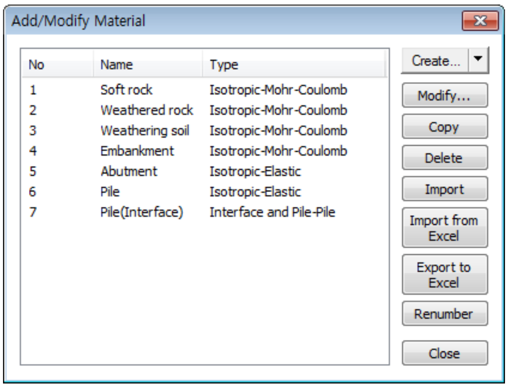
Section 1 Property/Coordinate/Function

1.1 Material

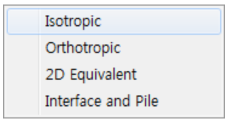
Overview

Define the general stiffness and nonlinear material properties of the ground and structure. For ground, additional permeability properties and drained/undrained conditions can be set.



Methodology

[Create] : Add a ground or structure material. The following 4 material types can be selected and the model type can be set for each material type.



The model type available for each material type is shown below. The ground/structure material properties and the material behavior properties are defined for each model type. Here, the elements used for ground modeling such as plane strain or solid can be assigned to a structure material that does not consider Ko effects or permeability properties.

Material type	Model type	Ground material	Structure material	Material behavior properties
Isotropic	Elastic	O	O	Linear elastic
	Tresca	O	O	Elasto-plastic
	von Mises (Nonlinear)	O	O	Elasto-plastic
	Mohr-Coulomb	O	O	Elasto-plastic
	Drucker Prager	O	O	Elasto-plastic
	Hoek Brown	O	O	Elasto-plastic
	Generalized Hoek Brown	O	O	Elasto-plastic
	Hyperbolic(Duncan-Chang)	O	X	Nonlinear elastic
	Strain Softening	O	X	Elasto-plastic
	Modified Cam Clay	O	X	Elasto-plastic
	Jardine	O	X	Nonlinear elastic
	D-min	O	X	Nonlinear elastic
	Modified Mohr-Coulomb	O	O	Elasto-plastic
	Soft Soil	O	X	Elasto-plastic
	Soft Soil Creep	O	X	Elasto-plastic
	User defined model	O	O	Elasto-plastic
	Modified UBCSAND	O	X	Elasto-plastic
	Sekiguchi-Ohta(Inviscid)	O	X	Elasto-plastic
	Sekiguchi-Ohta(Viscid)	O	X	Elasto-plastic
	Modified Ramberg-Osgood	O	O	Elasto-plastic
	Modified Hardin-Drnevich	O	O	Elasto-plastic
	Hardening Soil(small strain stiffness)	O	X	Elasto-plastic
	Generalized SCLAY1S	O	X	Elasto-plastic
	CWFS	O	O	Elasto-plastic
Orthotropic	Transversely Isotropic	O	O	Linear elastic
	Jointed Rock Mass	O	O	Elasto-plastic
	2D Orthotropic	X	O	Linear elastic
	Geogrid	X	O	Elasto-plastic
2D Equivalent	2 Dimensional equivalent	O	O	(Equivalent)Linear elastic
Interface /Pile	Interface	X	O	Elasto-plastic
	Shell Interface	X	O	Elasto-plastic
	User supplied - Shell Interface	X	O	Elasto-plastic
	Pile	X	O	Nonlinear elastic

[Isotropic] : Isotropic materials have the same properties in every direction and is used to define material behavior properties of most linear-elastic / nonlinear elastic / elasto-plastic materials.

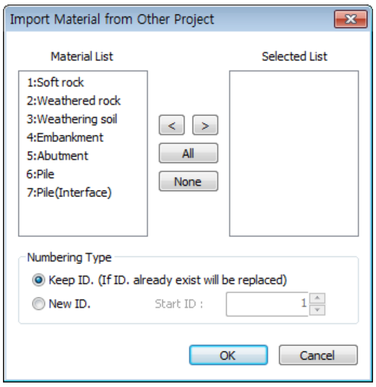
[Orthotropic] : Natural ground is generally layered and sloped, making it possible to have different strengths in each orthogonal direction. This option can also be used to define Jointed Rocks, which have different properties depending on the direction and behave differently according to the specific confinement conditions.

[2D Equivalent] : 2D equivalent linear analysis specific model. Use the converging strength and damping ratio from the equivalent linear method to consider the nonlinear and n1lastic behavior of materials.

[Interface / Pile] : Applied when simulating relative behavior (interface behavior) between ground and structure.

[Modify/Copy/Delete]
Modify the parameters of an added material. Copy can be used when adding multiple materials while only changing certain parameters.

[Import/Import from Excel / Export to Excel]
Import the material properties from a different model file with a saved material/property. This operation is useful when analyzing the existing project under the same conditions. Selecting the import file generates the material list containing all saved materials. The user can select the desired material.



The user can construct the frequently used material DB by importing or exporting the excel file containing the material properties.

[Renumber]
Change the registration number of a material. Repeating Add/Delete automatically sets the registration number to recently added number+1.

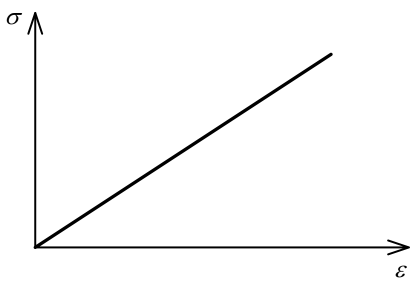
Refer to Ch.4 of the Analysis manual to understand the Finite element formulization and yield shape. The input parameters and behavioral characteristics for each material model are as follows.

1.2 General Material (Behavioral Properties)

Elastic

A linear elastic model where the stress is directly proportional to strain. The proportionality constants are the Elasticity modulus (E) and Poisson’s ratio (ν).

▸Linear elastic-stress strain behavior



Since the yield value is not defined, the calculated stress and strain of the linear elastic model can be highly unrealistic. Hence, it is recommended that the Mohr-Coulomb or other nonlinear material models be used for general analysis. However, this model is appropriate in modeling concrete or structural steel structures, which have a much higher strength than the ground.

Tresca

The Tresca criterion was originally developed to be used on yield conditions of metallic materials. In geotechnics, it is often used to simulate the ground material behavior during undrained conditions. This model has some flaws when applied to soil materials, such as no consideration of the effects of hydrostatic pressure acting on the yield surface. Firstly, the assumption that shear stress is unrelated to hydrostatic pressure (or confining pressure) is wrong for general soil behavior. Secondly, the yield stress is the same for compression and tension in this criterion, but soils generally have a much larger compressive strength than tensile strength, sometimes even negligible tensile strength.

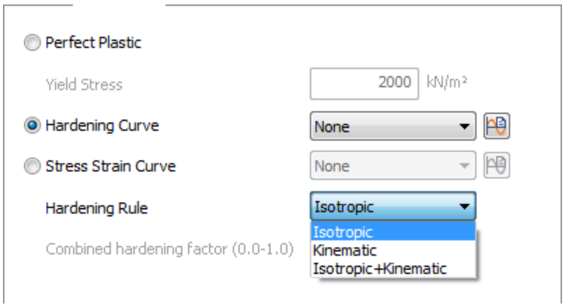
However, performing the total stress analysis for saturated soils under undrained conditions (called $\phi=0$ analysis) using the Tresca yield criterion still gives appropriate results. The Tresca yield criterion can be used because according to the results, the shear strength of the saturated soil is unrelated to the stress comp1nt of the hydrostatic pressure during undrained loading. In this case, because the maximum shear stress limit represents the undrained shear strength, the decision must be made from the results of the undrained triaxial compression test.

von Mises (Nonlinear)

The von Mises model is often used to define the behavior of ductile materials based on the principle that yielding occurs when the shear stress reaches the threshold value. This model can be applied to truss, embedded truss and pipe elements as well as geotechnical elements. It can also be applied when simulating anchors, nails or steel pipe piles made from steel.

The von Mises model has the same limitation as the Tresca model when applied to soil materials; it does not consider of the effects of hydrostatic pressure and the yield stress is the same for compression and tension. Like the Tresca criterion, the undrained strength of saturated soil can be appropriately presented using the von Mises yield criterion. This model is useful because it does not have the mathematical difficulty or analysis complexity caused by the hexagonal corners of the curved surface on the Tresca criterion.

As a material yield, hardening defines the change of yield surface with plastic straining, which is classified in to the three types : **Isotropic, Kinematic and Combined**.
Appropriate for all types of materials, which exhibit Plastic Incompressibility.



Perfect Plastic
Specify Initial Uniaxial (tensile) Yield Stress

Hardening Curve
Relation between plastic strain and stress(true stress) can be resulted from uniaxial compression / tensile test or shear test.

Stress Strain curve (optional)
Relation between strain and stress(true stress)

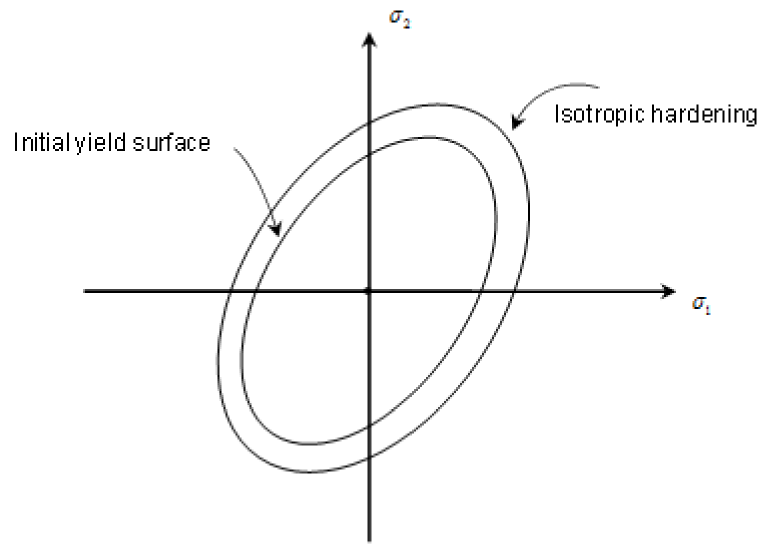
Hardening Rule
Isotropic, Kinematic and Combined (Isotropic + Kinematic)

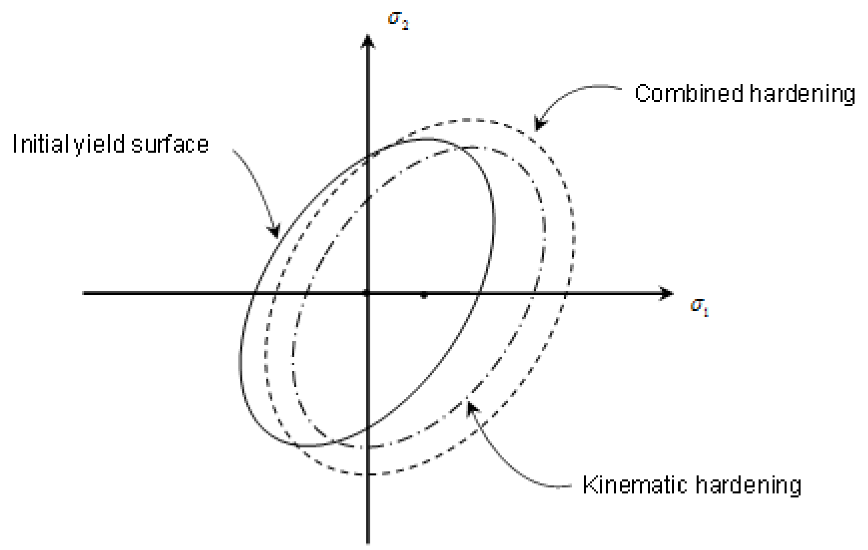
- Total increment of Plastic can be expressed by Isotropic and Kinematic Hardening as follows:

$$\sigma_y = \lambda_c h_y(0) + (1-\lambda_c) h_y(e_p)$$

- **Combined hardening factor (λc, 0~1)** represents the extent of hardening. '1' for Isotropic, '0' for Kinematic, and between '0~1' for Combined hardening.

▸Yield Surface for each hardening rule

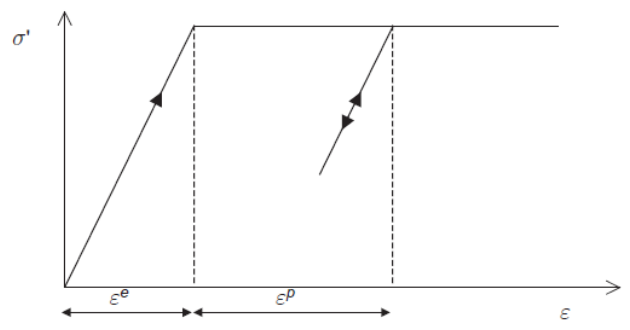




Mohr-Coulomb

The Mohr-Coulomb model is defined by an elasto-plastic behavior as shown in the figure below. This behavioral assumption shows reliable results for general nonlinear analysis of the ground and is widely used in simulating most terrain.

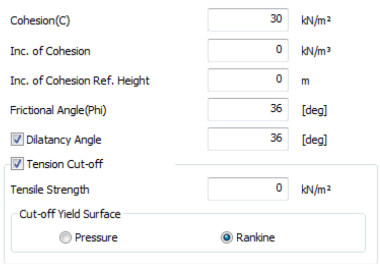
▸Material behavior of Mohr-Coulomb model



The Mohr-Coulomb yield criterion has 2 flaws when using geo-materials. First, the intermediate principal stress does not affect yield, which is a contradictory assumption to real soil test results. Second, the Meridian and Failure envelope of the Mohr-Circle is linear; so the Strength parameter (angle of friction) does not change with the Confining pressure (or Hydrostatic pressure). This criterion is accurate within a limited range of confining pressure but as the range difference increases, the accuracy decreases. However, this criterion is often used because it is easy to use and displays considerably accurate results within the general confining pressure range.

The major nonlinear parameters used to define the Coulomb yield criterion are as follows.

▸Mohr-Coulomb



Cohesion (C) , Friction angle (Φ)

Soils have different cohesion and friction angle depending on their type and these values are applied to the shear strength equation. Soils, unlike other construction materials, have very little resistance to tension and in most cases shear failure occurs. When an external force or self weight is applied, shear stress occurs in the ground. The strain increases with stress increase and as these progresses, it works along a plane causing what is known as shear failure. The shear stress induces shear resistance and the shear resistance limit is called shear strength. The shear resistance of soil is made up of 2 comp1nts: cohesion and friction angle.

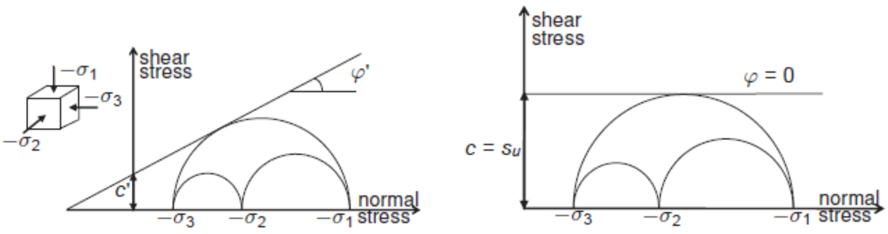
According to Coulomb, the shear strength of soil can be expressed in the following linear equation.

$\tau = c + \sigma \tan\varphi$ (c: Cohesion, σ: Normal stress, φ: Interior friction angle)

Cohesion is the shear strength when the interior friction angle is '0(zero)' according to the yield criterion. It can be defined as an undrained shear strength of cohesive soils. Sandy soils with no cohesion can be

defined as c=0, but to avoid errors in analysis, it is recommended that a value of at least 0.2 (kN/m2) be entered.
Defining the cohesion automatically sets the tensile strength by that amount. However, because tensile resistance is generally ignored for geo-materials, the Tension-Cutoff is set to prevent unrealistic resistance behavior to tension.

▸Mohr-Coulomb Failure envelope (Drained/Undrained)



Increment of Cohesion Reference Height

In general, the strength properties of the soil change with the depth and confining pressure; even within a ground layer composed of the same material. For example, defining a soil layer several meters deep as a ‘strength parameter’ may be a limitation in the detailed simulation of a ground behavior. The ground layer can be further subdivided and modeled, but this characteristic can be replaced by changing cohesion according to height. If the cohesion increases according to the height being '0(zero)', the cohesion has a constant value and if it is not '0(zero)', the cohesion is calculated with reference to a standard height **(reference height based on the Global Coordinates)** using the following equation.

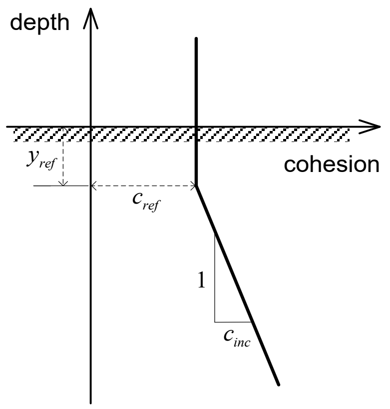
$$c = c_{ref} + \left(y_{ref} - y\right)c_{inc} \qquad \left(y \le y_{ref}\right)$$
$$c = c_{ref} \qquad \left(y > y_{ref}\right)$$

c_{ref} : Input cohesion value

c_{inc} : Incremental amount depending on cohesion depth

y_{ref} : Depth of c_{ref} measurement

▸ Conceptual diagram of cohesion increment



The y in the equation represent the integral point positions of an element where the finite element method calculation occurs. If the integral point position is higher than y_{ref} , the cohesion can be less than 0 in some places. To avoid this, use the c_{ref} value instead of further decreasing the cohesion value.

Dilatancy Angle

The dilatancy angle can be viewed as the volume increase rate according to shear strain. It is a type of strength parameter for roughness and is generally defined as dilatancy angle = interior friction angle-30˚.

Hence, if the interior friction angle is less than 30˚, the dilatancy angle is close to '0(zero)'. In real tests, a negative dilatancy angle can be defined for vey loose sandy soil but numerically, the dilatancy angle has a value between 0 and the interior friction angle.
For undrained analysis, the dilatancy angle must be set as '0(zero)' when the interior friction angle is '0(zero)'. The important thing is that the dilatancy angle is a parameter considered in analysis after changes have been made to the constitutive equation. If the effects of the dilatancy angle are not considered, the same value must be entered for the dilatancy angle and interior friction angle. In other words, not checking the 'Consider dilatancy angle' option automatically performs the analysis with the dilatancy angle equal to the interior friction angle.

Tensile Strength (Tension-CutOff)

Input the allowable tensile strength of the geo-material. In many cases, tension cracks can be observed on the natural ground surface rather than shear failure. Input the allowable tensile strength to assign tensile resistance to the geo-material. The tensile resistance of geo-materials are generally ignored and so the default setting is '0(zero)'. Not checking the tensile strength option uses a tensile stress, automatically calculated from cohesion and the interior friction angle, into the analysis.

Drucker Prager

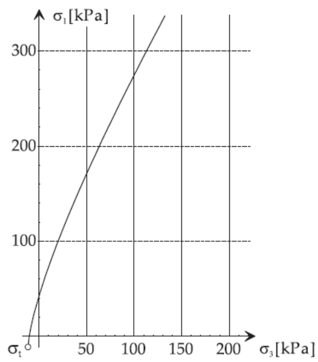
The Drucker-Prager model was developed by Drucker and Prager (1952) to solve the numerical problems that occur on the corners of the yield shape of the Mohr-Coulomb model. The internal algorithm is the same as the Mohr-Coulomb model, and the material constant can be related to the existing cohesion (c) and friction angle (ϕ) of the Mohr-Coulomb model.

▸ Drucker-Prager

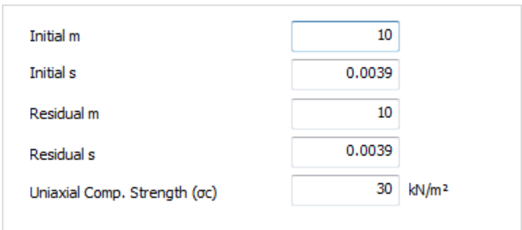
Hoek Brown

Geo-materials can be largely divided into rocks and soils. Rocks are more rigid than soils and their strength characteristics depend on the degree of weathering. The behavioral characteristics of rocks are divided by the fact that the change in stiffness due to stress is negligible. In particular, the shear stress and tensile stress of rocks have a larger effect on the overall behavioral characteristic than soils. Hoek and Brown (1980) suggested the concept of an equivalent continuum to define the stress decrease phenomenon in jointed rock mass failure. A yield function was proposed to distinguish between intact rock and broken rock and when the rock failure was defined by this function, certain parameter values that define the function could be decreased to simulate the stress decrease phenomenon. This method defines the unconfined compressive strength that could not be considered in the existing Mohr-Coulomb method and allows accurate and simple representation of rock behavior, making it a widely used analysis method even today. The shear strength of rocks can be expressed using the Mohr-Coulomb yield criterion and Hoek-Brown strength parameters within a certain stress range can be used to predict the cohesion and friction angle of the Mohr-Coulomb model.

▸Hoek-Brown yield criterion



The main nonlinear parameters of the Hoek-Brown criterion are as follows.



Initial m,s value

The initial m,s values are 1 of the empirical Hoek-Brown material constants for Intact rocks that classify rocks according to their grade (type). General m values are shown in the table below.

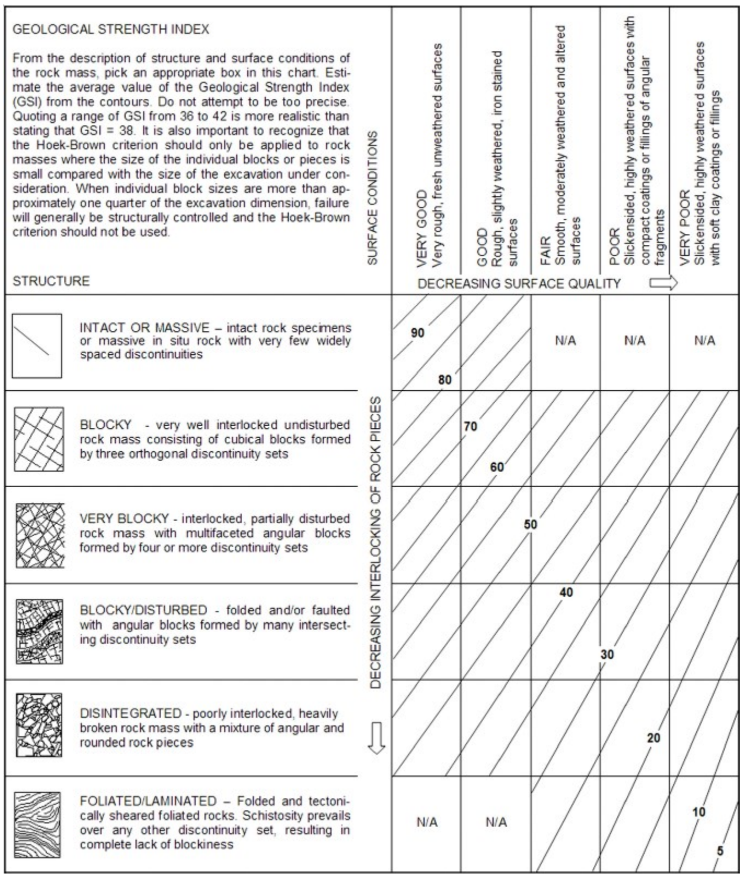
▸Hoek-Brown material constant m_i

Rock type	Class	Group	Texture			
			Coarse	Medium	Fine	Very fine
SEDIMENTARY	Clastic		Conglomerate (22)	Sandstone 19	Siltstone 9	Claystone 4
				Greywacke (18)		
	Non-Clastic	Organic		Chalk 7		
				Coal (8-21)		
		Carbonate	Breccia (20)	Sparitic Limestone (10)	Micritic Limestone 8	
		Chemical		Gypstone 16	Anhydrite 13	
METAMORPHIC	Non Foliated		Marble 9	Hornfels (19)	Quartzite 24	
	Slightly foliated		Migmatite (30)	Amphibolite 25 - 31	Mylonites (6)	
	Foliated*		Gneiss 33	Schists 4 - 8	Phyllites (10)	Slate 9
IGNEOUS	Light		Granite 33		Rhyolite (16)	Obsidian (19)
			Granodiorite (30)		Dacite (17)	
			Diorite (28)		Andesite 19	
	Dark		Gabbro 27	Dolerite (19)	Basalt (17)	
			Norite 22			
	Extrusive pyroclastic type		Agglomerate (20)	Breccia (18)	Tuff (15)	

The s constant can be calculated from the GSI (Geological Strength Index) when the rock is intact and has a value of '1'. The GSI according to rock grade is shown in the table below and the value can be generally predicted by calculating the rock classification results (RMR, RQD).
The s calculated from the GSI is as follows.

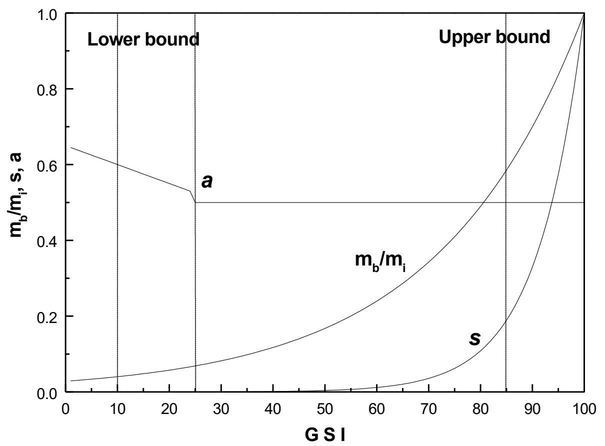
$$s = \exp\left(\frac{GSI - 100}{9}\right) (GSI > 25)$$
$$s = 0\ (GSI \leq 25)$$

▸ GSI (Hoek(1999))



The m,s relationship from the GSI assumed by a=0.5 from the Hoek-Brown model is as follows.

▸Relationship between GSI and m,s



Uniaxial Compressive Strength

Input the Uniaxial compressive strength test results for Intact Rock. The general numerical value for rock grade is as follows.

▸Uniaxial compressive strength

Grade	Term	Uniaxial Comp. Strength (MPa)	Field estimate of strength	Examples
R6	Extremely strong	> 250	Specimen can be chipped with a geological hammer	Fresh basalt, chert, diabase, gneiss, granite, quartzite
R5	Very strong	100 – 250	Specimen requires many blows of a geological hammer to fracture it.	Amphibolite, sandstone, basalt, gabbro, gneiss, granodiorite, limestone, marble, rhyolite, tuff
R4	Strong	50 – 100	Specimen requires more than one blow of a geological hammer to fracture it.	Limestone, marble, phyllite, sandstone, schist, shale
R3	Medium strong	25 – 50	Cannot be scraped or peeled with a pocket knife, specimen can be fractured with a single blow from a geological hammer.	Claystone, coal, concrete, schist, shale, siltstone
R2	Weak	5 – 25	Can be peeled with a pocket knife with difficulty, shallow indentation made by firm blow with point of a geological hammer.	Chalk, rocksalt, potash.
R1	Very weak	1 – 5	Crumbles under firm blows with point of a geological hammer, can be peeled by a pocket knife.	Highly weathered or altered rock.
R0	Extremely weak	0.25 – 1	Indented by thumbnail	Stiff fault gouge

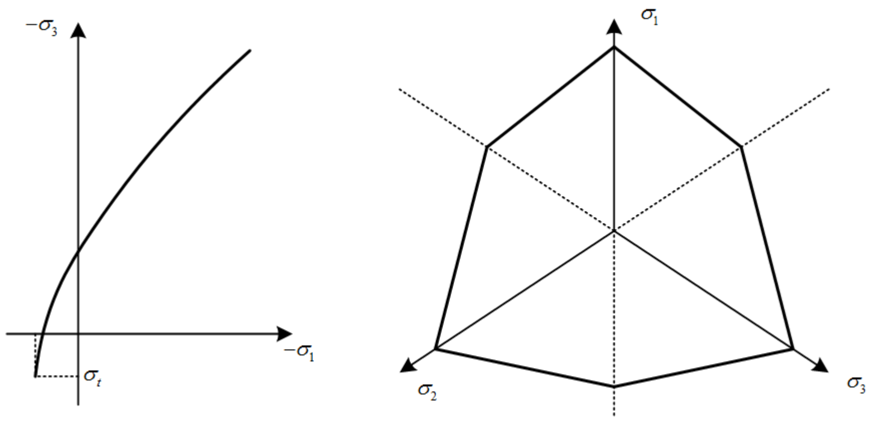
Generalized Hoek Brown

Representative model to simulate general rock behavior (stiffer and stronger than other types of soil). Hoek-Brown model is isotropic linear elastic behavior and Generalized Hoek-Brown is to link the empirical criterion to geological observations by means of one of the available rock mass classification schemes. All geological index was subsequently extended for weak rock masses. This model is also applicable for **Strength Reduction Method (slope stability analysis)**.

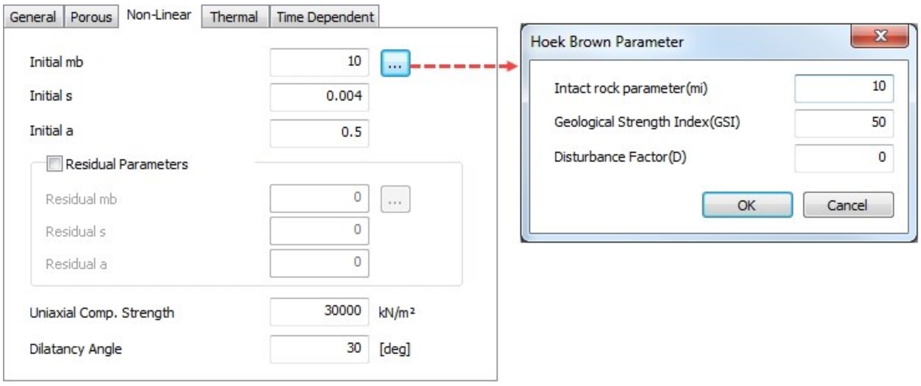
$$f_{HB} = (\sigma_1 - \sigma_3) - \sigma_{ci}\left(-\frac{m_b}{\sigma_{ci}}\sigma_1 + s\right)^a$$

$$\sigma_1 \geq \sigma_2 \geq \sigma_3$$

▸Failure surface in principle stress plane



The main nonlinear parameters of the Generalized Hoek-Brown criterion are as follows.



$$m_b = m_i \exp\left(\frac{GSI - 100}{28 - 14D}\right)$$

$$s = \exp\left(\frac{GSI - 100}{9 - 3D}\right)$$

$$a = \frac{1}{2} + \frac{1}{6}\left(e^{-GSI/15} - e^{-20/3}\right)$$

Refer to following Geological Index (Hoek, 1999)

Uniaxial Compressive Strength

▸Uniaxial Compressive Strength

Grade	Term	Uniaxial Comp. Strength (MPa)	Field estimate of strength	Examples
R6	Extremely strong	> 250	Specimen can be chipped with a geological hammer	Fresh basalt, chert, diabase, gneiss, granite, quartzite
R5	Very strong	100 – 250	Specimen requires many blows of a geological hammer to fracture it.	Amphibolite, sandstone, basalt, gabbro, gneiss, granodiorite, limestone, marble, rhyolite, tuff
R4	Strong	50 – 100	Specimen requires more than one blow of a geological hammer to fracture it.	Limestone, marble, phyllite, sandstone, schist, shale
R3	Medium strong	25 – 50	Cannot be scraped or peeled with a pocket knife, specimen can be fractured with a single blow from a geological hammer.	Claystone, coal, concrete, schist, shale, siltstone
R2	Weak	5 – 25	Can be peeled with a pocket knife with difficulty, shallow indentation made by firm blow with point of a geological hammer.	Chalk, rocksalt, potash.
R1	Very weak	1 – 5	Crumbles under firm blows with point of a geological hammer, can be peeled by a pocket knife.	Highly weathered or altered rock.
R0	Extremely weak	0.25 – 1	Indented by thumbnail	Stiff fault gouge

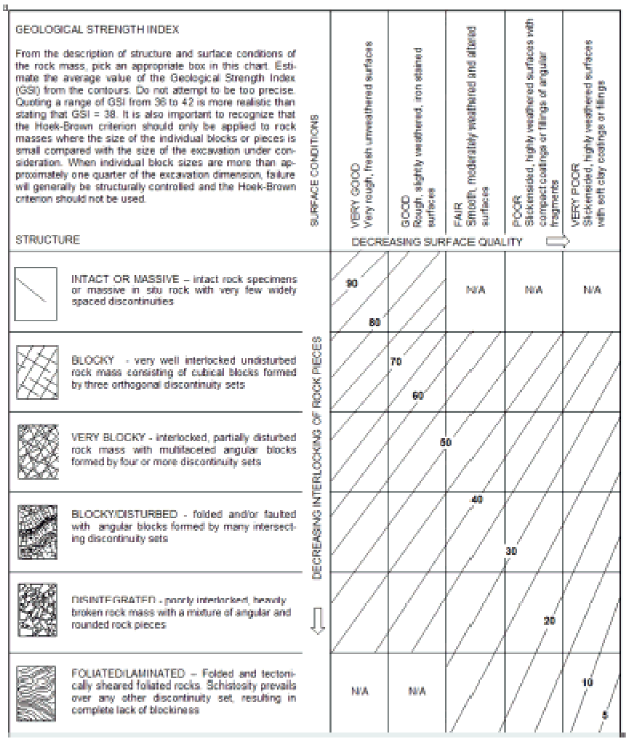
Intact Rock Parameter

▸Intact Rock Parameter

Rock type	Class	Group	Texture: Coarse	Medium	Fine	Very fine
SEDIMENTARY	Clastic		Conglomerate (20)	Sandstone 19	Siltstone 9	Claystone 4
				Greywacke (18)		
	Non - Clastic	Organic		Chalk 7		
				Coal (8-21)		
		Carbonate	Breccia (22)	Sparitic Limestone (10)	Micritic Limestone 8	
		Chemical		Gypstone 16	Anhydrite 13	
METAMORPHIC	Non Foliated		Marble 9	Hornfels (19)	Quartzite 24	
	Slightly foliated		Migmatite (30)	Amphibolite (25 - 31)	Mylonites (6)	
	Foliated*		Gneiss 33	Schists 4 - 8	Phyllites (10)	Slate 9
IGNEOUS	Light		Granite 33		Rhyolite (16)	
			Granodiorite (30)		Dacite (17)	
			Diorite (28)		Andesite 19	
	Dark		Gabbro 27	Dolerite (19)	Basalt (17)	
			Norite 22			
	Extrusive pyroclastic type		Agglomerate (20)	Breccia (18)	Tuff (15)	

Geological Strength Index (GSI)

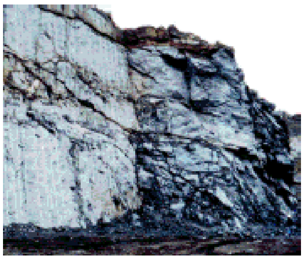
▸Geological Strength Index (GSI)



Guideline for estimating Disturbance Factor (D)

▶ Guideline for estimating Disturbance Factor (D)

Table 1: Guidelines for estimating disturbance factor *D*

Appearance of rock mass	Description of rock mass	Suggested value of *D*
	Excellent quality controlled blasting or excavation by Tunnel Boring Machine results in minimal disturbance to the confined rock mass surrounding a tunnel.	$D = 0$
	Mechanical or hand excavation in poor quality rock masses (no blasting) results in minimal disturbance to the surrounding rock mass. Where squeezing problems result in significant floor heave, disturbance can be severe unless a temporary invert, as shown in the photograph, is placed.	$D = 0$ $D = 0.5$ No invert
	Very poor quality blasting in a hard rock tunnel results in severe local damage, extending 2 or 3 m, in the surrounding rock mass.	$D = 0.8$
	Small scale blasting in civil engineering slopes results in modest rock mass damage, particularly if controlled blasting is used as shown on the left hand side of the photograph. However, stress relief results in some disturbance.	$D = 0.7$ Good blasting $D = 1.0$ Poor blasting
	Very large open pit mine slopes suffer significant disturbance due to heavy production blasting and also due to stress relief from overburden removal. In some softer rocks excavation can be carried out by ripping and dozing and the degree of damage to the slopes is less.	$D = 1.0$ Production blasting $D = 0.7$ Mechanical excavation

▶Engineering example for material verification (Slope Stability Analysis)

The Shear Strength Reduction Method for the Generalized Hoek-Brown Criterion
Hammah, R.E., Yacoub, T.E. and Corkum, B.C.
Rocscience Inc., Toronto, ON, Canada
Curran, J.H.
Lassonde Institute, University of Toronto, Toronto, ON, Canada

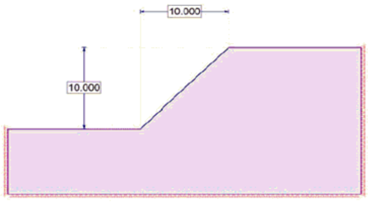


Figure 3. Geometry of the slope in Example 1.

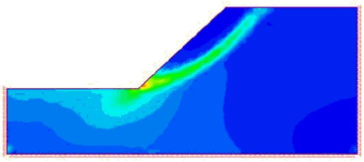
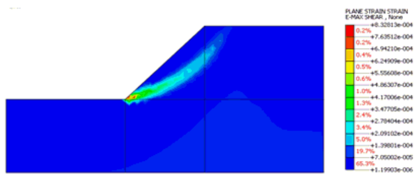


Figure 4. Contours of maximum shear strain in the slope at failure. The contours reveal the failure mechanism predicted by the SSR method.

[Reference - F.S. : 1.15]

Table 1. Properties of the rock mass in the Example 1slope

Property	Value
Young's modulus, E (MPa)	5000
Poisson's ratio, ν	0.3
Weight, γ (MN/m³)	0.025
Uniaxial compressive strength σ_{ci} (MPa)	30
GSI	5
Intact rock parameter m_i	2
Disturbance factor, D	0
Parameter m_b	0.067
Parameter s	2.5 x 10⁻⁵
Parameter a	0.619

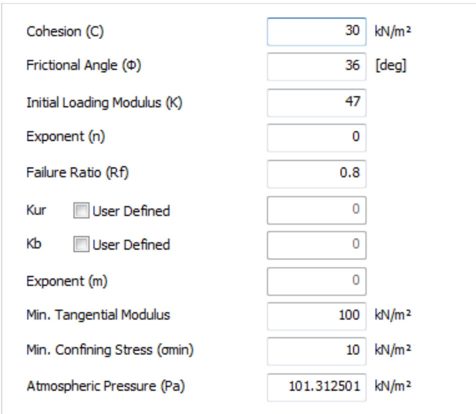


[GTSNX - F.S. : 1.19]

Hyperbolic (Duncan-Chang)

Ground stress-strain behavior becomes nonlinear as it approaches the yield criterion; nonlinear elastic models simulate such ground behavior by modifying the foundation modulus. The function proposed by Duncan and Chang (1970) is used to calculate the foundation modulus. The stress-strain curve of the function is a hyperbola and the foundation modulus is a function of confining stress and shear stress. This nonlinear elastic material model is very useful because it only needs material properties that can be easily obtained from the triaxial compression test or literature. The Duncan and Change nonlinear stress-strain curve represents a hyperbola between the axial strain space generated by shear stress ($(\sigma_1 - \sigma_3)$) and it can be defined according to stress state and stress path by 3 foundation moduli (Initial modulus (E_i), Tangent modulus (E_t), Unloading-reloading modulus (E_{ur})).

The main nonlinear parameters of the hyperbolic model are as follows.



The results of the triaxial compression test can be plotted on a vertical axis of E/p_a or B_m/p_a and a horizontal axis of σ_3/p_a. Set each axis to a log scale and the vertical axis value at σ_3/p_a =1 is the **Initial Loading Modulus (K)**. The **Initial Stiffness Exp1nt(n)** can be found from the slope when the vertical axis is E/p_a and the **Bulk Modulus Exp1nt(m)** can be found from the slope when the vertical axis is B_m/p_a. Here, the Bulk modulus B_m is defined by the following equation and can be predicted using the relationship with Poisson's ratio. Here, the Poisson's ratio is limited to values within 0 to 0.5.

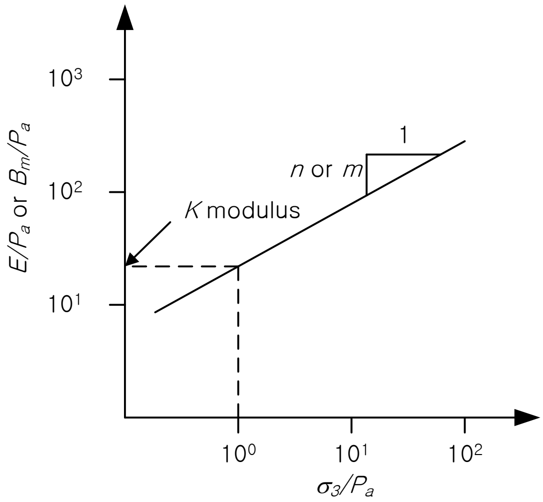
$$B_m = \frac{(\Delta\sigma_1 + \Delta\sigma_2 + \Delta\sigma_3)/3}{\Delta\varepsilon_v},$$

$\Delta\sigma$: Amount of principal stress change,

$\Delta\varepsilon_v$: Amount of volume strain change

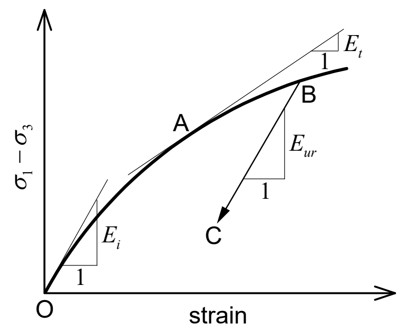
$$B_m = \frac{E}{3(1-2\nu)}$$

▸Set material property



The Duncan and Change nonlinear stress-strain curve can be defined according to stress state and stress path by 3 foundation moduli (Initial modulus (E_i), Tangent modulus (E_t), Unloading-reloading modulus (E_{ur})).

▸Nonlinear stress-strain behavior



Here, the **Failure Ratio (Rf)** can be found by the relationship between the Initial modulus (E_i) and Tangent modulus (E_t). The ratio of failure is the ratio between the asymptote of the hyperbola and the maximum shear strength and has a value between 0.75~1. A convergence problem can occur when the Tangent modulus (E_t) is too small and so the minimum Tangent modulus value is set as the atmospheric pressure (Pa). **Bulk modulus number (Kb)** can be calculated from the Bulk modulus (Bm) and Bulk modulus index (m).

$$B_m = K_b p_a \left(\frac{\sigma_3}{p_a} \right)^m$$

Here,

B_m : Bulk modulus,

K_b : Bulk modulus number,

m : Bulk modulus index

Unloading-reloading modulus number K_{ur} can be calculated from the unloading-reloading ratio E_{ur}.

$$E_{ur} = K_{ur} p_a \left(\frac{\sigma_3}{p_a} \right)^n$$

If the confining stress is '0(zero)' or negative (tensile state) when calculating the initial moduli, the moduli can be '0(zero)' or a negative value. Hence, a lower bound needs to be set for the confining stress and the set Minimum confining pressure is 0.01Pa.

The suggested parameter values depending on the density of sandy soils are as follows. (Duncan, J. M. and Chan, C. Y. (1970))

▸Table. Summary of stress-strain parameters for uniform fine silica sand

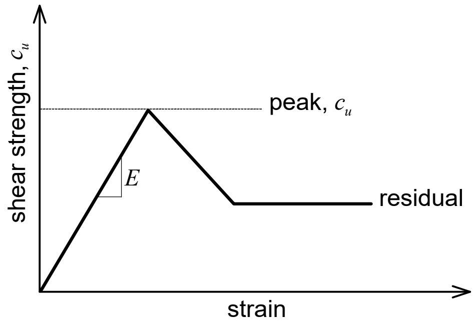
Relative density	ϕ_d	R_f	K	K_{ur}	n
100% (dense)	36.5	0.91	2000	2120	0.54
38% (loose)	30.4	0.90	295	1090	0.65
$C_d = 0$ for dense and loose sand					

* ϕ_d, C_d : Dry state friction angle and cohesion

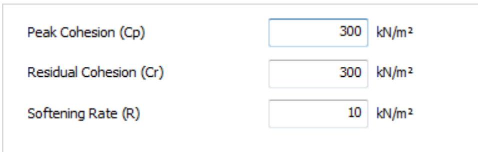
Strain Softening

The softening model consists of a linear region until it reaches the maximum shear strength. The softening region then enters a nonlinear region. Specify the Maximum residual strength, Residual strength and Softening rate to define the constitutional relationship below.

▸Strain softening constitution relationship

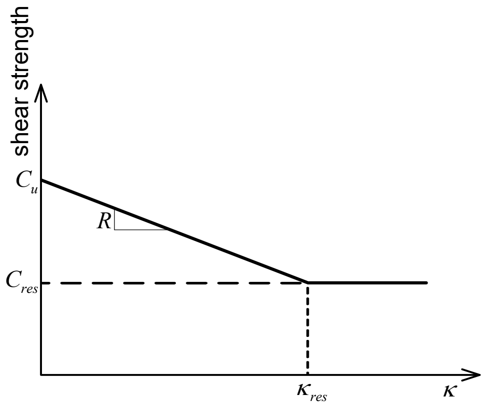


The main nonlinear parameters of the Softening model are as follows.



The maximum cohesion using the undrained shear strength and define residual cohesion, defined by the amount of stress reduction and softening rate to simulate the strength decrease with strain and bulk increase. In comparison with the Tresca model, the Tresca model behaves in a simple or perfectly plastic state for undrained conditions but this model decreases from the maximum strength and can simulate the effects of the residual strength.

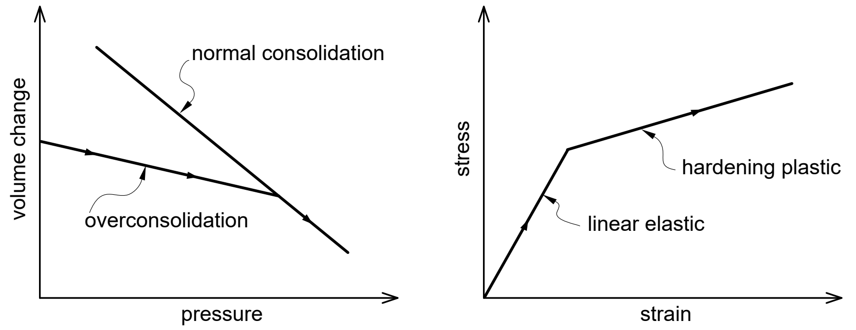
▸Softening behavior



Modified Cam Clay

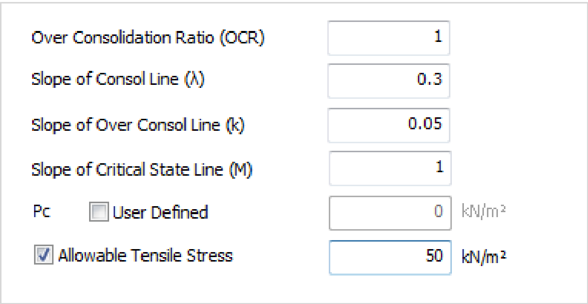
A model used to simulate clay materials. The general relationship between volume change and pressure in clay ground can be expressed using the concept of normal consolidation line and over-consolidation line. The over-consolidation line is also called the dwelling line and the stress increase (load) follows along over-consolidation line to the normal consolidation line. Passing the intersection with additional stress increase makes the stress state move down along the normal consolidation line. This has similar characteristics to the stress-strain curve of an elasto-hardening plastic model. Hence, the initial linear elastic region of the over-consolidation line can be corresponded to the hardening plastic region of the normal consolidation line.

▸Volume-Pressure vs Stress-Strain relationship



To use the Modified Cam Clay model, the initial void ratio, initial stress and initial Pre-consolidation pressure need to be defined. The Pre-consolidation pressure can be directly entered, or calculated from the initial stress and Over Consolidation Ratio (OCR). When both the OCR and Pre-consolidation pressure are entered, the Pre-consolidation pressure is preferentially used.

The main nonlinear parameters of the Modified Cam Clay model are as follows.



Symbol	Definition
κ	Over-consolidation line slope
λ	Normal consolidation line slope
M	Critical state line slope

The material properties of the ground are generally obtained from the 1-dimensional consolidation test and the Compression index C_c and Swelling index C_s can be obtained from the void ratio (e) - $\log_{10}(p)$ graph. The Compression index and Swelling index are related to the Normal consolidation line slope λ and Over-consolidation line slope κ by the following equations.

$$\lambda = \frac{C_c}{2.303} \quad , \quad \kappa = \frac{C_s}{2.303}$$

The Critical state line slope M can be assumed from the effective shear resistance angle (shear resistance angle from drained test).

$$M = \frac{6\sin\phi}{3-\sin\phi}$$

ϕ : Interior friction angle from triaxial compression test

The OCR value can be used to calculate the stress distribution of the in-situ state from the current loaded stress distribution. The stress for each depth is calculated from the entered OCR and because the ground

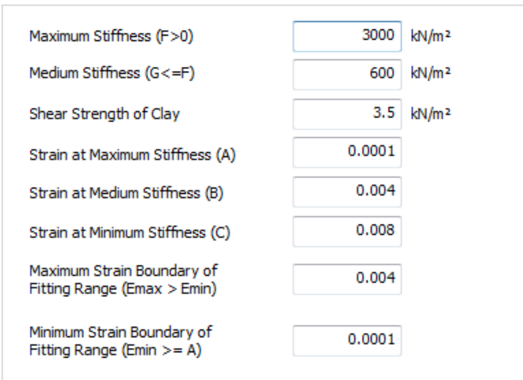
surface stress can be underestimated than the actual initial stress, the Pc (Pre-consolidation pressure) can be directly defined. When both the OCR and Pre-consolidation pressure are entered, the Pre-consolidation pressure is preferentially used in the analysis.
When the Pc is entered, the internal solver studies whether the Pc and in-situ stress state satisfy the yield function. If it does not, the Pc is recalculated.

Allowable Tensile Stress

MCC material models fundamentally do not allow tensile stress in the failure criteria (stress-strain relationship). However, various conditions can generate tensile stress, such as the heaving of neighboring ground due to embankment load during consolidation or uplift due to excavation. To overcome the material model limits and increase the applicability, analysis on tensile stress within the 'allowable tensile stress' range can be conducted.
The size of the allowable tensile stress is not specified, and requires repeated analysis to input a larger value than the tensile stress created from the overburden load (embankment) or failure behavior. Hence, the allowable tensile stress value needs to be set, to prevent divergence and halting of analysis results due to tensile failure during analysis.
Hence, the allowable tensile stress value needs to be set, to prevent divergence and halting of analysis results due to tensile failure during analysis.
However, when directly entering the pc (pre-consolidation load), the allowable tensile stress cannot surpass the pc value. When defining using the OCR, the pc value is automatically calculated internally by considering the size of the input allowable tensile stress.

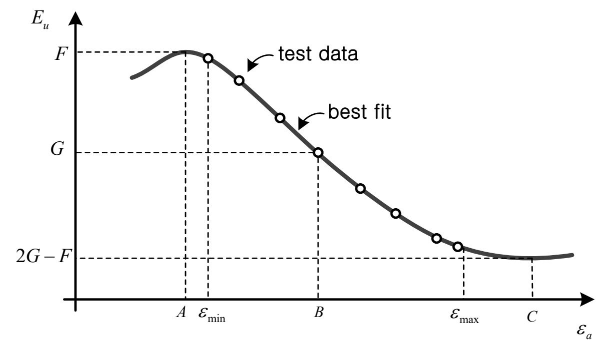
Jardine

The Jardine model is appropriate for geo-materials that display nonlinear behavior even with little strain. Jardine (1984) proposed this nonlinear elastic model to simulate clay like materials that display nonlinear behavior when the confining stress range is small and this model displays perfectly plastic behavior when the material stress is larger than the input shear stress. The main nonlinear parameters of this model are as follows.



Maximum Stiffness (F>0)	3000	kN/m²
Medium Stiffness (G<=F)	600	kN/m²
Shear Strength of Clay	3.5	kN/m²
Strain at Maximum Stiffness (A)	0.0001	
Strain at Medium Stiffness (B)	0.004	
Strain at Minimum Stiffness (C)	0.008	
Maximum Strain Boundary of Fitting Range (Emax > Emin)	0.004	
Minimum Strain Boundary of Fitting Range (Emin >= A)	0.0001	

The Jardine model is often applied to clay ground. Clays display nonlinear elastic behavior at small strains and the Jardine model was developed to consider this. The nonlinear equation can be derived from the Secant elasticity modulus and axial strain measured from the undrained triaxial compression test as shown below. The undrained triaxial compression test applies a load in the axial direction of a cylindrical sample and the stress along the side is maintained.



Here, F , G the maximum and average stiffness that satisfies the best fit line value, A , B , C the strain at the maximum, medium and minimum Elasticity modulus and ε_{max} , ε_{min} the maximum and minimum strain.

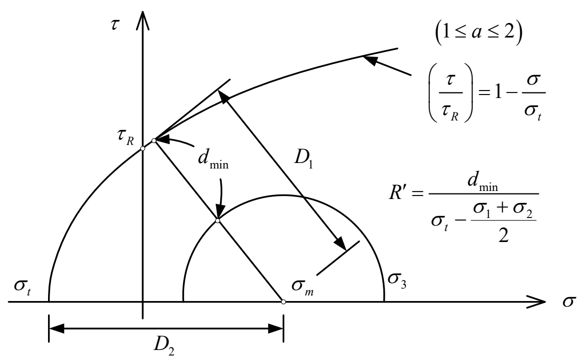
The shear strength of clay is needed because the Tresca model is used at the plastic region.

D-min

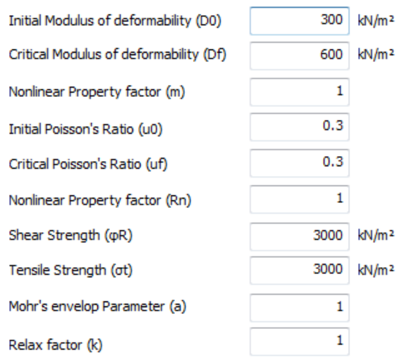
The D-min model is a secti1d linear model proposed by Japan Central Research Institute of Electric Power Industry (CRIEPI), Hayashi, Hibino and applied to general rocks (hard rock, soft rock etc.). In this model, the stiffness is different for each construction step, but the stiffness within 1 construction step is a constant value. In other words, the materials in the model have a fixed material property value for each load step and hence, do not require repeated analysis.

It is assumed that as the failure envelope approaches the Mohr circle, the interior bonding state of the rock eases and the Elasticity modulus decreases while the Poisson’s ratio increases. Hence, the relative distance between the Mohr circle and failure envelope in each section determines the Elasticity modulus and Poisson’s ratio.

▸Failure envelope and Mohr circle of D-min model



The main nonlinear parameters are as follows.



If the user enters the material properties at the initial state and limit state, the value proceeds to the limit state material property depending on the plasticity as shown below.

$$E = R^m (E_i - E_{cr}) + E_{cr}$$
$$\nu = R^n (\nu_i - \nu_{cr}) + \nu_{cr}$$

Here, E_i is the initial Elasticity modulus, E_{cr} is the limit Elasticity modulus, m is the nonlinear property factor, ν_i is the initial Poisson's ratio, ν_{cr} is the limit Poisson's ratio, n is the nonlinear material modulus. The Mohr's envelop parameter (a) and relax factor (k) increases with the increase in initial Elasticity modulus (E_i).The relationship between the Mohr's envelop parameter (a) and relax factor (k) with the initial Elasticity modulus (E_i) is shown below. The data was based on the triaxial compression test and put together by the Japan Road Traffic Information Center in 1986.

▸Table. Parameters based on initial Elasticity modulus (JARTIC, 1986)

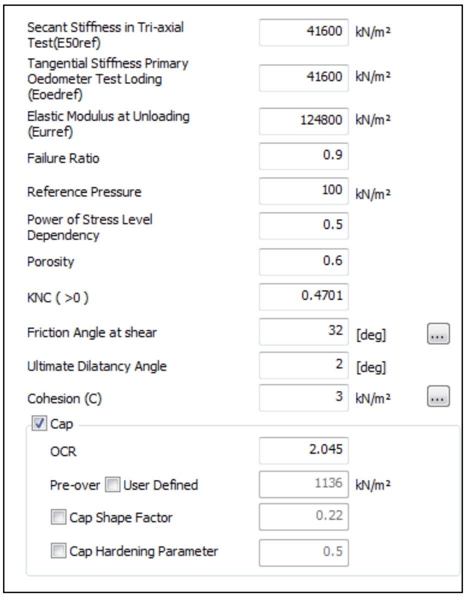
Initial Elasticity modulus (E_i , kgf/cm^2)	**Relax Factor (k)**	**Mohr's envelop parameter (a)**
$100 \le E_i < 1,000$	2.0	1.0
$1,000 \le E_i < 10,000$	4.0	2.0
$10,000 \le E_i < 100,000$	6.0	3.0
$100,000 \le E_i$	10.0	4.0

Modified Mohr-Coulomb

This model is an improvement on the Mohr-Coulomb model, generated by combining nonlinear elastic models and elasto-plastic models to make a suitable model for the behavioral characteristics of silt or sand based ground. The Modified Mohr-Coulomb model can simulate the Double stiffening behavior, which is not affected by the shear failure or compressive yield.

The axial strain and decrease in material stiffness caused by the initial deviatory stress is similar to the Hyperbolic (nonlinear elastic) model, but it is closer to the plastic theory than elastic theory and has differences in dilatancy angle consideration and yield cap application.

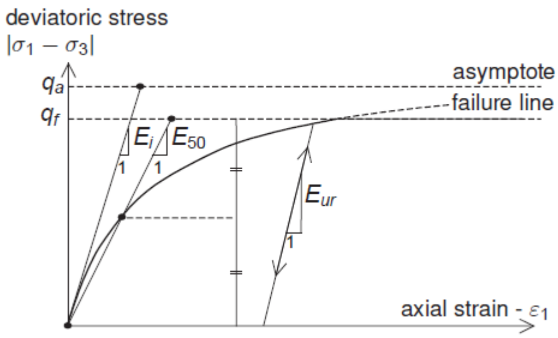
The main nonlinear parameters are as follows.



Elasticity modulus (Three types of reference stiffness modulus)

This model is a more detailed material model than the Mohr-Coulomb model and the Elasticity modulus can be set at different values for loading and unloading. In most cases, the Elasticity modulus for unloading is set larger to prevent uplift (bulging phenomenon) on the cutting surface due to stress release during excavation modeling. For a rough approximation, in case of hard soil (sand, OC clay), unloading stiffness is set equal to 3 times of secant stiffness in standard drained triaxial test. In case of soft soil, based on the relationship between compression and swelling index, unloading stiffness is approximately set equal to 10 times of secant stiffness.

▸Hyperbolic stress-strain relation



Failure Ratio

The ultimate deviatoric stress is derived from the Mohr-Coulomb failure criterion. As soon as the deviatoric stress reaches to the ultimate value, the failure criterion is satisfied and perfectly plastic yielding occurs. The ratio between the ultimate and the quantity in deviatoric stress is given by the failure ratio which must be smaller than1.

Reference Pressure

The reference stress used in the triaxial test of specific strengths on the nonlinear elastic curve. This can represent in-situ horizontal stress at mid-level of soil layer depending on OCR(Over Consolidation Ratio)

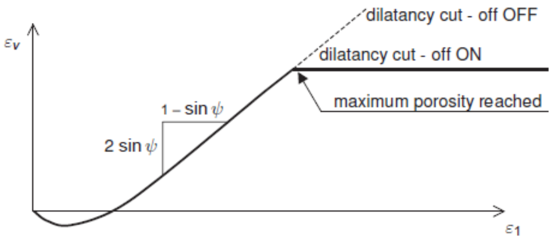
Power Law nonlinear elastic model coefficient

Hardening soil model is the stress dependency of soil stiffness. The reference stiffness modulus which is used in this model has the relationship with the confining stress dependent stiffness. In this relation, the amount of stress dependency is given by the power m. In case of soft clays, the power is recommended to be taken equal to 1. The value of m around 0.5 for hard soil like sand and OC clays can be used to simulate a logarithmic compression behavior. In general, m is in the range of 0.5 to 1.

Porosity

The void ratio is the volume ratio between voids and soil particles. Here, the porosity is the volume ratio between voids and the total soil including water. Hence, unlike the void ratio, the porosity cannot have a value larger than 1 and has a value of 0.6 in general. When soil experience shearing, dilating materials reach to the state of critical density. In order to include this soil behavior by means of dilatancy cut-off, the maximum porosity must be entered as advanced parameters. When the soil is subject to shear hardening, solver recalculates dilatancy angle.

▸Strain curve including dilatancy cut-off

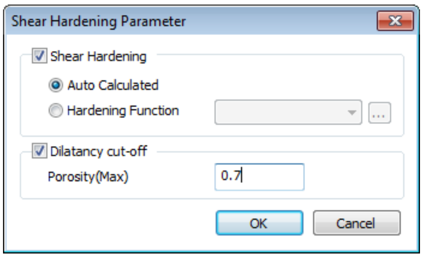


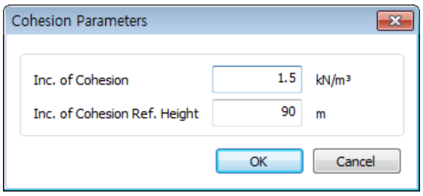
KNC

KNC is the percentage of K (Coefficient of earth pressure) in a normally consolidated ground. In other words, it is the effective horizontal stress ratio during maximum vertical stress. This can be expressed as **1-sin (Interior friction angle)** and because general clays have an interior friction angle of nearly zero, the value is close to 1. However, it cannot be smaller than '0(zero)'.

Friction angle at shear / Ultimate dilatancy angle / Cohesion

Same as the friction angle, dilatancy angle, cohesion parameters of the Mohr coulomb material model.

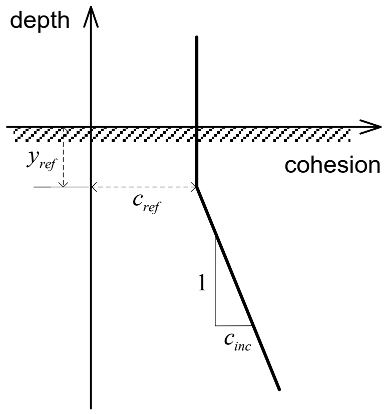




Shear hardening can be defined by equivalent plastic strain related to the mobilized shear resistance automatically. Shear yield surface can expand up to the Mohr-Coulomb failure surface.

Increment of Cohesion with depth can be defined with additional options as in Mohr-Coulomb model. The reference height must be inputted based on the Global coordinates.

▸ Conceptual diagram of cohesion increment



Cap (Compression hardening)

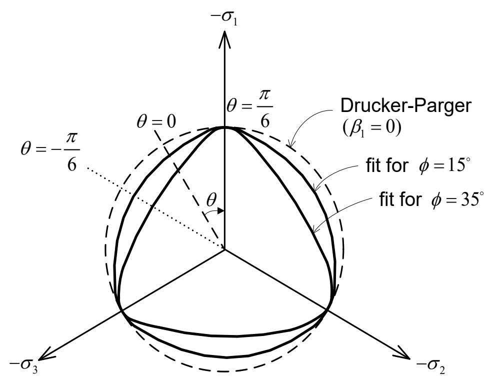
Compressive yield can happen when excessive compression forces occur on the ground. Normally, the compressive forces that cause yield are very large and the Mohr-Coulomb model has no problems in simulating real soils with it omitted. However, in order to simulate the compressive behavior more accurately, the model considers circles or ellipses when considering compressive failure, hence the name 'Cap'.

OCR / Pc (Pre-overburden pressure)

Yield surface of hardening soil model in p-q plane has the length of preconsolidation stress and its magnitude is determined by Cap shape factor and cap hardening parameter. The smaller value of cap shape factor(α) lead to steep caps underneath the Mohr-coulomb failure line. For this, the preconsolidation stress can be calculated from either the OCR(Over consolidation ratio) or the Pc(Pre-overburden pressure). The user can input the pressure at which compressive failure occurs.

Cap shape factor / Cap Hardening Parameter

Use this variable to change the shape of the Cap, a yield function shape. Refer to Ch.4 of the Analysis manual for a more detailed algorithm. These are considered automatically based on the relation between KNC and Eoedref rather than directly inputting parameters.

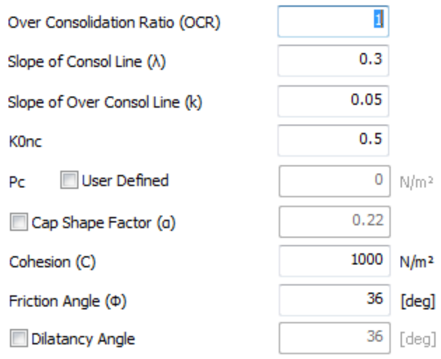


Following is the summary of parameters for the Modified Mohr-Coulomb model.

Parameter	Description	Reference value (kN, m)
Soil stiffness and failure		
E50ref	Secant stiffness in standard drained triaxial test	Ei x (2 – Rf) / 2 (Ei = Initial stiffness)
Eoedref	Tangent stiffness for primary oedometer loading	E50ref
Eurref	Unload / reloading stiffness	3 x E50ref
m	Power for stress-level dependency of stiffness	0.5 ≤ m ≤ 1 (**0.5** for hard soil, 1 for soft soil)
C (C_{inc})	Effective cohesion (Increment of cohesion)	Failure parameter as in MC model
φ	Effective friction angle	Failure parameter as in MC model
ψ	Ultimate dilatancy angle	0 ≤ ψ ≤ φ
Advanced parameters (Recommend to use Reference value)		
Rf	Failure Ratio (qf / qa)	0.9 (< 1)
Pref	Reference pressure	100
KNC	Ko for normal consolidation	1-sinφ (< 1)
Tensile strength	Cut off value for tensile hydrostatic pressure	-
Dilatancy cut-off		
Porosity	Initial void ratio	-
Porosity(Max)	Maximum void ratio	Porosity < Porosity(Max)
Cap yield surface		
OCR / Pc	Over Consolidation Ratio / Pre-overburden pressure	When entering both parameters, Pc has the priority of usage
α	Cap Shape Factor (scale factor of preconsolidation stress)	from KNC (Auto)
β	Cap Hardening Parameter	from Eoedref (Auto)

Soft Soil

The Soft Soil model is suitable for simulation of normally consolidated or near normally consolidated clay soils. The Soft Soil model has the nonlinear elastic characteristic which has the logarithmically relationship between volumetric strain and mean effective pressure. This is the same stress-dependent stiffness with Modified Cam-Clay.

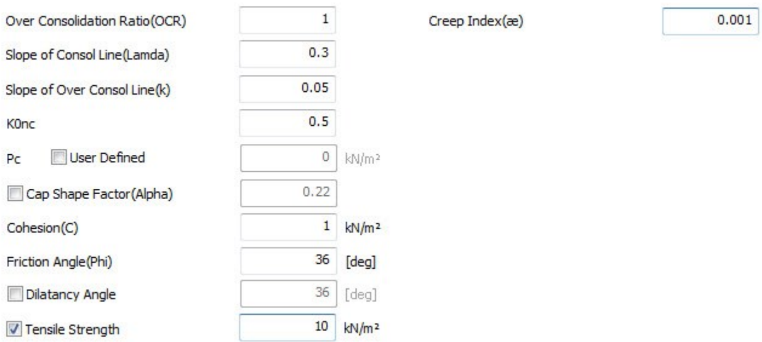


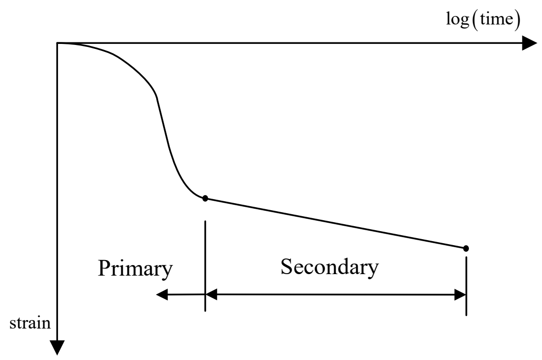
Following is the summary of parameters for the Soft Soil model.

Parameter	Description	Reference value (kN, m)
Soil stiffness and failure		
λ	Compression index	Cc / 2.303
κ	Swelling index	Cs / 2.303 (Cc / 5 for a rough estimation)
c	Cohesion	Failure parameter as in MC model
φ	Friction angle	Failure parameter as in MC model
ψ	Dilatancy angle	0
Advanced parameters (Recommend to use Reference value)		
KNC	Ko for normal consolidation	1-sinφ (< 1)
Cap yield surface		
OCR / Pc	Over Consolidation Ratio / Pre-overburden pressure	When entering both parameters, Pc has the priority of usage
α	Cap Shape Factor (scale factor of preconsolidation stress)	from KNC (Auto)

Soft Soil Creep

This is the model which is extended to a 3D-model based on the 1D-creep theory. In case that time-dependent behavior is critical, this model is applicable to estimate the creep from FE analysis. The stress dependent stiffness parameters can be estimated from compression and recompression index as in **Modified Cam Clay** model. Unlike MCC model, the user can consider secondary consolidation.



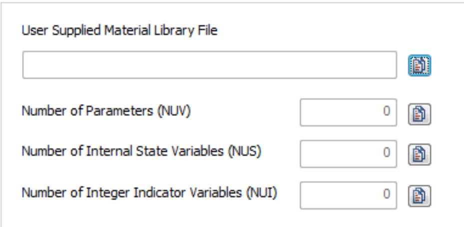


Following is the summary of parameters for the Soft Soil Creep model.

Parameter	Description	Reference value (kN, m)
Soil stiffness and failure		
λ	Compression index	Cc / 2.303
κ	Swelling index	Cs / 2.303 (Cc / 5 for a rough estimation)
μ	Creep index	Cc / 20 for a rough estimation
c	Cohesion	Failure parameter as in MC model
φ	Friction angle	Failure parameter as in MC model
ψ	Dilatancy angle	0
Tensile strength	Cut off value for tensile hydrostatic pressure	-
Advanced parameters (Recommend to use Reference value)		
KNC	Ko for normal consolidation	1-sinφ (< 1)
Cap yield surface		
OCR / Pc	Over Consolidation Ratio / Pre-overburden pressure	When entering both parameters, Pc has the priority of usage
α	Cap Shape Factor (scale factor of preconsolidation stress)	from KNC (Auto)

User Supplied Material

The User Supplied Material utilizes the material model generated by the user using the Pre/Post of the program. The total stress, interior variable, strain and strain increment amount is basically given at the integral points to calculate the stress and tangent slope converged to the User Supplied Material. Nonlinear elastic materials and nonlinear elasto-plastic materials are applied and the supported elements are planar strain, axis symmetry and solid element. Refer to the separately attached 'User Supplied Subroutine (Appendix)" for more information.



User Supplied Material library file
Select the User Specified Model DLL file path on Windows explorer.

Number of parameters
Input the number of parameters needed to define the materials. For example, input the material properties such as Elasticity modulus or Poisson's ratio needed for the User Supplied Material.

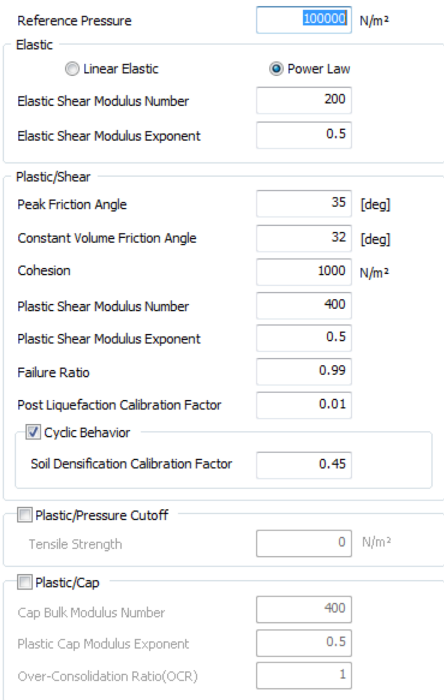
Number of internal state variables
Save the variables needed for the next repetitive calculation. In this case, real type variables are saved such as plastic strain.

Number of integer indicator variables
If the interior state variables were saved as real type variables, the integer indicator variables are saved as integer type variables such as plasticity state index.

Modified UBCSAND

This model is an effective stress model for predicting liquefaction behavior of sand under seismic loading. FEA NX Liquefaction Model is extended to a full 3D implementation of the modified UBCSAND model using implicit method. In elastic region, nonlinear elastic behavior can be simulated, elastic modulus changes according to the effective pressure applied. In plastic region, the behavior is defined by three types of yield functions: **shear (shear hardening), compression (cap hardening), and pressure cut-off**. In case of shear hardening, soil densification effect can be taken into account by cyclic loading.

* Note - **Implicit Method**: Explicit methods calculate the state of a system at a later time from the state of the system at the current time, while implicit methods find a solution by solving an equation involving both the current state of the system and the later one.



Elastic

Shear modulus is updated according to the effective pressure (p') based on the following equation. Allowable tensile stress (Pt) is calculated using cohesion and friction angle automatically. Poisson's ratio is constant and bulk modulus of elasticity will be determined by following relation.

$$G^e = K_G^e p_{ref}\left(\frac{p'+p_t}{p_{ref}}\right)^{ne}, \qquad K^e = \frac{2(1+\nu)}{3(1-2\nu)}G^e$$

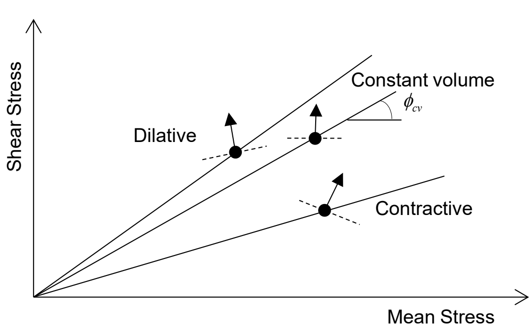
Plastic / Shear

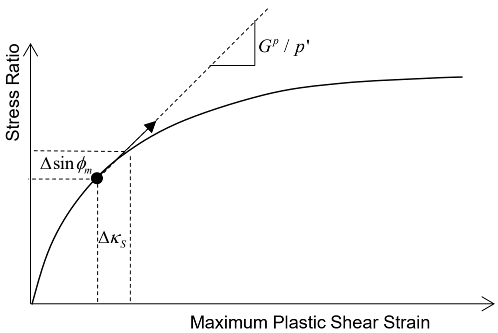
Depending on the difference between mobilized friction angle (Φ_m) and constant volume friction angle (Φ_{cv}), shear induces plastic expansion or dilation is predicted. The Plastic shear strain increment is related to the change in shear stress ratio assuming a hyperbolic relationship and can be expressed as follows.

$$\sin\psi_m = \sin\phi_m - \sin\phi_{cv}, \qquad \Delta\sin\phi_m = \frac{G^p}{p'}\Delta\kappa_s = K_G^p\left(\frac{p'}{p_{ref}}\right)^{np-1}\left\{1-\left(\frac{\sin\phi_m}{\sin\phi_p}\right)R_f\right\}^2\Delta\kappa_s$$

$$\Delta\kappa_s = \left|\Delta\varepsilon_1^p - \Delta\varepsilon_3^p\right|$$

▶Swelling/Shrinkage according to the direction of plastic strain
▶▶Plastic shear hardening behavior





Following is the summary of parameters for the Modified UBCSAND model.

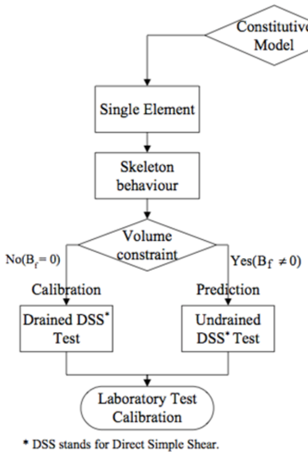
Parameter	Description	Reference
Pref	Reference Pressure	In-situ horizontal stress at mid-level of soil layer
Elastic (Power Law)		
K_G^e	Elastic shear modulus number	Dimensionless
ne	Elastic shear modulus exponent	Dimensionless
Plastic / Shear		
ϕ_p	Peak Friction Angle	Failure parameter as in MC model
ϕ_{cv}	Constant Volume Friction Angle	-
C	Cohesion	Failure parameter as in MC model
K_G^p	Plastic shear modulus number	Dimensionless
np	Plastic shear modulus exponent	Dimensionless
R_f	Failure ratio (qf / qa)	0.7~0.98 (< 1), decreases with increasing relative density
F_{post}	Post Liquefaction Calibration Factor	Residual shear modulus
F_{dens}	Soil Densification Calibration Factor	Cyclic Behavior
Advanced parameters		
Pcut	Plastic/Pressure Cutoff (Tensile Strength)	-
K_B^p	Cap Bulk Modulus Number	-
mp	Plastic Cap Modulus Exponent	-
OCR	Over Consolidation Ratio	Normal stress / Pre-overburden pressure

Model Calibration

Monotonic and cyclic drained Direct Simple Shear (DSS) test (Skeleton Response).

Single element test and calibration using Standard Penetration Test (SPT) - **($(N_1)_{60}$: Equivalent SPT blow count for clean sand**.

▶ Parameters and Equations for Calibration



$$K_G^e = 21.7 \times 20.0 \times (N_1)_{60}^{0.333}$$

$$30^0 < \phi_{cv} < 34^0$$

$$\nu = 0.0163$$

$$K_G^p = K_G^e (N_1)_{60}^{2} \times 0.003 + 100.0$$

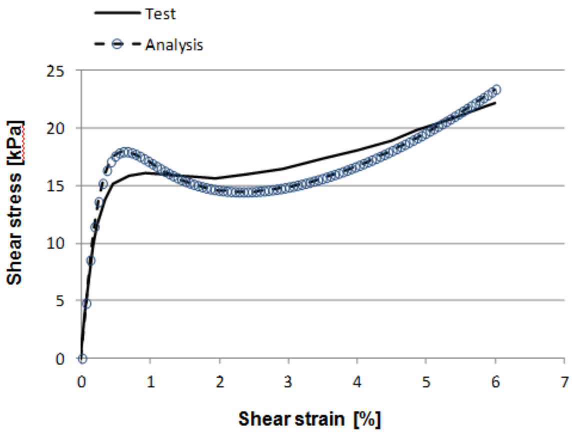
$$ne = 0.5$$

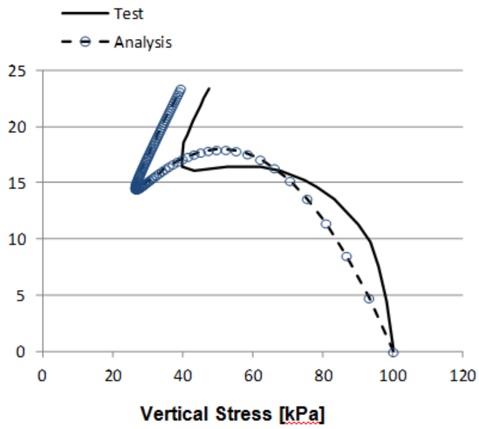
$$np = 0.4$$

$$\phi_p = \begin{cases} \phi_{cv} + (N_1)_{60}/10.0 & ((N_1)_{60} < 15.0) \\ \phi_{cv} + (N_1)_{60}/10.0 + \max\left(0.0, \dfrac{(N_1)_{60} - 15}{5}\right) & ((N_1)_{60} \geq 15.0) \end{cases}$$

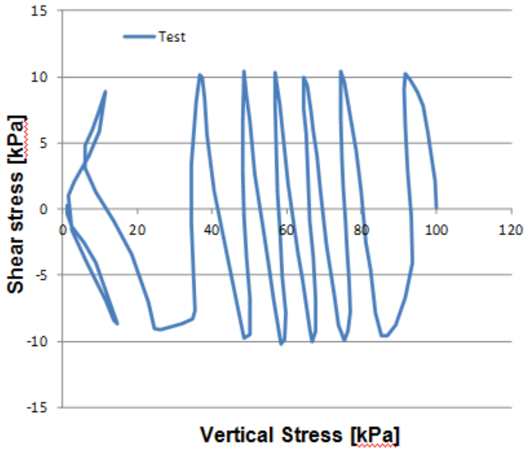
$$R_f = 1.1 \times (N_1)_{60}^{-0.15}$$

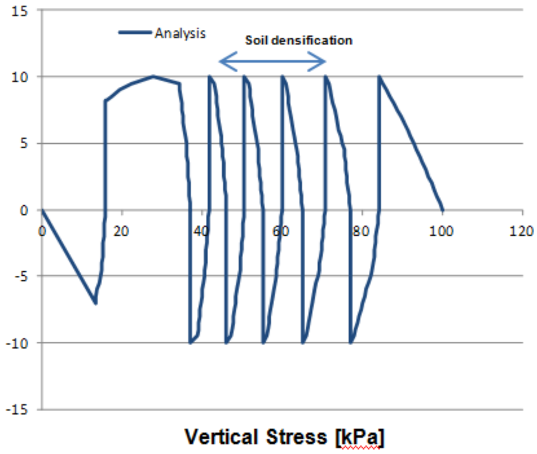
▶ Undrained DSS (Monotonic)





▶ Undrained DSS (Cyclic)





▶Estimation of liquefaction results

Pore Pressure Ratio (PPR)

The ratio of excessive pore pressure change and the initial effective pressure

$$PPR = -\frac{\Delta p_w}{p'_{init}} = \frac{p'_{init} - p'_{current}}{p'_{init}}$$

Δp_w	Excessive Pore Pressure Change
p'_{init}	Initial Effective Pressure
$p'_{current}$	Current Effective Pressure

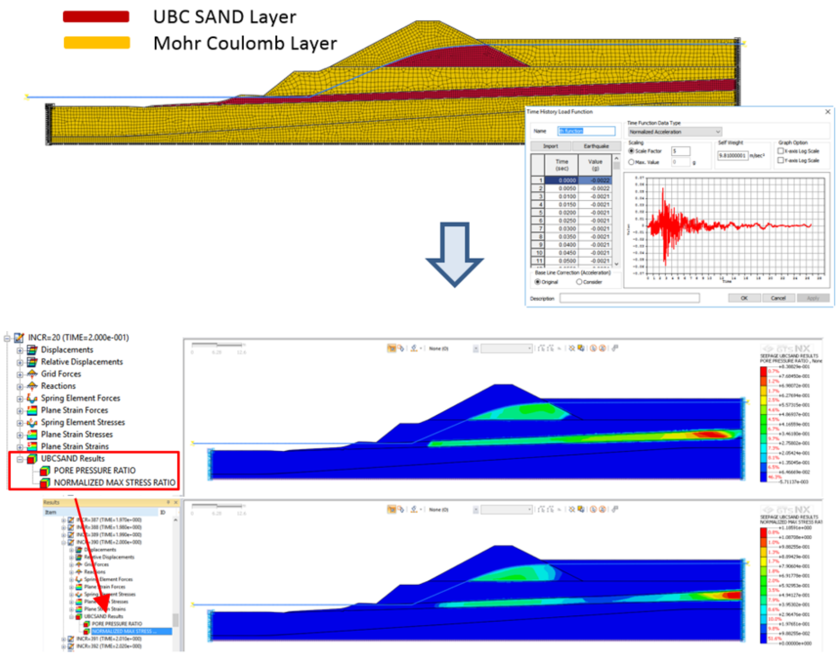
Normalized Max Stress Ratio

The ratio of mobilized friction angle and the peak friction angle

When the Max stress ratio is reached, the mobilized friction angle is close to the peak friction angle, liquefaction is triggered (1 = Liquefaction)

$$\max\left(\frac{\sin\phi_m}{\sin\phi_p}\right)$$

φ'_m	Mobilized Friction Angle
φ'_p	Peak Friction Angle

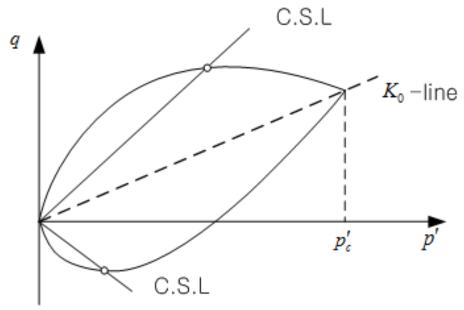


[Nonlinear Time History Analysis under the earth quake]

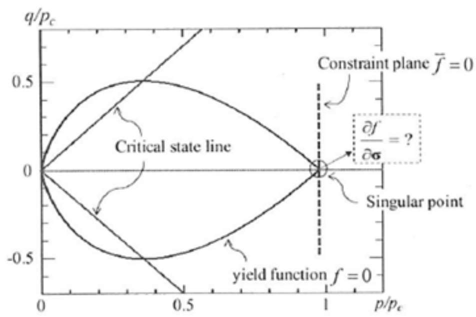
Sekiguchi-Ohta(Inviscid)

This model is one of the Critical state theory model which is similar to Modified Cam Clay model. The yield function has K0 dependent terms, therefore users have to always consider K0 condition (Earth pressure coefficient at rest) for initial stress of ground (K0 Anisotropy is not applicable). As the representative cohesive soil model, this can consider the elasto-plastic behavior, but time-independent one. If K0=1, Original Cam Clay model is equal to Sekiguchi-Ohta model.

▶ Yield Function

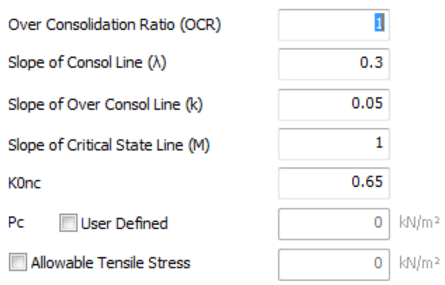


[Sekiguchi-Ohta (Inviscid)]



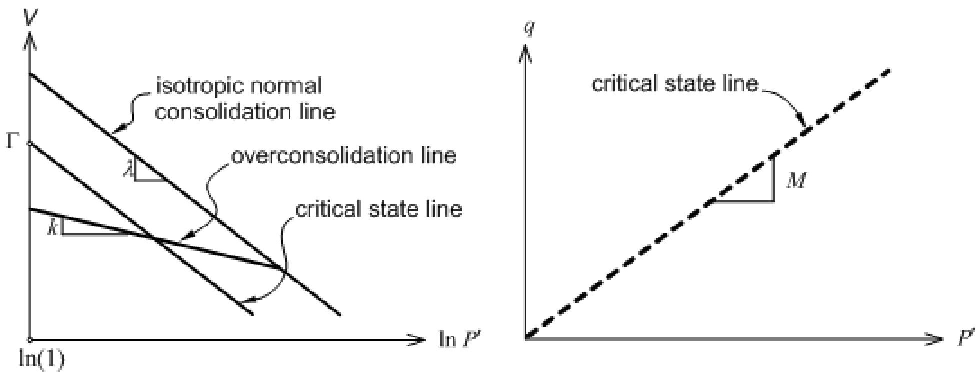
[Cam Clay]

Nonlinear Parameters



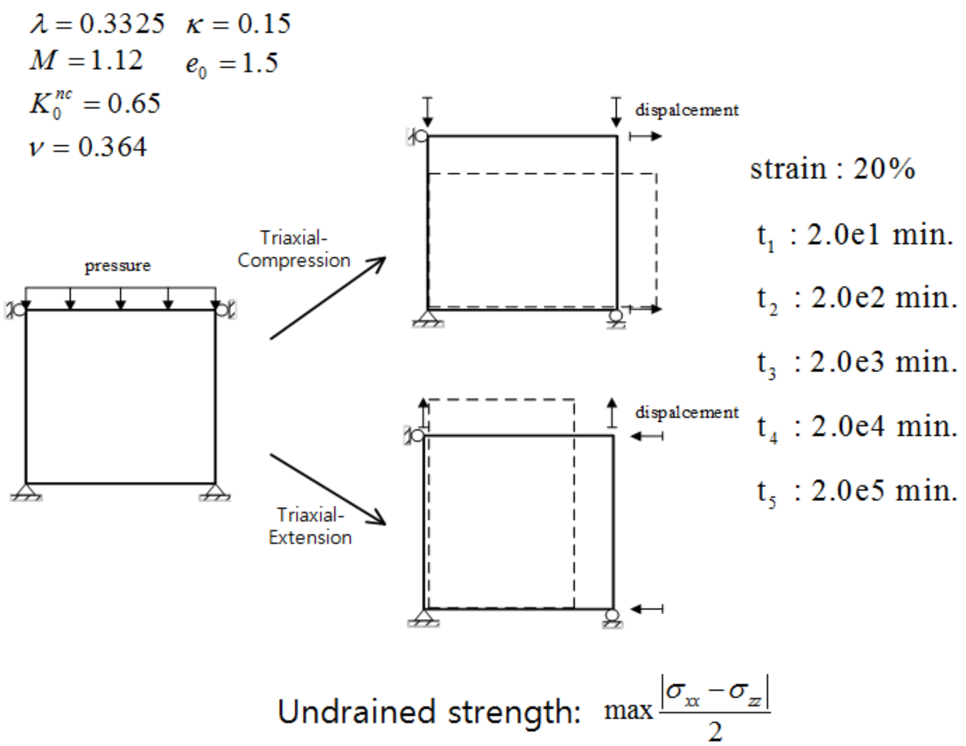
Following is the summary of parameters for the Sekiguchi-Ohta(Inviscid) model.

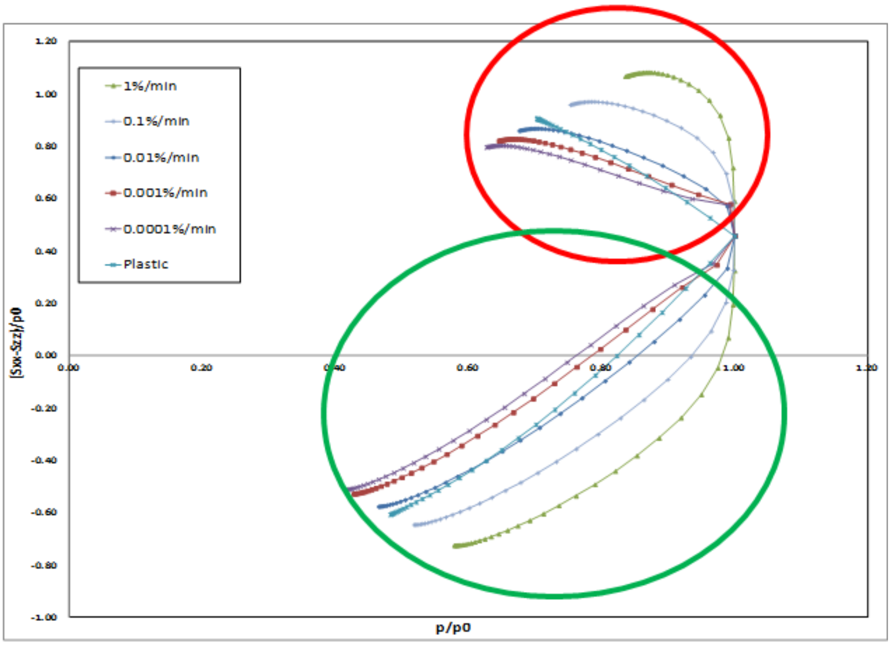
Parameter	Description	Reference value
Nonlinear		
λ	Slope of normal consolidation line	Cc / 2.303
κ	Slope of over-consolidation line	Cs / 2.303 (Cc / 5 for a rough estimation)
M	Slope of critical state line	6 x sinΦ' / (3-sinΦ') (Φ': Effective internal friction angle)
K0nc	K0 for normal consolidation	1-sinφ' (< 1)
Cap yield surface		
OCR / Pc	Over Consolidation Ratio / Pre-overburden pressure	When entering both parameters, Pc has the priority of usage
T_{allow}	Allowable Tensile Stress	* Note

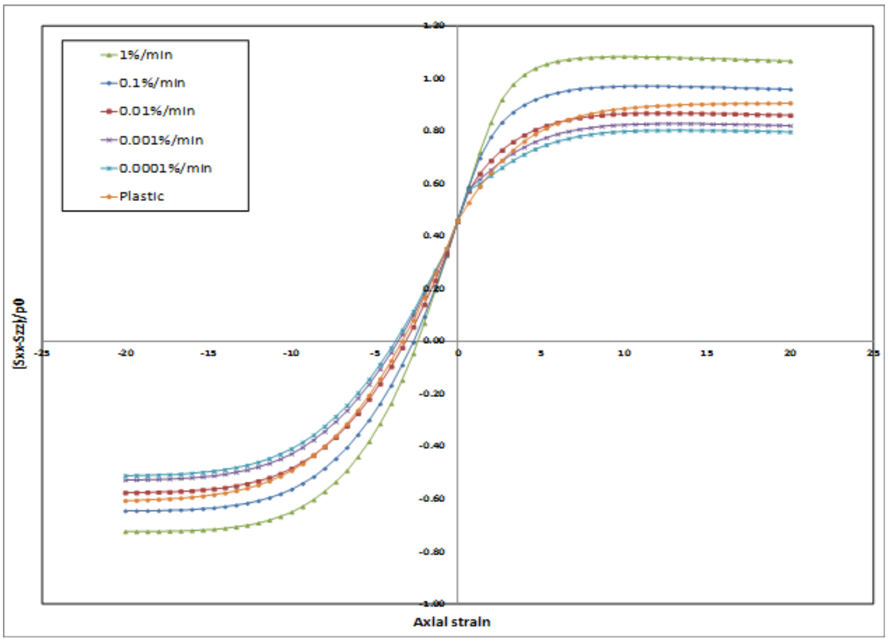


Allowable Tensile Stress
Fundamentally, this model does not allow tensile stress in the failure criteria (stress-strain relationship). However, various conditions can generate tensile stress, such as the heaving of neighboring ground due to embankment load during consolidation or uplift due to excavation. To overcome the material model limits and increase the applicability, analysis on tensile stress within the 'allowable tensile stress' range can be conducted.
The size of the allowable tensile stress is not specified, and requires repeated analysis to input a larger value than the tensile stress created from the overburden load (embankment) or failure behavior. However, when directly entering the pc (pre-consolidation load), the allowable tensile stress cannot surpass the pc value. When defining using the OCR, the pc value is automatically calculated internally by considering the size of the input allowable tensile stress.

Model Calibration
- Undrained Triaxial Compression and Extension test - Effect of Strain Rate.
- Undrained strength depends on the **rate of shearing** in **different ways** on the compressional and extensional sides of shearing.







Sekiguchi-Ohta(Viscid)

This model is one of the Critical state theory model which is similar to Modified Cam Clay model. The yield function has K0 dependent terms, therefore users have to always consider K0 condition (Earth pressure coefficient at rest) for initial stress of ground (K0 Anisotropy is not applicable). As the representative cohesive soil model, this can consider the elasto-plastic behavior, and time-dependent one like soft soil creep model.

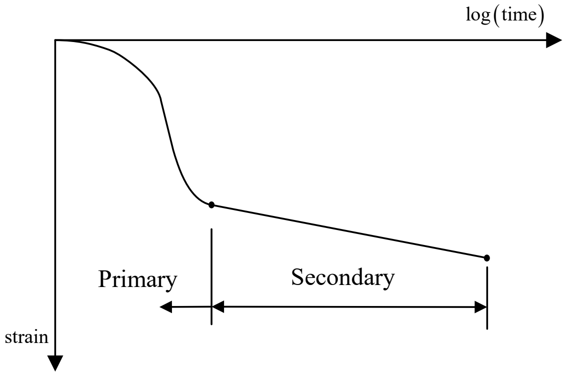
Soft Soil Creep	Sekiguchi-Ohta (viscid)
Always plastic state	Plastic state after yielding

Nonlinear Parameters

Following is the summary of parameters for the Sekiguchi-Ohta(Viscid) model.

Parameter	Description	Reference value
Nonlinear		
λ	Slope of normal consolidation line	Cc / 2.303
κ	Slope of over-consolidation line	Cs / 2.303 (Cc / 5 for a rough estimation)
M	Slope of critical state line	6 x sinΦ' / (3-sinΦ') (Φ': Effective internal friction angle)
K0nc	K0 for normal consolidation	1-sinφ' (< 1)
Cap yield surface		
OCR / Pc	Over Consolidation Ratio / Pre-overburden pressure	When entering both parameters, Pc has the priority of usage
T_{allow}	Allowable Tensile Stress	* Note
Time Dependent		
α	Coefficient of secondary consolidation	Cc / 20 for a rough estimation
$\dot{\varepsilon}_v$	Initial volumetric strain rate	* Note
t_0	Time when primary consolidation ends	* Note

▶ Time Dependent



Allowable Tensile Stress

Fundamentally, this model does not allow tensile stress in the failure criteria (stress-strain relationship). However, various conditions can generate tensile stress, such as the heaving of neighboring ground due to embankment load during consolidation or uplift due to excavation. To overcome the material model limits and increase the applicability, analysis on tensile stress within the 'allowable tensile stress' range can be conducted.

The size of the allowable tensile stress is not specified, and requires repeated analysis to input a larger value than the tensile stress created from the overburden load (embankment) or failure behavior. However, when directly entering the pc (pre-consolidation load), the allowable tensile stress cannot surpass the pc value. When defining using the OCR, the pc value is automatically calculated internally by considering the size of the input allowable tensile stress.

Sekiguchi Ohtamodel requires some material properties, which can be obtained by triaxial tests.
Following empirical relations can be used to estimate the additional soil parameters: **Karibe Method**

Input Parameters	Remarks
Plastic index	I_p
Compression index	C_c
Drainage distance	Unit: cm H
$\lambda = 0.015 + 0.007 I_p$	$\lambda = 0.434 C_c$
$e_0 = 3.78\lambda + 0.156$	
$\sin\phi' = 0.81 - 0.233 \log I_p$	
$\log c_v = -0.025 I_p - 0.25 \pm 1 (cm^2 / \min)$	
$T_v(90\%) = 0.848$	
$\dot{v}_0 = \dfrac{\alpha}{H^2 T_v(90\%) c_v}$	

Model Calibration

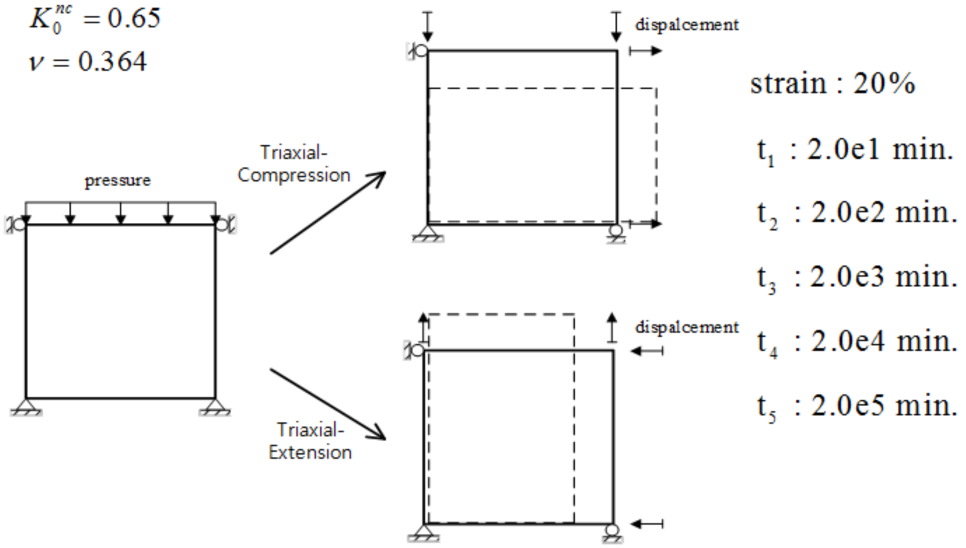
- Undrained Triaxial Compression and Extension test - Effect of Strain Rate.
- Undrained strength depends on the **rate of shearing** in **different ways** on the compressional and extensional sides of shearing.

$\lambda = 0.3325 \quad \kappa = 0.15$

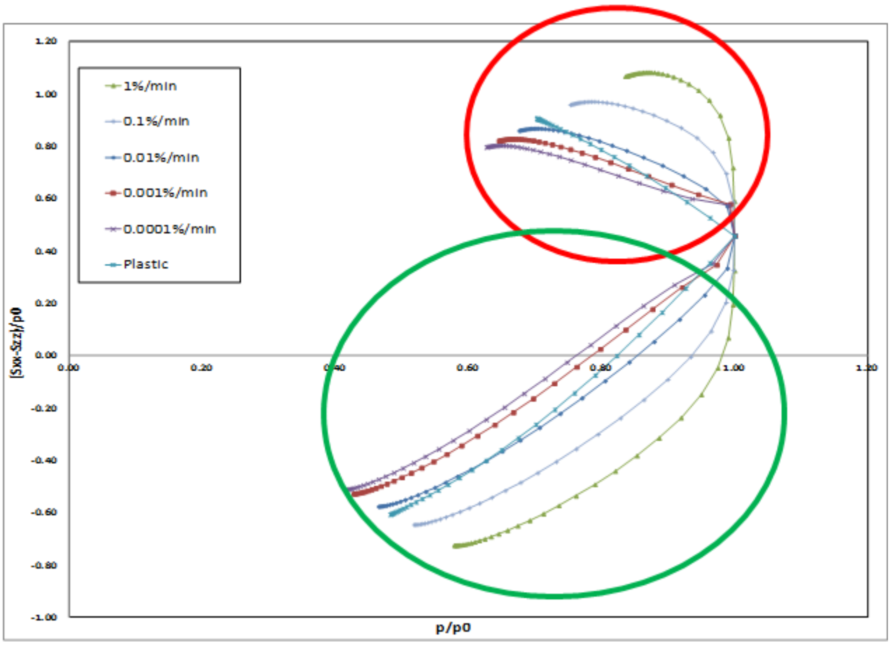
$M = 1.12 \quad e_0 = 1.5$

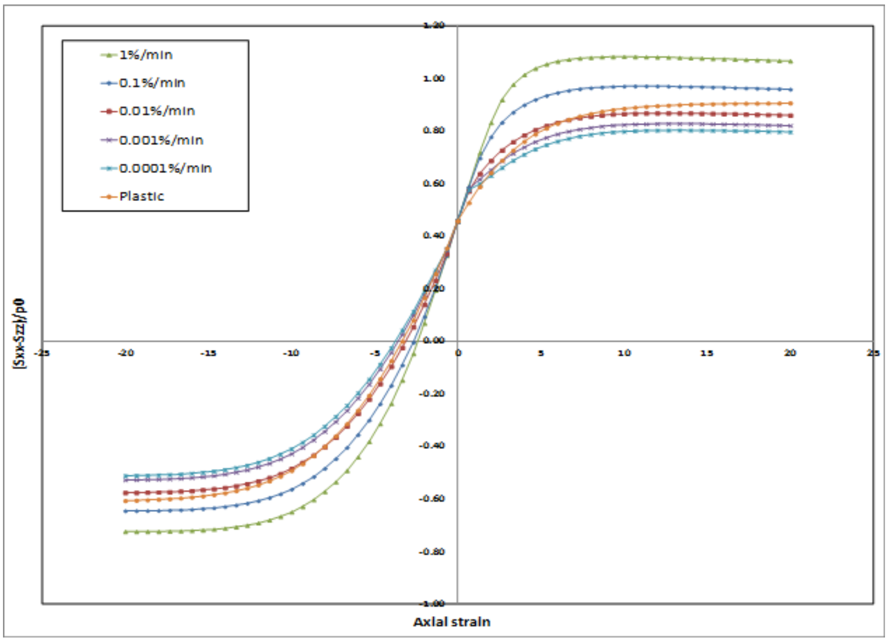
$K_0^{nc} = 0.65$

$\nu = 0.364$



Undrained strength: $\max \dfrac{|\sigma_{xx} - \sigma_{zz}|}{2}$

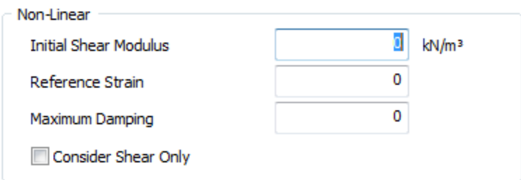




Modified Ramberg-Osgood

This model is one of Hysteresis models for inelastic hinge, an extension was made to 2D and 3D solid elements. This model can be applied to simulate crack or local (plastic) failure. (Refer to inelastic hinge). This model is applicable to all types of nonlinear static and dynamic analysis. (Nonlinear static, Construction Stage, Consolidation, Fully Coupled, SRM (Strength Reduction Method), Nonlinear Time History, Nonlinear Time History + SRM analysis.

Nonlinear Parameters



Following is the summary of parameters for the Modified Ramberg-Osgood model.

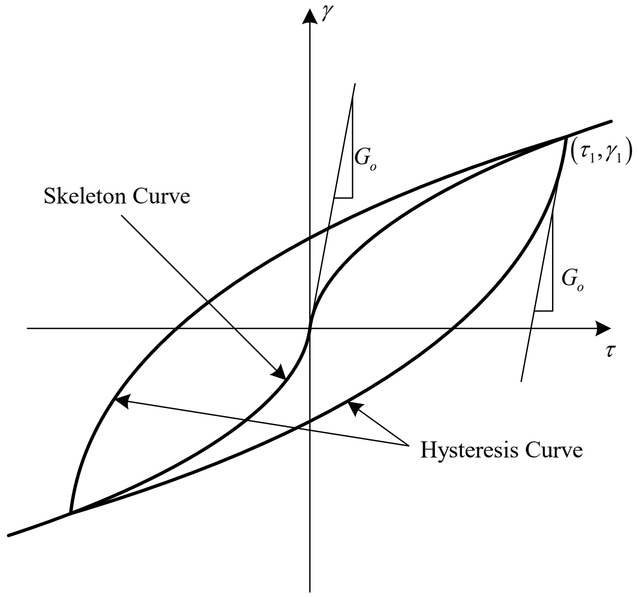
Parameter	Description	Reference
G_o	Initial Shear Modulus	-
γ_r	Reference Strain	$\gamma_r = \frac{\tau}{Go}$
h_{max}	Maximum Damping	0.05 (for soil), $h = \frac{1}{2\pi} \cdot \frac{\Delta W}{W}$
Shear Only	Check: Consider shear modulus for each direction separately (Gxy, Gyz, Gzx) Uncheck: Consider equivalent shear modulus (Geq)	

Yield Function

$$G_o\gamma = \tau + \alpha|\tau|^{\beta}\tau$$

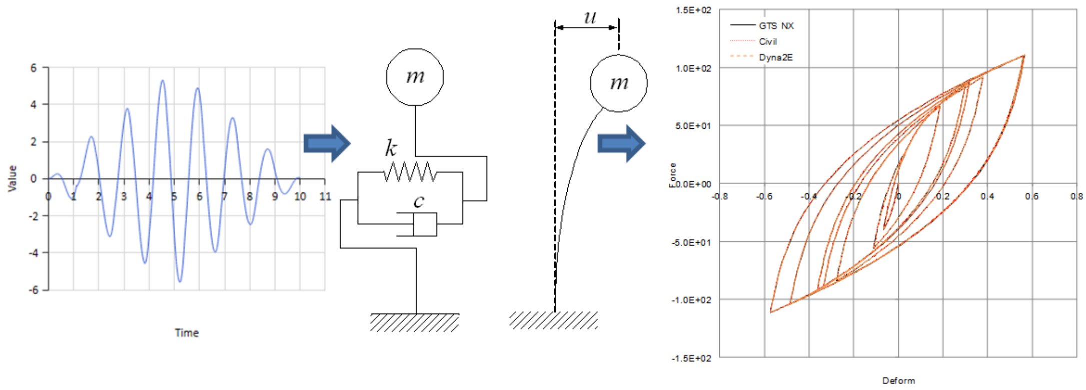
$$\beta = \frac{2\pi h_{\max}}{2-\pi h_{\max}}, \alpha = \left(\frac{2}{\gamma_r G_o}\right)^{\beta}$$

▶Modified Ramberg-Osgood model



Verification Example

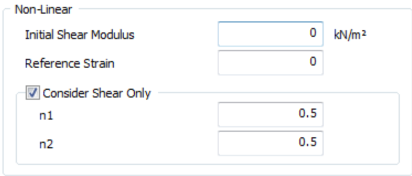
▶Load
▶▶System
▶▶▶Results



Modified Hardin-Drnevich

This model is one of Hysteresis models for inelastic hinge, an extension was made to 2D and 3D solid elements. This model can be applied to simulate crack or local (plastic) failure. (Refer to inelastic hinge). This model is applicable to all types of nonlinear static and dynamic analysis. (Nonlinear static, Construction Stage, Consolidation, Fully Coupled, SRM (Strength Reduction Method), Nonlinear Time History, Nonlinear Time History+SRM analysis. Hysteresis curves are formulated on the basis of the **Masing's rule**.

Nonlinear Parameters



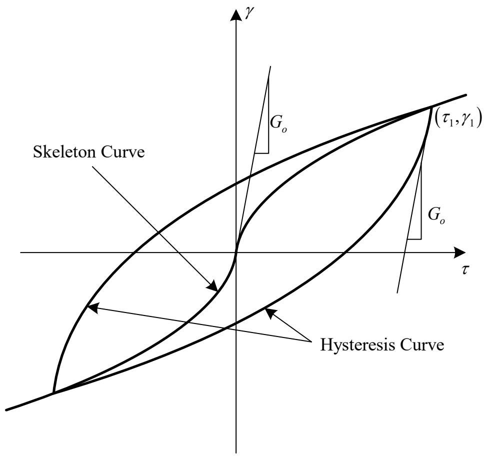
Following is the summary of parameters for the Modified Hardin-Drnevich model.

Parameter	Description	Reference
G_o	Initial Shear Modulus	$\tau = \dfrac{G_0\gamma}{1+\left\lvert\dfrac{\gamma}{\gamma_r}\right\rvert}$
γ_r	Reference Strain	
Shear Only	Check: Consider shear modulus for each direction separately (Gxy, Gyz, Gzx) Uncheck: Consider equivalent shear modulus (Geq) $G_0 = G_{0i} \times \left(\dfrac{\sigma'_{m0}}{\sigma'_{mi}}\right)^{n1}$ $\gamma_{0.5} = \gamma_{0.5i} \times \left(\dfrac{\sigma'_{m0}}{\sigma'_{mi}}\right)^{n2}$	

-n1: The Initial Shear Modulus index (0 < n1 < 1)
-n2: The Reference Shear Strain index (0 < n2 < 1)

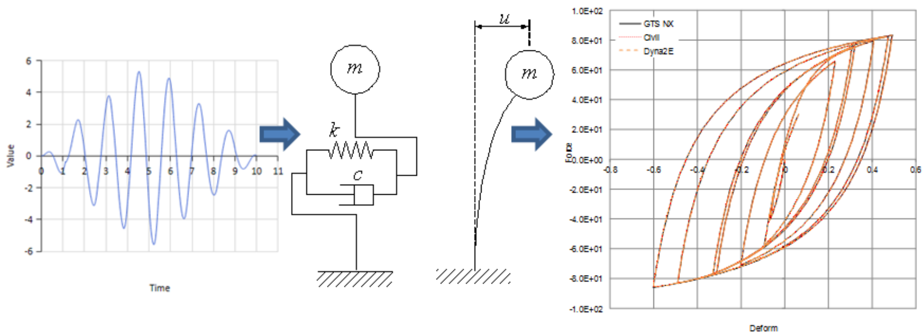
Yield Function

▶Modified Hardin-Drnevich model



Verification Example

▶Load
▶▶System
▶▶▶Results



Hardening Soil (small strain stiffness)

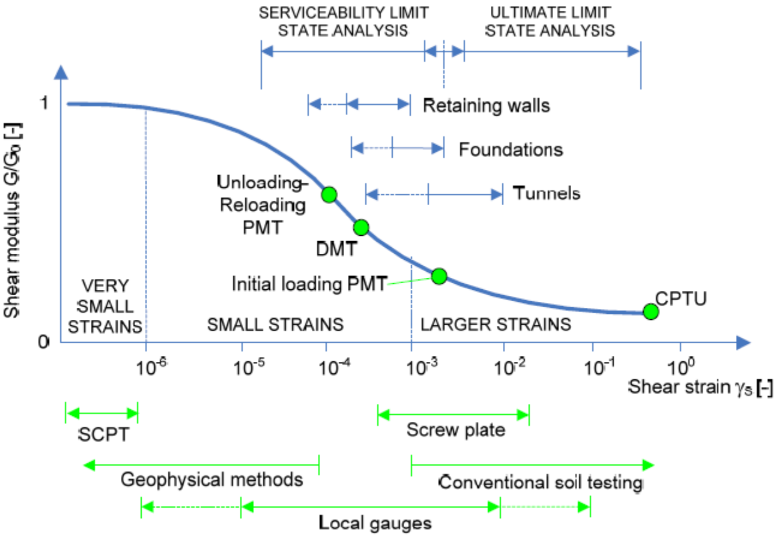
The Hardening Soil with small strain stiffness model is implemented by using the Modified Mohr-Coulomb model and Small strain overlay model, and needed two additional parameters as below:

G_0^{ref} : Initial or very small-strain shear modulus

$\gamma_{0.7}$: Shear strain at which the shear modulus is about 70% of the initial small-strain shear modulus

The strain range in which soils can be considered truly elastic is very small. With increasing strain range, soil stiffness decrease nonlinearly as the following graph.

▶Characteristic stiffness-strain behavior of soil with the ranges for typical geotechnical structures and different tests



To reflect the above characteristics, the Hardening Soil with small strain stiffness model uses the modified Hardin & Drnevich relationship as the following equation.

$$\frac{G_s}{G_0} = \frac{1}{1 + a\left|\frac{\gamma}{\gamma_{0.7}}\right|} \quad ,a = 0.385$$

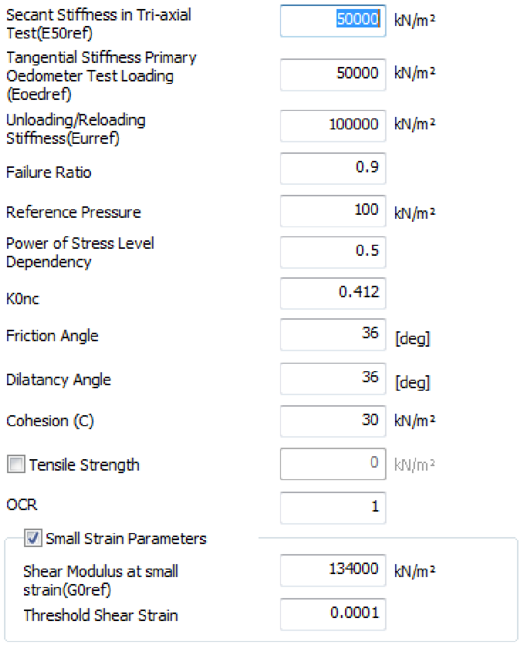
G_s : Shear modulus

G_0 : Initial shear modulus

γ : Shear strain

$\gamma_{0.7}$: Shear strain at which the shear modulus is about 70% of the small-strain shear modulus

Nonlinear Parameters



Following is the summary of parameters for the Hardening Soil (small strain stiffness) model.

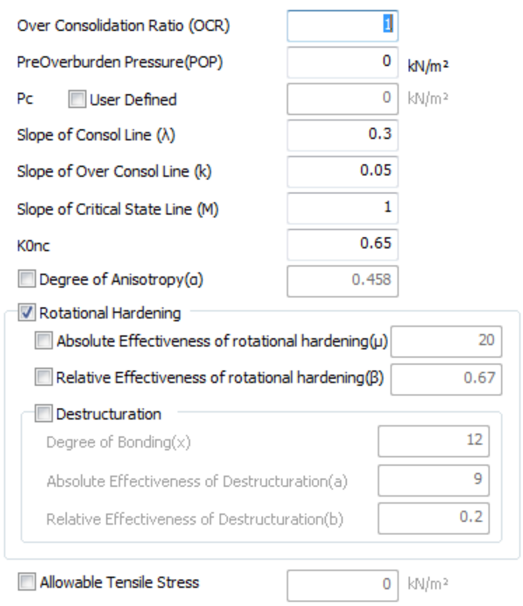
Parameter	Description	Reference value (kN, m)
Soil stiffness and failure		
E50ref	Secant stiffness in standard drained triaxial test	Ei x (2 – Rf) /2 (Ei = Initial stiffness)
Eoedref	Tangent stiffness for primary oedometer loading	E50ref
Eurref	Unload / reloading stiffness	3 x E50ref
m	Power for stress-level dependency of stiffness	0.5 ≤ m ≤ 1 (**0.5** for hard soil, 1 for soft soil)
C	Effective cohesion	Failure parameter as in MC model
φ	Effective friction angle	Failure parameter as in MC model
ψ	Ultimate dilatancy angle	0 ≤ ψ ≤ φ
Advanced parameters (Recommend to use Reference value)		
Rf	Failure Ratio (qf / qa)	0.9 (< 1)
Pref	Reference pressure	100
KNC	Ko for normal consolidation	1-sinφ (< 1)
Tensile strength	Cut off value for tensile hydrostatic pressure	-

Small strain stiffness		
Threshold Shear strain	Shear strain at which shear modulus has decayed to 70% of initial shear stiffness (G0ref)	$\gamma_{0.7} \approx \frac{1}{9G_0}\left[2C(1+\cos 2\varphi)-\sigma'_1(1+K_0)\sin 2\varphi\right]$
β	Shear modulus at small strain	$G_0^{ref} = 33\frac{(2.97-e)^2}{1+e}$

Generalized SCLAY1S

The Generalized S-CLAY1S model is a development of the earlier S-CLAY1 model and is a rotational hardening elasto-plastic model incorporating the influence of bonding and destructuration. The S-CLAY1 model assumes the triaxial stress state whereas the Generalized S-CLAY1S model considers to the general stress state as well. The Generalized S-CLAY1S model has the complex yield surface and needs additional constitutive parameters for anisotropy and destructuration.

Nonlinear Parameters



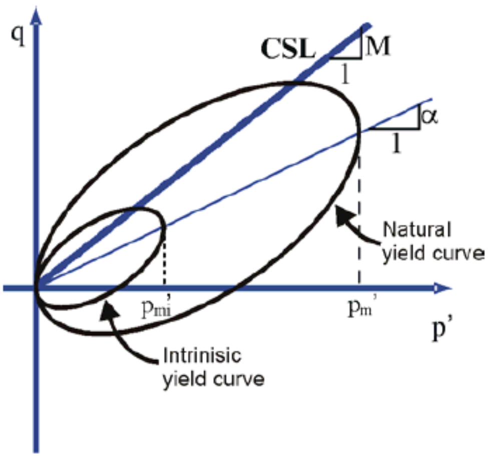
Following is the summary of parameters for the Generalized SCLAY1S model.

Parameter	Description	Reference value (kN, m)
OCR / Pc	Over Consolidation Ratio / Pre-overburden pressure	When entering both parameters, Pc has the priority of usage
POP	Pre-Overburden Pressure	-
λ	Compression index	Cc / 2.303
κ	Swelling index	Cs / 2.303 (Cc / 5 for a rough estimation)

M	Stress ratio at critical state	Triaxial test
K0nc	K0 for normal consolidation	1-sinφ (< 1)
α	Initial inclination of the yield curve	Anisotropy (estimated via φ')
μ	Absolute effectiveness of rotational hardening	Anisotropy (typical values: 10/λ ~ 20/λ)
β	Relative effectiveness of rotational hardening	Anisotropy (estimated via φ')
x	Initial bonding effect	Destructuration
a	Absolute effectiveness of destructurational hardening	Destructuration (typical values: 8~11)
b	Relative effectiveness of destructurational hardening	Destructuration (typical values: 0.2~0.3)

Yield Function

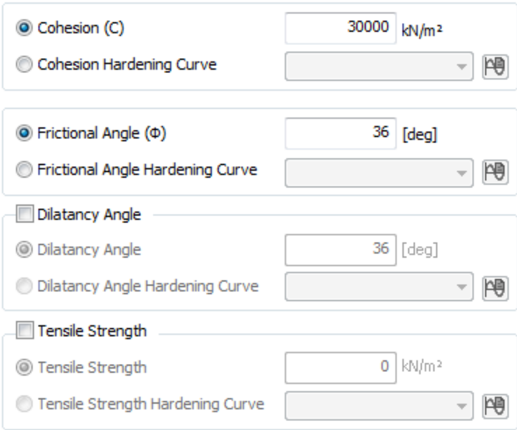
▶Generalized SCLAY1S model

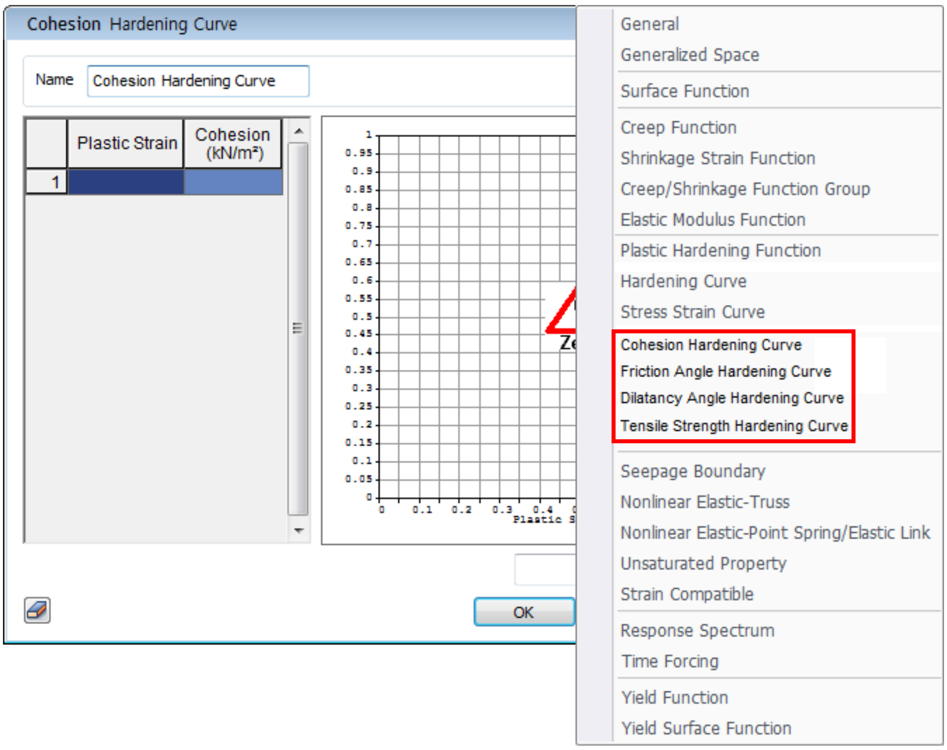


CWFS

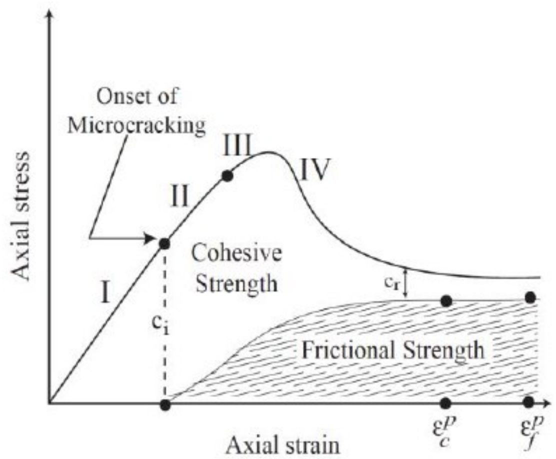
When a tunnel or an underground structure is excavated in deep geological environments, the failure process is affected and eventually dominated by stress-induced fractures growing preferentially parallel to the excavation boundary. This fracturing is generally referred to as brittle failure by spalling and slabbing. Continuum models with traditional failure criteria such as Hoek-Brown or Mohr-Coulomb model have not been successful in prediction of the extent and depth of brittle failure. The cohesion weakening and frictional strengthening (CWFS) model is known to predict brittle failure well. The general conditions (General, Porous and Time Dependent) are same with Mohr-Coulomb model, but the hardening/softening behavior with table using Mohr-Coulomb yield surface can be considered in the nonlinear parameters.

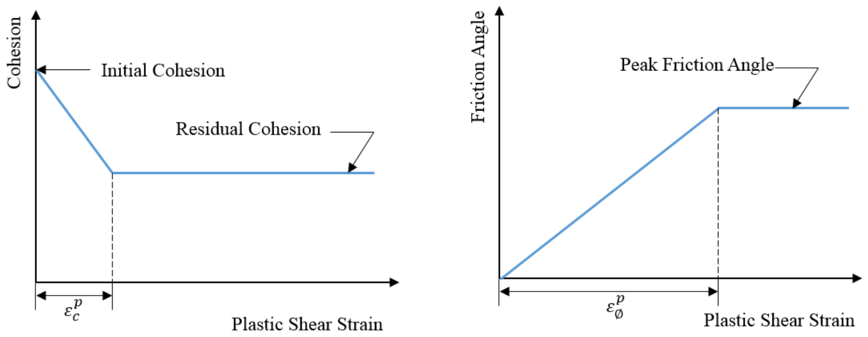
Nonlinear Parameters





Mobilisation of the strength components

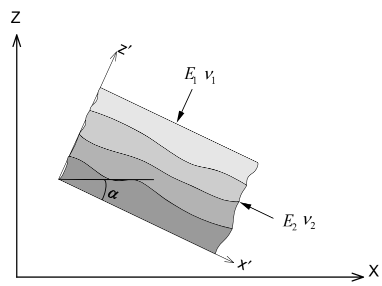




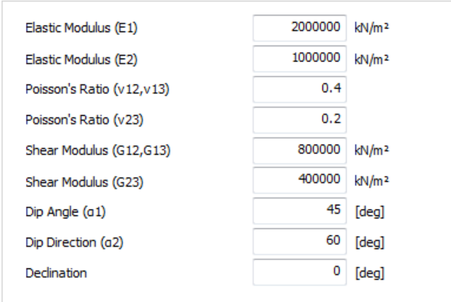
Transversely Isotropic

Natural ground is generally layered and sloped, making it possible to have different strengths in each orthogonal direction. The figure below shows a soil layer with an angle α between the global x axis and the element x' axis and displays perpendicular anisotropy (orthotropy) with the x' axis and z' axis of the element.

▸Orthotropic model



This orthotropy is simulated by assigning different stiffness to the tangent and normal direction to the stratification (fault) orientation. Generally, the normal direction stiffness decreases in comparison to the tangential stiffness and the anisotropic shear strength is defined by the Shear modulus (G). For fully isotropic case, E_1 , ν_1 is equal to E_2 , ν_2 respectively and G is defined by $E_1/2(1+\nu_1)$.



Transversely isotropic materials are material models defined by an isotropic transverse section with a vertical axis to the section. The physical properties are the same within the transverse section and the vertical direction has different properties.

► Out of transverse plane material properties : E_1 , $\nu_{12}(=\nu_{13})$, $G_{12}(=G_{13})$

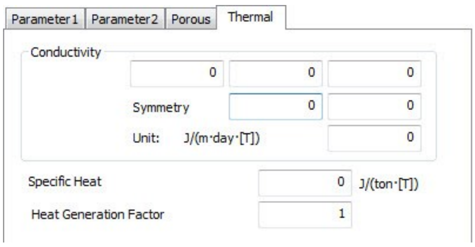
► In transverse plane material properties : $E_2(=E_3)$, ν_{23} , G_{23}

Here, E_1 is the Elasticity modulus in the vertical axis to the section, ν_{12} , ν_{13} and G_{12} , G_{13} are the Poisson's ratio and Shear modulus of the surfaces generated by the vertical and section with the other axes respectively.

The local coordinate system is defined by the dip angle α_1 and dip direction α_2. Because the reference axis of the inclined plane and horizontal plane ($\mathbf{N}$ and X respectively) are not identical, use the auxiliary angle α_3 that subtracts the declination (angle formed between the 2 axes) from α_2 when setting the actual transformation matrix.

$$\alpha_3 = \alpha_2 - declination$$

Thermal



Conductivity: the ability to conduct thermal energy.

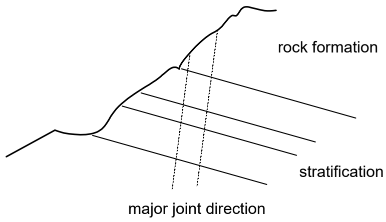
Specific Heat: the amount of heat required to raise single unit mass of a substance by single temperature unit. (required for transient heat transfer problems)

Heat Generation Factor: the value of the heat load multiplied by the exothermic coefficient used as the load vector for heat transfer analysis is the total exothermic load applied to the object.

Jointed Rock Mass

Geo-materials have different material properties depending on the direction and behave differently according to the direction specific conditions. This material property is called Anisotropic. Jointed Rock models pose both transversely isotropic properties and anisotropic plastic properties, making it an Anisotropic elasto-perfectly plastic model. The Jointed rock model is appropriate in modeling the behavior rock formations with joined stratifications as shown below.

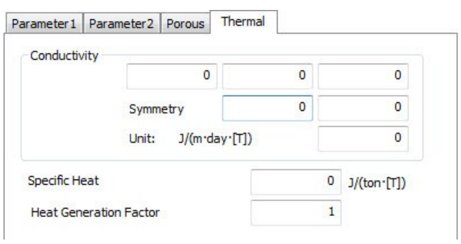
▸ Schematic diagram of Jointed Rock model



It is appropriate when a series of connections or set of connections (joint, Fault etc.) exists. These connection sets must not be filled with fault clay and the spacing must be smaller than the size of the structure. The direction of the connection can be defined in a maximum 3 directions.

The Elasticity modulus, Poisson's ratio, cohesion of each joint, friction angle is the same as that of the Mohr-coulomb model.
The method to define the dip direction, dip angle and deflection angle is the same as the Transversely Isotropic model, but 3 joints can be entered in this model. Here, the anisotropic elastic behavior is defined by the alpha1 and alpha2 defined on the first joint and the other 2 joints are only used when defining plastic deformation.

Thermal



Conductivity: the ability to conduct thermal energy.

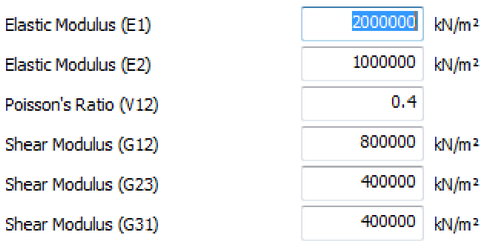
Specific Heat: the amount of heat required to raise single unit mass of a substance by single temperature unit. (required for transient heat transfer problems)

Heat Generation Factor: the value of the heat load multiplied by the exothermic coefficient used as the load vector for heat transfer analysis is the total exothermic load applied to the object.

2D Orthotropic

This model is applicable to 2D element type such as Shell, Plane Stress and 2D Geogrid. Users can define different values of stiffness along each direction, so can define geometrically orthotropic with significant different stiffness in horizontal and vertical direction.

Parameters

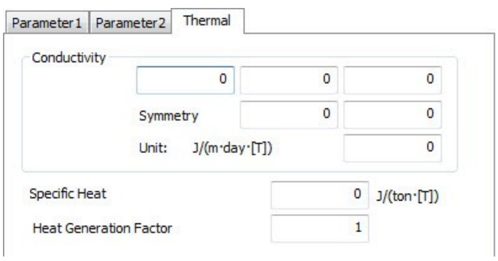


‣Stress-strain relation in 2D

$$\begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} \dfrac{E_1}{1-\nu_{12}\nu_{21}} & \dfrac{\nu_{21}E_1}{1-\nu_{12}\nu_{21}} & 0 \\ \dfrac{\nu_{12}E_2}{1-\nu_{12}\nu_{21}} & \dfrac{E_2}{1-\nu_{12}\nu_{21}} & 0 \\ 0 & 0 & G_{12} \end{bmatrix} \begin{Bmatrix} \varepsilon_{11}-\alpha_{11}\Delta T \\ \varepsilon_{22}-\alpha_{22}\Delta T \\ \gamma_{12} \end{Bmatrix}$$

$$\begin{Bmatrix} \tau_{31} \\ \tau_{23} \end{Bmatrix} = \begin{bmatrix} G_{31} & 0 \\ 0 & G_{23} \end{bmatrix} \begin{Bmatrix} \gamma_{31} \\ \gamma_{23} \end{Bmatrix}$$

Thermal

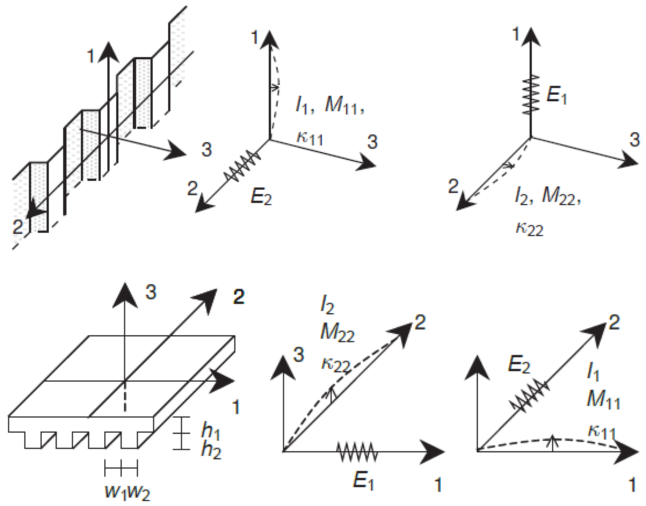


Conductivity: the ability to conduct thermal energy.

Specific Heat: the amount of heat required to raise single unit mass of a substance by single temperature unit. (required for transient heat transfer problems)

Heat Generation Factor: the value of the heat load multiplied by the exothermic coefficient used as the load vector for heat transfer analysis is the total exothermic load applied to the object.

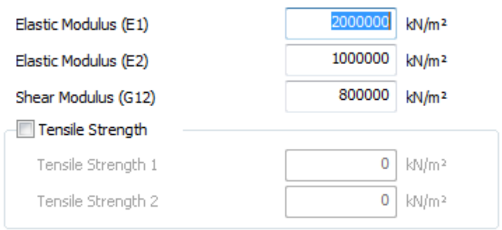
▸Engineering example for geometrical orthotropic



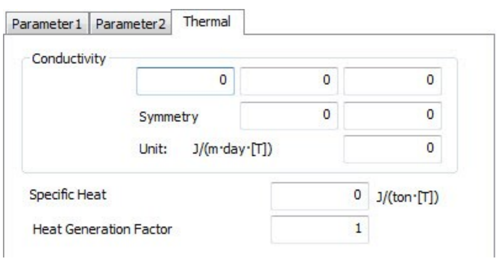
Geogrid

The Geogrid model is an orthotropic material which has tension only behavior and can be only assigned to 1D/2D Geogrid property. 1-direction and 2-direction behave independently each other. It shows tension only nonlinear elastic behavior without the 'Tensile Strength' option, and it shows plastic behavior under load conditions that exceed the tensile strength if this option is selected. In case of 1D Geogrid element, E2, G12 and Tensile Strength 2 are not considered.

Parameters



Thermal



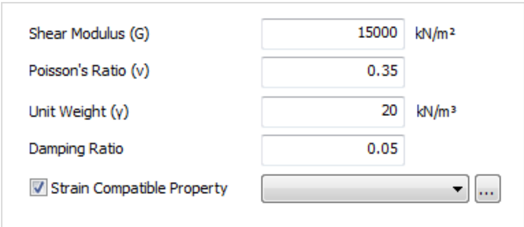
Conductivity: the ability to conduct thermal energy.

Specific Heat: the amount of heat required to raise single unit mass of a substance by single temperature unit. (required for transient heat transfer problems)

Heat Generation Factor: the value of the heat load multiplied by the exothermic coefficient used as the load vector for heat transfer analysis is the total exothermic load applied to the object.

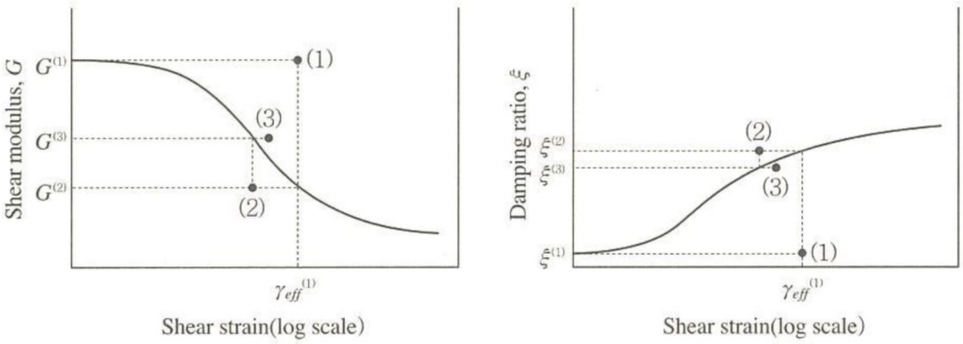
2D Equivalent

Is a material that simplifies complex nonlinear behavior of geo-material properties into equivalent linear properties and allows ground-structure analysis under dynamic loading. Shear strain due to earthquakes or other dynamic loading is constantly changing and the effective shear strain concept is used to set an equivalent linear value. The required input parameters and application process are as follows.



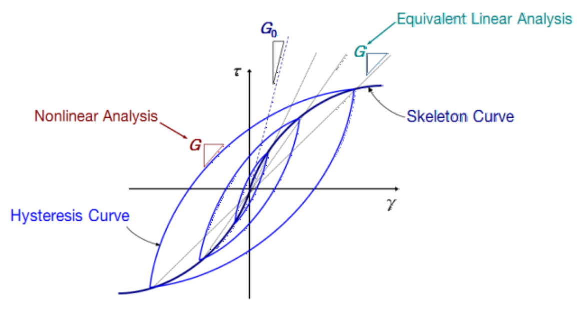
Process	Analysis process
1	Assume initial Shear modulus and damping ratio
2	Compute ground response and strain hysteresis from initial values
3	Use maximum strain from computed strain hysteresis to calculate effective strain
4	Use equivalent linear damping ratio corresponding to effective strain and Shear modulus to re-compute the ground response and strain hysteresis
5	Repeat process 2~4 until the calculated Shear modulus and damping ratio error variation is within the tolerance

▶Convergence of nonlinear Shear modulus and damping factor using the equivalent linear method

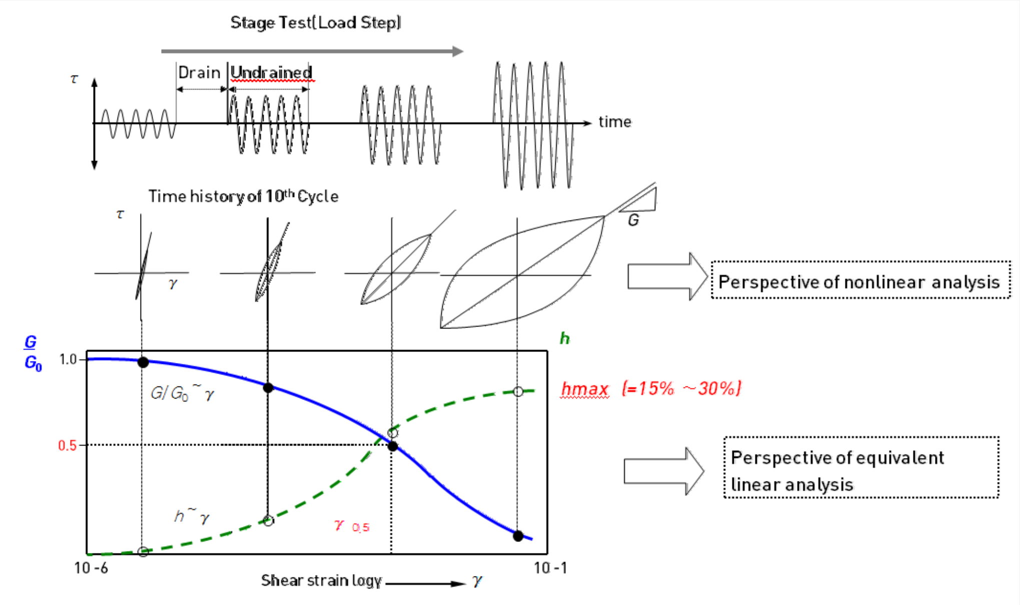


Tip

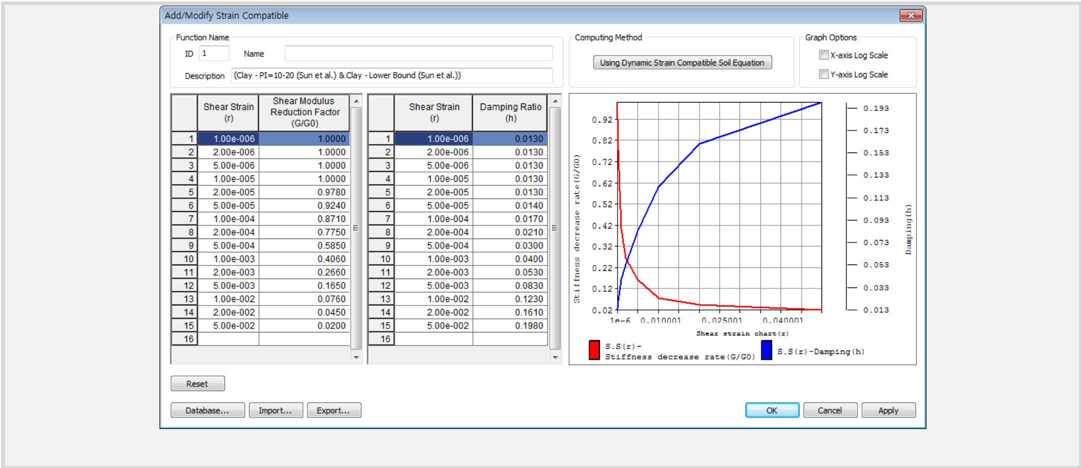
The stress-strain relationship graph for equivalent linear analysis and nonlinear analysis can be expressed as follows.



In the equivalent linear method, the nonlinear characteristics of the geo-material can be expressed as a function of the ratio of maximum shear modulus and shear modulus G/G_0 and shear strain γ, and a function of damping ratio h and shear strain γ. These material properties can be found from the dynamic strain test as shown below.



Strain compatibility properties can be set using a function of Shear modulus and damping ratio to strain when considering the nonlinear, inelastic behavior of the ground. If the function is not defined, the geo-material is assumed to be linear and an entered (fixed) Shear modulus and damping ratio is used in the analysis. Various DB exist for each type of ground. Refer to 'Function' section for more detailed information.


Add/Modify Strain Compatible
Function Name
ID 1
Name
Description (Clay - PI=10-20 (Sun et al.) & Clay - Lower Bound (Sun et al.))
Computing Method
Using Dynamic Strain Compatible Soil Equation
Graph Options
X-axis Log Scale
Y-axis Log Scale
Shear Strain (r)
Shear Modulus Reduction Factor (G/G0)
Shear Strain (r)
Damping Ratio (h)
1 1.00e-006 1.0000 1 1.00e-006 0.0130
2 2.00e-006 1.0000 2 2.00e-006 0.0130
3 5.00e-006 1.0000 3 5.00e-006 0.0130
4 1.00e-005 1.0000 4 1.00e-005 0.0130
5 2.00e-005 0.9780 5 2.00e-005 0.0130
6 5.00e-005 0.9240 6 5.00e-005 0.0140
7 1.00e-004 0.8710 7 1.00e-004 0.0170
8 2.00e-004 0.7750 8 2.00e-004 0.0210
9 5.00e-004 0.5850 9 5.00e-004 0.0300
10 1.00e-003 0.4060 10 1.00e-003 0.0400
11 2.00e-003 0.2660 11 2.00e-003 0.0530
12 5.00e-003 0.1650 12 5.00e-003 0.0830
13 1.00e-002 0.0760 13 1.00e-002 0.1230
14 2.00e-002 0.0450 14 2.00e-002 0.1610
15 5.00e-002 0.0200 15 5.00e-002 0.1980
16 16
Stiffness decrease rate(G/G0)
Damping(h)
Shear strain chart(r)
S.S(r)-Stiffness decrease rate(G/G0)
S.S(r)-Damping(h)
Reset
Database...
Import...
Export...
OK
Cancel
Apply

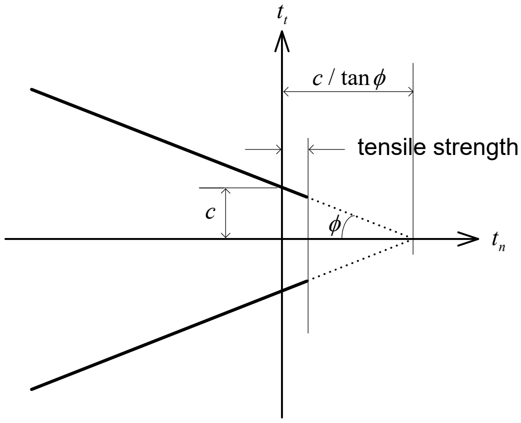
Interface (Interface / Shell Interface)

Interface

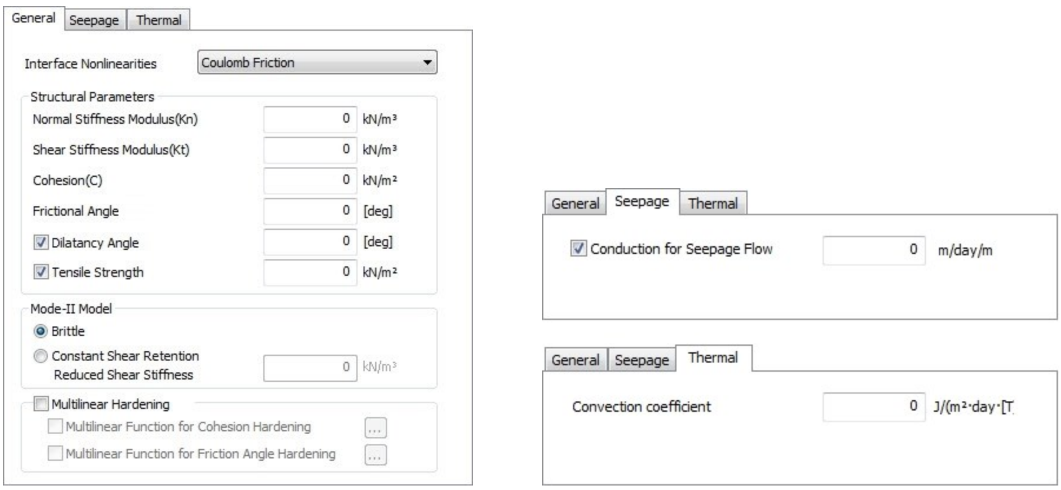
The interface behavioral model was developed to simulate the boundary (interface) behavior between same or different materials. The interface behavioral model is not only used in geo-technology but also throughout architecture and civil engineering in general to define the behavior of various interfaces. The interface behavioral model is based on **Coulomb's law of friction** (1785) and follows the assumption that the frictional force of an interface is proportional to the coefficient of friction and the confining forces perpendicular to the normal direction acting on the interface.

This model is mostly used to simulate rock joints or structure-ground interfaces such as friction pile-ground interface, earth retaining wall-ground interface, lining-ground interface etc.

▸Coulomb Friction function



The main nonlinear parameters of the interface model are as follows. The user can also define the coefficient of permeability or stiffness to simulate interface behavior.

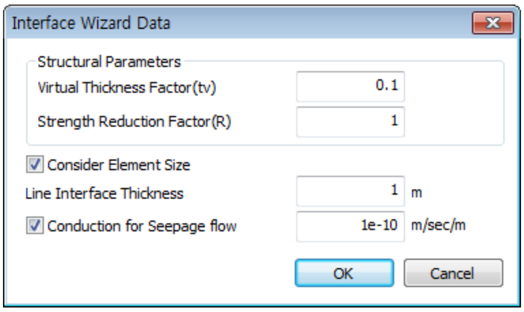


[Normal stiffness modulus (Kn)]

Is the Elasticity modulus for bonding and un-bonding behavior in the normal direction to the interface element. The general range is 10~100 times the smallest value of the Elasticity modulus on the oedometer of adjacent elements.

[Shear stiffness modulus (Kt)]
Is the Elasticity modulus for slip behavior in the normal direction to the interface element. The general range is 10~100 times the smallest value of the shear strength of adjacent elements.

The nonlinearity of the interface needs to be computed by applying the Coulomb Friction criterion and using the stiffness parameters along with experimentation (relative displacement-frictional force curve), but an empirical formula can be used to predict the interface behavior between 2 materials. The empirical formula uses a virtual thickness (tv) and strength reduction factor (R). When creating an interface element, the following Wizard can be used for automatically calculate, according to the element properties of the neighboring ground element, using the 2 parameters (tv, R).



Tip

The interface material can be defined using the following equation. Using the stiffness of adjacent elements and nonlinear parameters, the virtual thickness (tv) and strength reduction factor (R) is applied. R x (Fn + Ft x tan(phi) - C) = 0 → R x (Kn x un + Kt x ut x tan(phi) - C) = 0
The Wizard can be used to simplify this process.

$$K_n = E_{oed,i} / t_v$$
$$K_t = G_i/t_v$$
$$C_i = R \times C_{soil}$$
$$phi_i = \tan^{-1} (R \times \tan (phi_{soil}))$$

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

(v_i =Interface Poisson's ratio=0.45, the interface is used to simulate the non-compressive frictional behavior and automatically calculates using 0.45 to prevent numerical errors.)
t_v = Virtual thickness(Generally has a value between 0.01~0.1, the higher the stiffness difference between ground and structure, the smaller the value)
$G_i = R \times G_{soil}$ ($G_{soil} = E/(2(1+ v_{soil}))$), R = Strength Reduction Factor

The general Strength reduction factor for structural members and neighboring ground properties are as follows.

- Sand/Steel : R ≈ 0.6~0.7
- Clay/Steel : R ≈ 0.5
- Sand/Concrete : R ≈ 0.8~1.0
- Clay/Concrete : R ≈ 0.7~1.0

In case of multiple soil layer the same structural component, the smaller value of R is recommended.

Checking the Element size consideration calculates the interface material properties considering the average length(line), average area(face) of the neighboring ground element when creating an interface.

In other words, the average length(l), average area(A) are multiplies to the virtual thickness in the equation below to calculate the tangent, normal direction stiffness of the interface.

Kn = Eoed,i / (l or $\sqrt{A}$ x tv) , Kt = Gi / (l or $\sqrt{A}$ x tv)

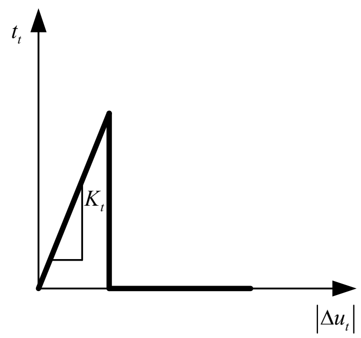
If the consideration is not checked, the unit length(area) is applied.
The thickness is defined separately for a line interface. The thickness is an important element when using the interface on a ground material that displays hardening behavior. Generally, the neighboring ground particle size is input, but if an accurate numerical value is not available, the default value from the program is used. For a 3D model, like the 1 in the example above, the surface interface does not need a thickness.

When defining the stiffness against seepage for an interface element, the "permeability coefficient" can be defined to be the same as the permeability coefficient of the ground. If the option is not checked, the layer is considered to be impermeable.

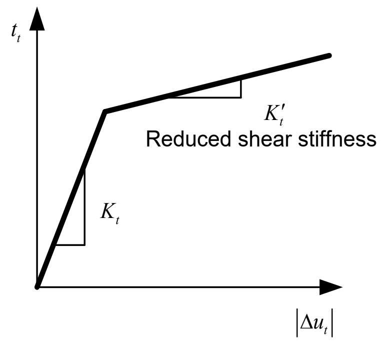
When the dilatancy angle and tensile strength is defined, a smaller or equal value needs to be defined for the interface element and the cohesion; friction angle can be multiplied with the strength reduction factor. For the interface dilatancy angle, the same angle can be applied as the ground when the ground is under rigid body motion without strength reduction (R=1). When considering strength reduction, entering '0(zero)' is the general definition for rigid body motion.

[Mode-II Model]
The Mode-II model expresses shear behavior and defines the tangential slip behavior or the interface. For the 2 models below, the failure envelopes are shown for when the 'Constant Shear Retention' function is considered suitable in terms of numerical analysis stability etc.

- Brittle model
  The structure does not receive any loading if the vertical force is higher than the tensile strength.



- Constant Shear Retention
  Apply the input value on the shear direction such that the structure can receive loading in that direction.



K_t' : Reduced Shear Stiffness

[Multi-linear Hardening]

If a function is entered in the multi-linear hardening, the cohesion and friction angle used in the Coulomb friction failure criterion changes with plastic displacement. Note that the cohesion and friction angle both need to increase as the plastic displacement increases. This behavioral characteristic must be defined by experimentation and is mainly used for research purposes than practical purposes.

Seepage

Conduction for Seepage flow:Sets allowable flow rate at the interface.

Thermal

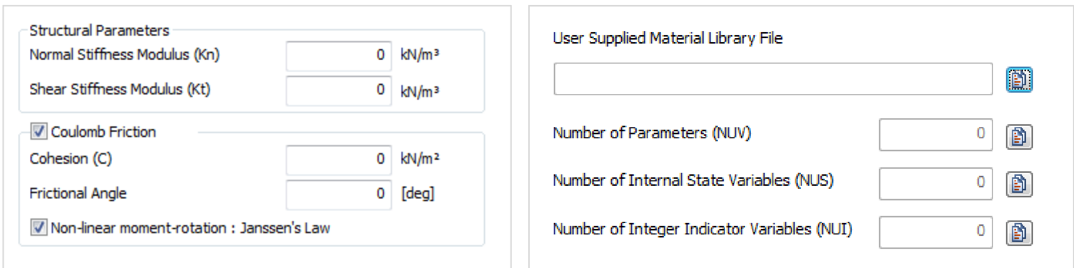
Convection coefficient:Controls allowable heat exchange at the interface.

Shell Interface

The interface element was developed to simulate the interface behavior just like a general face element. Here, the interface element is also capable of resisting the rotational force between plates.

- Tensile force is not transmitted to the load or moment.
- Linear behavior is observed for small rotations or shear force.
- Nonlinear elastic behavior for large rotations. (Janssen's law)
- Plastic behavior for large shear force. (Coulomb friction)

Nonlinear behavior at the plate interface element follows the Coulomb friction law for movement and Janssen's law for rotation. The relative move displacement and interface force follows the Coulomb friction model with some restrictions. The tensile strength is set as 0 for the Tension Cut-off function, the dilatancy angle and interior friction angle are identical and the asymmetrical material property matrix is not defined. The stiffening function does not need to be defined separately.



For the User supplied shell interface, it is the same as the User Supplied Material model.

Pile (Pile & Pile Tip)

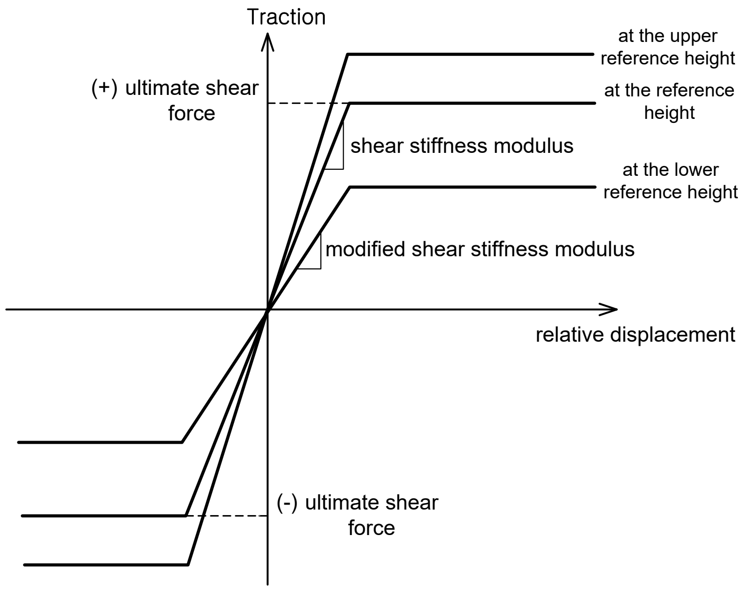
The pile element behavior is the interfacial behavior between the parent element and pile elements such as a beam or a truss. The interfacial behavior for a pile can divided into 2 normal direction behavior and 1 tangent direction behavior. Like the interface element, the shear/vertical stiffness is defined for simulation of interface behavior but here, it is assumed that the 2 normal direction (vertical) motions undergoes identical rigid body motion as the parent element and the tangent direction (shear) behavior undergoes nonlinear elastic motion.

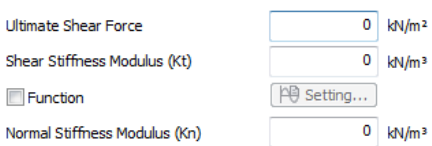
The pile tip element behavior is the interfacial behavior between the parent element and the 1 tip of the pile element. It is assumed that the normal direction behavior with respect to the element coordinate axis at the pile tip undergoes identical rigid body motion as the parent element and the tangent direction behavior undergoes nonlinear elastic motion. For the pile element, it is assigned either multiple curves as a nonlinear material model or a value for fully plastic behavior.

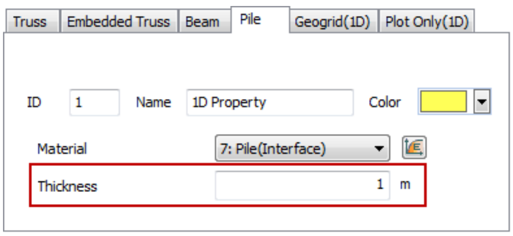
Pile and pile tip elements express nonlinear behavior through default stiffness as well as bearing power or ultimate strength. The nonlinear behavior can be defined by functions and a 3D table is supported for piles to define different function for different depths.

When defining the shear stiffness of the pile using yield strength, assume that the slope and yield strength undergoes fully elasto-plastic behavior depending on height as shown below. The user can also define the strain-frictional force relationship curve for different heights when defining the stiffness as a function.

▸Relative displacement-Frictional force relationship







Input a certain coefficient for the normal direction. The nonlinear analysis is analyzed linearly. For the shear direction, use the input coefficient and analyze if linear. For nonlinear analysis, also consider the ultimate shear strength and analyze as a fully plastic model. If the shear stiffness is differs for each depth, a function must be used.

Ultimate Shear Force : Input the ultimate shear resistance force(kN) of the pile for a load in the axial direction in stress units, by dividing it with the pile length(m) and pile element thickness(m). The frictional force of the pile is output as the force per unit length(kN/m) and the Ultimate frictional force becomes the input [Ultimate shear force(kN/m2) x Pile element thickness(m)].

Shear Stiffness Modulus (Kt) : The slope of the linear section of the friction stress(kN/m2)-relative displacement(m) curve that has the units of kN/m3. When the analysis results of the tangent direction frictional force(kN/m) and relative displacement(m) of a pile is drawn with respect to the applied load size, the graph has a linear slope(kN/m2) until the ultimate frictional force and this becomes the input [Shear stiffness modulus(kN/m3) x Pile element thickness(m)].

Function : To specify the nonlinear elastic behavior in the tangent direction, define the Friction stress(kN/m2)-Relative displacement(m) curve for each depth, rather than entering the Ultimate shear force and Shear stiffness modulus.

Normal Stiffness Modulus (Kn) : The slope of the linear section of the relationship graph between the ground resistance to a horizontally applied force(kN), which is expressed as stress by dividing it with the pile length(m) and pile element thickness(m), and the relative displacement. It is the same concept as the Lateral subgrade reaction modulus, calculated from general p-y analysis. When the analysis results of the tangent direction frictional force(kN/m) and relative displacement(m) of a pile is drawn with respect to the applied load size, it is the linear slope(kN/m2), which can be expressed as the input [Shear stiffness modulus(kN/m3) x Pile element thickness(m)].
When using the Lateral subgrade reaction modulus formula proposed by the design code, input the calculated coefficient into the Normal Stiffness Modulus and the pile element thickness can be input as a unit width(1m).

Thermal
Convection coefficient: Controls allowable heat exchange at the interface.

Tip

Because the pile size, length, neighboring ground material properties all affect the pile element parameters, it is ideal to use the results of a loading test. However, if there are no test results, the Ultimate Shaft Resistance, Lateral Subgrade Reaction Modulus and End Bearing Capacity of a pile can be calculated using the formulas proposed by the design code and the neighboring ground parameters (unit weight, cohesion, friction angle etc.).

1. Using loading test results
For example, if a load of 1000kN was found before failure from the pile loading test and the pile length was 10m, the Ultimate shear force is [1000kN/10m/1m] = 100kN/m2. Here, the 1m is the unit length of the input pile element thickness.
The Shear stiffness modulus is the slope of the linear section on the relationship graph with relative displacement until the 1000kN load is applied. If we assume that the relative displacement at 1000kN is 0.01m, the Shear stiffness modulus becomes [100kN/m2 / 0.01m] = 10000kN/m3.

2. Using the Ultimate Shaft Resistance results

For each design code, various formulas are suggested to predict the Ultimate bearing capacity of a pile according to ground and pile section properties. For example, if the calculated Ultimate shaft resistance is 50kN/m2 and the Shaft surface area of the pile(equivalent circumference) is 3m, input 50kN/m2 for the Ultimate shear force and 3m for the Pile element thickness respectively, or input 150kN/m2 for the Ultimate shear force and the unit length of the Pile element thickness 1m. The Pile element length is automatically taken into account.

Because the load-relative displacement has no relationship, if the allowable settlement is assumed to occur at the Ultimate bearing capacity, the Shear stiffness modulus applied to the analysis can be inferred with reference to the allowable settlement. If the allowable settlement is 0.025m for the example above, the Shear stiffness modulus is [150kN/m2 / 0.025m] = 6000kN/m3. If the Pile element thickness is 3m, entering [50kN/m2 / 0.025m] = 2000kN/m3 still gives the same results.
However, if a different numerical value from the unit length is input for the Pile element thickness, be aware that the same Normal stiffness modulus is equally applied.

3. Applying the interface Wizard interaction formula

Pile elements are also used to estimate the ground-structure mutual behavior with interface elements. Because the neighboring ground material has a larger affect than the pile stiffness or sectional properties, the shear/normal stiffness of the pile element can be inferred using the interaction formula that calculates the tangent/normal stiffness on the interface Wizard.

$$[K_n = E_{oed,i} / L \times t_v \ , K_t = G_i / L \times t_v]$$

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

(v_i =Interface Poisson's ration=0.45, the interface is used to simulate the non-compressive frictional behavior and automatically calculates using 0.45 to prevent numerical errors.)
t_v = Virtual thickness (Generally has a value between 0.01~0.1, the higher the stiffness difference between ground and structure, the smaller the value)
$G_i = R \times G_{soil}$ ($G_{soil} = E/(2(1+ v_{soil}))$), R = Strength Reduction Factor

The general Strength reduction factor for structural members and neighboring ground properties are as follows.

- Sand/Steel = R : 0.6~0.7
- Clay/Steel = R : 0.5
- Sand/Concrete = R : 1.0~0.8
- Clay/Concrete = R : 1.0~0.7

When inferring the shear/normal stiffness of the pile using this method, the Ultimate shear force can be found using the Shear stiffness modulus, as calculated in example 2, and the allowable displacement that is to be applied in the analysis.

Examples 2 and 3 are only acceptable suggestions when the test results are not available. For accurate behavior predictions, the load test must be conducted, or the design parameters need to be computed through repeated analysis.

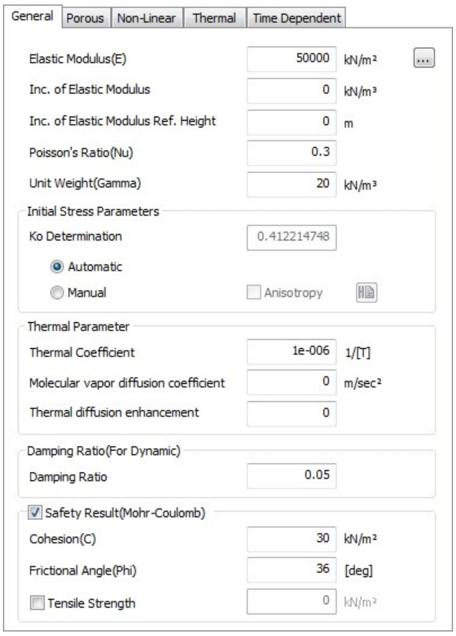
1.3 Material Properties

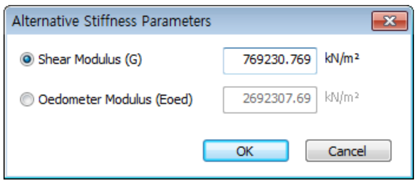
Default parameter (General)

The input parameters and units for defining the default stiffness and initial condition of each model are listed in the table below

Input parameter	Definition	Units
Elasticity modulus (E)	Elasticity modulus	kN/m2
Elasticity modulus increment	The Elasticity modulus increment amount depending on height (slope)	kN/m3
Reference height	Reference height of Elasticity modulus increment	m
Poisson's ratio (ν)	Poisson's ratio	-
Unit weight (γ)	Unit weight of entire unsaturated soil(γt)	kN/m3
Initial stress (Ko)	Coefficient of earth pressure (initial stress parameter)	-
Temperature coefficient	Coefficient for calculating temperature loading	1/[T]
Damping ratio	Material damping ratio (only applied to dynamic analysis)	-

▸General parameter

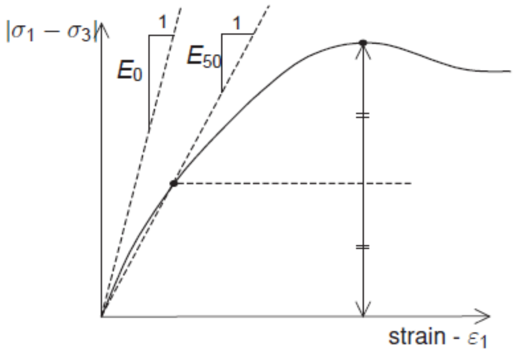




Elastic modulus (E)

This parameter defines the default initial stiffness of the material. The user can specify the Elasticity modulus, or use the Shear modulus (G) or Oedometer Elasticity modulus (E_{oed}) from the Oedometer test. The initial stiffness is very important because geo-materials display nonlinear behavior from the early stages of loading. The initial stiffness can be defined from the stress-strain curves of the triaxial compression test. It is realistic to use the E_0 for materials that display linear (elastic) behavior until a large strain but for general geo-materials, E_{50}, the slope of the tangent at 50% of the stress, is appropriate as an initial stiffness. When simulating unloading and reloading due to excavation during construction step analysis, it is better to use E_{ur} instead of E_{50} to realistically simulate the ground behavior.
Hence, it is important to set the stress path and stress range (size) when using the initial stiffness to simulate the ground behavior. To simulate detailed behavior, various nonlinear material models can be used.

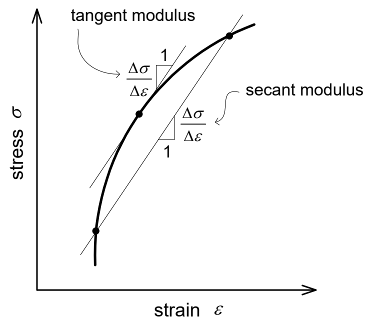
▸Triaxial compression test result graph



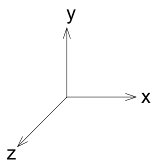
The use of K(bulk modulus) and G(shear modulus) may be debatable use to the continuity issues associated with the ground, but it can be expressed more simply and clearly than E or v and is convenient to use. The following figure briefly explains the mechanical significance of K and G.

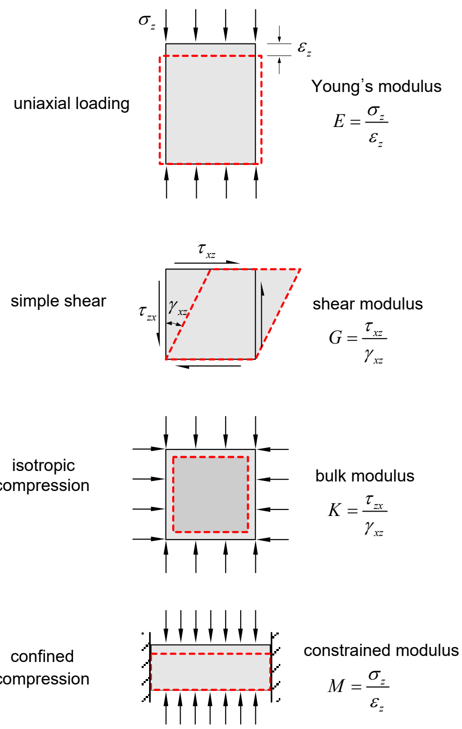
▸Various types of Elasticity modulus

According to the magnitude of the stress increment



According to the loading condition





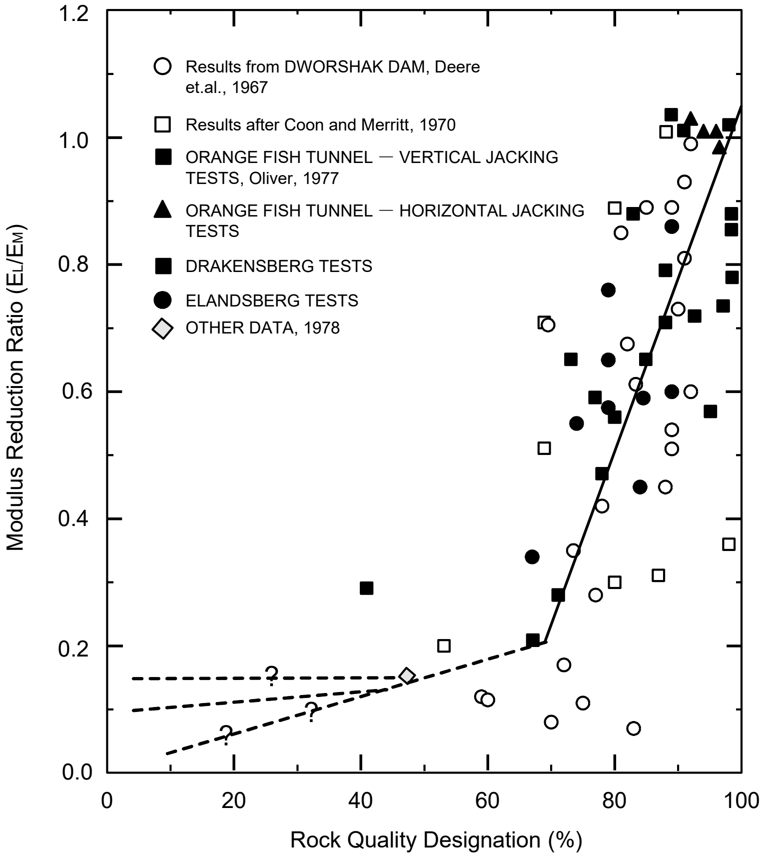
The Elasticity modulus values gained from on-site tests can be 1 of the many elasticity moduli discussed above and can be modified appropriately for real situations.

▸The Elasticity modulus and Poisson's ratio for rock and other materials

Geo-material	Elasticity modulus (tonf/m2)	Poisson's ratio
amphibolite	9.4~12.1 ×106	0.28~0.30
anhydrite	6.8 ×106	0.30
siabase	8.7~11.7 ×106	0.27~0.30
siorite	7.5~10.8 ×106	0.26~0.29
solomite	11.0~12.1 ×106	0.30
sunite	14.9~18.3 ×106	0.26~0.28
deldspathic gneiss	8.3~11.9 ×106	0.15~0.20
gabbro	8.9~11.7 ×106	0.27~0.31
granite	7.3~8.6 ×106	0.23~0.27
ice	7.1 ×106	0.36
limest1	8.7~10.8 ×106	0.27~0.30
marble	8.7~10.8 ×106	0.27~0.30
mica Schist	7.9~10.1 ×106	0.15~0.20
obsidian	6.5~8.0 ×106	0.12~0.18
oligoclasite	8.0~8.5 ×106	0.29
quartzite	8.2~9.7 ×106	0.12~0.15
rock salt	3.5 ×106	0.25
slate	7.9~11.2 ×106	0.15~0.20
aluminum	5.5~7.6 ×106	0.34~0.36
steel	20.0 ×106	0.28~0.29

The Elasticity modulus in the table above is for small, intact rock samples tested in the lab. Hence, when considering the site conditions, a reduced elasticity modulus needs to be used considering the discontinuous surfaces within large scale rocks. The figure below is a graph of actual data showing the relationship between the RQD (Rock Quality Designation) and the Elasticity modulus reduction ratio. An RQD is the percentage of the sum of the lengths of cracks that are over 10cm and exist on the 100cm situ core against the total length. An RQD of 100% does not mean the core is an intact rock. However, a higher RQD means a higher quality rock and the RQD decreases with more weathering.

▸RQD- Modulus reduction ratio (EL/EM) relationship



As shown on the figure, an RQD of 70% already needs to decrease the lab Elastic modulus by 20%.

Increment of Elastic modulus

In general, the strength properties of the soil change with depth and confining pressure, even within a ground layer composed of the same material. To take this characteristic into account, increase or decrease in the Elastic modulus can be simulated with reference to a reference height (standard height). If the elastic increase according to height is '0(zero)', the Elastic modulus has a constant value and if it is not '0(zero)', the Elastic modulus is calculated with reference to a standard height using the following equation.

$$E = E_{ref} + \left(y_{ref} - y\right)E_{inc} \qquad \left(y \le y_{ref}\right)$$
$$E = E_{ref} \qquad \left(y > y_{ref}\right)$$

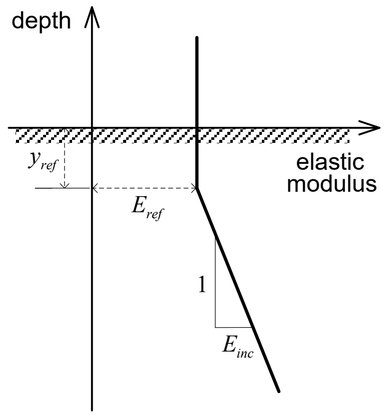
Here,

E_{ref} : Input elastic modulus value

E_{inc} : Incremental slope of elastic modulus

y_{ref} : Depth of E_{ref} measurement

▸Schematic diagram of Elastic modulus increment



The y in the equation represent the integral point positions of an element where the finite element method calculation occurs. If the integral point position is higher than y_{ref} , the elastic modulus value can be less than 0 in some places. To avoid this, use the E_{ref} value instead of further decreasing the E value.

Poisson's ratio(ν)

Poisson's ratio is a proportional constant from the stress-strain relationship and displays the material volume change associated with loading. As ν approaches 0.5, the material becomes an incompressible solid and closer to 0 means the material is elastic, showing large volume changes even at small loads. The initial stress ratio due to self weight $K_0 = \sigma_h / \sigma_v$ can be related to the ratio in the uniaxial compression state by $K_0 = \nu/(1-\nu)$. If K_0 is not used to define the initial in-situ stress, the horizontal stress is calculated from the vertical stress using the entered ν . For geo-materials, the general Poisson range is within 0.3~0.4 and entering a value larger than 0.49 can cause numerical errors. Hence, if K_0 is larger than 1, for example over-consolidated ground, the Poisson's ratio cannot be calculated and the value must be entered directly.

Shear modulus(G)

The Shear modulus is automatically calculated from the Elastic modulus and Poisson's ratio using the following equation derived from Hooke's law. If the value is directly entered, the Elastic modulus changes.

$$G = \frac{E}{2(1+\nu)}$$

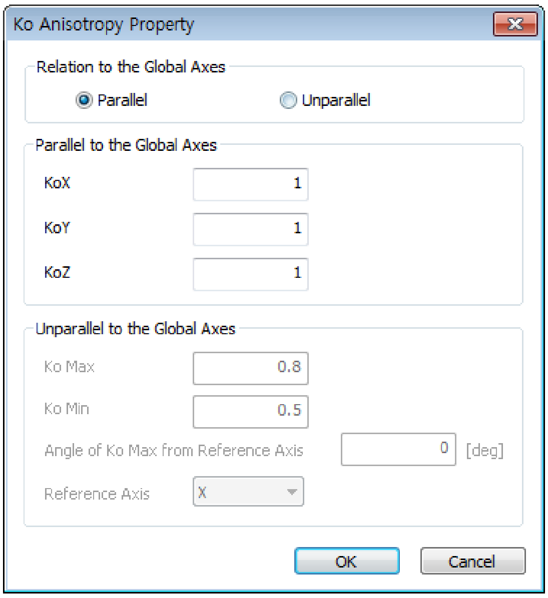
Oedometer Elastic modulus (E_{oed})

The Oedometer modulus can be calculated from the Elastic modulus and Poisson's ratio using the following equation.

$$Eoed = \frac{(1-\nu)E}{(1-2\nu)(1+\nu)}$$

Initial stress (K_0)

K_0 is the Coefficient of earth pressure, which is defined as the ratio of the initial vertical/horizontal stress ($K_0 = \sigma_h / \sigma_v$). The anisotropic property can be set with reference to the Global Coordinate System.



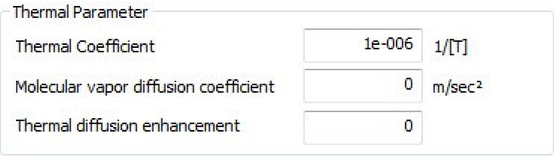
Firstly, select yes/no on whether the Global Coordinate System direction and anisotropic property match and set the lateral pressure index in each axis or any direction depending on the selected options.

When the 2 properties **do match**, the lateral pressure index is set in each axis direction but a value of '1' in the direction of gravity cannot be defined depending on the work environment (2D/3D).

When the 2 properties **do not match**, the lateral pressure index direction is set by entering the angle with respect to the reference axis. The reference axis exists to set the lateral pressure index direction. For a 2D work environment, the 'X-Y' plane is fixed and only the 'X' axis can be selected, with all initial shear stress at '0(zero)'. For 3D, each axis apart from the gravitational direction can be selected. For example, if the gravitational direction is the 'Z' axis and the reference axis is set as the 'X' axis, the angle can be entered on the 'X-Z' plane will be the maximum lateral pressure angle and all initial shear stress in the XY and YZ direction will be '0(zero)'.

The in-situ stress state, where the soil it not disturbed by excavation or fill, can be expressed using the Coefficient of earth pressure and self weight. In other words, realistic results can be obtained from applying K0 after modeling the in-situ ground for the 1st step of construction during analysis. This is true for flat foundations but for inclined foundations, it is recommended that another construction process be added to converge the stress found using K0 for equilibrium.

Thermal Parameter



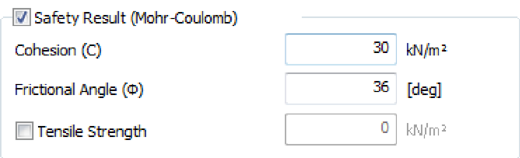
Thermal Coefficient - describes how the size of an object changes with a change in temperature. Specifically, it measures the fractional change in size per degree change in temperature at a constant pressure.

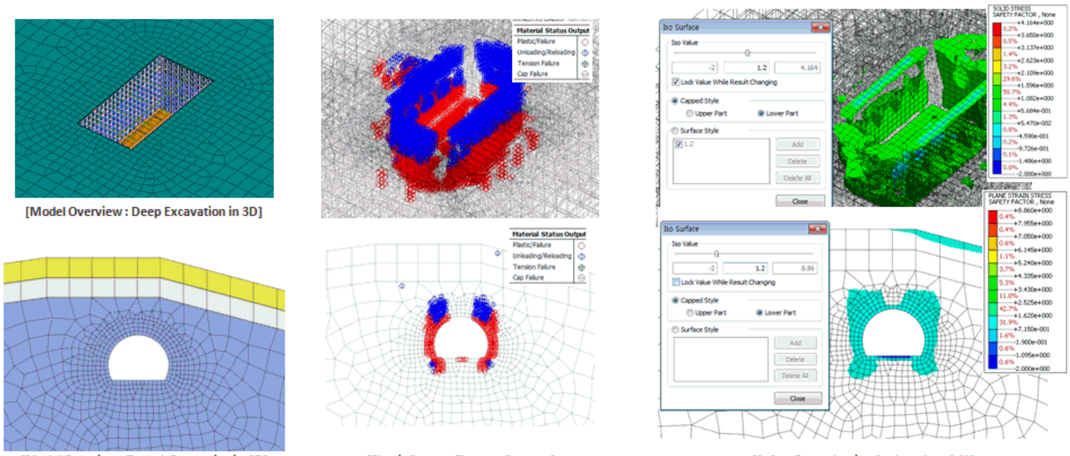
**Molecular vapor diffusion coefficient* - the gas diffusion coefficient of a porous medium, which indicates the change in gas density over time.

*Thermal diffusion enhancement (factor) - controls the degree of gas flow according to the temperature gradient (unit less).

Safety Result (Mohr-Coulomb)

- Cohesion, Friction Angle and Allowable tensile strength (optional) can be defined as the failure criteria.
- Stress status of material for each construction stage can be represented by Factor of Safety based on Mohr-Coulomb failure criteria.
- The ratio of generated stress to stress at failure for each element will be calculated automatically.
- Users can figure out stable, potential failure and plastic failure area directly.
- Check factor of safety for each element - (**2D : Plain Strain Stresses > SAFETY FACTOR , 3D : Solid Stresses > SAFETY FACTOR**)
- In case that Safety Factor is less than 1(or 1.2), it can be identical with plastic failure region.





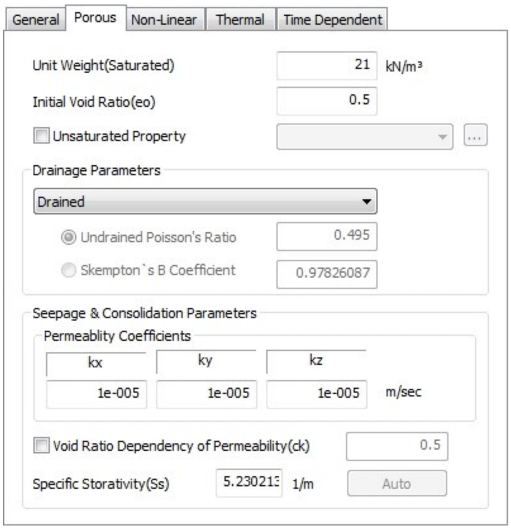
[Model Overview : Tunnel Excavation in 2D] [Plastic Status : Element Stresses] [Safety Factor (region for less than 1.2)]

Seepage, Drained/Undrained Parameter

The input parameters for the permeability characteristics and drained/undrained conditions of the foundation are as follows.

Input parameter	Definition	Units
Unit weight(saturated)	Saturated state unit weight	kN/m^3
Initial void ratio(e_o)	Initial void ratio	-
Unsaturated characteristic	Unsaturated characteristic function setting (negative pore water pressure-water content-permeability ratio)	-
Drainage parameter	Drained/Undrained condition	-
Permeability coefficient	GCS direction - Saturated permeability constant	m/sec
Void ratio dependent permeability ratio (ck)	Permeability ratio dependent on void ratio	-
Specific storage(Ss)	Volume ratio of water inflow/outflow	1/m

▸ Permeability parameter



Initial void ratio (e0)

The initial void ratio of the foundation used in consolidation analysis and stress-seepage coupled analysis. It is the volume ratio between the voids and soil particles within the soil and the value is less than 1 for most soils. The value can be larger than 1 for clays or organic soils, but the value depends greatly on the sampling method or compaction. Generally, coarse grain sand has a value of 0.6~0.8 and high density sand with an even size distribution has a value of 0.3. The void ratio can be even 2~3 for fine grained soils.

Unsaturated Property

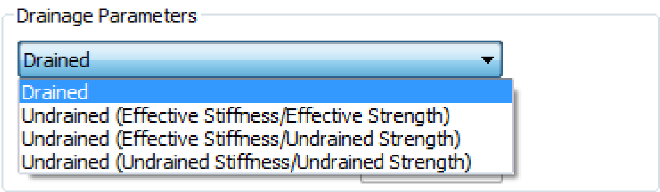
Set to consider the unsaturated property of the foundation. It is a required property for unsteady infiltration analysis and is used to consider the partial degree of saturation of the foundation for nonlinear (construction step)/consolidation analysis etc. Because real foundations are unsaturated and have a certain ratio of air, unsteady infiltration analysis needs to consider unsaturated characters of the soil for more realistic results. If the unsaturated properties are not considered, it is assumed that the ground is saturated and hence, the infiltration analysis with time cannot be examined.
Unsaturated property defines the change in permeability coefficient and water content (Degree of saturation) in the unsaturated region depending on the size of the negative pore water pressure. There are 2 methods to define the unsaturated property; directly defining (define individually) the permeability function and water content function using the pressure head (negative pore water pressure) or defining the relationship

between pressure head-volumetric water content (degree of saturation)-permeability ratio (define relationship). Refer to "Function>Unsaturated characteristic function" for more information.

Drainage parameters

The pore water pressure in stress analysis can be divided into normal state pore water pressure and abnormal state pore water pressure - the excess pore water pressure generated between soil particles due to external loading under undrained conditions. An excess pore water pressure of nearly 0 is called the drainage condition and is generally used for drainage analysis of sand, which has a large permeability. However when simulating clay, which has a very small permeability and water cannot be drained out during sudden loading, undrained analysis is appropriate. The initial state, where the excess pore water pressure has not yet dissipated, is seen as the most unstable state and the pore water pressure is determined by the volume change of the foundation due to compressibility and permeability coefficient.

Undrained Poisson's ratio and Skempton(B) coefficient are parameters used to calculate the bulk modulus of elasticity for water. The undrained Poisson's ratio has a standard value of 0.495 with a compressibility of nearly '0(zero)' and the Skempton coefficient expresses the saturation, with 1 meaning full saturation.
The materials for the unsaturated analysis are as follows.
Please refer to Ch.4 of the Analysis manual for more detailed information.



Drained/Undrained Materials	Useable Material models
Drained	All geo-materials
Undrained (Effective stiffness /Effective strength)	Elastic, Mohr-Coulomb, Drucker-Prager, Duncan-Chang, Hoek-Brown, Strain Softening, Modified Cam-clay, Jardine, D-min, Modified Mohr-Coulomb, User-supplied, Transversely Isotropic
Undrained (Effective stiffness / Undrained strength)	Mohr-Coulomb, Drucker-Prager, Modified Mohr-Coulomb
Undrained (Undrained stiffness / Undrained strength)	Elastic, Mohr-Coulomb, Drucker-Prager, Modified Mohr-Coulomb

Permeability coefficients (kx,ky,kz)

The permeability coefficient represents the permeability characteristics (velocity) of the foundation and is used in infiltration analysis and consolidation analysis. The permeability coefficient for each direction can be defined on the GCS. The input value is the saturated permeability coefficient and becomes the standard for computing the permeability ratio (Kunsat / Ksat) due to negative pore water pressure when defining an unsaturated property function.

Void ratio dependency permeability (ck)

The permeability coefficient is a measurement of how much the groundwater within a foundation moves in a unit time and is dependent on the water content and the void ratio change Δe . The larger water content, the larger the flow channel and hence, the value is largest when the foundation is saturated. The water content depends on the pore water pressure and hence, the permeability coefficient is also dependent on the pore water pressure. The change in void ratio is considered in consolidation analysis as well as stress-seepage coupled analysis and is calculated from the initial void ratio.

To express the change in pore water pressure, FEA NX uses the permeability ratio function $k_r = k_r(p)$ depended on saturated pore water pressure coefficient $\mathbf{k}_{sat}$ and pore water pressure change and the void ratio dependent permeability ratio c_k dependent on void ratio change Δe . The unsaturated permeability coefficient depending on void ratio change Δe is given by the following equation.

$$\mathbf{k} = 10^{\frac{\Delta e}{c_k}} \ k_r(p) \ \mathbf{k}_{sat}$$

Specific storativity (Ss)

The Specific storage is the water volume inflow or outflows from the unit volume of the aquifer due to water level rise or fall in a confined aquifer. A coefficient can be directly entered or automatically calculated for compressible fluids.
The change in volumetric water content for pore water pressure in infiltration and consolidation analysis can be expressed by the porosity and degree of saturation.

$$\frac{\partial \theta}{\partial p} = S\frac{\partial n}{\partial p} + n\frac{\partial S}{\partial p}$$

The first clause is the slope of the volumetric water content under saturated conditions that can be expressed using the specific storage.

$$S\frac{\partial n}{\partial p} = \frac{\partial V_v}{\partial h}\frac{\partial h}{\partial p} = \frac{S_s}{\gamma}$$

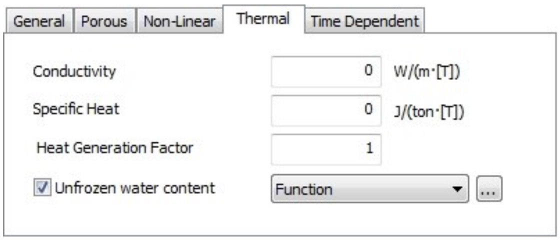
When the material drainage property is set to undrained, the specific storage is automatically calculated using the undrained Poisson's ratio (v_u) and the Effective elastic modulus (E') and Poisson's ratio (v'), entered in the general parameters.

$$K_f = BK_u = \frac{E'(\nu_u - \nu')}{(1-2\nu')(1+\nu')(1-2\nu_u)}$$

$$S_s = \frac{n\gamma_w}{K_w} = \frac{\gamma_w}{K_f}$$

$$S_s = \gamma_w \frac{(1-2\nu')(1+\nu')(1-2\nu_u)}{E'(\nu_u - \nu')}$$

Thermal

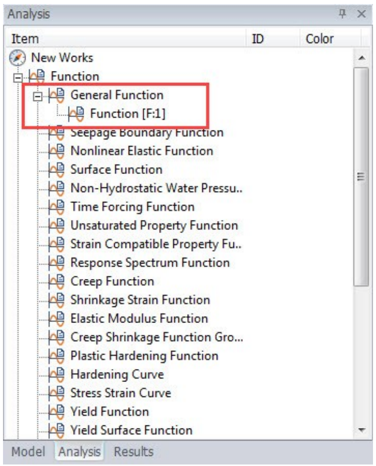


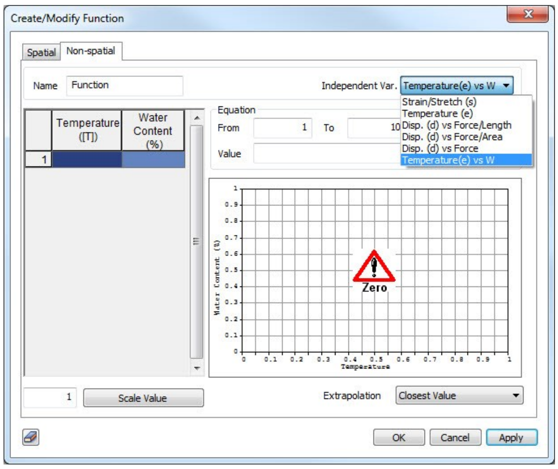
Conductivity: the ability to conduct thermal energy.

Specific Heat: the amount of heat required to raise single unit mass of a substance by single temperature unit. (required for transient heat transfer problems)

Heat Generation Factor: the value of the heat load multiplied by the exothermic coefficient used as the load vector for heat transfer analysis is the total exothermic load applied to the object.

Unfrozen water content: indicates floating water content in soil / rock. It is given as a temperature-dependent function as a unique characteristics of the ground.



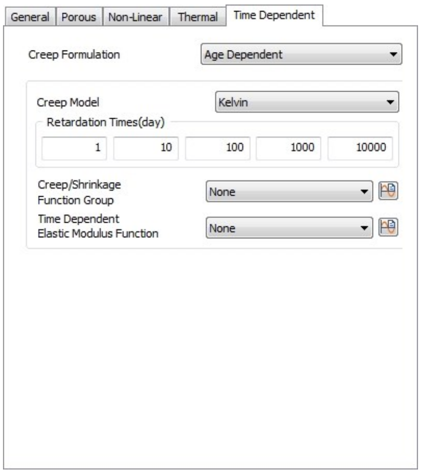


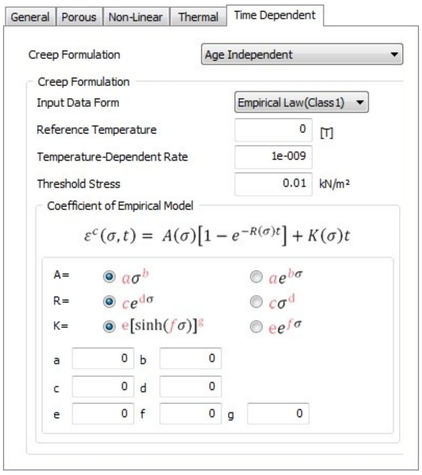
Time Dependent

This is to define Creep Formulation to simulate time-dependent behavior of concrete structures. Following constitutive models are available for concrete structures, ***Elastic, Tresca, von Mises, Mohr-Coulomb, Drucker Prager, and Hoek Brown***.

Two types of creep formulation are available to define Time-dependent behavior of material, Age Dependent and Age Independent. Refer to analysis reference Ch.4-Section5 in detail.

▸ Creep Formulation

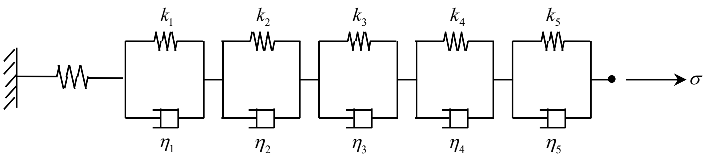




Age Dependent

The stiffness of concrete changes with time, and the creep and shrinkage may cause unexpected deformation. The creep strain of concrete depends on the time of stress occurrence even under the same applied load. FEA NX supports aging-Kelvin model and aging Viscous model excluding the spring from Kelvin model.

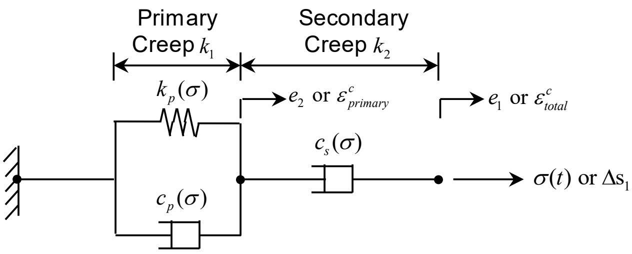
▸ Aging - Kelvin creep model



Age Independent

FEA NX can take into account the primary and secondary creep. The user can use two types of empirical law to define the creep behavior.

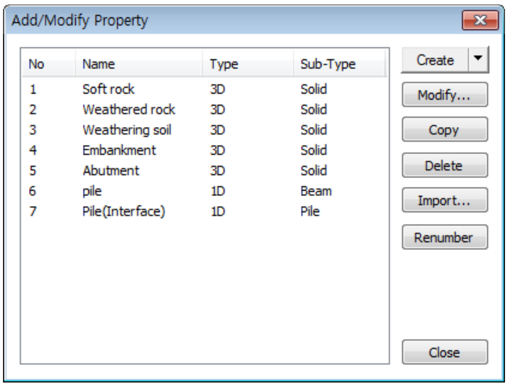
▸ Kelvin-Maxwell creep model



1.4 Property

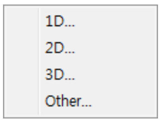
Overview

Add/define ground and structural properties. This function defines the property of each mesh set when a ground or structure mesh is generated. For ground, it determines which material to use and for structures, it adds section size, shape (stiffness) and horizontal spacing etc. The horizontal spacing is the 1D spacing between structural members in the horizontal direction of a 2D model. Tapered section option, where the size (thickness) of a beam or plate changes, is also available.



Methodology

[Create]
Creates 1 of the following 4 property types and adds a material type and section characteristic for each property.



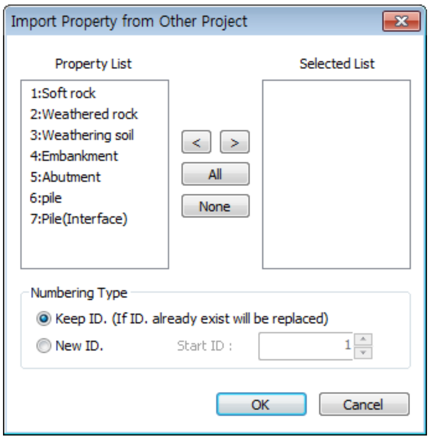
The supported model type for each property type is as follows. The section shape/size, stiffness depending on spacing and selectable materials is defined.

Property type	Model type	Ground property	Structural property	Nonlinear property
1D	Geogrid(1D)	X	O	Tension Only
	Plot only(1D)	X	X	-
	Truss	X	O	From Material
				Tension Only/Hook
				Compression Only/Gap
				Nonlinear Elastic
	Embedded truss	X	O	From Material
				Tension Only/Hook
				Compression Only/Gap
				Nonlinear Elastic
	Beam	X	O	-
	Embedded Beam	X	O	-
	Pile	X	O	-
2D	Geogrid(2D)	X	O	Tension Only
	Plot only(2D)	X	X	-
	Gauging shell	X	O	-
	Axisymmetric	O	X	-
	Shell	X	O	-
	Plane stress	X	O	Tension Only
	Plane strain	O	X	-
3D	Solid	O	X	-
Others	Rigid link	X	O	-
	Pile tip	X	O	-
	User Supplied Behavior for Shell Interface	X	O	-
	Point spring	X	O	Linear Elastic
				Tension Only
				Compression Only
				Hook
				Gap
				Nonlinear Elastic
	Matrix spring	X	O	-
	Free Field	O	X	Free Field
				Absorbent Boundary
	Interface	X	O	-
	Shell interface	X	O	-
	Infinite	O	X	From Adjacent Element
				User Defined
	Elastic link	X	O	Linear Elastic
				Rigid Body
				Tension Only
				Compression Only
				Nonlinear Elastic
	Seepage Cut Off	O	X	-

Refer to Ch.3 of the Analysis manual for detailed information on each input parameter and behavioral characteristic.

[Modify/Copy/Delete]
Modify the parameters of an added material. Copy can be used when adding multiple materials while only changing certain parameters.

[Import]
Import the material properties from a different model file with a saved material/property. This operation is useful when analyzing the existing project under the same conditions. Selecting the import file generates the material list containing all saved materials. The user can select the desired material.



1.5 Section Property

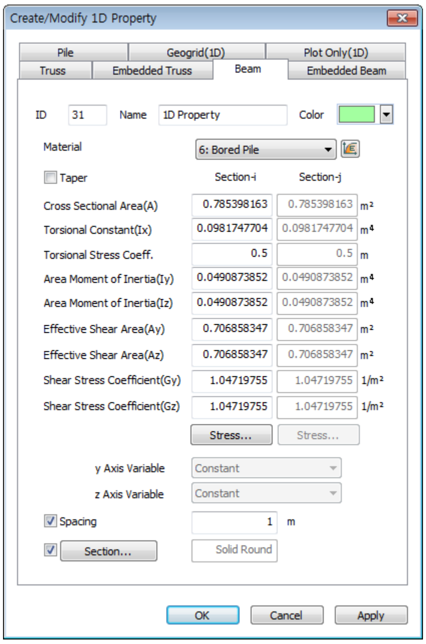
Overview

The section material properties need to be defined for a 1 dimensional truss, embedded truss or beam element. Here, the truss and embedded truss only need the cross-sectional area but the beam needs other material properties such as total cross section, torsional rigidity, first area moment of inertia and second area moment of inertia to consider torsion, bend and shear.

The thickness of the element needs to be defined to specify the 2D plane stress element, 2D geogrid element, plate element, plane strain element, axial symmetry element, linear interface element etc. Here, the plane strain element, axial symmetry element and linear interface element have an interior unit weight of 1 and the user can define the used units according to the thickness.
The plane stress element, 2D geogrid element and plate element use the thickness value entered by the user. Here, the plate element has a rotor float and because nonlinear analysis is possible, a separate integration is performed in the thickness direction.

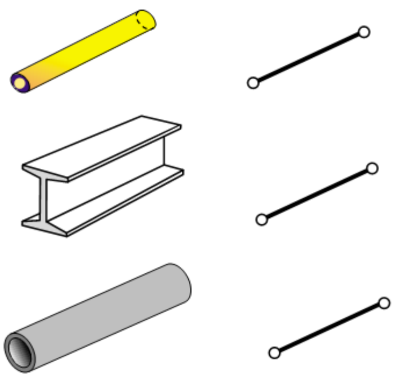
1D element

1D element is an element that is made up of 2(primary) or 3(secondary) nodes and has the geometric property of length. Because 3D shapes are expressed as 1D elements, the section (size, shape) needs to be defined and this is modeled as a 2D element for calculations.
FEA NX provides various shapes, as shown in the figure below. The position can also be set when defining the sectional properties.

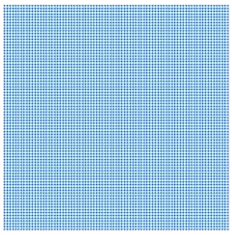


▸Actual model

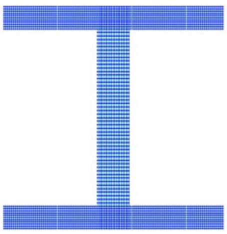
▸▸Finite element model



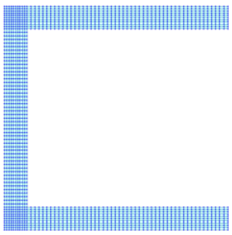
▸Automatic section modeling



QUAD-4 6400 elements
<Solid square>

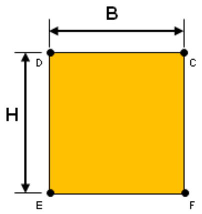


QUAD-4 3400 elements
<H section>

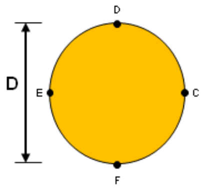


QUAD-4 1700 elements
<Channel>

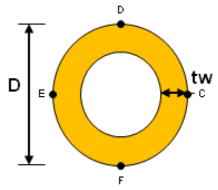
▸ Section shape and size specification



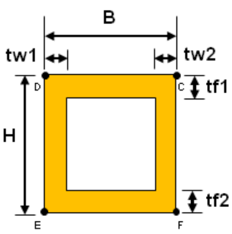
<Solid Rectangle>



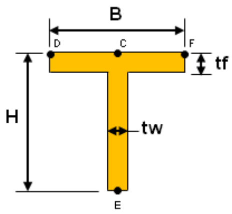
< Solid Round>



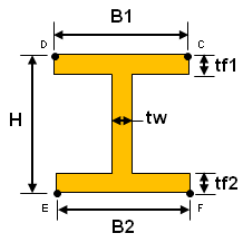
<Pipe>



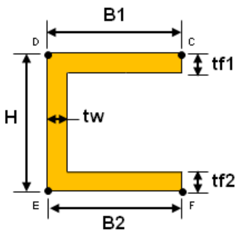
<Box>



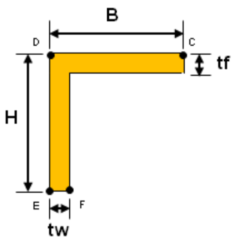
<T-section>



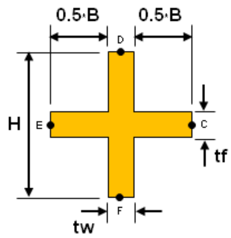
<H-section>



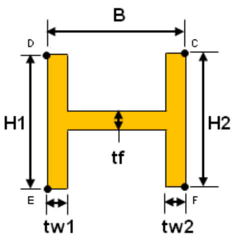
<Channel>



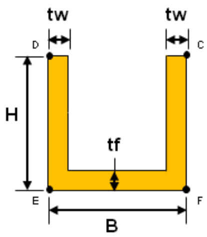
<Angle>



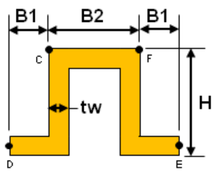
<Cross>



<I-section>



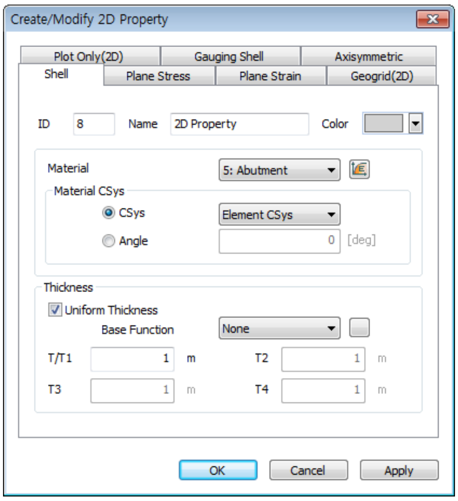
<Channel 1>



<Hat>

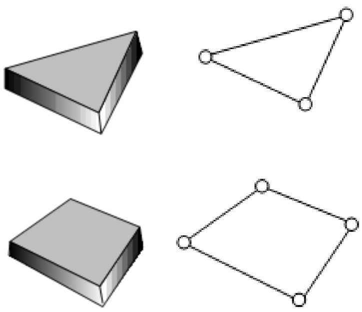
2D element

2D elements are Triangles or Quadrilaterals with the geometric property of area. Because 3D shapes are expressed as 2D elements, the thickness needs to be defined. The thickness can be set the same or tapered.



▸Actual model

▸▸Finite element model

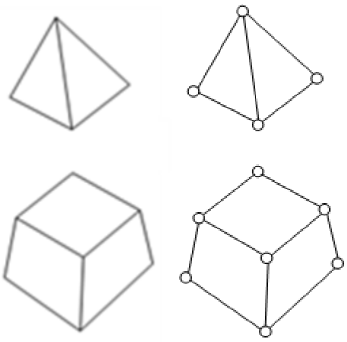


3D element

3D elements are Tetrahedron or Hexahedrons, Bricks with the geometric property of volume.

▸Actual model

▸▸Finite element model

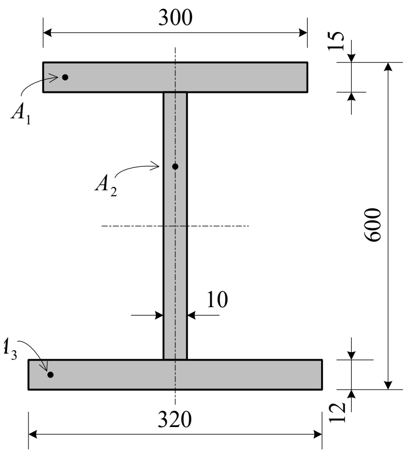


Cross sectional area (A)

The cross sectional area is used to calculate the axial stiffness when a tensile or axial force acts on the member or the stress on a member. The calculations for the H section are as follows.
There are 2 methods to calculate the cross sectional area in the FEA NX. The first method uses the provided database to input the dimensions of a section and automatically calculate the area. For the second method, the user calculates the area directly and inputs the value. The first method is convenient, but because it does not consider the decrease in area due to bolts in the connection or rivet holes, the area entered using the second method may provide more accurate results.

▸Example of cross sectional area calculation

Area =∫dA=Aone+ A2 + A3
=(300×one5)+ (573×one0)
+ (320×one2) = one4,070



Torsional stiffness(Ixx)

The torsional stiffness resists the torsional moment and is expressed as follows.

$$I_{xx} = \frac{T}{G\theta}$$

Here,

I_{xx}	: Torsional stiffness
T	: Torsional moment or torque
θ	: Torsional angle (angle of twist)
G	: Shear modulus

The torsional stiffness is the stiffness that resists the torsional moment and is different from the polar moment of inertia that decides the shear stress due to torsion. (However, the 2 are the same when considering a circular cross-section or a thick cylindrical section)
The torsional stiffness can be calculated from Saint-Venant's principle as shown below.

$$T = G\theta \int \left[\left(\frac{\partial \omega}{\partial z} + y \right) y - \left(\frac{\partial \omega}{\partial y} - z \right) z \right] dA$$

ω is the warping function $\omega(y,z)$ that can be calculated using the Finite element method as shown below.

$$\int\left(\frac{\partial\delta\omega}{\partial y}\frac{\partial\omega}{\partial y}+\frac{\partial\delta\omega}{\partial z}\frac{\partial\omega}{\partial z}\right)dA=\int\left(\frac{\partial\delta\omega}{\partial y}z-\frac{\partial\delta\omega}{\partial z}y\right)dA$$

Because $T=I_{xx}G\theta$, the torsional stiffness comp1nt is expressed as the following equation.

$$I_{xx}=\int\left[\left(\frac{\partial\omega}{\partial z}+y\right)y-\left(\frac{\partial\omega}{\partial y}-z\right)z\right]dA$$

Effective shear area (Asy, Asz)

The effective shear area is needed to calculate the shear stiffness that resists the shear force acting on the y axis or z axis of the element coordinate system. If the effective shear area is not entered, the shear strain in that direction is ignored.

$$A_{sy}=S_{ky}A$$
$$A_{sz}=S_{kz}A$$

Here,

S_{ky} : Effective shear factor that resists shear force in the y axis of the element coordinate system
S_{kz} : Effective shear factor that resists shear force in the z axis of the element coordinate system
A_{sy} : Effective shear area that resists shear force in the y axis of the element coordinate system
A_{sz} : Effective shear area that resists shear force in the z axis of the element coordinate system

When the interior section material properties are calculated or entered from the database, the shear stiffness comp1nt is automatically considered and the effective shear factor is calculated using the warping function $\varphi(y,z)$ from the shear force caused by bending moment and the warping function $\omega(y,z)$ from the Saint-Venant principle.

$$\frac{1}{S_{ky}}=\frac{A}{V_y}\left[\frac{I_{\varphi y}I_{yy}-I_{\varphi z}I_{yz}}{I_{yy}I_{zz}-I_{yz}^2}+\frac{\nu}{2(1+\nu)}\frac{\left(C_{zz}I_{yy}-C_{yy}I_{yz}\right)}{I_{yy}I_{zz}-I_{yz}^2}\right]$$
$$\frac{1}{S_{kz}}=\frac{A}{V_z}\left[\frac{I_{\varphi z}I_{zz}-I_{\varphi y}I_{yz}}{I_{yy}I_{zz}-I_{yz}^2}+\frac{\nu}{2(1+\nu)}\frac{\left(C_{zz}I_{zz}-C_{yy}I_{yz}\right)}{I_{yy}I_{zz}-I_{yz}^2}\right]$$

Here,

$$I_{\varphi y}=\int\varphi y\,dA,\qquad I_{\varphi z}=\int\varphi z\,dA$$
$$C_{yy}=\int\tau_{xz}\left(y-y_o\right)^2dA,\qquad C_{zz}=\int\tau_{xy}\left(z-z_o\right)^2dA$$
$$y_o=\frac{\int\left(\frac{\partial\omega}{\partial z}+y\right)y^2dA}{2\int\left(\frac{\partial\omega}{\partial z}+y\right)ydA},\qquad z_o=\frac{\int\left(\frac{\partial\omega}{\partial y}+z\right)z^2dA}{2\int\left(\frac{\partial\omega}{\partial y}+z\right)zdA}$$

Area moment of inertia(Iyy, Izz)

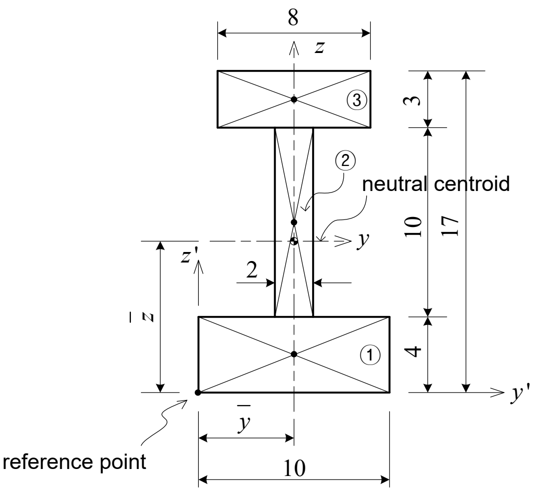
The area moment of inertia is used to calculate the flexural stiffness that resists the bending moment and is calculated from the centroid axis of the section using the following equation.

- Area moment of inertia about the y axis of the element coordinate system

$$I_{yy} = \int z^2 dA$$

- Area moment of inertia about the z axis of the element coordinate system

$$I_{zz} = \int y^2 dA$$



▸Table. First area moment of inertia and calculation of centroid

Section element	b	h	A_i	$\bar{z}_i$	Q_{yi}	$\bar{y}_i$	Q_{zi}
①	10	4	40	2	80	5	200
②	2	10	20	9	180	5	100
③	8	3	24	15.5	372	5	120
total	-	-	84	-	632	-	420

A_i : Area

$\bar{z}_i$: Distance from the reference point to the centroid of the section element in the z´-axis direction

$\bar{y}_i$: Distance from the reference point to the centroid of the section element in the y´-axis direction

Q_{yi} : First moment of area relative to the reference point in the y´-axis direction

Q_{zi} : First moment of area relative to the reference point in the z´-axis direction

- Calculate position of neutral axis ($\bar{Z}$, $\bar{Y}$)

$$\bar{Z}=\frac{\int \bar{z}dA}{Area}=\frac{Q_y}{Area}=\frac{632}{84}=7.5238$$
$$\bar{Y}=\frac{\int \bar{y}dA}{Area}=\frac{Q_z}{Area}=\frac{420}{84}=5.0000$$

- Calculate second area moment of inertia (I_{yy} , I_{zz})

▸Table. Second area moment of inertia example

Section element	A_i	$\bar{Z}-\bar{z}_i$	I_{y1}	I_{y2}	I_{yy}	$\bar{Y}-\bar{y}_i$	I_{z1}	I_{z2}	I_{zz}
①	40	5.5328	1224.5	53.3	1277.8	0	0	333.3	333.3
②	20	1.4672	43.1	166.7	209.8	0	0	6.7	6.7
③	24	7.9762	1526.9	18.0	1544.9	0	0	128.0	128.0
total		2794.5	238.0	3032.5		0	468.0	468.0	

$$I_{y1}=A_i\times\left(\bar{Z}-\bar{z}_i\right)^2,\quad I_{y2}=\frac{bh^3}{12},\quad I_{yy}=I_{y1}+I_{y2}$$
$$I_{z1}=A_i\times\left(\bar{Y}-\bar{y}_i\right)^2,\quad I_{z2}=\frac{hb^3}{12},\quad I_{zz}=I_{z1}+I_{z2}$$

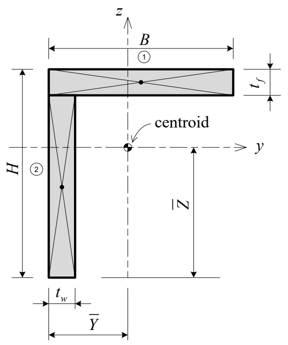
Area product moment of inertia (Iyz)

The area product moment of inertia is used to calculate the stress comp1nt of an asymmetrical section and the definition is as follows.

$$I_{yz}=\int y\cdot z dA$$

H, pipe, box, channel, tee type sections have at least 1 axis of symmetry out of the y,z axis on the element coordinate system and hence Iyz=0. For angle type sections, it does not have any angle of symmetry (Iyz≠ 0) and so, the stress comp1nt needs to be calculated.
The calculations for the area product moment of inertia of an angle type section are shown in the figure below.

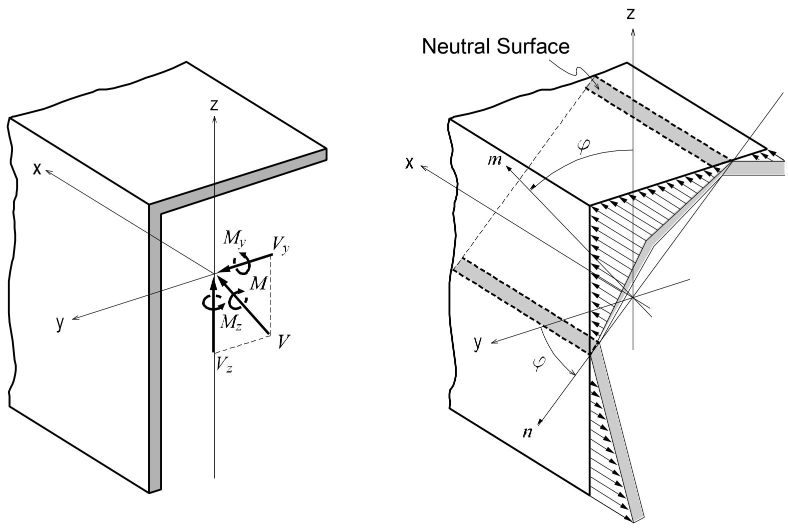
▸ Area product moment of inertia calculations for an angle section



section element	A_i	e_{zi}	e_{zi}
①	$B\times t_f$	$B/2-\bar{Y}$	$(H-t_f/2)-\bar{Z}$
②	$(H-t_f)\times t_w$	$t_w/2-\bar{Y}$	$(H-t_f/2)-\bar{Z}$

$$\begin{aligned}I_{yz}&=\sum A_i\times e_{yi}\times e_{zj}\\&=\left(B\times t_f\right)\times\left(B/2-\bar{Y}\right)\times\left\{\left(H-t_f/2\right)-\bar{Z}\right\}\\&\quad+\left\{\left(H-t_f\right)\times t_w\right\}\times\left(t_w/2-\bar{Y}\right)\times\left\{\left(H-t_f/2\right)-\bar{Z}\right\}\end{aligned}$$

▸ Flexural stress distribution diagram for asymmetrical section



The neutral axis is the axis that passes the points where the flexural stress due to the bending moment is '0(zero)' within the member. The neutral axis is perpendicular to the m-axis and the n-axis.

Because the flexural stress due to bending moment is '0' on the neutral axis, the neutral axis direction can be found by the following equation.

$$\left(M_y \times I_{zz} + M_z \times I_{yz}\right) \times z - \left(M_z \times I_{yy} + M_y \times I_{yz}\right) y = 0$$

$$\tan\phi = \frac{y}{z} = \frac{M_y \times I_{zz} + M_z \times I_{yz}}{M_z \times I_{yy} + M_y \times I_{yz}}$$

The general equation used to calculate the flexural stress due to bending moment is as follows.

$$f_b = \frac{M_y - M_z\left(I_{yz}/I_{zz}\right)}{I_{yy} - \left(I_{yz}^2/Izz\right)} \times z + \frac{M_z - M_y\left(I_{yz}/I_{yy}\right)}{I_{zz} - \left(I_{yz}^2/I_{yy}\right)} \times y$$

If this is a H type section, $I_{yz} = 0$ and,

$$f_b = \frac{M_y}{I_{yy}} \times z + \frac{M_z}{I_{zz}} \times y = f_{by} + f_{bz}$$

Here,

I_{yy} : Second area moment of inertia about the y axis of the element coordinate system,

I_{zz} : Second area moment of inertia about the z axis of the element coordinate system,

I_{yz} : Area product moment of inertia,

y : Elemental y axis distance from neutral axis to where the flexural stress is calculated,

z : Elemental z axis distance from neutral axis to where the flexural stress is calculated,

M_y : Bending moment about the y axis of the element coordinate system,

M_z : Bending moment about the z axis of the element coordinate system

The shear stress due to shear force acting in the y axis and z axis direction of the element coordinate system can be calculated using the following equation.

$$\tau_y = \frac{V_y}{b_z \times \left(I_{yy} \cdot I_{zz} - I_{yz}^2\right)} \times \left(I_{yy} \cdot Q_z - I_{yz} \cdot Q_y\right) = \left(\frac{I_{yy} \cdot Q_z - I_{yz} \cdot Q_y}{I_{yy} \cdot I_{zz} - I_{yz}^2}\right) \times \left(\frac{V_y}{b_z}\right)$$

$$\tau_z = \frac{V_z}{b_y \times \left(I_{yy} \cdot I_{zz} - I_{yz}^2\right)} \times \left(I_{zz} \cdot Q_y - I_{yz} \cdot Q_z\right) = \left(\frac{I_{zz} \cdot Q_y - I_{yz} \cdot Q_z}{I_{yy} \cdot I_{zz} - I_{yz}^2}\right) \times \left(\frac{V_z}{b_y}\right)$$

Here,

V_y : Shear force acting in the y axis direction of the element coordinate system,

V_z : Shear force acting in the z axis direction of the element coordinate system,

Q_y : First area moment of inertia about the y axis of the element coordinate system,

Q_z : First area moment of inertia about the z axis of the element coordinate system,

b_y : Section thickness at the point where shear stress is calculated in the normal direction to the elemental y axis,

b_z : Section thickness at the point where shear stress is calculated in the normal direction to the elemental z axis

First moment of area (Qy, Qz)

The first moment of area is used to calculate the shear stress at an arbitrary point on the section and the shear stress can be calculated using the following equation.

$$Q_y = \int z dA$$

$$Q_z = \int y dA$$

For a section that is symmetrical about the y, z or both axis, the shear strength on an arbitrary point can be calculated using the following equation.

$$\tau_y = \frac{V_y \cdot Q_z}{I_{zz} \cdot b_z}$$

$$\tau_z = \frac{V_z \cdot Q_y}{I_{yy} \cdot b_y}$$

Here,

V_y : Shear force acting in the y axis direction of the element coordinate system,

V_z : Shear force acting in the z axis direction of the element coordinate system,

I_{yy} : Second area moment of inertia about the y axis of the element coordinate system,

I_{zz} : Second area moment of inertia about the z axis of the element coordinate system,

b_y : Section thickness at the point where shear stress is calculated in the normal direction to the elemental y axis,

b_z : Section thickness at the point where shear stress is calculated in the normal direction to the elemental z axis

Element thickness

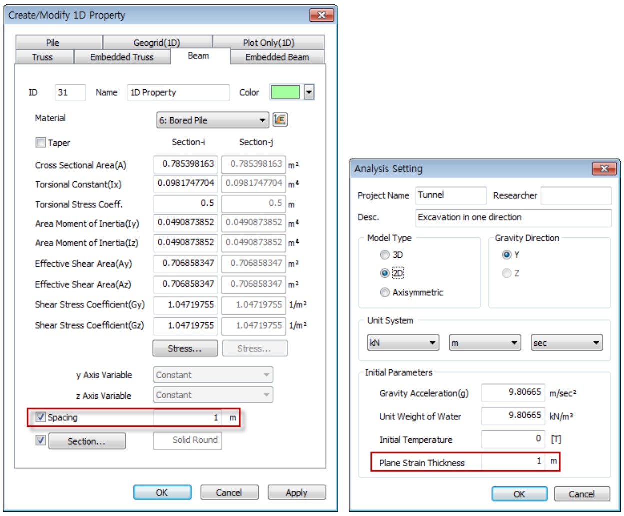
On the FEA NX, the thickness needs to be defined to specify the 2D plane stress element, 2D geogrid element, plate element, plane strain element, axial symmetry element, linear interface element etc. Here, the plane strain element, axial symmetry element and linear interface element have an interior unit weight of 1.

The plane stress element, 2D geogrid element and plate element use the thickness value entered by the user. The plate element has a rotor float and because nonlinear analysis is possible, a separate integration is performed in the thickness direction.

Spacing

This functionality is in the 1D element property which is activated only in 2D project setting. Since this option is used to consider the 1D element force per each element when the user introduce the 1D element more or less than one along the axis of horizontal direction (thickness direction) in 2D model.

If the user uncheck the spacing option, on the FEA NX, the spacing will be regarded as Plane Strain Thickness in the analysis setting, meaning that the unit thickness based on the selected unit system.



On the FEA NX, spacing is used to calculate the stiffness of the element and output the member force per each element.

$$K^* = K / n = \frac{E \times A / n}{L}$$

where, n = spacing, L = length , A = area, K* = stiffness considering spacing.

$$f = \sigma A^* = E\varepsilon(nd)\frac{A}{n} = E\varepsilon(d)A\,,\; f^* = nf$$

where, f* = member force

For instance,

if $E = 1000N/m^2$, $L = 10m$, $A = 1m^2$, and applied external load $P = 1000N$,

$$F = K^* D = \frac{EA}{L} D = \frac{1000N/m^2 \times \left(1m^2 / 0.5\right)}{10m} D = 200N/m \times D \text{ , } D = F/K = 5m$$

$$\varepsilon = D/L = \frac{5m}{10m} = 0.5$$

$$\sigma = E\varepsilon = 500N/m^2$$

$$f = \sigma A = 500N/m^2 \times (1m^2/0.5) = 1000N$$

the member force calculated is as follows,

$$f^* = nf = 0.5 \times 1000N = 500N$$

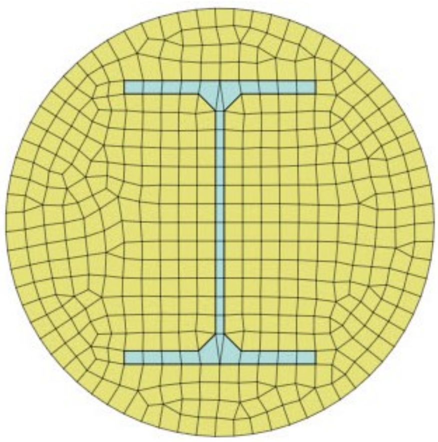
1.6 Complex Cross-Section Property

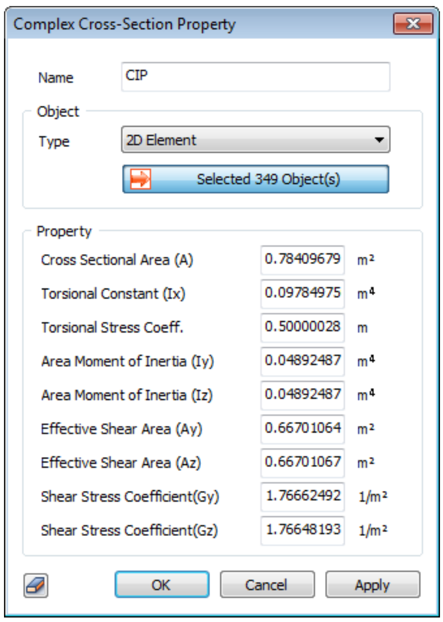
Overview

Calculate the section property in the non-formalized cross-section. (Cross Sectional Area (A), Torsional Constant (Ix), Torsional Stress Coefficient, Area Moment of Inertia (Iy, Iz), Effective Shear Area (Ay, Az), Shear Stress Coefficient (Gy, Gz))
The section property is calculated automatically from the material property by selecting 2D elements assigned ground material, and this can be used for 1D structural property.
A value close to the exact solution can be obtained under the more dense mesh.

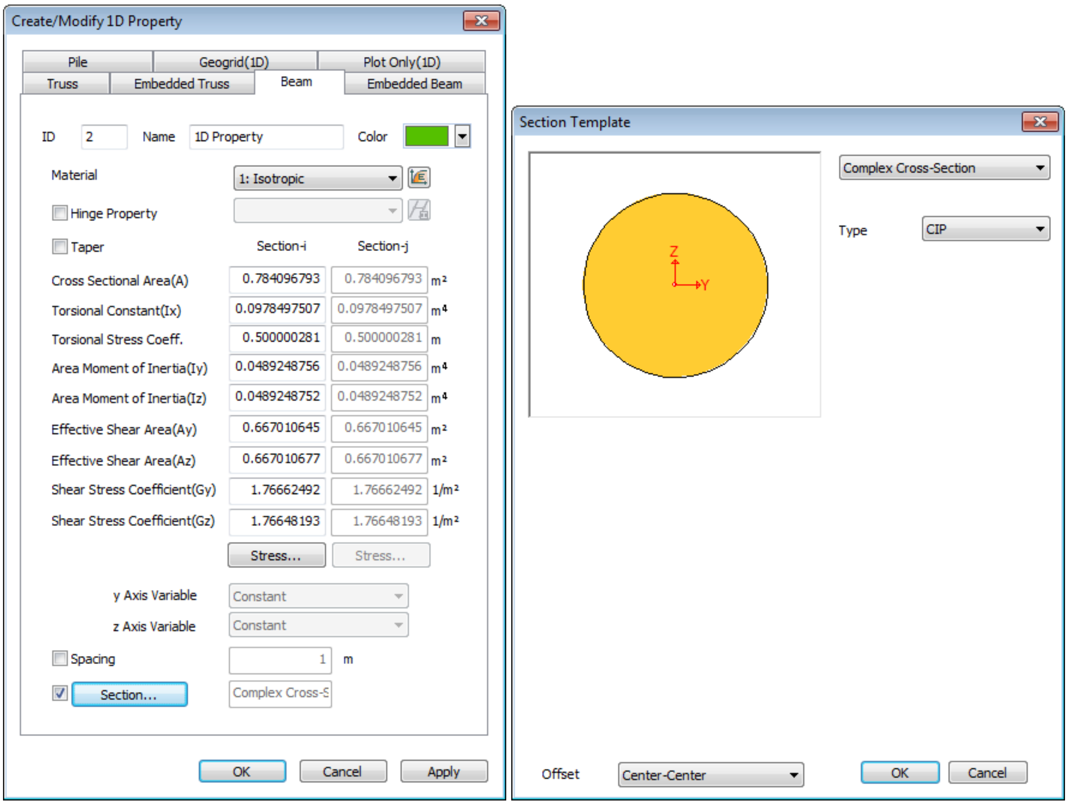
Methodology

[Auto-calculation of complex cross-section property]
Create 2D mesh for the complex cross-section.
Click ‘Add’ in the Complex Cross-Section Property window. If 2D mesh is selected, the cross-section property is calculated automatically.
Save the cross-section property.





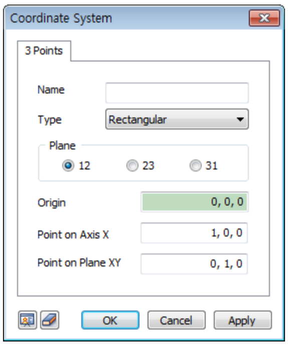
[Create/Modify 1D Property > Section Template]
The complex cross-section property can be assigned when creating 1D structural element.
The registered complex cross-section property is available if the 'Complex Cross-Section' is selected in the 'Section Template'.



1.7 Coordinate System

Overview

Add a result output coordinate system for 2D/3D element. The default rectangular/cylindrical coordinate system is defined and another coordinate system can be added by arbitrarily defining 1 of the three planes on the coordinate system. The added coordinate system can be used as the material coordinates when the properties of the 2D / 3D elements are defined. For 2D structural members, it is especially hard to unify the element coordinate system in 1 direction depending on the element shape. It is useful to set the output coordinate system to check the member forces of the entire structure according the same direction and sign convention.



Methodology

There are 2 types of coordinate systems; rectangular and cylindrical. The coordinate system can be set by defining 1 of the three planes of the coordinate system 12, 23, 31 (XY,YZ,ZX) by entering three points. In other words, the plane that passes the three points becomes 1 of the 12, 23, 31 planes and the other 2 planes are automatically determined by the direction of the reference plane.

Select the plane, origin point, a point on 1 axis of the selected plane and a point on the selected plane in this order to specify the position and direction of the reference plant.

1.8 Function

Overview

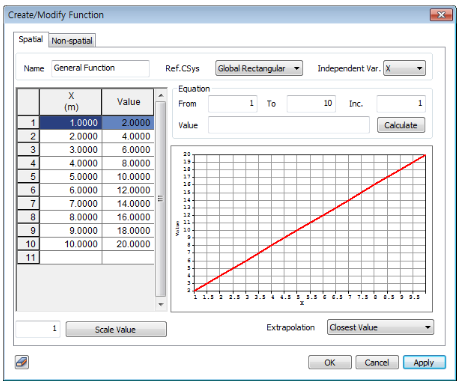
When setting the analysis conditions (boundary, loading etc.) after the mesh has been generated, the values that change with position and time can be registered as a function. The provided function types are as follows and the characteristics and applicable range for each function is given.

Methodology

Function type	Applicable range		
	Material/Property	**Boundary condition**	**Loading**
General (Spatial)	-	Water level	Force, Displacement, Pressure, Prestress, Initial equilibrium force, Dynamic nodal, Dynamic surface
General (Non-spatial)	Pile, Pile tip	-	-
Generalized space	-	Water level	Force, Displacement, Pressure, Prestress, Initial equilibrium force, Dynamic nodal, Dynamic surface
Surface Function	-	Water level	-
Non-Hydrostatic Water Pressure	-	Water level	-
Creep / Shrinkage Strain Function	Time-dependent (User-defined)	-	-
Creep/Shrinkage Function Group	Time-dependent (Design code based)	-	-
Elastic Modulus Function	Time-dependent (Design code based)	-	-
Plastic Hardening Function	Shear Hardening (User-defined)	-	-
Hardening Curve	von Mises	-	-
Stress-Strain Curve	von Mises	-	-
Cohesion Hardening Curve	CWFS	-	-
Frictional Angle Hardening Curve	CWFS	-	-
Dilatancy Angle Hardening Curve	CWFS	-	-
Tensile Strength Hardening Curve	CWFS	-	-
Seepage boundary	-	Node head, Node flux, Surface flux	-
Nonlinear elastic (Truss element)	Truss, Embedded truss	-	-
Nonlinear elastic (Point spring/ Elastic link)	Point spring, Elastic link	-	-
Unsaturated property	Isotropic, Orthotropic	-	-
Strain compatible	2D Equivalent	-	-
Response spectrum	-	-	Response spectrum
Time forcing	-	-	Ground acceleration, Time varying static, Dynamic nodal, Dynamic surface

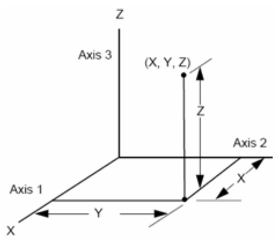
General function (Spatial)

Changes in position (coordinates) of a value with respect to the rectangular or cylindrical coordinate system can be set as function and used when specifying the load. The water level conditions can be directly specified on the screen, but registering the water level according to the coordinate change as a function allows this to be used as the water line during the construction step.

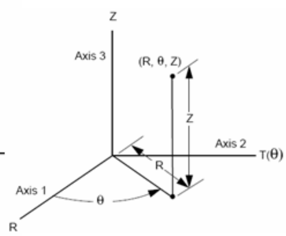


Input the independent variables (X,Y,Z or R,TH,Z) according to the reference coordinates and the values of the set variables in the table to generate a function . Already made functions can be copy + pasted from Excel.

▸Reference coordinate system standard



<Rectangular Coordinate>



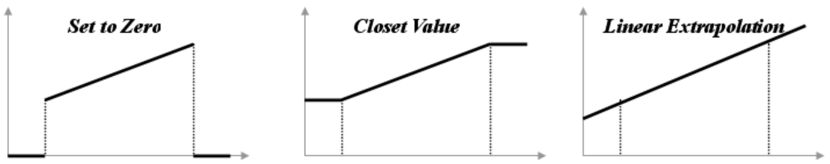
<Cylindrical Coordinate>

Equation
The Equation can be set without defining the values of the independent variables.
For example, when defining the function Y=2*X on the Cartesian coordinate system, specify the X coordinate range (start, end) and the X coordinate increment first. Input 2*X and press the calculate button to automatically generate a function as shown above.
When defining the function Sin(T) between a certain angle with reference to the T coordinate on the cylindrical coordinate system , specify the angle range (start, end) and increment angle and then input sin(T) to generate a sin function within the specified angle range.

Scale Value
Value multiplied to the defined function value. The initial value is set as 1. For example, to increase all defined functions by 2 times, input 2 for the scale value.

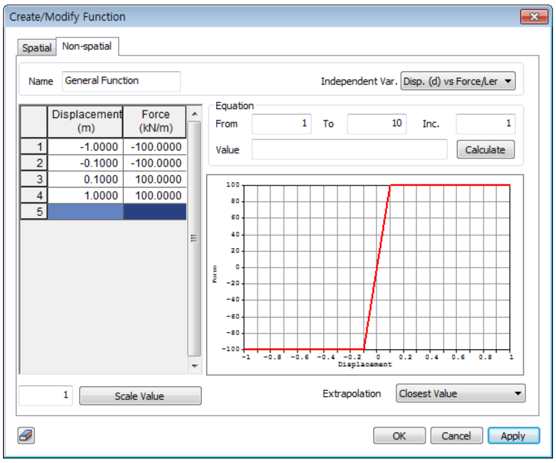
Extrapolation
Assign a function value to values outside the independent variable range. The user can select whether to set the function value outside the range to be 0, the same as the closest function value or determined through linear extrapolation.

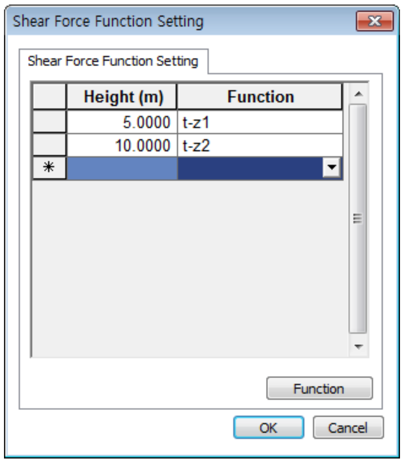
▸Extrapolation



General function (Non-spatial)

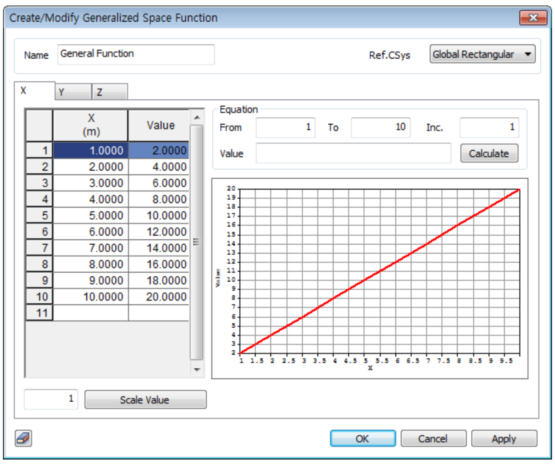
Used to specify the shear stiffness and spring stiffness of pile or pile tip elements as a function. The relative displacement vs Force/Area is used as a shear stiffness function and the relative displacement vs Force/Length is used as a spring stiffness function at the pile tips. Different pile shear stiffness functions can be set for each depth when defining the pile material properties.





Generalized space function

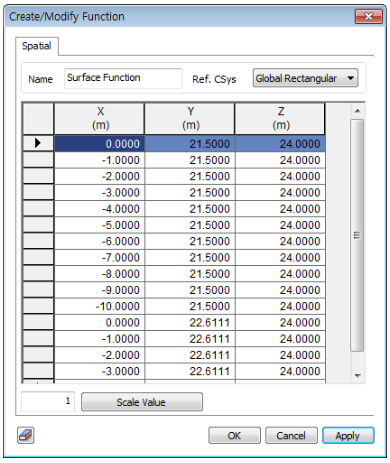
The purpose and functions are the same as the general function (spatial). However, the generalized space function is able to generate a function that considers all 3 axis directions while the general function (spatial) is a 1 dimensional function that can only set 1 independent variable axis. The input method and detailed functions are the same as the general function (spatial).



Input the independent variables (X,Y,Z or R,TH,Z) according to the reference coordinates and the values of the set variables in the table to generate a function . Already made functions can be copy + pasted from Excel.

Surface function

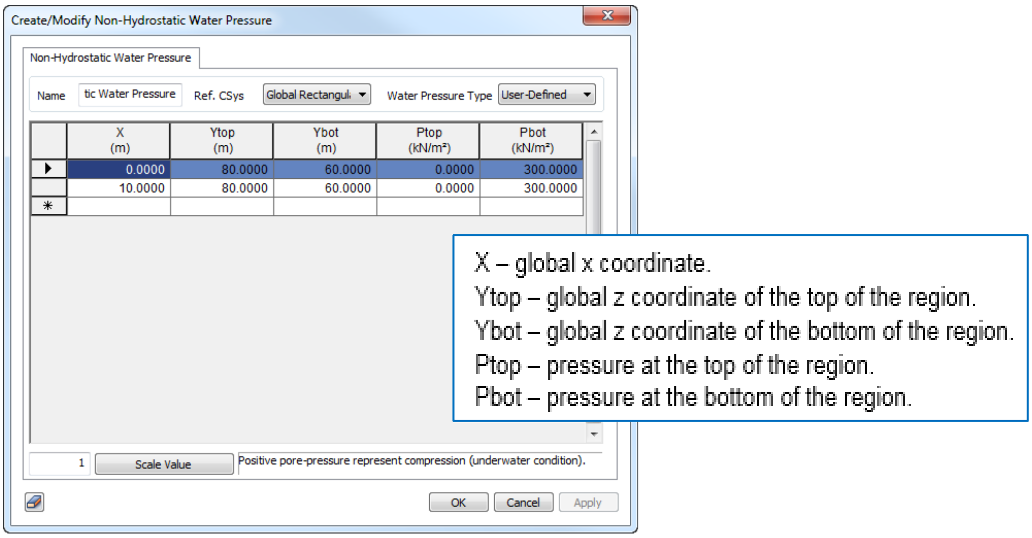
Used to specify the water level surface in 3D space. The 3D water level surface can be generated by entering the variable values with reference to the global or cylindrical coordinates as shown in the table below, but it can also be generated using the Boundary Condition > Water Level operation and selecting a surface in space to automatically extract the coordinate information. Here, the X axis spacing determines the precision of the water surface and for suddenly changing sections, the spacing needs to be carefully set following the element node positions.



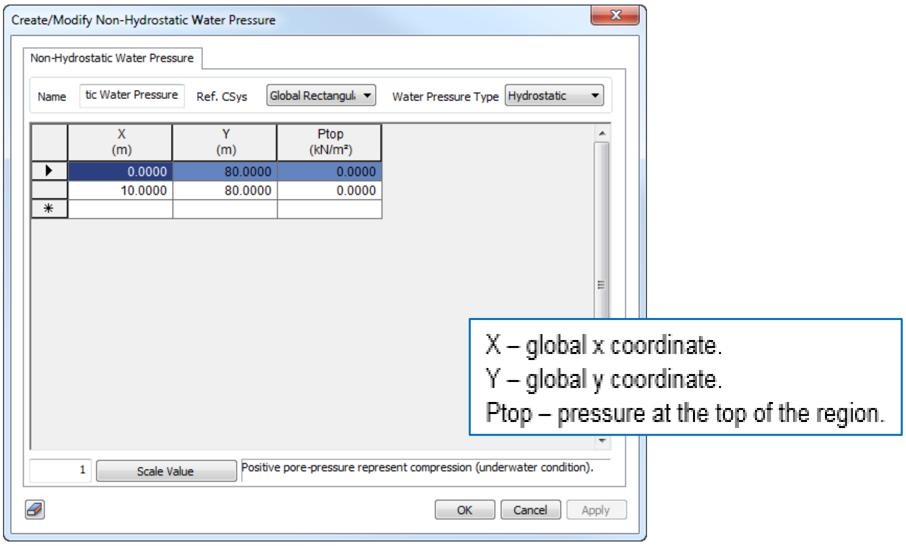
Non-hydrostatic Water Pressure

Define the water pressure at the top and bottom depending on the position of the model. Applicable in integer conditions and user-defined conditions when specifying the mesh set level.

- User-defined condition: the user can directly define the water pressure at the top and bottom of the assigned mesh by using the specific water pressure function.



- Hydrostatic condition: this function is used to define the water pressure at the top of the allocated mesh as hydrostatic



Name – input name for the function.

Ref. CSys – reference coordinate system for the function definition.

Water Pressure Type – select the type of function for selection in Analysis Case > Analysis Control > Define Water Level for Mesh Set. (User Defined or Hydrostatic condition)

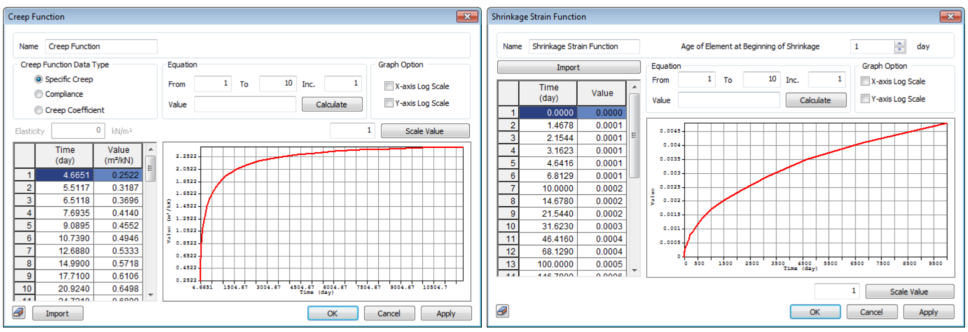
Scale factor to multiply all input data.

Creep / Shrinkage Strain Function

Define the time dependent creep property and shrinkage strain of concrete material. These functions are available if User Defined is selected in the Code field.

There are three types of Creep function data types.

- ✓ Specific Creep ; Strain per unit stress excluding immediate deformation
- ✓ Creep Function ; Strain per unit stress including immediate deformation
- ✓ Creep Coefficient : Ratio of creep strain to elastic strain

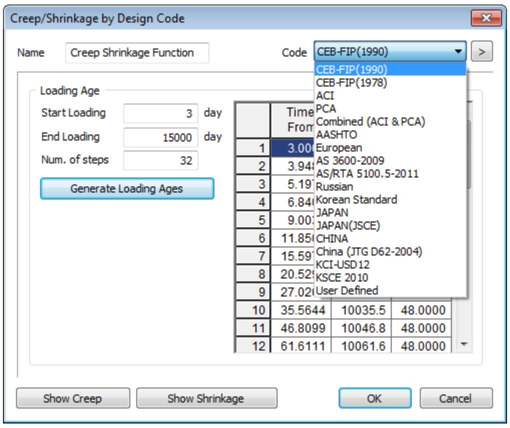


A frequently used Creep Function may be saved in and recalled from a file. The data file, fn.TDM retains the following form:

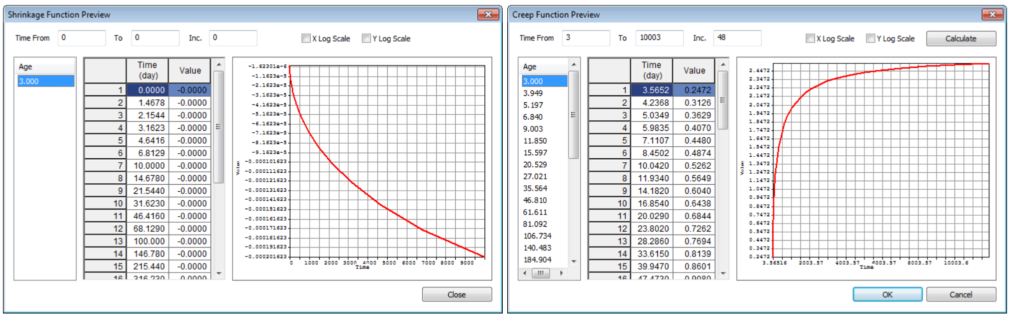
* Unit, in, kip	
* Data	Assign the units (optional items)
20, 0.9934	Enter the data in the form of 'day' and 'value' (mandatory items)
40,1.2182	-
60, 1.3705	-
80,1.4883	-
100, 1.5854	-
120, 1.6683	-
140, 1.7408	-
160, 1.8054	-
180, 1.8636	-
200, 1.9166	-
220, 1.9653	-
240, 2.0103	-
260, 2.0252	-

Creep / Shrinkage Function Group

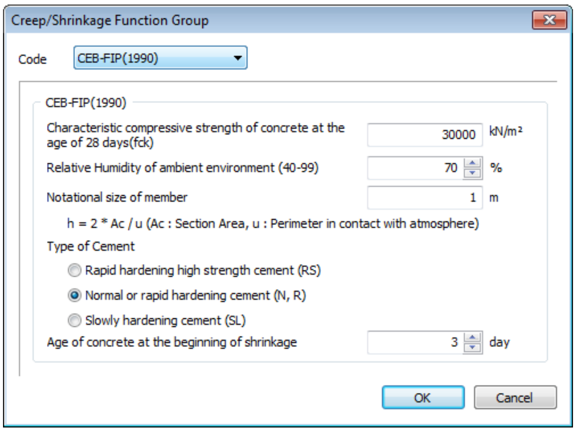
The user can define Creep/Shrinkage Function based on the embedded Design Codes as follows.



▸Show Creep / Shrinkage



[CEB-FIP(1990)]



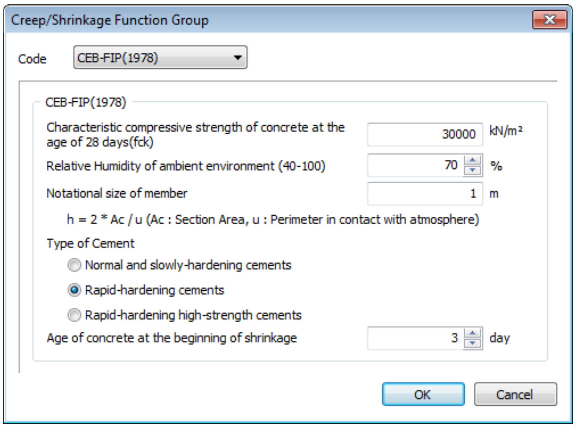
Notational size of member

- Conceptual (Equivalent) size of structure. Two times of equivalent area divided by perimeter of the member

Age of concrete at the beginning of shrinkage

- The number of days elapsed after pouring of concrete, when the shrinkage is assumed to start

[CEB-FIP(1978)]



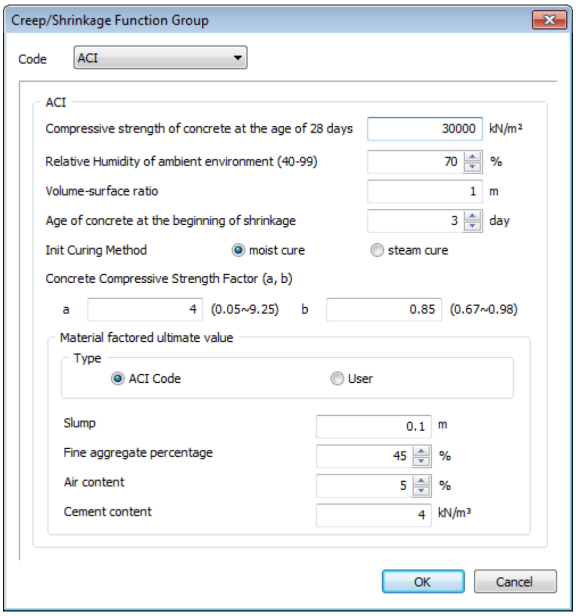
Notational size of member

- Conceptual (Equivalent) size of structure. Two times of equivalent area divided by perimeter of the member

Age of concrete at the beginning of shrinkage

- The number of days elapsed after pouring of concrete, when the shrinkage is assumed to start

[ACI]



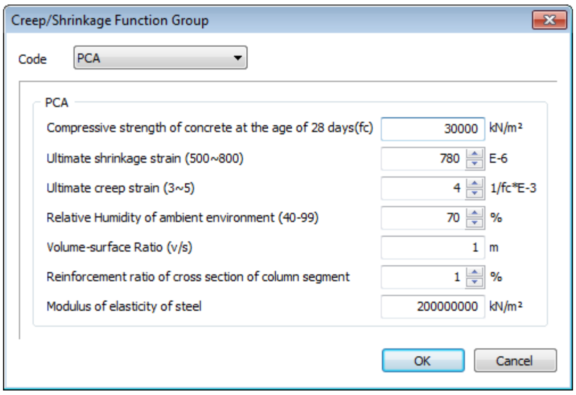
<u>**Age of concrete at the beginning of shrinkage**</u>

- The number of days elapsed after pouring of concrete, when the shrinkage is assumed to start

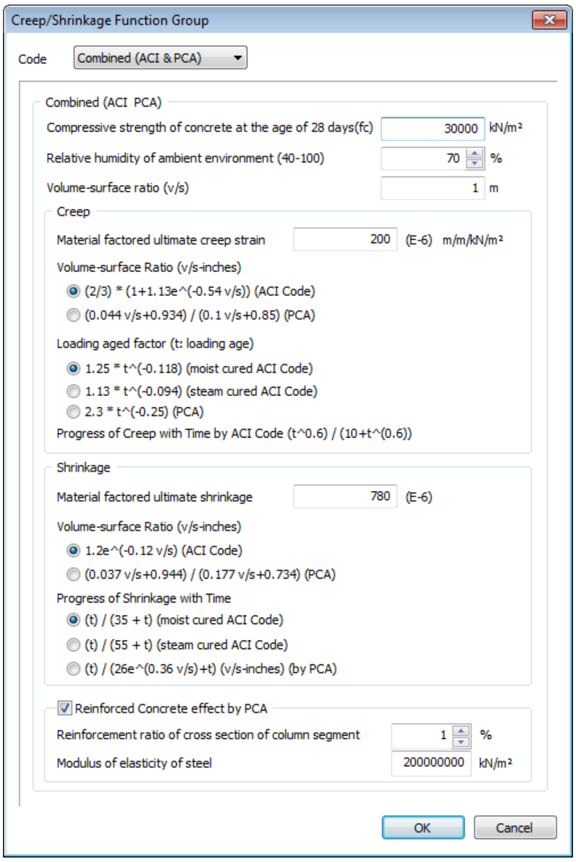
<u>**Material factored ultimate value**</u>

- The user may enter the ultimate values considering concrete properties by ACI code or User type.

[PCA]



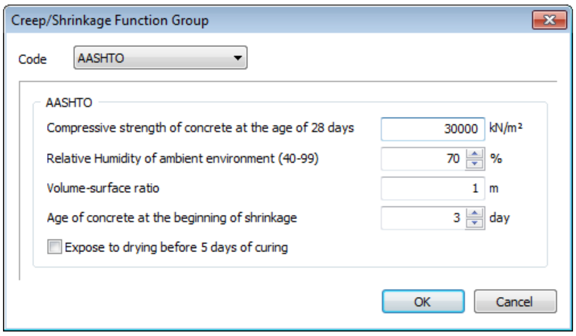
[Combined (ACI & PCA)]



Material factored ultimate creep strain / Shrinkage

- The user may enter the ultimate values considering concrete properties by User type.

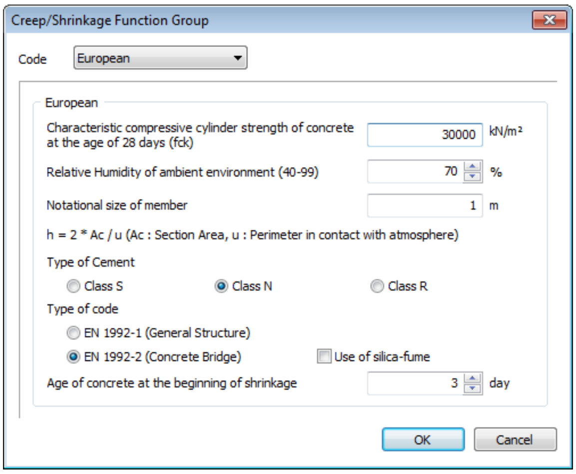
[AASHTO]



Expose to drying before 5 Days of curing

- If this option is checked on, the influence of Creep & Shrinkage is increased by 20% (ref. AASHTO 5.4.2.3)

[European]



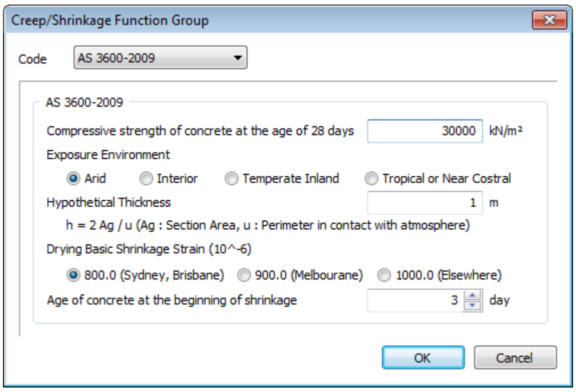
Notational size of member

– Conceptual (Equivalent) size of structure. Two times of equivalent area divided by perimeter of the member

Age of concrete at the beginning of shrinkage

– The number of days elapsed after pouring of concrete, when the shrinkage is assumed to start

[AS 3600-2009]



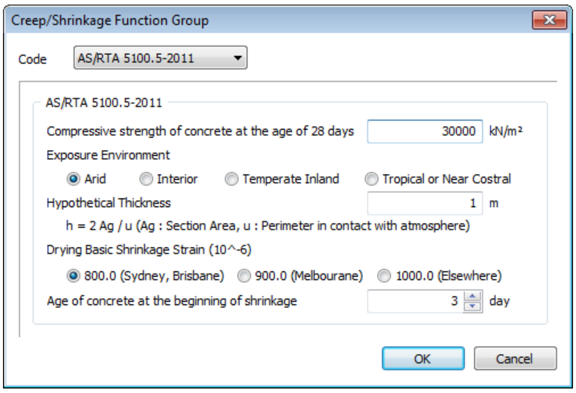
Hypothetical Thickness

– Conceptual (Equivalent) size of structure. Two times of equivalent area divided by perimeter of the member

Age of concrete at the beginning of shrinkage

– The number of days elapsed after pouring of concrete, when the shrinkage is assumed to start

[AS/RTA 5100.5-2011]



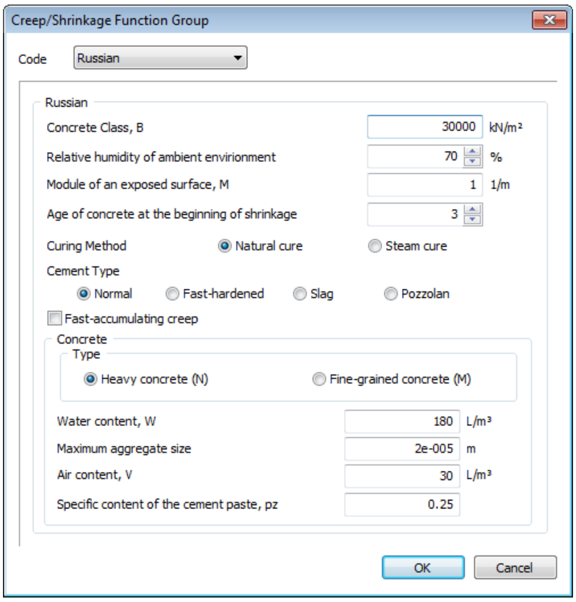
Hypothetical Thickness

- Conceptual (Equivalent) size of structure. Two times of equivalent area divided by perimeter of the member

Age of concrete at the beginning of shrinkage

- The number of days elapsed after pouring of concrete, when the shrinkage is assumed to start

[Russia]



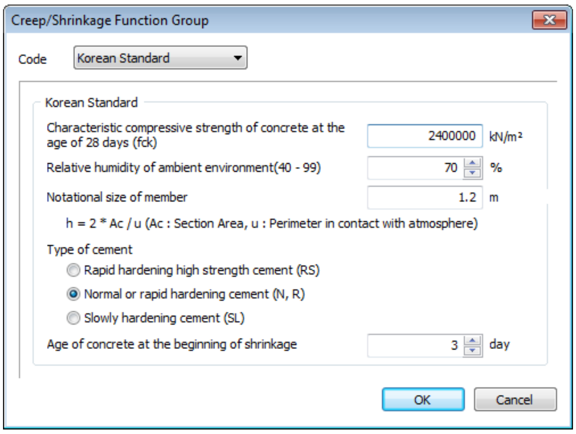
Age of concrete at the beginning of shrinkage

- The number of days elapsed after pouring of concrete, when the shrinkage is assumed to start

Fast – accumulating creep

- If this option is checked on, the influence of Creep & Shrinkage will be increased by % based on Russian code

[Korean Standard]



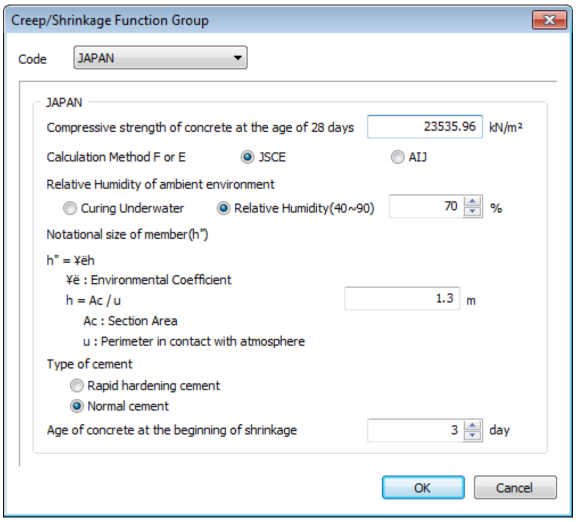
Notational size of member

– Conceptual (Equivalent) size of structure. Two times of equivalent area divided by perimeter of the member

Age of concrete at the beginning of shrinkage

– The number of days elapsed after pouring of concrete, when the shrinkage is assumed to start

[Japan]



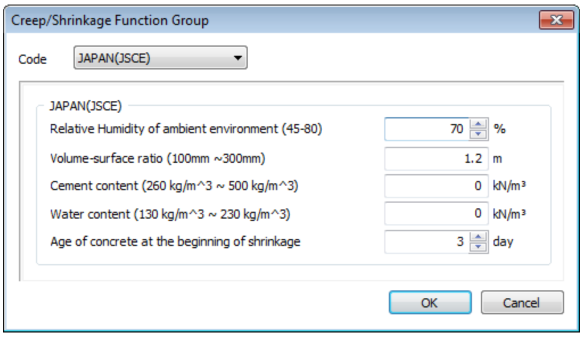
Notational size of member

– Conceptual (Equivalent) size of structure. Equivalent area divided by perimeter of the member considering Environmental Coefficient.

Age of concrete at the beginning of shrinkage

– The number of days elapsed after pouring of concrete, when the shrinkage is assumed to start

[Japan (JSCE)]



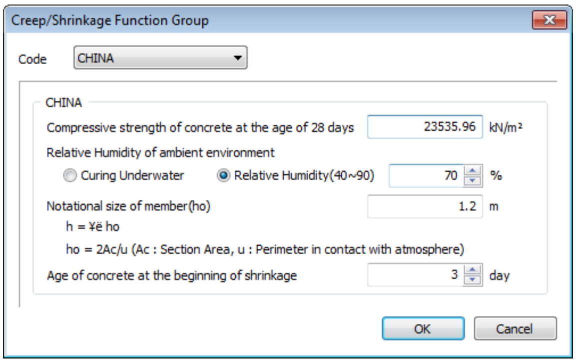
Cement content / Water content

- Required to input each content per unit volume to generate Creep/Shrinkage Function automatically

Age of concrete at the beginning of shrinkage

- The number of days elapsed after pouring of concrete, when the shrinkage is assumed to start

[CHINA]



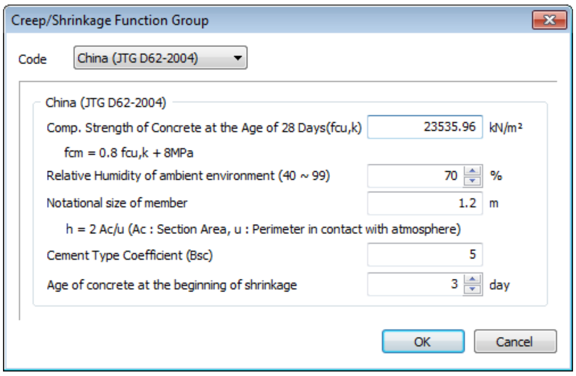
Notational size of member

- Conceptual (Equivalent) size of structure. Two times of equivalent area divided by perimeter of the member considering Environmental coefficient

Age of concrete at the beginning of shrinkage

- The number of days elapsed after pouring of concrete, when the shrinkage is assumed to start

[China (JTG D62-2004)]



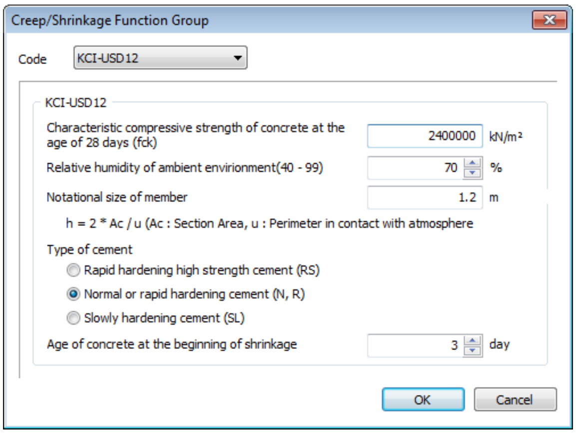
Notational size of member

– Conceptual (Equivalent) size of structure. Two times of equivalent area divided by perimeter of the member.

Age of concrete at the beginning of shrinkage

– The number of days elapsed after pouring of concrete, when the shrinkage is assumed to start

[KCI-USD 12]



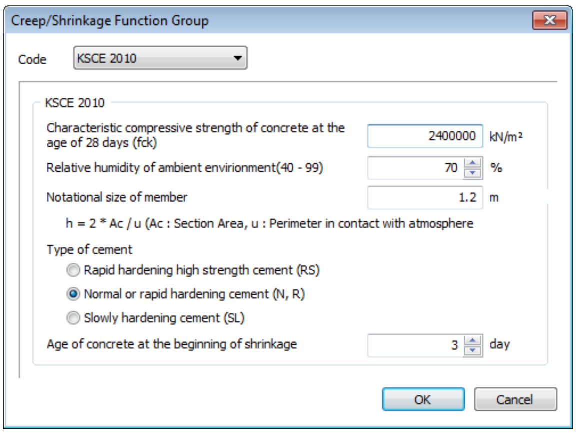
Notational size of member

– Conceptual (Equivalent) size of structure. Two times of equivalent area divided by perimeter of the member

Age of concrete at the beginning of shrinkage

– The number of days elapsed after pouring of concrete, when the shrinkage is assumed to start

[KSCE 2010]



Notational size of member

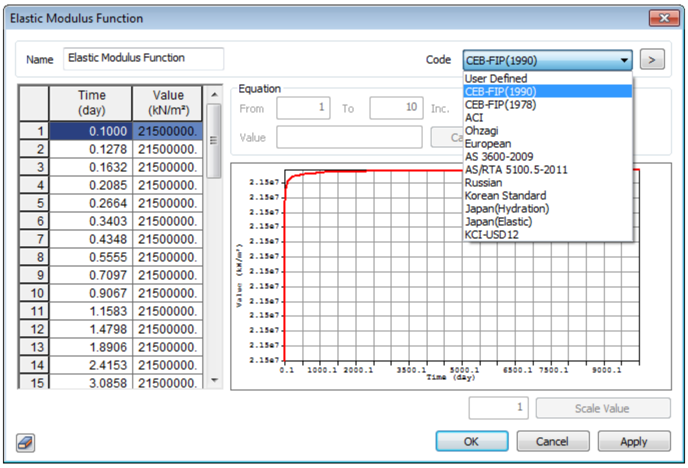
- Conceptual (Equivalent) size of structure. Two times of equivalent area divided by perimeter of the member

Age of concrete at the beginning of shrinkage

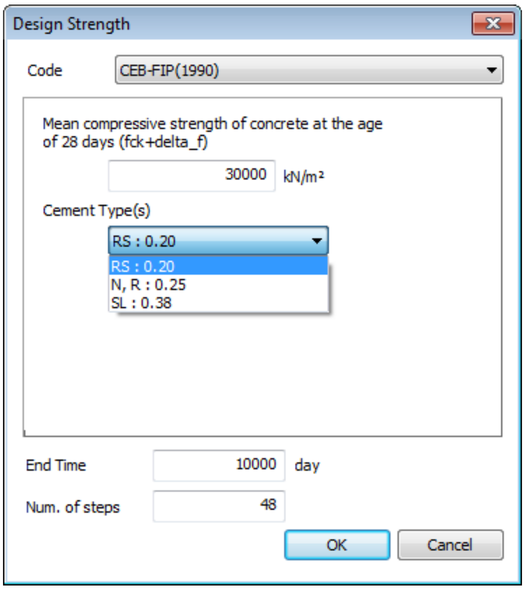
- The number of days elapsed after pouring of concrete, when the shrinkage is assumed to start

Elastic Modulus Function

Define Time-dependent Elastic modulus function based on selected design code. It is required to input End Time of function with the number of steps.



[CEB-FIP(1990)]



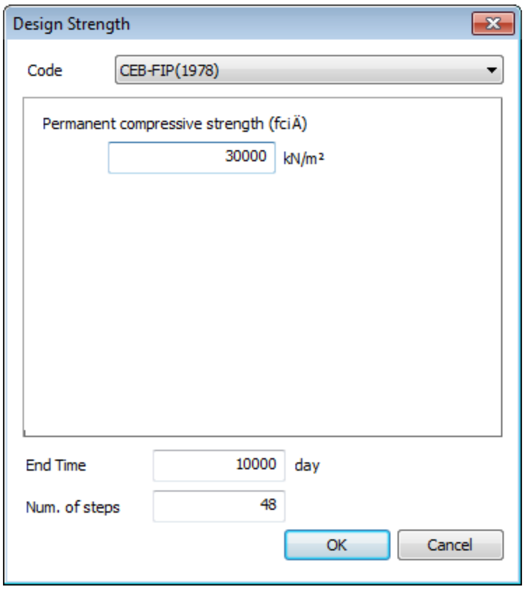
Specify the Concrete Compressive Strength at 28 Days and Cement Type

RS – Rapid hardening high strength cements

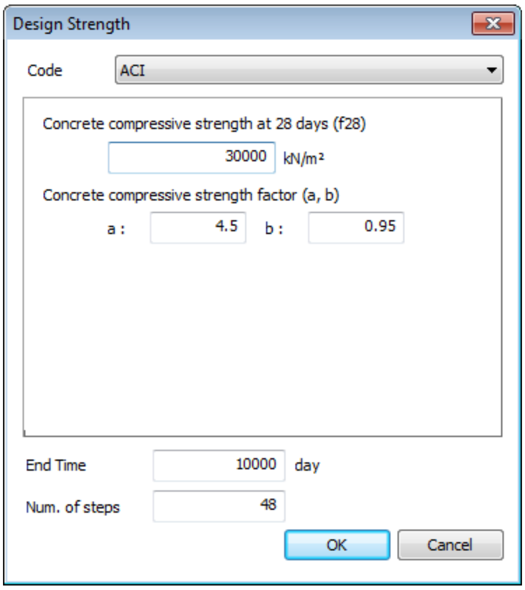
N,R – Normal or rapid hardening cements

SL – Slowly hardening cements

[CEB-FIP(1978)]



[ACI]



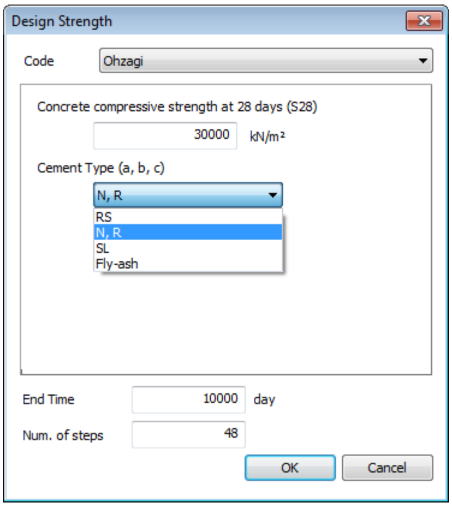
Modulus of elasticity, which will be reflected in the analysis, is calculated using the compressive strength of concrete and weight density

$$E_c = w_c^{1.5} 0.043 \sqrt{f_c'} \text{ (in MPa)}$$

w_c : Density of concrete, kg/m3

f_c' : Compressive strength of concrete, MPa

[Ohzagi]



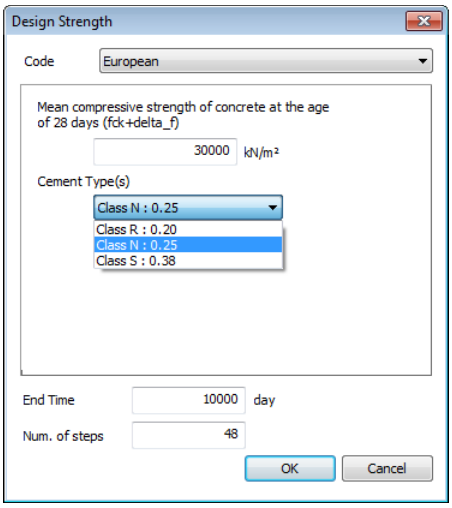
The equation proposed by Ohzagi is used to define the change of compressive strength of concrete. Specify the Concrete Compressive Strength at 28 Days and Cement Type

RS – Rapid hardening high strength cements

N,R – Normal or rapid hardening cements

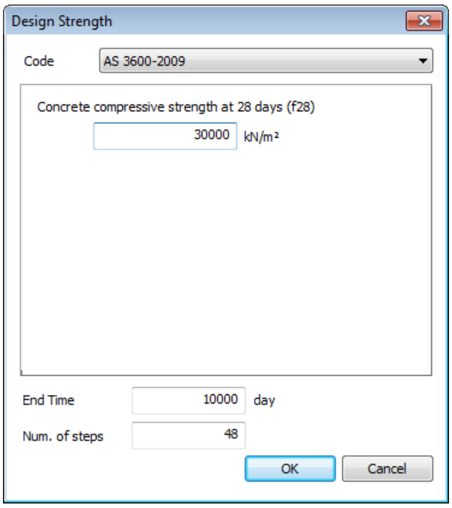
SL – Slowly hardening cements
Fly ash – Fly ash cementing material

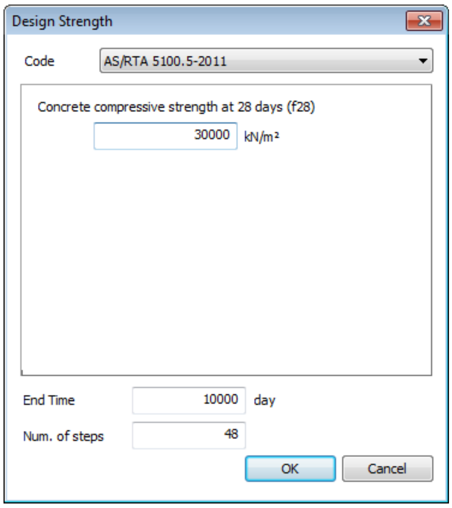
[European]



Specify the Concrete Compressive Strength at 28 Days and Cement Type
RS – Rapid hardening high strength cements
N,R – Normal or rapid hardening cements
SL – Slowly hardening cements

[AS 3600-2009] / [AS/RTA 5100.5-2011]





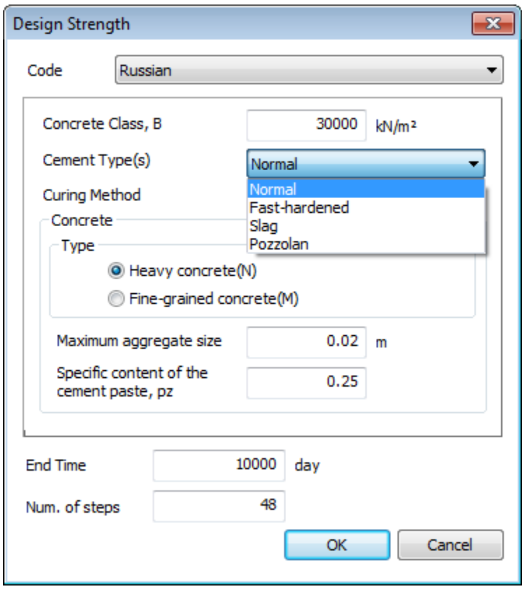
Mean modulus of elasticity of concrete at the appropriate age is calculated as follows.

$$f_{cmi} \leq 40 \text{ MPa}, \; E_{cj} = (\rho^{1.5}) \, x \, (0.043 \sqrt{f_{cmi}})$$
$$f_{cmi} > 40 \text{ MPa}, \; E_{cj} = (\rho^{1.5}) \, x \, (0.024 \sqrt{f_{cmi}} + 0.12)$$

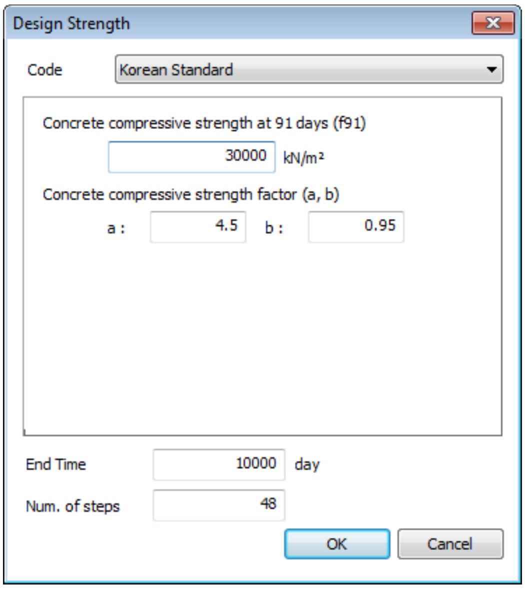
Since there is no equation for Compressive Strength at the appropriate age in Australian Standard, it is calculated based on the following equation specified in CEB-FIP 1978.

$$f(t)=\frac{1.451849874\times t^{0.75}\times f_c'}{t^{0.75}+5.5}$$

[Russian]



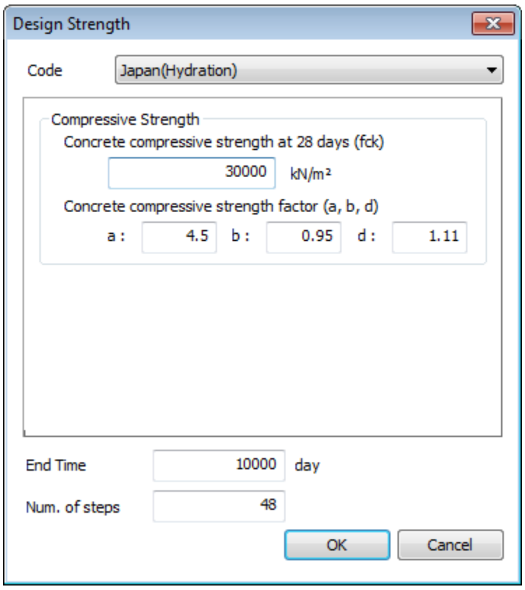
[Korean Standard]



Specify the Concrete Compressive Strength at 91 Days and Strength Factor (a,b)

Cement Type	a	b
Rapid strength	2.9	0.97
Normal	4.5	0.95
Moderate heat	6.2	0.93

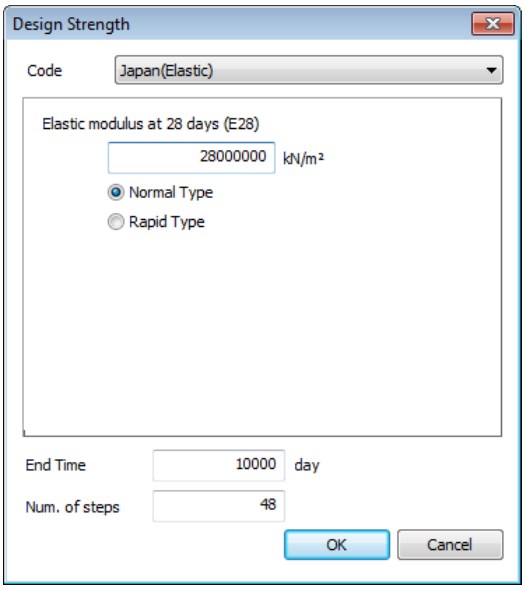
[Japan (Hydration)]



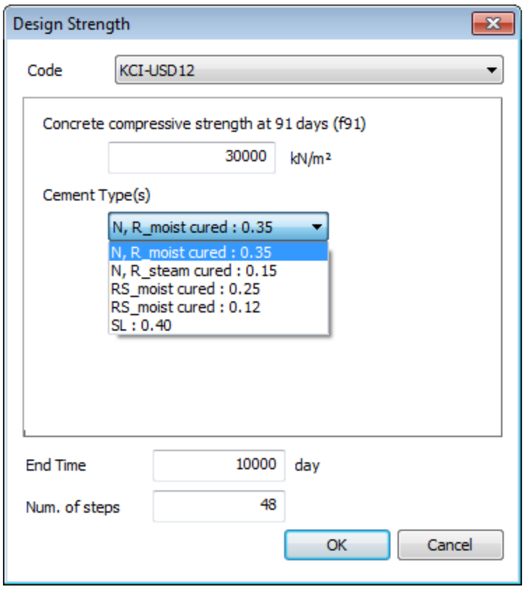
Specify the Concrete Compressive Strength at 28 Days and Strength Factor (a,b,d)

Cement Type	a	b	d
Rapid strength	4.5	0.95	1.11
Normal	6.2	0.93	1.15
Moderate heat	2.9	0.97	1.07

[Japan (Elastic)]



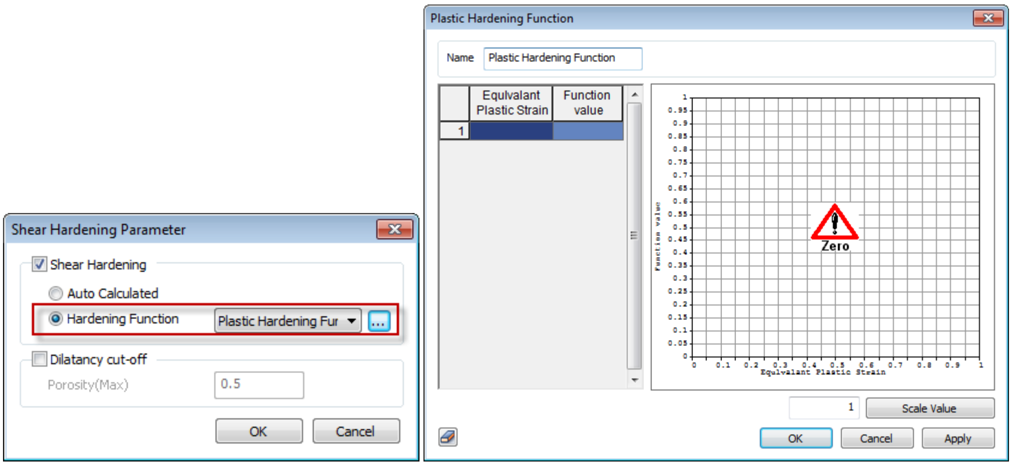
[KCI-USD12]



Specify the Concrete Compressive Strength at 91 Days and Strength Factor (a,b)
N,R – Normal or rapid hardening cements
RS – Rapid hardening high strength cements
SL – Slowly hardening cements

Plastic Hardening Function

Shear Hardening in Modified Mohr-Coulomb model can be defined by Equivalent plastic strain related to the mobilized shear resistance. Using "Auto-Calculated" option, solver recalculates friction angle based on the deviatoric plastic strain.

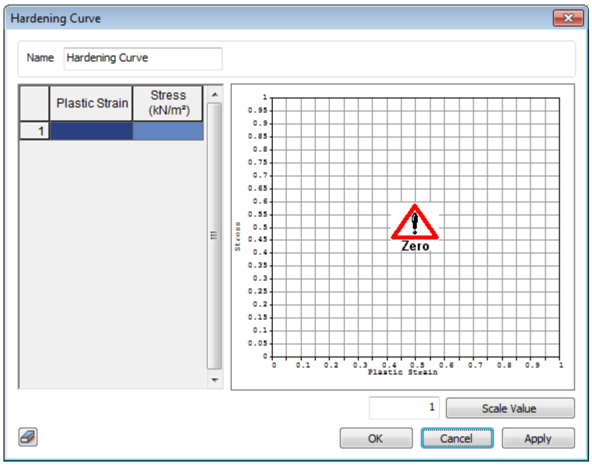


Hardening Curve

Plastic strain and stress are directly inputted. It is used when defining hardening curve within von Mises model. Plastic strain begin once the material starts yielding so it can be calculated using following equation.

$$e_p = \varepsilon - \varepsilon^{el} = \varepsilon - \frac{\sigma}{E}$$

Where, E : Young's modulus



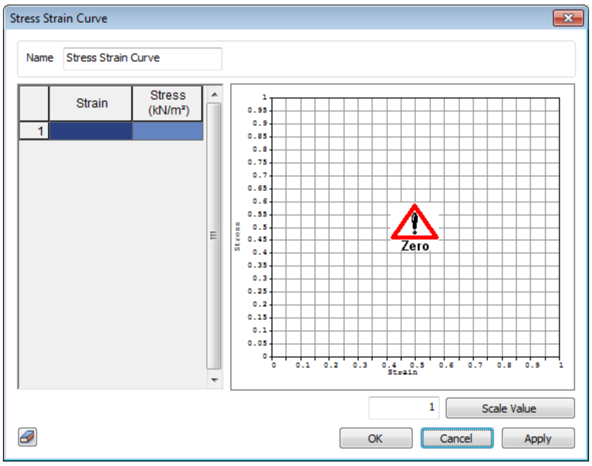
Stress-Strain Curve

Stress and stain are directly inputted. It is used when defining Stress-Strain Curve within von Mises model. When Load-Displacement Curve is already known, true strain can be calculated using following equation.

$$\varepsilon = \log\left(\frac{L_0 + d}{L_0}\right) = \log\left(\frac{L}{L_0}\right), \quad \sigma = \frac{Pe^{\varepsilon}}{A_0}$$

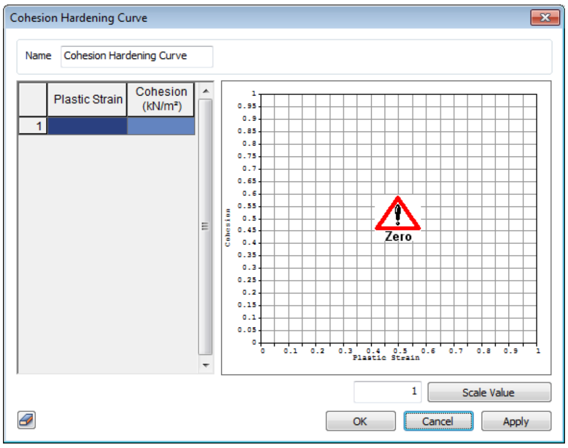
Where, L_0, L : Length before/after deformation

A_0 : Area before deformation



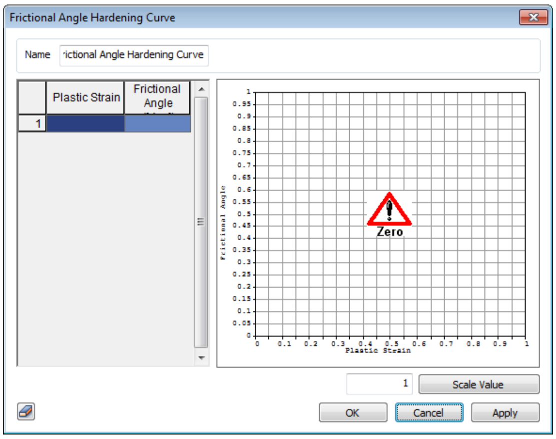
Cohesion Hardening Curve

Plastic strain and cohesion are directly inputted. It is used when describing Cohesion Hardening Curve within CWFS(Cohesion Weakening and Frictional Strengthening).



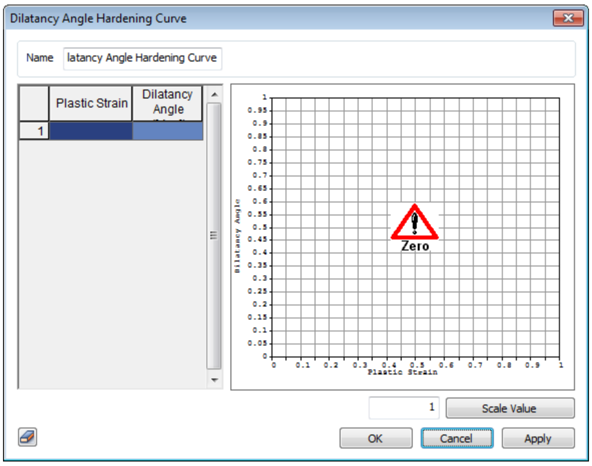
Frictional Angle Hardening Curve

Plastic strain and frictional angle are directly inputted. It is used when describing Frictional Hardening Curve within CWFS(Cohesion Weakening and Frictional Strengthening).



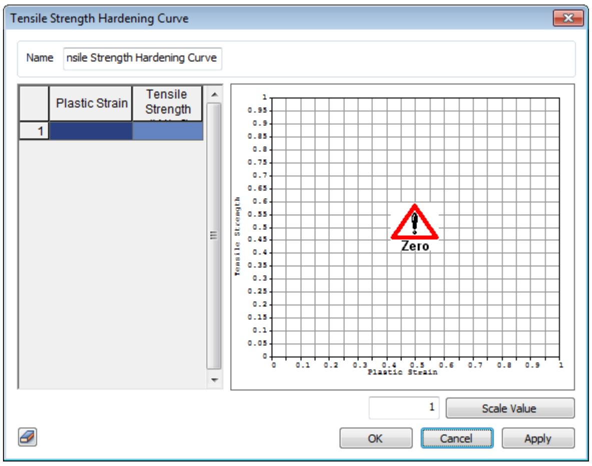
Dilatancy Angle Hardening Curve

Plastic strain and dilatancy angle are directly inputted. It is used when describing Dilatancy Hardening Curve within CWFS(Cohesion Weakening and Frictional Strengthening).



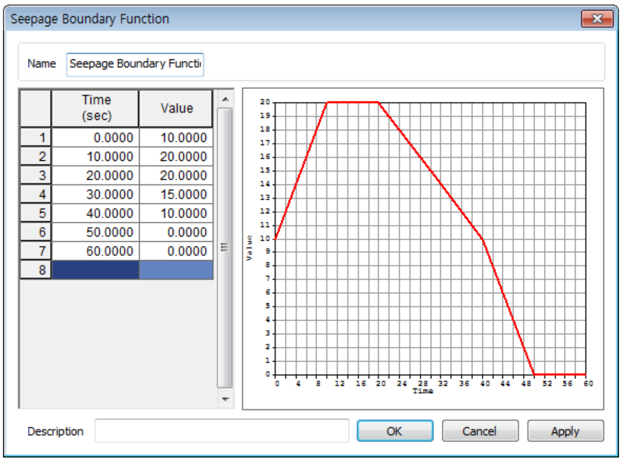
Tensile Strength Hardening Curve

Plastic strain and tensile strength are directly inputted. It is used when describing Tensile Strength Hardening Curve within CWFS(Cohesion Weakening and Frictional Strengthening).



Seepage Boundary function

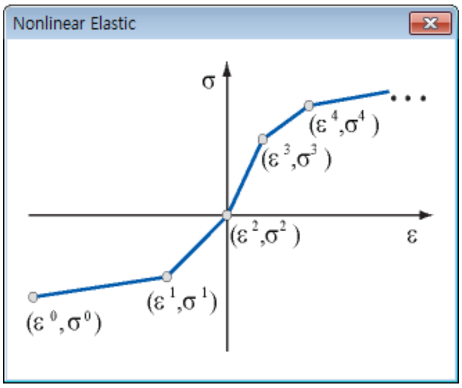
Used to simulate the head and flow rate difference with time such as node head, node flux rate, surface flux. Setting the values (head/flux) according to the time flow that fits the specified time unit generates a function that can be applied to unsteady infiltration analysis.



For unsteady infiltration analysis, the time steps for result verification are set separately and the time function for the steps is used in the analysis. When the time step is beyond the range of the function, the function value is automatically calculated using linear interpolation. In other words, to apply a 0 function value to time steps outside the range, the same function value (0) needs to be entered for arbitrary time steps when creating the function as shown in the figure above.

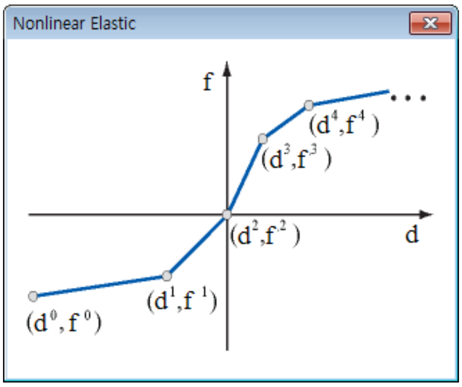
Nonlinear elastic function (Truss element)

The behavioral characteristics of truss elements or embedded truss elements can be defined as nonlinear elastic. The stress change according to strain of the truss element can be directly generated as a function and used. The function can be applied to the tensile (compression) test results of a structural member (truss affiliated) or to the deformation behavior properties of a general steel member.



Nonlinear elastic function (Point spring/ Elastic link)

The behavioral characteristics of spring or elastic link elements can be defined as nonlinear elastic. The spring/link stiffness according to the element strain can be defined to generate a function.



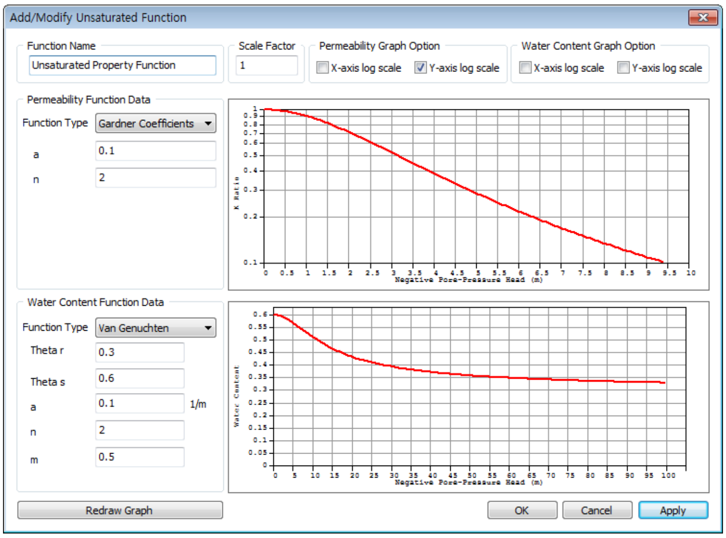
For steady infiltration analysis that assumes a saturated ground, the unsaturated properties are not taken into account even if they are entered. For the time dependent unsteady analysis, the unsaturated properties of the ground must be considered. Also, because real foundations are unsaturated and have a certain ratio of air, unsteady infiltration analysis needs to consider unsaturated characters of the soil for more realistic results.

Unsaturated property defines the change in permeability coefficient and water content (Degree of saturation) in the unsaturated region depending on the size of the negative pore water pressure. There are 2 methods to define the unsaturated property; directly defining (define individually) the permeability function and water content function using the pressure head (negative pore water pressure) or defining the relationship between pressure head-volumetric water content (degree of saturation)-permeability ratio (define relationship).

Individually

Define the permeability function data and water content function data. Based on the experimental data of unsaturated soil, the coefficients for each function type can be defined using Curve Fitting or the data can be directly input through customize. If the experimental data is entered directly, the negative pore water pressure size is entered by its absolute value and the permeability ratio is found by dividing the unsaturated value with the saturated state value.

▸ Individual consideration



The provided permeability functions and coefficients are listed below.

[Gardner coefficient]

$$R_K = \frac{1}{1 + a|h|^n}$$

$\mathbf{R_k}$: Permeability ratio (permeability coefficient according to h increase / permeability coefficient at h=0)
a, n : Experimental constants predicted from Curve Fitting of unsaturated experimental data
[Frontal function]

$$R_k = \begin{cases} l & if \quad H = 0 \\ (R_{k0} - l)\dfrac{H}{H_0} + l\varphi & if \quad 0 < H < H \\ R_{k0} & if \quad H \geq H_0 \end{cases}$$

K ratio (R_k) : Permeability ratio (permeability coefficient according to h increase / permeability coefficient at h=0)
H_o : Head at which the permeability coefficient no longer decreases

[Van Genuchten function (Permeability ratio)]

$$\mathbf{R_k} = \frac{[\mathbf{1} - (\mathbf{ah})^{\mathbf{n-1}} \cdot \{\mathbf{1} + (\mathbf{ah})^{\mathbf{n}}\}^{-\mathbf{m}}]^2}{[\mathbf{1} + (\mathbf{ah})^{\mathbf{n}}]^{\mathbf{m}/2}}$$

$\mathbf{R_k}$: Permeability ratio (permeability coefficient according to h increase / permeability coefficient at h=0)
a, n, m : Experimental constants predicted from Curve Fitting of unsaturated experimental data

[Van Genuchten (Water content)]

$$\theta_w = \theta_\gamma + \frac{\theta_s - \theta_\gamma}{\left[1 + (ah)^n\right]^m}$$

θ_w: Volume water content
θ_r : Residual volume water content
θ_s : Saturated volume water content
a, n, m : Experimental constants predicted from Curve Fitting of unsaturated experimental data

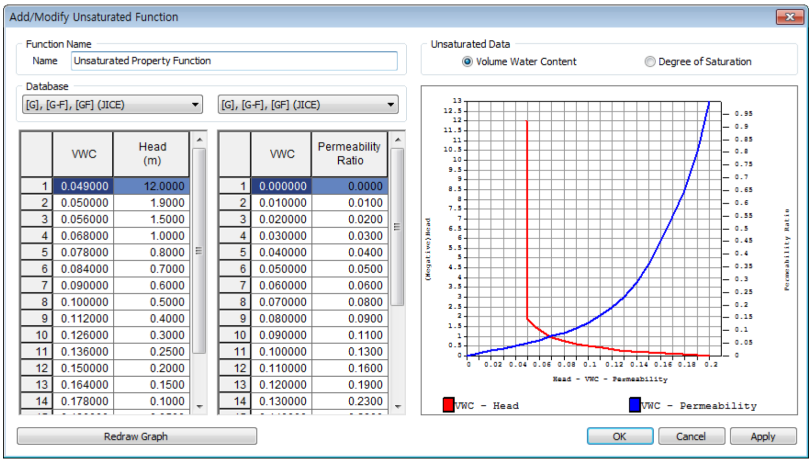
Relation

The unsaturated material property data can be set on the ground type according to the JICE (Japan Institute of Construction Eng.) criteria. The pressure head-volumetric water content (degree of saturation)-permeability ratio relationship function for each type of ground is shown below.

Pressure head(P)-Volumetric water content(T) -Permeability ratio (K)		**Pressure head(P)-Degree of saturation(Sr) -Permeability ratio(K)**	
(T)- (P)	(T)- (K)	(Sr)- (P)	(Sr)- (K)
Gravelly soil [G], [G-F], [GF] (JICE)	Gravelly soil [G], [G-F], [GF] (JICE)	Gravelly soil [G], [G-F], [GF] (JICE)	Gravelly soil [G], [G-F], [GF] (JICE)
Sandy soil [S], [S-F], [SF] (JICE)	Sandy soil [S], [S-F], [SF] (JICE)	Sandy soil [S], [S-F], [SF] (JICE)	Sandy soil [S], [S-F], [SF] (JICE)
Sandy soil [SF] (JICE)	Cohesive soil [M], [C] (JICE)	Sandy soil [SF] (JICE)	Cohesive soil [M], [C] (JICE)
Cohesive soil [M], [C] (JICE)	User specified	Cohesive soil [M], [C]	User specified

		(JICE)	
User specified		User specified	

‣Dual consideration

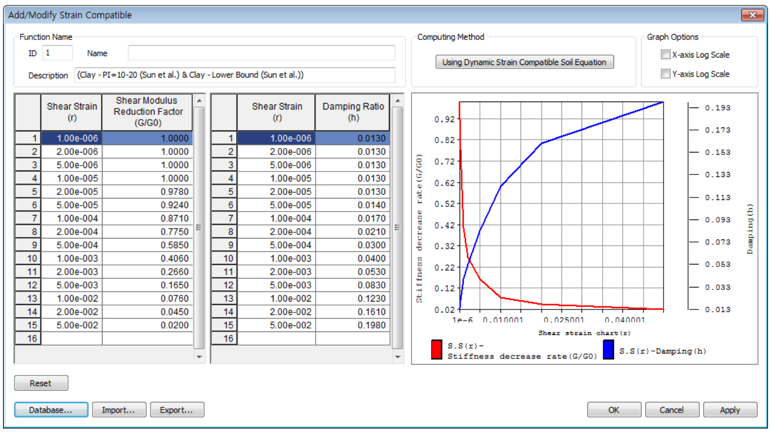


For unsteady analysis, the negative pressure head (pore water pressure) is calculated for each time step (construction step) and applied, renewing the relative permeability coefficient. The volumetric water content (degree of saturation), according to the calculated pressure head, is found and the relative permeability, according to the found volumetric water content (degree of saturation), is renewed for each step.

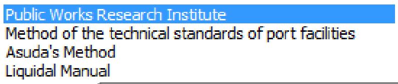
Strain compatible function

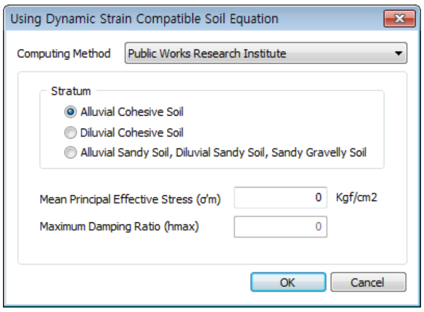
For the 2D equivalent linear analysis, the shear modulus and damping ratio are set as functions of strain to consider the nonlinearity and inelastic behavior of the ground. If the function is not defined, the ground material is assumed to be linear and the entered (fixed) shear modulus and damping ratio is applied. Generally, the ground displays decreasing shear modulus and increasing damping ratio as the shear strain increases. For complex nonlinear behavior of the ground, the material properties are simplified to linear equivalent properties. Repeated calculations from the assumed initial values can compute the converging shear modulus and damping ratio.

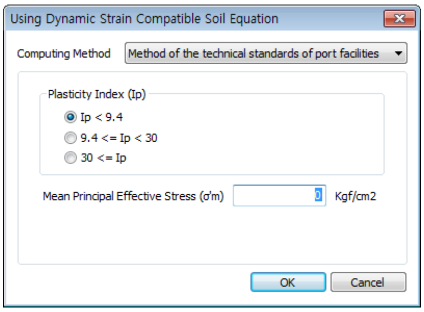
‣Dynamic Strain compatible function

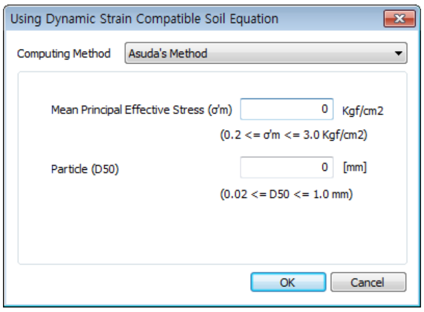


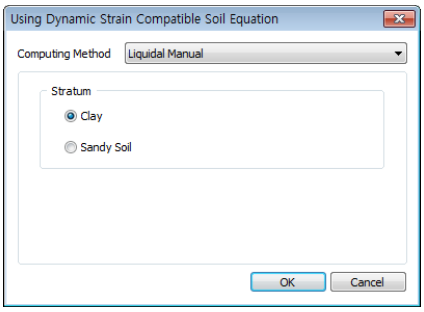
Material properties can be defined from various existing database. The following empirical equation can be used to generate a function following the stratum properties.





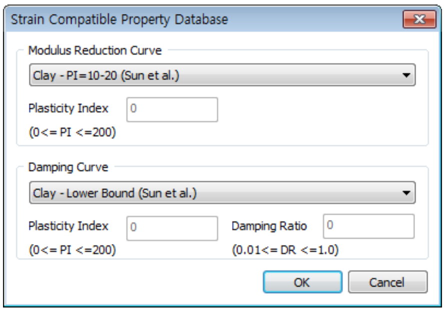






Database

A strain function stored from existing research can be imported from the database. An example DB is shown below.

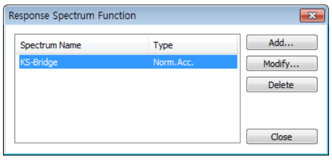


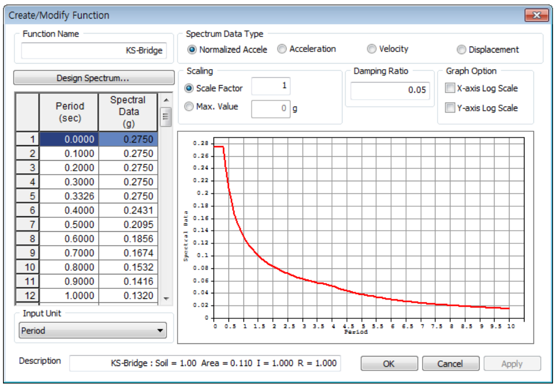
Modulus Reduction Curve	Clay - PI=5-10 (Sun et al.0)	**Damping Curve**	Clay - Lower Bound (Sun et al.0)
	Clay - PI=10-20 (Sun et al.0)		Clay - Average (Sun et al.0)
	Clay - PI=20-40 (Sun et al.0)		Clay - Upper Bound (Sun et al.0)
	Clay - PI=40-80 (Sun et al.0)		Clay (Idriss 1990)
	Clay - PI=80+ (Sun et al.0)		Gravel (Seed et al.0)
	Clay (Seed and Sun 1989)		Linear
	Gravel (Seed et al.0)		Rock
	Linear		Rock (Idriss)
	Rock		Sand (Idriss 1990)
	Rock (Idriss)		Sand (Seed & Idriss) - Lower Bound
	Sand (Seed & Idriss) - Lower Bound		Sand (Seed & Idriss) - Average
	Sand (Seed & Idriss) - Average		Sand (Seed & Idriss) - Upper Bound
	Sand (Seed & Idriss) - Upper Bound		Vucetic - Dobry
	Sand (Seed and Idriss 1970)		-
	Vucetic - Dobry		

Response spectrum function

Define the spectrum function used in response spectrum analysis. Because the response spectrum analysis uses a linearly interpolated spectrum function value for the natural periodicity of the structure, compact spectrum values are needed for areas where the curvature of the spectrum curve changes rapidly. The period range of the spectrum function must contain all natural periods of the structure.

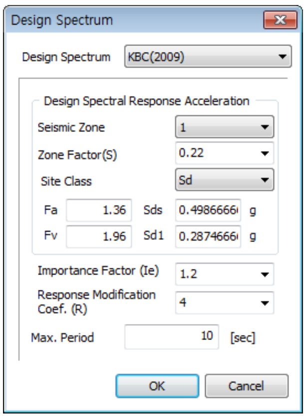
The spectrum data types are normalized acceleration (acceleration spectrum, gravitational acceleration), acceleration, velocity and displacement spectrum and changing the time function data format only changes the application format, not the data format. The scale factor is a gradient modulus for the entered data and the entire data can be scaled to fit the specified maximum value. In the 'Damping Ratio' space, input the damping ratio applied on the response spectrum but if the damping ratio of the target structure is different, the input spectrum data is processed to fit the structural damping ratio.





Design spectrum function

Use the built-in design spectrum. The default built-in design spectrum types are as follows.



► Korea(Bridge) : Korea, Traffic Specification

► Japan(Bridge02) : Japan, Load instruction and Dynamic Analysis of Building

► China(JTJ004-89) : China, Specifications of Earthquake Resistant Design for Highway Engineering

► KBC 2009 : Korea, Korea Building Code-Structural 2009

► KBC 2005 : Korea, Korea Building Code-Structural 2005

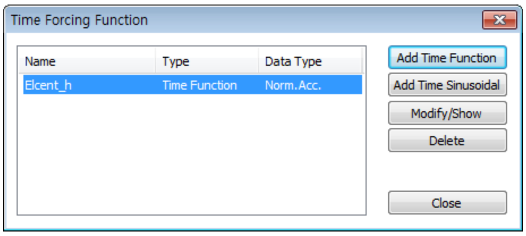
► IBC2000(ASCE7-98) : USA, International Building Code 2000

► UBC(1997) : USA, Measure of Uniform Building Code 97 (Standards)

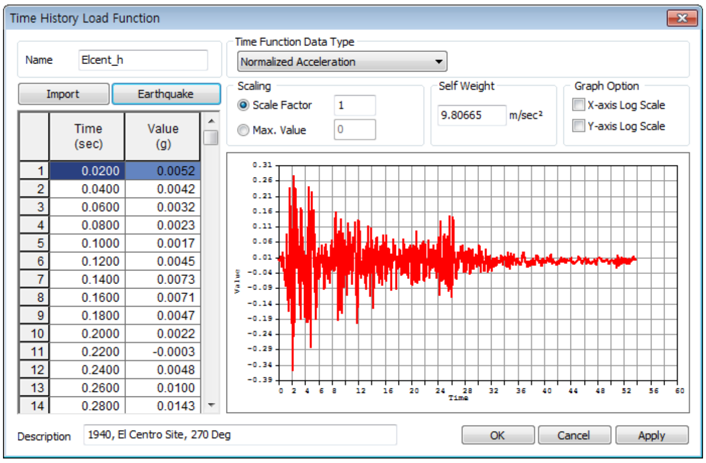
► EURO(2004H-ELASTIC) : Europe, Sesismic Design Measure of Structures

Time forcing function

The time forcing function is applied to the loading conditions (ground acceleration, time varying static, dynamic nodal, dynamic surface) applied to linear/nonlinear time history analysis. A function for the time history loading value is formed and changing the time function data format only changes the application format, not the data format.[edit] The scale factor is a gradient modulus for the entered data and the entire data can be scaled to fit the specified maximum value.



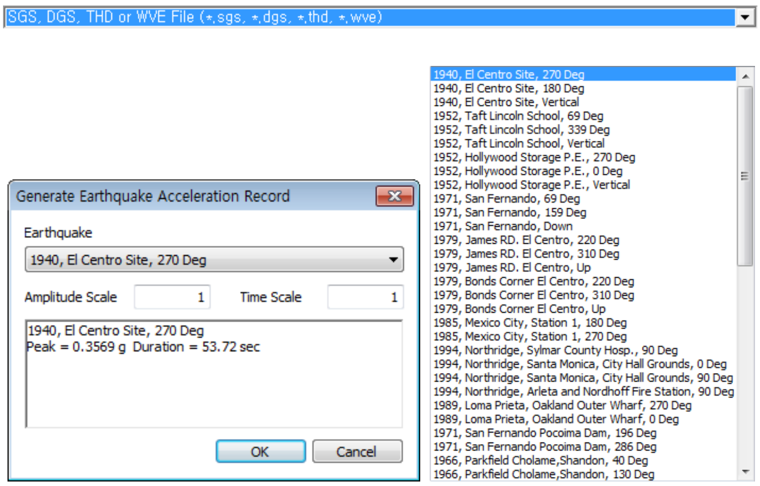
Add Time Function



The defined function is also applicable for Dynamic node (surface) loading and time varying static loading. Specifying 'Force' or 'Moment' uses the time history load as a 'dynamic node (surface) load' input and specifying 'Normalized Acceleration' or 'Acceleration' uses it to input the 'ground acceleration'. Specifying 'Normal' uses the time history load as the change in static load with time for 'time varying static load' or 'dynamic surface load'.

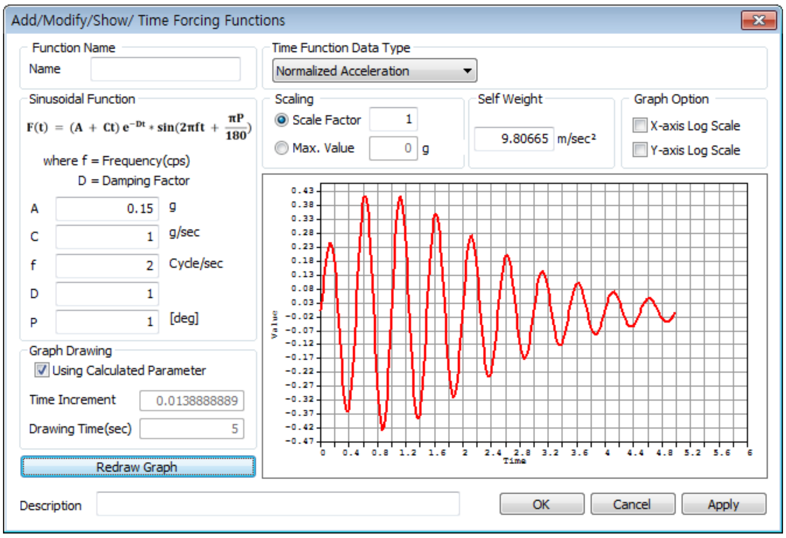
Import/Earthquake

Save and import frequently used time history loading or select earthquake acceleration data from the program DB. There are a total of 32 types of earthquake acceleration.



Add Time Sinusoidal

A sine function can be used to define the time history loading. A, C are constants, f is the frequency of the input load, D is the damping factor and P is the phase angle. If the time history load is entered as a harmonic function, input the necessary sine function variables and click [Redraw Graph] to view the loading on the right hand side.

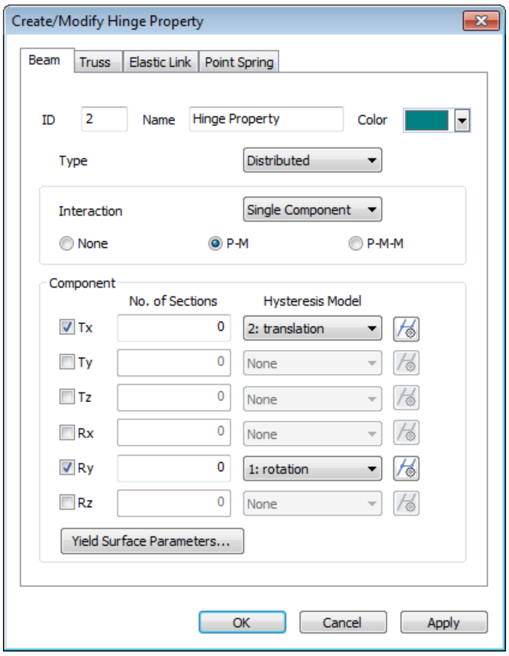


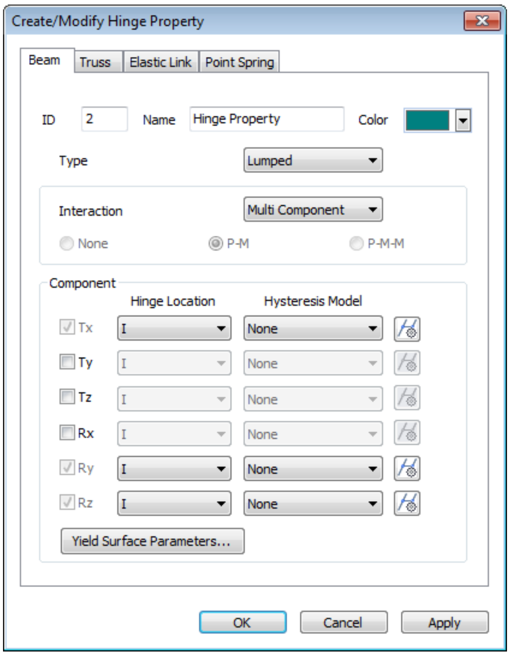
1.9 Hinge

Overview

When crack or yield occurs due to irregular cyclic load such as seismic load, very complex behavior appears since displacement history to the current affects to the restoring force and displacement relationship. It is called that hysteresis model which regulates this relationship and is considered to inelastic hinge at inelastic element.

Inelastic hinges are available in nonlinear, construction stage, consolidation, fully coupled, nonlinear time history analysis, SRM/SAM (Slope stability analysis) and nonlinear time history analysis with SRM and element results can be displayed.





Methodology

- **Type**

[Beam - Lumped] : It concentrates the inelastic behavior represented by rotational and translational springs at each end and the center. And the remaining parts are assumed to behave elastically. Inelastic hysteresis behaviors are defined by skeleton curves, which are empirical hysteresis models. The axial component is represented by a spring at the center and two translational components are represented by springs at each end defined by force-displacement relationships. The two flexural components, My and Mz, are represented by springs defined by the relationship between moment and angle of rotation at either I or J or at both ends.

[Beam – Distributed] : Unlike lumped hinges, it assumes inelastic behavior throughout the member. The plastic hinge locations in the length direction of a member assigned by the user are defined as the integration points. The flexibility matrix of a section, which represents the distribution of internal forces, is calculated through the integration points. The number of integration points can be 1 and between 3 and 20. If the number of integration points is 2, the moment at the free end of a cantilever beam does not come to exactly zero due to the inherited characteristic of the integration method. Therefore, two integration points are not permitted. Inelastic hysteresis behavior can be defined by 2 models, empirical Skeleton and Fiber. The hinge behaviors can be expressed by force - deformation relationships in each axis direction, and the hinge hysteresis behavior of the flexural components can be expressed by the relationships of moment and angle of rotation. Inelastic behaviors can be defined for 3 axis components and 2 flexural (My & Mz) components.

[Truss] : The axial component is represented by a spring at the center defined by the force-displacement relationship. The inelastic hysteresis behavior of a spring is defined by a skeleton model.

[Spring / Elastic Link] : Unlike Lumped and Distributed hinges, which are influenced by the inelastic properties of materials and members, the inelastic plastic hinge properties for the corresponding linear properties of each component of Property Type defined in General Link Properties are defined. The elastic stiffness of each component is defined by effective stiffness and acts as the initial stiffness in inelastic analysis. The inelastic hysteresis behavior of a spring is defined by a skeleton model. The inelastic properties of a spring can be defined for all 3 translational and 3 rotational directions.

- **Interaction**

The type of considering interaction between axial force and moment is selected.

[None] : Interaction between axial force and moments is not considered.

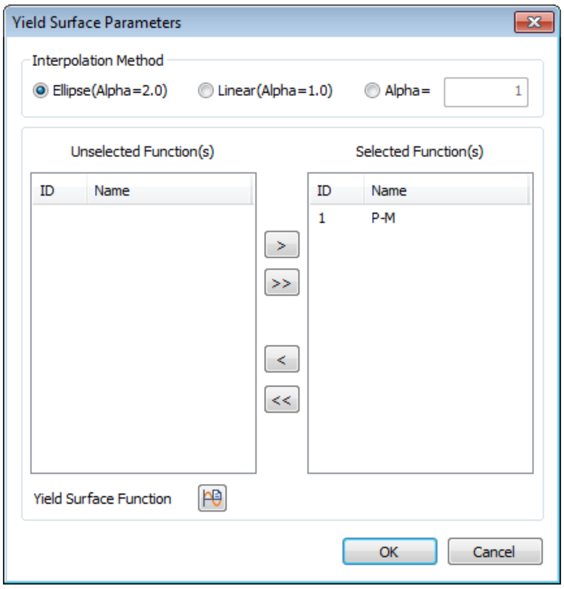
[P-M in Strength Calculation] : N-M interaction in time history analysis is reflected by calculating the flexural yield strength of a hinge considering the effect of axial force. In this method, the interaction between biaxial bending moments is ignored. The axial force is assumed to act with each directional bending moment independently when the hinge status is evaluated at each time step. Recalculation of bending moment yield strength reflecting axial force is carried out in a loading condition, which satisfies the following three conditions:

1) It must be the first among the sequential time history load cases, which will be consecutively analyzed.
2) Inelastic static analysis must be carried out.

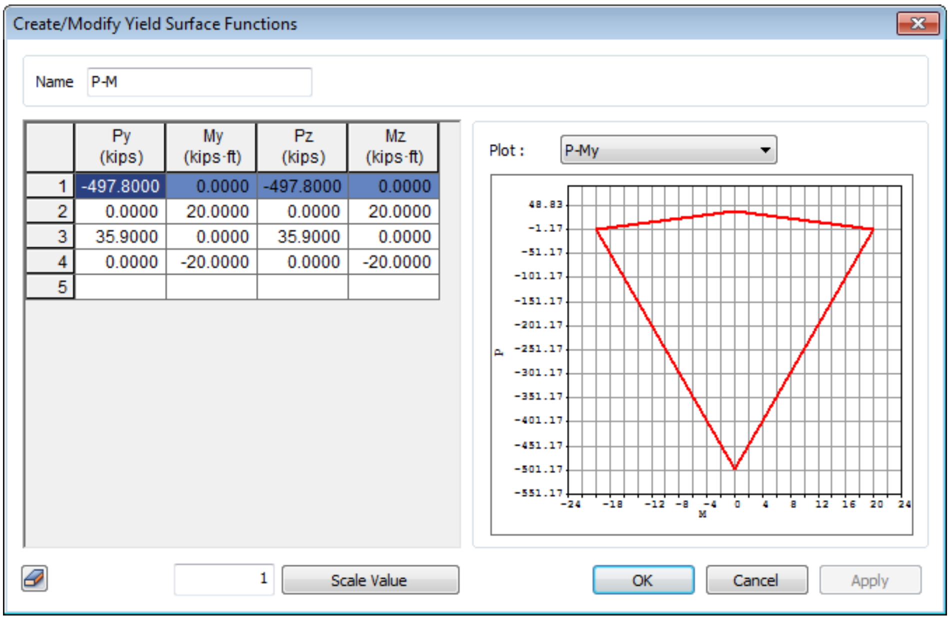
The elements are inelastic beam elements assigned with hinge properties to which P-M interaction is applied. The initial axial and bending moment at this time are determined by the combination of linear elastic analysis results of all the static loads contained in Time Varying Static Load. The factors used in the combination are defined by the Scale Factors specified in Time Varying Static Load.

[P-M-M in Status Determination] : This method uses a multi-axis hinge hysteresis model in inelastic time history analysis. Interaction between axial force and biaxial moments is realized by applying the plasticity theory. The interaction is considered at each time step through evaluating the status of inelastic hinges using the 3-dimensional yield surface. FEA NX supports the kinematic hardening type.

▶Yield Surface Parameter



▶Yield Surface Function



- **Interaction**

If "P-M in Strength Calculation" or "P-M-M in Status Determination" is selected in Interaction Type, enter the related data for P-M interaction curve and 3-D yield surface.

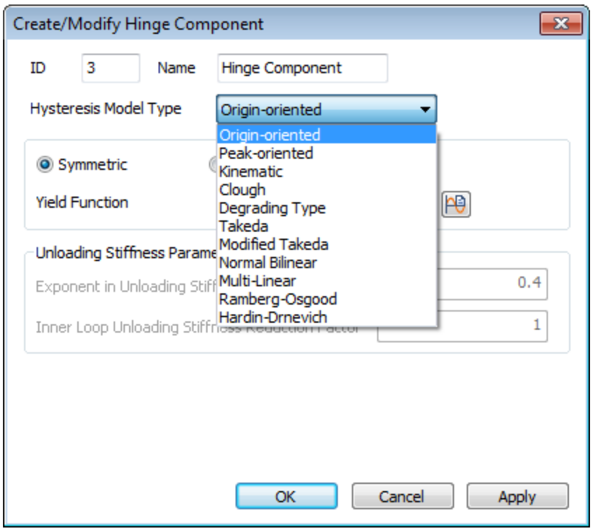
[Component]
Select the components of sectional strength for which properties will be entered. The Spring Type permits properties in all directional components, whereas Lumped and Distributed types permit all but the Mx component.

[Hinge Location]
Select the locations of lumped inelastic hinges. Axial component is fixed to the center of a member. I-end, j-end or both ends can be selected for the bending moment components.

[Num. of Sections]
Enter the number of integration points for inelastic hinges of the distributed type. Up to 20 sections are permitted, and moment - curvature relationships are calculated at all the sections corresponding to the points.

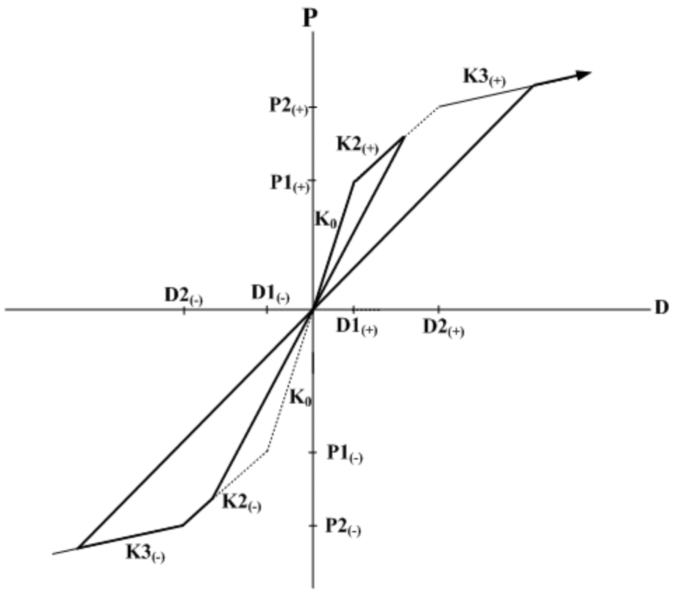
[Hysteresis Model]
Select a hysteresis model for an inelastic hinge.

Hinge Components (Single)

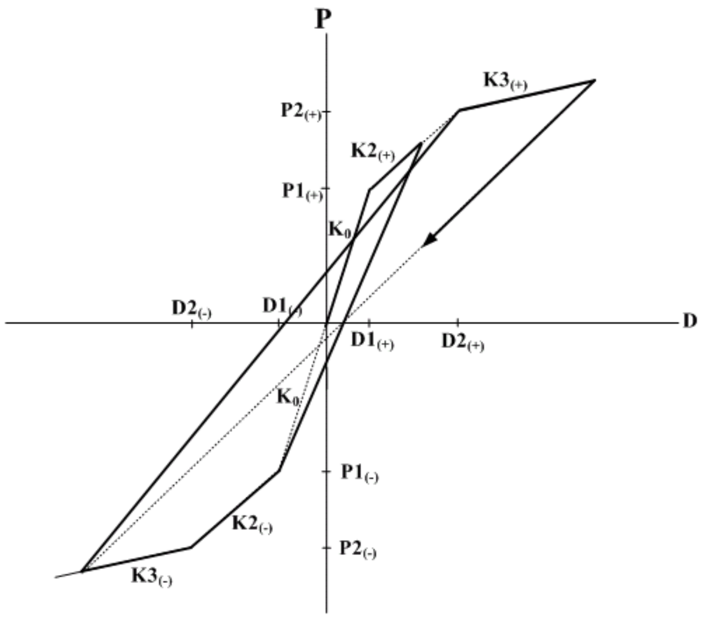


- **Hysteresis Model Type**

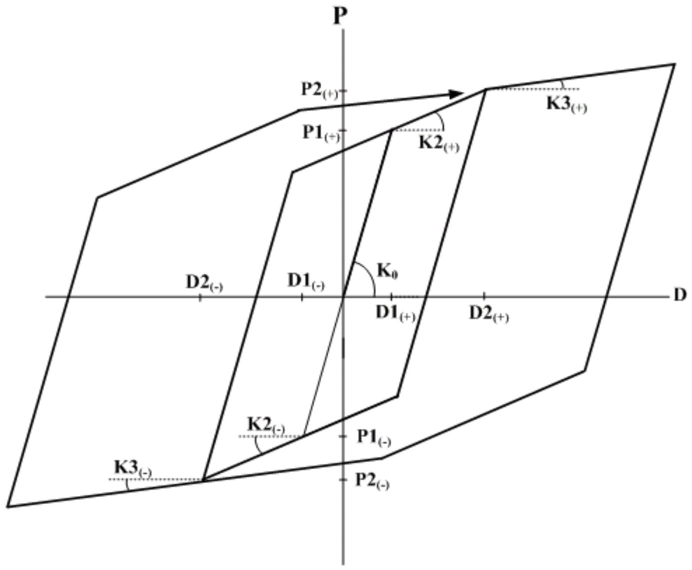
[Origin-oriented] : Response points at the initial loading move along a trilinear skeleton curve. Response points at unloading move toward the origin and again move along the skeleton curve after reaching the opposite skeleton curve.



[Peck-oriented] : Response points at the initial loading move along a trilinear skeleton curve. Response points at unloading move toward the point of maximum displacement on the opposite side. If the first yielding has not occurred on the opposite side, the response points move toward the first yielding point on the skeleton curve.



[Kinematic] : Response points at the initial loading move along a trilinear skeleton curve. The unloading stiffness is identical to the elastic stiffness. It shows the tendency of strength increase with the increase in loading. This is used to model the Bauschinger effect of metallic materials. Accordingly, it is cautioned that energy dissipation may be overestimated for concrete. Due to the characteristic of the model, only the positive (+) and negative (-) symmetry is permitted for the strength reduction ratios after yielding.



[Clough] : Response points at the initial loading move along a bilinear skeleton curve. Unloading stiffness is obtained from the elastic stiffness reduced by the equation below. As the deformation progresses after yielding, unloading stiffness reduces gradually.

$$K_r^+ = K_o \left(\left| \frac{D_1^+}{D_{\max}} \right| \right)^\beta \le K_o$$

$$K_r^- = K_o \left(\left| \frac{D_1^-}{D_{\min}} \right| \right)^\beta \le K_o$$

K_o : Initial elastic stiffness

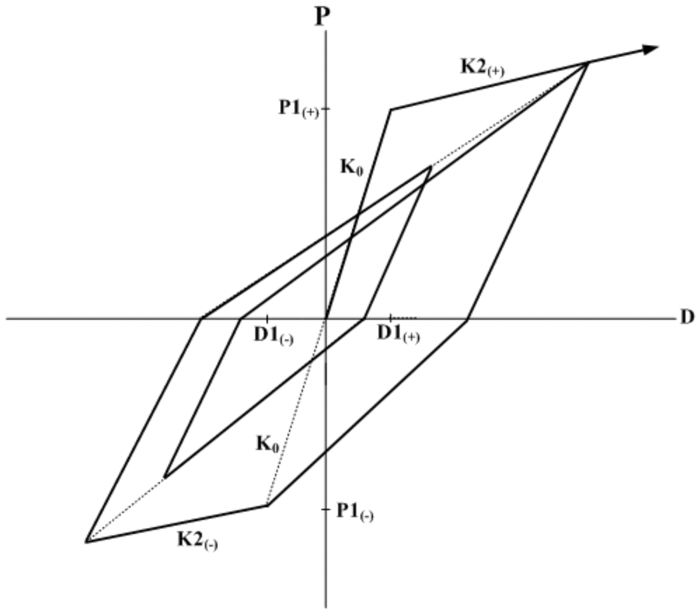
$D_1^\pm$: Yield displacement in the region of the first unloading

$D_{\max}$: Maximum displacement in the region of tension

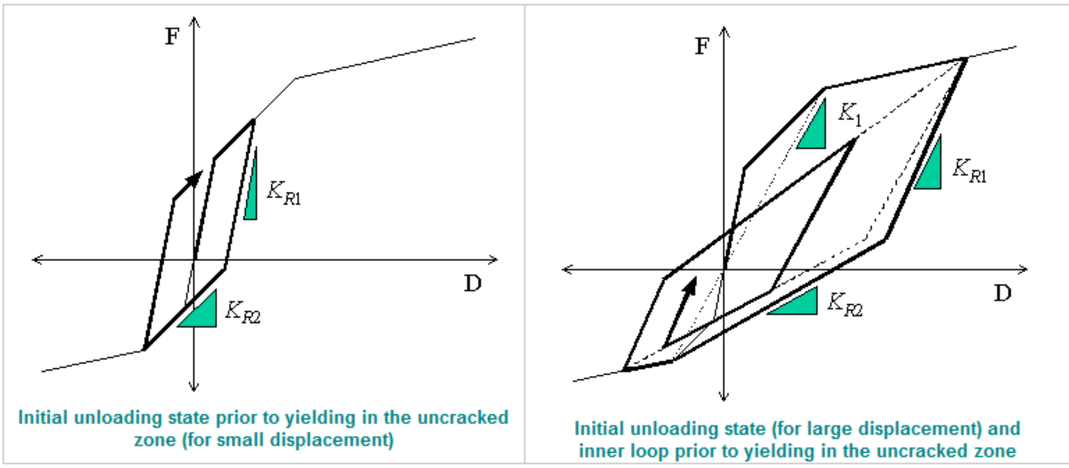
$D_{\min}$: Maximum displacement in the region of compression

β : Constant for determining unloading stiffness

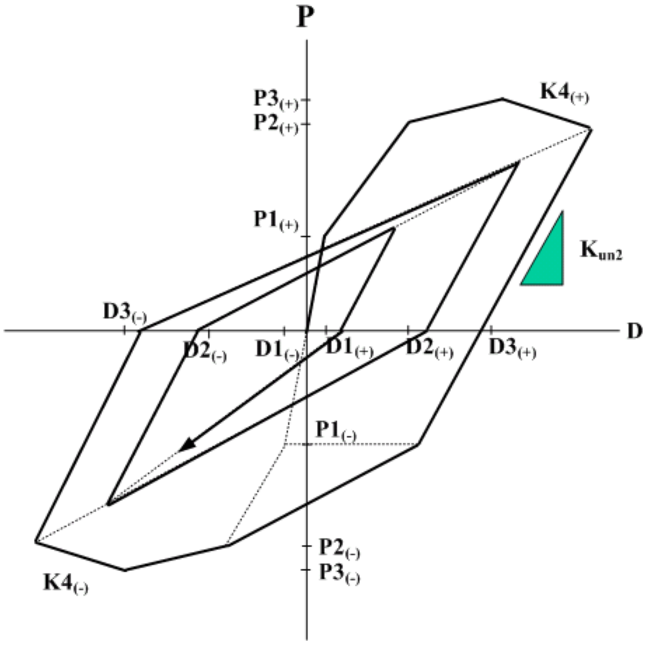
If the sign of loading changes in the process of unloading, response points move toward the point of maximum displacement in the zone of progressing direction. If yielding has not occurred in this zone, the response points move toward the yield point on the skeleton curve. If unloading becomes loading without changing the loading sign, the response points move along the unloading path. If the loading continually increases, loading continues on the skeleton curve again.



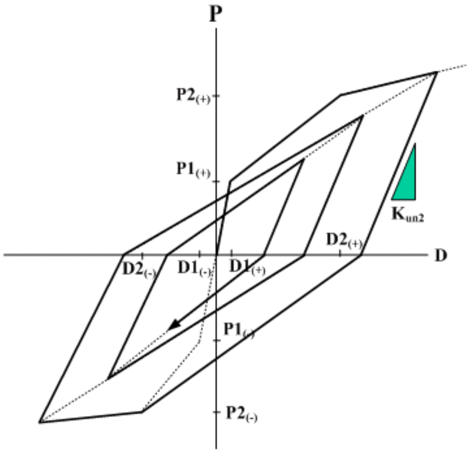
[Degrading] : Response points at the initial loading move along a trilinear skeleton curve. The load-displacement coordinates at unloading move to the path of reaching the maximum deformation point on the opposite side due to the change of unloading stiffness once in the middle. If yielding has not occurred on the opposite side, the first yielding point is assumed to be the maximum deformation point.

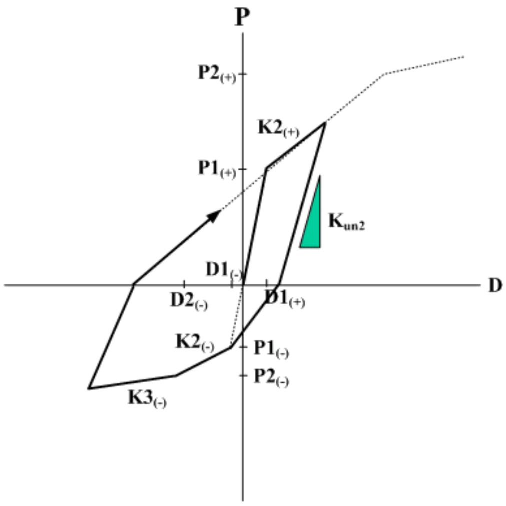


[Takeda] : Response points at the initial loading move along a tetralinear skeleton curve. If the current displacement or deformation, D, does not exceed D3, the hysteresis rules are identical to the Original Taketa hysteresis. If the current displacement or deformation, D, exceeds D3, response points move along the slope K4. For unloading, response points move by the same rules as the Original Taketa hysteresis. The Takeda tetralinear hysteresis model can be applied to beam element and General Link of Spring Type of Lumped Type and Distributed Type.

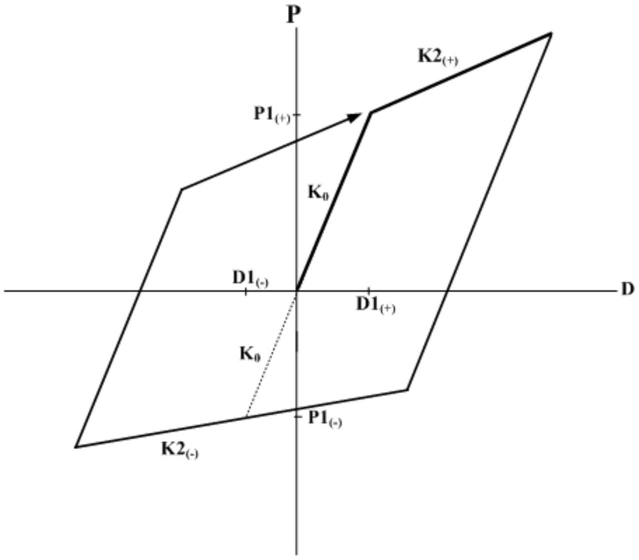


[Modified Takeda] : Response points at the initial loading move along a trilinear skeleton curve. If the current displacement or deformation, D, exceeds D2 for the first time or the maximum deformation point up until now, response points move along the trilinear skeleton curve. If unloading takes place from this straight line toward the opposite direction, the points move along the slope K_{un2} until the point of the restoring force becoming 0. If the restoring force goes beyond the 0 point, the points move toward the maximum deformation point on the opposite side. Even in the case where unloading takes place from the straight line directed toward the maximum deformation point from the point of the 0 restoring force, the points move along the slope K_{un2} until the points reach the 0 restoring force. After the point of 0 restoring force is passed, the points move toward the maximum deformation point on the opposite side. The Modified Takeda type hysteresis model can be applied to beam element and General Link of Spring Type of Lumped Type and Distributed Type.

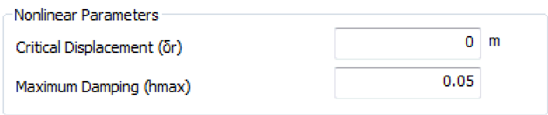


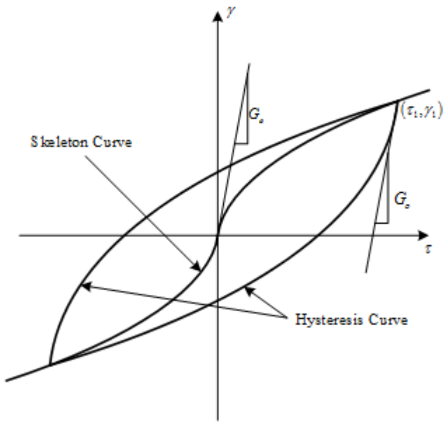


[Normal Bilinear] : Response points at the initial loading move along a bilinear skeleton curve. The unloading stiffness is identical to the elastic stiffness. The Normal Bilinear type hysteresis model can be applied to beam element and General Link of Spring Type of Lumped Type and Distributed Type.

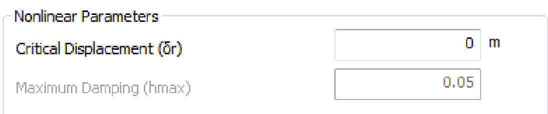


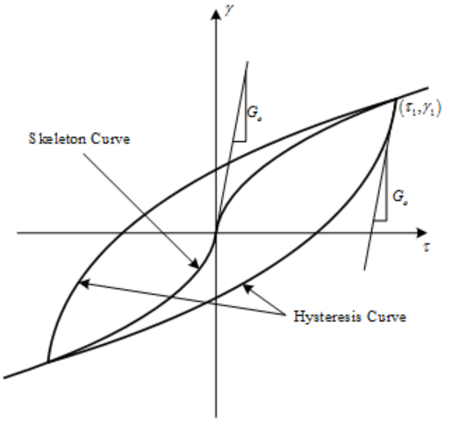
[Modified Ramberg-Osgood]

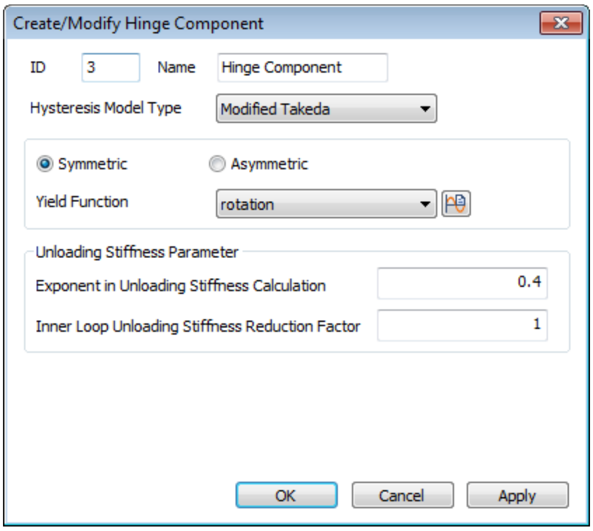




[Modified Hardin-Drnevich]



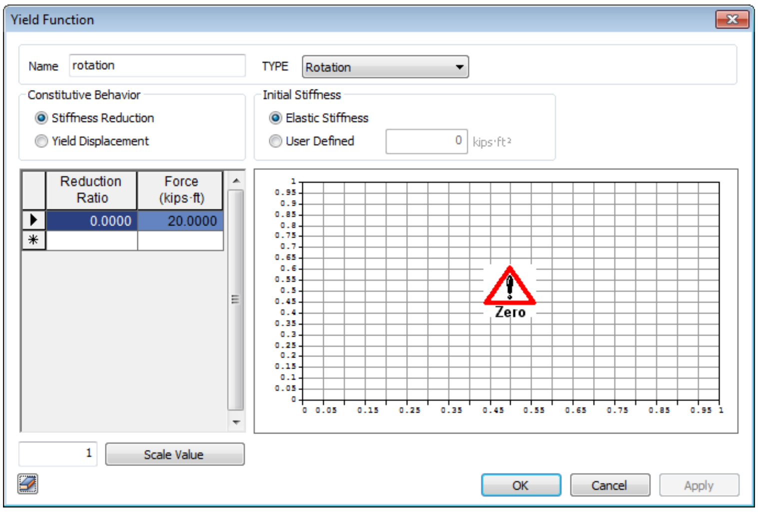




- **Symmetric / Asymmetric**

Select the type of Skeleton Curve.

- **Yield Function**



[Stiffness Reduction Ratio] : Enter the stiffness reduction ratios of a sloped skeleton curve when Strength - Stiffness Reduction Ratio is selected for Input Type.

[Yield Displacement] : Enter the yield displacement of a sloped skeleton curve when Strength - Yield Displacement is selected for Input Type.

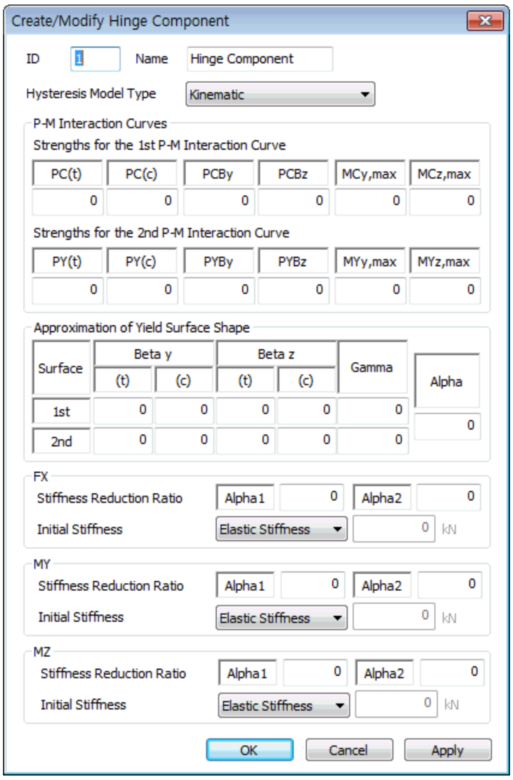
[Force (Yield Strength)] : Yield strength is specified. It is user defined based on material and section properties. The user specifies positive (+) values regardless of tension (t) or compression (c). The program treats compression as negative (-) internally.

- **Unloading Stiffness Parameter**

[Exponent in Unloading Stiffness Calculation]
This is an option used to determine the unloading stiffness of the outer loop used in the Clough and Takeda type models among hysteresis models of skeleton curves. This is used to reflect the effect of reduction in stiffness, which occurs as the deformation progresses after yielding. The unloading stiffness is determined by the elastic stiffness reduced by the yield displacement and maximum displacement in the zone where unloading begins and the exponent entered here.

[Inner Loop Unloading Stiffness Reduction Factor]
This is used to determine the unloading stiffness of the inner loop. The inner loop is formed when unloading occurs before reaching the target point on the skeleton curve while reloading after the loading sign changes in the process of unloading. The unloading stiffness of the inner loop is calculated by multiplying the unloading stiffness of the outer loop by the reduction ratio for the unloading stiffness of the inner loop.

Hinge Components (Multi)



[P-M Interaction Curves]

Enter the P-M interaction curve data required to calculate 3-dimensional yield surfaces. All strength values must be entered with positive sign. Sign convention for plotting P-M curve is positive for compression and negative for tension.

Strengths for the 1st P-M Interaction Curves

PC(t): First yield strength subject to pure tension force

PC(c): First yield strength subject to pure compression force

PCBy: Axial force at the time of balanced failure in the first yield interaction curve for the y-axis moment of the section

PCBz: Axial force at the time of balanced failure in the first yield interaction curve for the z-axis moment of the section

MCy,max: Maximum bending yield strength in the first yield interaction curve for the y-axis moment of the section

MCz,max: Maximum bending yield strength in the first yield interaction curve for the z-axis moment of the section

Strengths for the 2nd P-M Interaction Curves

PY(t): Second yield strength subject to pure tension force

PY(c): Second yield strength subject to pure compression force

PYBy: Axial force at the time of balanced failure in the second stage yield interaction curve for the y-axis moment of the section

PYBz: Axial force at the time of balanced failure in the second yield interaction curve for the z-axis moment of the section

MYy,max: Maximum bending yield strength in the second yield interaction curve for the y-axis moment of the section

MYz,max: Maximum bending yield strength in the second yield interaction curve for the z-axis moment of the section

[Approximation of Yield Surface Shape]

On the basis of P-M interaction curve, the parameters for 3-dimensional yield surface are either user defined or auto-calculated. If some items are auto-calculated and the remainder is to be user defined, Auto-calculation should be performed first, and then necessary items can be modified after converting to User Input. In case of Alpha, only user defined entry is possible. The value of each parameter is used in the equation of yield surface displayed in the dialog box.

Beta y, Beta z, Gamma: Being the exponential powers of P-My or P-Mz interactions, different values can be entered for the first and second yields. For Beta y and Beta z on the other hand, two separate values representing the ranges of larger and smaller axial forces relative to the axial force at the time of balanced failure can be entered.

Alpha: Exponent for My-Mz interaction for the 1st and 2nd yielding

[Stiffness Reduction Ratio]

Enter the stiffness reduction ratios of a sloped skeleton curve when Strength - Stiffness Reduction Ratio is selected for Input Type.

α1: Ratio of stiffness immediately after the first yielding divided by the initial stiffness

α2: Ratio of stiffness immediately after the second yielding divided by the initial stiffness

[Initial Stiffness]

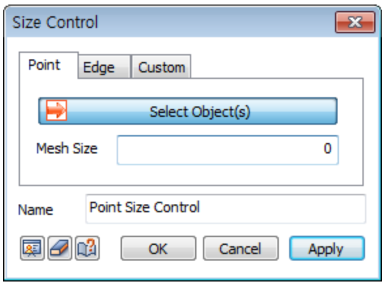
The initial stiffness used in inelastic analysis is either selected or entered by the user.

Elastic Stiffness: elastic stiffness of a member is used as the initial stiffness for inelastic analysis.
User Defined: the user directly enters the initial stiffness if the Input Type is Strength - Stiffness Reduction Ratio.

Section 2 Control

2.1 Size Control

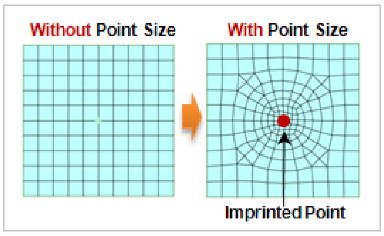
Point



Overview

Specify the size of elements surrounding the selected point. It is only applied to points imprinted on the surface or points that exist as interior objects. Points that exist on sub-shapes, such as the corners of a surface, are automatically ignored during the mesh creation process.

Methodology

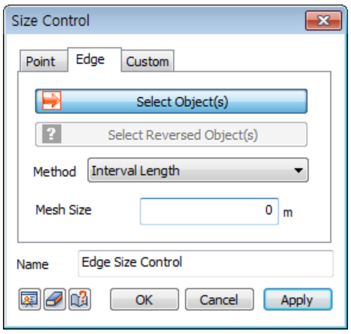


Specify the point and input the mesh size with respect to the current length unit to generate a mesh around the selected point.

Tip

'Seeding' is defined as the 'position where the node will be generated'.
If a seed is assigned to the shape and the shape is then modified, all seeding information is erased. Hence, seeding assignment is best d1 just before the mesh creation, after the geometrical shape has been finished. Seed information is the value that is applied first during mesh creation and the value is continuously applied until it is erased or removed.
The generated seed information can be registered on the Control mesh column of the Model tree by the specified name. The seed information can be erased/modified using the Context Menu.

Edge seed



Overview

Select edges (lines) and use the following 5 methods to preset the position (mesh size) of the new mesh node. The node spacing can be directly entered or a selected line can be divided into constant or linearly changing segments.

Methodology

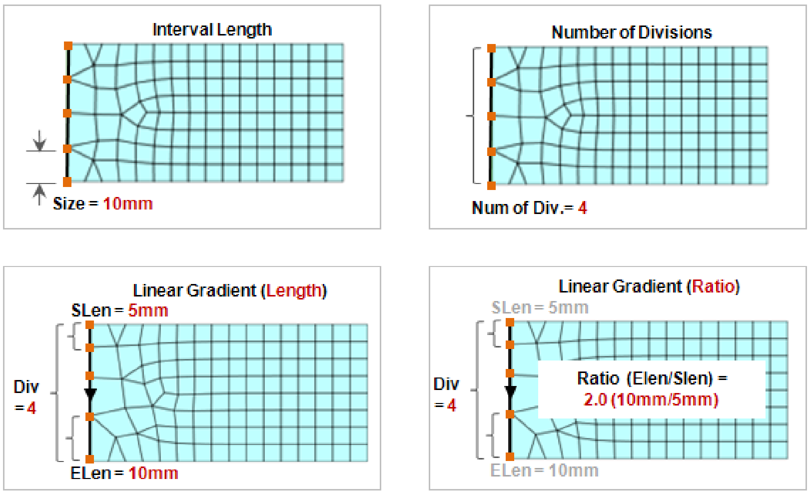
[Interval Length] : Input the node spacing in the current length unit.
[Number of Divisions] : Divides the selected line into multiple divisions, specified by the input number.
[Linear Grading (Length)] : Input the spacing between the start and end points of a line to automatically set the node positions through linear interpolation.
[Linear Grading (Ratio)] : Input the spacing ratio (end/start) between the start and end points of a line.

▸Interval Length

▸▸Number of Divisions

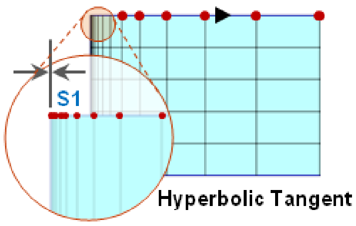
▸▸▸Linear Grading (Length)

▸▸▸▸Linear Grading (Ratio)



[Hyperbolic Tangent] : Input the start length and number of divisions to specify the nodes positions considering the total length of the line and number of divisions.

▸Hyperbolic Tangent



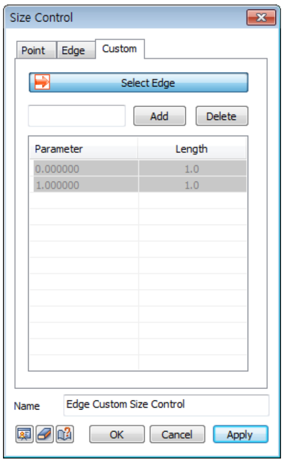
Tip

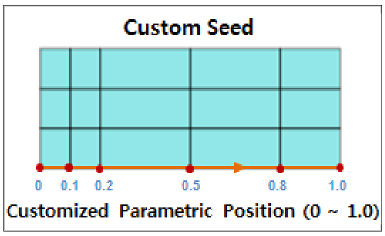
The number of divisions and certain intervals to the Linear Gradient (Length/Ratio) method. Entering the number of divisions makes it a priority. The certain interval defines the section of the selected line where linear change will take place. Input 0 for the start point and 1 for the end point. For an input of 0.3, the line will be divided by the specified constant value until 0.3 of the length from the start point. From that point, linear change takes place. Entering a negative value applies the certain interval value from the end point. For an input of -0.3, the line will be divided by the specified constant value until 0.3 of the length from the end point. The rest of the line will undergo linear change. Check the symmetric seed option to generate seeds that are symmetric about the center of the line.

Custom seed

Specify the node positions on the selected edge by entering the coordinates in the table directly. Entering a ratio between 0~1 automatically calculates the node position (length) of the selected line.

▸Custom seed

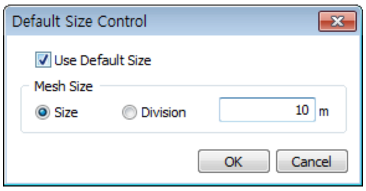




2.2 Default Size Control

Overview

The mesh size and number of divisions used in the mesh creation process can be preset. The default mesh size is not forced upon and is the value entered in the Generate mesh dialog box. Hence, the user can input a different value to generate a different size mesh during the mesh creation process.



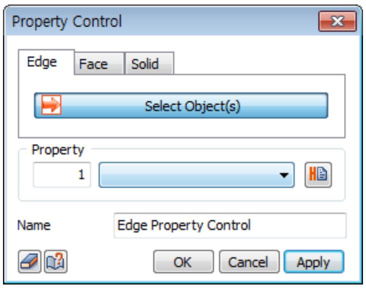
Methodology

Check the 'Use Default Size' to apply the default setting to all Generate mesh dialog boxes. The current length unit can be set directly or can be defined by the number of divisions of the selected shape.

2.3 Property Control

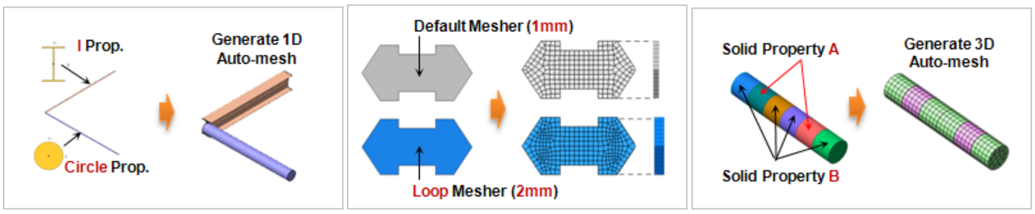
Overview

Pre-assign the properties (Edge, Face, Solid) of the geometry shape.



Methodology

▸Edge (Property Control)
▸▸Face (Property Control)
▸▸▸Solid (Property Control)

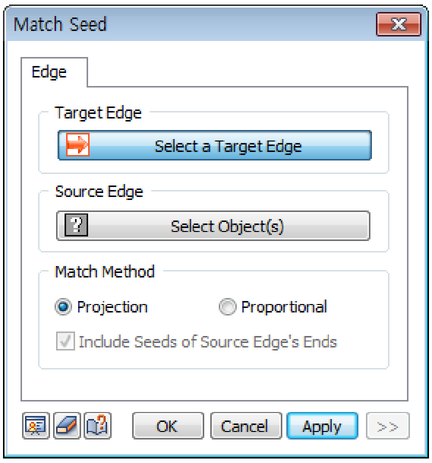


The element properties can be assigned simultaneously as the mesh creation. However, this function can be used to pre-assign the properties of the geometry shape and to automatically generate elements using the assigned properties. Pre-assigning a property give it priority over other property assignments during mesh creation.

2.4 Match Seed

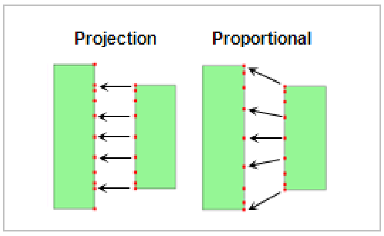
Overview

Specify the seed by transferring the seed information of a seeded edge to another edge. It is generally used to make the node positions of 2 separated edges similar to generate a homogeneous mesh or to ease the share node operation at a point with a very small gap.



Methodology

▸Match Seed (Projection/Proportional)



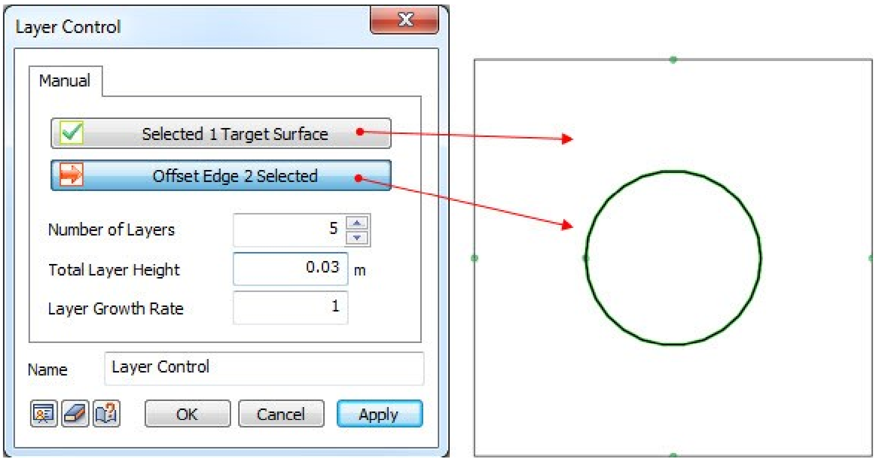
Select the target edge and the already seeded source edge to transfer the seed information from the source edge to the target edge.

[Projection] The source edge seeds are projected on to the target edge in the minimum distance direction.
[Proportional] Set the ratio such that the target edge has the same node division ratio as the source edge.

2.5 Layer Control

Overview

This tool creates several layers of mesh around holes (circular closed shapes) to obtain more accurate stress concentration result



Methodology

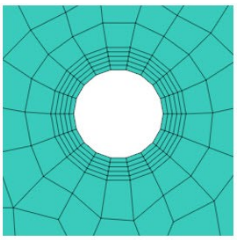
Select the target surface where circular shape is present and edges from circular shape (closed edges domain).

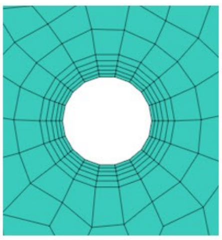
Number of Layers
Specify the number of layers to be offset (minimum value 1).

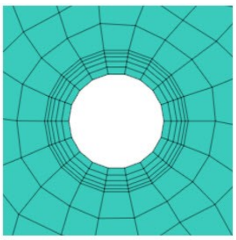
Total Layer Height
Specifies the height of the total number of boundary layers.

Layer Growth Rate
Proportionally adjusts the height value as the layer advances when the number of boundary layers is 2 or more.

Example:
When 1 is input, it is represented by the same height.
If it is larger than 1, it becomes larger.
If it is smaller than 1 a layer is created with increasingly smaller heights.


Layer Growth Rate 1


Layer Growth Rate 1.2

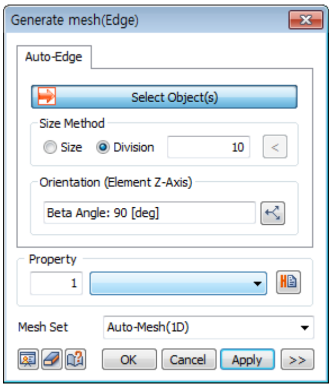

Layer Growth Rate 0.8

Section 3 Generate

3.1 1D

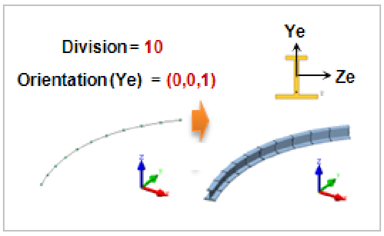
Overview

Generate a 1D structural mesh by selecting an edge. It is mostly used to generate independent structural elements that do not require node connections with neighboring ground elements.



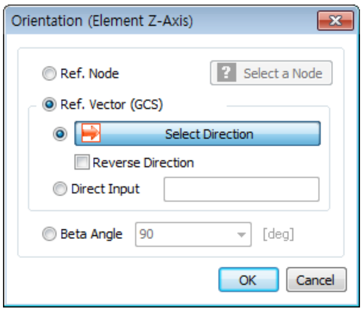
Methodology

▸Generate 1D mesh



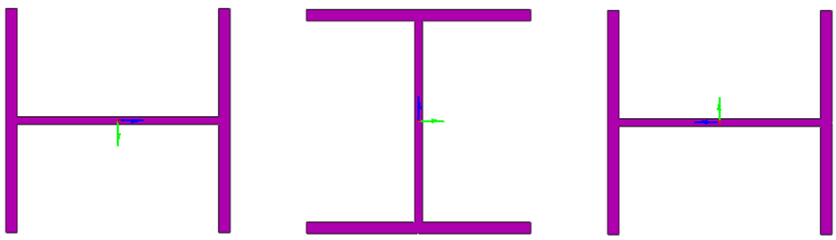
Embedded trusses or Pile elements are interface elements that do not require node connections with neighboring ground elements. These elements can set independent mesh sizes or number of divisions to generate meshes independently.
The assigned properties can be set or new properties can be added during Mesh creation.

[Orientation (Element Z-Axis)]
This function is used to unify the direction property of a 1D element to 1 direction or to set the major and minor axes directions. Adjust the Z axis direction by checking the element coordinate axis and assigning with reference to the Beta angle.



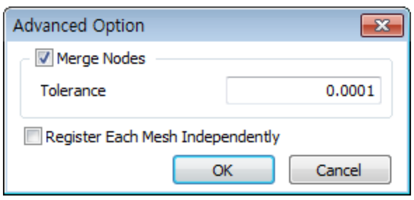
- Reference node: Select the reference node for the sectional direction of the 1D element. The element Z coordinate direction is set with reference to the selected node.
- Reference Vector (GCS): Set the Z coordinate direction of the selected element using the GCS direction or the input vector direction.
- Beta Angle: Angles 0, 90 and 180 can be chosen, and the selected Beta angle rotates the element by that angle with reference to the X axis.

▸Beta angle:0
▸▸Beta angle:90
▸▸▸Beta angle:180



Advanced Option (>>)

Small gaps within the tolerance can be automatically joined for mesh creation. When creating a mesh for 2 or more edges, individual independent mesh sets can be generated.

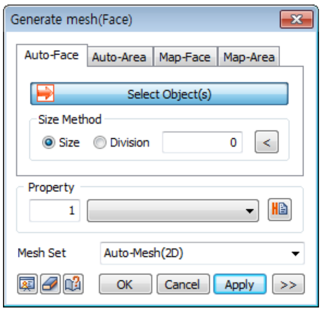


3.2 2D

Auto-Face

Overview

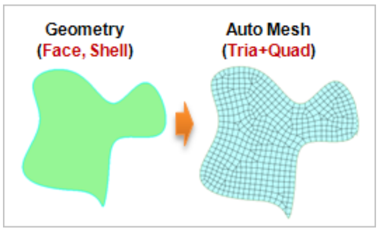
Generate a 2D mesh by selecting a surface. It is mostly used to generate a mesh in the ground or specified area of a 2D model.



Methodology

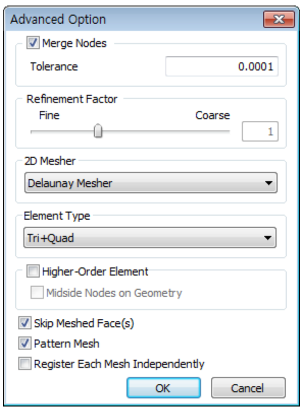
Define the mesh size directly or input the number of divisions for the surface forming edge to set the mesh size. Assigned properties can be specified during mesh creation, and the name of the mesh set can be pre-determined.

▸Auto-generate surface mesh



Advanced Option (>>)

Additional options are provided depending on the mesh generation method. The mesh shape, mesh density and generation algorithm can be set. The initial settings take into account the efficiency and accuracy depending on the geometric shape for the best mesh generation. The detailed settings are as follows.



[Merge Nodes]

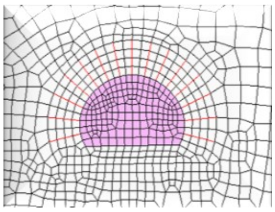
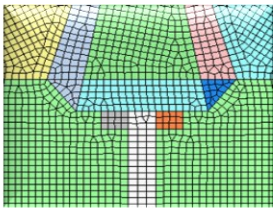
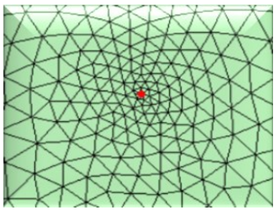
Merge 2 or more nodes within the tolerance during mesh generation. Nodes separated by tiny gaps are the main sources of error during analysis, and small gaps within the tolerance can be automatically joined for mesh creation.

[Refinement Factor]
Select the mesh size (mesh density) that will be created in the interior of the selected shape. A more compact mesh is created as the value approaches [Fine]. Fine meshes provide more detailed result analysis, but it is important to consider analysis time and efficiency when selecting the mesh density.

[2D Mesher]
Select the mesh generation algorithm for mesh generation. The user can select between three options; Loop Mesher, Grid Mesher and Delaunay Mesher. The generated shape and process change with the selected algorithm.

- Loop: Mesh generation method and shape based on Looping algorithm
- Grid: Complex mesh generation based on Modified grid algorithm
- Delaunay: Indirect mesh generation based on Delaunay triangulation algorithm

▸Loop
▸▸Grid
▸▸▸Delaunay

[Element type]
Generate a mesh with the selected shape. The user can select between a triangle, quadrilateral or a combination of the 2. Quadrilateral meshes provide a more stable analysis, but for complex geometric shapes where quadrilateral meshes are difficult to generate, it is better to generate a triangular mesh.

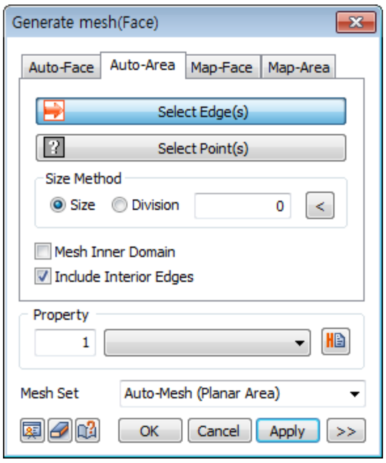
[Higher-Order Element]
Generate another node between mesh nodes to create a higher order mesh. Higher order adds a computation point and therefore, a more detailed analysis is possible, but the analysis time becomes longer. It is recommended that meshes be created with reference to the mesh shape and mesh density, and the higher order meshes be created only when necessary, depending on the analysis method. For example, generate a higher order mesh for the strength reduction method on a slope face where analysis of detailed deformation sections is necessary.

Check the [Skip Meshed Face(s)] to prevent overlapping generation of meshes on a surface with an existing mesh. The homogeneity option can be set to make the mesh sizes as uniform as possible. Also, when generating meshes on multiple surfaces at the same time, the mesh sets can be individualized by each surface or grouped into 1 mesh set.

Auto-Area

Overview

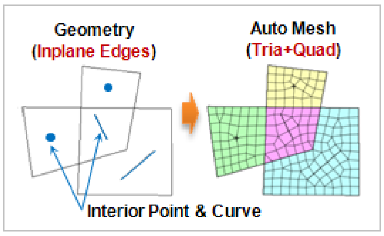
Select lines that form a closed area to generate a mesh in that domain. It does not create a surface for a complex 2D shape and is used to generate a mesh in all areas.



Methodology

Define the mesh size directly or input the number of divisions for the area forming edge to set the mesh size.
If the closed domain is within another closed domain, use the [Mesh Inner Domain] option to divide the domain and auto-generate. If an interior line (edge) or point is selected, the option identifies the position of the line or point and creates a node. In particular, the [Include Interior Edges] option can extract sub-shapes from an edge and is a vital option when generating a structural element.

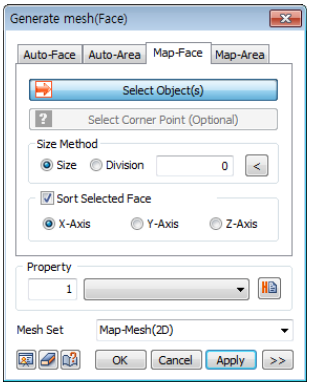
▸Auto-generate area element



Map-Face

Overview

Select a surface and auto-generate a quadrilateral mesh. The Mapped mesh maps the selected shape with square domains and generates a mesh on the mapped domains.

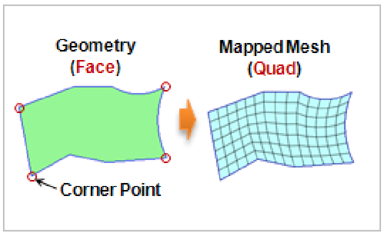


Methodology

Define the mesh size directly or input the number of divisions on the surface forming edge.
The target shape must have at least 4 outlines for the mesh to be generated and if the shape has too many outlines, the automatic mapping may not work. In this case, select 4 corners to manually map the edges. The 4 selected points become the corners of the square domain and lines between each point are mapped by 1 side of the square domain.

For mapped domains, mesh cannot be created if the opposing outlines (the group of outlines opposite each other after mapping) have different number of generated mesh elements. Hence, it is convenient to work after entering the seed information on the edge beforehand.

▸Create mapped mesh surface



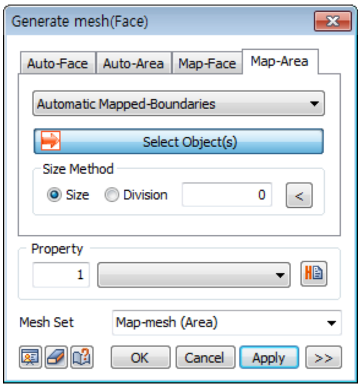
Tip

Generating a mesh by selecting multiple surfaces in random order may not generate a mesh because the number of divisions on the left and right are not the same. To avoid this, specify the mesh generation order as X axis, Y axis or Z axis and align the surfaces in order before generating a mesh.

Map-Area

Overview

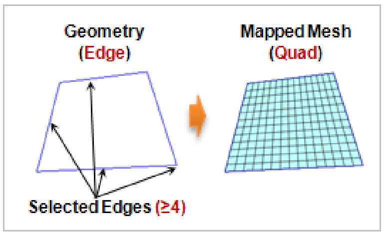
Select lines that form a closed domain to generate a quadrilateral mesh.

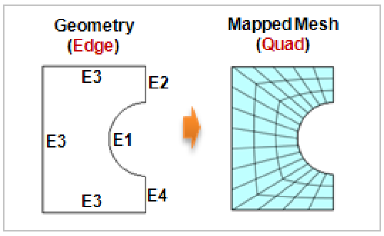


Methodology

Define the mesh size directly or input the number of divisions on the surface forming edges. Select [Automatic Mapped-Boundaries] or [Manual Mapped Boundaries] edge composition to select edges that form a closed domain.

‣Automatic Mapped-Boundaries
‣‣Manual Mapped Boundaries





[Automatic Mapped-Boundaries]
Generate a 2D mapped mesh by automatically selecting edges that form the desired domain. The mesh may not be created due to the model shape and algorithm properties.

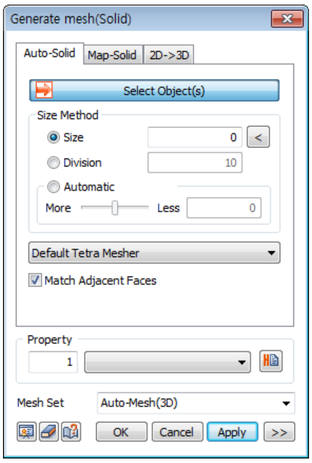
[Manual Mapped Boundaries]
When the 2D mapped mesh cannot be automatically generated, the edges are divided into 4 explicit groups and then selected. All edges within an edge group must be connected and the edge groups need to be specified in order, either in the clockwise or counter clockwise direction. Finally, the selected edges must form 1 closed domain.

3.3
3D

Auto-Solid

Overview

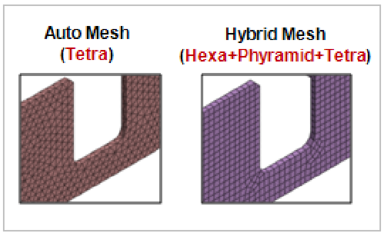
Automatically generate a mesh on a 3D solid shape. The mesh shape can be selected between a tetrahedral and a hexahedron centered hybrid (tetrahedral + hexahedron combination shape).



Methodology

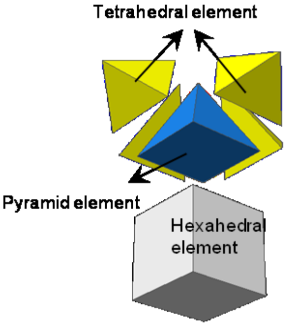
Define the mesh size directly or input the number of divisions on the solid forming edges to set the mesh size. Also, the mesh density can determine the mesh size by the “More” or “Less” option in [Automatic].

▸Auto-generate solid mesh



[Mesh generator]
The default tetrahedron element and hexahedral centered hybrid element are both supported. Like the 2D mesh, quadrilateral-based meshes provide more stable analysis than triangle-based meshes. Hybrid elements are formed by combining a pyramid and tetrahedron on the hexahedron base.

▸Hybrid mesh shape

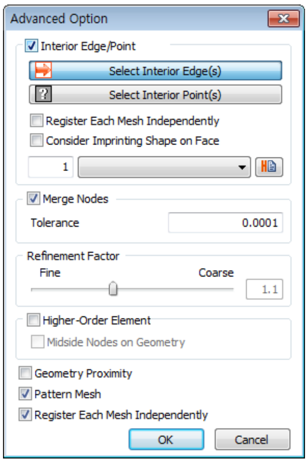


[Match Adjacent Faces]

The most important part in mesh generation is the node connection between adjacent elements. If the nodes are not connected, errors can occur during analysis and if the “Fit adjacent mesh” option is checked off, the nodes may not be connected even though an interface forms between the 2 solids. Unless the node is removed separately, always keep this option checked when generating a mesh.

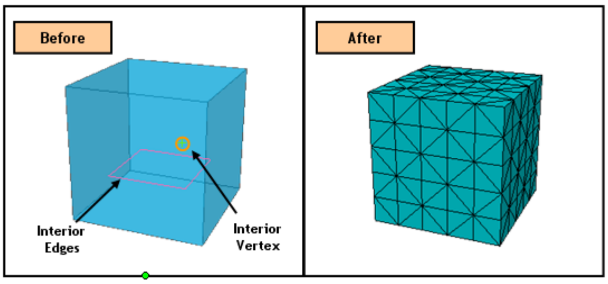
Advanced Option (>>)

The mesh shape and mesh density options can be set additionally depending on the mesh generation method. The detailed settings are as follows.



[Interior Edge/Point]
Generate a solid mesh considering the position and size of the interior edges inside a solid. If the whole edge is within the solid, just selecting the edge creates a solid mesh that considers the interior edge. However, if the edge is touching or penetrating the exterior boundary surface, the intersecting point needs to be imprinted on the surface of the solid and the mesh division point needs to be set to generate a solid mesh. Selecting an interior point creates a mesh node at the point when the mesh is created.
The additionally selected interior edges can be assigned a 1D structural property during mesh generation.

▸Include interior edge/point



[Merge Nodes]
Merge 2 or more nodes within the tolerance during mesh generation. Nodes separated by tiny gaps are main sources of error during analysis and small gaps within the tolerance can be automatically joined for mesh creation.

[Refinement Factor]
Set the interior mesh density in addition to the mesh size to create a better quality mesh.
[Higher-Order Element]

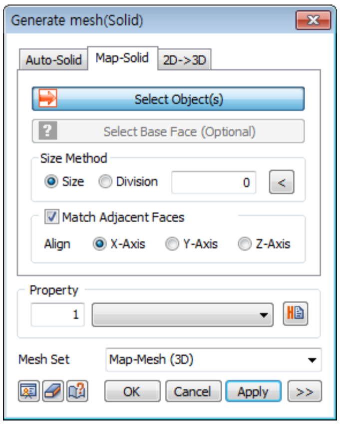
Generate another node between mesh nodes for a more detailed analysis, but the analysis time becomes longer per additional node.

Other options are available, such as [Geometry Proximity], which divides meshes that have a size difference during the mesh generation process, and {Pattern mesh}, which makes the mesh sizes as uniform as possible.

Map-Solid

Overview

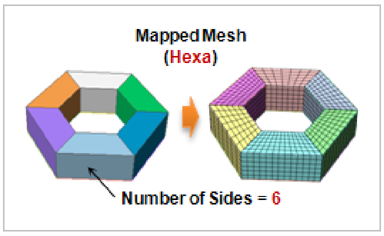
Automatically generate a hexahedral mesh on 3D solid.



Methodology

Define the mesh size directly or input the number of divisions on the corner edges to set the mesh size. Mapped meshes use only hexahedral shapes to generate the mesh and so, the seed information on the 2 opposing surfaces needs to be identical. Hence for complex shapes, the solid needed to be divided appropriately or the seed information must be input in advance.

▸Auto-generate hexahedral mesh



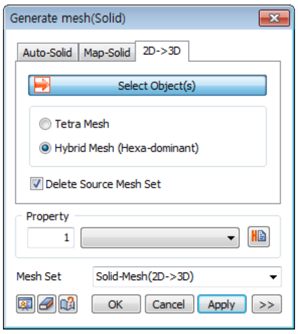
[Select Base Face]
Use the Sweep Based Mapped Mesh algorithm to generate a mesh when the 3D mapped mesh is not generated by the Full Mapping algorithm. However, the mapped mesh may not be generated if the shape is complex and the reference surface cannot be automatically found. In this case, directly select the reference surface for Sweep.

[Match Adjacent Mesh]
Generating a mesh by selecting multiple solids in random order may not generate a mesh because the number of divisions on the left and right are not the same. To avoid this, specify the mesh generation order as X axis, Y axis or Z axis and align the surfaces in order before generating a mesh.

2D -> 3D

Overview

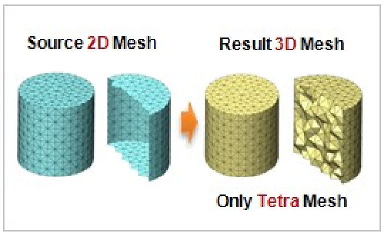
Generate a tetrahedral or hybrid (hexahedral centered) mesh using 2D mesh information for a 2D mesh enclosed domain with no solids.



Methodology

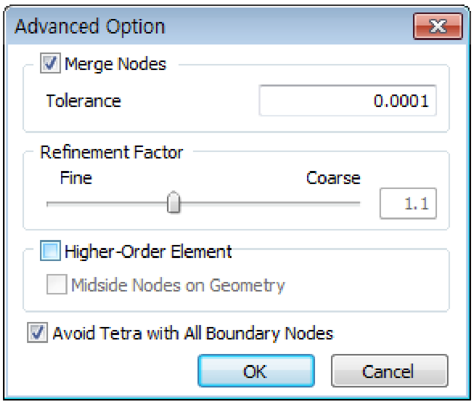
The 2D mesh must completely surround the domain where the 3D mesh will be created without any free edges. The original 3D mesh can be deleted or kept.

▸2D→3D Mesh generation



Advanced options (>>)

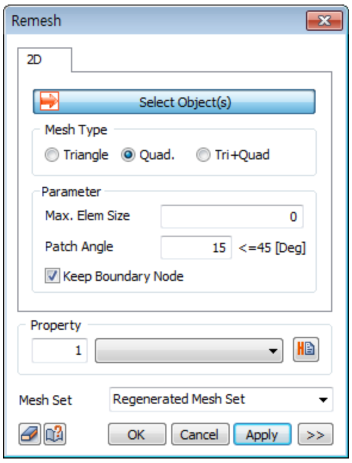
The following advanced options can be set when generating a 3D mesh from a 2D mesh. The nodes within the tolerance can be automatically merged to prevent analysis error and the interior mesh density of the solid mesh can be specified. Selecting the [Avoid Tetra with All Boundary Nodes] option prevents the 4 nodes of the tetrahedral element from all being on the boundary surface. This option puts at least 2 elements in the thickness direction of a thin solid and granting it a boundary condition can prevent the 4 nodes of the tetrahedral element from all being restricted.



3.4 Remesh

Overview

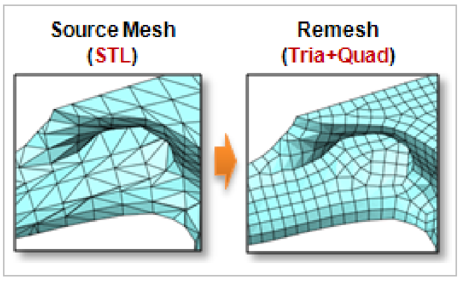
Regenerate a 2D mesh using already created 2D mesh information to define the surface. This option can be used to edit the mesh size and shape.



Methodology

When selecting the target for mesh generation, surfaces that do not have an existing 2D mesh are not selected. The mesh type can be selected from triangular, quadrilateral and triangular+quadrilateral combined elements.

▸Regenerate 2D mesh



[Maximum Element Size]
Specify the maximum Element size of the new mesh. However, if the target surface is seeded, the seed is also applied to the mesh regeneration process. Hence, to specify a new mesh size, the existing seeds need to be deleted before regeneration.

[Patch Angle]
Input the maximum angle between the selected existing element and the new generated element. The angle needs to be smaller than 45 degrees.

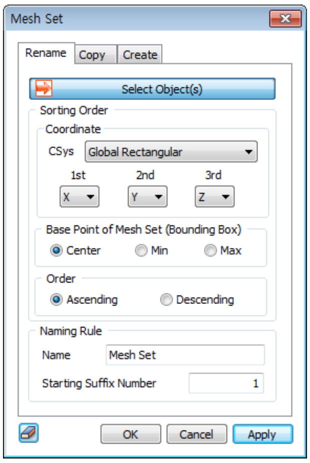
[Keep Boundary Nodes]
Decide whether to keep the boundary nodes. The boundary node positions may not be changed, even if the element shape or size is changed, to connect with other nodes of adjacent meshes.

Section 4 Mesh Set

4.1 Rename

Methodology

Change the name of the mesh set by adding numeric suffixes in the specified order. Because the data used for construction step is classified by the name of the mesh set, it is recommended to classify the mesh sets by domain or property in advance. It is especially useful to use the Wizard to automatically set the construction step.



Methodology

Select all the mesh sets that need name changes and set the classified order and name. For example, selecting 3 mesh sets and entering “Excavation” for the name and “3” for the Start number changes the name of the mesh sets in the classified order to “Excavation-003”, “Excavation-004”, “Excavation-005”.

[Coordinate]
Specify the mesh set order with respect to the Global Rectangular or Cylindrical coordinate system. Specify the order according to priority by specifying each coordinate axis direction in 1^{st}, 2^{nd}, or 3^{rd} priorities. The 1^{st} axis is given the most preference, and the 2^{nd} axis becomes the order reference when the coordinates of the 1^{st} axis are equal. Hence, if the target is assigned an order in the 1^{st} axis, the order does not change even if the coordinates in the 2nd axis is different. If the coordinates are the same in both the 1st and 2nd axes, the 3rd axis determines the order.

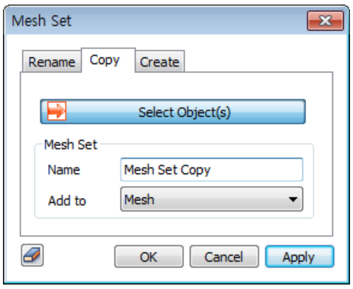
[Base Point of Mesh Set (Bounding Box)]
Specify the coordinates of the calculation position used for order comparison. For example, selecting the center compares the position of the mesh set using the coordinates of the center point of the mesh set boundary box.

[Order]
Increase or decrease the suffix number in order of specification.

4.2 Copy

Overview

Copy the generated mesh set. The copied mesh can be used in various ways. For example, the mesh can be generated without further geometrical modeling when comparing different materials or properties on each copied element to the original element; or when the same element is uniformly placed for each construction step.



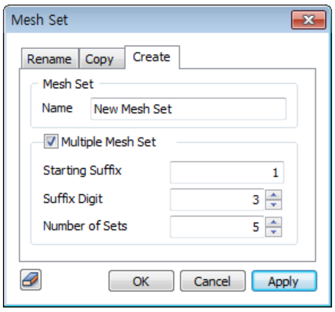
Methodology

Select the target mesh set and input the new position to copy all the nodes and elements of the original mesh set to the new position 1 by 1.

4.3 Create

Overview

Create multiple empty mesh sets according to the specified name rule. This function is useful for defining an element in 1 set into multiple sets.

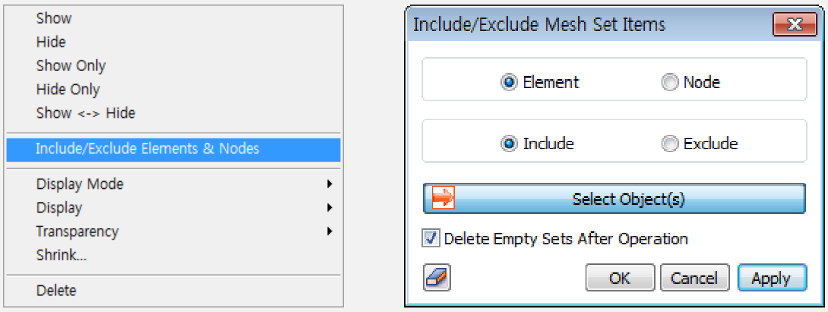


Methodology

Input the name of the newly generated mesh set. When creating multiple mesh sets, input the starting number and number of mesh sets.

Tip

Data used in analysis is classified by mesh sets. Hence, the mesh set needs to be subdivided depending on the analysis case. Generating an empty mesh set, clicking it with the right mouse button and bringing up the Context Menu allows the [Include/Exclude of Elements and Nodes] for each mesh set, as shown below.



<Include/Exclude Elements/Nodes>

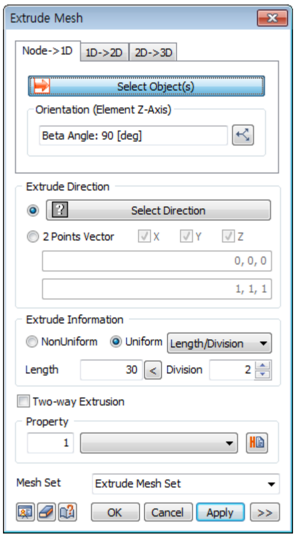
Section 5 Protrude

5.1 Extrude

Node -> 1D

Overview

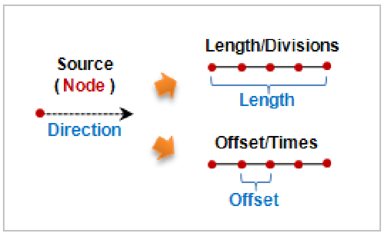
Generate a 1D element by extruding a node in the set direction. This function can be used to create a structural element without drawing a line.



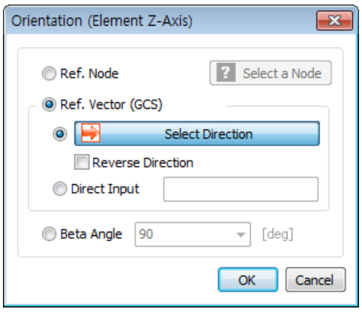
Methodology

Select the element node to extrude and specify the extrude direction and length. The element can be created by extruding in 1 direction or 2 directions.

▸Extrude node

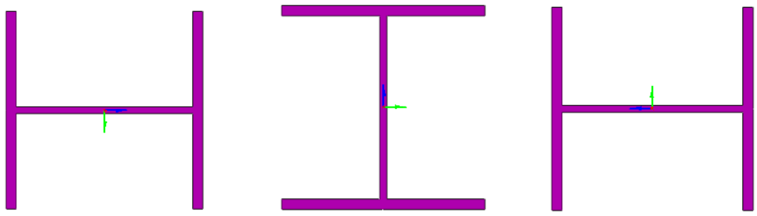


[Orientation (Element Z axis)]
This function is used to unify the direction property of a 1D element to 1 direction or to set the major and minor axis directions. Adjust the Z axis direction by checking the element coordinate axis and assign with reference to the Beta angle.



- Reference Node : Select the reference node for the sectional direction of the 1D element. The element Z coordinate direction is set with reference to the selected node.
- Reference Vector (GCS) : Set the Z coordinate direction of the selected element using the GCS direction or the input vector direction.
- Beta Angle : Angles 0,90,180 can be chosen and the selected Beta angle rotates the element by that angle with reference to the X axis.

▸Beta angle:0
▸▸Beta angle:90
▸▸▸Beta angle:180



Extrude Direction

Select the element extrude direction with reference to the GCS or input the start and end points of the direction vector using the 2-point vector function. It is useful when extruding in an arbitrary direction.

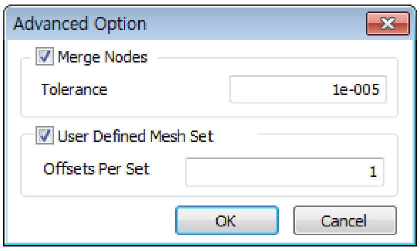
Extrude Information

Set the total length and division of 1D element which will be created. The division spacing can be set as either uniform or nonuniform. Entering a negative value for length (Offset,Times) extrudes in the opposite direction to the axis or vector direction.

[Nonuniform]
Specify the offset length and number simultaneously. The length can be listed using a comma (,) or as number@length for continuously repeating extrude operations.
For example, entering 10@3 creates 10 elements with a length of 3 each and entering 2,3,4 creates 3 elements, each with a length of 2,3 and 4.
[Uniform]
Set the offset length or number, or input the total length and division spacing.

Advanced Option (>>)

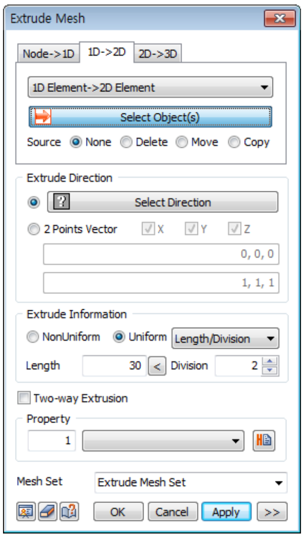
When checking the User defined mesh set, the generated elements are classified and registered in different mesh sets depending on the number of inputs in the [Offsets per Set] option. Entering a number registers the mesh sets in the specified uniform offset spacing and entering 2 or more numbers register the mesh sets in the specified uniform offset spacing and the rest are registered separately on the mesh set. When entering 2 or more numbers, the sum of input numbers should not be more than the total number of offsets.



1D -> 2D

Overview

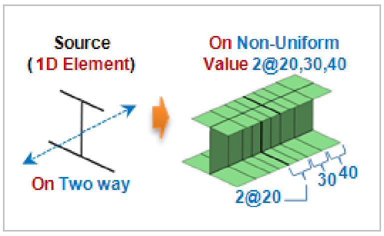
Generate a 2D element by extruding a 1D element, element edge (element outline) or line (edge).
Here, the used edge needs to be seeded (have seed information) or needs to be connected to the mesh.

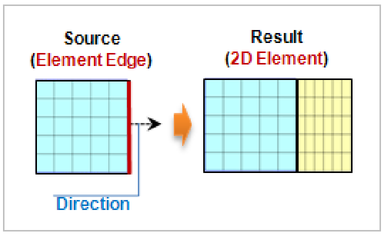


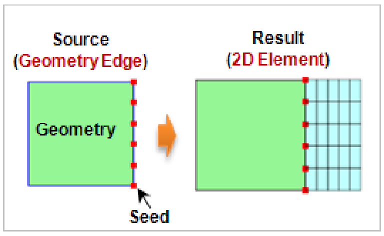
Methodology

Select the 1D element, element side or edge to be extruded and set the extrude direction, length and number of divisions. The element can be created by extruding in 1 direction or 2 directions. The original element used in extrude can be deleted/moved/copied. For move, the used element is moved to the end of the extruded element.

▸1D element
-> 2D element extrude
▸▸Element edge
->2D element extrude
▸▸▸Geometry edge
->2D element extrude







Extrude Direction

Select the element extrude direction with reference to the GCS or input the start and end points of the direction vector using the 2-point vector function. It is useful when extruding in an arbitrary direction.

Extrude Information

Set the total length and division of 2D element will be created. The division spacing can be set as either uniform or non-uniform. Entering a negative value for length (offset, spacing) extrudes in the opposite direction to the axis or vector direction.

[Non-uniform]
Specify the offset length and number simultaneously. The length can be listed using a comma (,) or as number@length for continuously repeating extrude operations.
For example, entering 10@3 creates 10 elements with a length of 3 each and entering 2,3,4 creates 3 elements, each with a length of 2,3 and 4.

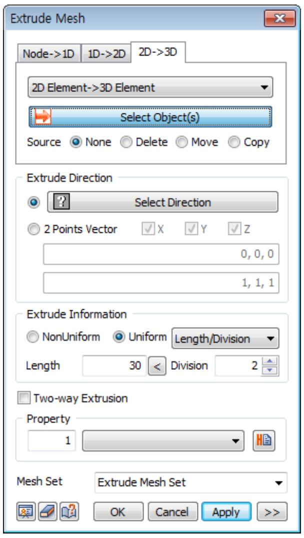
[Uniform]
Set the offset length or number, or input the total length and division spacing.

The shape of the 2D element can be selected from a triangle or a quadrilateral in advanced options (>>).

2D -> 3D

Overview

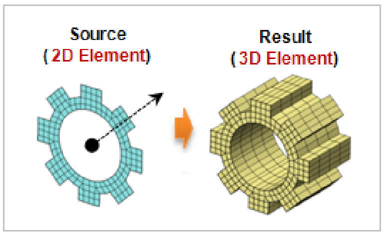
Generate a 3D element by extruding a 2D element of a boundary surface of a 3D element.

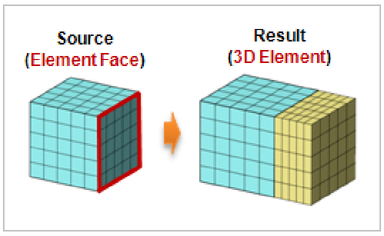


Methodology

Select the 2D element or 3D element face to be extruded and set the extrude direction, length and number of divisions. The element can be created by extruding in 1 direction or 2 directions. The original element used in extrude can be deleted/moved/copied. For move, the used element is moved to the end of the extruded element.

▸2D element
->3D element extrude
▸▸Element face
->3D element extrude





Extrude Direction

Select the element extrude direction with reference to the GCS or input the start and end points of the direction vector using the 2 points vector function. It is useful when extruding in an arbitrary direction.

Extrude Information

Set the total length and division of 3D element which will be created. The division spacing can be set as either uniform or nonuniform. Entering a negative value for length (offset, spacing) extrudes in the opposite direction to the axis or vector direction.

[Nonuniform]

Specify the offset length and number simultaneously. The length can be listed using a comma (,) or as number@length for continuously repeating extrude operations.

For example, entering 10@3 creates 10 elements with a length of 3 each and entering 2,3,4 creates 3 elements, each with a length of 2,3 and 4.

[Uniform]

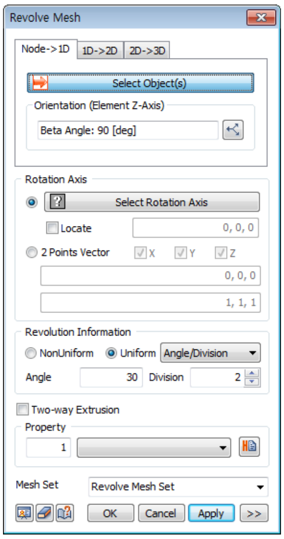
Set the offset length or number, or input the total length and division spacing.

5.2 Revolve

Node -> 1D

Overview

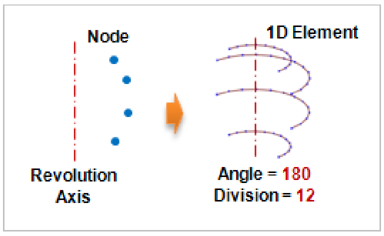
Generate a 1D element by rotating a node about the specified revolution axis by the input angle. It can be used to create various arch or circle shaped 1D elements without using geometry shape.



Overview

Generate a 1D element by rotating a node about the specified rotation axis by the input angle. It can be used to create various arch or circle shaped, 1D elements without using geometry shape.

▸Node revolve extrude



Rotation Axis

Select the rotation axis with reference to the GCS or input the start and end points of the direction vector using the 2 points vector function. '2 points vector' is useful when setting the reference axis in an arbitrary direction. When using the GCS, use the [Locate] option to set the revolution axis position using its coordinates. The revolution axis is moved to the coordinate position and the node is rotated about the moved axis.

Revolution Information

Set the rotation angle and division of 1D element which will be created. The division spacing can be set as either uniform or non-uniform. Entering a positive angle rotate extrudes in the counterclockwise direction and entering a negative angle rotate extrudes in the clockwise direction.

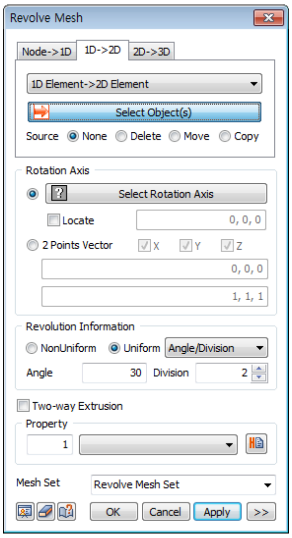
[Non-uniform]
Specify the rotation angle. The angle can be listed using a comma (,) or as number@angle for continuously repeating angles.
For example, entering 10@20 continuously creates 10 elements that are rotated 20 degrees from the previous element and entering 10,20,30 creates 3 elements, each rotated by 10 degrees, 20 degrees and 30 degrees.

[Uniform]
Set the rotation angle and number, or input the total rotation angle and division spacing.

1D-> 2D

Overview

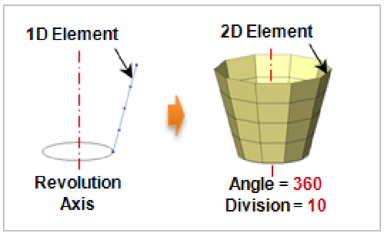
Generate a 2D element by revolve extruding a 1D element, element side (element outline) or line (edge). Here, the used edge needs to be seeded (have seed information) or needs to be connected to the mesh.

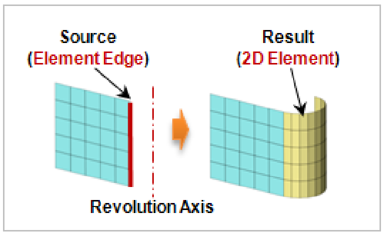


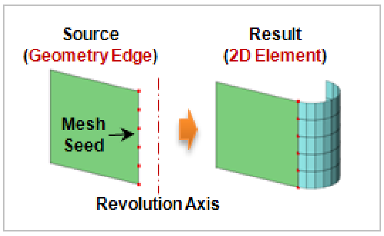
Methodology

Select the 1D element, element side or edge to be revolve extruded and set the rotation axis, rotation angle and number of divisions. The element can be created by extruding in 1 direction or 2 directions. The original element used in extrude can be deleted/moved/copied. For move, the used element is moved to the end of the extruded element.

▸1D element
->2D revolve extrude
▸▸Element edge
->2D revolve extrude
▸▸▸Geometry edge
->2D revolve extrude







Rotation Axis

Select the rotation axis with reference to the GCS or input the start and end points of the direction vector using the 2 points vector function. It is useful when setting the reference axis in an arbitrary direction. When using the GCS, use the [Locate] option to set the revolution axis position using its coordinates. The revolution axis is moved to the coordinate position and the node is rotate extruded about the moved axis.

Revolution Information

Set the rotation angle and division of 1D element will be created. The division spacing can be set as either uniform or non-uniform. Entering a positive angle rotate extrudes in the counterclockwise direction and entering a negative angle rotate extrudes in the clockwise direction.

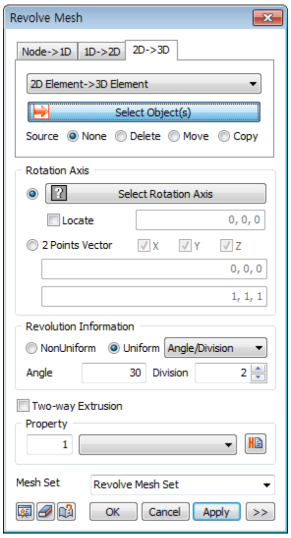
[Non-uniform]
Specify the rotation angle. The angle can be listed using a comma (,) or as number@angle for continuously repeating angles.
For example, entering 10@20 continuously creates 10 elements that are rotated 20 degrees from the previous element and entering 10,20,30 creates 3 elements, each rotated by 10 degrees, 20 degrees and 30 degrees.

[Uniform]
Set the rotation angle and number, or input the total rotation angle and division spacing.
The shape of the 2D element can be selected from a triangle or a quadrilateral in advanced options (>>).

2D-> 3D

Overview

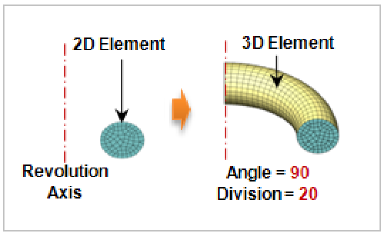
Generate a 3D element by rotate extruding a 2D element or a boundary surface of a 3D element.

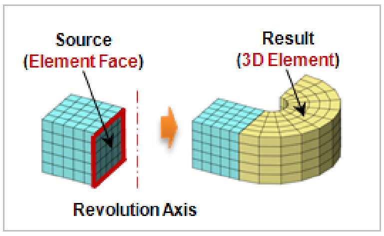


Methodology

Select the 2D element or 3D element boundary surface to be revolve extruded and set the rotation axis, rotation angle and number of divisions. The element can be created by extruding in 1 direction or 2 directions. The original element used in extrude can be deleted/moved/copied. For move, the used element is moved to the end of the extruded element.

▸2D element
->3D revolve extrude
▸▸Element Face
->3D revolve extrude





Rotation Axis

Select the rotation axis with reference to the GCS or input the start and end points of the direction vector using the 2 points vector function. It is useful when setting the reference axis in an arbitrary direction.
When using the GCS, use the [Locate] option to set the revolution axis position using its coordinates. The revolution axis is moved to the coordinate position and the node is rotate extruded about the moved axis.

Revolution Information

Set the rotation angle and division of 1D element will be created. The division spacing can be set as either uniform or non-uniform. Entering a positive angle rotate extrudes in the counterclockwise direction and entering a negative angle rotate extrudes in the clockwise direction.

[Non-uniform]
Specify the rotation angle. The angle can be listed using a comma (,) or as number@angle for continuously repeating angles.
For example, entering 10@20 continuously creates 10 elements that are rotated 20 degrees from the previous element and entering 10,20,30 creates 3 elements, each rotated by 10 degrees, 20 degrees and 30 degrees.

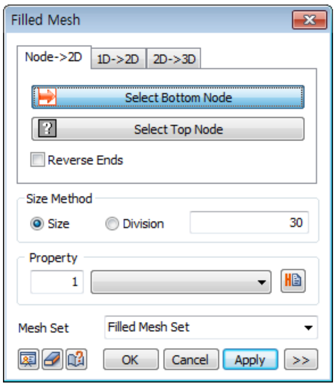
[Uniform]
Set the rotation angle and number, or input the total rotation angle and division spacing.

5.3 Fill

Node-> 2D

Overview

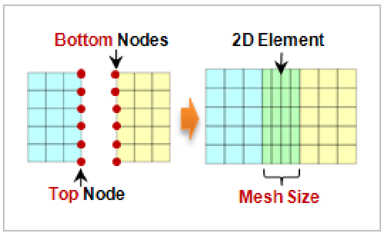
Generate a 2D surface element between 2 nodes. It is useful when creating elements in a certain area using edit/add mesh.



Methodology

Set the top and bottom nodes of the 2D element to set the total domain and define the mesh size by entering the mesh size directly or the number of divisions between nodes. The number of top nodes and bottom nodes must be the same and the selected nodes are assigned a number in order. The same numbered nodes correspond and generate a 2D element. To reverse the corresponding order, use the [Reverse Ends] option.

▸Node->2D element

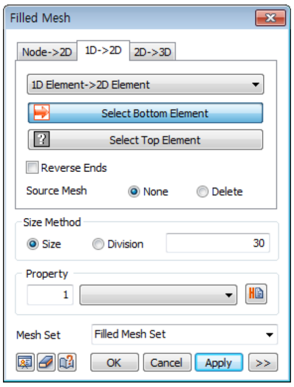


The shape of the 2D element can be selected from a triangle or a quadrilateral in advanced options (>>).

1D-> 2D

Overview

Generate a 2D surface element that connects 1D elements or 2 edges. It is useful when creating a 2D connecting element between structural elements or surface elements.

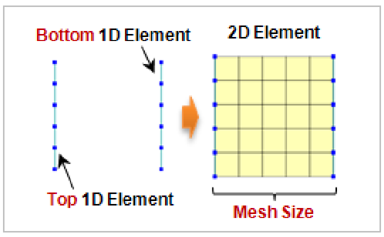


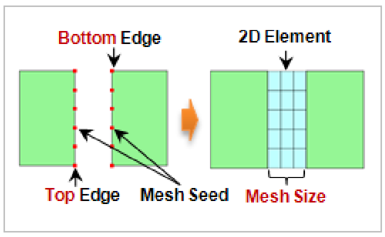
Methodology

Select the top and bottom nodes or edges of the 2D element and define the mesh size by entering the mesh size directly or the number of divisions between edges or nodes. The number of top nodes and bottom elements must be the same when using a 1D element and if a line (edge) is used, the input seed information for the top and bottom lines (edges) need to be the same. The bottom element, or start point of a line and top element, or start point of a line correspond in order and generate a 2D element. If the start point of the bottom element and end point of the top element correspond such that the element is twisted, use the [Reverse Ends] option to reverse the corresponding order.

▸1D element->2D element

▸▸Geometry edge
-> 2D element



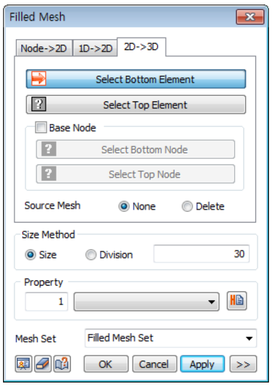


The shape of the 2D element can be selected from a triangle or a quadrilateral in advanced options (>>).

2D-> 3D

Overview

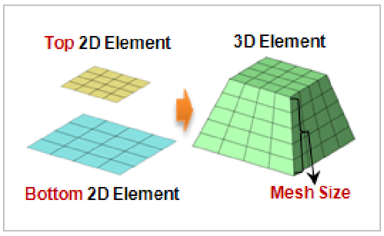
Generate a 3D element that connects 2D elements. It is useful when adding elements to a certain area to edit the model.



Methodology

Select the top and bottom 2D elements and define the 3D element size by entering the mesh size directly or the number of divisions between the selected 2D elements. The number of top and bottom elements of each selected 2D element need to the same and the element position or shape needs to be similar to allow 3D element fill. If the corresponding pairs cannot be found automatically, the corresponding reference nodes need to be specified manually. Each reference node needs to exist on the outline of the element.

▸2D->3D element



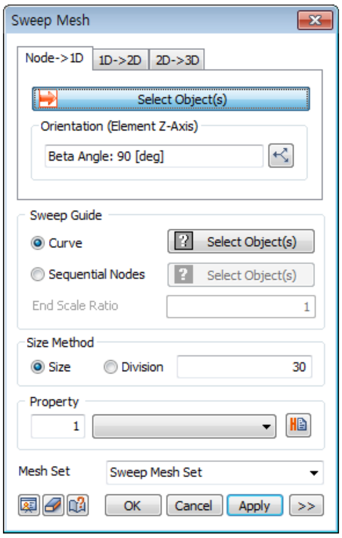
The shape of the 2D element can be selected from a triangle or a quadrilateral in advanced options (>>).

5.4 Sweep

Node -> 1D

Overview

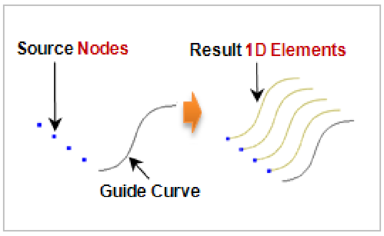
Generate a 1D element by sweeping a node about a selected guide curve. It is useful to create complex elements continuously.



Methodology

Select the element node to extrude and select the sweep guide curve or node positions that can substitute the guide curve. Define the mesh size directly or input the number of divisions for the selected curve.

▸Node sweep extrude



Sweep Guide

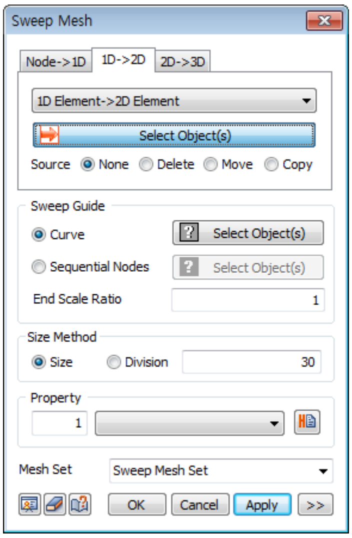
[Curve]
Only 1 guide curve can be selected. Hence, the curves for a complex shape need to be created into a single wire in advance.

[Sequential Nodes]
When selecting multiple nodes, be aware that the extrude direction is specified by the selection order. When selecting the nodes directly, the spacing between the nodes becomes the element size so the size does not need to be set.

1D-> 2D

Overview

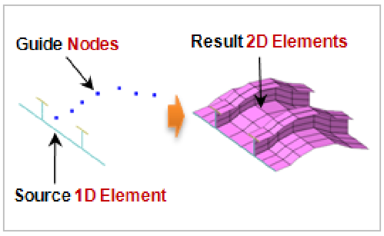
Generate a 2D element by extruding a 1D element, element side (element outline) or line (edge) along a guide curve. Here, the used edge needs to be seeded (have seed information) or needs to be connected to the mesh.

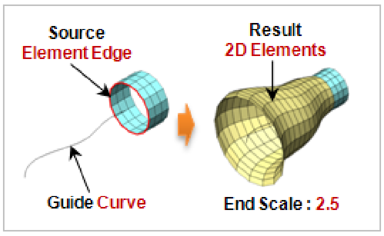


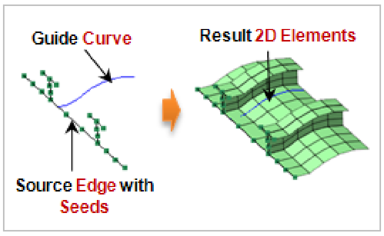
Methodology

Select the 1D element, element side or seeded line (edge) to be extruded and directly select the sweep guide curve or node positions that can substitute the guide curve. The element size can be directly entered or defined by the number of divisions on the selected extrude curve. The original element used in extrude can be deleted/moved/copied. For move, the used element is moved to the end of the extruded element.

▸1D element->2D Sweep extrude
▸▸Element edge ->2D Sweep extrude
▸▸▸Geometry edge ->2D Sweep extrude







Sweep Guide

[Curve]
Only 1 guide curve can be selected. Hence, the curves for a complex shape need to be created into a single wire in advance.

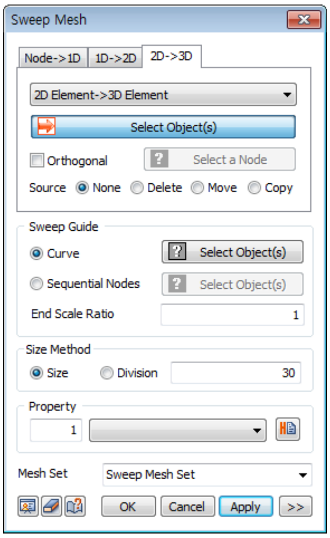
[Sequential Nodes]
When selecting multiple nodes, be aware that the extrude direction is specified by the selection order. When selecting the nodes directly, the spacing between the nodes becomes the element size so the size does not need to be set.

[End Scale Ratio]
Adjust the scale ratio at the end of the element generation to set the ratio for the extruded mesh size.
For example, entering 2 generates an element that is 2 times larger at the extrude finish point.

2D-> 3D

Overview

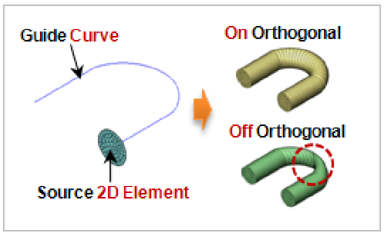
Generate a 3D element by extruding a 2D element or 3D element boundary surface along a guide curve.

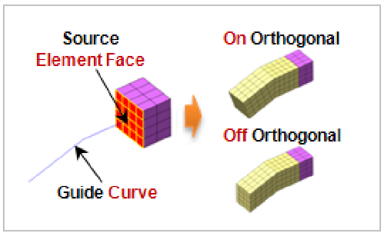


Methodology

Select the 2D element or 3D element boundary surface and select the sweep guide curve or node positions that can substitute the guide curve. Define the mesh size directly or input the number of divisions for the selected curve. The original element (2D element) used in extrude can be deleted/moved/copied. For move, the used element is moved to the end of the extruded element.

▸2D element
->3D Sweep extrude
▸▸Element Face
->3D Sweep extrude





Orthogonal

Extrude along the selected guide line or in the orthogonal direction to the selected node direction to generate an element. It is especially useful when rotate extrude along a curve.

Sweep Guide

[Curve]
Only 1 guide curve can be selected. Hence, the curves for a complex shape need to be created into a single wire in advance.

[Sequential Nodes]
When selecting multiple nodes, be aware that the extrude direction is specified by the selection order. When selecting the nodes directly, the spacing between the nodes becomes the element size so the size does not need to be set.

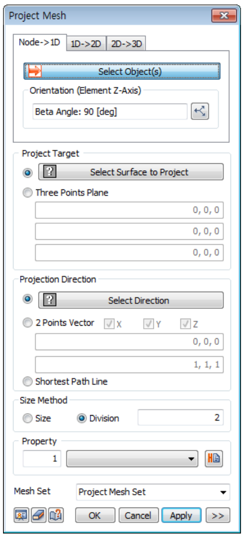
[End Scale Ratio]
Adjust the scale ratio at the end of the element generation to set the ratio for the extruded mesh size.
For example, entering 2 generates an element that is 2 times larger at the extrude finish point.

5.5 Project

Node -> 1D

Overview

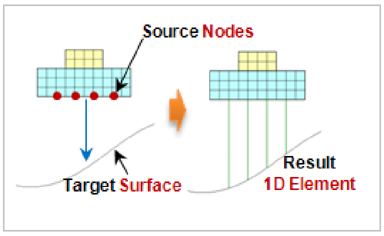
Generate a 1D element by project extrude a node. It is useful when generating an element in the same direction as the target surface (projection target).



Methodology

Select the element nodes to project extrude and specify the projection target (target surface) and projection direction. Define the mesh size directly or input the number of divisions for the projection distance.

▸Project extrude node



Project Target

Select an existing surface of a geometry shape or an element boundary surface as the projection target (target surface) or select 3 arbitrary points that pass the target surface using the [Tree points Plane] function.

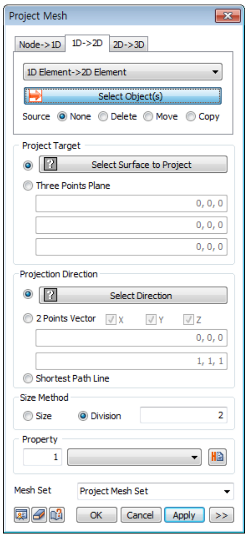
Projection Direction

Set the projection direction using the axis direction on the GCS or the direction of an arbitrary vector that connects the start and end points. Use the [Shortest Path Line] to automatically set the shortest distance direction between the node and projection target (target surface) in the normal direction.

1D-> 2D

Overview

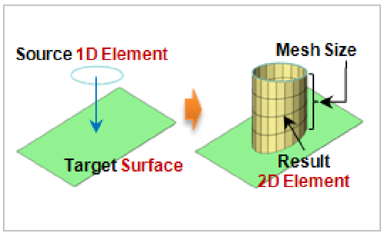
Generate a 2D element by project extruding a 1D element, element side (element boundary line) or edge onto the target surface. Here, the used edge needs to be seeded, or connected to a mesh.

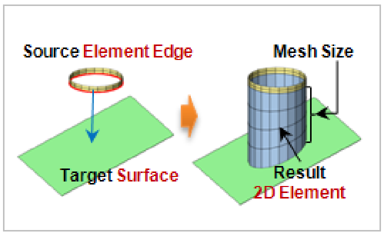


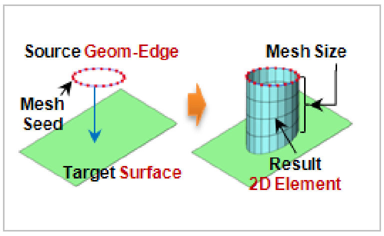
Methodology

Select the 1D element, element side or seeded line (edge) to be extruded and set the projection target (target surface) and direction. The element size can be directly entered or defined by the number of divisions on the distance to the projection. The original element used in extrude can be deleted/moved/copied. For move, the used element is moved to the end of the extruded element.

▸1D element
->2D Project extrude
▸▸Element edge
->2D Project extrude
▸▸▸Geometry edge
->2D Project extrude







Project Target

Select an existing surface of a geometry shape or an element boundary surface as the projection target (target surface). Or, select 3 arbitrary points that pass the target surface using the [Tree Points Plane] function.

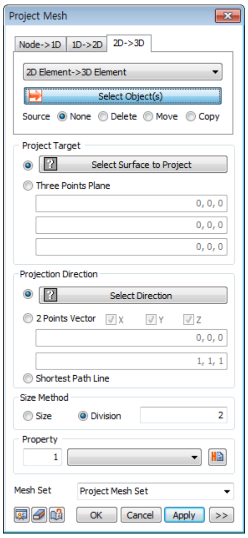
Projection Direction

Set the projection direction using the axis direction on the GCS or the direction of an arbitrary vector that connects the start and end points. Use the [Shortest Path Line] to automatically set the shortest distance direction between the node and projection target (target surface) in the normal direction.

2D-> 3D

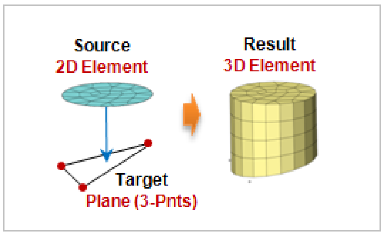
Overview

Generate a 3D element by project extruding a 2D element or 3D element surface (element boundary surface).

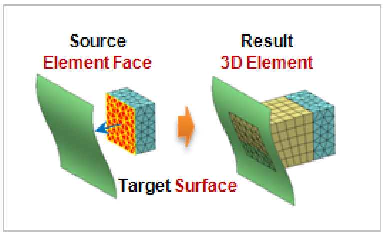


Methodology

Select the 2D element or element face to project extrude and set the projection target (target surface) and direction. The element size can be directly entered or defined by the number of divisions on the distance to the projection. The original element (2D element) used in extrude can be deleted/moved/copied. For move, the used element is moved to the end of the extruded element.



▸2D element
->3D Project extrude



▸▸Element face
->3D Project extrude

Project Target

Select an existing surface of a geometry shape or an element boundary surface as the project target (target surface). Or, select 3 arbitrary points that pass the target surface using the [Three points Plane] function.

Projection Direction

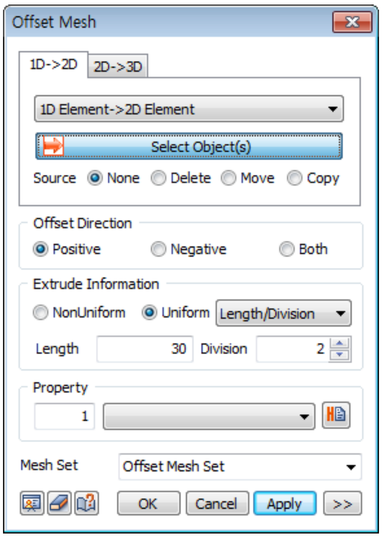
Set the projection direction using the axis direction on the GCS or the direction of an arbitrary vector that connects the start and end points. Use the [Shortest Path Line] to automatically set the shortest distance direction between the node and projection target (target surface) in the normal direction.

5.6 Offset

1D-> 2D

Overview

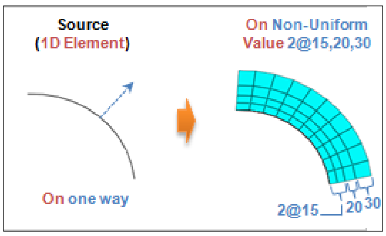
Generate a 2D element by offset extruding a 1D element, element side or line (edge). Here, the used edge needs to be seeded (have seed information) or needs to be connected to the mesh.

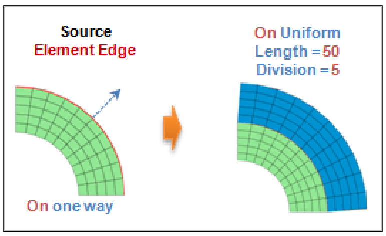


Methodology

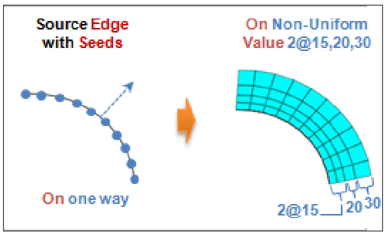
Select the 1D element, element side or seeded line (edge) to be extruded and directly select the offset direction, extrude length and number of element divisions. The element can be created by extruding in 1 direction or 2 directions. The original element (1D) used in extrude can be deleted/moved/copied. For move, the used element is moved to the end of the extruded element.

▸1D element
->2D Offset extrude
▸▸Element edge
->2D Offset extrude





▸Geometry edge
->2D Offset extrude



Offset Direction

Set the extrude direction in the selected object [Positive] direction (normal direction), [Negative] direction (opposite normal direction), or in both directions.

Extrude Information

Set the total width and division of 2D element will be created. The division spacing can be uniform or non-uniform. Entering a negative value for length (offset, spacing) extrudes in the opposite direction to the axis or vector direction.

[Non-uniform]
Specify the offset length and number simultaneously. The length can be listed using a comma (,) or as number@length for continuously repeating extrude operations.
For example, entering 10@3 creates 10 elements with a length of 3 each and entering 2,3,4 creates 3 elements, each with a length of 2,3 and 4.

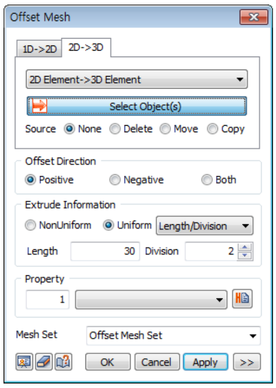
[Uniform]
Set the offset length or number, or input the total length and division spacing.

The shape of the 2D element can be selected from a triangle or a quadrilateral in advanced options (>>).

2D-> 3D

Overview

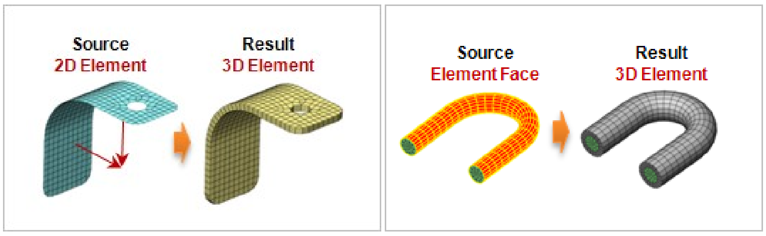
Generate a 3D element by offset extruding a 2D element or a 3D element boundary surface.



Methodology

Select the 2D element or 3D element boundary surface and set the offset direction, extrude length and number of divisions. The element can be created by extruding in 1 direction or 2 directions. The original element (2D) used in extrude can be deleted/moved/copied. For move, the used element is moved to the end of the extruded element.

▸2D element
->3D Offset extrude
▸▸Element face
->3D Offset extrude



Offset Direction

Set the extrude direction in the selected object [Positive] direction (normal direction), [Negative] direction (opposite normal direction), or in [Both] directions.

Extrude Information

Set the total width and division of 2D element will be created. The division spacing can be uniform or non-uniform. Entering a negative value for length (offset, spacing) extrudes in the opposite direction to the axis or vector direction.

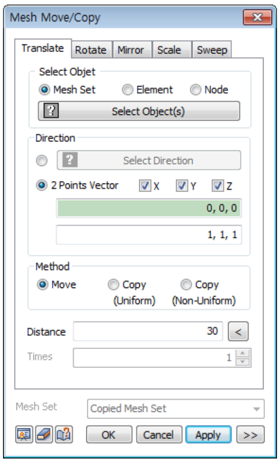
[Non-uniform]
Specify the offset length and number simultaneously. The length can be listed using a comma (,) or as number@length for continuously repeating extrude operations.
For example, entering 10@3 creates 10 elements with a length of 3 each and entering 2,3,4 creates 3 elements, each with a length of 2,3 and 4.
[Uniform]
Set the offset length or number, or input the total length and division spacing.

Section 6 Transform

6.1 Translate

Overview

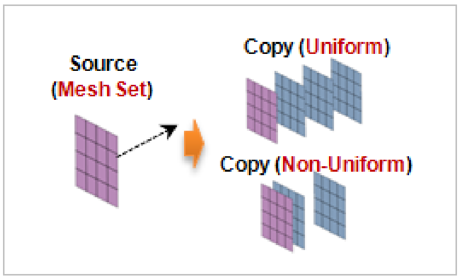
Move/copy a node, element or mesh set. It is useful for moving the element position or copying the same element to a specified distance.



Methodology

Select the node, element or mesh set to move (or copy) and define the direction.

‣Translate (Move/Copy)



Direction

Select the move/copy direction with respect to the GCS or the direction of a vector that connects 2 arbitrary points. For the 2 points vector function, select (deselect) a particular coordinate axis and select only the directional comp1nt of a vector defined by a start and end point.

Method

Move the position of the selected element or copy and move the element to a specified or arbitrary distance.

[Move]
Directly input the move distance or using the 2 points vector function, automatically calculate the actual move distance with reference to the start and end points. Select the button to check the automatically calculated distance.

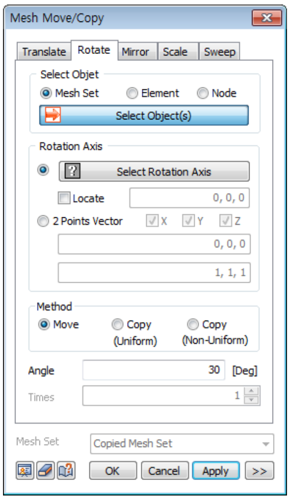
[Copy (Uniform)]
Specify the copy distance and number of copies. Entering a negative value copies the object in the opposite direction to the set direction.

[Copy (Non-Uniform)]
The length can be listed using a comma (,) or as number@length for continuously repeated moves (copies). For example, entering 10@3 creates 10 elements with a move length of 3 each and entering 2,3,4 creates 3 elements, each with a move/copy length of 2,3 and 4.

6.2 Rotate

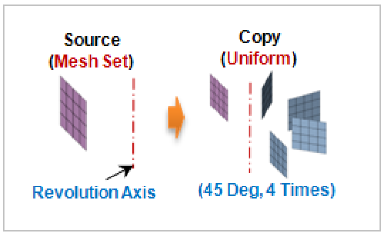
Overview

Rotate a node, element or mesh set about the rotation axis to move/copy. It is useful for continuously generating a mesh that forms a certain angle.



Methodology
Select the node, element or mesh set to rotate move (or copy) and define the revolution axis.

▸Rotate(Move/Copy)



Rotation Axis

Select the move/copy revolution axis with respect to the GCS or for the 2 points vector function, input the start and end point coordinates of the direction vector. It is useful when setting the reference axis in an arbitrary direction.

When using the GCS, use the [Locate] option to set the rotation axis position using its coordinates. The rotation axis is moved to the specified coordinate position and the node is rotated and moved/copied about this axis.
For the 2 points vector function, select (deselect) a particular coordinate axis and select only the directional comp1nt of a vector defined by a start and end point.

Methodology

Specify the rotation angle and rotation copy angle. A uniform angle or arbitrarily set non-uniform angle can be used to rotate and move/copy.

[Move]
Directly input the rotation angle.

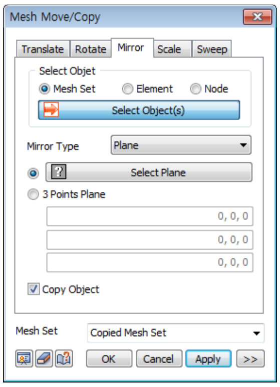
[Copy (Uniform)]
Set the rotation angle and number of repetitions. Input a negative to rotate in the opposite direction to the set direction.

[Copy (Non-uniform)]
The angle can be listed using a comma (,) or as number@angle for continuously repeating angles.
For example, entering 10@30 continuously creates 10 elements that are rotated 30 degrees from the previous element and entering 20,30,40 creates 3 elements, each rotated and moved/copied by 20 degrees, 30 degrees and 40 degrees.

6.3 Mirror

Overview

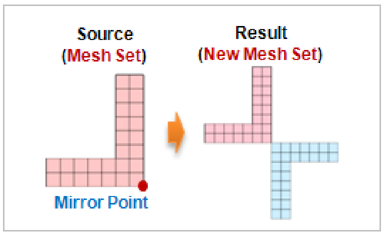
Mirror a node, element or mesh set and move/copy. If a section shape is symmetrical, model only half and use the mirror (copy) function to easily model the whole section.

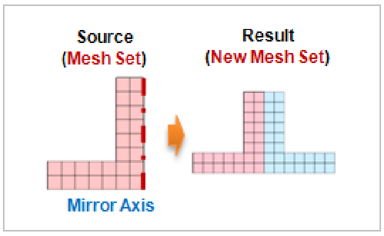


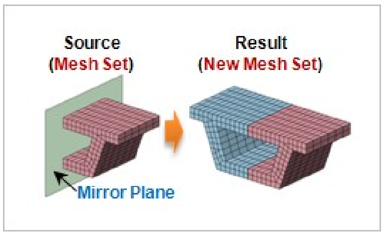
Methodology

Select the node, element or mesh set to mirror move/copy and select the mirror type. The mirror types are point, axis and plane. Use the copy target option to use mirror move and copy simultaneously.

▸Mirror point
▸▸Mirror axis
▸▸▸Mirror plane







Mirror Type

[Point] : Directly select the mirror point or input the coordinates.
[Axis] : Select the mirror axis on the GCS or define an arbitrary vector axis that connects the start and end points.
[Plane] : Select the mirror plane or define an arbitrary plane that passes 3 points.

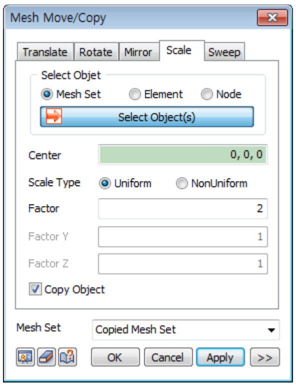
Tip

Generating an element using the mirror move operation reverses its coordinates with respect to the original element.
For example, the normal direction is reversed for 2D elements that are mirrored about an axis or point. This creates an error during analysis and the element coordinate system needs to be united from Mesh > Element > Parameter.

6.4 Scale

Overview

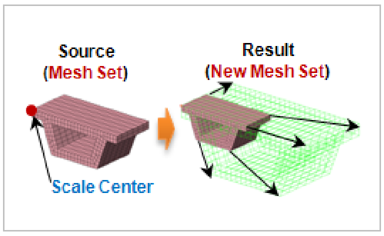
Scale up/down a node, element or mesh set with reference to a scale center.



Methodology

Select the node, element or mesh set to scale up/down and specify the scale center. Apply a constant ratio with respect to the scale center or selectively apply a scale factor on the axial direction of the GCS.
Check the copy target option to use scale up/down and copy simultaneously

▸Scale



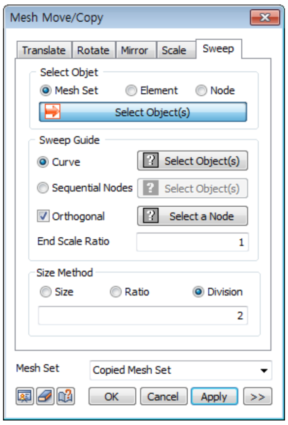
Scale type

[Uniform] : Scale up/down the selected target in all axial directions of the GCS uniformly.
[NonUniform] : The scale factor can be applied differently for each axial direction on the GCS.

6.5 Sweep

Overview

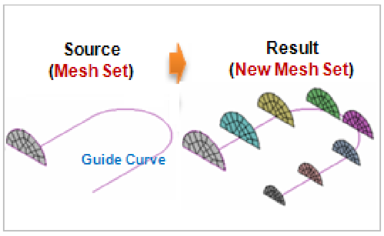
Move a node, element or mesh set along a sweep guide and copy.



Methodology

Select the node, element or mesh set and directly select the sweep guide or node positions that can substitute the guide curve. Specify the center point. Apply a certain ratio with respect to the center point or apply a selective scale factor on the axial directions of the GCS.

▸Sweep pattern



Sweep Guide

[Curve]

Only 1 guide curve can be selected. Hence, the curves for a complex shape need to be created into a single wire in advance.

[Sequential Nodes]

When selecting multiple nodes, be aware that the extrude direction is specified by the selection order. When selecting the nodes directly, the spacing between the nodes becomes the element size so the size does not need to be set.

[Orthogonal]

The copied shape is always perpendicular to the sweep guide and the reference point can be set by selecting a node.

[End Scale Ratio]

Adjust the scale factor at the end of the element generation to set the ratio for the mesh size.

For example, entering 2 generates an element that is 2 times larger at the sweep finish point.

Size Method

[Size] : Directly specify the mesh size. For Sweep move/copy, size signifies the sweep interval.

[Ratio] : Define the sweep interval and number with a ratio between 0~1. For example, entering 0.3,0.5,1 moves/copies the mesh at 0.3,0.5,1 ratio positions on the total length of the sweep guide.

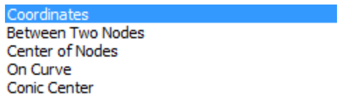
[Divisions] : Define the sweep interval by the number of divisions on the total length of the sweep guide.

Section 7 Node

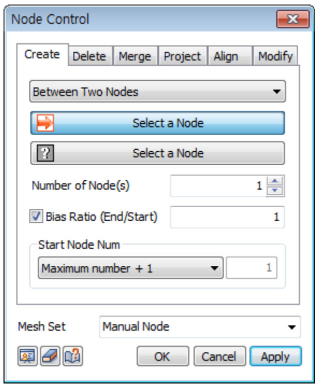
7.1 Create

Overview

Add a node in the desired position on the 3D work space. There are a total of 5 node creation methods.



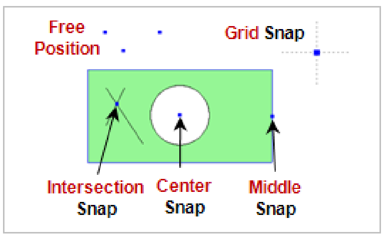
The position and number can be freely adjusted depending on the creation method. The additional node number can be set as the minimum non-overlapping and available number or the maximum number+1.



Methodology

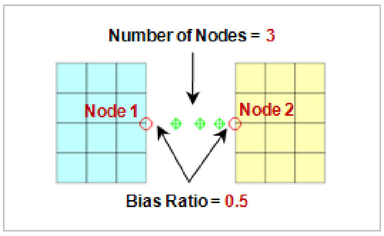
[Coordinates]
Directly input the node coordinates or select the position on the screen using snap point.

‣Create node : Coordinates



[Between 2 nodes]

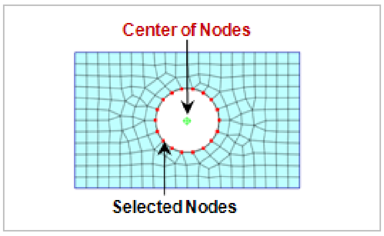
▸Create node : Between 2 nodes



Create an additional node between the 2 selected nodes. Select the number of nodes to create and when creating more than 1 node, adjust the spacing by entering the ratio between the starting space and ending space of nodes,

[Center of nodes]

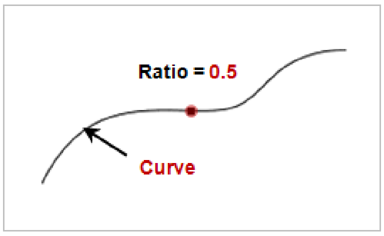
▸Create node : Center of nodes



Create an additional node at the center of the shape created by selecting 2 or more nodes.

[On curve]

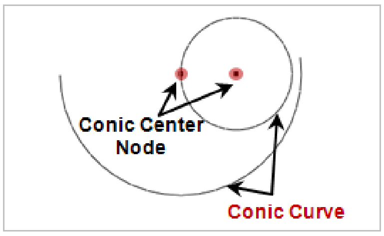
▸ Create node : On curve



Add a node on the selected curve. The node position can be specified by directly entering its distance from the start or end point, or by entering a ratio between 0~1. The [Reverse Direction] option can be used to reverse the direction of the start point and end point.

[Conic center]

▸ Create node : Conic center

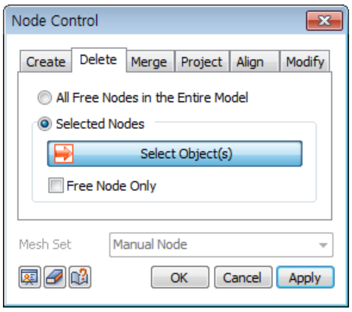


Automatically create a node at the center point of selected arc (circle). The node can also be created by using the snap position for arc centers.

7.2 Delete

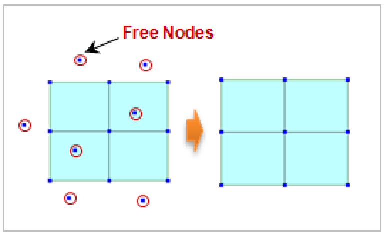
Overview

Delete a node.



Methodology

▸Delete node

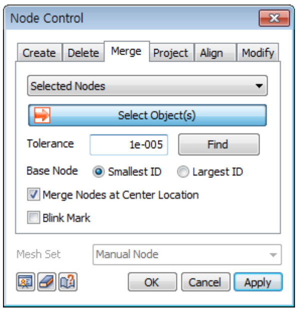


The free nodes that are not connected to an element can be deleted and selected. However, deleting a node that forms an element will delete the element as well.

7.3 Merge

Overview

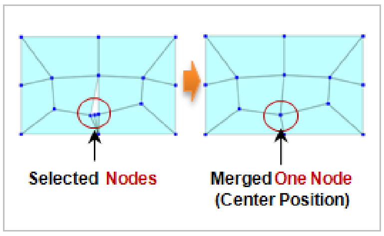
Merge 2 or more nodes into 1, or merge two nodes by selecting be moved and be kept node continuously. It is mostly used to connect nodes between elements that are slightly apart during the modeling modification process. It is useful when checking for model errors due to free edges by checking the free nodes that are not connected to an element.



Methodology

[Selected Nodes]

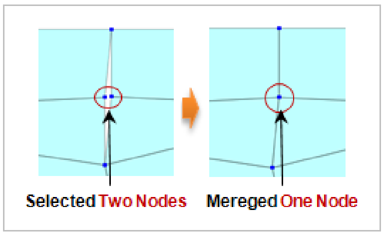
▸Merge : Selected Nodes



Select the nodes to merge and define the merge criterion. The tolerance is the allowable limit of the merger; nodes with spacing smaller than the tolerance are merged. Selecting 'find' automatically searches for nodes below the tolerance. The merged node positions are specified by the node number (Smallest ID/Largest ID) or the Center of Nodes (Merge Nodes at the Center Location). The [Blink Mark] function indicates the Free Nodes on the screen, which can be used to distinguish modeling errors such as Free Edges.

[2-Nodes]

▸Merge : 2-Nodes

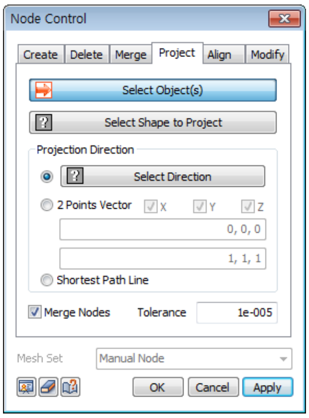


Merge 2 nodes by selecting the moving node and maintained node. If a Free Edge is created on a small area, the Merge 2-Nodes function can be used to edit easily.

7.4 Project

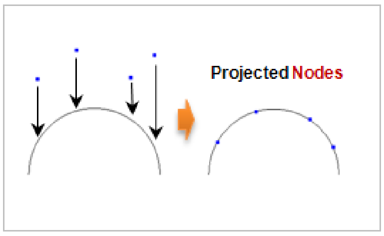
Overview

Project the node onto the target surface or edge.



Methodology

▸Node projection

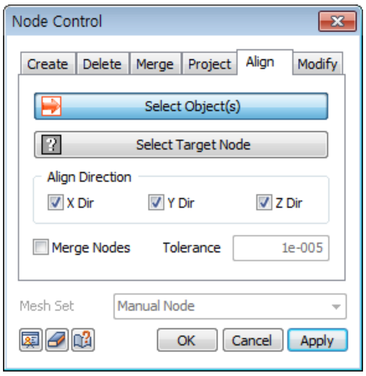


Select the nodes to project; the target shape can be selected between a surface and edge. The projection direction can be selected using the axis direction on the GCS or the direction of an arbitrary vector that connects the start and end points. Use the [Shortest Path Line] to automatically set the shortest distance direction between the node and projection target (target surface) in the normal direction. Selecting all nodes of the element moves the whole mesh. Selecting only some nodes, the unselected nodes remain in their position and the element shape and size automatically changes with the projection distance.

7.5 Align

Overview

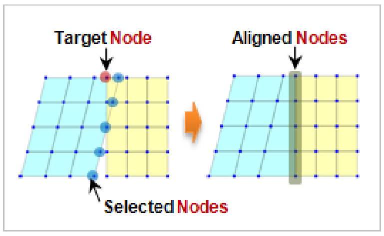
Align/move the selected nodes with reference to the target node. It is useful when editing the shape with respect to the target node and alignment direction.



Methodology

Select the nodes to align and the alignment reference node to move the target nodes in the alignment direction. The alignment direction can be selected from 1 of the X, Y, Z axis directions of the GCS. Selecting all nodes of the element moves the whole mesh. Selecting only some nodes, the unselected nodes remain in their position and the element shape and size automatically changes with the projection distance.

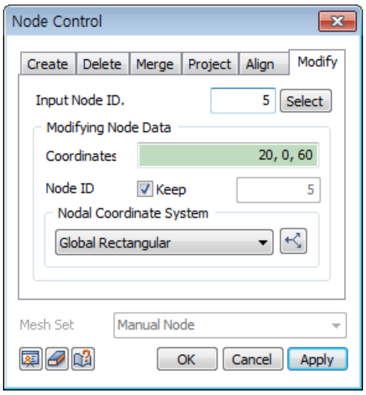
▸Node alignment



7.6 Modify

Overview

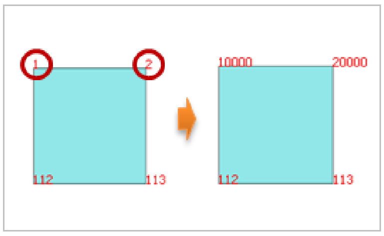
Select individual nodes to edit the node number and node coordinate system.



Methodology

Select the node to edit or input the node number to automatically print the coordinates of that point. The node number can be kept or a new node can be assigned, provided that the new number does not conflict with an existing node number. Simultaneously, the coordinate system of the selected node can be changed. The node coordinate system change can be d1 simultaneously for multiple nodes in the coordinate system function.

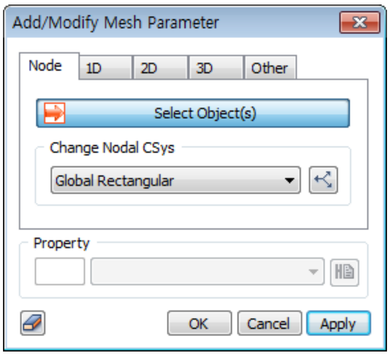
▸Edit node



7.7 Coordinate System

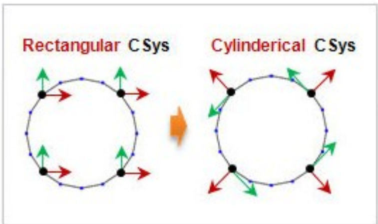
Overview

Change the node coordinate system.



Methodology

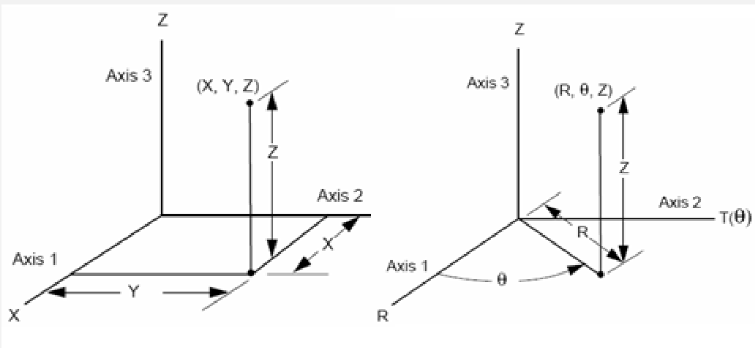
▸Change node coordinate system



Select the node to change the coordinate system. The changed coordinate system can be selected from the Rectangular coordinate system, Cylindrical coordinate system and the arbitrary setting using the 3-point plane.

Tip

The Rectangular coordinate system and Cylindrical coordinate system can be distinguished as follows.



<Rectangular Coordinate> <Cylindrical Coordinate>

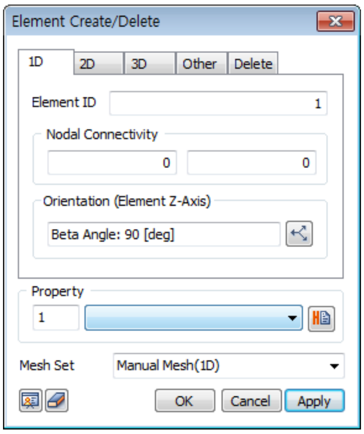
Section 8 Element

8.1 Create

1D

Overview

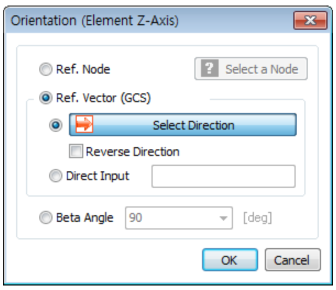
Create a 1D element connecting 2 nodes. This is useful when creating structural elements (pile, embedded truss, truss element) which do not require connections to the neighboring ground individually.



Methodology

Input the element ID that becomes the end points of the 1D element. Already created nodes can be selected in order on the screen. The element ID is automatically set to maximum previous number+1. Note that the number being directly entered cannot overlap with an existing node ID. Structural properties that will be assigned can be set or added to the created element and the mesh set can be created separately.

[Orientation (Element Z axis)]
This function is used to unify the direction property of a 1D element to 1 direction or to set the major and minor axis directions. Adjust the Z axis direction by checking the element coordinate axis and assign with reference to the Beta angle.



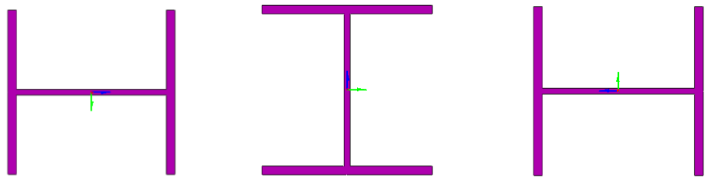
- Reference Node: Select the reference node for the sectional direction of the 1D element. The element Z coordinate direction is set with reference to the selected node.
- Reference Vector (GCS): Set the Z coordinate direction of the selected element using the GCS direction or the input vector direction.

- Beta Angle: Angles 0,90,180 can be chosen and the selected Beta angle rotates the element by that angle with reference to the X axis.

▸Beta angle:0

▸▸Beta angle:90

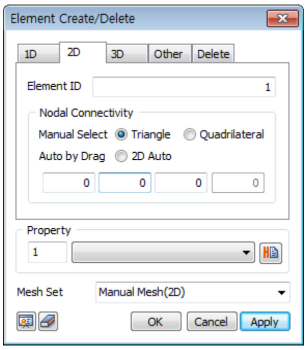
▸▸▸Beta angle:180



2D

Overview

Create a 2D element connecting 2 nodes. This is useful when creating an arbitrary surface strain element in an area where mesh auto-generation has failed.



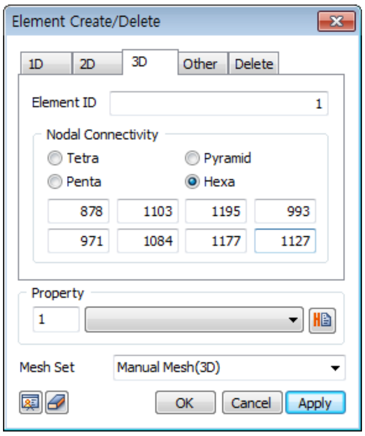
Methodology

Create a triangular or rectangular element depending on the number of selected nodes. Directly input the node numbers that will become the corners of the 2D element, or select already created nodes in order on the screen. The element ID is automatically set to maximum previous number+1 and when entering the number directly, the number cannot overlap with an existing node number. Properties that will be assigned can be set or added and the mesh set can be created separately. Using the automatic generation function generates a mesh set immediately after node selection.

3D

Overview

Create a 3D element filling the space between selected nodes. It is useful when creating individual 3D elements of a complex geometry.



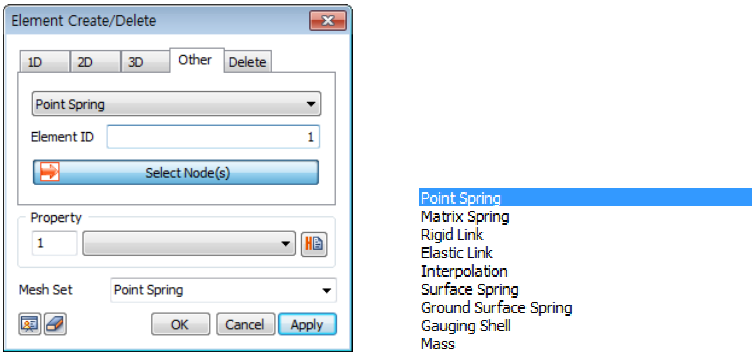
Methodology

Create a tetrahedral, pyramidal, pentahedral, hexahedral shape depending on the number of selected nodes. Directly input the node numbers that will become the corners of the 3D element, or select already created nodes in order on the screen. The element number is automatically set to maximum previous number+1 and when entering the number directly, the number cannot overlap with an existing node number. Properties which will be assigned can be set or added and the mesh set can be created separately.

Other

Overview

Create a spring, node link or Pile affiliated elements as shown below. The assigned properties can be defined depending on the created element type.



Methodology

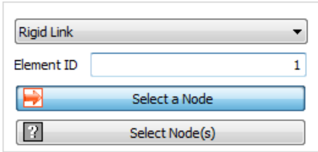
[Point spring]

Create a spring with a constant stiffness on the selected node. The constraints on deformation and rotation with reference to the GCS are defined by the spring constant and damping coefficient. It is mostly used as a flexible support condition for ground or a constraint condition for dynamic analysis.

[Matrix Spring]

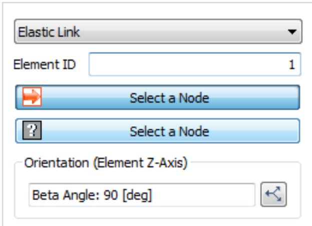
Has the same function as the Point spring. However, the spring constant for deformation and rotation can be directly input into a matrix when defining the characteristics.

[Rigid links]



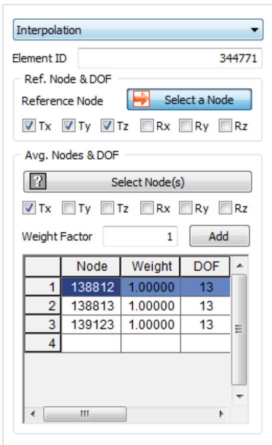
Create a link element that connects 2 selected nodes. Select the first node (reference) and select multiple nodes that become the connection target. It is used to simulate the rigid behavior between 2 nodes under deformation and rotation and the constraint direction can be defined the with reference to the GCS.

[Elastic link]



Connect 2 nodes with a spring that has a constant stiffness. Select the first node (reference) and select another node to create the link. Like the point string, the property is defined by the constant stiffness to deformation and rotation.

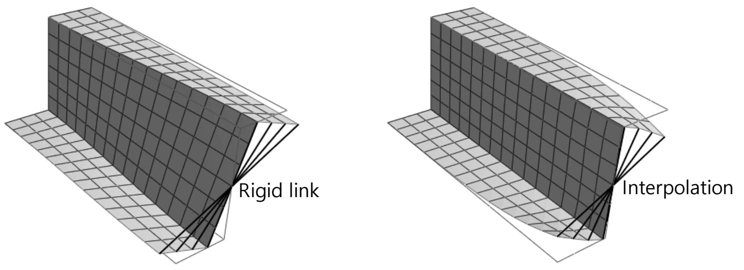
[Interpolation]



This function simulates the behavior of the standard (reference) node by weighing the average behavior of the selected nodes. It restricts the movement between connected nodes, similar to the rigid link. However, the interpolation element allows the relative behavior of a node due to movement at multiple different nodes. Hence, the average behavior at the multiple other nodes determines the movement of the reference node (dependent node).
Select the nodes to restrict and the degree of freedom, and then select the nodes to take the average values from. The weight of each node can be applied.

▸Rigid link

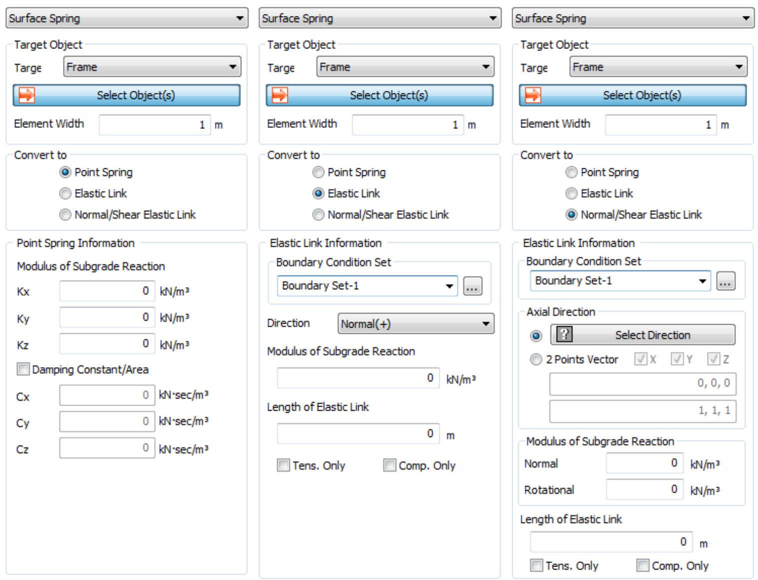
▸▸Interpolation



[Surface spring]

Create a point spring or elastic link by entering the spring stiffness per unit area at the support point of an element.

This is used to consider the flexible support condition of the ground during foundation analysis or underground structure analysis. Entering the spring stiffness per unit area automatically converts it to the spring or link acting on the node by considering the selected element area.



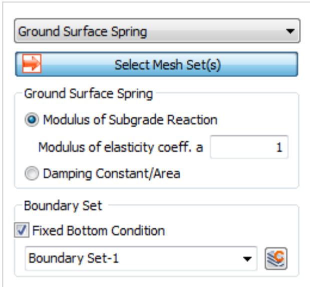
The surface spring inputs are as follows.

- Frame: Create a point spring or elastic link on a 1D element node. Input the width to calculate the support stiffness per unit length of a beam element.
- Planar: Create a spring or link by selecting a 2D element.
- Solid-Face: Specify an arbitrary surface on 3D Solid and create a spring or link at all nodes connected to that surface.
- Element Edge: Select an outline of a 2D element and create a spring or link at the nodes connected to that outline.

The input elastic link per unit area is a way to define the spring constant. Other options are available, such as [Tension only] or [Compression only].

[Ground surface spring]

This automatically creates the elastic/viscous boundary elements needed for dynamic analysis. Selecting a mesh automatically creates boundary conditions at the left/right/floor of the selected mesh and the spring constant is automatically calculated from the material/characteristics assigned to the element.



For dynamic analysis, the bottom of the model (floor surface) is often assigned a fixed condition (displacement constraint) to simulate bedrock conditions. Check the [Fixed Bottom Condition] to set this condition more easily.

Tip

⁎ How to create an elastic boundary element

- The elastic spring is used as a ground boundary condition for Eigen value analysis and Response spectrum analysis.
- Creating an elastic spring can be hard for beginners and the elastic spring element can be created from the following steps.

1. Use the elastic modulus of the ground to compute Kv0. (The Equation is shown below.)

$$k_{vo} = \frac{1}{30}\alpha \cdot E_0, \quad k_{ho} = \frac{1}{30}\alpha \cdot E_0$$

Modulus of deformation E0 from the following test methods (kfg/cm2)	a	
	Regular time	During earthquake
1/2 of E0 from the cyclic curve of the plate load test, d1 using a rigid circular plate of 30cm diameter	1	2
E0 measured in the borehole	4	8
E0 from the unconfined or tri-axial compression test on a specimen	4	8
E0 estimated by the N value from the Standard Penetration test when E0=28N	1	2

Here, E0: Elastic modulus of the ground, a Coefficient depending on test condition

2. Re-calculate the Subgrade Reaction Modulus Kv(= Kh) using the computed Kv0.

$$K_v = k_{v0}\left(\frac{B_v}{30}\right)^{3/4}$$

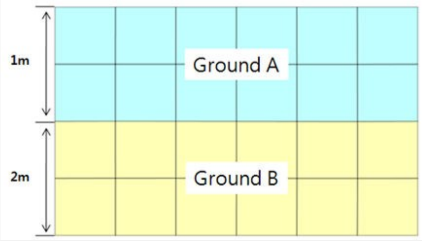
Here, $B_v = \sqrt{A_v}$

The area Av becomes the area where the subgrade reaction spring will be installed.

If the model exists like the following figure,

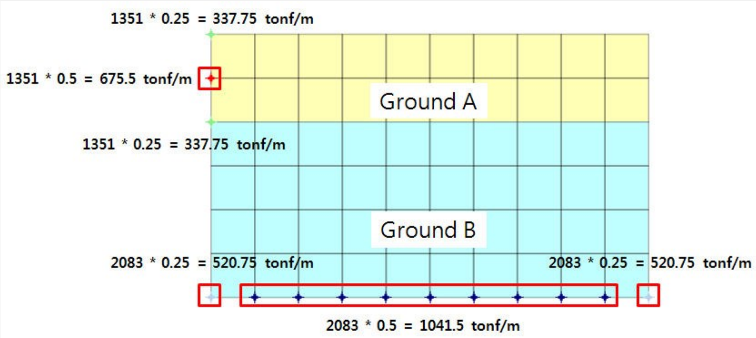
Area of Ground A is Av=1m(Left length of model)*1m(Unit width of 2D analysis)=1m^2, Bv becomes 1m=100cm.

Using the same method, the unit width of Ground B is Bv=√(20000)cm=141.42136 cm.



Ultimately, the Subgrade Reaction Modulus K can be computed and a point spring is created on the node, considering the area of the element.

	E (tonf/m^2)	Ky0	A (cm)	B	K (tonf/m^3)	α
Ground A	1000	3.3333	1.00E + 04	100	1351.186643	1
Ground B	2000	6.6667	2.00E + 04	1414213562	2083.845925	1



The spring coefficient of the floor (Z direction) is created with the same value as the X direction. (Element length x Width (1m) = Cross sectional area, so only consider the effective length of the element.)
2 overlapping boundary elements are created where the ground and ground meet.

*** How to create a viscous boundary element**

- The viscous boundary element required as a model boundary condition for time history analysis.
- The viscous boundary element can be created from the following steps.

1. Compute Cp, Cs

   Cp, Cs can be calculated using the equation below.

$$C_p = \rho \cdot A \cdot \sqrt{\frac{\lambda + 2G}{\rho}} = W \cdot A \cdot \sqrt{\frac{\lambda + 2G}{W \cdot 9.81}} = c_p \cdot A$$

$$C_s = \rho \cdot A \cdot \sqrt{\frac{G}{\rho}} = W \cdot A \cdot \sqrt{\frac{G}{W \cdot 9.81}} = c_s \cdot A$$

   Here, $\lambda = \frac{v \cdot E}{(1+v)(1-2v)}$, $G = \frac{E}{2(1+v)}$,

   λ : Bulk modulus, G : Shear modulus, E : Elastic modulus, v : Poisson's ratio, A : Cross-section area

2. The cross-section area is automatically considered until the surface spring is created, so only the Cp,

Cs needs to be computed.

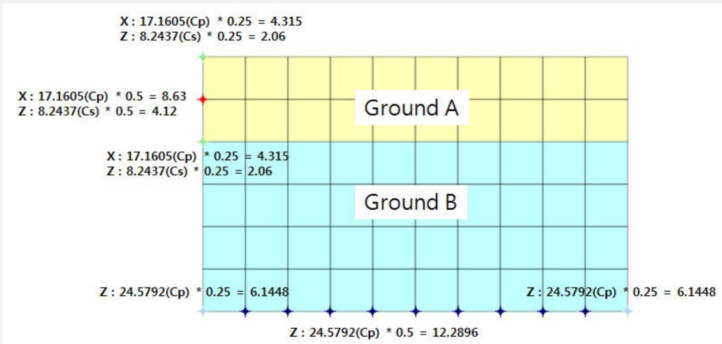
	Elastic modulus	Bulk modulus	Shear modulus	Unit weight	Poisson's ratio	P wave	S wave
	E (tonf/m^2)	λ (tonf/m^2)	G (tonf/m^2)	W (tonf/m^3)	v	Cp (tonf·sec/m^3)	Cp (tonf·sec/m^3)
GroundA	1000	864.1975309	370.3703704	1.8	0.35	17.1605	8.2437
GroundB	2000	1459.531181	751.8796992	2	0.33	24.5792	12.381

Multiplying the Cp, Cs (tonf·sec/m^3 units) to the cross-section area eventually leads to the spring stiffness of the viscous boundary element in tonf·sec/m units.

The shaded cell parameters are the physical properties of the ground the user inputs during modeling and the Bulk modulus and Shear modulus are calculated using the Elastic modulus and Poisson's ratio. Hence, there is no need to input additional values when creating a viscous boundary element.

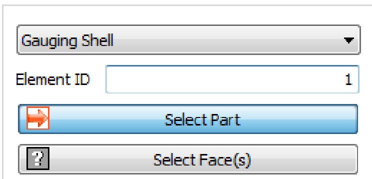
When creating the viscous boundary element automatically, the spring is automatically created by considering the element area (effective length*unit width) as shown below. Input the Cp value for the normal direction coefficient at the point of spring creation and input the Cs value for the parallel direction.

For example, the Cx of the spring coefficient created on the left/right of the model is the Cp of each ground and Cz becomes the Cs value. The bottom spring coefficient Cz becomes the Cp value.

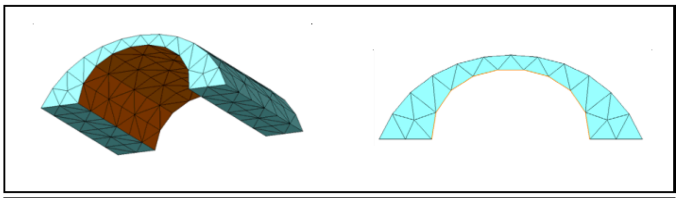


[Gauging shell]

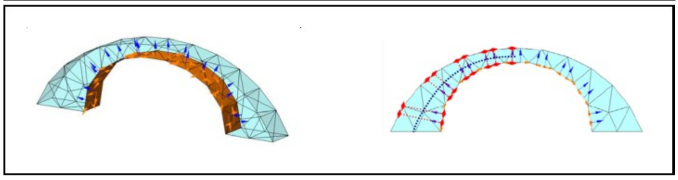
Create a shell element to check the force and moment on the surface of a solid element structure. To create a gauging shell, select the base solid element and then select the element surface on the solid to extrude the gauging shell from. The stiffness of the gauging shell is calculated by applying the stiffness increment coefficient to the stiffness of the solid element. The thickness of the selected solid is automatically considered and the thickness of each element is calculated.



▶Select element surface

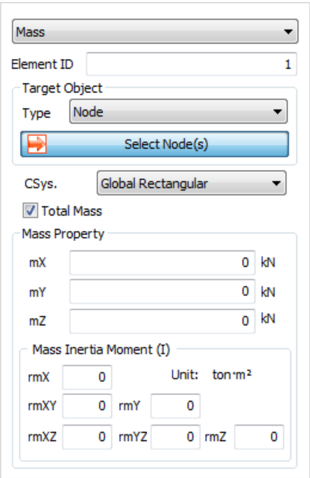


▸Gauging shell thickness
(Length of red dotted line)



[Mass]

Input the lumped mass on an arbitrary point. It is used to convert the loading into mass and apply it to the analysis.

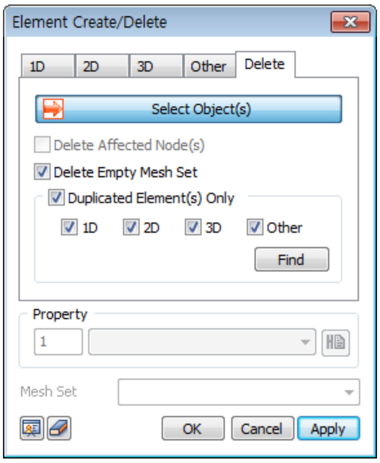


Check total mass to automatically divide and input the lumped mass data entered in the mass property onto a selected node. The sum of the divided lumped mass data on a node, created using the total mass option, is equal to the entered lumped mass data. Entering the loading using the converted mass value and selecting the total mass option allows easy application of mass data for Eigen value analysis, Response spectrum, analysis, Time history analysis etc. The lumped mass data is input with respect to the GCS and the moment of inertia (I) is defined according to the set unit system.

8.2 Delete

Overview

Delete an element. It is possible to select the element to delete on the model tree or work screen and press the Delete key, but this function provides the following options.



Delete Affected Node

The selected element is deleted and the nodes that no longer have a connection with the element are also deleted.

Delete Empty Mesh Set

The empty mesh set that no longer has an element is deleted along with the selected element.

Duplicated Element only

Overlapping elements are different elements that have the same node and can be the source of analysis error. Changing the properties of an identical element can use the element property change boundary condition and so, it is better to delete the overlapping element created during modeling. This function searches for overlapping elements out of the selected elements and deletes all but 1 overlapping elements.

8.3 Modify

Overview

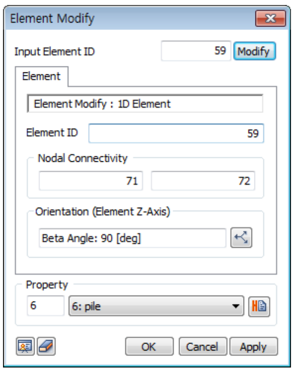
Edit the properties of the selected element. Select the element to edit on the work screen or use the ID select function to input the element ID. The element property modification can also be d1 through Element > Parameters.

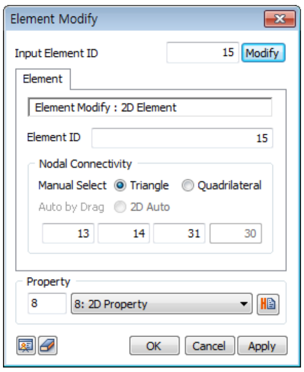
The input information needed for editing, depending on the selected element type, is automatically printed as shown below.

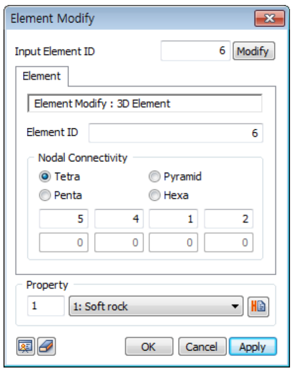
‣1D element

‣‣2D element

‣‣‣3D element





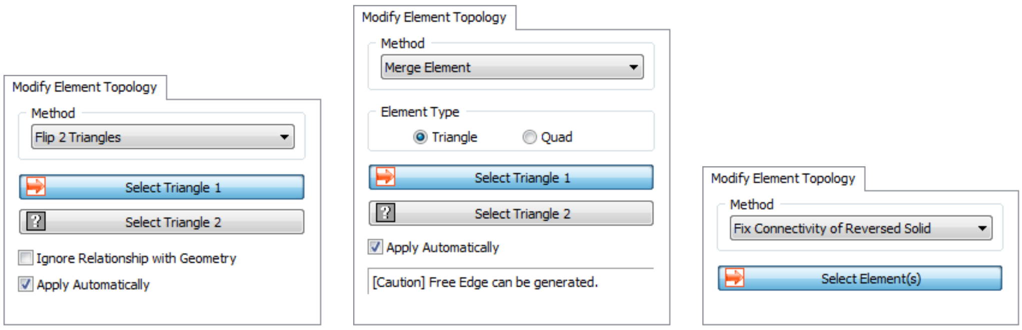


Edit Geometry through changing node information or change property. This function is useful when modifying details of the specific element.

8.4 Modify Topology

Overview

Modify the shape after element generation. There are 3 methods to choose from, depending on the element type and modification purpose.



Methodology

[Flip 2 Triangles]
Modify the shape of 2 2D triangle elements that share a side by selecting them consecutively. The nodes are kept. When modifying an element shape, the relationship with the geometry shape can be ignored, or the shape can be automatically renewed by setting the auto-apply option and selecting 2 elements.

[Merge Element]
Merge 2 connected triangles, or quadrilaterals, into 1 element by selecting them consecutively.
However, a B-Spline can form if the node connection between the adjacent elements is ignored.

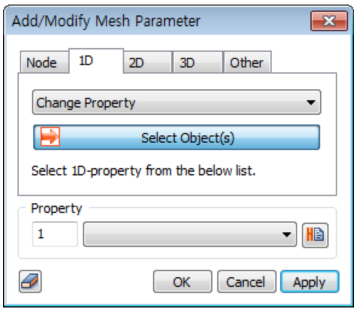
[Fix Connectivity of Reversed Solid]
An incorrect element can be formed during mesh 'import' or when the normal direction is defined in the opposite direction accidentally. In this case, this function modifies the node connections of a solid element. Use the [Front-Back Color] in View mode (mesh) to view the element shape on the screen and to check for elements in the opposite normal direction, indicated by the different color.

8.5 Parameters

1D

Overview

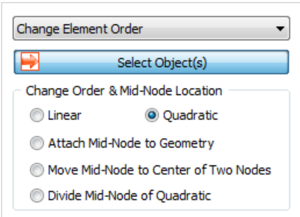
Change the properties, difference, coordinate system of a 1D element or add the offset distance or the end boundary conditions (fixed, hinge roller etc.) of a beam element. The addable and changeable items are listed below.



Methodology

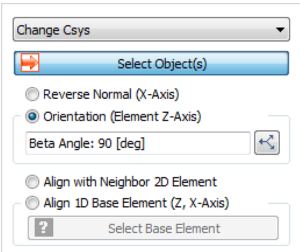
[Change Property]
Change the property assigned to a 1D element during its creation. Select the 1D element to modify and specify the property to change.

[Change Element Order]



Add or delete the nodes between a 1D element to change it into a higher order or lower order element. Changing the position of the middle node to the start or end point for high order element can create an arbitrary middle node on the geometry shape. A 1D high order element can be divided into 2 low order elements with reference to the middle node.

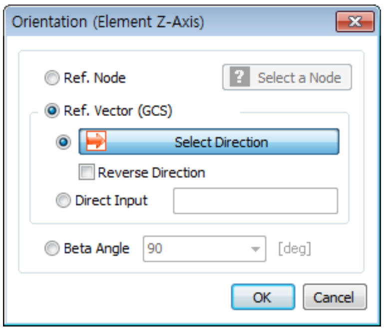
[Change Coordinate System]



The analysis results of a structural element are printed with reference to the element coordinate axis. Hence, it is important to check the coordinate system of the structural element to make sure it is in 1 direction. Switch the element x axis direction or specify the element y axis direction to unify the major and minor axis directions of the structural elements. The direction can be unified with adjacent 2D elements.

Orientation (Element Z-axis)

This function is used to unify the direction property of a 1D element to 1 direction or to set the major and minor axis directions. Adjust the Z axis direction by checking the element coordinate axis and assign with reference to the Beta angle.

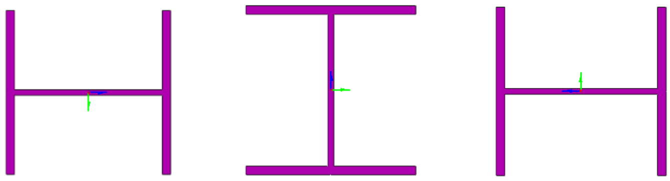


- Reference Node: Select the reference node for the sectional direction of the 1D element. The element Z coordinate direction is set with reference to the selected node.
- Reference Vector (GCS): Set the Z coordinate direction of the selected element using the GCS direction or the input vector direction.
- Beta Angle: Angles 0, 90,180 can be chosen and the selected Beta angle rotates the element by that angle with reference to the X axis.

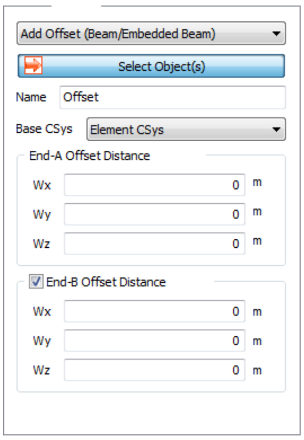
▸Beta angle:0

▸▸ Beta angle:90

▸▸▸ Beta angle:180



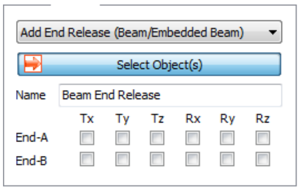
[Add Offset (Beam/Embedded Beam)]



Define the offset distance by considering the sectional properties (shape) of the structural element. When defining the properties, the offset can be set within the boundary of the section shape, or outside the boundary using the extra features of offset. The offset is the eccentric distance between the position of the geometry shape (line), needed to create a structural element, and the reference axis, where the loading is applied and the results are calculated. It is mostly used to express the connection between structural members or the combined section of 2 meeting members.

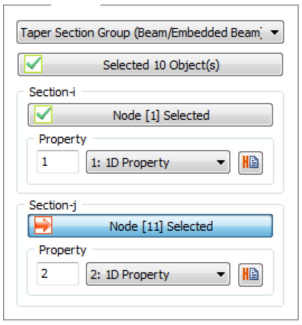
Specify the reference coordinates for the offset distance calculation and input uniform or non-uniform to set the offset distance for each axial direction.

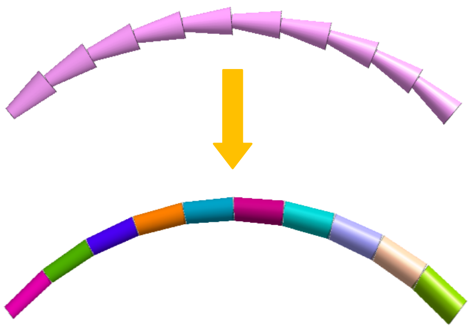
[Add End Release (Beam/Embedded Beam)]



Specify the end point boundary conditions for a 1D element. It is mostly applied when specifying the connection conditions between structural members such as hinge, roller etc.

[Taper Section Group (Beam/Embedded Beam)]



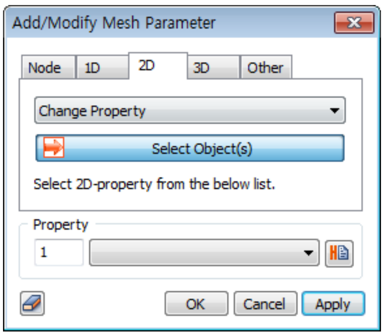


Members designated to a tapered section are grouped and calculate the section size automatically to define a constant tapered section regardless of the divided state of the elements. Firstly, select all elements that configure section changing area, and select the node at the beginning and the end of section changing area into the node of section i and j respectively. The tapered section is calculated by section property assigned to section i and j. The complex section changing area can be modeled quickly without creating the property of tapered section as the number of elements within section changing area.

2D

Overview

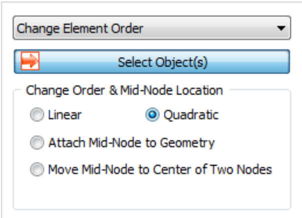
Change the properties, difference, coordinate system of a 2D element or add the offset distance or the end boundary conditions (fixed, hinge roller etc.) of a surface element. The addable and changeable items are listed below. The material coordinate system is defined as the result printing coordinate system for the 2D element and a separate print coordinate system (material coordinate system) can be added to certain selected elements.



Methodology

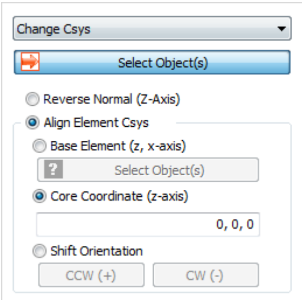
[Change Property]
Change the property assigned to a 2D element during its creation. Select the 2D element to modify and specify the property to change.

[Change Element Order]



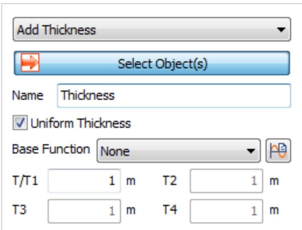
Add or delete nodes between a 2D element to change it into a higher order or lower order element. Changing the position of the middle node to the start or end point for high order element can create an arbitrary middle node on the geometry shape.

[Change Coordinate System]



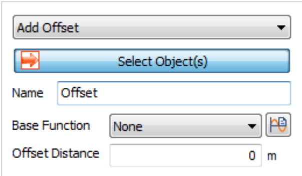
For a 2D structural element, the print coordinate system (material coordinate system) can be defined separately. Here, if the print coordinate system (material coordinate system) is set as the element coordinate system, it is important to check the coordinate system of the structural element to make sure it is in 1 direction. Switch the element z axis direction or unify the normal directions of the selected elements to the normal direction of the reference element. When using centered coordinates, use the direction vector from the reference point to each element to modify the normal direction of each element.

[Add Thickness]



The thickness of a 2D element is defined in the section properties, but an additional thickness can be specified for selected elements. Select the 2D element and change the thickness by specifying the thickness for each element node.

[Add Offset]



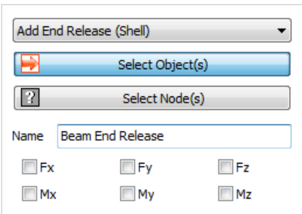
Define the offset distance by considering the sectional properties (shape) of the structural element. The offset is the eccentric distance between the positions of the geometry shape (surface), needed to create a structural element, and the reference axis, where the loading is applied and the results are calculated. The distance can be specified by a function on the GCS and the direction moves to the normal direction (+,-) of the 2D element. When applying the function, the input offset distance becomes the scale factor that is multiplied to the function for calculation.

[Add Material Orientation]

Apart from the material coordinate system specified on the 2D element property, a separate material coordinate system (print coordinate system) can be additionally set for selected elements. The coordinate system can be defined as follows.

- Coordinate System: Specify the material x direction in the X,Y,Z direction of the GCS. Both the Global rectangular coordinate system and Global cylindrical coordinate system can be used.
- Angle: Specify the material x direction by setting the normal direction of the element as the revolution axis and rotating the coordinate system by the specified angle.
- Reference Vector: Input or use the selected space vector direction to specify the material X direction.
- Coordinate system and Angle: Specify the material X direction as the rotated direction of the reference coordinate axis on the selected coordinate plane.

[Add End Release (Shell)]

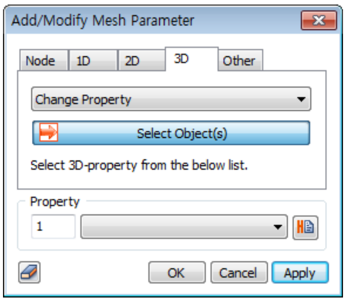


Specify the selected node boundary conditions for a 2D element. The axial direction and rotation conditions can be released for each element.

3D

Overview

Change the properties and difference of a 3D element.

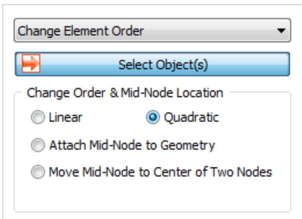


Methodology

[Change Property]

Change the property assigned to a 3D element during its creation. Select the 3D element to modify and specify the property to change.

[Change Element Order]

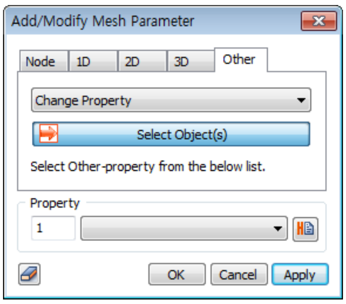


Add or delete nodes between a 3D element to change it into a higher order or lower order element. Changing the position of the middle node to the start or end point for high order element can create an arbitrary middle node on the geometry shape.

Other

Overview

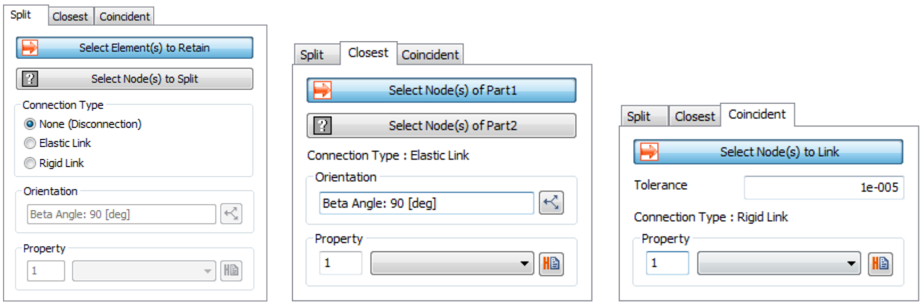
Use to change the assigned other properties of an element. It is applied to spring, link, interface affiliated elements and select the element to define a fitting property.



8.6 Connection

Overview

Cut the connection between shared nodes or create a spring or rigid link element between nodes. The free edge/faces created by the node separation can be used to create an interface element.



Methodology

[Split]
Select the element that contains the node which will be separated. Then, select the nodes to divide or create a link element on. The connection type can be chosen between only node separation, create elastic link after node separation or create rigid link. When creating an elastic link, specify the coordinate system of the link element.

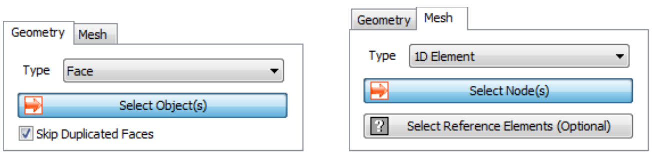
[Closest]
Select 2 groups of nodes and create an elastic link between the 2 closest nodes between the groups.

[Coincident]
Create a rigid link between selected nodes that are within the tolerance range. The tolerance is the allowable limit of the node connection, and the operation is only applied when the distance between the nodes is smaller than the tolerance. It is used to automatically connect 2 barely separated nodes.

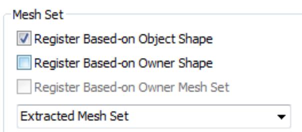
8.7 Extract

Overview

Extract a sub-element from an existing mesh on a geometry shape or a mesh set. The extracted mesh is connected to the existing meshes by nodes and is often used to create structural elements that are connected to the neighboring ground by nodes. During extract, the element properties can be defined or added and for 1D elements, the element coordinate system can be specified.



The extracted mesh can be divided by target shape, possessed shape or mesh to create a mesh set. This option automatically divides the mesh sets when extracting elements from multiple solids simultaneously.



Methodology

[Geometry]
Extract 1D/2D elements that have kept their node position from the edge/surface used for mesh creation. The [Skip Duplicated Faces] option can be applied when extracting a 2D element from a surface. This option extracts the mesh from only the non-overlapping surfaces of the selection. If the selected target surface lies on multiple solids in a complex manner, select all the solids and use [Skip Duplicated Faces] to extrude only the outermost surface of the solid.

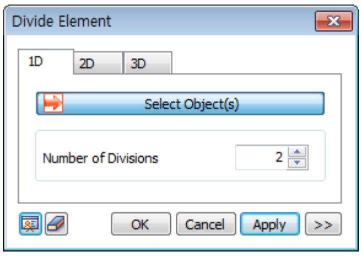
[Mesh]
Extrude a 1D/2D element from the free edge/surface of an existing mesh. Only the selected nodes on the free edge/surface are extruded automatically and selecting the reference mesh extrudes only the nodes on the specified mesh.

8.8 Divide

1D

Overview

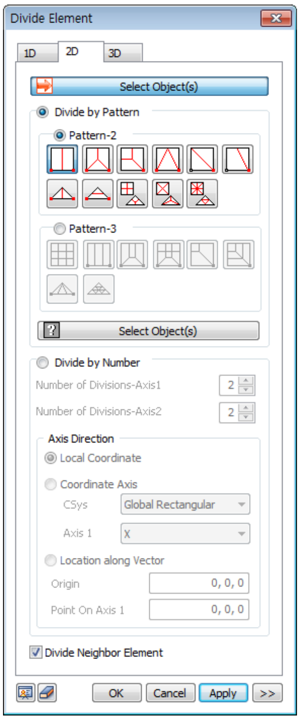
Divide the 1D element by the set number. It is used to divide a 1D high order element into 2 low order elements.



2D

Overview

Divide a 2D element into multiple elements using a pattern or entering the number of divisions. If the geometry shape is complex, the shape (quality) of the equal size mesh created by the auto-mesh function may not be good. In this case, use the divide function to increate the Aspect ratio. Set the divide pattern or the number of divisions in the axial direction. Use the [Divide Neighbor Element] option to maintain the node connections with adjacent elements during element division.



Methodology

[Divide by Pattern]
Specify the 2D element and pattern. Then, select the reference node.

Pattern -2: Use 1 of the 11 patterns to divide the element outline into 2 segments. The red point that appears in the pattern indicates the position of the reference node. The white points indicate the points that ignore the reference node.

Pattern -3: Use 1 of the 8 patterns to divide the element outline into three segments. The red point that appears in the pattern indicates the position of the reference node.

[Divide by Number]
Evenly divide the 2D element by the input number of divisions. However, it can only be applied to rectangular elements that are defined in 2 axial directions. Set the number of divisions in the axis 1, 2 direction and define the axial direction as follows. Input the number of divisions in the axial direction of an element edge that has makes up the smallest angle between the specified axis directions.

Axis Direction

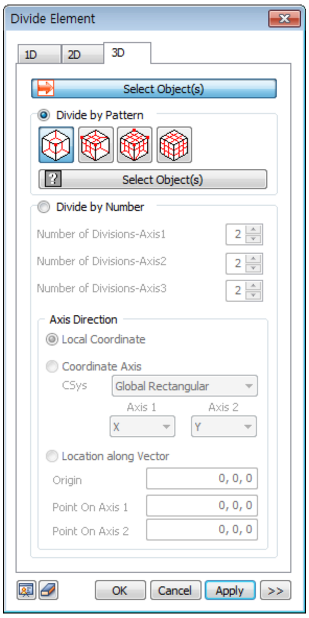
[Local Coordinate]
Use axis 1 as the x axis direction of the element coordinate system.

[Coordinate Axis]
Use the coordinate system to specify axis 1. The axis can be select from the Global rectangular system or the Global cylindrical system.

[Location along Vector]

Use the input vector as axis 1. Input the start point and another arbitrary point on axis 1 to create a vector.

3D element



Overview

Divide a 3D element into multiple elements using a pattern or entering the number of divisions. Only hexahedral elements can be divided.

Methodology

[Divide by Pattern]
Use 1 of the 4 preset patterns to divide the element. The red point that appears in the pattern indicates the position of the reference node. Specify the 3D element and pattern and select the reference node.

[Divide by Number]
Evenly divide the 3D element by the input number of divisions. Set the number of divisions in the axis 1,2,3 direction and define the axial direction as follows. Input the number of divisions in the axial direction of an element edge that has makes up the smallest angle between the specified axis direction.

Axial Direction
[Local Coordinate]
Use axis 1 as the x axis direction of the element coordinate system.

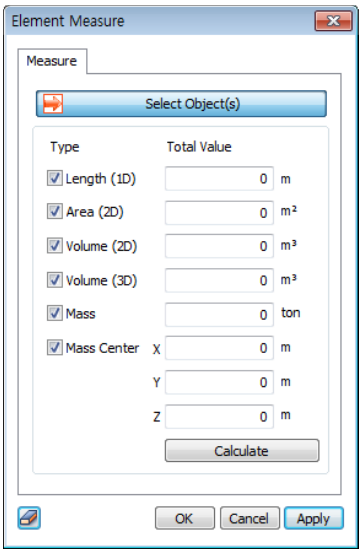
[Coordinate Axis]
Use the coordinate system to specify axis 1 and axis 2. The axis can be select from the Global rectangular system or the Global cylindrical system.

[Location along Vector]
Use 2 input vectors as axis 1 and axis 2. Input the start point and any other arbitrary points on axis 1 and axis 2 to create 2 vectors.

8.9 Measure

Overview

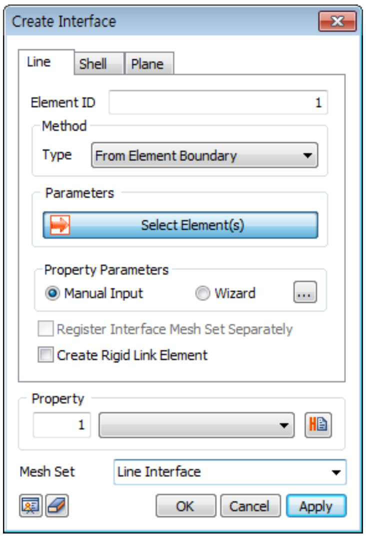
Measure the material and geometric information of a selected element. It is useful to check the Length (1D), Area (2D) and Volume (3D) of an element. Select the element to be measured and use calculate to check the total sum of all selected elements. For Volume (2D), use the property (thickness) information of a 2D element and the element area to calculate the hypothetical volume.



8.10 Interface

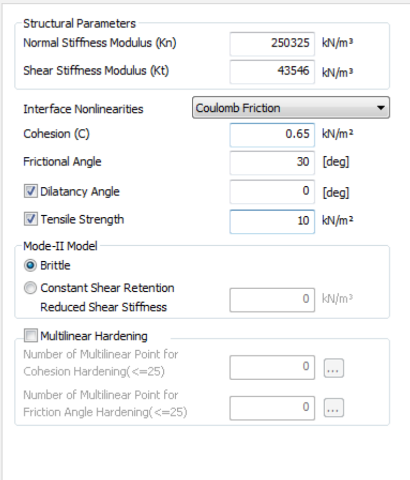
Overview

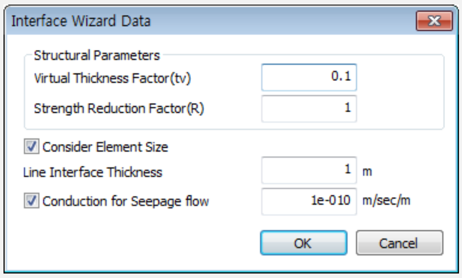
Create a homogeneous or heterogeneous material interface where slip or isolation can happen. It is used to simulate the interface behavior between ground and structural members with a large relative stiffness difference. The interface generation method is classified by the work environment (2D/3D) and the target. The properties must be defined in order to create an interface element. The properties can be directly entered, or can be automatically calculated from the properties of adjacent elements using Wizard.



Tip

Interface element generation immediately separates the connected nodes of that position and creates an element with specific stiffness in the normal and tangent directions. Hence, because the nodes are left separated for certain steps where the interface element is not used (ex. Initial in-situ ground), the nodes need to be connected by a rigid link to prevent analysis error. Reversely, when the interface element is used, the rigid links need to be removed during analysis. Use the [Create Rigid Link Element] option to automatically create a Link element connecting 2 nodes.





The interface material can be defined using the following equation. Using the stiffness of adjacent elements and nonlinear parameters, the virtual thickness (tv) and strength reduction factor (R) is applied. Interface material stiffness and parameters are applied differently according to the relative stiffness difference between neighboring ground or structural members. The Wizard can be used to simplify this process.

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

$K_t = G_i/t_v$

$C_i = R \times C_{soil}$

$phi_i = \tan^{-1} (R \times \tan (phi_{soil}))$

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

(v_i =Interface Poisson's ration=0.45, the interface is used to simulate the non-compressive frictional behavior and automatically calculates using 0.45 to prevent numerical errors.)

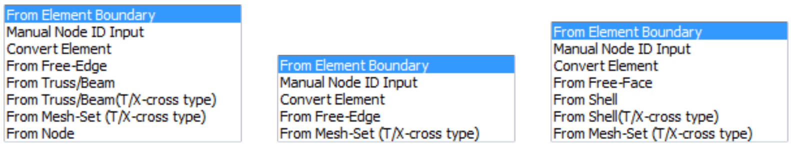
t_v = Virtual thickness(Generally has a value between 0.01~0.1, the higher the stiffness difference between ground and structure, the smaller the value)

$G_i = R \times G_{soil}$ ($G_{soil} = E/(2(1+ v_{soil}))$), R = Strength Reduction Factor

Methodology

The interface can be created using the following methods, depending on the work environment and comp1nts or shape.

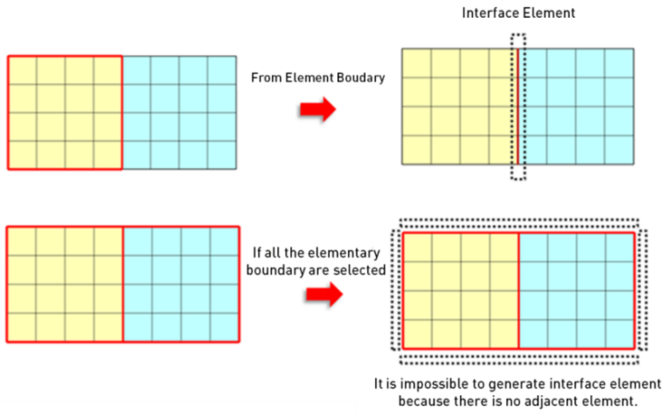
▸Line Interface
▸▸Shell Interface
▸▸▸Plane Interface



- From Element Boundary

Create an interface element at the boundary position between the selected element and adjacent element, as shown in the figure below. Selecting all elements cannot create an interface element because there are not adjacent elements. If the created interface element is within a mesh, the interface element has a wedge shape, as shown.

▸From element boundary



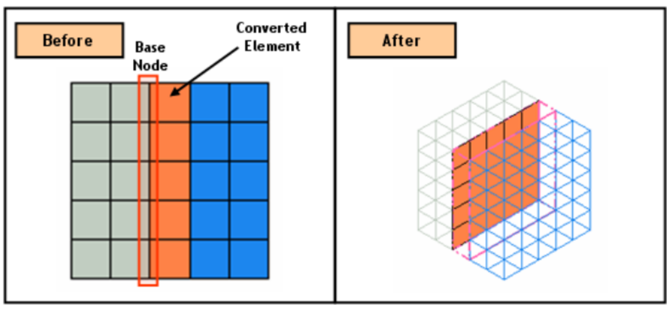
- Manual Node ID Input

Directly input the node ID to create an interface element. The nodes are divided into 2 groups: Side 1 and Side 2 that determine the shape of the interface element. The number of input nodes on Side1 and Side2 must be the same and the shape for each size is as follows.

- Convert Element

Convert a general 1D, 2D, 3D element into an interface element. Because general elements do not have a consistent node order, the base reference node needs to be selected additionally.

▸Convert element



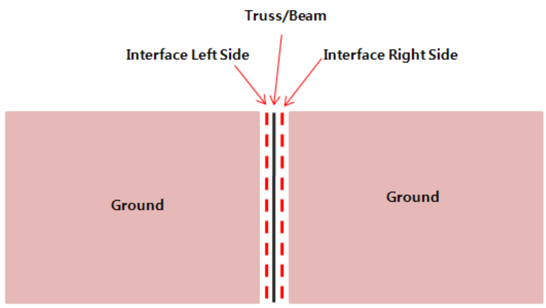
- From Free-Edge

Create an interface element from opposing free surface/edge. The free edges (free surface for 3D) with no node connections need to be selected on each side and if the free edge (free surface) does not exist, use the Connection > Divide function to divide beforehand.

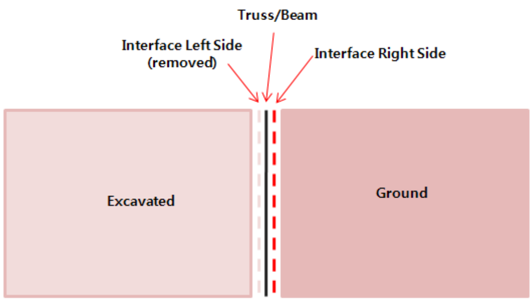
- From Truss/Beam (Use plate element)

Create an interface element using the truss/beam element. For 3D, use a plate element. Creating an interface element for structural elements such as truss/beam/plate generates an interface element on both sides of the element. Hence if the interface creation method is truss/beam, set the “add mesh set separately” option for the interface element to separately create the interface element in each direction.

▸Use structural element



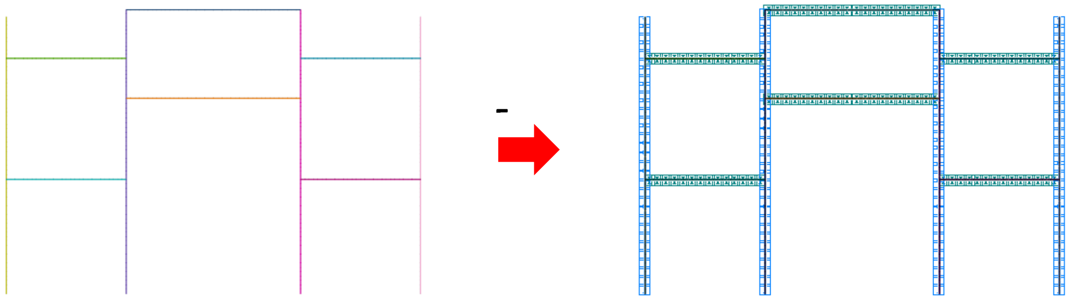
Also, if the ground element connected to the interface element is removed during construction, the interface element is also removed to prevent analysis errors.



- From Truss/Beam (T/X-cross type)

The interface elements are created at the location where truss/beam elements cross T or X-shape. Shell elements can be selected in 3D model. The ‘Register Interface Mesh Set Separately’ option isn’t available since interface elements are T or X-shape.

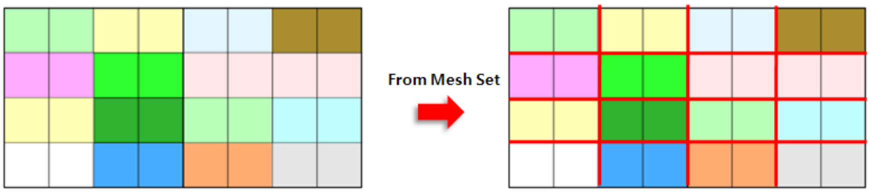
From Truss/Beam (T/X-cross type)



- From Mesh-Set (T/X-cross type)

  Create an interface element at the T- or X-cross of the selected mesh set. This method can be used when the interface elements intersect, such as in masonry structures.

From Mesh-Set (T/X-cross type)



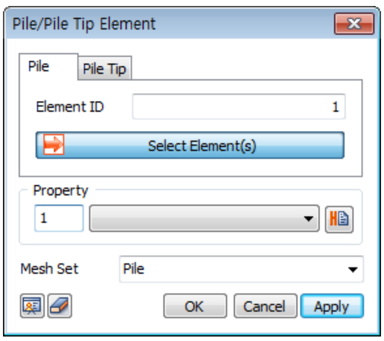
- From Node

  Create an interface element between adjacent elements, using the relationship between the selected nodes. For 3D, select a segment point if the interface element is not created properly.

8.11 Pile/Pile tip

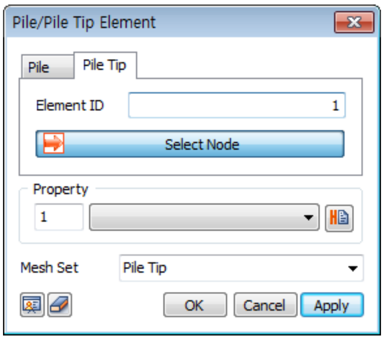
Pile

Piles are an interface affiliated element in embedded beam element form that does not require node connections with the neighboring ground. It is used to check the relative displacement and friction behavior between the beam element and the ground. Create a beam element and neighboring ground element and then select the beam element to create a pile element.



Pile tip

The pile tip element is added when defining the relative behavior between the ground element and 1 end node of a pile. Create a pile element. Then, select the tip. The stiffness of the pile tip is defined by the end bearing capacity and spring constant.

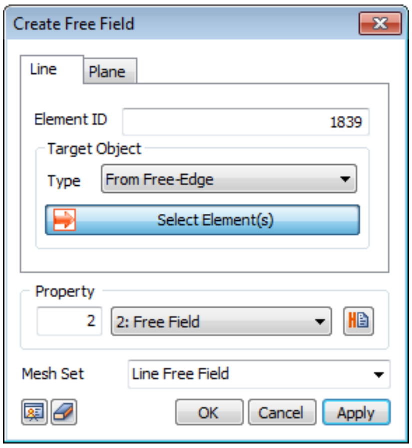


8.12 Free Field Element (Infinite Element for Dynamic Analysis)

Overview

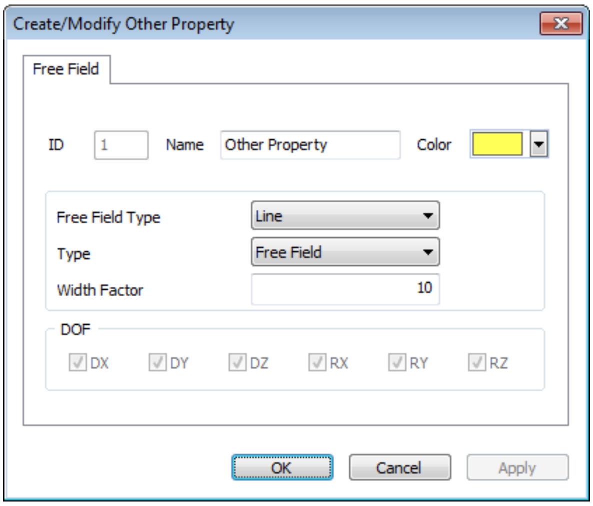
For the seismic analysis, users need to model infinite ground to eliminate the boundary effect caused by reflection wave. Since it is not possible to model infinite ground, users can apply Free Field Element at the boundary.

Free Field Element enables to apply traction resulted from Free Field Analysis to the ground boundary and then, eliminate reflection wave using absorbent boundary condition.



Methodology

Select free edges in 2D and free faces in 3D to define Free Field Elements.



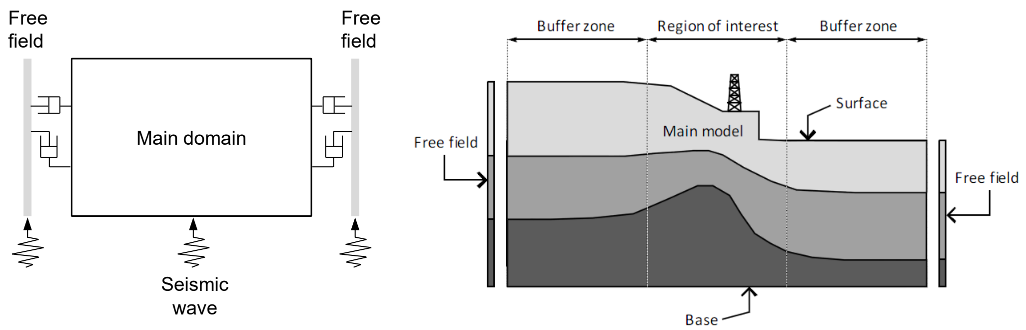
- **Free Field**
  Enable to simulate infinite ground boundary.

- **Absorbent Boundary**
  Enable to eliminate reflection wave at the ground boundary.

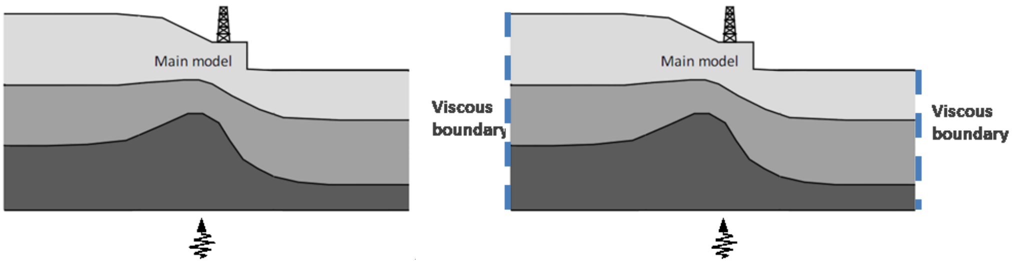
- **Width Factor (Penalty Parameter)**
  In order to minimize the size effect of the model, users have to input more than 104, This value will be multiplied by model width (In case of 2D, this is plain strain thickness (unit width).

- **DOF (Degree Of Freedom for damping)**
  Users can select specified DOF for damping effect.

‣Schematic Overview of Free Field Element
‣‣Free field effect (O), Absorb reflection (O)



‣Free field effect (X), Absorb reflection (X)
‣‣Free field effect (X), Absorb reflection (O)

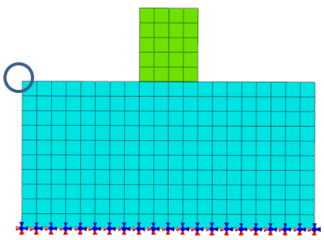


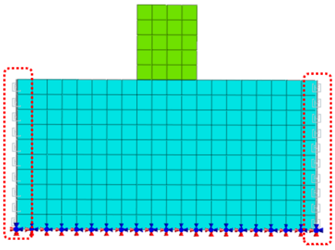
- **Verification Example**
  Free field element can result in identical behavior with infinite ground model.

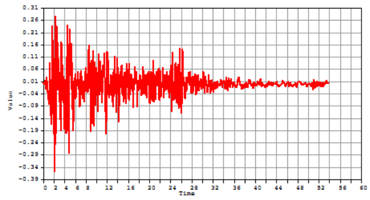
‣None

‣‣Free field

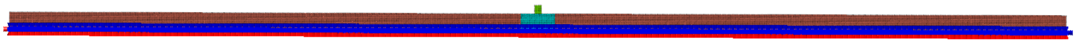
▶▶▶Ground



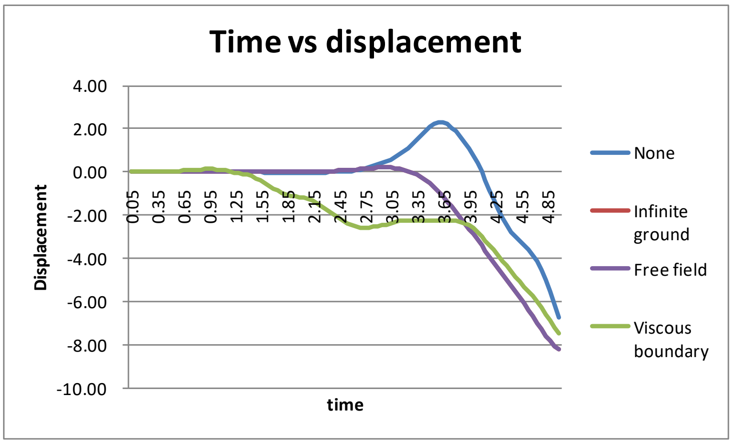




‣Infinite ground



‣Displacement results for each case

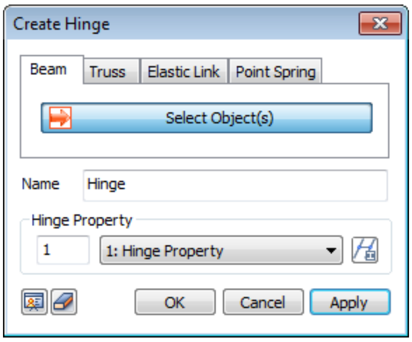


8.13 Hinge

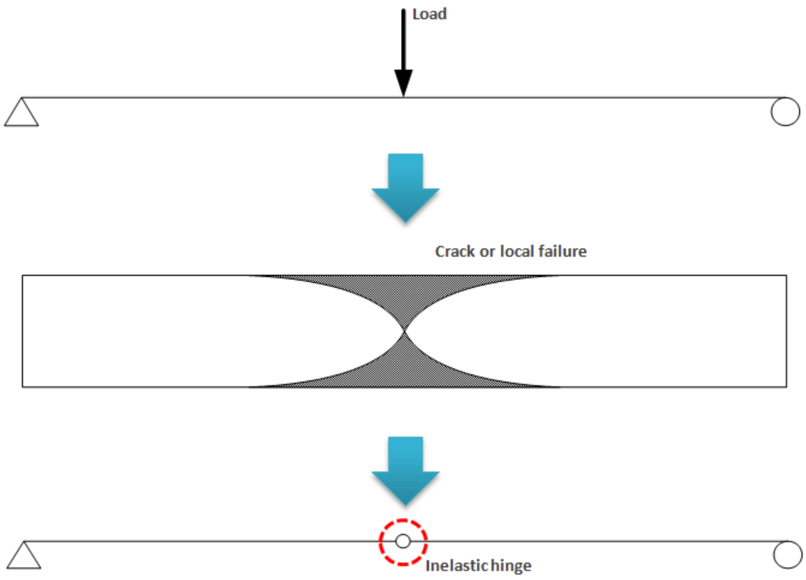
Methodology

- **Inelastic Hinge**
  Inelastic hinge can be applied to the structural elements to simulate crack or local (plastic) failure. Applicable in Nonlinear Static and Time History Analysis as follows : Nonlinear, Construction Stage, Consolidation, Fully Coupled, SRM (Slope Stability).
  Following properties are available to define inelastic hinge : **Beam, Truss, Elastic Link and Point Spring**

‣Hinge Properties



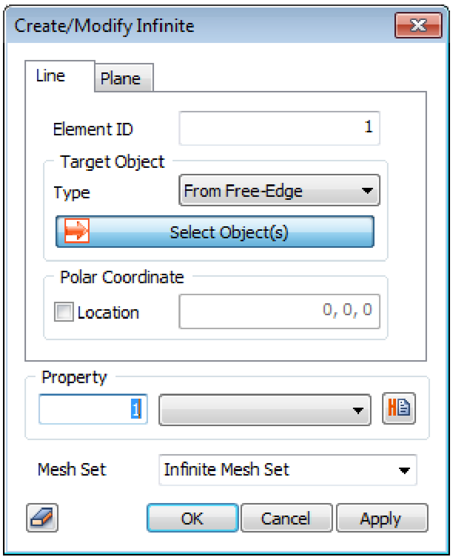
▸Schematic overview of Inelastic Hinge



8.14 Infinite Element

Overview

Without modeling all region, a model can be reduced considering the area of interests by assigning infinite element on the boundary which does not have significant effects on the result. All analysis types are applicable except dynamic analysis. (Free field element is used in dynamic analysis.)
It isn't possible to use all kind of load & boundary condition (Self Weight, Constraint, Nodal Head, Review, etc.) in the node or element of infinite element.

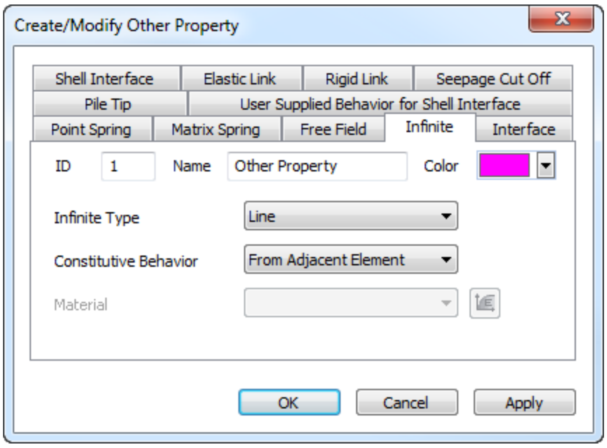


Methodology

It is divided into 'Line, and 'Plane' tab according to the type. For the adjacent element with infinite element, analysis is only for Quad / Wedge / Hexa typed Low/High order element with 'Plane Strain / Axisymmetric / Solid' property.

- **Target Object Type**
  Infinite element can be created by selecting 2D element, ‘Free Edge’ in the case of 'Line' type. Infinite element can be created by selecting 3D element, ‘Free Face’ in the case of 'Plane' type.

- **Property**
  Select Line/Plane type infinite element



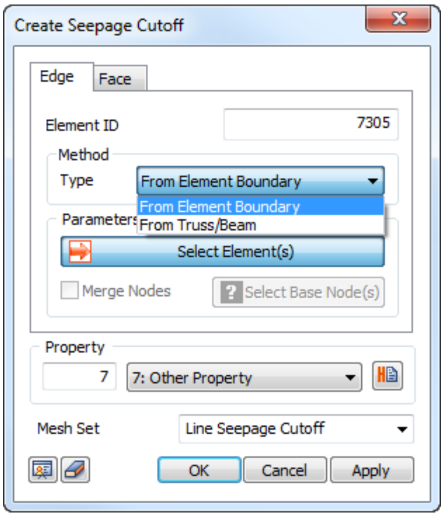
- **Infinite Type**
  Select infinite element type. It is divided into ‘Line’ type and ‘Plane’ type for 2D model and 3D model respectively.

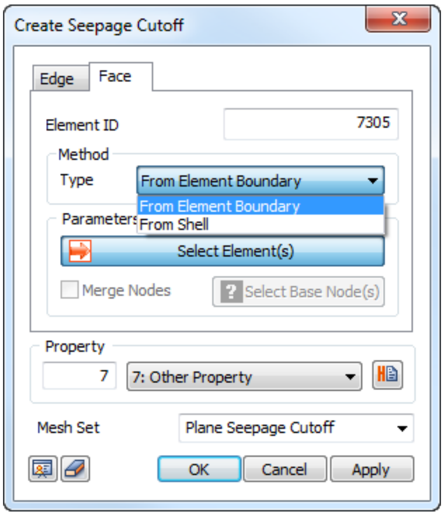
- **Constitutive Behavior**
  Select Constitutive Behavior of infinite element. It is divided into ‘From Adjacent Element’ type which adopt material property of adjacent element and ‘User Defined’ type which can assign other material property for only infinite element. ‘Material’ is directly inputted for only 'User Defined' type.

8.15 Seepage Cut Off Element

Overview

The Seepage Cut Off (SCO) element can be used to simulate the behavior of structural waterproof members. Main role of the element is to control seepage DOF on the line or face.





Methodology

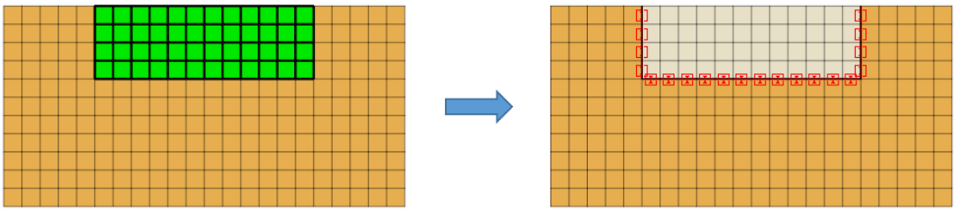
In 2D analysis, select a line element (From Element Boundary/Truss/Beam element) to define the impermeability.

In 3D analysis, the impermeability can be defined by selecting the face element (From Element Boundary/Shell element).

▸From element boundary

- From Element Boundary

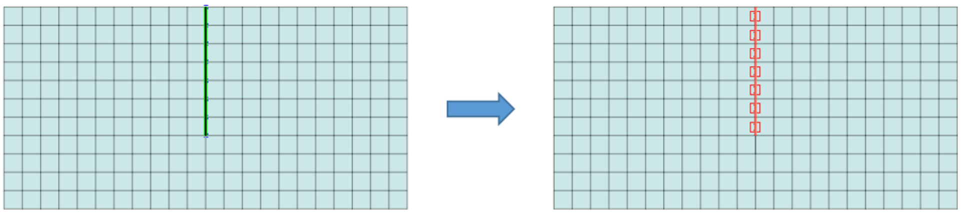
  Create an SCO element at the boundary position between the selected element and adjacent element, as shown in the figure below. Selecting all elements cannot create an SCO element because there are not adjacent elements.

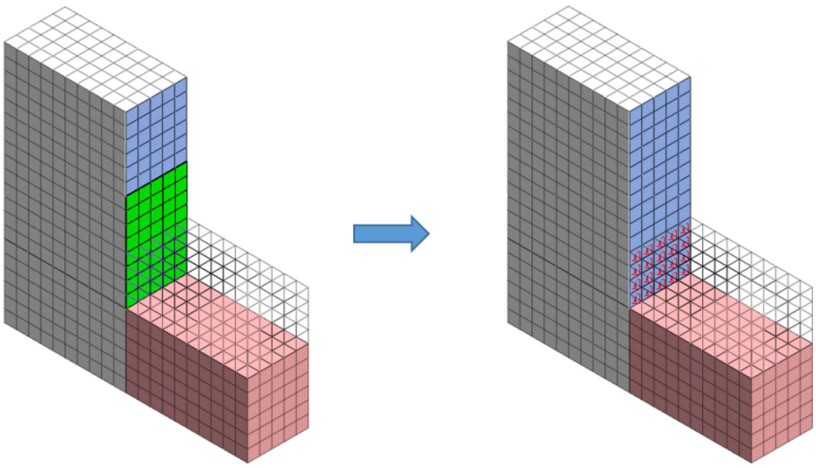


▸Use structural element

- From Truss/Beam (for 3D case - Shell element)

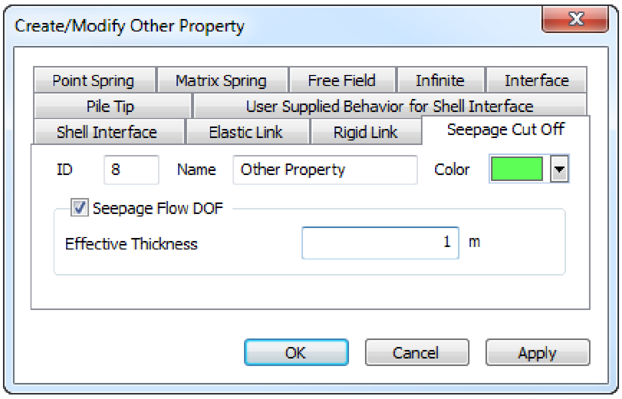
  Create an SCO element using the truss/beam element. For 3D, use a plate element. Creating a seepage cut off element for structural elements such as truss/beam/shell generates “impermeable interface” element on both sides of the element.





- **Property**

  Checking the Seepage Flow DOF option allows for control the flow rate/impermeability characteristics. (※ When checked: consider the effect of seepage, when not checked: consider the impermeability effect)



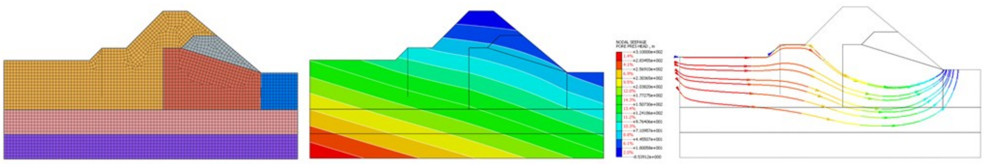
The effective thickness used in Seepage Flow DOF is the effective thickness for the imaginary size used for the calculation of the seepage flow and it defines the width of the structural member.

▸Schematic overview of Seepage Cut Off element results

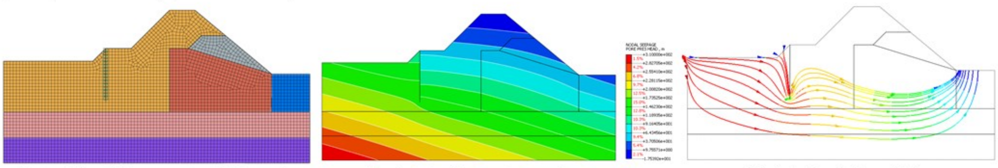
- **Results comparison**

– 2D analysis case (via Truss/Beam element type).

–

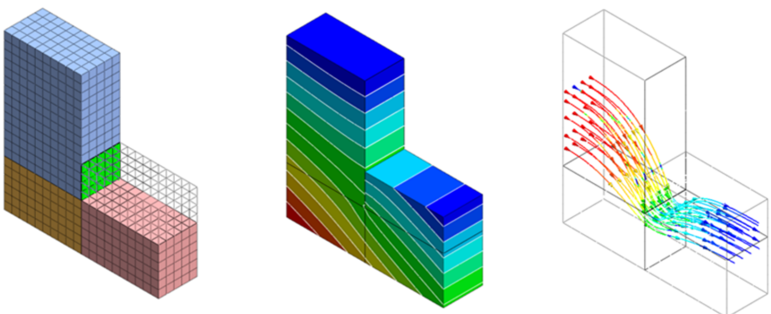


[2D Model - without Seepage Cut Off element] [Analysis Results-Pore Pressure] [Analysis Results-Flow Net]



[2D Model - with Seepage Cut Off element] [Analysis Results-Pore Pressure] [Analysis Results-Flow Net]

– 3D analysis case (via Shell element type).



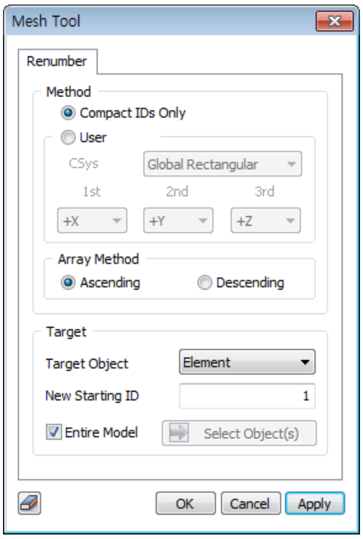
[3D Model with Seepage Cut Off element] [Analysis Results-Pore Pressure] [Analysis Results-Flow Net]

Section 9 Tools

9.1 Renumber

Overview

When mesh and node numbers are recomposed, results can be checked in an organized manner.



Methodology

Select the target nodes and elements and input the starting number. The organization can be set for the whole model, or only for the selected nodes (elements). Use the [Compact IDs Only] option to organize the node number continuously starting from 1, and the coordinate system priority for the organization reference can be selected from User defined. Assign the minimum number with reference to the first axis. The numbers are then organized by the second axis within the range that does not affect the first axis. The numbers are organized again by the third axis within the range that does not affect the first and second axes.

For example, if the first is X, second is Y, third is Z, then the organization is first d1 by considering the X coordinates specified by the first priority. The number increases from the small X coordinate to the large X coordinate, and if the X coordinates are the same, then the next priority Y coordinates are considered. For elements, XYZ coordinates of the center of gravity is used for organization.

The number order can be selected from Ascending/Descending order.

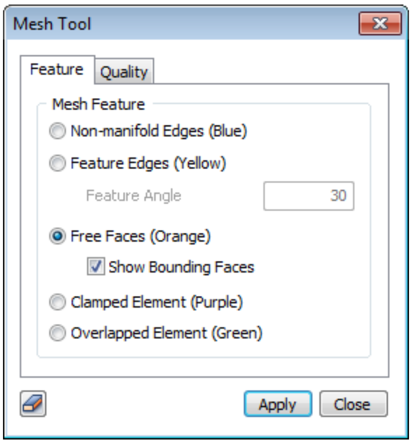
9.2 Check

Mesh Topology

Overview

Analyze the mesh information and sort the mesh depending on classification for easy distinction. Apply the check mesh menu to view all existing meshes regardless of the view/hide status to display all calculated free lines, free surfaces, and etc. on the screen.

When checking a free surface, the mesh is displayed as a wireframe and it may be difficult to check for free surfaces existing within the interior of the model. In this case, the mesh check is performed with all target meshes hidden.



Methodology

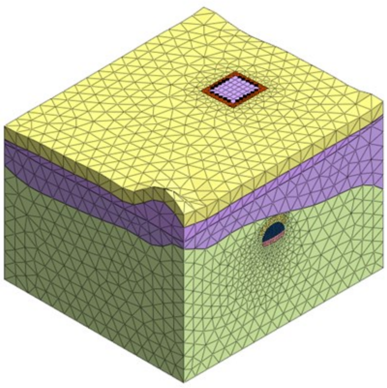
[Non-Manifold Edges (Blue)]
Check Non-Manifold edges in blue. A Non-Manifold edge is formed at the meeting edge of three or more element surfaces.

[Feature Edge (Yellow)]
Check the feature edge in yellow. A feature edge is the line that has a sudden change in the model shape, and has an angle larger than the feature division angle between 2 element surfaces.

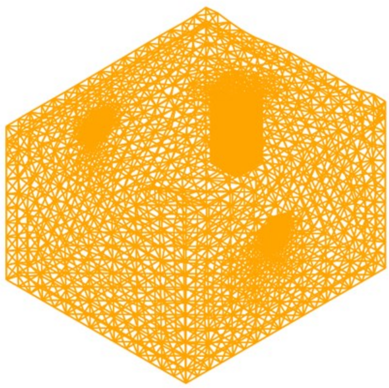
- Feature Angle
  Input the feature angle to calculate the feature edge.

[Free Faces (Orange)]
Check the free surface in orange. The free surface is the section where the nodes between 3D elements are not connected.

- Show Bounding Faces
  This is an option to show/hide the boundary faces among free faces. It is useful to check free faces located inside in case of 3D complex model.



[Model]



[Check on the option]

[Check off the option]

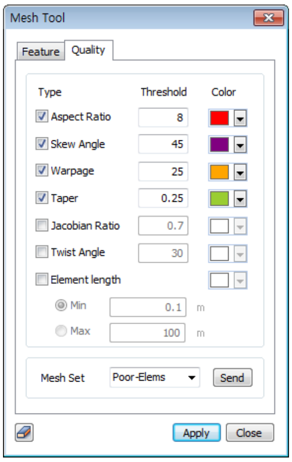
[Clamped Element (Purple)]
Check the clmped element in purple. Search for elements with all nodes on the free surface in each 3D element. If the boundary condition is all applied on the free surface of the mesh, the confined element does not affect the analysis.

[Overlapped Element (Green)]
Check overlapped elements that have the same node information at the same position. The elements are highlighted in green, and the Element > Delete function can be used to delete the overlapping element.

Mesh Quality

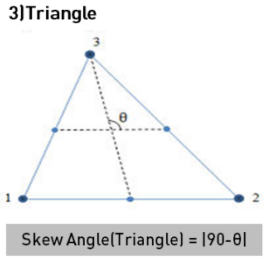
Overview

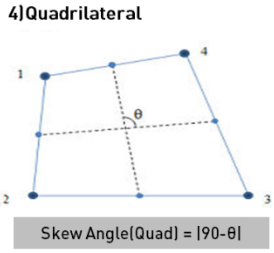
The relative size between connected elements and the shape and quality of a mesh have a larger effect on the analysis results than its absolute size. Hence, after creating a mesh, it is important to check and modify the quality of the mesh, especially for 3D elements. Input the reference value to color the elements that do not meet this standard.



[Aspect Ratio]
Length ratio between the width and length, or the ratio of the longest side to the shortest side of a 2D element. For example, a square has the same width and length and therefore has an aspect ratio of 1. As a shape digresses from the square shape, the aspect ratio becomes smaller. A value closer to 1 is ideal. This ratio has significant effect on the analysis result and if the value is very small, it may be hard to obtain normal analysis results.

[Skew Angle]
How much the shape digresses from the rectangular shape (90 degrees), measured in angles. A quadrilateral forms a 90 degree angle, the inclination angle is 0 degrees and this value increases as the shape strays from the quadrilateral. For a solid element, the inclination angle is checked for each face and the smallest value is chosen as the inclination angle. A value closer to 0 is better.



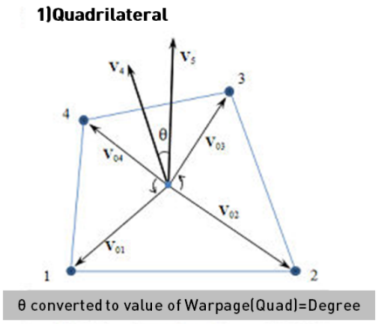


3)Tetrahedron
Max value for each of the 4 faces

4)Pentahedron
Max value for each of the 5 faces

5)Hexahedron
Max value for each of the 6 faces

[Warpage]
Evaluates how much the shape is out of the plane. For a quadrilateral 2D element with all nodes on the same plane, the value is 0. The value increases as the shape strays from the plane. For a solid element, the warpage is checked for each rectangular face and the smallest value is chosen as the warpage value. A value closer to 0 is better. This item has significant effect on the analysis result and if the value is very large, it may be hard to obtain normal analysis results.



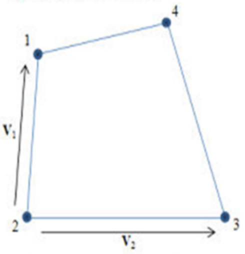
$$V_4 = \frac{V_{04} + V_{01}}{\|V_{04} + V_{03}\|}$$
$$V_5 = \frac{V_{02} + V_{02}}{\|V_{02} + V_{02}\|}$$
$$V_4 \cdot V_5 = \|V_4\| \cdot \|V_5\| \cos\theta$$
$$\theta = \cos^{-1}(V_4 \cdot V_5) \quad (\|V_4\| \cdot \|V_5\| = 1)$$

2)Pentahedron
Max value for each of the 3 side faces

3)Hexahedron
Max value for each of the 6 faces

[Taper]
Geometrically calculates how much the quadrilateral digresses. It is not applied to triangular elements. A quadrilateral has a value of 1, and the value decreases as it digresses (becomes closer to a triangular shape) from its rectangular shape. For a solid element, the taper value is checked for each face and the smallest value is chosen as the taper value. A value closer to 1 is better.

1)Quadrilateral



$$Taper(Quad) = \frac{J_{max} + J_{avg}}{J_{avg}}$$

$$V_3 = V_1 \times V_2$$
$$J_j = \frac{\|V_2\|}{4} \quad (i = 1,2,3,4)$$
$$J_{max} = \max(J_j)$$
$$J_{avg} = (J_1 + J_2 + J_3 + J_4)/4$$

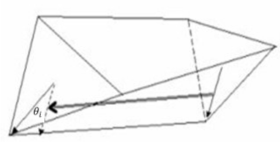
2)Pentahedron(Wedge)
Max value for each of the 3 side faces

3)Hexahedron
Max value for each of the 6 faces

[Jacobian Ratio]
The Jacobian determinant is calculated at each Gauss integral point on the mesh the Jacobian ratio is the ratio between the largest and smallest Jacobian determinant value. For 2D elements, the Jacobian determinant is calculated on the element projected onto a plane. For solid elements, the Jacobian determinant is calculated directly. If the quadrilateral element is not convex, the negative value is outputted and the analysis is not performed properly. A higher value is better.

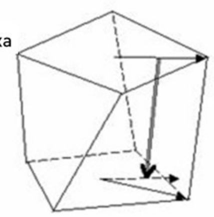
[Twist(Solid)]
Value that represents the twist between 2 opposing faces in a solid.

Wedge



$$Twist = 1 - \frac{Max(\theta_i)}{120°}$$

Hexa



$$Twist = 1 - \frac{Max(\theta_i)}{90°}$$

[Element length]
Check the edge length of an element. Here, the minimum and maximum values can be set.

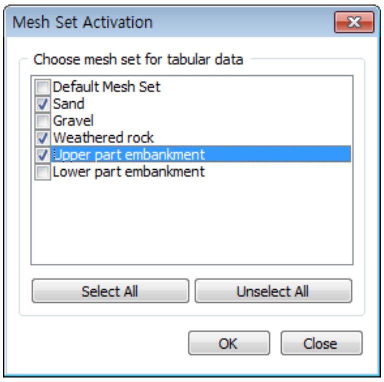
Mesh set

After the quality check has finished, press ‘Send’ to define a different mesh set. The user can specify the name of the mesh set.

9.3 Table

Overview

The node and mesh information for all mesh sets in a model can be printed onto a table. Activate all created mesh sets and select mesh set to extract the overlapping nodes and element information from.



Node table

Selecting an element prints the coordinate information of all nodes in the selected element. The model information can be modified on the table through Add, Edit, Delete.

	No.	CSys	X (m)	Y (m)	Z (m)
▶	58054		-13.400000	21.500000	17.500000
	58055	1:Global Rectangular	-13.400000	31.500000	17.500000
	58056	2:Global Cylindrical	-14.900000	21.500000	17.500000
	58057	1:Global Rectangular	-14.150000	21.500000	17.500000
	58058	1:Global Rectangular	-14.900000	31.500000	17.500000
	58059	1:Global Rectangular	-14.150000	31.500000	17.500000
	58060	1:Global Rectangular	-13.400000	29.500000	18.650000
	58061	1:Global Rectangular	-13.400000	27.500000	18.650000
	58062	1:Global Rectangular	-12.500000	30.500000	20.700000
	58063	1:Global Rectangular	-12.500000	31.500000	21.500000
	58064	1:Global Rectangular	-12.500000	30.500000	21.500000

Element table

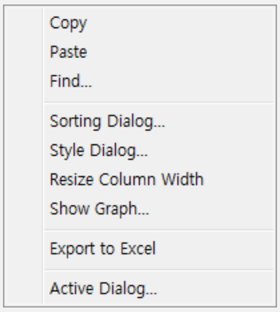
Selecting an element prints the element information divided into element type tabs. The node connections and node number can be checked and the element can be edited on the table through Add, Edit, Delete.

	No.	Type	Property	Node1	Node2	Node3	Node4	Node5	Node6	Node7	Node8	Node9	Node10
▶	84759	Hexahedron	5:Concrete	58159	58236	58218	58168	58160	58254	58227	58167		
	84760	Hexahedron	5:Concrete	58228	58173	58174	58210	58238	58172	58175	58219		
	84761	Hexahedron	5:Concrete	58236	58235	58217	58218	58254	58252	58226	58227		
	84762	Hexahedron	5:Concrete	58229	58228	58210	58211	58240	58238	58219	58220		
	84763	Hexahedron	5:Concrete	58235	58234	58216	58217	58252	58250	58225	58226		
	84764	Hexahedron	5:Concrete	58230	58229	58211	58212	58242	58240	58220	58221		
	84765	Hexahedron	5:Concrete	58234	58233	58215	58216	58250	58248	58224	58225		
	84766	Hexahedron	5:Concrete	58231	58230	58212	58213	58244	58242	58221	58222		
	84767	Hexahedron	5:Concrete	58233	58232	58214	58215	58248	58246	58223	58224		

1D | 2D | 3D | Point Spring | Matrix Spring | Rigid Link | Elastic Link | Interpolation | Line Interface | Plane Interface | Gauging Shell | Pile | Pile

Tip

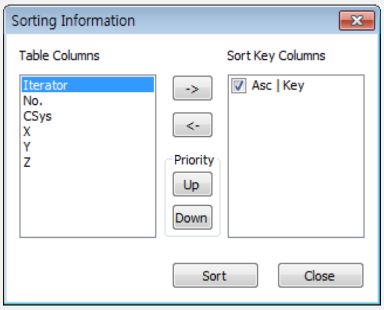
Click the right mouse button on the node/element table to bring up the Context Menu. The functions available in the menu are as follows.



Copy/Paste/Find

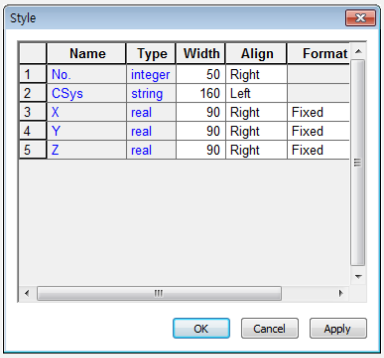
Copy the selected area on the table and paste it into a different table on excel or search particular number

Sorting Dialog



Organize the table information according to a certain principle. Each column title is listed in the Table column and the chosen align principle is moved to the Align key column. The checked Asc in the Align key column organizes in ascending order and the top column on the Align key column is given priority. The priority can be changed by using the Up, Down button.

Style Dialog



Change the expression style of each data on the table. The name of each column is listed and Integer, real, string is automatically listed for each string type. The width, alignment (Left, Right, Center) format, digits or each column can be set. The data format can be chosen from the 4 options below.

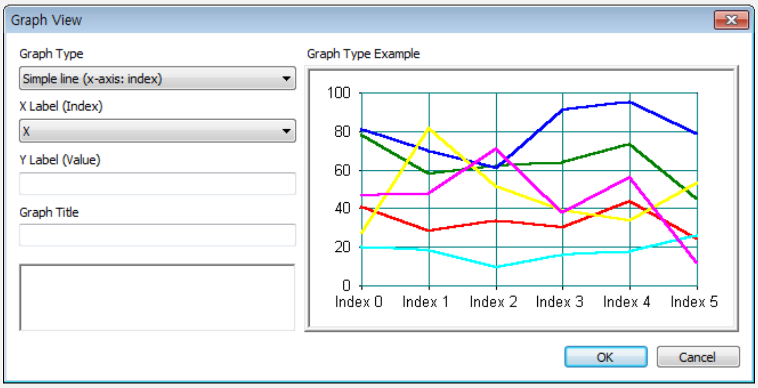
- Default : The data is expressed in 11 digits.
- Fixed : The decimal place is specified by the input value.
- Scientific : The input value is used as the number of significant figures, which are expressed in exp1ntial form. It is useful for expressing very small numerical data such as the permeability coefficient.
- General : Rounding off at the digit, automatically determined by the program.

Resize Column Width

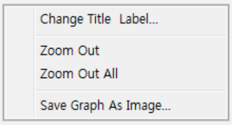
Minimize the width of each column. It is useful when there are many columns.

Show Graph

Select 2 or more columns to draw a graph. The graph type can be chosen from the 25 built-in options.



The graph can be drawn by setting the graph type and X axis, Y axis labels and click the right mouse button on the graph table to bring up the Context Menu. The functions available in the menu are as follows.



Edit the graph title and the label of each axis. Select an arbitrary point on the graph with the left mouse button and drag to magnify. In this case, use Zoom out to return to the original state. The graph can be saved as an image file (*.bmp, *.jpg. etc.)

Export to Excel / Active Dialog

Auto-save the printed table as an excel file and re-set the mesh set to print the node/element information from using the Active dialog box.