Chapter 9
2D Tunnel Lining

Workflow

	: Geometry > Point & Curve > Tunnel	6
	: Mesh > Generate > 1D	6
	: Mesh > Element > Parameters	7
	: Mesh > Tools > Renumber	8
	: Static/Slope Analysis > Load > Self Weight	10
	: Static/Slope Analysis > Load > Beam Load	10
	: Static/Slope Analysis > Load > Comb. Set	12
	: Mesh > Element > Create	13
	: Static/Slope Analysis > Boundary > Constraint	14
	: Analysis > Analysis Case > General	14
	: Analysis > Analysis > Perform	15

2D Tunnel Lining

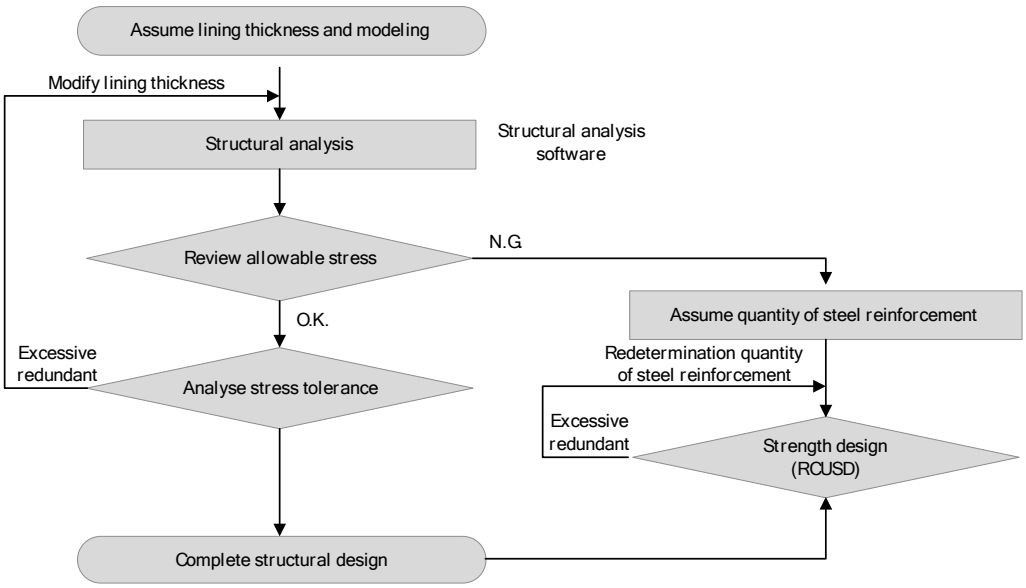
Section 1
Overview

1.1 Learning Purpose

Structural design of tunnel lining is processed by structural analysis first and then cross section design. Generally we perform allowable stress method which is an additional design method in the standard of plain concrete. If the allowable stress is exceeded, we apply ultimate strength method.

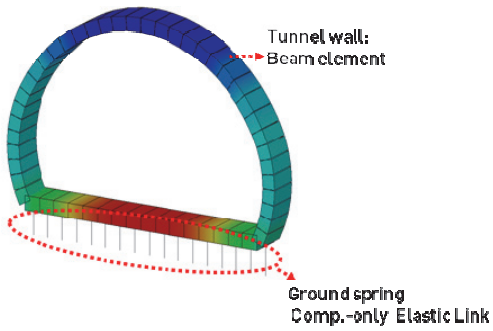
Analysis process of tunnel lining is illustrated in the following schema. Choose lining thickness and arrangement of steel reinforcing bars according to loading and boundary conditions.

▶Flow diagram of lining analysis



Generally, use continuum of beam elements to model the concrete lining, and consider the interaction with ground as elastic spring which only bears compressive force.

▶Lining modeling



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

- Modeling of tunnel lining
- Modeling of compression- only spring

- Beam element force and combined load set

1.2 Model and Analysis Summary

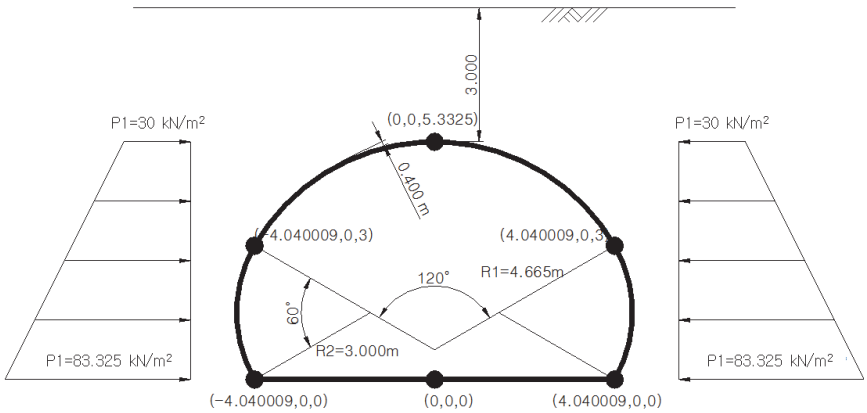
The model used in this tutorial is simplified for instruction purpose, so it might be different with real project application.

Data and loads for tunnel excavation area are shown in the following table.

Tunnel Type		Open cut tunnel
Height of Soil above the tunnel		3m
Soil Property	Unit Weight (γs)	20 kN/m^3
	Internal Friction Angle (ϕ)	30°
	Coefficient of Earth Pressure (K0=1-sinϕ)	0.5
Material (Concrete)	Design Criterion Strength (fck)	27000 kN/m^2
	Unit Weight (γ)	25 kN/m^3
	Elastic Modulus (Ec)	2.4645x10^7 kN/m^3
Tunnel Shape		3 Centered Tunnel (R1 = 4.665m, R2 = 3.0m, A1 = 60° , A2 = 60°)
Length		14m

Tunnel section and calculated static earth pressure is shown in the diagram below. Self weight, Vertical earth pressure, and Horizontal earth pressure are applied to the tunnel structure.

▶Open cut tunnel mimic



Tip

In open cut tunnel analysis only static earth pressure and passive earth pressure are considered. However, excavated tunnel analysis needs to consider yielding load instead of earth pressure.

Section 2

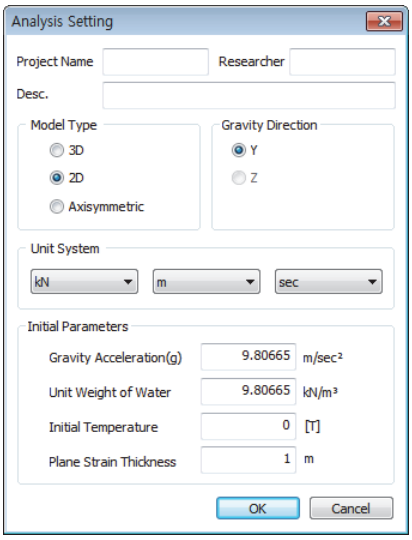
Analysis Setting

[Open the attached start file (09_lining_start).]

∗ : Analysis > Analysis Case > Setting

- Set model type, gravity direction, initial parameters and unit system for the analysis. The unit system can be changed whenever you want during the modeling process and after performing the analysis. The input parameters will be automatically converted by the right unit system.
- This is a 2D model which uses SI unit system (kN, m).

▶Analysis setting



Section 3
Define Material and Property

3.1 Definition of Ground and Structural Materials

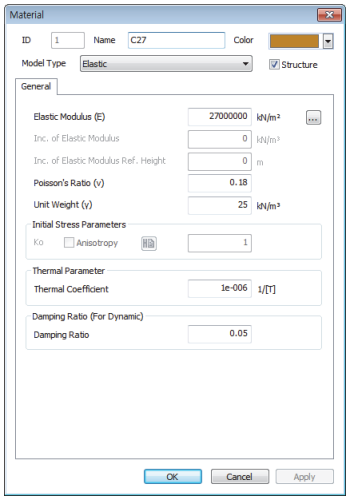
Materials used in this model are listed in the following table. Check [Structure] option for structural member.

▶Table. Structure material

[Unit : kN, m]

Name	C27
Type	Isotropic
Model type	Elastic
Elastic modulus(E)	2.7e7
Poisson's Ratio(v)	0.18
Unit Weight(r)	25

▶Define structure material



3.2 Definition of Properties

Lining uses beam elements which resist axis/shear/moment. Material properties are defined in the following table. Cross section stiffness is calculated automatically based on cross section shape.

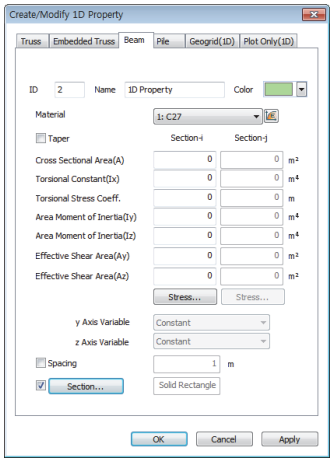
▶Table. Structure property

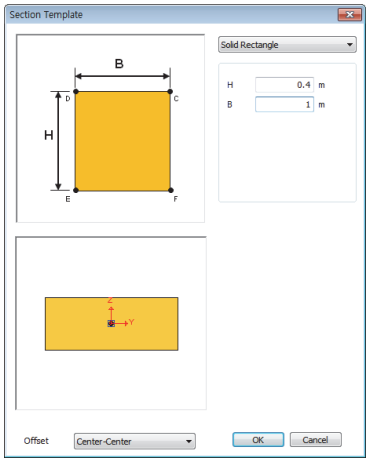
[Unit : kN, m]

Name	Lining
Type	1D
Model type	Beam
Material	C27
Spacing	-
Section Shape	Rectangle
Section Size	H=0.4, B=1

▶Define structure property

▶▶Define cross-section shape





Section 4
Modeling

Since the purpose of this tutorial is to study 2D geometry / mesh generation, loading condition setting and member design, you can start the tutorial by opening the start file in which basic materials and properties have already been predefined.

4.1 Modeling Geometry

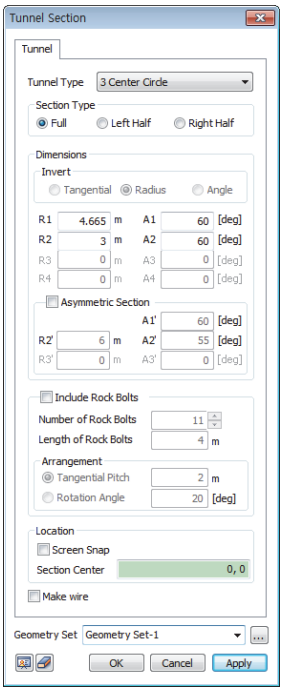
∗ : Geometry > Point & Curve > Tunnel

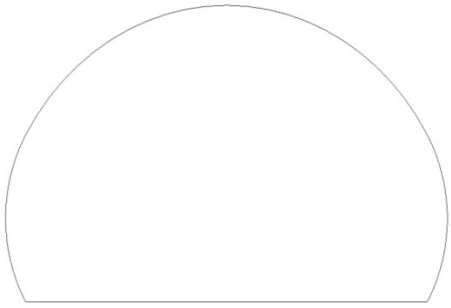
Create tunnel geometry. GTS NX Offers Tunnel-section creation tool for convenience.

- For [Tunnel Type] select '3 Center Circle'. And choose section type 'Full'.
- Input R1=4.665, A1=60, R2=3, A2=60.
- Uncheck [Make wire] option. And click [OK] to create tunnel.

▶Generating tunnel section

▶▶Generated tunnel edge





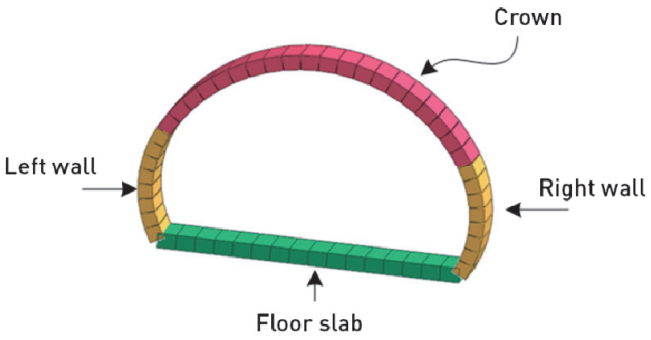
4.2 Generate Mesh

∗ : Mesh > Generate > 1D

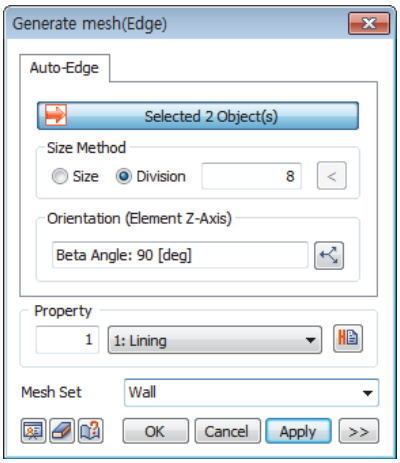
Model the tunnel lining with beam elements which bear axis/shear/moment.

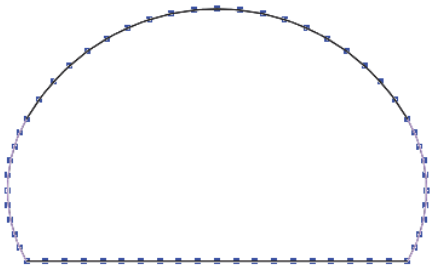
- Select left/right wall, and enter [Division] to '8'.
- Name the mesh set as [Wall], and press [Apply] button.
- In the same way generate crown and floor slap elements.
  (Crown: Division 20, Floor slab: Division 20)

▶Open cut tunnel



▶Generate mesh(Line)
▶▶Generated mesh(Node)



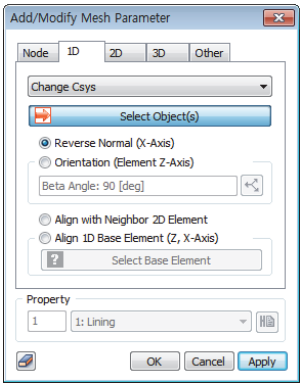


* : Mesh > Element > Parameters

This is process of changing element coordinate systems. Element coordinate system is the coordinate system based on which member force and stress results are outputted. Especially for beam elements it is also the coordinate system based on which shearing rigidity and bending rigidity directions are inputted. Therefore it is very important to understand its definition accurately. For element such as 'truss element' or tensile/compressive only element, which has only axial direction rigidity, only x axis is meaningful. Although y and z axes have no real meanings, they are necessary to define the arrangement direction of the member section for graphic display.

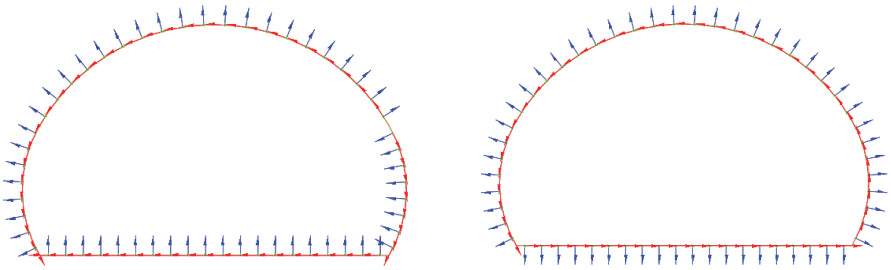
- In Work Tree > Mesh, Select wall, crown, floor slab.
- Mouse right click > Context menu > Display, Check 'Element CSys'.
- Select Mesh > Element > Parameter > 1D tab.
- In the Dropdown menu, select [Change Csys].
- Use 'Reverse Normal (X-Axis)'and set elements into the same direction.

▶Change CSys



▶Before alignment

▶▶After alignment



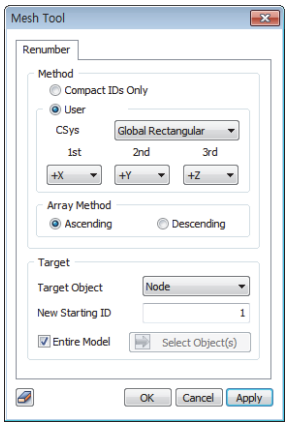
∗ : Mesh > Tools > Renumber

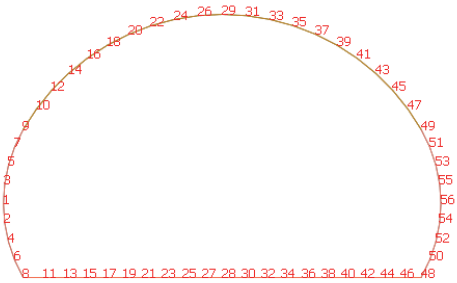
Sort the numbers of the generated nodes and elements according to specific standard.
By knowing number of node and element in advance, you can easily distinguish node and element after analysis.

- Select wall, crown, floor slab mesh in the Work Tree > Mesh.
- In Mouse right click > Context menu > Display, Check [Display Node ID].
- Select Mesh > Tools > Renumber.
- Choose method to 'User' and set standard coordinate system to 'Global Rectangular'. Set 1st standard to '+X', 2nd to '+Y'. Choose [Array Method] to 'Ascending'. Select target type 'Node' and input '1' as starting ID.
- Click [Apply] button.
- In the same way show element number and renumber elements.

▶Renumber

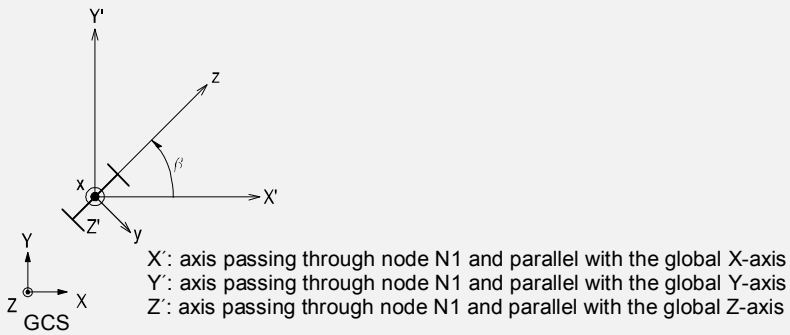
▶▶Modified node number



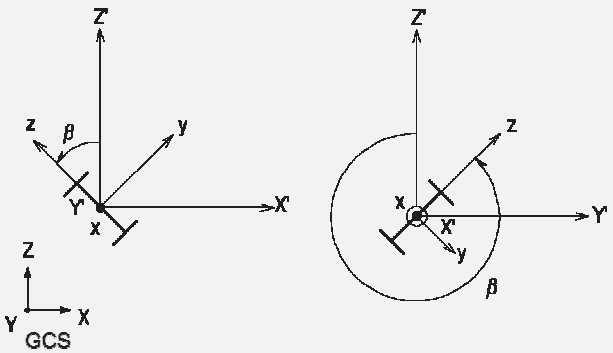


Tip

For convenience, GTS NX uses 'Beta angle (β)' concept to determine the directions Y and Z axes of element coordinate system.
For any line element, X axis signifies direction from node N1 to node N2 (Refer to image). If X axis of element coordinate system is parallel to Z axis of global coordinate system, beta angle becomes an angle between X axis of GCS and Z axis of element coordinate system. Sign of the Beta angle follows 'right-hand rule' with X axis as the axis of rotation. And if X axis of line element is not parallel to global Z axis, beta angle become vertical angle between Z axis of GCS and x-z-plane of element coordinate system.



(a) In case of Vertical member (If X axis of line element is parallel to global Z axis)



(b) In case of Horizontal or diagonal member (If X axis of line element is not parallel to global Z axis)

[Concept map of Beta Angle]

Section 5
Analysis Setting

5.1 Setting Load Condition

Loads applied to this model are self weight, horizontal earth pressure, and vertical earth pressure.

∗ : 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.

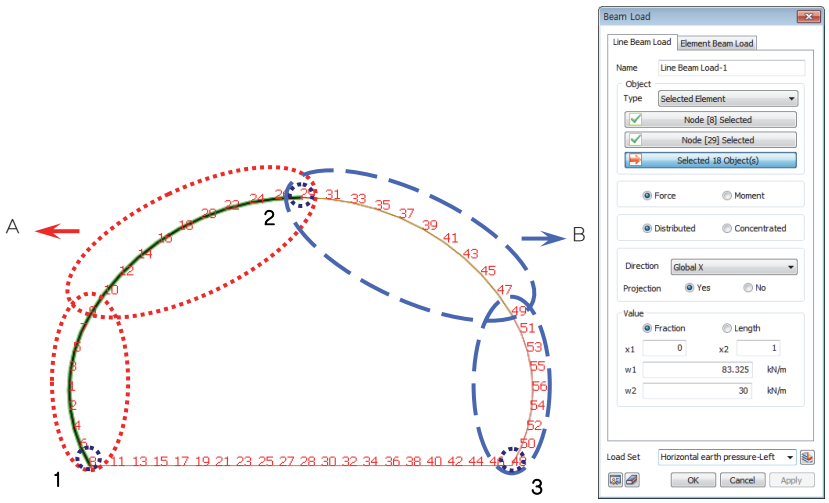
- Set the [Name] to 'Gravity-1' and enter 'Self weight' as the [Load set] name.
- Put -1 for Gy value.
- Click [OK].

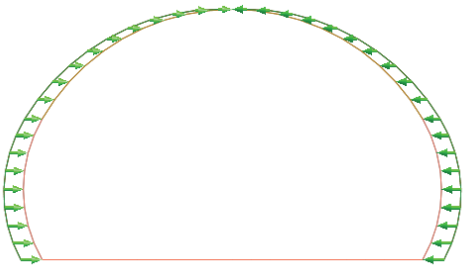
∗ : Static/Slope Analysis > Load > Beam Load

Apply horizontal earth pressure. The horizontal earth pressure is trapezium shaped as suggested on page 2. Use [Beam Load] to apply the load.

- In Work Tree > Mesh, Select wall, crown, floor slab mesh.
- Mouse right click > Context menu > Display, Check [Display node ID].
- Execute Static/Slope Analysis > Load > Beam Load.
- Select object type to 'Selected Element'.
- Name the load as 'Horizontal earth pressure - left'
- According to image below select first node as '1' and second node as '2'.
- Select elements of 'A'.
- Select direction to 'Global X'.
- Input 0, 1, 83.325, 30 for X1, x2, w1, w2.
- Name the load set as 'Horizontal earth pressure', Click [Apply].
- In the same way, generate 'Horizontal earth pressure-right' about node 3, 2, B. (Input load size to (-) value.)

▶Input beam load(Horizontal earth pressure)
▶▶Input beam load



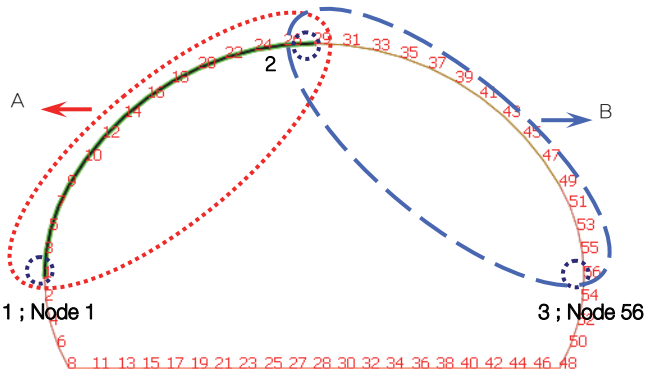


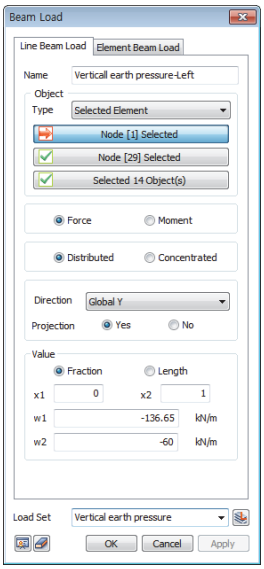
* : Static/Slope Analysis > Load > Beam Load

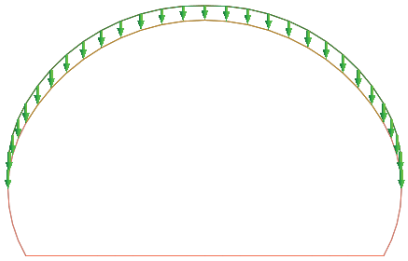
Apply vertical earth pressure. Vertical earth pressure differs according to depth. Use [Beam Load] to load the pressure.
Since the overburden is 3m, depth of tunnel wall end node (node 1) is 6.8325m. So if soil unit weight is 20kN/㎥, vertical earth pressure is 136.65 kN/㎡ at the end of side wall and 60 kN/㎡ at tunnel crown.

- In [Line Beam Load] tab select type as 'Selected Element'.
- Name the pressure as [Vertical earth pressure-Left].
- According to image below Select '1' for first node, '29' for second node and 'A' for element.
- Select direction to 'Global Y'.
- Input 0, 1, -136.65, -60 for X1, x2, w1, w2.
- Name the load set as 'Vertical earth pressure', Click [Apply].
- In the same way, generate 'Horizontal earth pressure-right' about node 56, 29, B.

▶Input beam load(Vertical earth pressure)
▶▶Enter bean load







* : Static/Slope Analysis > Load > Comb. Set

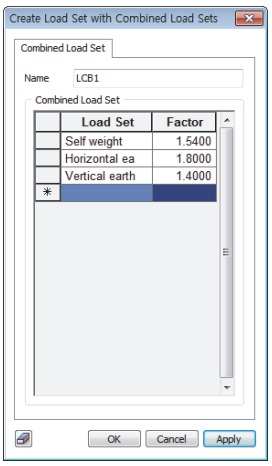
Ground spring is modeled by using compression only function of elastic link. Loading conditions can't be combined linearly because each load performs nonlinear analysis. In another word, by calculating results in each loading condition first and then combining them together using [result combination] function, we cannot obtain the correct result. Therefore it is necessary to combine load sets and transfer them to a unit static load using [Combine static load set] function before the analysis.

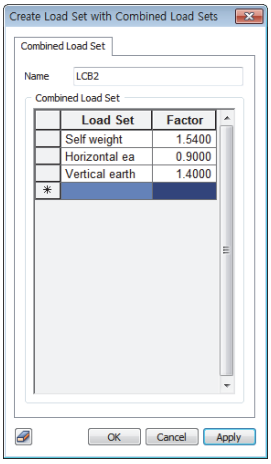
The load combinations which we are interested in cross section design are listed below.

LCB 1 : 1.54 self weight + 1.8 Horizontal earth pressure + 1.4 Vertical earth pressure
LCB 2 : 1.54 Self weight + 0.9 Horizontal earth pressure + 1.4 Vertical earth pressure

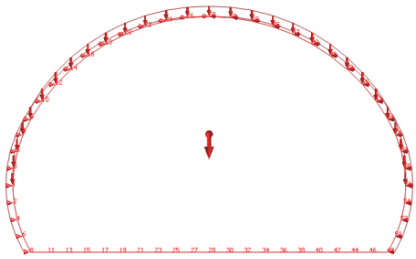
- Name combined load set to 'LCB1'.
- Select load set of Self weight, Horizontal earth pressure, and Vertical earth pressure. And Input each factor value '1.54', '1.8', '1.4'.
- Click [Apply].
- In the same way generate LCB2.

▶Combined load set





▶Combined load



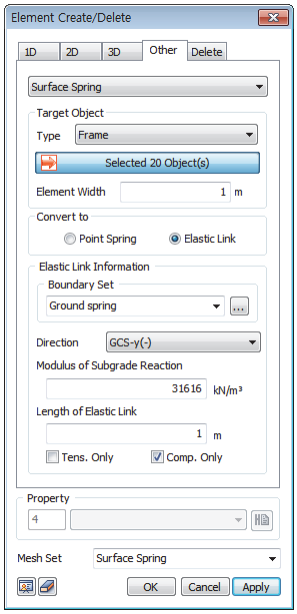
5.2 Setting Boundary Conditions

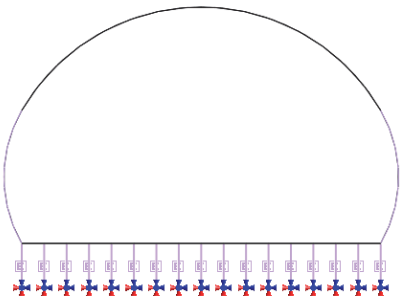
* : Mesh > Element > Create

Simulate ground spring in the analysis model using surface spring. Select nodes where the ground springs are going to be applied, and input coefficient of soil reaction. Then spring coefficient of each node is automatically calculated by considering intervals between selected nodes.
Since foundation of tunnel excavation part cannot resist tension force, use ground spring as compression-only element and then perform boundary nonlinear analysis. This is open cut excavation tunnel. So apply ground spring only to the floor slab for boundary condition.

- Select [Other] tab.
- In the Dropdown menu select surface spring.
- Select elements of floor slab.
- Set width of element to '1(m)', and Convert to 'Elastic Link'.
- Input 'Ground spring' at boundary set, choose direction 'GCS-y(-)'.
- Input Modulus of Subgrade Reaction=31616 kN/m³, Length of Elastic Link= 1m. And select [Compression Only] option.
- Name the mesh set as 'Surface Spring', press [OK] button.

▶Generate surface spring
▶▶Generated surface spring





Tip

Subgrade Reaction Modulus in vertical direction at the tunnel bottom is calculated according to subgrade reaction calculating formula of 'highway bridge design code'.

$K_{v0} = \frac{1}{30}\alpha E_0 = \frac{1}{30}x1x1400 = 46.667\ kgf/cm^3$

$B_v = \sqrt{A_v} = \sqrt{800x1400} = 1058.3\ cm$

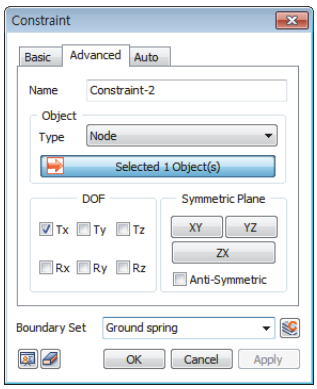
$E_0 = 28N = 28x50 = 1400\ kgf/cm^2$

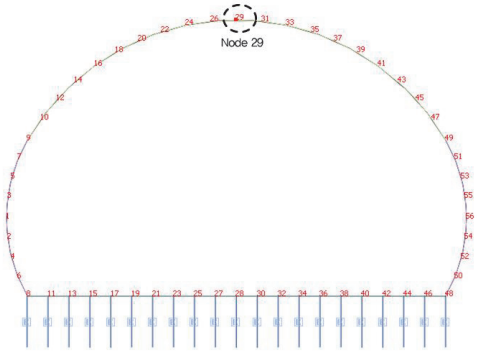
$K_v = K_{v0}\left(\frac{B_v}{30}\right)^{-3/4} = 46.667\ x\ \left(\frac{1058.3}{30}\right)^{-3/4}$ = 3.224 kgf/cm³ = 3224 tonf/m³=31616 kN/m³

* : Static/Slope Analysis > Boundary > Constraint

Although constraint in X direction is not an obligation since earth pressure is loaded symmetrically, to prevent numerical error, we will still input X direction constraint. Set constraint at peak of the tunnel crown.

- Verify if object type in [Advance] tab is set to 'node'.
- Select 'Node 29' as object. Check 'Tx' at DOF.
- Name the boundary set as 'Ground spring', Click [OK].



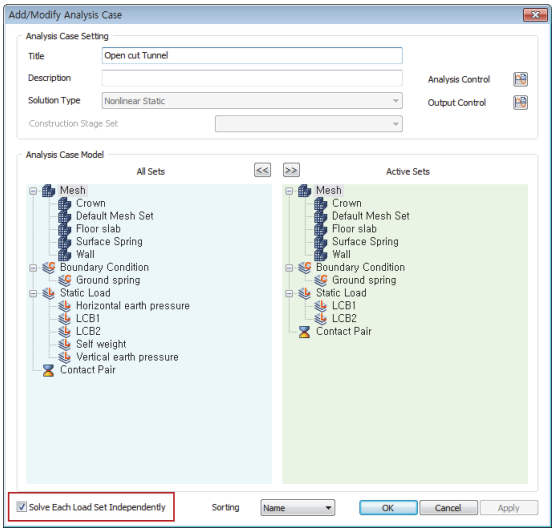


5.3 Setting Analysis Case

* : Analysis > Analysis Case > General

In the analysis case, data of the model are defined for the analysis. Analysis and output types can be controlled in the advance options. In the Linear/Nonlinear static analysis, it is possible to perform the analysis separately according to each activated load set.

- Name the analysis case as 'Open cut tunnel'.
- Select solution type 'Nonlinear Static'.
- Move entire mesh set, Ground spring, LCB1, and LCB2 to [Activate Sets].
- Select [Solve Each Load Set Independently] option. And press [OK].



5.4 Perform Analysis

* : Analysis > Analysis > Perform

Press the [Perform] button to run the analysis. When the analysis completes, the software automatically switches to [Post-Mode] in which you can check results. To modify model and other conditions after the analysis, you can switch back to [Pre-Mode].

- Perform analysis.

Tip

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].
The model is automatically saved before being analyzed. The information about analysis is saved in *.OUT file in the same place where the model file is saved.
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 results of nodes and elements after the analysis are also plotted in *.OUT file.

Section 6
Results and Generate Reports

After the analysis it is possible to verify deformation, member force of each combined load set. In this tutorial, the main result items which need to be checked are listed below

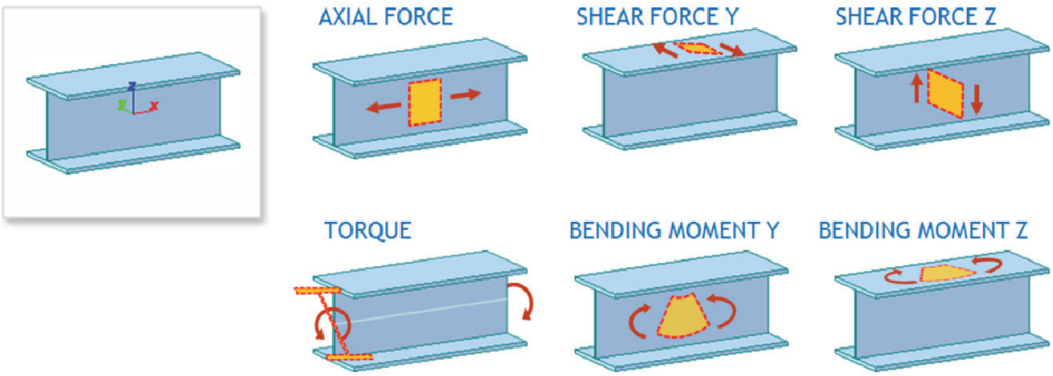
- Verify member force about each combined load

6.1 Verify Member force about Each Combined Load

Member force and stress are plotted according to element coordinate systems, so you have to check alignment of element coordinate systems before verifying the result.

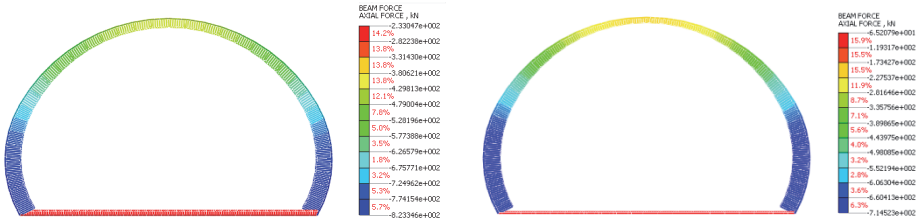
Member forces of Beam element are shown below

▶Member force generation of beam element



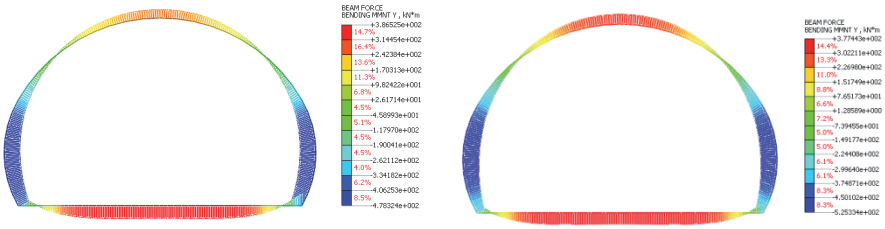
- Select Work Tree > Results > LCB1 > Beam Element Forces > AXIAL FORCE. In the same way check axial force diagram of LCB2.

▶Axial force(LCB1)
▶▶Axial force(LCB2)



- Select Work Tree > Results > LCB1 > Beam Element Forces > BENDING MOMENT Y. In the same way check moment diagram of LCB2.

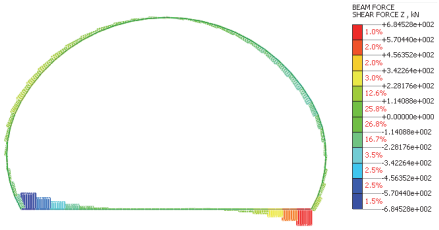
▶Moment(LCB1)
▶▶Moment(LCB2)

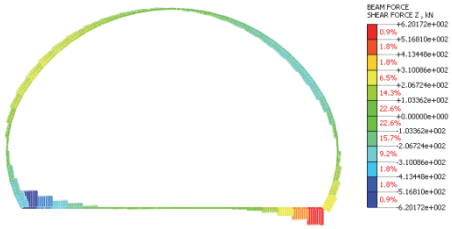


- Select Work Tree > Results > LCB1 > Beam Element Forces > SHEAR FORCE Z. In the same way check shear force diagram of LCB2.

▶Shear force(LCB1)

▶▶Shear force(LCB2)





Tip

(+) signifies tension and (-) signifies compression for beam element in MIDAS software.