Chapter 8
2D Tunnel Construction Stage

Workflow

	: Main Menu > Import > DXF 2D (Wireframe)...	7
	: Geometry > Point & Curve > Intersect	7
	: Geometry > Remove > Face/Edge	7
	: Mesh > Control > Size Ctrl	8
	: Mesh > Generate > 2D	8
	: Mesh > Element > Parameters	9
	: Mesh > Element > Extract	10
	: Mesh > Element > Parameters	11
	: Static/Slope Analysis > Load > Self Weight	12
	: Static/Slope Analysis > Boundary > Constraint	12
	: Static/Slope Analysis > Boundary > Change Property	13
	: Static/Slope Analysis > Construction Stage > Stage Set	14
	: Analysis > Analysis Case > General	17
	: Analysis > Analysis > Perform	17

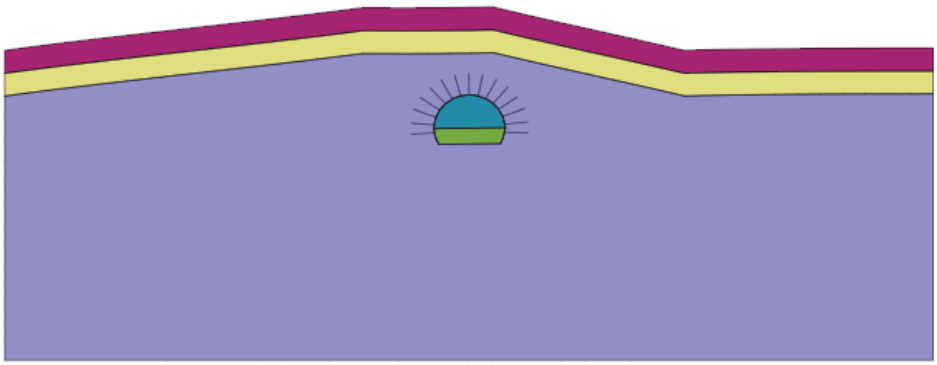
2D Tunnel Construction Stage

Section 1
Overview

1.1 Learning Purpose

One of the major objectives of the tunnel analysis is to figure out the stability of the given design and the adequacy of the construction process. By processing numerical analysis, you can distinguish the displacement and subsidence which arise around the tunnel before tunnel construction. Also you can comprehensively analyze stress of shotcrete and rockbolts to predict in advance problems during construction.

Ground characteristic values, ground initial stresses and support characteristic values etc. are the input data for the analysis. Since the initial stresses have a major influence on the tunnel behavior, it is necessary to process stability evaluation of the tunnel by changing parameters (Coefficient of lateral earth pressure K0).



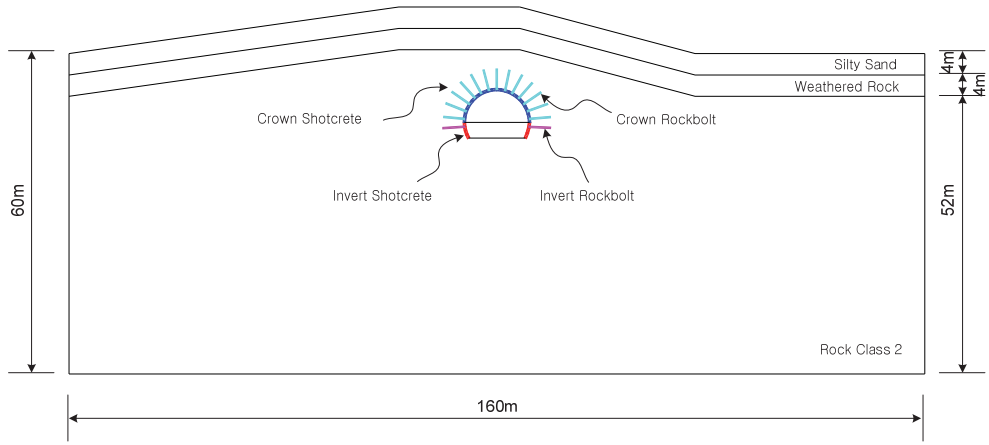
In this tutorial, the following main concepts will be explained:

- Import geometry shapes
- Define ground/structure materials
- Generate meshes
- Modeling primary tunnel supporting structures (shotcrete and rockbolt)
- Change parameters (before analysis, during analysis)
- Define construction stage(check LDF)
- Parametric analysis
- Results - plotting graphs and tables of the construction stage results

1.2 Model and Analysis Summary

This model represents a cross-sectional shape of the mountain terrain with a tunnel inside. The ground consists of 4m of silty sand layer on the top. Beneath the silty sand layer, there are weathered rock layer and rock class 2. After the excavation of the tunnel body, the installation of primary supports (shotcrete and rockbolt) is performed. And at last the stage of shotcrete hardening is performed.

▶Analysis cross-section



Section 2

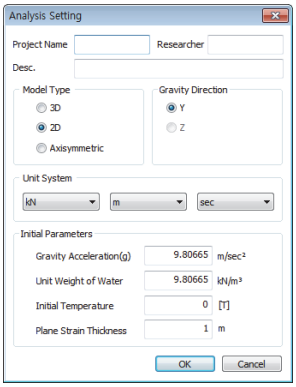
Analysis Setting

[Open the attached start file (08_1arch_start)]

∗ : Analysis > Analysis Case > Setting

- Set Model type, Gravity direction and Initial parameters. Check the Unit system which will apply to the analysis. The unit system can be changed both during the modeling process and after performing analysis. The input parameters will automatically converted to the right unit system.
- This tutorial is a 2D model and uses SI unit system (KN, m, s).

▶Analysis setting



Tip

Through Unit converter located in the bottom right of the monitor, Unit can be easily change and apply to the analysis. 2D model uses XY plane as a work plane. For 3D model, the gravity direction can be set as Y or Z.

Section 3
Define Materials and Properties

3.1 Define Ground and Structure Materials

For material modal type, apply 'Mohr-Coulomb' for the ground and 'Elastic' for the structure, which doesn't consider Nonlinearity. Define 4 types of material, according to the different Ko values of the Rock classes

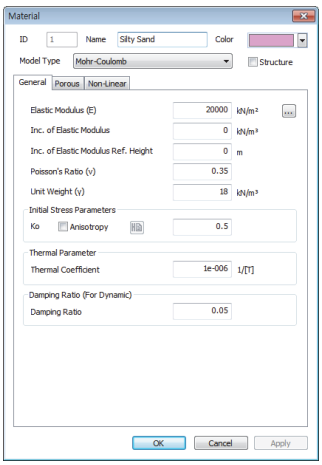
Materials for ground and structure are shown in the following tables.

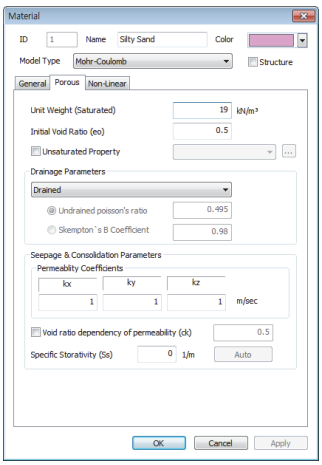
▶Table. Ground material

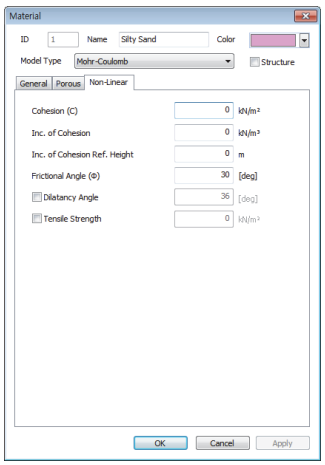
[Unit : KN, m]

Name	Silty sand	Weathered rock	Rock Class 2-1	Rock Class 2-2	Rock Class 2-3	Rock Class 2-4
Material	Isotropic	Isotropic	Isotropic	Isotropic	Isotropic	Isotropic
Model Type	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb
General						
Elastic Modulus(E)	2.0E+04	2.0E+05	1.0E+07	1.0E+07	1.0E+07	1.0E+07
Poisson's Ratio(v)	0.35	0.33	0.22	0.22	0.22	0.22
Unit Weight(r)	18	21	26	26	26	26
Ko	0.5	0.5	0.5	1.0	1.5	2.0
Porous						
Unit Weight (Saturated)	19	22	27	27	27	27
Initial Void Ratio(eo)	0.5	0.5	0.5	0.5	0.5	0.5
Drainage Parameters	Drained	Drained	Drained	Drained	Drained	Drained
Non-Linear						
Cohesion(C)	0	30	3000	3000	3000	3000
Frictional Angle	30	33	43	43	43	43

▶Define ground material -General
▶▶Define ground material-Porous
▶▶▶Define ground





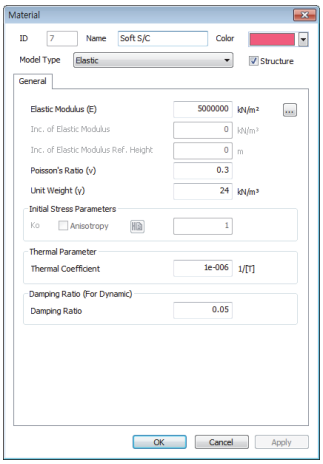


[Unit : KN, m]

▶Table. Structure material

Name	Soft S/C	Hard S/C	Rockbolt
Material	Isotropic	Isotropic	Isotropic
Model Type	Elastic	Elastic	Elastic
Elastic Modulus(E)	0.5E+07	1.5E+07	2.10E+08
Poisson's Ratio(v)	0.3	0.3	0.3
Unit Weight(r)	24	24	78

▶Define structure material



Tip

Material properties can be imported from an existing analysis file and Excel file that is organized according to the suitable form.
Structure material can be defined by checking [Structure] which is next to model type. If this [Structure] is checked, water-related parameters will not be considered.

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 shotcrete which resists to axial/shear/moment and use truss element for rockbolt which only resists to axial forces.

Beam element is a 1dimensional line element defined by 2 or 3 nodes. Beam element can be used when the structure's length or height is significantly larger than the dimension of its cross-section which receives bending loads. If the ratio of cross-section width to length or height is over 1/5, the effect of shear deformation should not be neglected. In this case, instead of using beam element, it is advisable to use shell element or solid element.

Truss element is a 1 dimensional line element defined by 2 nodes. It is widely used to model structure members whose lengths are longer than their cross-sectional widths such as space trusses and diagonal braces. Mostly it is used to model structure element for which flexural behavior can be ignored. For example, anchor, nail, rockbolt, etc.

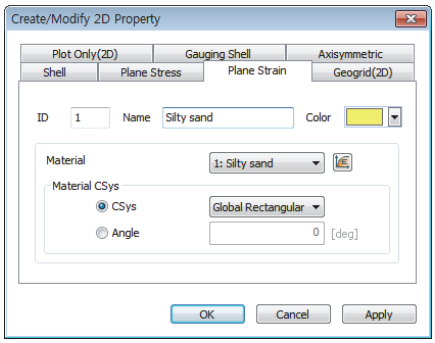
Ground properties are described in the following table

▶Table. Ground property

[Unit : KN, m]

Name	Silty sand	Weathered rock	Rock Class 2-1	Rock Class 2-2	Rock Class 2-3	Rock Class 2-4
Type	2D	2D	2D	2D	2D	2D
Sub-Type	Plane Strain	Plane Strain	Plane Strain	Plane Strain	Plane Strain	Plane Strain
Material	Silty sand	Weathered rock	Rock Class 2-1	Rock Class 2-2	Rock Class 2-3	Rock Class 2-4

▶Define ground property

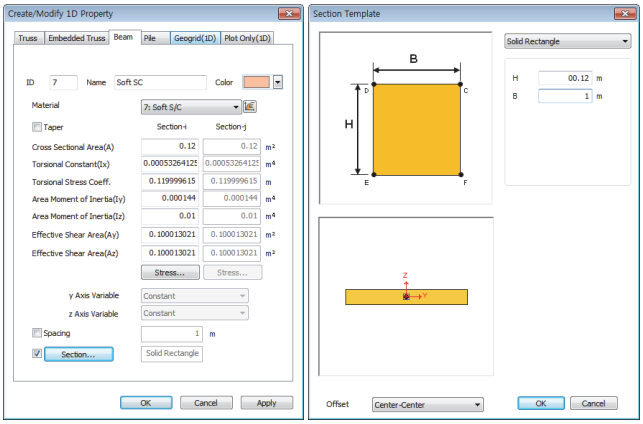


Structure properties are described in the following table. The cross-section stiffness will be automatically calculated based on the cross-section shape.

[Unit : KN, m]

Name	Soft S/C	Hard S/C	Rockbolt
Type	1D	1D	1D
Model Type	Beam	Beam	Truss
Material	Soft S/C	Hard S/C	Rockbolt
Spacing	-	-	1.5
Section	Rectangle	Rectangle	Solid Round
Section Size	H=0.12, B=1	H=0.12, B=1	D=0.025

▶Define property



Section 4
Modeling

Since the purpose of this tutorial is to study 2D 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 Modeling Geometry

∗ : Main Menu > Import > DXF 2D (Wireframe)...

DFX file of AutoCAD over version R13 can be imported.

- Import 08_1arch.dxf.

∗ : Geometry > Point & Curve > Intersect

This function intersects selected edges at cross positions.

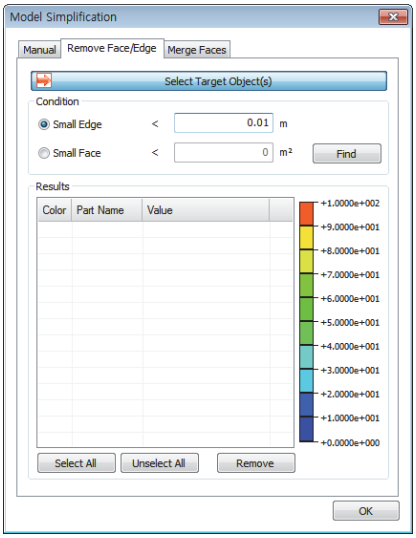
- Select all the lines and click on [OK].

∗ : Geometry > Remove > Face/Edge

After breaking the intersecting lines, check if there are small edges.

- Select all the lines, and type 0.01 for the [Small Edge] value.
- Click [Find] button. Lines smaller than 0.01m will be shown on the [Results] window.
- Check the locations of the lines. Click on [Remove] to delete the unnecessary ones.

▶Model simplification-Face/edge



Tip

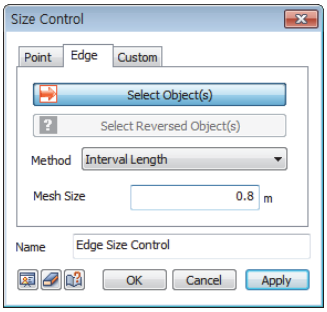
Extra nodes can be generated on the spots of unnecessary small edges and faces. These small nodes can cause abnormal displacement and stress concentration. Therefore small edge and face must be removed.

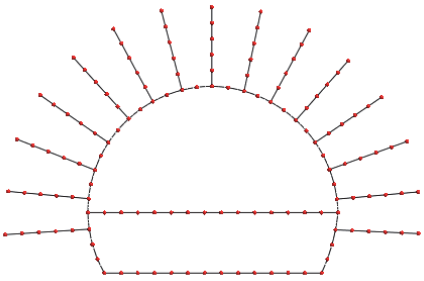
4.2 Generate mesh

* : Mesh > Control > Size Ctrl

Assign mesh size to points and edges.

- Select all the edges of the tunnel.
- Type 0.8 for the mesh size and click on [Apply].





Tip

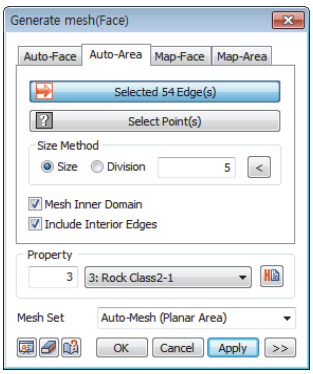
Mesh shape and quality is very important for finite element analysis. In case of ground where plastic failure occurs because of shearing strain, it is recommended to use hexahedral-based mesh for 3D and tetrahedral-based mesh for 2D to get a reliable analysis result. These types of meshes can be generated by using [Default Tetra Mesher] or hexahedral-based [Hybrid Mesher].
Generally speaking smaller the mesh sizes are, better the analysis quality is. 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. One of the tricks is to generate denser meshes around locations where you want to see the result.
To view the specified edge seed for mesh size control, check Work Tree > Mesh Control > Size Control.

* : Mesh > Generate > 2D

Generate 2D Mesh to the specified planar area which is defined by boundary lines.

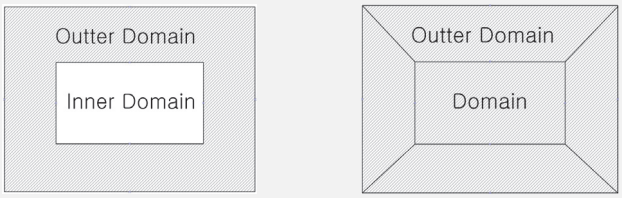
- Choose [Auto-Area] tap.
- Select all the edges (54 edges)
- Set the mesh size as 5.
- Check the [Mesh Inner Domain] option.
- Select Rock Class2-1 as the mesh property.

▶Generate mesh(Auto-Area)



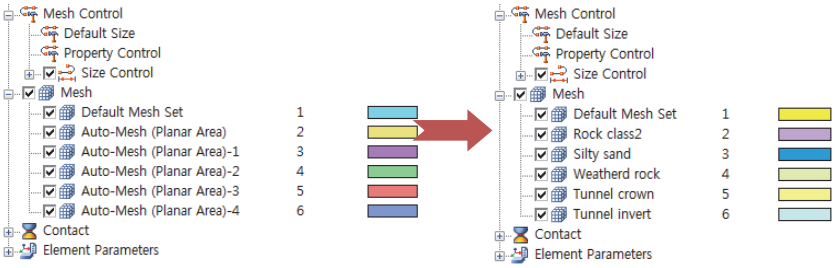
Tip

If the specified planar area is divided into 2 sections (interior and external), meshes can be generated in both section by checking [Mesh Inner Domain] option.
Interior domain is an area which exists separately at the inner part of the whole area without connection to the outer boundary. For example, if a small rectangle is located inside of a larger rectangle, the small rectangle is an inner domain. But in case of the small rectangle is connected to the outer rectangle with lines, the small rectangle is no longer an inner domain.



(a) Inner domain exists (b) No inner domain

- To change name of a mesh set, select the mesh set from the Work Tree > Mesh, press [F2] on keyboard to change the name.

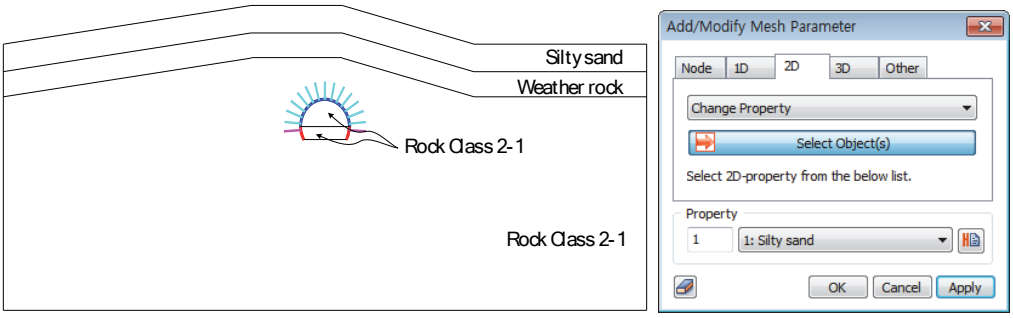


* : Mesh > Element > Parameters

Change the property of meshes which are already generated.

- Select 2D tap.
- Select the mesh sets and property that you want to change to. Click on [Apply] button. Refer to the figure below for the property information. (it is more convenient to select the mesh sets from the work tree)
- Property information can be directly confirmed on the screen by activating Display mode (Mesh) > Property Color.

▶Change property



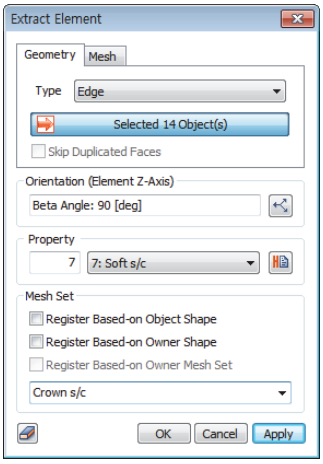
* : Mesh > Element > Extract

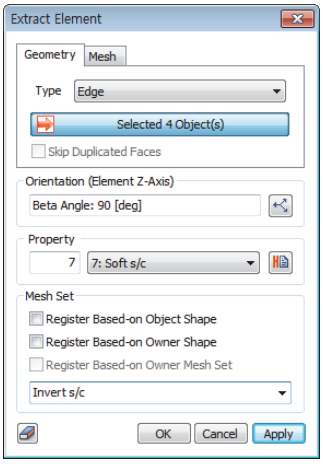
Primary supports (shotcrete and rockbolt) are going to be generated in this step. Use beam element for shotcrete which resists the axial/shear/moment forces, and truss element for rock bolt which only resists the axial force. Use [Extract] function, because both beam and truss must share the nodes with the ground.

- Select the edges of the crown shotcrete. (refer to analysis cross-section figure on page 2)
- Assign the property as soft s/c.
- Type 'Crown s/c' as a mesh set name.
- With the same method, generate the mesh of 'Invert s/c', 'Crown rockbolt', 'Invert rockbolt'.

▶Generate crown shotcrete

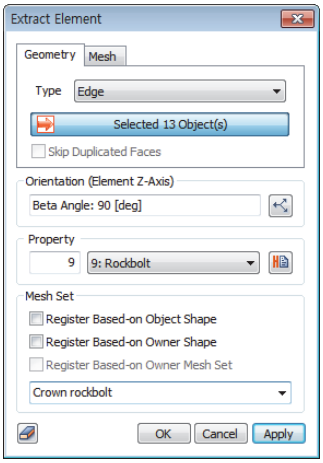
▶▶Generate invert shotcrete

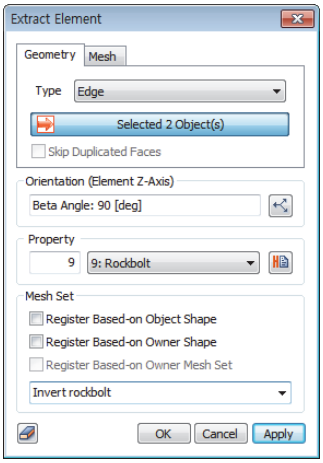




▶Generate crown rockbolt

▶▶Generate invert rockbolt



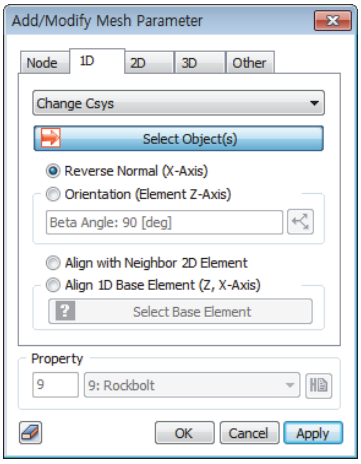


✱ : Mesh > Element > Parameters

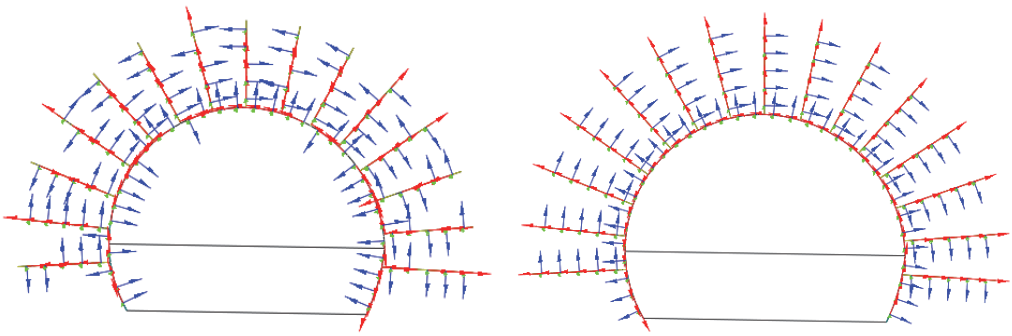
This step sets element coordinate systems. The member force of a Beam/Truss element is plotted based on its element coordinate system. Since the coordinate system of an element is determined by the order in which this element is generated, the coordinate system of the elements must be oriented correctly in order to have a consistent sign of the member force.

- Select 'Crown s/c', 'Invert s/c', 'Crown rockbolt', 'Invert rockbolt' from the Work Tree > Mesh.
- Mouse right click and select Display > Element CSys from the context menu
- Choose [Change CSys] from the dropdown Menu of the Mesh > Element > Parameters > 1D tap.
- Use [Reverse Normal (X-Axis)], to align elements into one direction.

▶Mesh parameters – Change coordinate system



▶Before the alignment
▶▶After the alignment



Section 5
Analysis setting

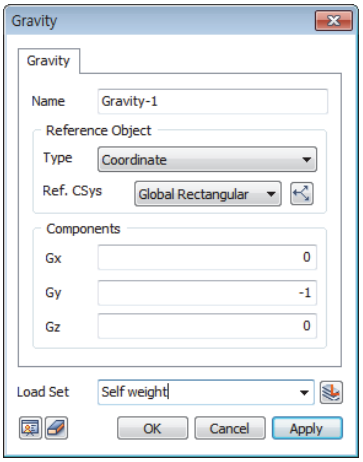
5.1 Define Loading Conditions

∗ : Static/Slope Analysis > Load > Self Weight

The self weight is calculated automatically by multiplying the unit weight of the ground, the acceleration of gravity and the scale factor in the appropriate direction to be defined in the Self Weight dialog box.

- Input 'Gravity-1' for [Name] and 'self weight' for [Load Set]
- For the [Components] put '-1' for the scale factor at Gy.
- Select [Apply].

▶Setting gravity



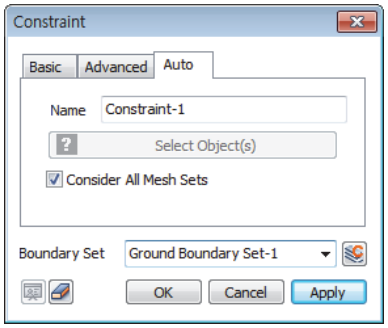
5.2 Setting Boundary Condition

∗ : Static/Slope Analysis > Boundary > Constraint

This process assigns boundary conditions to the analysis model based on global coordinate system.

- Under the [Auto] tab, give a name to the constraint, select all mesh sets and choose the target [Boundary set].

▶Setting boundary condition



Tip

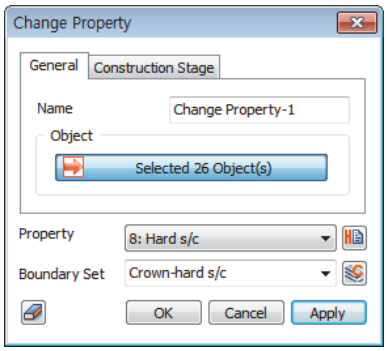
In 2D analysis, we constrain horizontal displacement of the left and right borders, and constrain both horizontal and vertical displacements of the bottom. GTS NX automatically detects the boundaries of the model and generates boundary conditions.

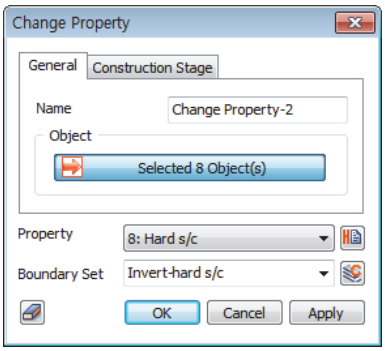
∗ : Static/Slope Analysis > Boundary > Change Property

According to the construction stage, stiffness of the shotcrete changes from soft to hard. To realize this change, use [Change Property].

- Type 'Change Property-1' at [Name] and 'Crown-hard s/c' at [Boundary Set].
- Select 'Crown s/c' as object.
- Since the Property will change to 'Hard shotcrete', select the 'Hard s/c' at the [Property].
- Click [Apply].
- Generate the boundary condition of the 'Invert-hard s/c' in the same way.

▶Setting stiffen condition for crown shotcrete
▶▶Setting stiffen condition for invert Shotcrete

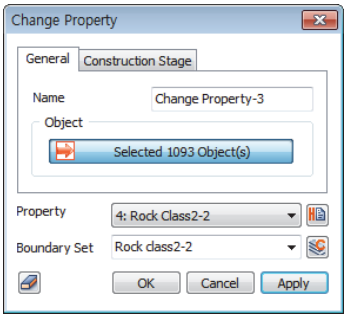


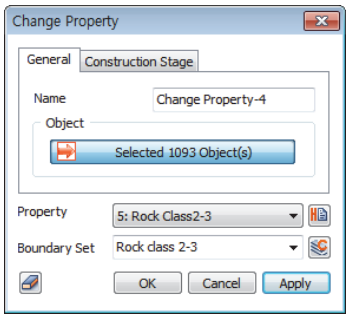


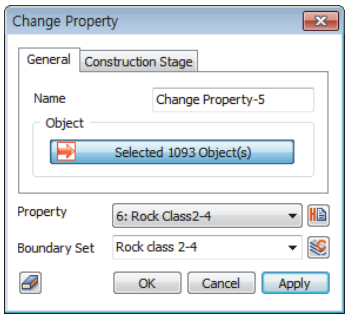
Use [Change Property] to define Rock Class2 with different K0 values for result comparison.

- Type 'Change Property-3' at [name], and put 'Rock class 2-2' at [Boundary Set].
- Select 'Rock Class 2', 'Tunnel crown', 'Tunnel invert' as object.
- Select the [Property] to Rock class 2-2.
- Click [Apply].
- Generate boundary conditions of 'Rock class2-3', 'Rock class2-4' in the same way.

▶Changing ground property(K0=1.0)
▶▶Changing ground property(K0=1.5)
▶▶▶Change ground







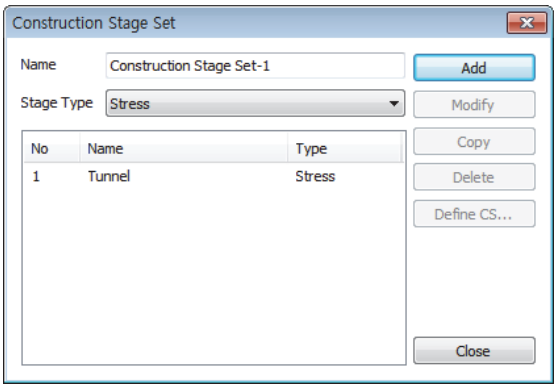
5.3 Define Construction Stages

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

In this process we define construction stages to check analysis results in each step. To define construction stages, mesh sets should be pre-divided. And then loads and boundary conditions are assigned to relevant mesh sets in each step. Starting from the original ground stress condition, each construction stage such as tunnel excavation, supports installation (shotcrete and rock bolt), and supports hardening etc. can be defined using this function.

- Type 'Tunnel' at the [Name].
- Select [Stress] as stage type and click on [Apply].

▶Generating construction stage set



- Generate construction stage by clicking [Define construction Stage].
- Composition of the construction stage is listed below.

Stage 1 – Stage Name : Foundation

Activated Data-Mesh:[Silty sand], [Tunnel crown], [Tunnel invert], [Weathered rock], [Rock class2]
Activated Data-Boundary Condition : [Ground boundary set-1]
Activated Data-Static load : [Self weight]

Stage 2 – Stage Name : Null stage

Check [Clear Displacement].
In general construction stage, the stress resulted from self weight is initial stress. In GTS NX, the initial stress can be calculated by both K0 method and self-weight analysis method.

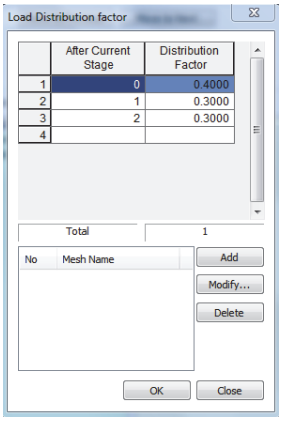
- K0 Method
- The K_0 method uses the constant K_0 to calculate the horizontal stress from the vertical stress and sets it as the in-situ stress. To use this method, the vertical stress σ_v needs to be calculated first using self-weight and this value is further used to calculate the horizontal stress through $\sigma_h = K_0\sigma_v$. In this case, the shear stress is assumed to be zero. If the ground surface is not flat, the calculated stress state and self-weight is not in equilibrium. So when the ground surface is not flat and K0 method is used to calculate initial stress, the analysis must be performed by using unbalanced force between resisting force according to the self weight and initial stress condition in order to make equilibrium condition. This can be performed by setting a [NULL Stage] in which none of the conditions is changed.

- Self-weight analysis method
  If the ground surface is flat, this method is the same as the K0=υ /(1- υ) of K0 method. But if it's not, the results come out differently with the K0 method because the horizontal displacement and the shear stress occur. Therefore, generally it is

recommended to use self-weight analysis method if the ground surface is not flat. In this case the K0 value can't be set over 1, you must use K0 method to use K0 value over 1.

Stage 3 – Stage Name : Crown Excavation

Deactivated Data-Mesh: [Tunnel crown]
To prevent all the unbalanced forces occurring from the excavation and from loading at the same time, use LDF (Load Distribution Factor) to split the force.
Check [LDF], enter [Distribution Factor] as 0.4, 0.3, 0.3.



Tip

During the tunnel excavation, a 3-dimensional structure is formed around the tunnel shield. Therefore, a load transfer phenomenon (Arching Effect) toward longitudinal and transverse directions occurs. This Arching effect occurring in two or more planes will scatter the ground load in a three-dimensional way. Displacement also occurs around the tunnel and the entire shield. To consider this effect in numerical analysis, 3-dimensional finite-element analysis must be directly performed. However, from the results of recent documents and researches, it has been demonstrated that the effect of 3-dimensional tunnel process can be reflected in 2 dimensions. Therefore, the general application of shield process and 3 dimensional ground behavior generated by supports installation are modeled by setting LDF (Load Distribution Factor).
In construction stage, LDF postpones the tunnel excavation effect by dividing the load and applying part of the load into the stages after the excavation stage. The load in each distribution stage is defined by a factor ratio from 0 to 1. And the sum of ratios in all stages must be 1. The LDF should not overlap each other.

Stage 4 – Stage name : Crown Support

Activated Data-Mesh: [Crown s/c], [Crown rockbolt]

Stage 5 – Stage Name ; Harden Crown Support

Activated Data-Boundary Condition: [Crown-hard s/c]

Stage 6 – Stage Name : Invert Excavation

Check [LDF], enter [Distribution Factor] to 0.4, 0.3, 0.3.
Deactivated Data-Mesh : [Tunnel invert]

Stage 7 – Stage Name: Invert Support

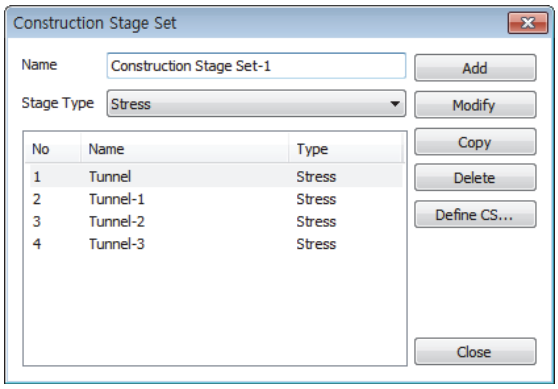
Activated Data-Mesh: [Invert s/c], [Invert rockbolt]

Stage 8 – Stage Name ; Harden invert support

Activated Data-Boundary Condition: [Invert-hard s/c]

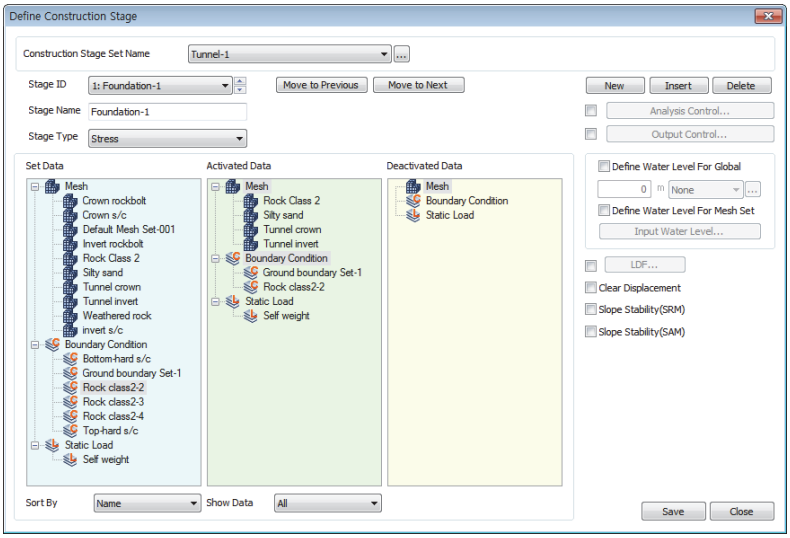
- To review the impact of the rock class transition and K0 transition, add k0=1.0, 1.5, 2.0 construction stage set.
- Select [Tunnel] and copy as shown below.

▶Generating construction stage.



- In the initial construction stage of [Tunnel-1], move [Rock class2-2] to [Activated data] to modify K0 value from 0.5 to 1.0.
- In the same way as the former step, move [Rock class2-3] and [Rock class 2-4] to [Activated data] in the initial construction stage of [Tunnel-2] and [Tunnel-3].

▶Define construction stage



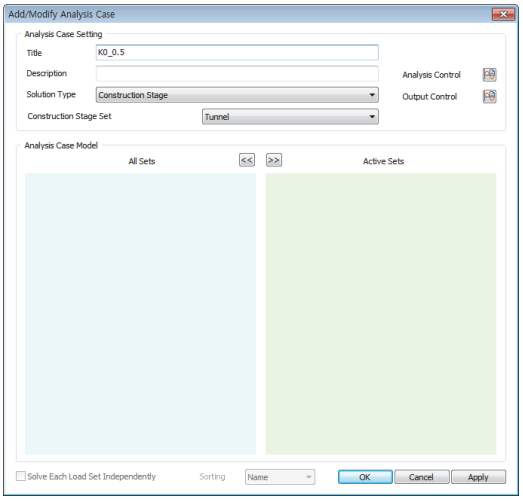
5.4 Define Analysis Case

∗ : Analysis > Analysis Case > General

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.

- Name title as 'K0_0.5'.
- Set the [Solution type] as 'Construction Stage' and [Construction Stage Set] as 'Tunnel'.
- In Analysis Control > General, check [Initial Stage for Stress Analysis] and select '1: Foundation'. Also check the [Apply K0 Condition].
- In the same way, generate analysis case for K0=1.0, 1.5 2.0. (Change [Construction Stage Set] to 'Tunnel-1', 'Tunnel-2', 'Tunnel-3', and set [initial stage for stress analysis] as 'Foundation'.)

▶Generating analysis case
▶▶Analysis control



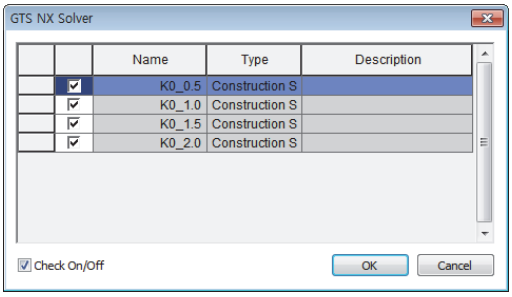
5.5 Perform Analysis

* : Analysis > Analysis > Perform

Perform analysis. If the analysis is done, it will automatically switch to 'Post-Mode (Confirmation of the Results)'. For the further modification of the model and options, you have to switch back to the 'Pre-Mode'.

- It is possible to select the analysis cases, and analyze only the selected cases.

▶Analysis perform manager



Tip

The information occurring from the analysis process is printed out in the [Output Window]. Especially if the warning occurs, be aware that the results may not be normal.
Before generating analysis the model is automatically saved. The information about analysis is

saved in *.OUT file as text file format in the same file with the saved model file.
When analyzing a model, the results can be controlled by Analysis > Analysis Case > General > Output Control option. And if you set the output option as [Binary and Text], the analysis results on nodes and elements are also plotted in *.OUT file.

.

Section 6
Result

After the analysis, results such as displacement, stress, axial member force of the each construction stage can be viewed by clicking on the relevant item on the result tree. All of the results can be shown as contour and table. The main result items for this model are shown below.

- Displacement of the Tunnel – Crown Displacement, Convergence
- Generated axial force of member and stress of Shotcrete
- Axial force of Rockbolt

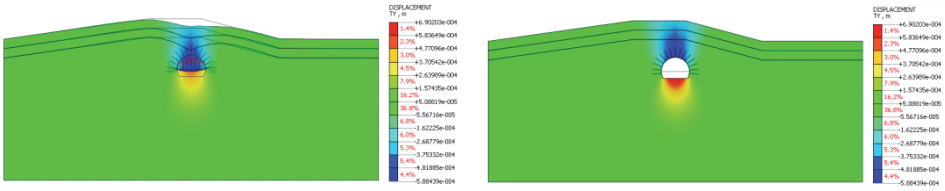
6.1 Verifying Displacement

The tendency of the displacement can be found in the result tree. TX, TY, TZ refers to X, Y, Z direction based on GCS. Therefore the crown displacement of the tunnel can be seen in [TY TRANSLATION], and the convergence can be seen in [TX TRANSLATION].
The item (V) refers to the result item to which both contour and vector are available. By using GTS NX, displacement and principal stress can be plotted as contour and vector.

- Select the stage (Harden invert support) to check the results in the Work tree > Results > K0_0.5, and Select Displacement > TY TRANSLATION (V).

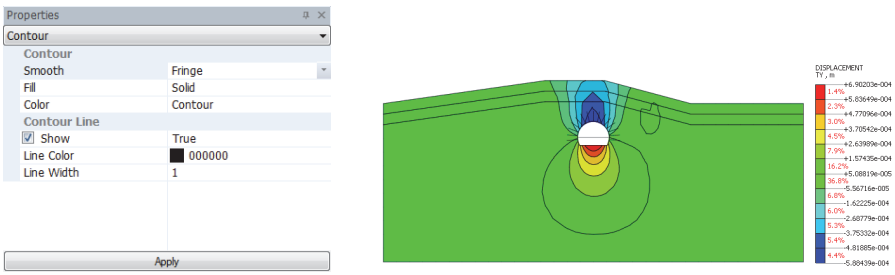
- Select Result > General > Deform > Undeformed.

▶Deformed
▶▶Undeformed



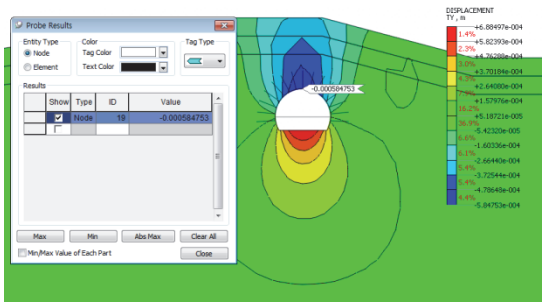
- Select Result > General > Smooth > Fringe. If you set the contour to [Fringe], it is easier to understand the tendency. The property of the contour can be changed by using the property window in the left bottom of the screen.

▶Property window > Contour
▶▶Fringe+Show contouor line

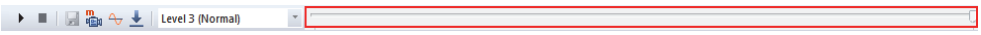


- Check the value of the selected node, by the Result > Advanced > Probe function.

▶Probe results



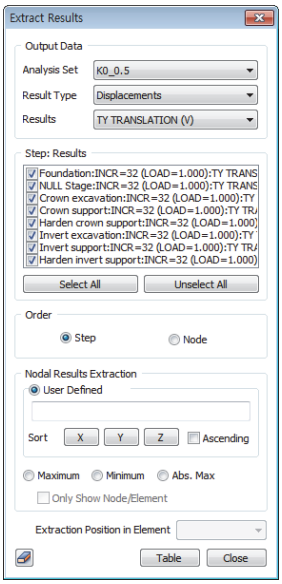
- The result units can be converted by the real-time Unit Converter lies in the right bottom of the screen.
- Scroll the movement bar at the bottom of the screen to see the result stage by stage

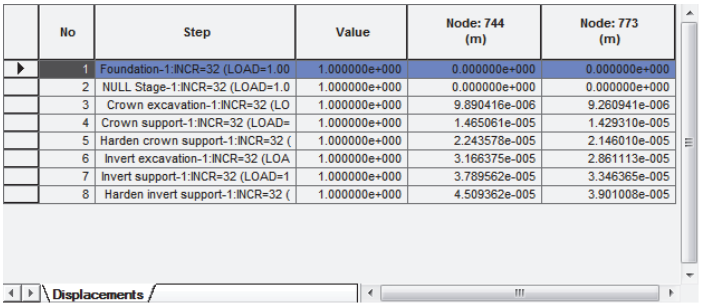


- Use Result > Advance > Extract to extract the result values of specific elements or nodes according to the construction stage. The results are plotted as table format. The table can be Copied / Pasted into the MS-EXCEL.

▶Extract results

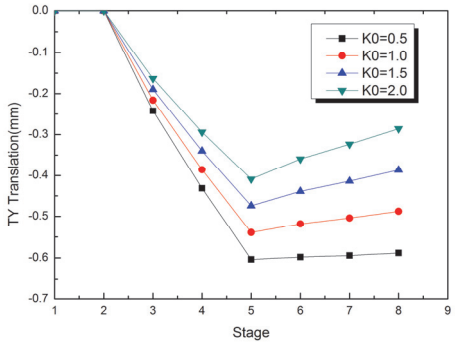
▶▶The result of the extraction

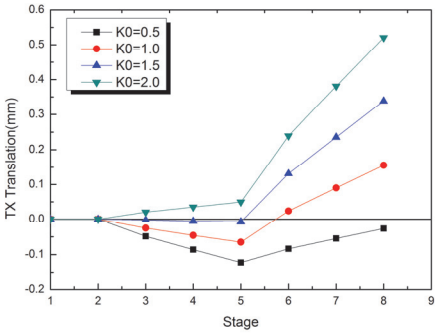




No	Step	Value	Node: 744 (m)	Node: 773 (m)
1	Foundation-1:INCR=32 (LOAD=1.00	1.000000e+000	0.000000e+000	0.000000e+000
2	NULL Stage-1:INCR=32 (LOAD=1.0	1.000000e+000	0.000000e+000	0.000000e+000
3	Crown excavation-1:INCR=32 (LO	1.000000e+000	9.890416e-006	9.260941e-006
4	Crown support-1:INCR=32 (LOAD=	1.000000e+000	1.465061e-005	1.429310e-005
5	Harden crown support-1:INCR=32 (	1.000000e+000	2.243578e-005	2.146010e-005
6	Invert excavation-1:INCR=32 (LOA	1.000000e+000	3.166375e-005	2.861113e-005
7	Invert support-1:INCR=32 (LOAD=1	1.000000e+000	3.789562e-005	3.346365e-005
8	Harden invert support-1:INCR=32 (	1.000000e+000	4.509362e-005	3.901008e-005

Displacements



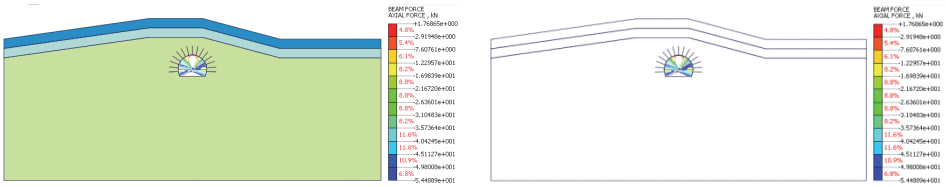


6.2 Member Axial Force and Shotcrete Stress

Shotcrete is modeled as a beam element. In case of Beam element, the axial force, shear force and moment can be plotted.

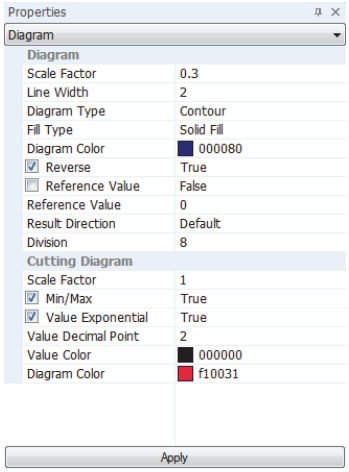
- Select the stage (Harden invert support) from the Work tree > Results > K0_0.5, and Select Beam Element Forces > AXIAL FORCE.
- Set Result > General > No Results > Exclude

▶AXIAL FORCE
▶▶No results > exclude



- Scale/Type/Fill Type/Division etc. can be assigned at Property Window > Diagram.

▶Property window(Diagram)
▶▶Solid fill+Reverse

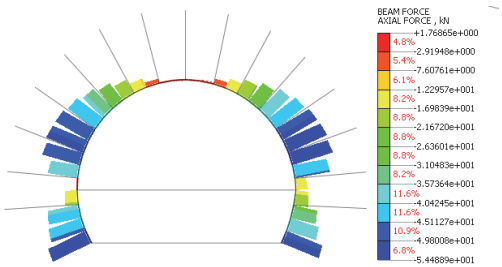


Properties

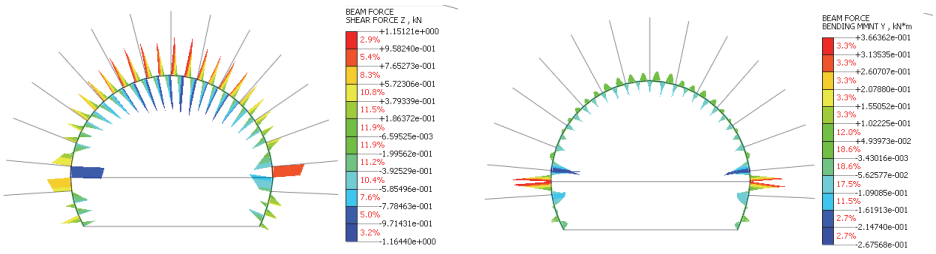
Diagram

Diagram	
Scale Factor	0.3
Line Width	2
Diagram Type	Contour
Fill Type	Solid Fill
Diagram Color	000080
☑ Reverse	True
☐ Reference Value	False
Reference Value	0
Result Direction	Default
Division	8
Cutting Diagram	
Scale Factor	1
☑ Min/Max	True
☑ Value Exponential	True
Value Decimal Point	2
Value Color	000000
Diagram Color	f10031

Apply



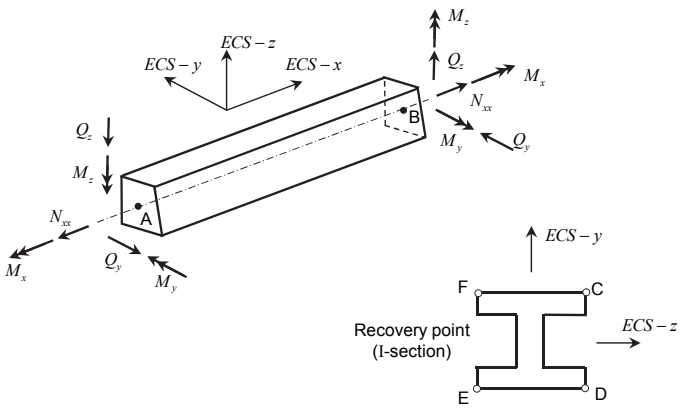
- Select the stage (Harden invert support) from the Work tree > Results > K0_0.5, and Select Beam Element Forces > SHEAR FORCE Z and BENDING MOMENT Y. check the sheer force and the moment.



The stress results from a beam element are listed in the following table.

	Results	Explanation
Stress	Axial stress	Position : Section A-B of each segment σ_{xx}
	Torsional stress	Position : Section A-B of each segment Calculate from Torsion stress coefficient c ($\tau = Tc / J$)
	Shear stress	Position : Section A-B of each segment Calculate from shear modulus of cross-section area $\tau_{xy} = Q_y / (S_{ky} A)$, $\tau_{xz} = Q_z / (S_{kz} A)$
	Point stress	Position : Section A-B of each segment Bending Stress induced by user defined location σ_{xx}
	Max/Min stress	Position : Section A-B of each segment Max/Min of the sum of Axial stress and Point stress among location C~F
	Von-mises stress	Position : Section A-B of each segment $\sigma_v = \sqrt{\sigma_{xx}^2 + 3(\tau_{xy}^2 + \tau_{xz}^2)}$

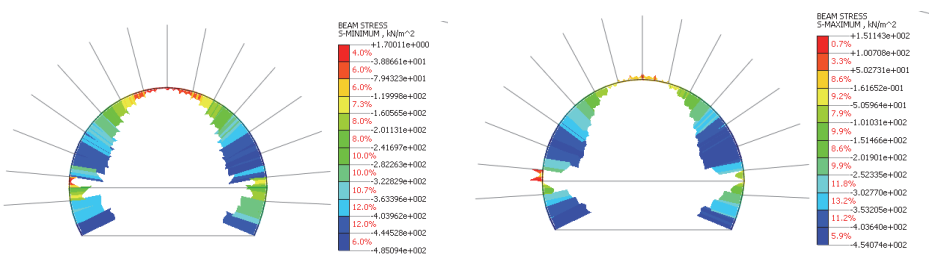
▶Result of output location and direction of Beam element



Sign convention of GTS NX is that (+) is represented for tension and (-) is represented for compression. In tunnel analysis, the bending compression stress of the shotcrete has to be checked if it is within the acceptable limits. Therefore, to check the bending compressive stress, check the max (-) value of Beam Element Stresses > S-MIN, S-MAX.

- Select the stage (Harden invert support) to check the results in the Work tree > Results > K0_0.5, and Select Beam Element Stresses > S-MIN. Check the bending compressive stress.

▶Combined stress(min)
▶▶Combined stress(max)



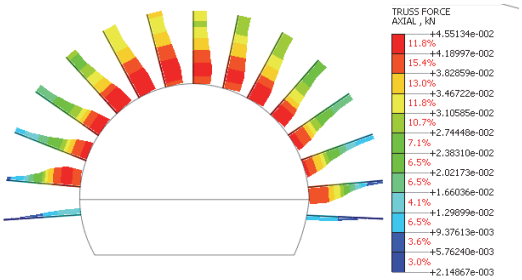
6.3 Axial Force of Rockbolt

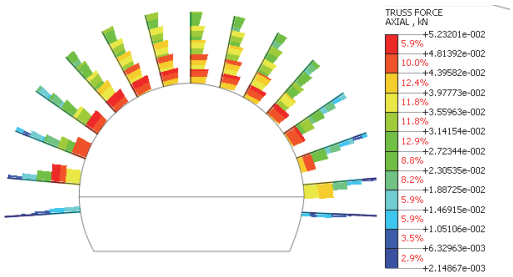
Rockbolt is modeled as truss element. A truss element can carry axial force.

- Select the stage (Harden invert support) in the Work tree > Results > K0_0.5, and Select Truss Element Forces > AXIAL FORCE. Check the axial force.
- The [Nodal Average] is an option which shows the average results of adjacent nodes to make the contour or diagram look smoother.

▶Check the nodal average

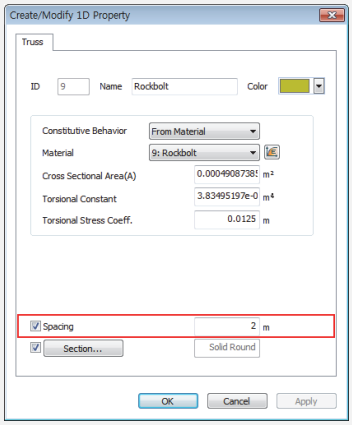
▶▶uncheck the nodal average





Tip

In 2D analysis, the CTC (spacing) which refers to distance between 1D elements is entered in [Static/Slope Analysis > Prop./ CSys./ Func. > Property].



In GTS NX, member axial force is output by the value of axial force per number of members ($f^* = nf$). CTC is included in the member axial force by multiplying CTC to the number of members.