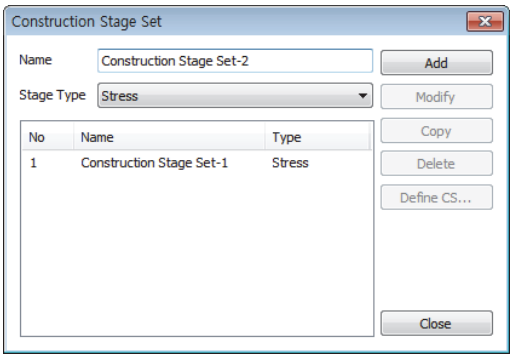
5.3 Define Construction Stages

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

This process sets construction stages to verify the results in each stage: foundation under initial stress status, excavation (or banking), support structure installations, loading etc. Mesh sets should be separated beforehand according to the construction stage.

- Set the [Stage type] to 'Stress'.
- Click on [Add] to create construction stage set.
- Click on [Define Construction stage] to define construction stages.
- The construction stage should be defined as below.

▶Setting construction stage



Stage 1 - Name : Foundation

- Activated Data-Mesh : [Stage1~5 excavation], [Buried layer], [Colluvium], [Weathering soil], [Rigid link mesh]
- Activated Data-Boundary Condition : [Ground boundary]
- Activated Data-Static Load : [Self weight]
- Check [Clear Displacement] option.
- Save and click [New] to define next stage.

Stage 2 – Name : Install Walling and Plus peg

- Activated Data-Mesh : [Sheet Pile], [Plus peg], [Plane interface]
- Activated Data-Boundary Condition : [Constraint rotation]
- Deactivated Data-Mesh : [Rigid link mesh]
- Save and click [New] to define next stage.

Stage 3 – Name : Stage1 excavation and install Stage 1 strut

- Activated Data-Mesh : [Stage1 walling], [Stage1 strut]
- Deactivated Data-Mesh: [Stage1 excavation]
- Save and click [New] to define next stage.

Stage 4 - Name : Stage2 excavation and install Stage 2 strut

- Activated Data-Mesh : [Stage2 walling], [Stage2 strut]
- Deactivated Data-Mesh: [Stage2 excavation]
- Save and click [New] to define next stage.

Stage 5 – Name : Stage3 excavation and install Stage 3 anchor

- Activated Data-Mesh : [Stage3 walling], [Stage3 anchor]
- Activated Data-Static Load : [Stage3 anchor tension]
- Deactivated Data-Mesh: [Stage3 excavation]
- Save and click [New] to define next stage.

Stage 6 – Name : Stage4 excavation and install Stage 4 anchor

- Activated Data-Mesh : [Stage4 walling], [Stage4 anchor]
- Activated Data-Static Load : [Stage4 anchor tension]
- Deactivated Data-Mesh : [Stage4 excavation]
- Save and click [New] to define next stage.

Stage 7 - Name: Final Excavation

- Deactivated Data-Mesh : [Stage5 excavation]
- Save and close.

5.4 Setting Analtsis Case

This process sets analysis method and model data for the analysis. The analysis and output types could be controlled using the advanced options. For construction stage analysis, because the data for the analysis has been formerly set, the [Analysis Case Model] is deactivated.

* : Analysis > Analysis Case > General

- Type in the name of the analysis case and select 'Construction Stage' as solution type.
- Set Analysis > General > Initial Stage > Initial Stage for Stress Analysis to '1:Foundation'. Check [Apply K0 Condition].
- Click on [OK].

5.5 Perform Analysis

Perform analysis and output the results. After the analysis, the software is automatically switched to [Post-Mode] (checking results). You can switch back to the [Pre-Mode].

* : Analysis > Analysis > Perform
    - Perform analysis

During the analysis, you can check the calculation process in real-time. Messages such as whether the results converge or not, warnings and errors can be checked through [Output Window].

Section 6
Results

After the analysis you can check the results such as displacements, stresses, member forces of each construction stage in the Result Tree. All the results can be displayed in the form of contour, table, and graph. The main result items which need to be checked in this tutorial are listed below.

- Sheet Pile – Horizontal displacement, bending stress, shear stress
- Vertical displacement at bottom part of excavation and ground solids
- Walling and Strut – Bending stress, shear stress
- Anchor – Maximum axial force
- Interface – Relative displacement and friction between wall and ground

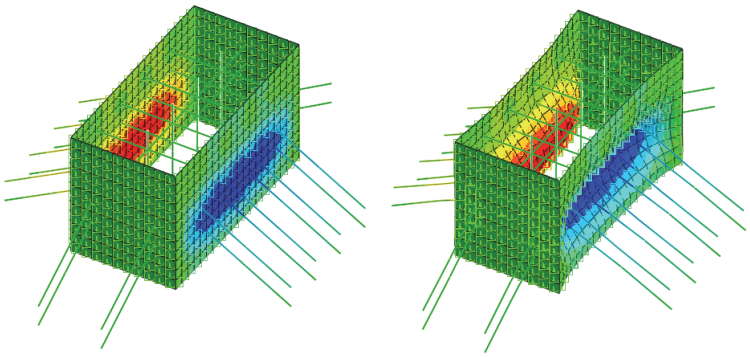
6.1 Verify Displacement

TX, TY, TZ represent displacements in X, Y, Z directions. Horizontal displacement and settlement tendency according to banking and surface loading can be verified in TX, TZ. '(V)' refers to the result items which can be represented by both contour and vector at the same time. In GTS NX, it is possible to show contour/vector simultaneously for displacements and principal stresses.

Check the result at the last stage of 'Final excavation'.

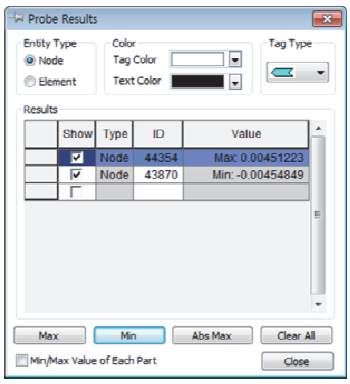
- Select the last stage in the Result Tree. And select Displacement > TX TRANSLATION (V).
- From Result > General > Deform, you can directly see the deformation in X direction.
  (Scale of deformation shape can be set in the property window. you can see the difference by checking [Actual Deformation] in the Result > Show/Hide.)

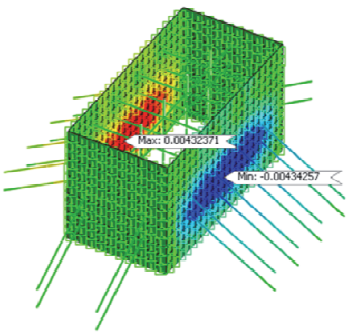
▶Horizontal displacement (unreformed)
▶▶Horizontal displacement (deformed)



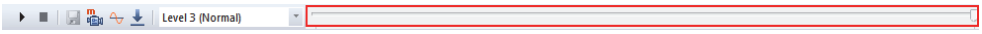
- It is possible to see the values of specific elements or nodes using Result > Advanced > Probe. You can also locate the max/ min/ abs max values on the model.

▶Probe





- By moving the simulation bar at the bottom of the work window, it is possible to simulate the results changing during the whole process of construction stages.



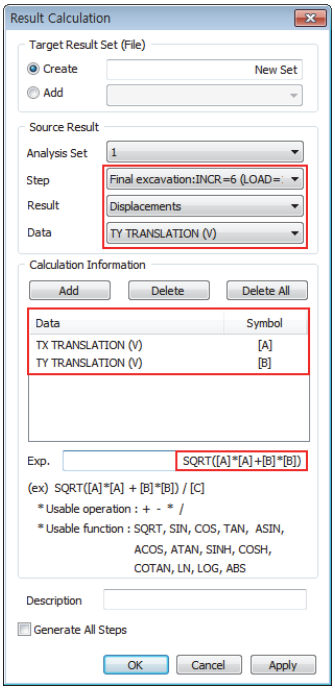
Add/Create the result items that you want to see.

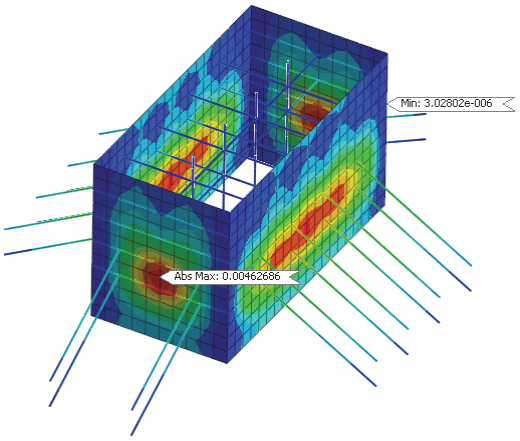
- By creating formulas in Result > Result > Calculation, you can make up any result item you want. Choose XY to see the deformations in two directions and combine the results to make a new result item.
- Select [Step: final excavation], [Result type: Displacement]. Select data of TX and click on [Add]. Add data of TY to the list too. Then items [A] and [B] will be generated.
- To create the TXY displacement result item, enter the formula below at the [Expression].

SQRT([A]*[A]+[B]*[B])

- Click on [OK] to add the new result item according to the formula. You can see the results in the form of contour, graph, table etc.

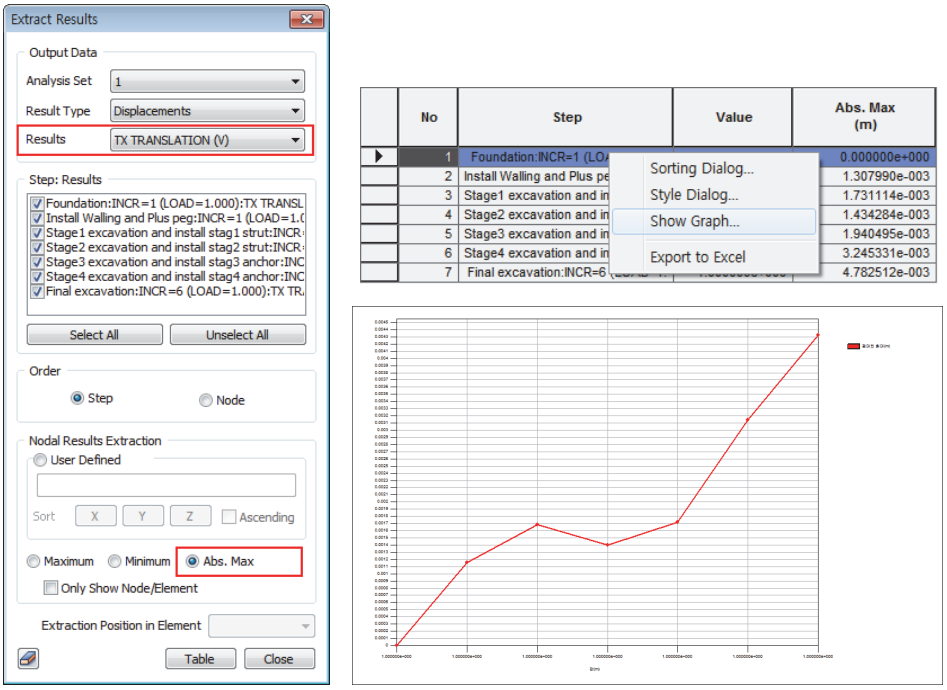
▶Generate XY direction displacement item
▶▶XY direction displacement





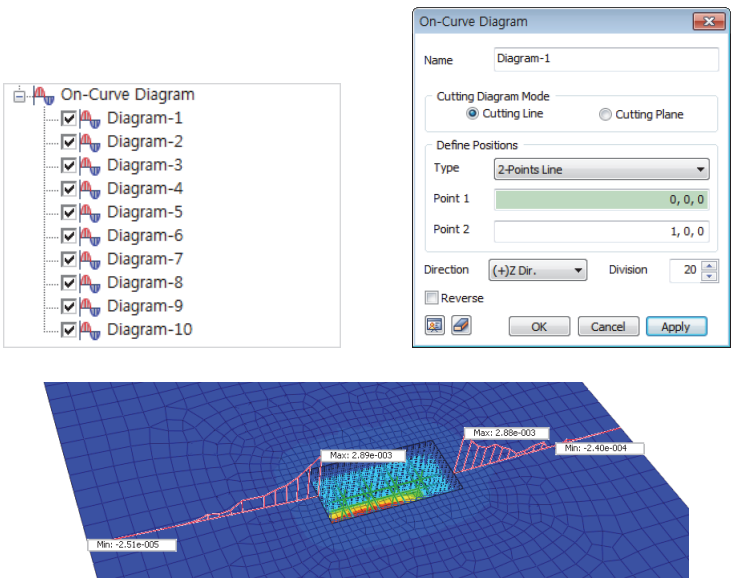
- Use [Extract] function to export maximum displacement results in each construction stage. By right clicking the mouse, you can plot graph based on data selected in the table.

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



- In the same way verify the settlement of the excavation part by selecting Displacement > TZ TRANSLATION(V). To draw the diagram directly on the model use Result > Advance > Cutting Diagram. Set line/point/face to plot the diagram as the image below. Choose the result item, and the result diagram will automatically renew. Added diagram will be registered in the Result Tree. Use checkbox to Show/Hide each diagram.

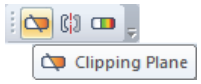
▶Generate On-Curve diagram
▶▶Extract settlement diagram



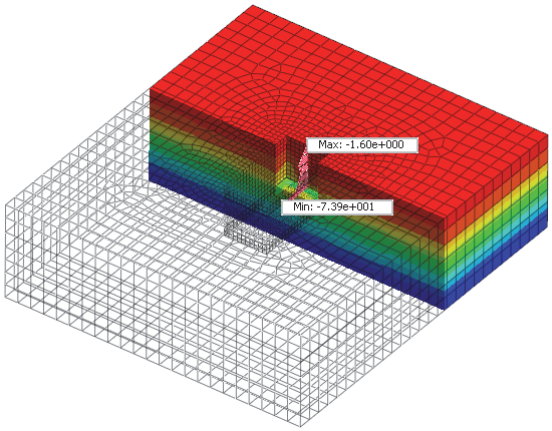
6.2 Verify Stresses

You can see the ground stresses in the 'Solid Stresses' of Result Tree. S-XX, S-YY, S-ZZ represent stresses in X, Y and Z directions. Using [On-Curve Diagram] it is possible to plot stress distribution on the cutting line.

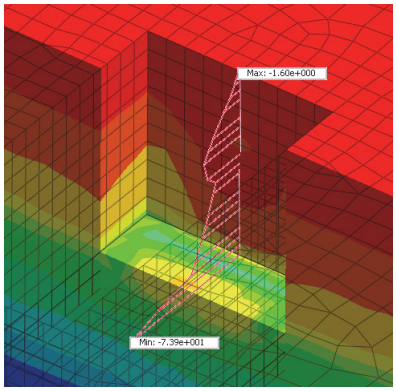
- Select Solid Stresses > S-YY of final excavation stage from the Result Tree.
- You can see the interior distribution of the ground stresses by using Advanced View Control Toolbar > Clipping Plane.



▶Ground horizontal stresses SYY



▶▶Diagram of SYY about depth



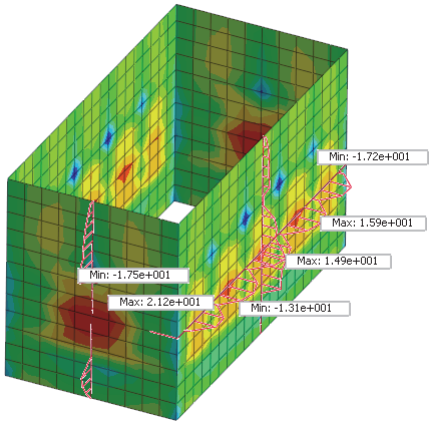
Check member forces/stresses on each structure and sheet pile. Sheet pile can be verified in 'Forces/Stresses' and 'Beam, Truss Element Forces/Stresses' for 1D member. Each result for structure member is plotted based on element coordinate system as by default. If you need to change it, change the coordinate system when you define material/property or in the [Output Control] when you create the analysis case.

Verify the results on sheet pile of final excavation stage.

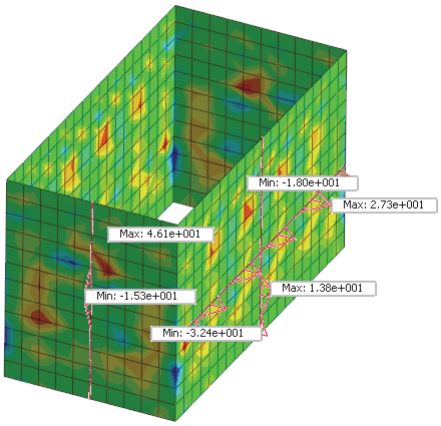
- Verify the moment of the sheet pile by selecting Shell Element Forces > BENDING MOMENT XX in the final excavation stage.
- After that, verify maximum shear force in TRANSVERSE SHEAR FORCE XZ.
- If you select Result > General > No Results > Exclude, you can hide all the other structures and display only the structure member which you are checking (sheet pile) in the work window.

You can see that most maximum member forces concentrate around connections with other structures such as walling, struts, anchors etc.

▶Sheet Pile moment diagram

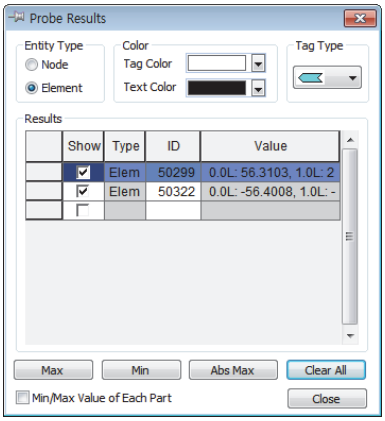


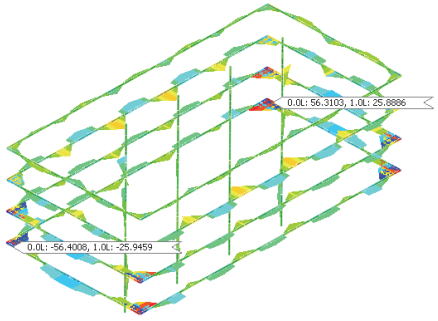
▶▶Sheet Pile shear force diagram



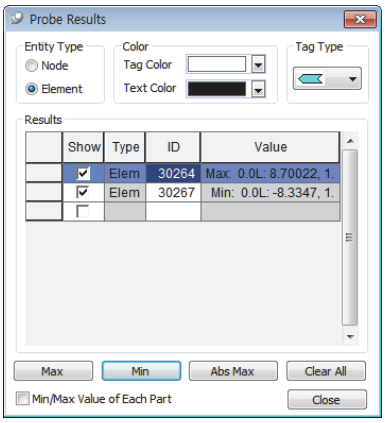
- Check maximum moment of walling, plus pegs, struts by selecting Beam Element Forces > BENDING MOMENT Y of the final excavation stage in the Result Tree. It is possible to plot result of each member that you want to see by Show/Hide check box in the Model Tree.

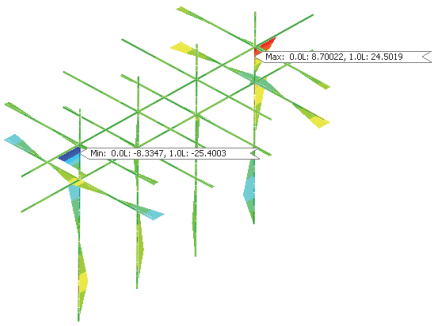
▶Walling moment





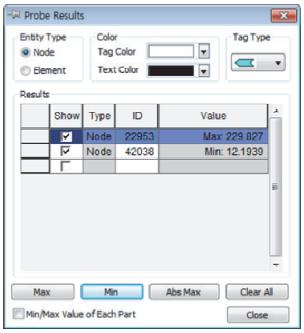
▶▶Plus pile, strut moment

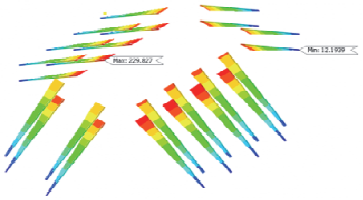




- Verify axial force on anchors by checking Truss Element Forces > Axial Force of final excavation stage in the Result Tree.

▶Anchor axial force(Consider pretension force)



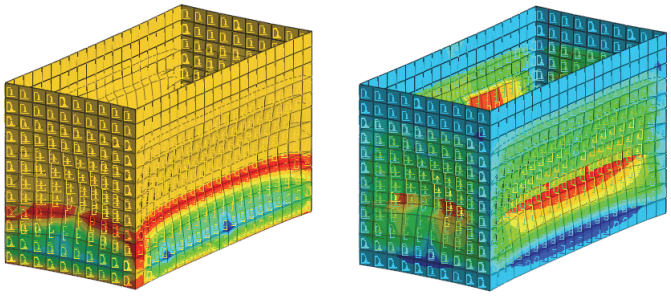


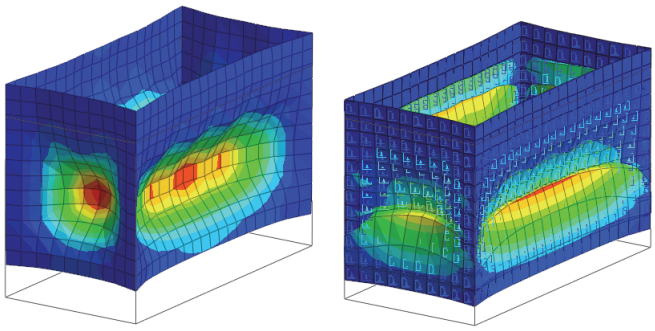
6.3 Verify Friction/Relative Displacement of Wall Interface

Sheet pile and ground have relatively large difference in rigidity. To simulate the separation behavior of these two, interface elements were applied. You can verify stress and relative displacements in normal direction and two tangential directions on the wall interface.

- Verify the friction between wall and ground by selecting Interface Stresses > TANGENTIAL Y of the final excavation stage in the Result Tree. As the excavation progresses, you can see the generation of large friction at the bottom of the wall.
- Click Interface Relative Displacement > PLASTIC TANGENTIAL Y, and verify the relative displacement between ground and wall.

▶Interface friction
▶▶Relative displacement



- Compare the sheet pile wall displacement with interface total displacement by selecting Displacements > TOTAL TRANSLATION (V) of the final excavation stage.

▶Sheet Pile total displacement
▶▶Interface total displacement