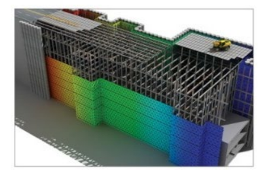
Release Note

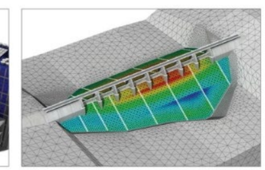
Release Date: June 2024

Product Version: GTS NX 2024(v1.1)

GTS NX

Geo-Technical analysis System New eXperience





Integrated Solver Optimized for the next generation 64-bit platform

Finite Element Solutions for Geotechnical Engineering

MIDAS

Enhancements

Analysis

1.1 Bowl Model

1.2 Multiple Shear Mechanism Consideration Option

1.3 Fluid Element(Sloshing)

1.4 SRM Inclusion Elements

1.5 Rayleigh Damping by Element(Material)

1.6 Newmark-β Method

1.7 Coupled Stress, Seepage and Time History Analyses

1.8 Saturated an Unsaturated Soil Properties

Integrated Solver Optimized for the next generation 64-bit platform

Finite Element Solutions for Geotechnical Engineering

MIDAS

Enhancements

Pre/Post Processing

Integrated Solver Optimized for the next generation 64-bit platform

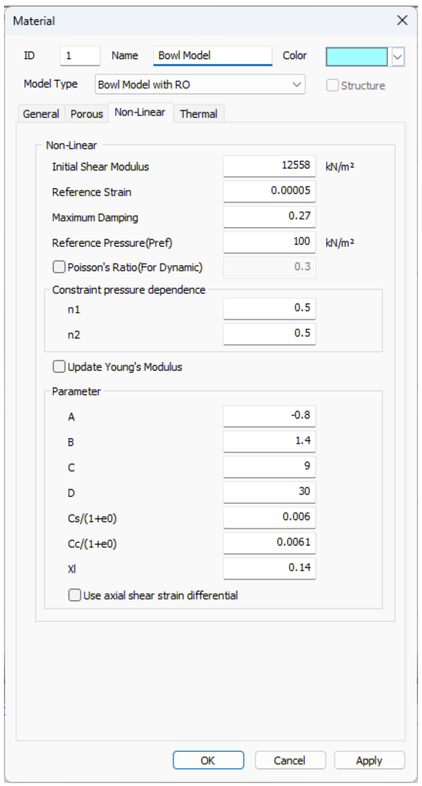
Finite Element Solutions for Geotechnical Engineering

MIDAS

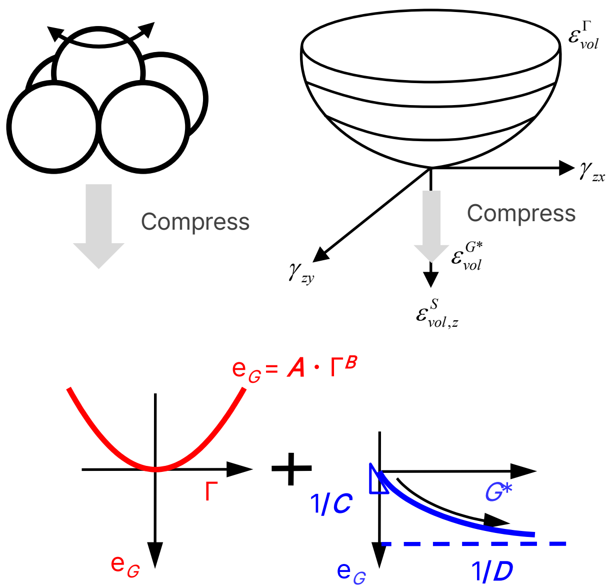
1.1 Bowl Model

This model was proposed by Fukutake & Matsuoka to model multidirectional simple shear-induced dilatancy and is applied to the Modified Ramberg-Osgood model to consider liquefaction due to seismic loading.

Mesh > Prop./Csys./Func. > Material > Isotropic > Bowl model(with RO)



[Bowl model(with RO)]



[Bowl Model]

The incremental volume deformation of soil is generally composed of the incremental deformation due to shear and the incremental deformation due to compression. $\varepsilon_{vol} = \varepsilon_{vol}^{s} + \varepsilon_{vol}^{c}$

Volumetric deformation by shear: $\varepsilon_{vol}^{s} = \varepsilon_{vol}^{\Gamma} + \varepsilon_{vol}^{G*}$

In the Bowl model, when shear occurs, soil particles are considered to move along the bowl as they rise in contact with surrounding particles. : $\varepsilon_{vol}^{\Gamma} = A\Gamma^{B}$

Also, the bowl itself undergoes volumetric deformation as shear disturbance occurs, compressing outward.

$$\varepsilon_{vol}^{G*} = \frac{G*}{C + DG*}$$

Volumetric deformation due to compression is determined by the relationship between the initial mean effective stress and the current mean effective stress of the bowl model:

$$\varepsilon_{vol}^{c} = \frac{C_s}{1+e_0}\log\frac{\sigma_{b,m}^{'}}{\sigma_{0,m}^{'}}$$

Assuming the condition of no drainage, the mean effective stress of the Bowl model at the state where total volumetric deformation becomes 0 would be...

$$\sigma_{b,m}^{'} = \sigma_{0,m}^{'} 10^{-\frac{1+e_0}{C_s}\varepsilon_{vol}^{s}}$$

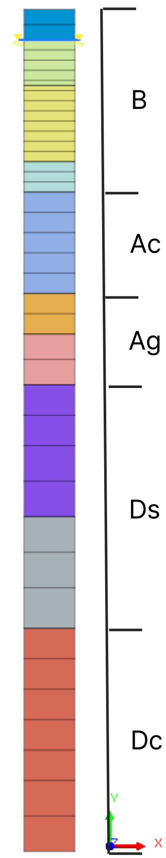
Using the average effective stress of the bowl model, the parameters of the modified Ramberg-Osgood model are modified to match the current ground condition, considering the liquefaction effect.

1.1 Bowl Model

Compared to other material models, it has fewer parameters, can be easily determined from experimental values and estimated values, and has a short analysis time, so it is a liquefaction model that can be easily used in practice.

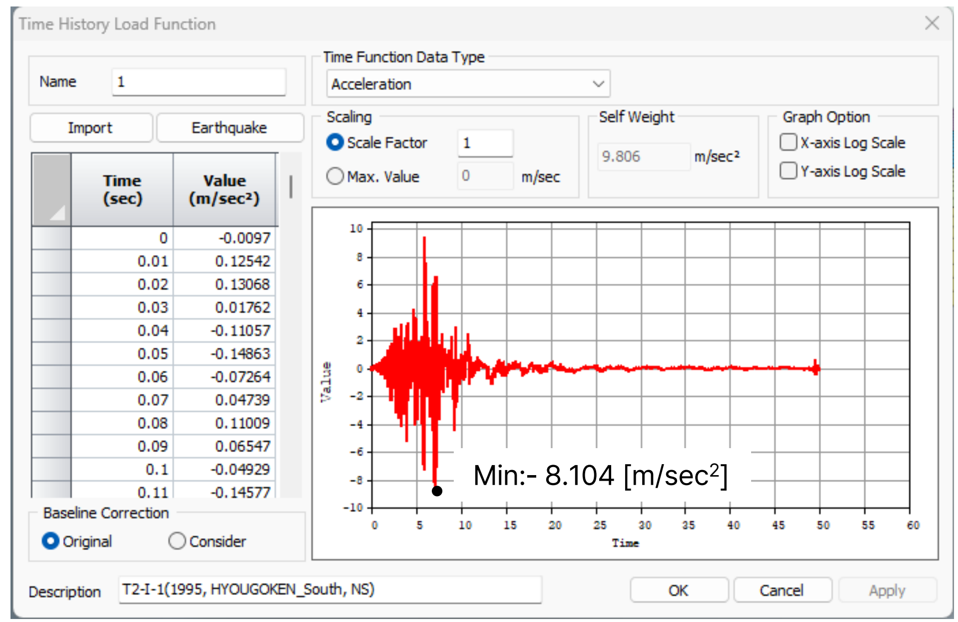
Mesh > Prop./Csys./Func. > Material

GL(m)	Soil Layer		Wave (m/s)	Unit vol. weight (kN/m3)
0 ~ −3	Back Fill	B	140	18.53
−3 ~ −7.5			170	21.28
−7.5 ~ −15			200	21.18
−15 ~ −18			220	21.57
−18 ~ −28	Clay	Ac	180	15.69
−28 ~ −32	Sand	Ag	245	17.65
−32 ~ −37			305	17.65
−37 ~ −50	Sand	Ds	305	18.14
−50 ~ −61			350	18.14
−61 ~ −83	Clay	Dc	303	17.65



[Fixed End(E+F Input)]

B、Ag Layers → Modified Ramberg-Osgood+Bowl Model
Ac、Ds、Dc Layers → Modified Ramberg-Osgood Model

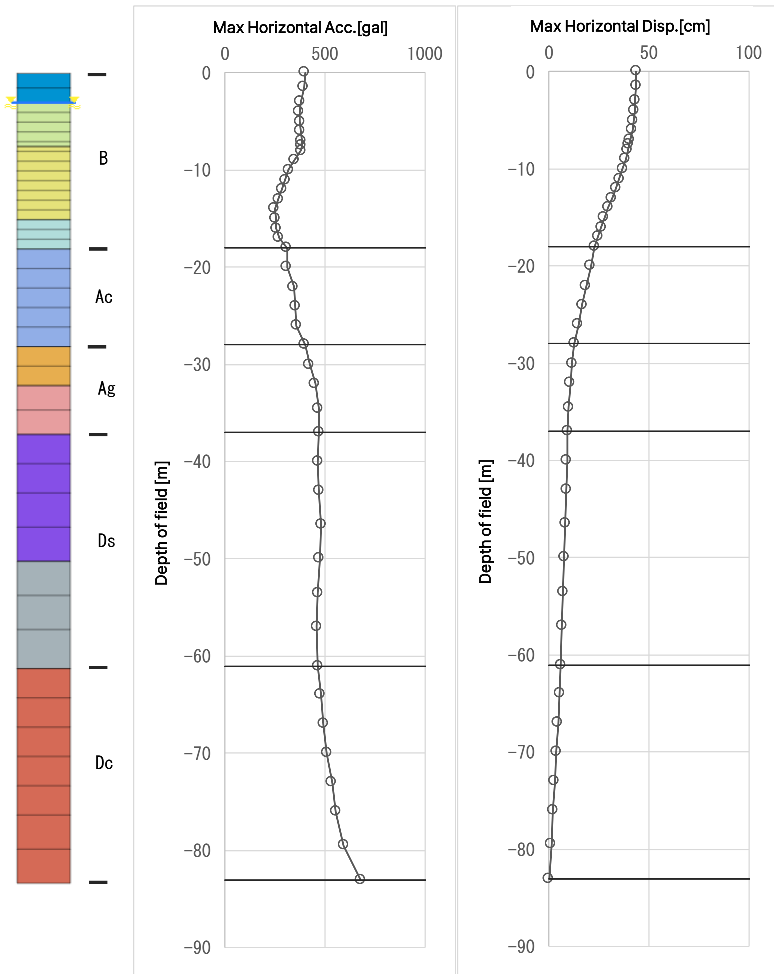


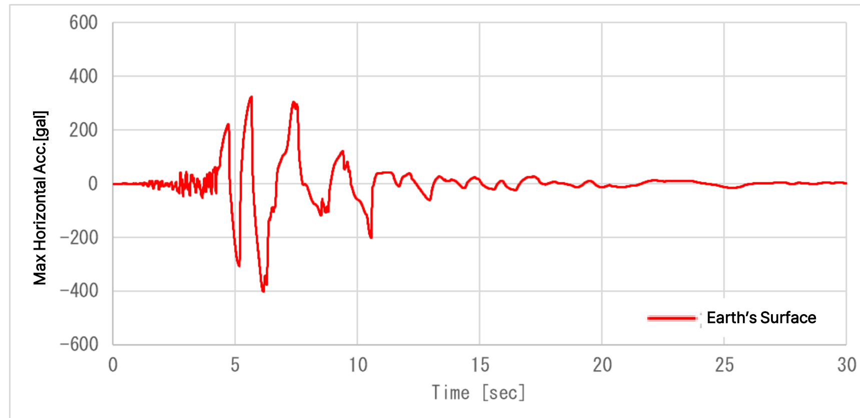
[Ground Acceleration]

1.1 Bowl Model

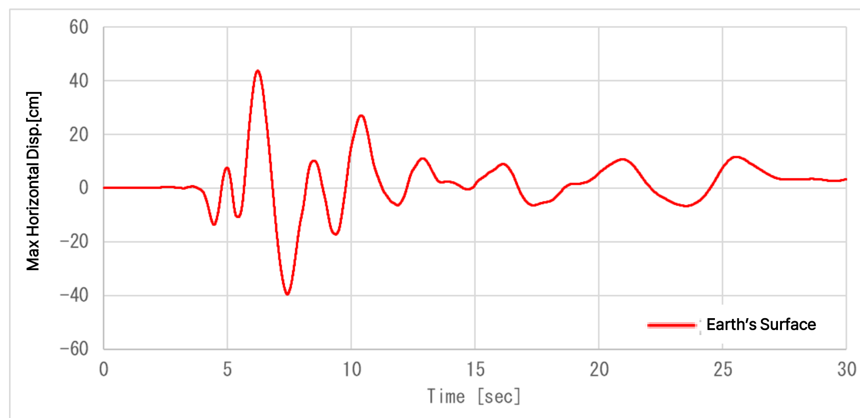
During an earthquake in the depth direction, it can be confirmed that the acceleration is attenuated, and the displacement increases through the response on the maximum acceleration and maximum displacement indicators.

Mesh > Prop./Csys./Func. > Material





Surface Horizontal Acceleration – Time Graph



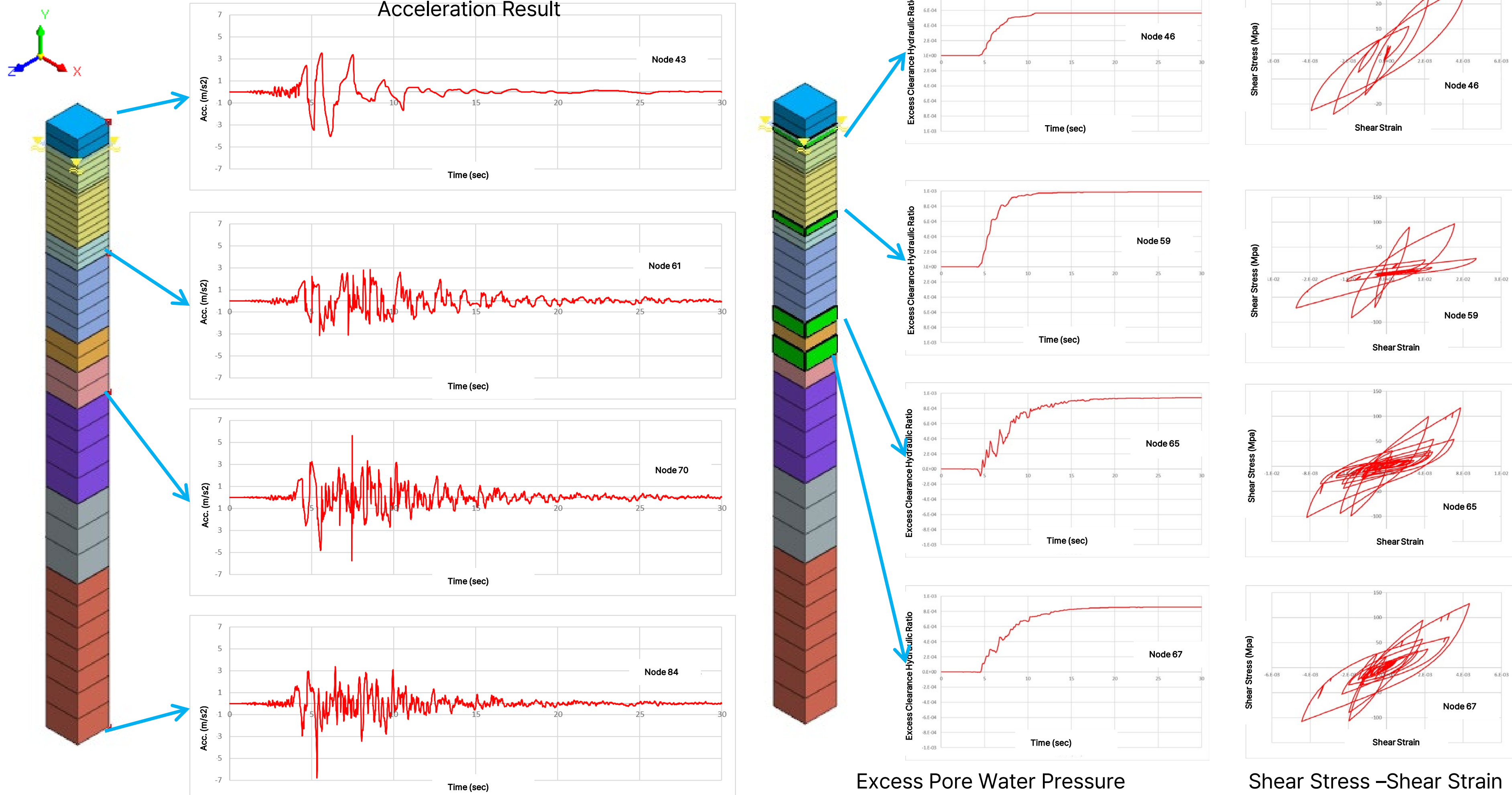
Surface Horizontal Displacement – Time Graph

1.1 Bowl Model

As the acceleration of the focal point is transmitted to the surface, In the liquefaction layer, excess pore water pressure increases and shear stiffness decreases. This can be confirmed from the shear stress-shear strain relationship.

Mesh > Prop./Csys./Func. > Material

Acceleration Result



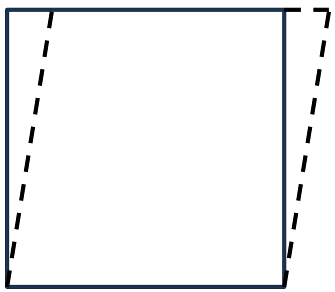
Excess Pore Water Pressure

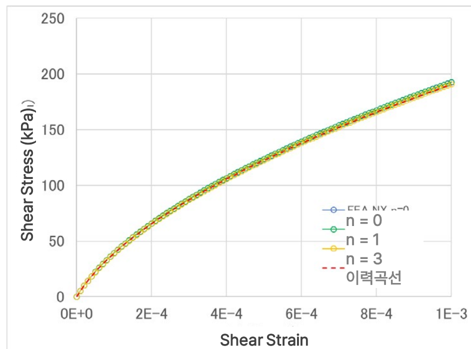
Shear Stress –Shear Strain

1.2 Multiple Shear Mechanism Consideration Option

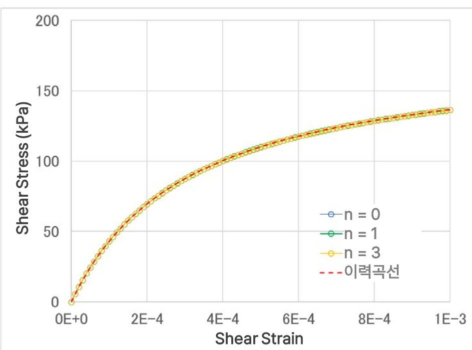
The option considering multi-shear mechanisms allows for an extension of functionality in material models (such as the Modified Ramberg-Osgood model, Modified Hardin-Drnevich model, GHE-S model), where only shear stress is considered. This extension enables the reflection of the rotation of the principal stress axes in the material model.

Analysis > Analysis Case > General > Analysis Type : Nonlinear Analysis/ Construction Stage Analysis / Nonlinear Time History Analysis / Nonlinear Time History Analysis + SRM > Analysis Control

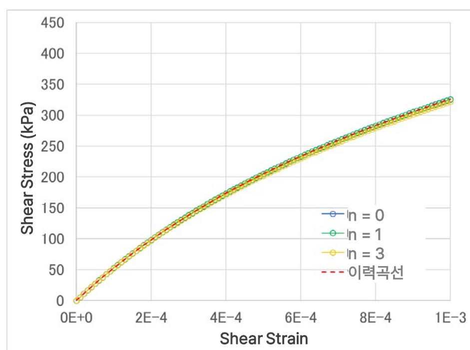




RO Model

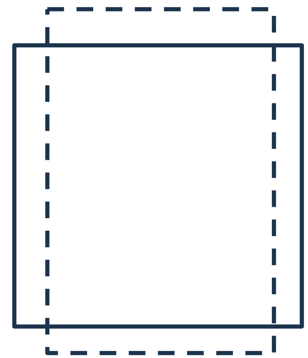


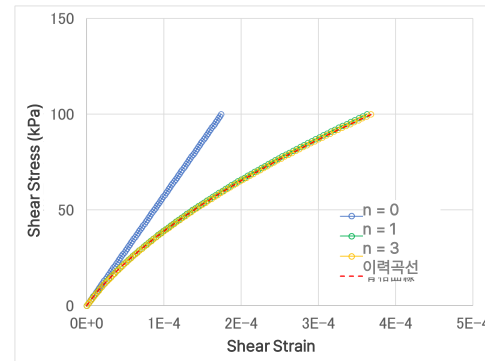
HD Model



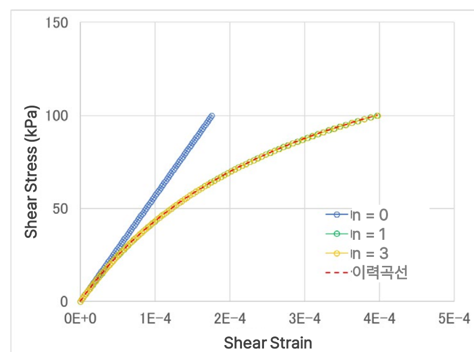
GHE-S Model

[Simple Shear]

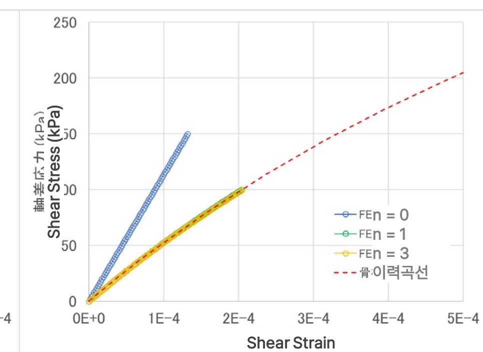




RO Model



HD Model



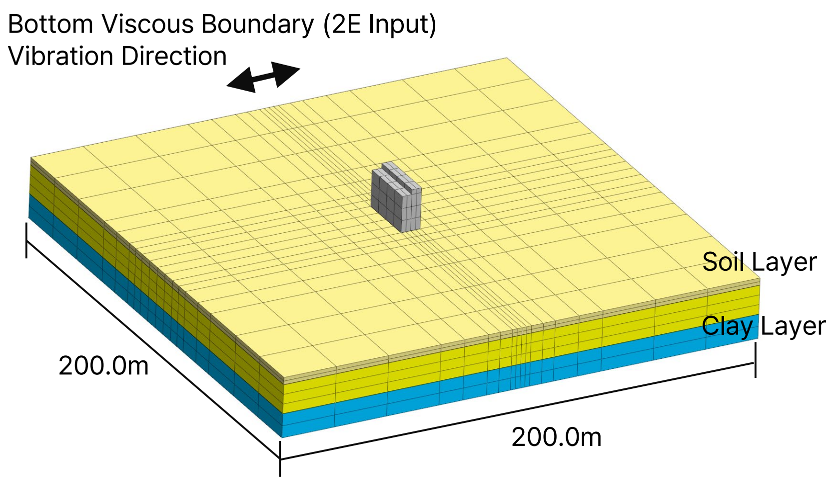
GHE-S Model

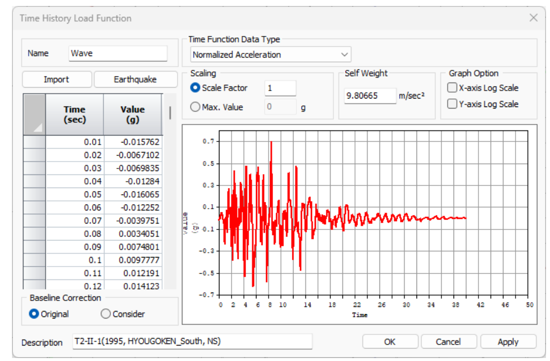
[Multi Shear]

1.2 Multiple Shear Mechanism Consideration Option

The option considering multi-shear mechanisms allows for an extension of functionality in material models (such as the Modified Ramberg-Osgood model, Modified Hardin-Drnevich model, GHE-S model), where only shear stress is considered. This extension enables the reflection of the rotation of the principal stress axes in the material model.

Analysis > Analysis Case > General > Analysis Type : Nonlinear Analysis/ Construction Stage Analysis / Nonlinear Time History Analysis / Nonlinear Time History Analysis + SRM > Analysis Control





[Ground Acceleration]

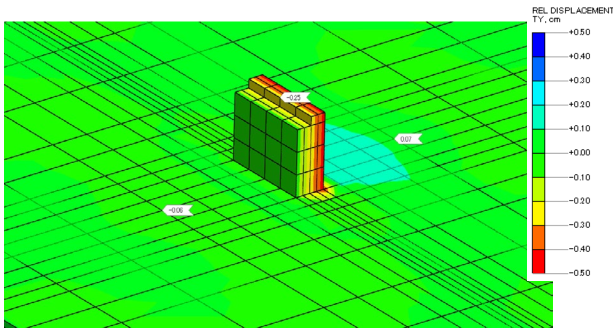
	Ground Model	Unit vol. weight [kN/m²]	Standard Confining Pressure [kN/m²]	Shear Modulus of E [kN/m²]	Refer. Strain	Confining Pressure Dependence Coff.	Poisson's Ratio	Max. Damping Ratio
Fill Layer	Modified R-O	17.60	18.00	64980.00	3.39e-04	0.50	0.33	0.30
Sand Layer	Modified R-O	17.60	66.00	64980.00	1.29e-03	0.50	0.33	0.30
Clay Layer	Modified R-O	16.70	120.00	38250.00	3.97e-03	0.50	0.33	0.20

[Ground Material Properties]

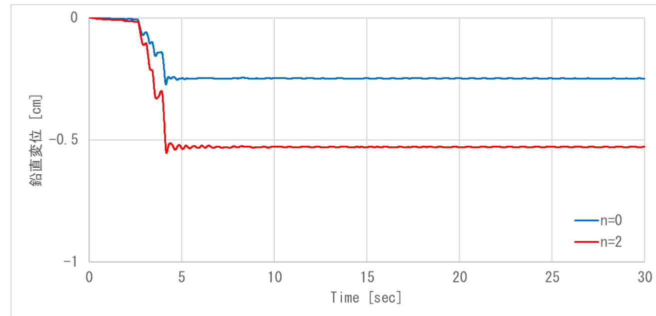
1.2 Multiple Shear Mechanism Consideration Option

The option considering multi-shear mechanisms allows for an extension of functionality in material models (such as the Modified Ramberg-Osgood model, Modified Hardin-Drnevich model, GHE-S model), where only shear stress is considered. This extension enables the reflection of the rotation of the principal stress axes in the material model.

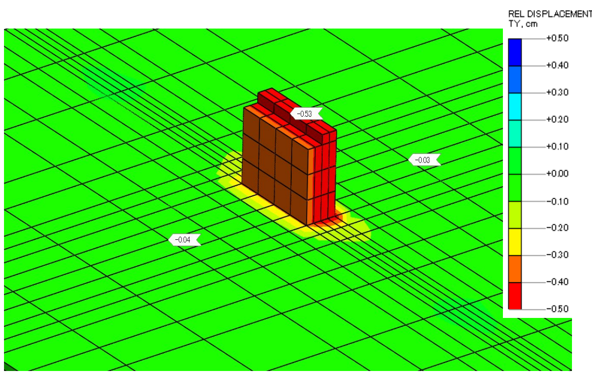
Analysis > Analysis Case > General > Analysis Type : Nonlinear Analysis/ Construction Stage Analysis / Nonlinear Time History Analysis / Nonlinear Time History Analysis + SRM > Analysis Control



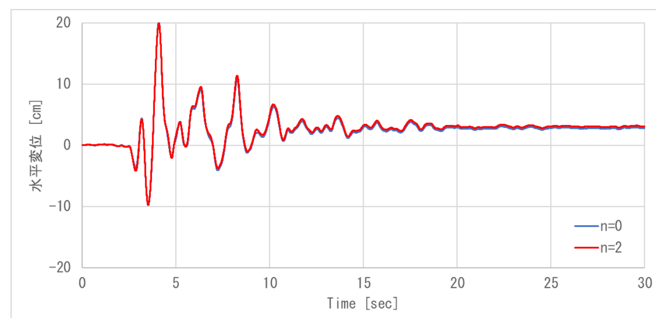
[Relative Displacement Multiple Shear Mechanism (n=0)]



[Vertical Displacement History]



[Relative Displacement Multiple Shear Mechanism (n=2)]

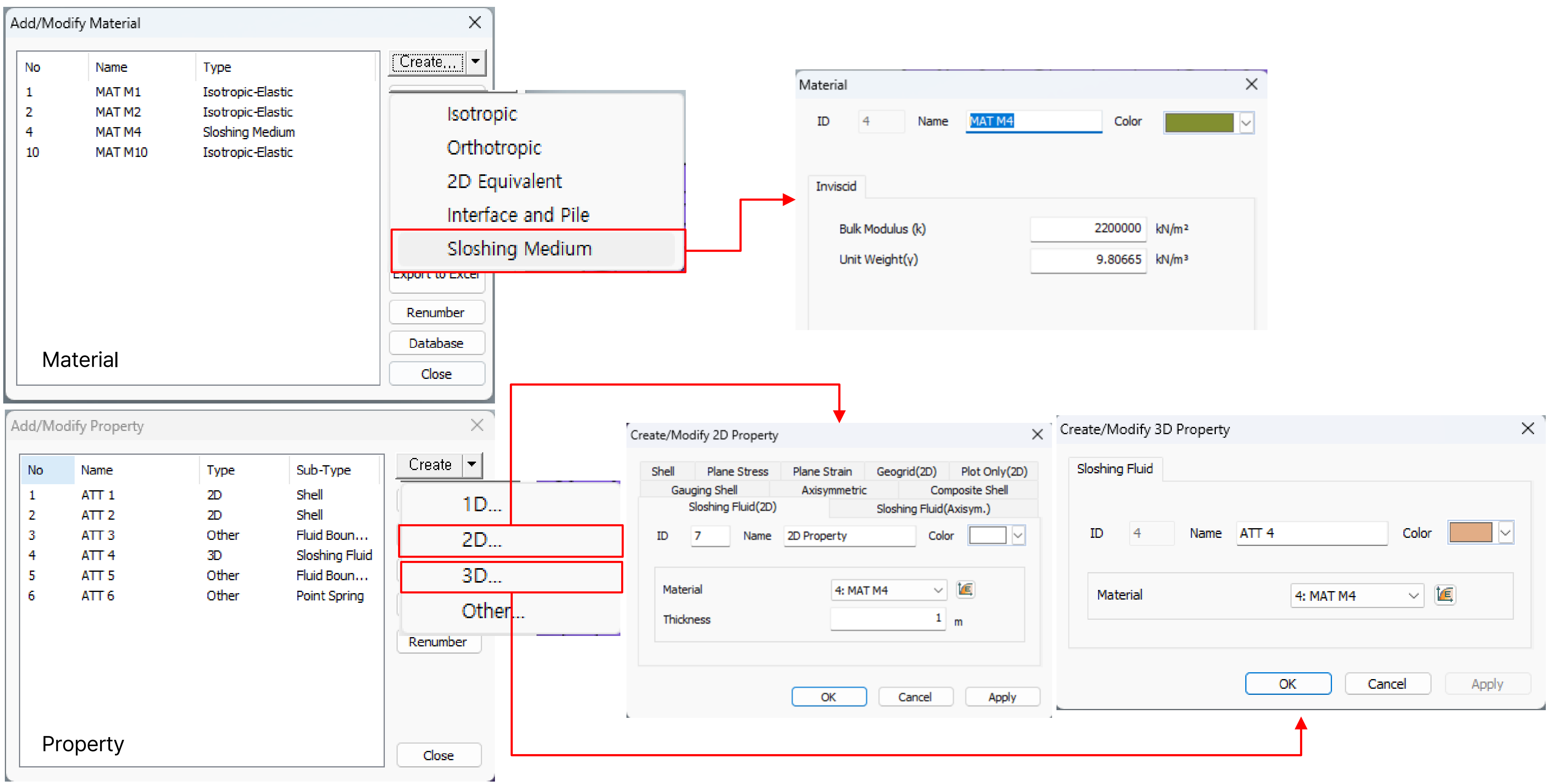


[Horizontal displacement history]

1.3 Fluid Element (Sloshing)

A fluid element that simulates water in structures and liquid gas in LNG has been added. It calculates wave height and pressure during earthquakes, predicting tank stake proximity and pressure. This Sloshing Medium also models reservoir sloshing conditions during earthquakes, serving as an alternative to Westergaard's Added Mass.

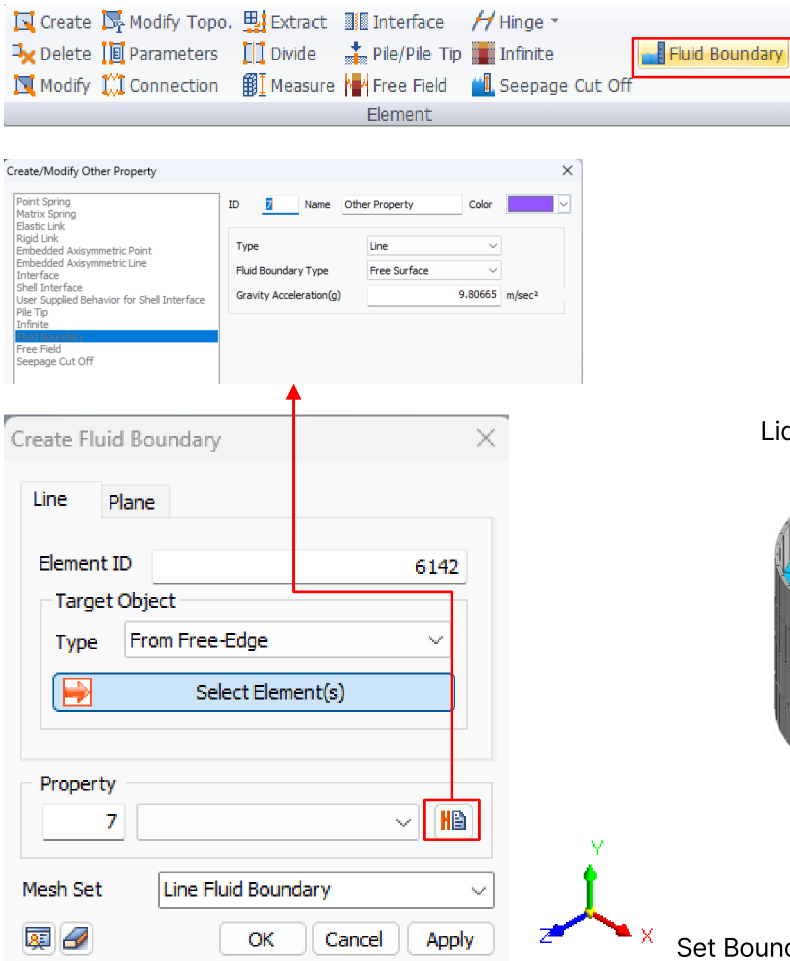
Mesh > Prop./CSys./Func. > Material

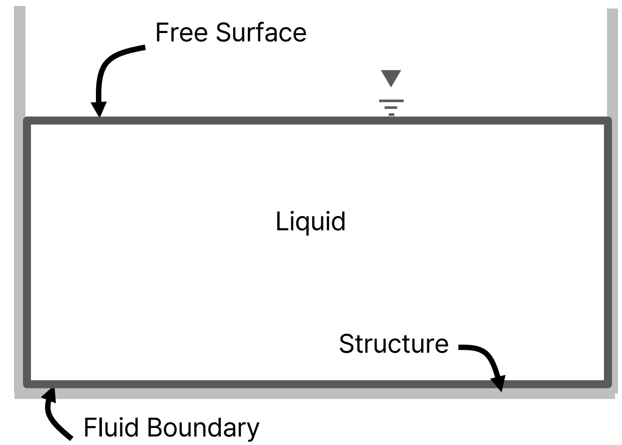


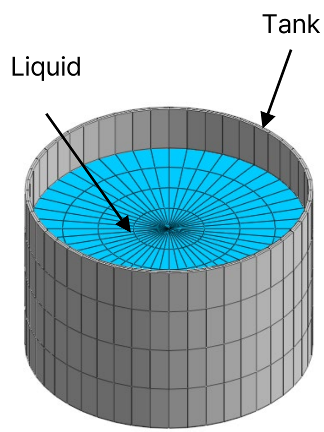
1.3 Fluid Element (Sloshing)

A fluid element that simulates water in structures and liquid gas in LNG has been added. It calculates wave height and pressure during earthquakes, predicting tank stake proximity and pressure. This Sloshing Medium also models reservoir sloshing conditions during earthquakes, serving as an alternative to Westergaard's Added Mass.

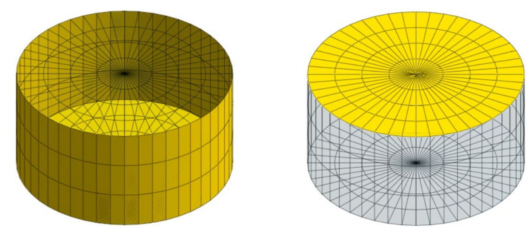
Mesh > Element > Fluid Boundary







Analysis Model



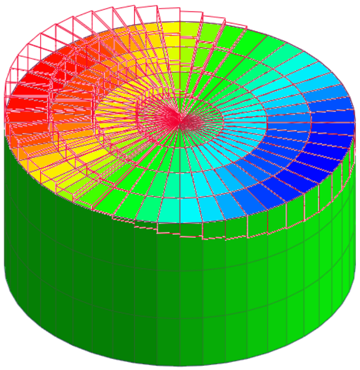
Fluid Boundary
(Interface between Structure and liquid)

Set Boundary element on the part in contact with the object and on the top of the liquid

1.3 Fluid Element (Sloshing)

A fluid element that simulates water in structures and liquid gas in LNG has been added. It calculates wave height and pressure during earthquakes, predicting tank stake proximity and pressure. This Sloshing Medium also models reservoir sloshing conditions during earthquakes, serving as an alternative to Westergaard's Added Mass.

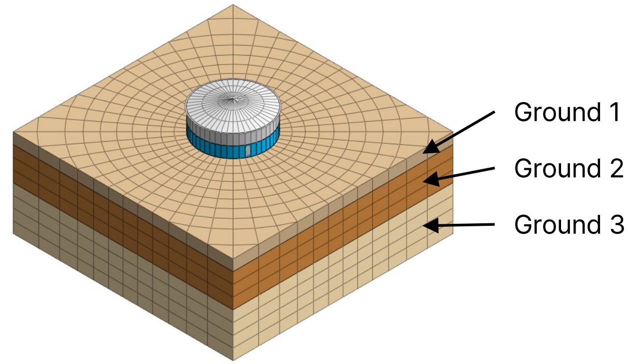
Mesh > Prop./CSys./Func. > Material

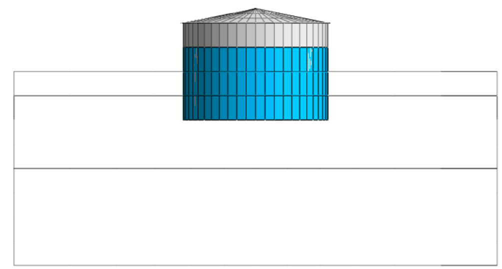


Velocity Potential Theory

Natural Cycle
$$T_{si} = \frac{2\pi}{\omega_i} = 2\pi\sqrt{\frac{R}{\varepsilon_i g}\coth\left(\varepsilon_i \frac{H}{R}\right)}$$

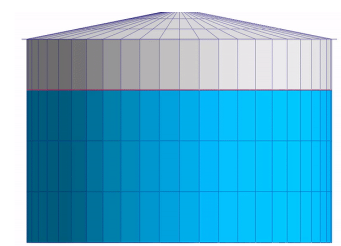
Natural Frequency
$$f(Hz) = \frac{1}{2\pi}\cdot\sqrt{\frac{(2n-1)\cdot\pi\cdot g}{L}\cdot\tanh\left(\frac{(2n-1)\cdot\pi\cdot H}{L}\right)}$$





Perform an eigenvalue Analysis including liquid elements and compare the natural frequency and natural period as follows.
εi is the i th root of dJ1(r)/dr=0 、and is calculate as ε1=1.84118.

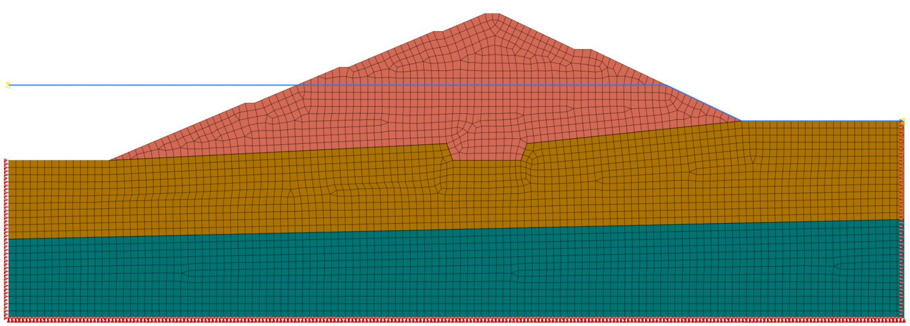
	Ts(s)	f(Hz)
Theoretical value	6.43	0.141
Analysis value	6.35	0.157



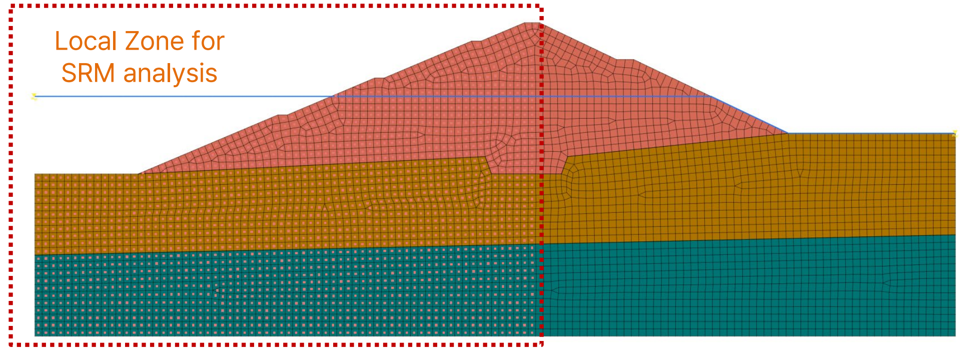
1.4 SRM Inclusion Elements

By default, the strength reduction method (SRM) assesses the entire model's stability, identifying vulnerable sections globally. For specific area analysis (Local Stability), SRM Inclusion Zones can be used. For example, in dam models, you can analyze each side independently. (※ Applicable only in Construction Stage Analysis.)

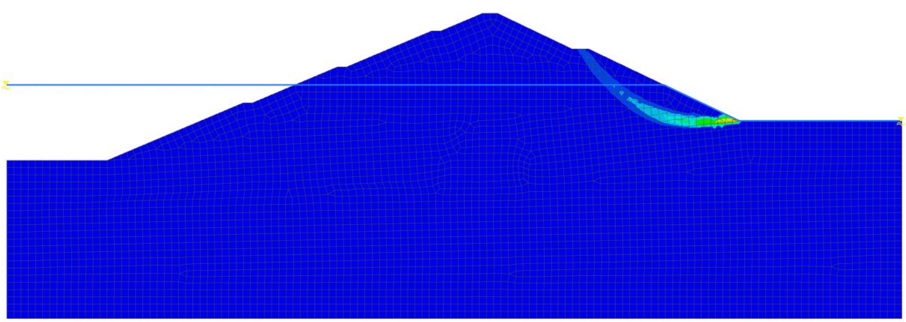
Static/Slope Analysis > Boundary > SRM Inclusion



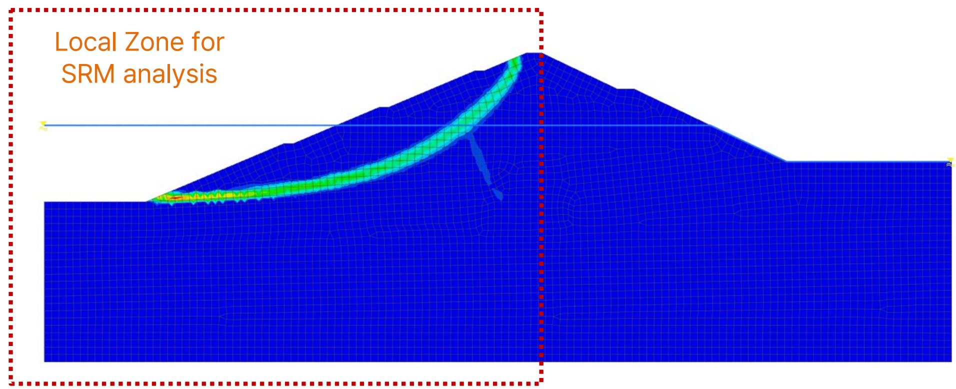
[Model]



[SRM Inclusion defined on the Left Side of the DAM]



[Global Probable Failure Surface]

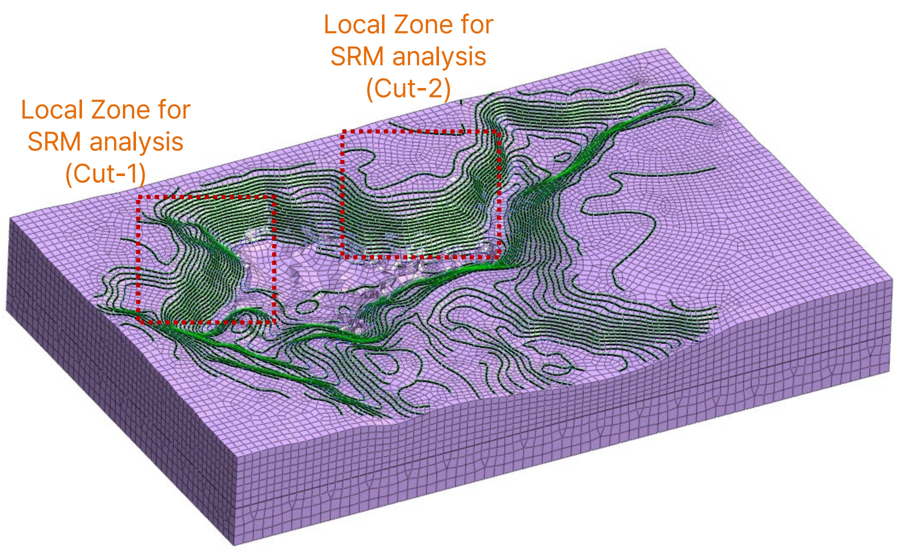


[Probable Failure Surface in the Local Area (SRM Inclusion Zones)]

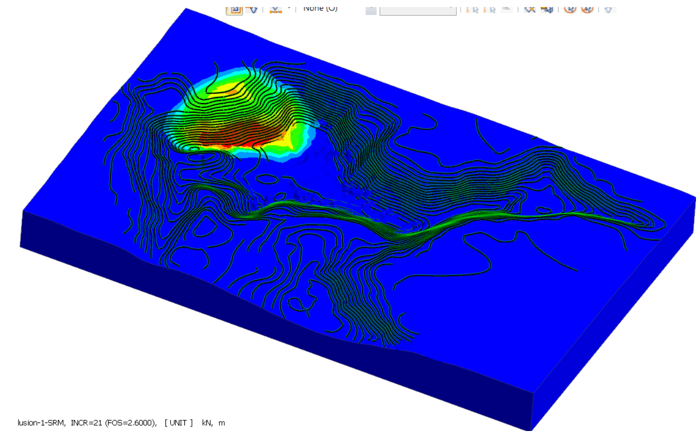
1.4 SRM Inclusion Elements

Another Application, in the case of an Open Pit Mine Models, you can independently analyze the stability of each 'Cut' of the Open Pit Mine Model.

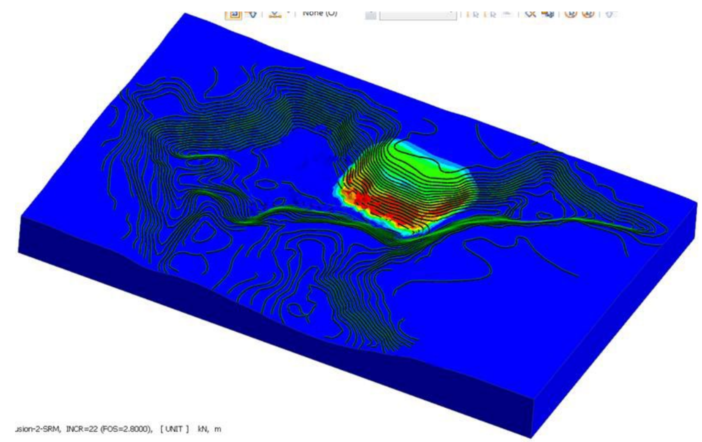
Static/Slope Analysis > Boundary > SRM Inclusion



[Open Pit Model with Local Zones (Cut 1 & 2) for FoS and Corresponding Failure Surface Determination]



[Shear Zone (Probable Failure Surface) of Cut 1 of the Open Pit Mine]

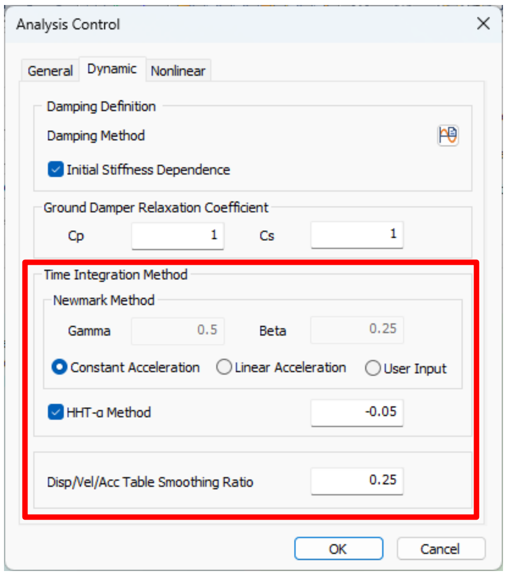


[Shear Zone (Probable Failure Surface) of Cut 2 of the Open Pit Mine]

1.5 Newmark-β Method

Until the previous version, the HHT-α method was the default numerical integration scheme. The new version adds the Newmark-β method, allowing users to choose between Newmark-β and HHT-α for analysis. Newmark-β offers three input methods, with Constant Acceleration recommended for stability. HHT-α, a generalized form of Newmark, has a default αH value of -0.05 in GTSNX.

Analysis Case > Analysis Control



Newmark Method: In the direct integration method, the Newmark method is used for numerical integration of the equations of motion, and two parameters related to this, Gamma and Beta, are input.

Constant Acceleration Method:
This method assumes that the acceleration of the structure remains constant over each time step interval, automatically inputting Gamma (=1/2) and Beta (=1/4). According to this assumption, in the analysis based on direct integration, the interpretation results can prevent divergence regardless of the value of the time increment.

Linear Acceleration Method:
This method assumes that the acceleration of the structure changes linearly over each time step interval, automatically inputting Gamma (=1/2) and Beta (=1/6). According to this assumption, in the analysis based on direct integration, if the time increment is more than 0.551 times the shortest period contained in the structure, the interpretation results may diverge.

User Input:
Users input the values of Gamma and Beta directly.

Displacement/Velocity/Acceleration Damping Coefficient: In co-analysis, to prevent deterioration of convergence due to abrupt changes, the curve inputted in the solver is smoothed for use. Entering '0' means no smoothing is applied.

※ Control of the Newmark method according to the time integration method cannot be done on a stage-by-stage basis in the construction stage analysis, so it has been added as a global setting. Consequently, even in general step-by-step analyses, the dynamic analysis tab is displayed, but the control values in this dynamic analysis tab are only reflected in the analysis when performing stress-nonlinear time history analysis.

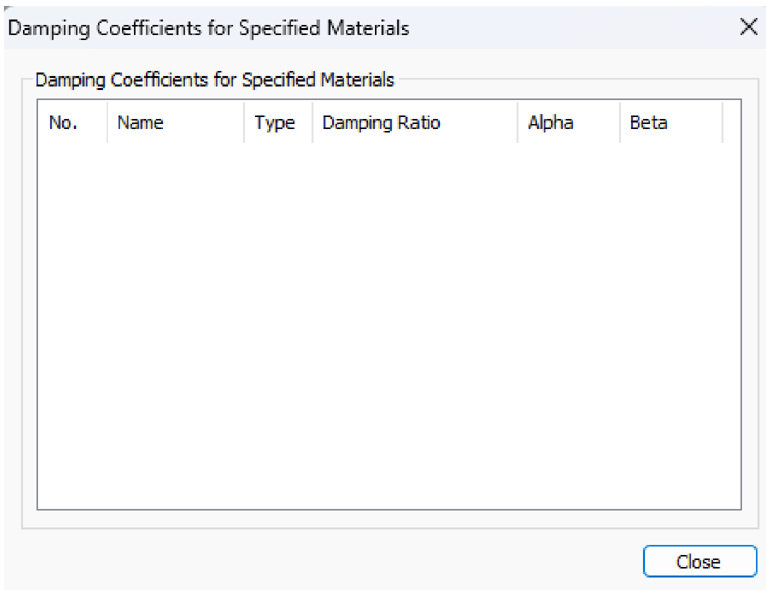
1.6 Rayleigh Damping by Element(Material)

During seismic analysis, the superstructure, substructure, and ground all have different attenuation coefficients
Therefore, in the analysis, a function is installed to calculate the attenuation coefficients α and β for each material.

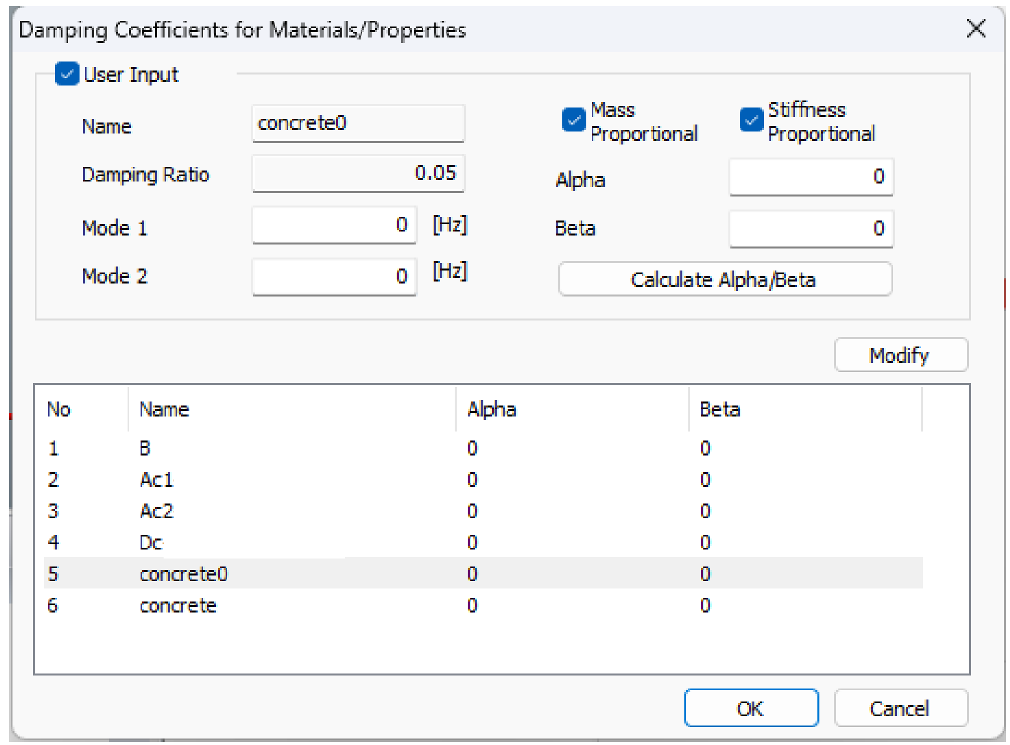
Analysis > Analysis Control > Dynamic > Damping Method

In the previous version, the α and β of all the materials are calculated using the inputted frequencies of the model.

In the new version, user has an option to input frequencies of each material and calculate α and β separately.



[Previous Version]

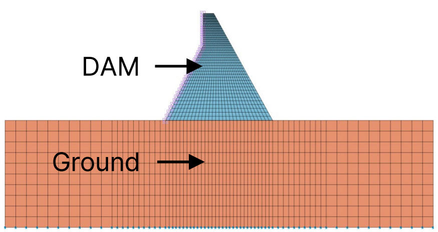


[New Version]

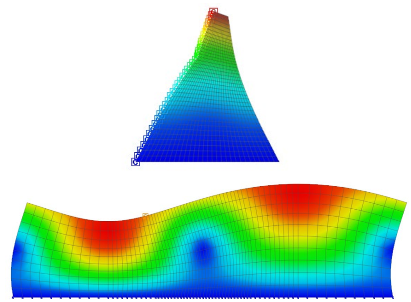
1.6 Rayleigh Damping by Element(Material)

During seismic analysis, the superstructure, substructure, and ground all have different attenuation coefficients
Therefore, in the analysis, a function is installed to calculate the attenuation coefficients α and β for each material.

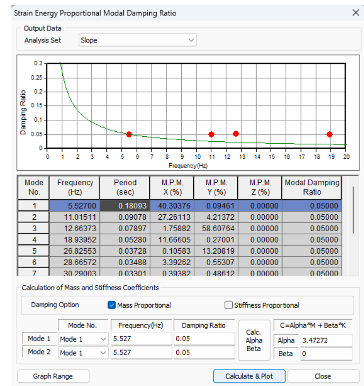
Analysis > Analysis Control > Dynamic > Damping Method



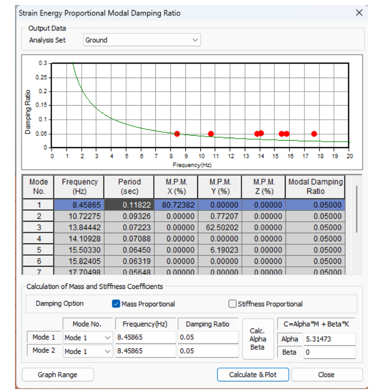
① Analysis Model



② Eigenvalue Analysis of Structures and Ground

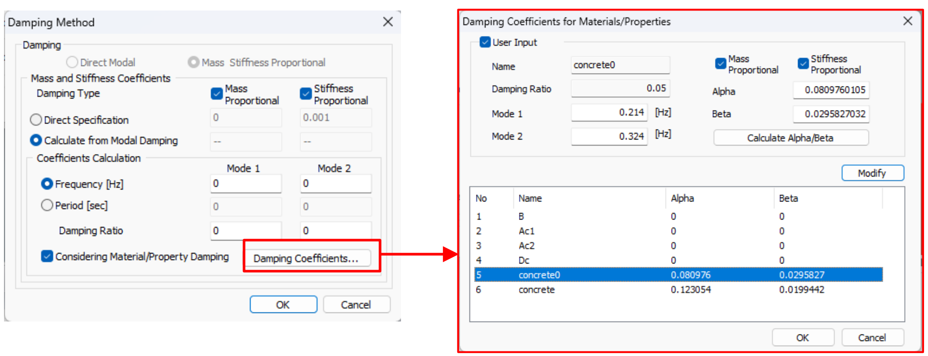


[Damping Constant of DAM]

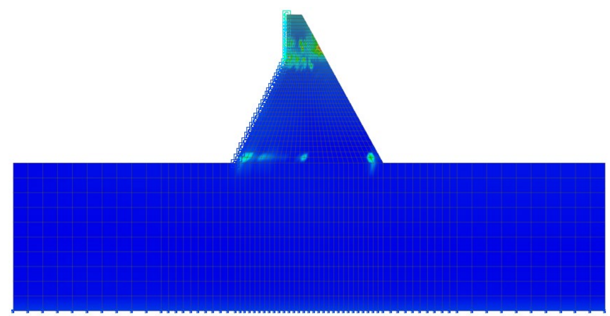


[Damping Constant of the Ground]

③ Damping Constant for each mode



④ Damping Coefficients for Material



⑤ Perform Dynamic Analysis

1.7 Coupled Stress, Seepage and Nonlinear Time History Analysis

In the new version, user can couple Stress, Seepage, Slope and Nonlinear Time History Analysis.
For Example, in case of earthen dam, one can consider the effects of construction sequence, seepage, and earthquake for the assessment in a single analysis.

Static/Slope Analysis > Construction Stage > Stage Set > Stress-Seepage-Slope-Nonlinear Time History

1.8 Saturated and Unsaturated Soil Properties

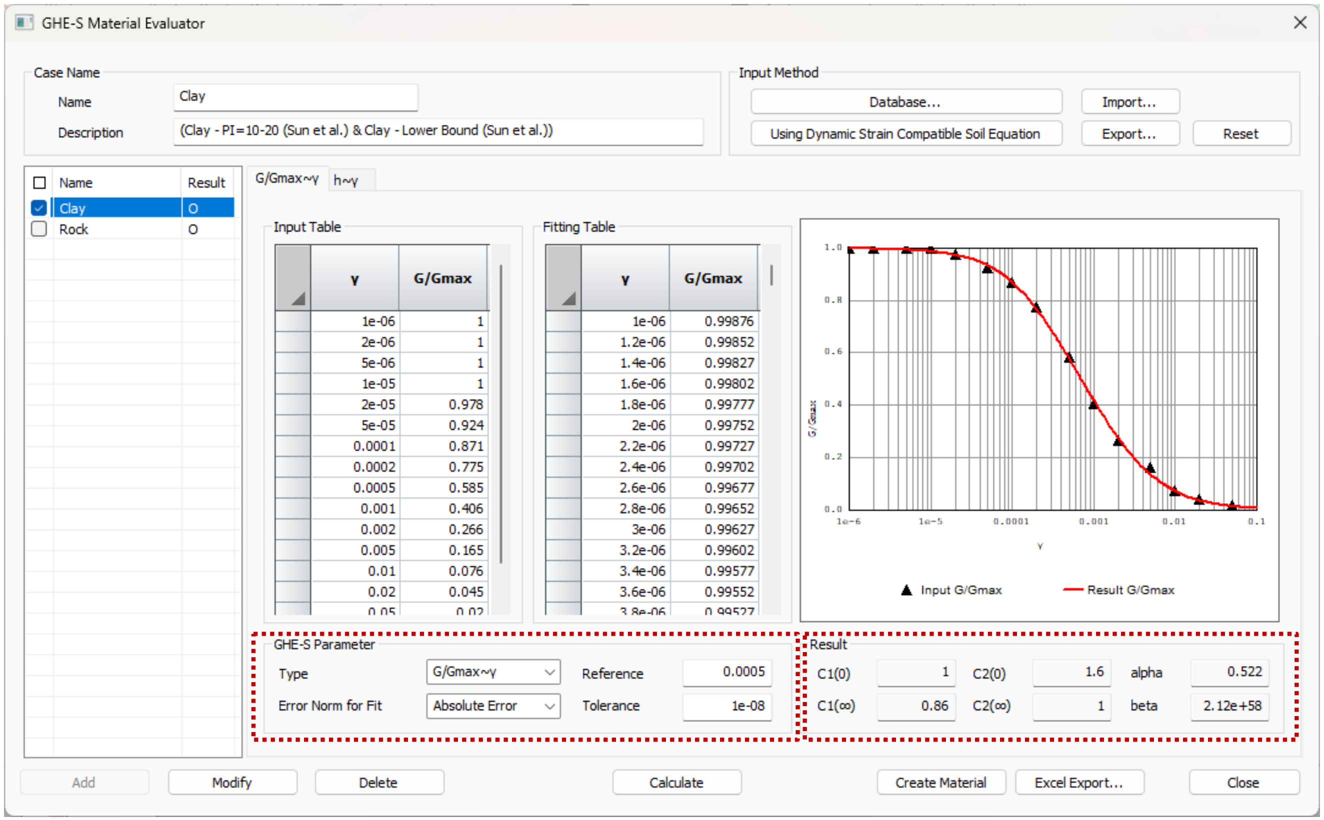
The strength parameters such as C & phi varies in both saturated and unsaturated conditions for a material. In the new version, the user can define two different properties of the same material in both Unsaturated and Saturated Conditions.
And software automatically takes the respective properties of the material depending on the pore pressure developed when ‘Auto Change Property By Pore Pressure’ boundary condition is defined.

Mesh > Element > Parameters > 2D/3D > Auto Change Property by Pore Pressure

2.1 Material Evaluator (GHE-S Model)

The Japanese railway dynamic nonlinear material model employs the GHE (General Hyperbolic Equation) proposed by Tatsuoka and Shibuya for the skeleton curve and hysteresis law improves upon the Massing law for the stress-strain relationship to satisfy $G/G_0\sim\gamma$ and $h\sim\gamma$ relationships.
When $G/G_0\sim\gamma$ and $h\sim\gamma$ relationship experimental data are entered, the parameters required for the material definition are automatically calculated.

Dynamic Analysis > Tools > Material Evaluation > GHE-S Model



[GHE-S Model Material Evaluation]

※ In the definition of the existing GHE-S model, the nonlinear tab's sub-material evaluation function has been moved to the tool position .

Type:

Choose whether to estimate parameters from the raw experimental data $G/[\![G\_max\sim\gamma]\!]$ or from the normalized data.

Error Norm for Fit:

These are the criteria used to evaluate errors when estimating data.

Relative Error:

(True Value - Approximate Value) / True Value

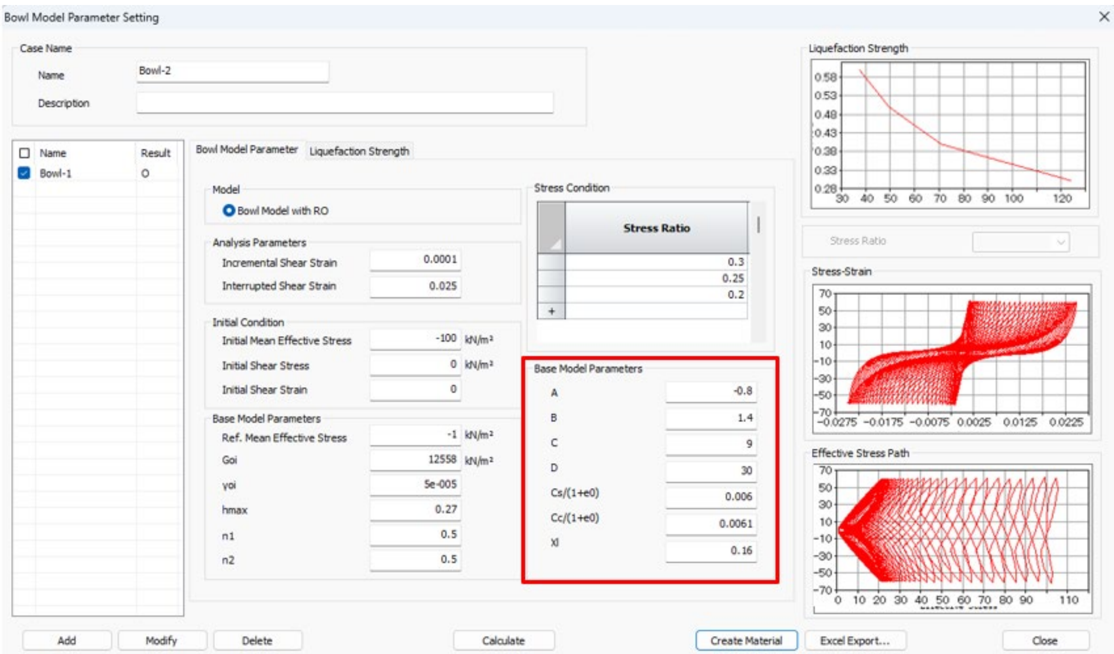
Absolute Error:

True Value - Approximate Value

2.1 Material Evaluator (Bowl Model)

The model proposed by Fukutake & Matsuoka for modeling dilatancy due to multi-directional simple shear is applied to the Modified Ramberg-Osgood model to account for liquefaction caused by seismic loading. When experimental values and estimated values are input, the parameters necessary for material definition are automatically calculated.

Dynamic Analysis > Tools > Material Evaluator > Bowl Model



[Bowl Model Material Evaluation]

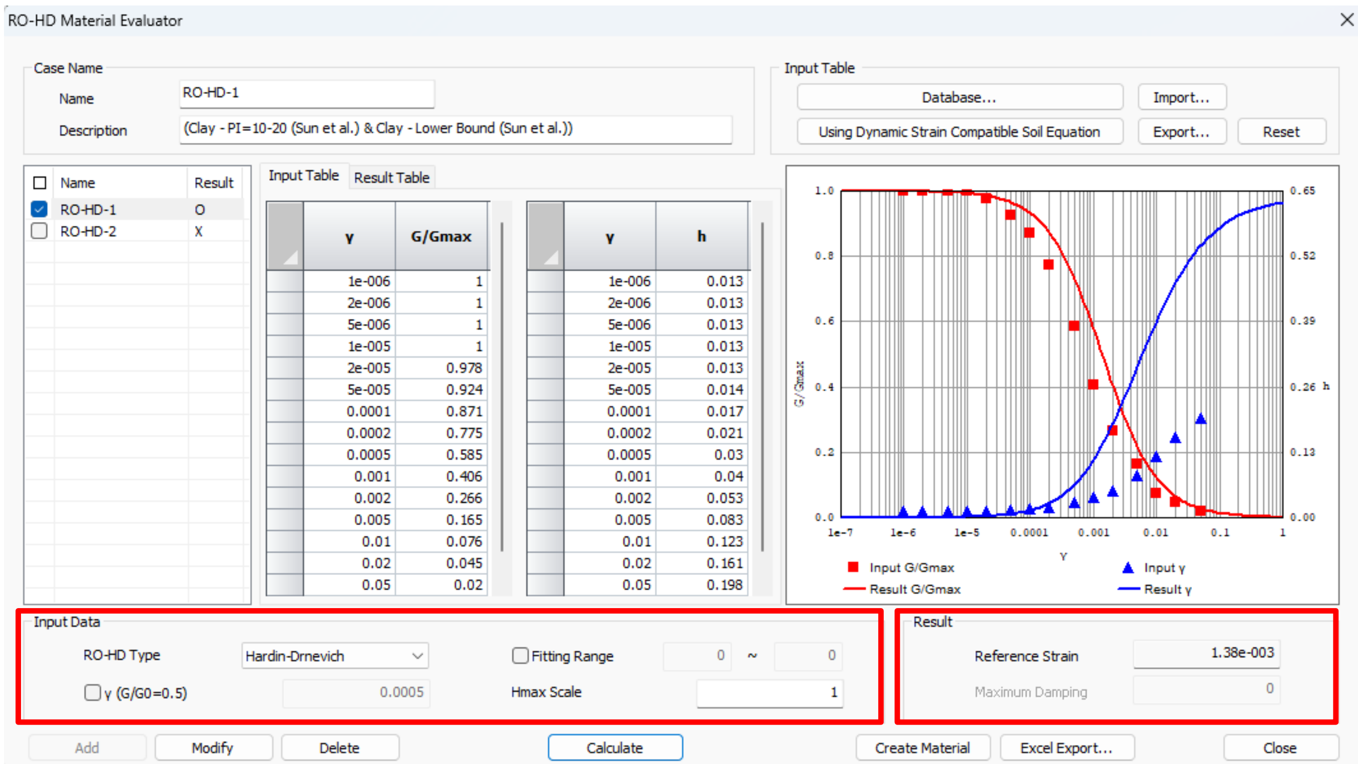
	Description
A, B	Represents the swelling component within dilatancy. The larger the magnitude of |A|, the more pronounced the dilatancy.
C, D	Indicates the compressive component in the dilatancy. 1/C is the slope of the dilatancy with respect to the early shear, 1/D is a tangent of a hyperbola
Cs/(1+e0)	Cs: Dilatancy index e0: Initial void ratio
Cc(1+e0)	Cc: Compression index e0: Initial void ratio
Xi	The lower limit of liquefaction strength *Rl*,

[Bowl Parameters]

2.1 Material Evaluator (RO/HD Model)

In the hysteretic material model, when experimental data of G/G0～γ and h~γ relationships are input, the parameters necessary for defining the material. Reference strain for Hardin-Drnevich (HD), and reference strain and maximum damping ratio for Ramberg-Osgood (HD) are automatically calculated.

Dynamic Analysis > Tools > Material Evaluator > R-O/H-D Model



[RO/HD Model Material Evaluation]

RO/HD Type:

Select the material model type.

Range:

Specify the range of shear strain (ϒ) to be applied in the interpolation.

ϒ:

Check this option if you want to use a specific value for the reference shear strain when G/G0 = 0.5.

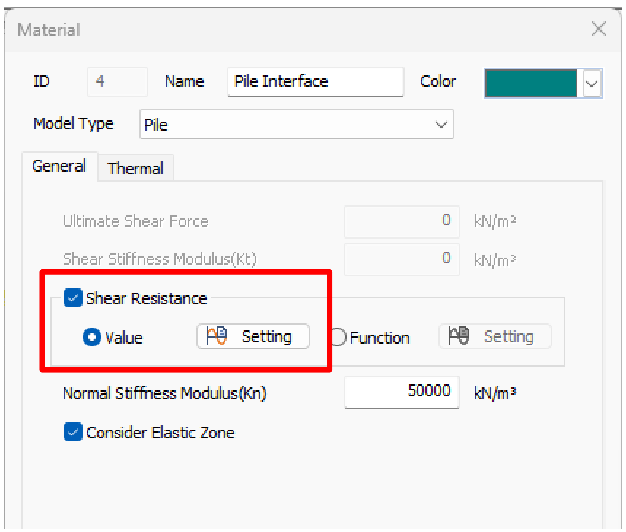
Hmax Scale:

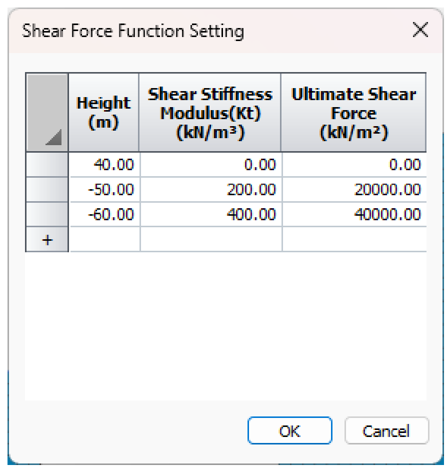
Enter the scale of the maximum damping constant.

2.2 Skin Friction vs Depth in Pile Interface

Now, defining Skin Friction vs. Depth for the Pile Interface is simpler. Users can directly input the global pile depth and corresponding ultimate shear force (skin friction). Previously, individual pile interfaces for each layer were required. This update offers three methods for defining the Pile Interface:
1. Direct definition of Skin Friction and stiffness for the entire pile. 2. Skin Friction vs. Depth & Shear Stiffness vs. Depth. 3. Direct P-y Curve definition vs. Depth.

Mesh > Prop./Csys./Func. > Material > Interface and Pile > Pile





Shear Resistance:

Select the methods, 'Value' or 'Function'

Value:

In this method, we need to define 'Ultimate Shear Force vs Height' and 'Shear Stiffness Modulus vs Height'

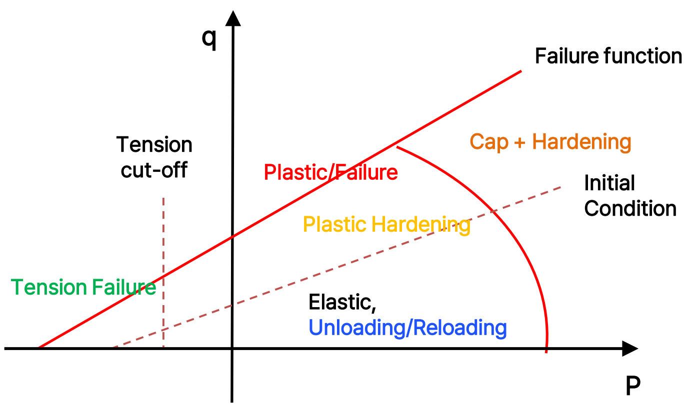
Height :

The Global Depth in the model is to be entered in the Height Column

2.3 Plastic Status Contour Improvement

In the Hardening Soil and Modified Mohr Coulomb material models, a new feature now distinguishes and outputs regions of plastic deformation or failure as Plastic Hardening and Cap+Hardening areas post-analysis. Furthermore, users can easily identify these areas by toggling the marking feature on or off through the properties window.

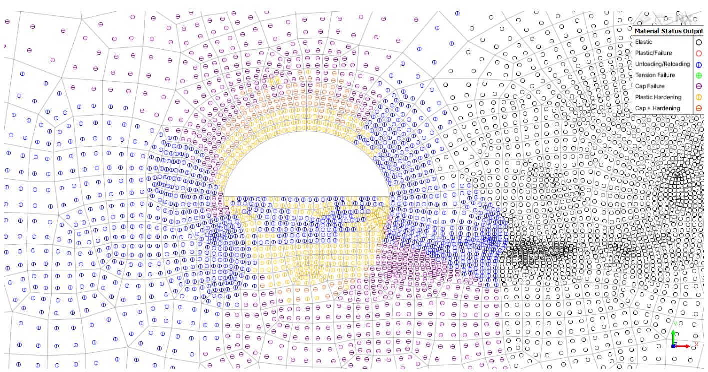
Results Works Tree >Plane Strain Stresses/Solid Stresses > Plastic Status

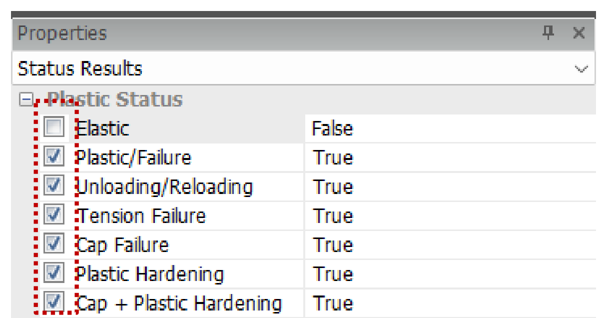


Material Status Output	
Elastic	
Plastic/Failure	
Unloading/Reloading	
Tension Failure	
Cap Failure	
Plastic Hardening	
Cap + Hardening	

- **Elastic** : When in the elastic region.
- **Failure / plastic** : When shear failure occurs
- **Unloading or reloading** : When the state changes due to the addition or removal of loads.
- **Tension / tension failure** : When failure occurs in the tension region.
- **Cap failure** : When failure occurs in the compression yield region.
- **Plastic hardening** : When the state is between the initial state and the failure state.
- **Cap + hardening** : When shear failure has occurred, and the state is in the cap region

Properties Works Tree > Status Results

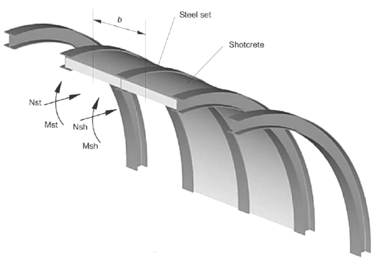


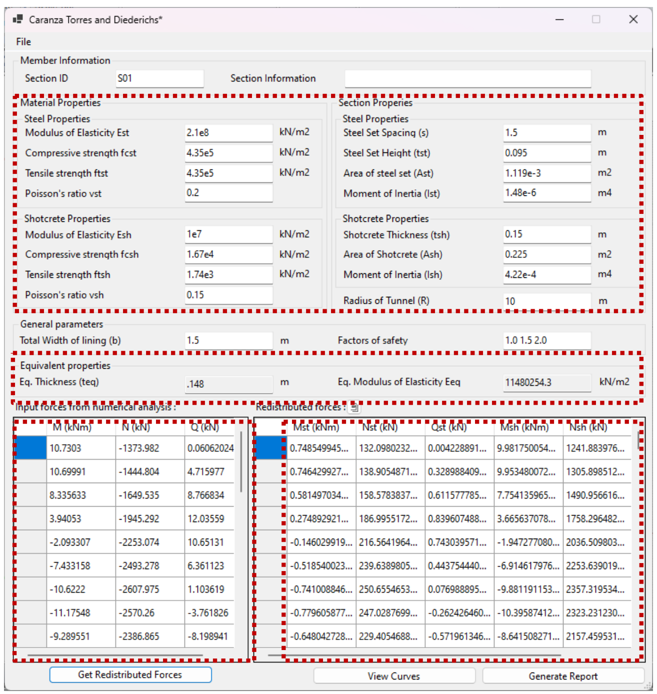


2.4 Tunnel Lining Plots

Tunnel designers commonly use the Carranza-Torres and Diederichs (2009) method to check the capacity of composite linings (steel sets embedded in shotcrete). This method calculates Equivalent Section properties and draws Demand-Capacity plots (M-N & Q-N) separately for Steel Sets/Ribs/Lattice Girders and Shotcrete, based on analysis results of Bending Moment (M), Shear Force (Q), and Axial/Hoop Force (N).

Tools > Options > Tunnel Lining Plots

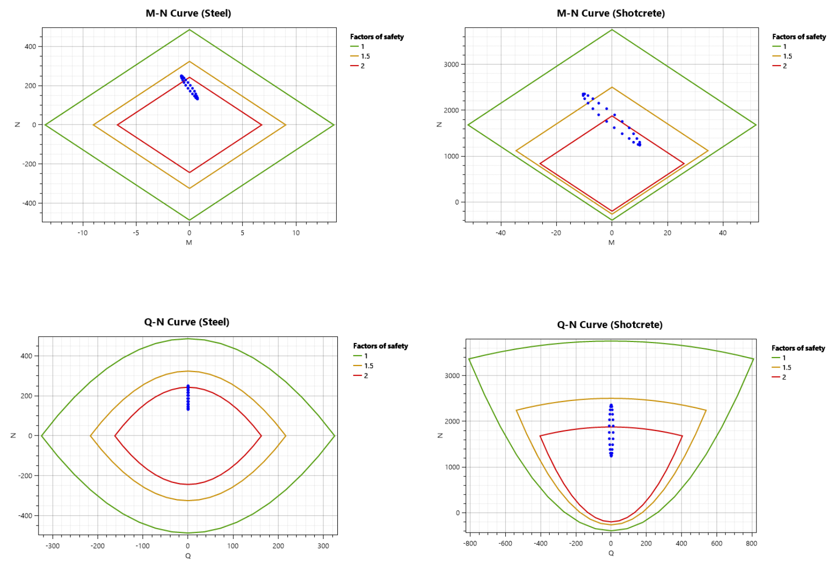




Shotcrete and Steel Section Properties as Input

Equivalent Section Properties

M, N & Q Inputs from Analysis Results

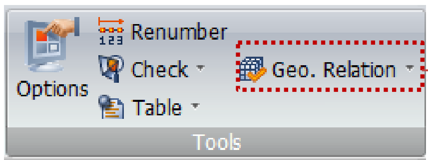


[Redistribution of forces to Steel and Shotcrete]

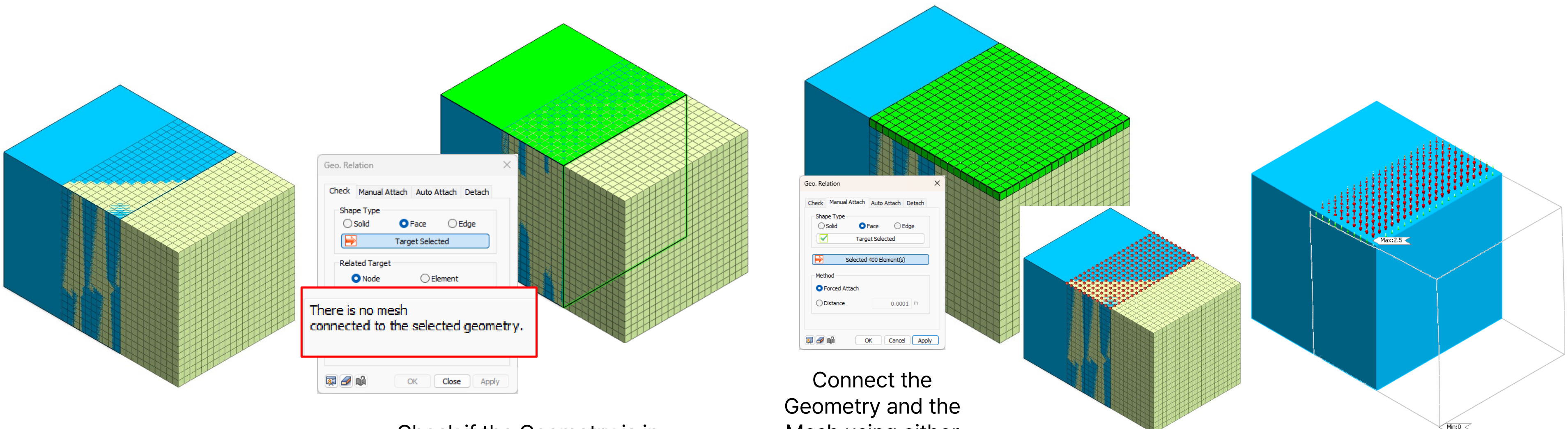
2.5 Geometry and Mesh Connection (Geo-Relation)

In earlier versions, moving or deleting geometric shapes before extracting sub-shapes from the meshed geometry could disrupt the geometry-mesh connection, necessitating mesh regeneration. However, in GTS NX 2024v1.1, users can automatically reconnect using manual editing or tolerance ranges. This enhancement streamlines tasks like load assignment and element extraction.

Mesh > Tools > Geo-Relation



- **Check** : Verify Geometry-Mesh Connectivity
- **Manual Attach** : Manually Connect the Target Geometry
- **Auto Attach** : Automatically connect the target shape (including sub shapes) within a tolerance
- **Detach** : Disconnect geometry mesh connection



Geometry and Mesh are Over Lapping

Check if the Geometry is in connection with Mesh or using 'Check'

Connect the Geometry and the Mesh using either 'Manual Attach' or 'Auto Attach'

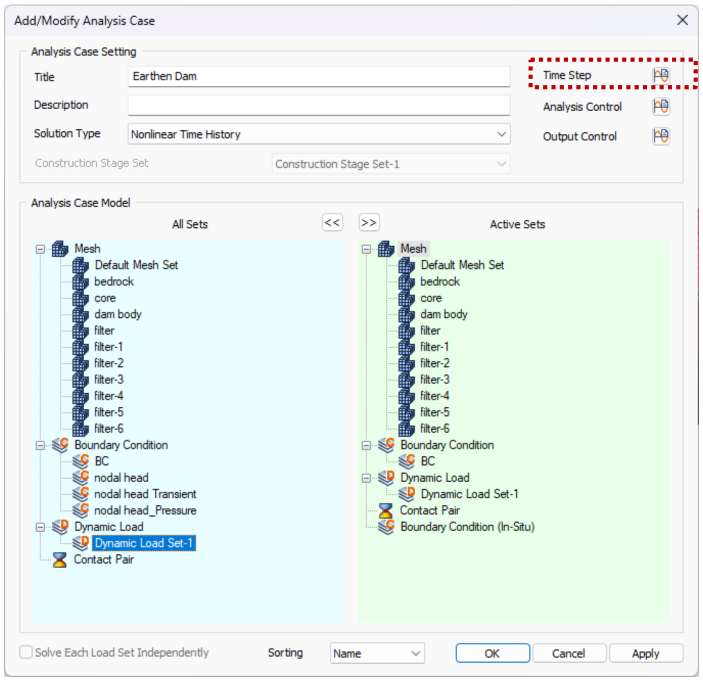
Check the connectivity

Applied Load in the Analysis

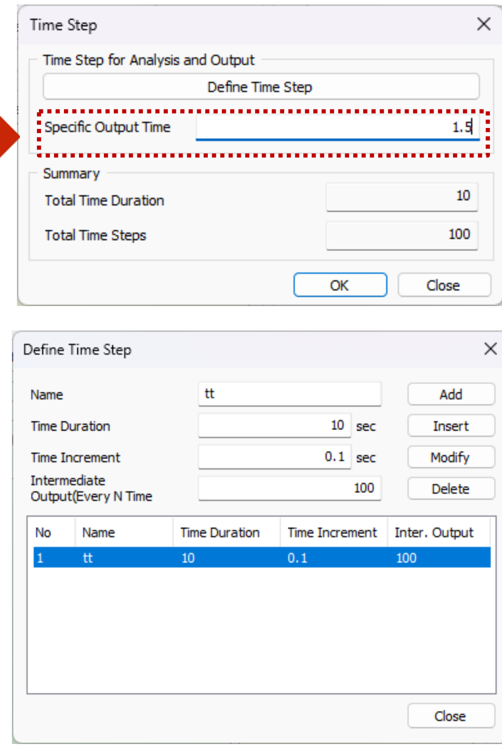
2.6 Random Setting of Dynamic Analysis Output Time

Previously, when defining time steps, results were only output at the times set for intermediate results. However, a new feature has been added to allow results to be output at specific times. For example, if the time interval is set to 0.01 seconds and the intermediate results output is set to 100, results are output every 1 second. Now, by entering the desired specific times for result output, additional result items can also be output at those specified times.

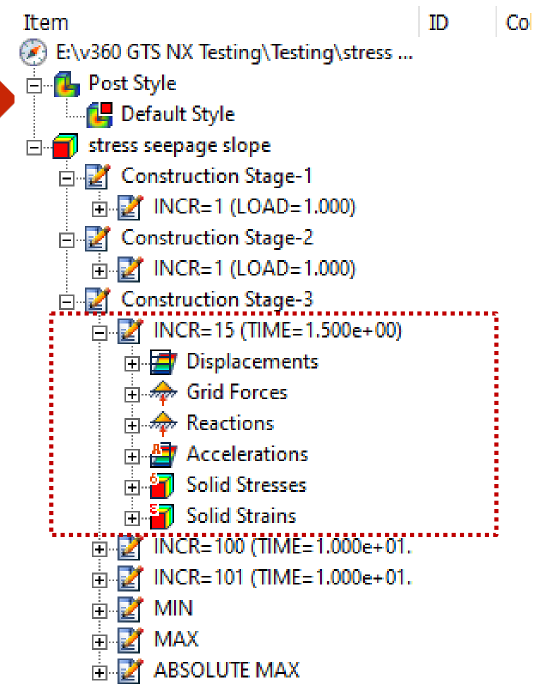
Analysis > Analysis Case > General > Time History Analysis > Analysis Control > Specific Output Time



[Analysis Case Definition]



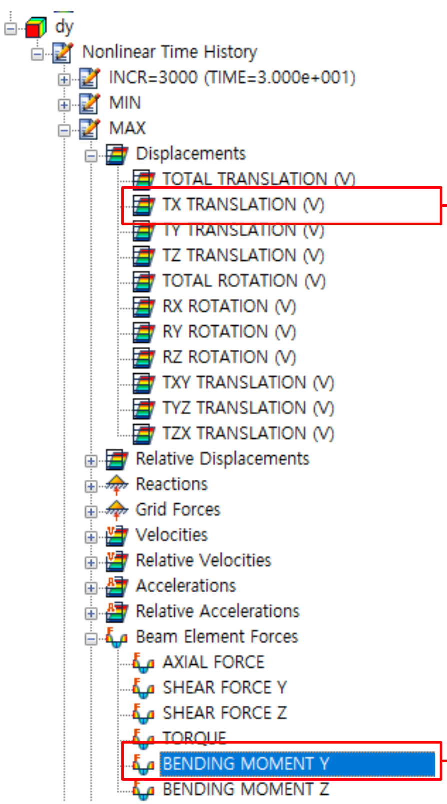
[Time Steps Definition]



2.7 Dynamic Analysis Min/Max value occurrence time output

Now the users can be able to find the Time of Occurrence of the Min/Max/Abs Max results at each node.

Results Tree > MIN, MAX, ABSOLUTE MAX (Occurrence time output)



Node	TX TRANSLATION (V) (m)	
	Value	Time (sec)
1	3.480e-001	5.510e+000
2	3.480e-001	5.510e+000
3	3.480e-001	5.510e+000
4	3.480e-001	5.520e+000
5	3.482e-001	5.520e+000
6	3.483e-001	5.520e+000
7	3.487e-001	5.530e+000
8	3.494e-001	5.540e+000
9	3.504e-001	5.550e+000
10	3.518e-001	5.560e+000
11	3.562e-001	5.580e+000
12	3.634e-001	5.620e+000
13	3.617e-001	5.650e+000
14	3.270e-001	5.690e+000
15	4.405e-001	2.105e+001
16	4.887e-001	2.104e+001
17	3.480e-001	5.510e+000

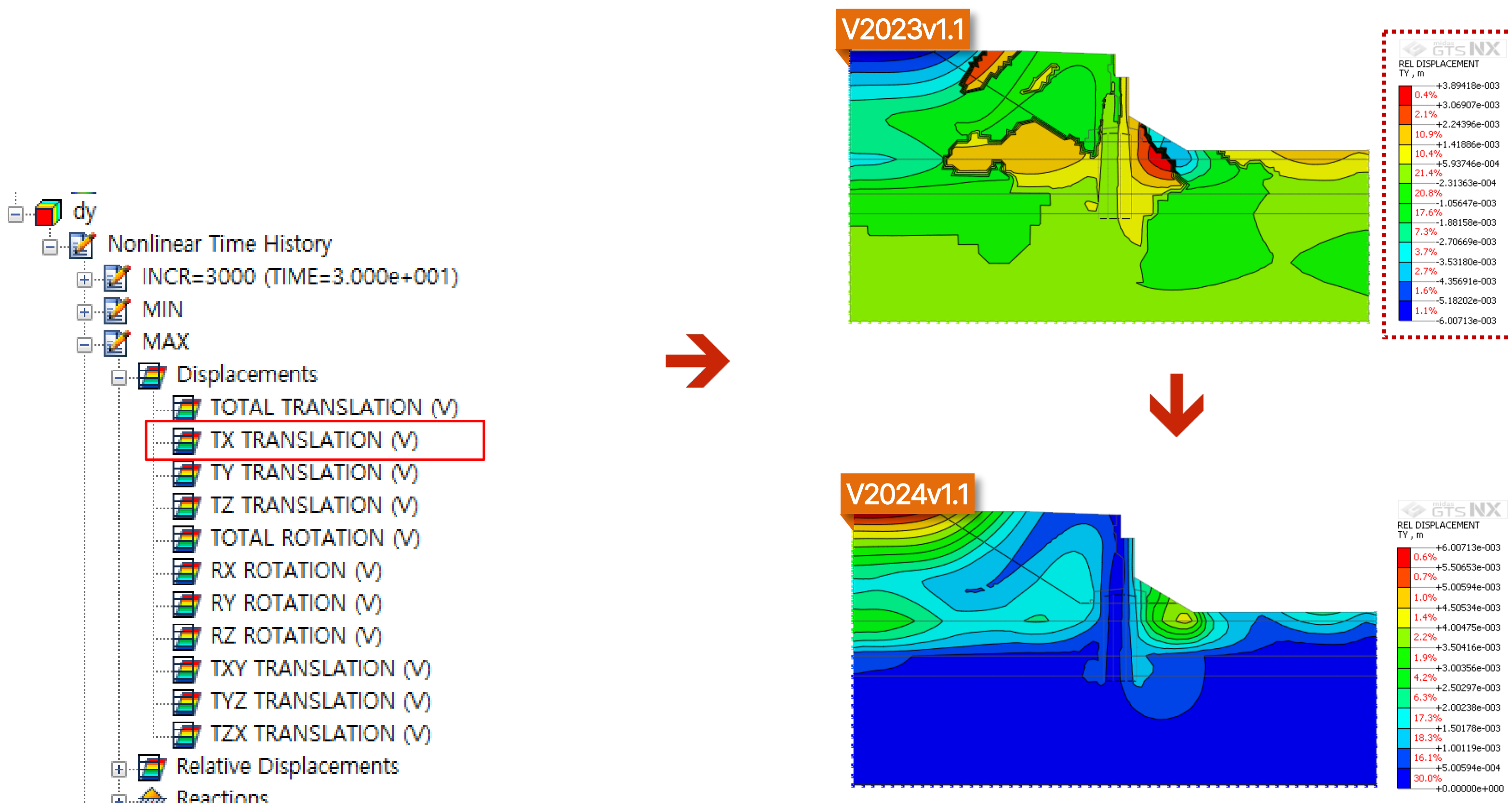
No.	BENDING MOMENT Y 0/4 (kN·m)		BENDING MOMENT Y 1/4 (kN·m)		BENDING MOMENT Y 1/4 (kN·m)		BENDING MOMENT Y 2/4 (kN·m)	
	Value	Time (sec)	Value	Time (sec)	Value	Time (sec)	Value	Time (sec)
1	4.561e+004	6.350e+000	4.510e+004	6.350e+000	4.510e+004	6.350e+000	4.459e+004	6.350e+000
2	4.351e+004	6.350e+000	4.299e+004	6.350e+000	4.299e+004	6.350e+000	4.247e+004	6.350e+000
3	4.138e+004	6.360e+000	4.087e+004	6.360e+000	4.087e+004	6.360e+000	4.036e+004	6.360e+000
4	3.927e+004	6.360e+000	3.875e+004	6.360e+000	3.875e+004	6.360e+000	3.822e+004	6.360e+000
5	3.711e+004	6.360e+000	3.657e+004	6.370e+000	3.657e+004	6.370e+000	3.607e+004	6.370e+000
6	3.495e+004	6.370e+000	3.388e+004	6.370e+000	3.388e+004	6.370e+000	3.281e+004	6.380e+000
7	3.067e+004	6.380e+000	2.965e+004	6.390e+000	2.965e+004	6.390e+000	2.868e+004	6.390e+000
8	2.662e+004	6.410e+000	2.576e+004	6.410e+000	2.576e+004	6.410e+000	2.494e+004	6.420e+000
9	2.316e+004	6.430e+000	2.242e+004	6.430e+000	2.242e+004	6.430e+000	2.177e+004	6.440e+000
10	2.040e+004	6.470e+000	1.956e+004	6.480e+000	1.956e+004	6.480e+000	1.921e+004	6.500e+000
11	1.896e+004	6.560e+000	2.031e+004	6.700e+000	2.031e+004	6.700e+000	2.273e+004	6.700e+000
12	2.758e+004	6.700e+000	2.861e+004	6.700e+000	2.861e+004	6.700e+000	2.965e+004	6.700e+000
13	3.189e+004	6.690e+000	3.020e+004	6.680e+000	3.020e+004	6.680e+000	2.858e+004	6.680e+000
14	2.575e+004	6.660e+000	2.068e+004	6.650e+000	2.068e+004	6.650e+000	1.571e+004	6.640e+000
15	6.680e+003	6.060e+000	5.010e+003	6.060e+000	5.010e+003	6.060e+000	3.340e+003	6.060e+000

No.	BENDING MOMENT Y 2/4 (kN·m)		BENDING MOMENT Y 3/4 (kN·m)		BENDING MOMENT Y 3/4 (kN·m)		BENDING MOMENT Y 4/4 (kN·m)	
	Value	Time (sec)	Value	Time (sec)	Value	Time (sec)	Value	Time (sec)
1	4.459e+004	6.350e+000	4.409e+004	6.350e+000	4.409e+004	6.350e+000	4.358e+004	6.350e+000
2	4.247e+004	6.350e+000	4.194e+004	6.350e+000	4.194e+004	6.350e+000	4.145e+004	6.360e+000
3	4.036e+004	6.360e+000	3.986e+004	6.360e+000	3.986e+004	6.360e+000	3.935e+004	6.360e+000
4	3.822e+004	6.360e+000	3.770e+004	6.360e+000	3.770e+004	6.360e+000	3.717e+004	6.360e+000
5	3.607e+004	6.370e+000	3.556e+004	6.370e+000	3.556e+004	6.370e+000	3.505e+004	6.370e+000
6	3.281e+004	6.380e+000	3.182e+004	6.380e+000	3.182e+004	6.380e+000	3.083e+004	6.380e+000
7	2.868e+004	6.390e+000	2.776e+004	6.400e+000	2.776e+004	6.400e+000	2.689e+004	6.400e+000
8	2.494e+004	6.420e+000	2.418e+004	6.420e+000	2.418e+004	6.420e+000	2.342e+004	6.420e+000
9	2.177e+004	6.440e+000	2.121e+004	6.450e+000	2.121e+004	6.450e+000	2.072e+004	6.460e+000
10	1.921e+004	6.500e+000	1.915e+004	6.510e+000	1.915e+004	6.510e+000	1.936e+004	6.540e+000
11	2.273e+004	6.700e+000	2.516e+004	6.700e+000	2.516e+004	6.700e+000	2.758e+004	6.700e+000
12	2.965e+004	6.700e+000	3.074e+004	6.690e+000	3.074e+004	6.690e+000	3.183e+004	6.690e+000
13	2.858e+004	6.680e+000	2.706e+004	6.670e+000	2.706e+004	6.670e+000	2.566e+004	6.660e+000
14	1.571e+004	6.640e+000	1.110e+004	6.060e+000	1.110e+004	6.060e+000	6.601e+003	6.070e+000
15	3.340e+003	6.060e+000	1.670e+003	6.060e+000	1.670e+003	6.060e+000	1.140e-009	8.690e+000

2.8 Improve Dynamic Analysis ABSOLUTE MAX(Absolute value output)

Previously, the ABSOLUTE MAX results displayed the actual values after considering the signs, based on absolute value comparisons across the entire time period. However, we have now changed it to display the absolute values directly, to facilitate consistent variability analysis when reviewing ABSOLUTE MAX results.

Results Tree > ABSOLUTE MAX

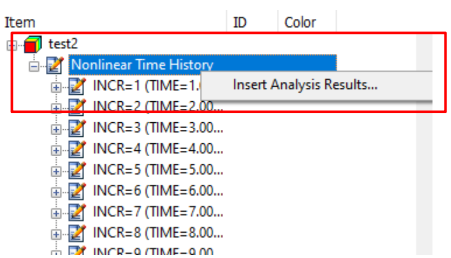


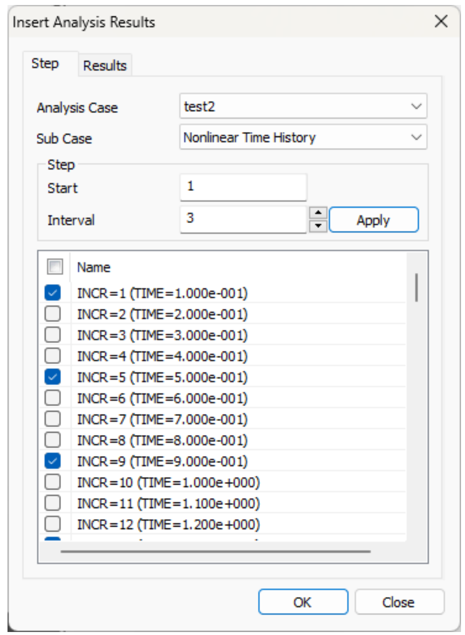
[Positive/Negative Result → Change Output format(ABS)]

2.9 Customization of Results Display

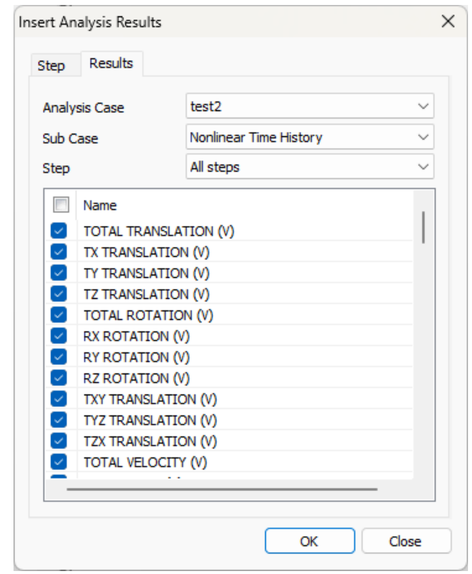
Previously, loading results in dynamic analysis or construction stage analysis could be time-consuming, especially with numerous time steps or stages. In the new version, users can select specific parts of the output results to display, ensuring faster output speed in models with many large steps and stages, like nonlinear time history analysis or construction stage analysis.

Results Tree > Analysis Case > Analysis





[Select the Steps to be seen using 'Interval']



[Select the Results to be shown in Respective Steps]

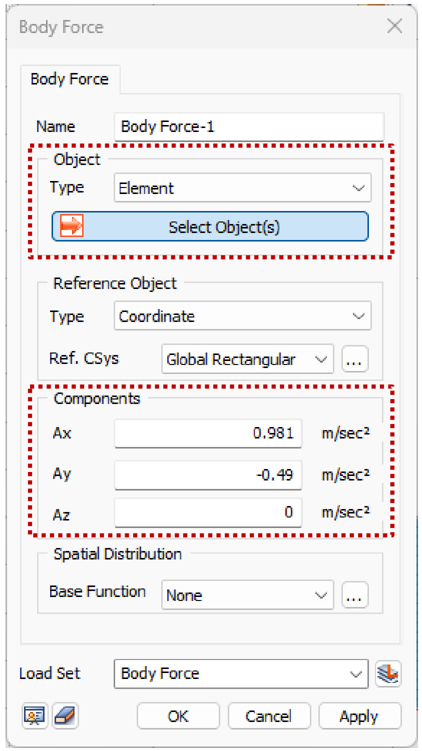
2.10 Body Force

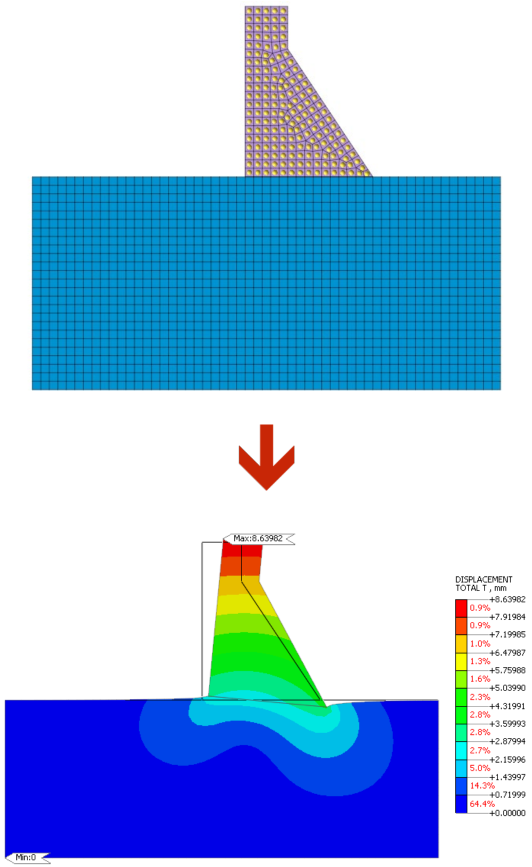
A new load set is introduced to assign the accelerations (pseudo static loads) for respective Elements/Mesh Sets.
In the case of Pseudo Static Loads, user needs to input the Accelerations directly (seismic coefficients*acceleration due to gravity) in the body force definition.

Static/Slope Analysis > Load > Define Body Force

Select the Elements/Mesh Sets to with Body force needs to be applied

Input the Acceleration Components.
For Pseudo Static Load.
Component = Seismic Coeff * 9.81 m/sec^2

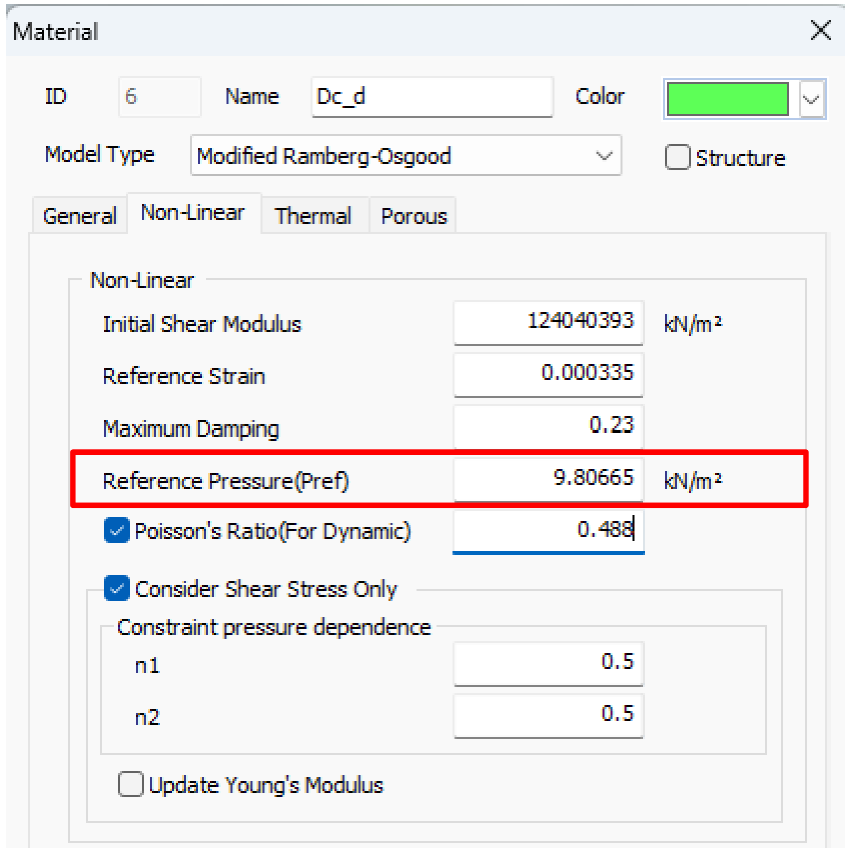




2.11 HD/RO/GHE-S Function (Confining Pressure)

An input item has been added to allow input of the standard confining pressure. Under the standard confining pressure used in the 3-axis compression experiment You can directly enter shear stiffness and reference strain rate or by using the Material Evaluator.

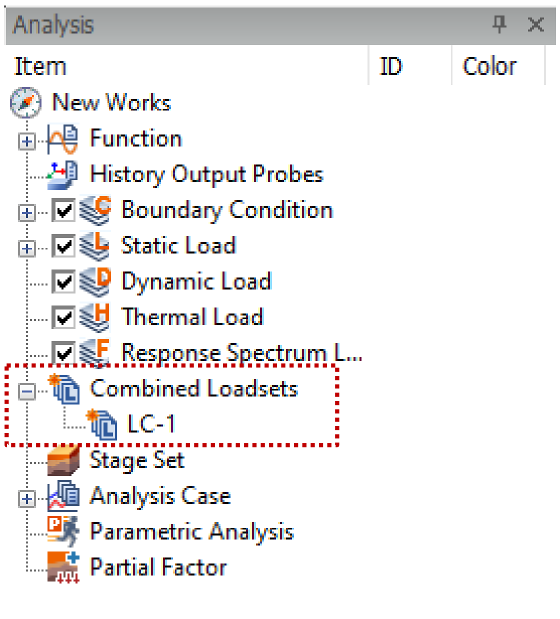
Mesh > Material > Isotropic > Modified Ramberg-Osgood > Nonlinear



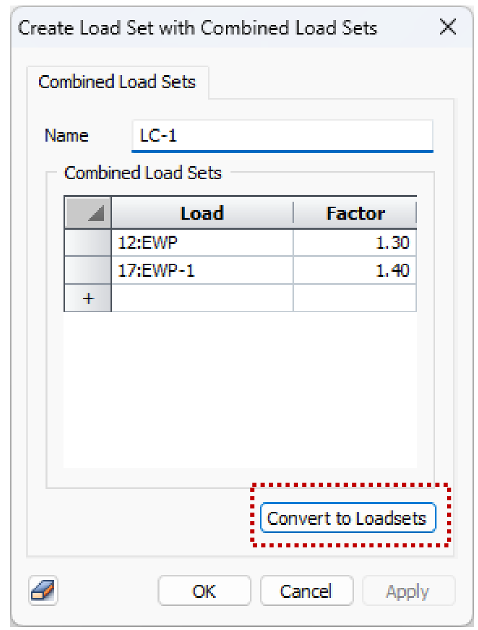
2.12 Accessing the Load Combination & Convert to Loadsets

Previously, it was tough to access the generated Load Combination. Now the user can access the generated load combination and corresponding load factors used. In addition, you can convert the Load Combination into a Load Sets

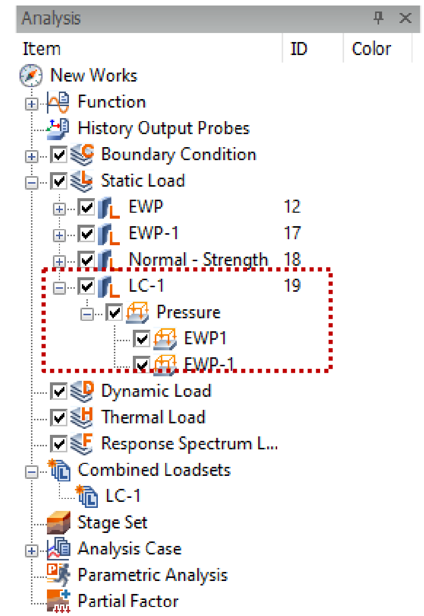
Analysis Workstree > Combined Loadsets



[Accessing the defined Load Combination]



[Converting to Loadsets]



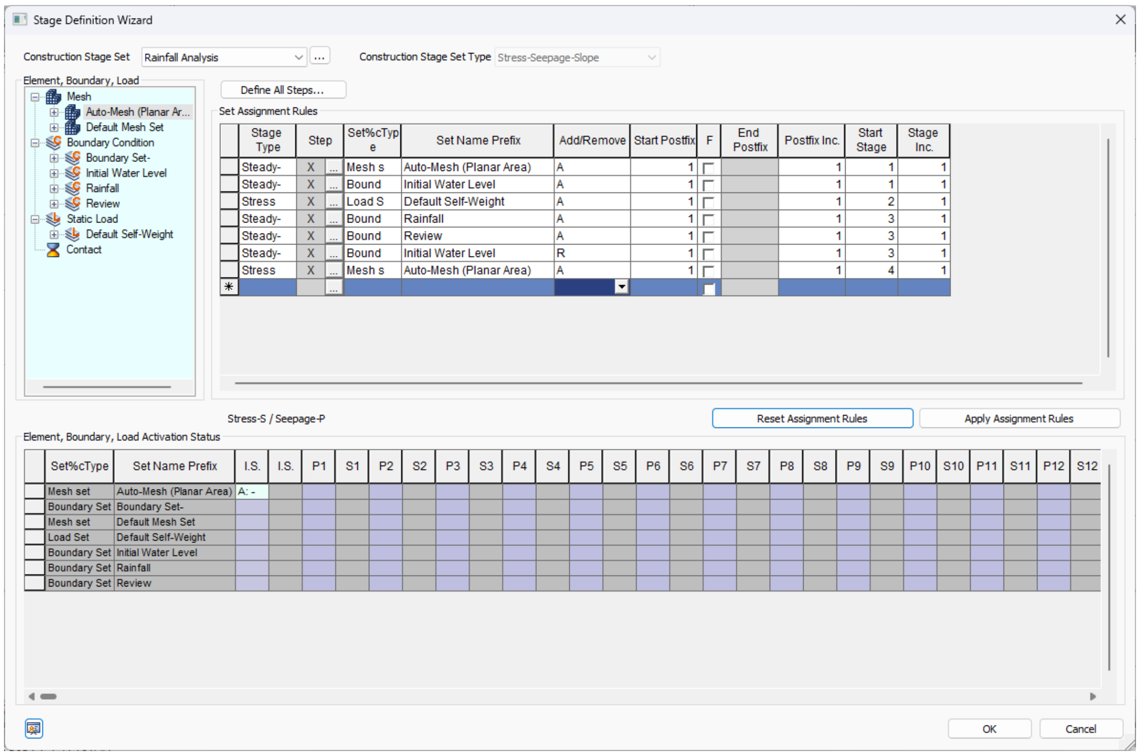
[Converted Loadsets]

※ 'Load Combination Set' which is not converted to a 'Load Set' can also be used as a 'Load Set' in the Analysis

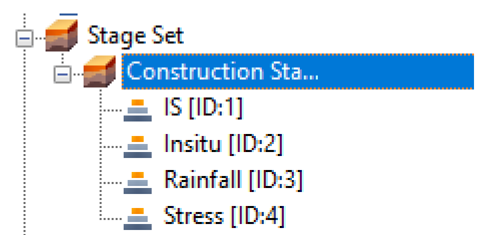
2.13 Construction Stage Wizard Function Improvement

Previously, the construction stage wizard was limited to single-type analysis. Now, it supports configuring stages for coupled Seepage-Stress unidirectional analyses. Sequential definition is possible for infiltration and stress stages; other cases require separate modifications in the construction stage set.

Static/Slope/Seepage/Consolidation Analysis > Construction Stage > Stage Wizard



[Construction Stage Wizard]

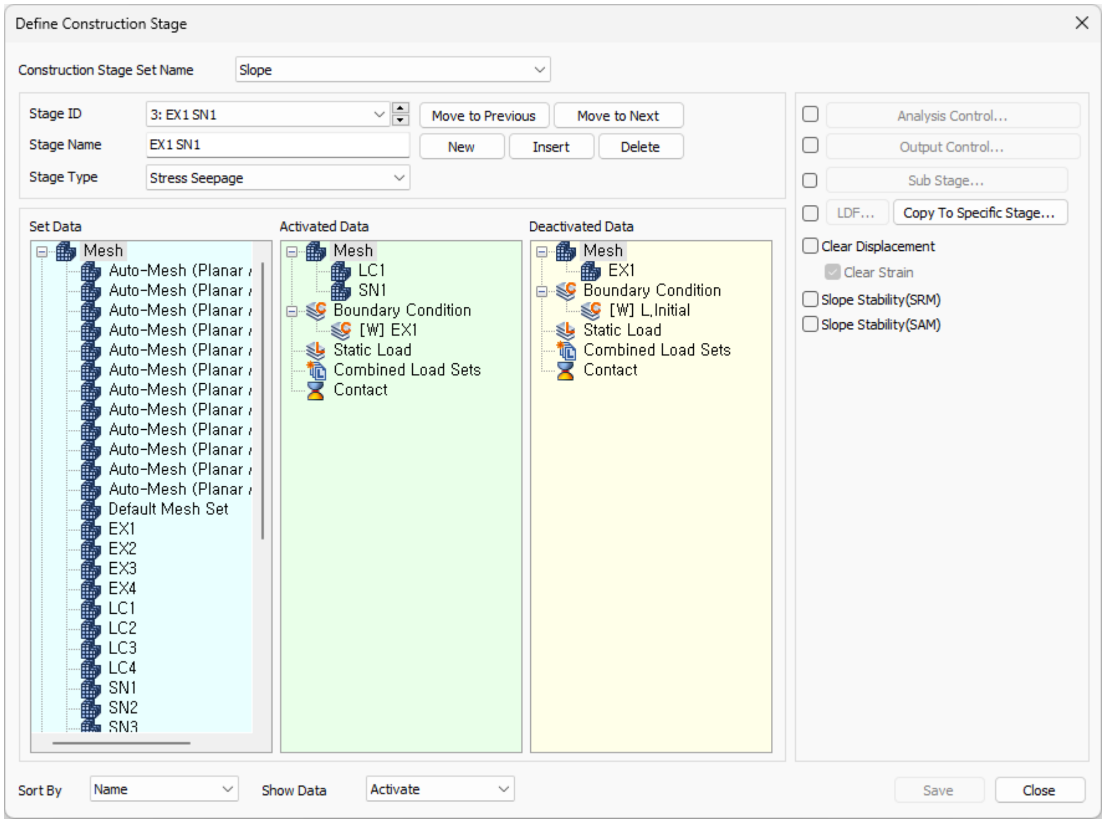


[Construction Stage Set]

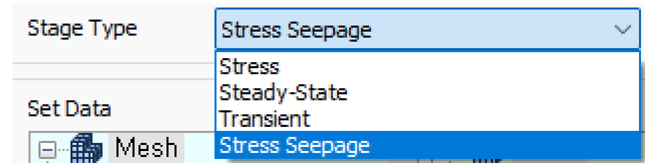
2.14 Additional Construction Stage Type

Previously, the construction stage for semi-coupled analysis considering seepage and stress required defining the seepage and stress stages separately. However, a new functionality has been added that allows the construction stages to be easily configured using the 'stress seepage' stage type, which defines both seepage and stress stages in the same window.

Static/Slope/Seepage/Consolidation Analysis > Construction Stage > Stage Set > Stage Type [Stress-Seepage-Slope] > Stage Type [Stress Seepage]



[Construction Stage Window]

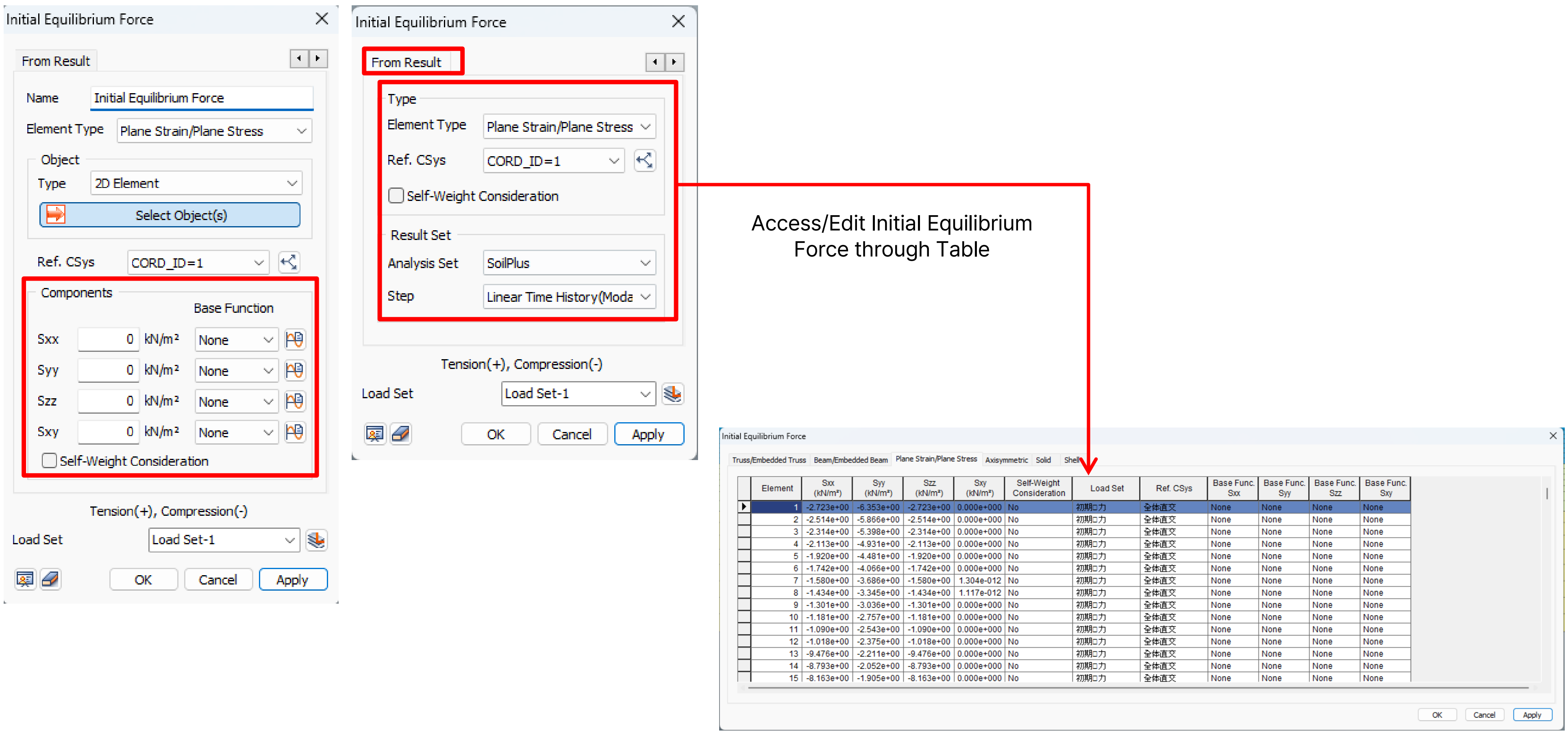


[Stage Type 'Stress Seepage']

2.15 Initial Equilibrium Force and Initial Stress Table Functions

Now, initial equilibrium forces for different elements (truss/embedded truss, beam/embedded beam, plane strain/plane stress, axisymmetric, solid, shell) can be automatically generated from analyzed results. Previously, users manually input these forces, but now they're generated from analysis results. Moreover, static analysis results (stress, internal forces) can be set as initial conditions for dynamic analysis, facilitating dynamic analysis based on these initial conditions.

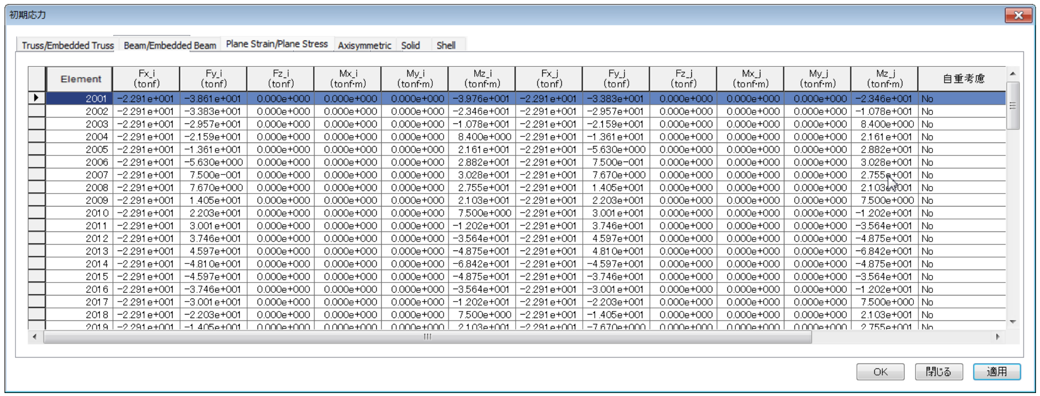
Static Analysis > Static Load > Initial Equilibrium Force/Initial Equilibrium Force Table



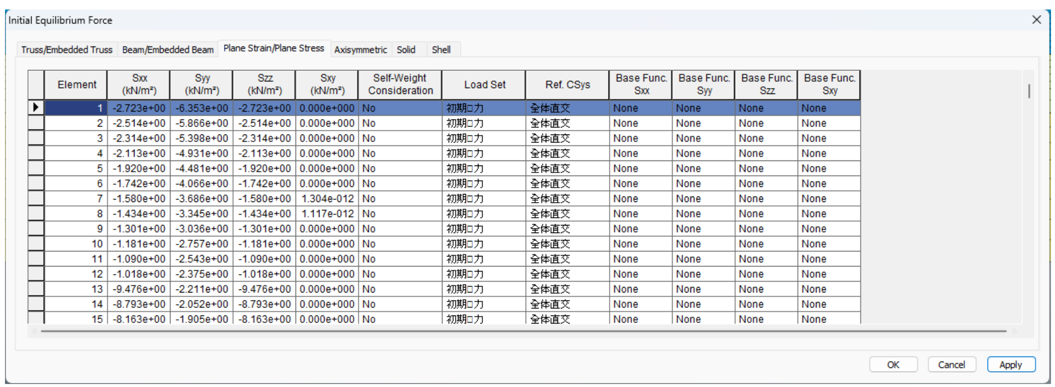
2.15 Initial Equilibrium Force and Initial Stress Table Functions

Now, initial equilibrium forces for different elements (truss/embedded truss, beam/embedded beam, plane strain/plane stress, axisymmetric, solid, shell) can be automatically generated from analyzed results. Previously, users manually input these forces, but now they're generated from analysis results. Moreover, static analysis results (stress, internal forces) can be set as initial conditions for dynamic analysis, facilitating dynamic analysis based on these initial conditions.

Static Analysis > Static Load > Initial Equilibrium Force



[Initial Equilibrium Force of Beam Element]

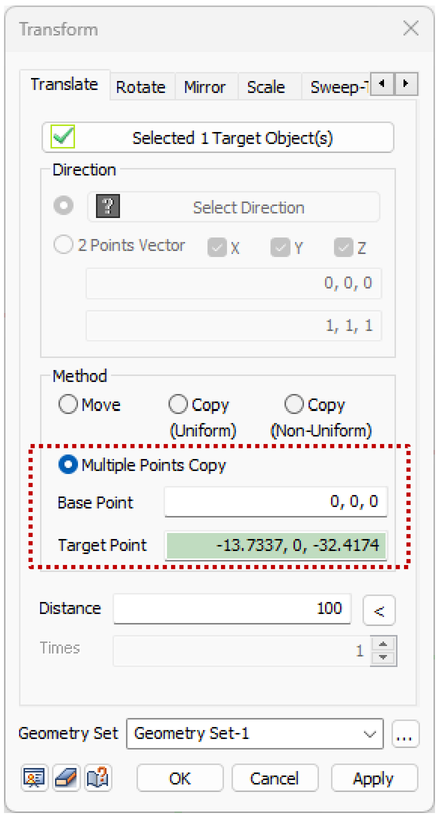


[Initial Stress of Plane Strain/Plane Stress Element]

2.16 Multiple Copy Objects Relative to Base Point

User can now be able to copy the Geometry Multiple times to different locations using the ‘Multiple Points Copy’ option.

Geometry > Transform > Translate > Multiple Points Copy

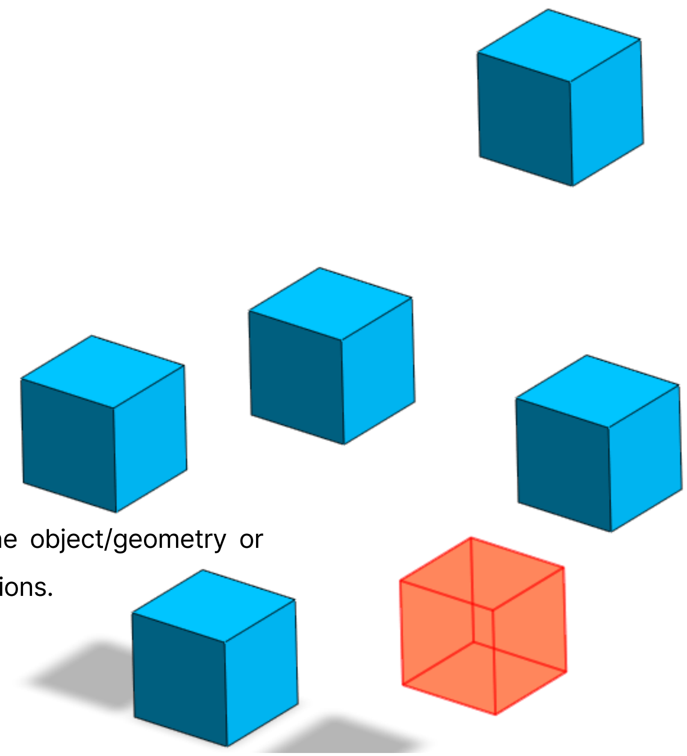


Base Point:

Select the Base Point to copy the object

Target Point:

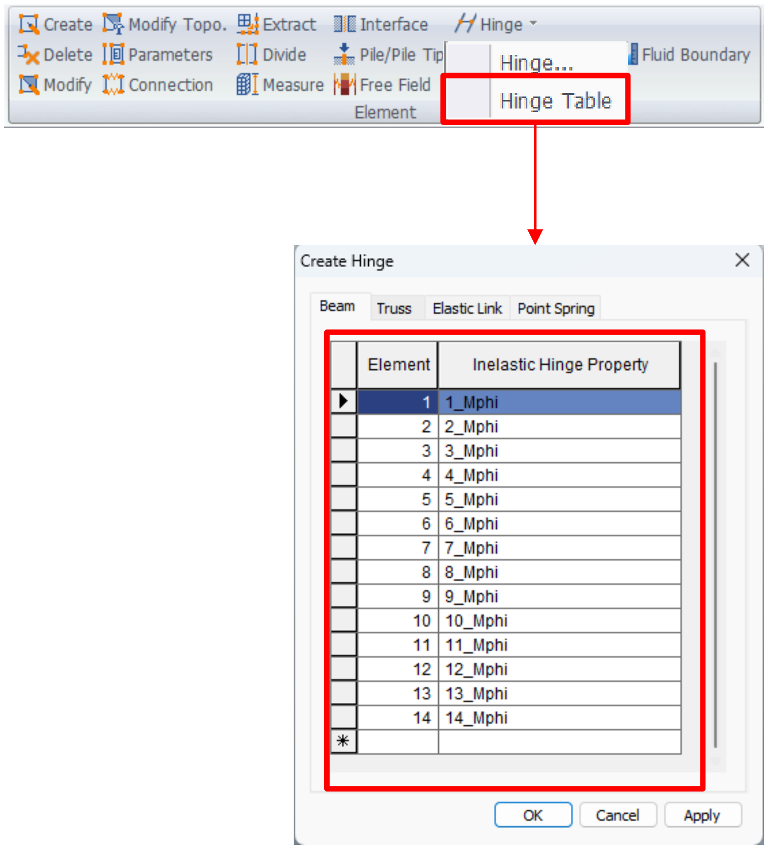
Either **manually** input the coordinate to copy the object/geometry or use the **cursor** to copy the object to multiple locations.



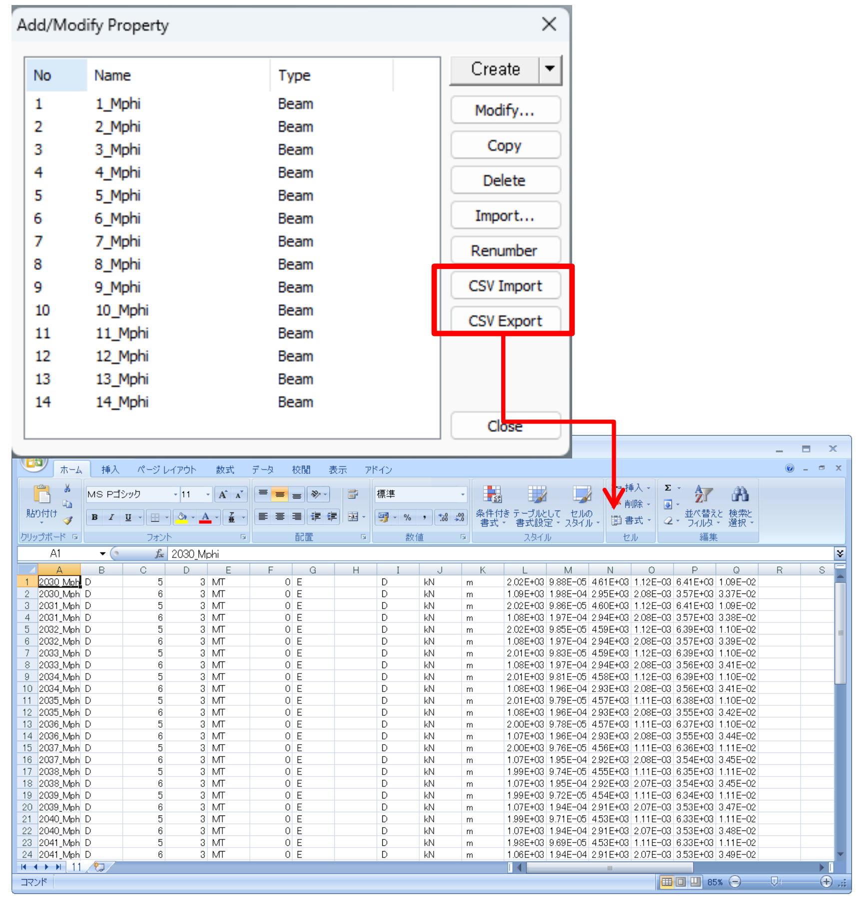
2.17 Hinge (M-Φ Data) Assign Table

When assigning inelastic hinge properties (M-Φ) to structural elements, it was previously necessary to repetitively set these properties for each element when dealing with many structural members. This process has been improved with a new feature that allows users to easily assign hinge properties through a table. Additionally, a feature has been added to facilitate the import and export of hinge property files from a CSV file when defining hinge properties.

Mesh > Element > Hinge Table



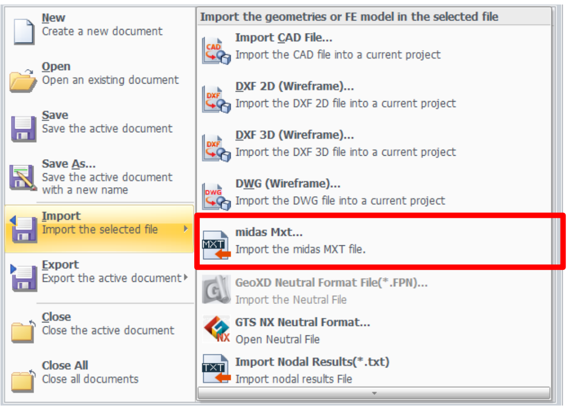
Mesh > Prop./Csys./Func. > Hinge > Hinge Properties

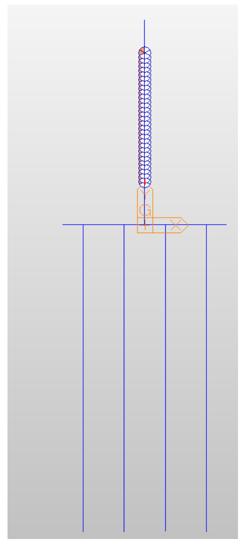


2.18 Midas Civil Inelastic Hinge Data

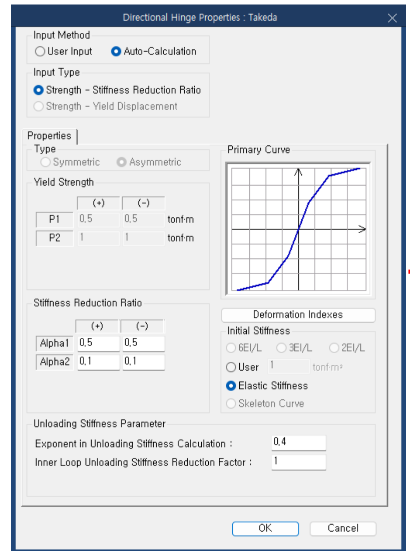
Previously, the inelastic hinge data assigned to elements in Midas Civil could not be imported into GTS NX via the .mxt format. Now, the user can import the inelastic hinge data into GTS NX using mxt format and proceed with nonlinear analysis involving soil continuum and structural elements.

File > Import > midas Mxt



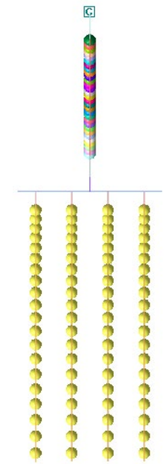


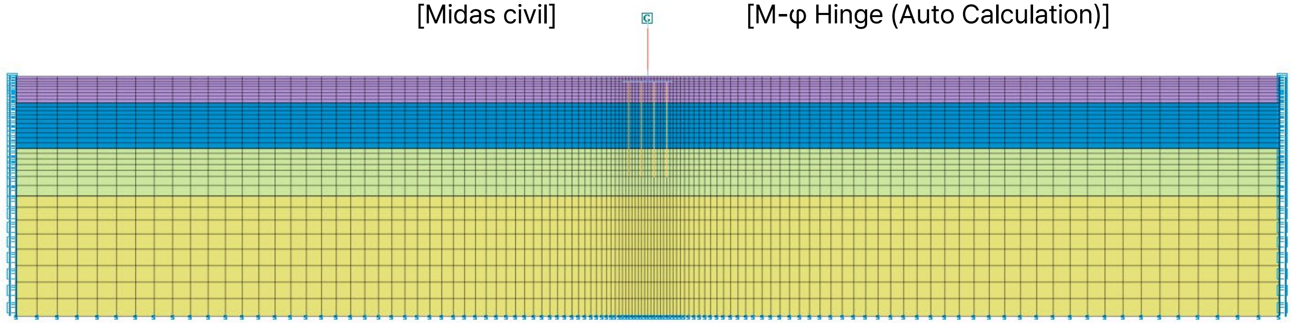
[Midas civil]



[M-φ Hinge (Auto Calculation)]

∗.mxt



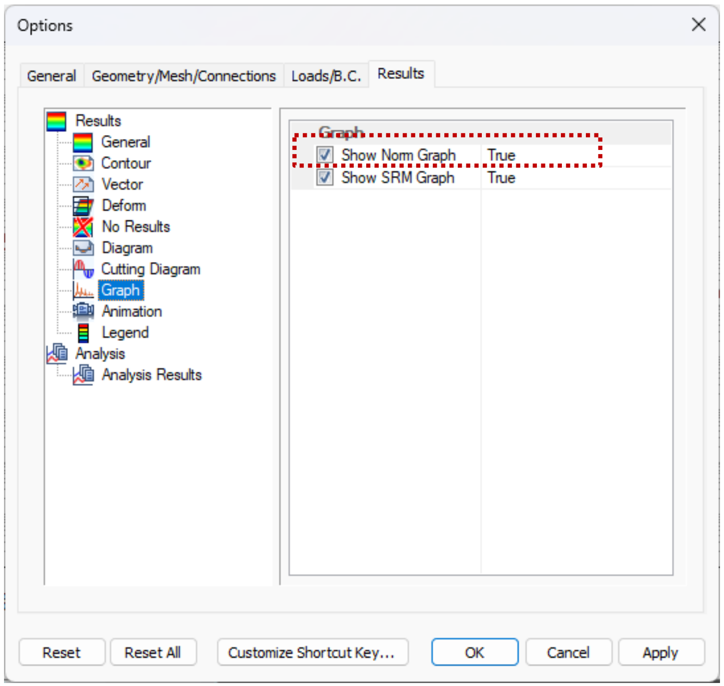


[Structure + Ground Analysis]

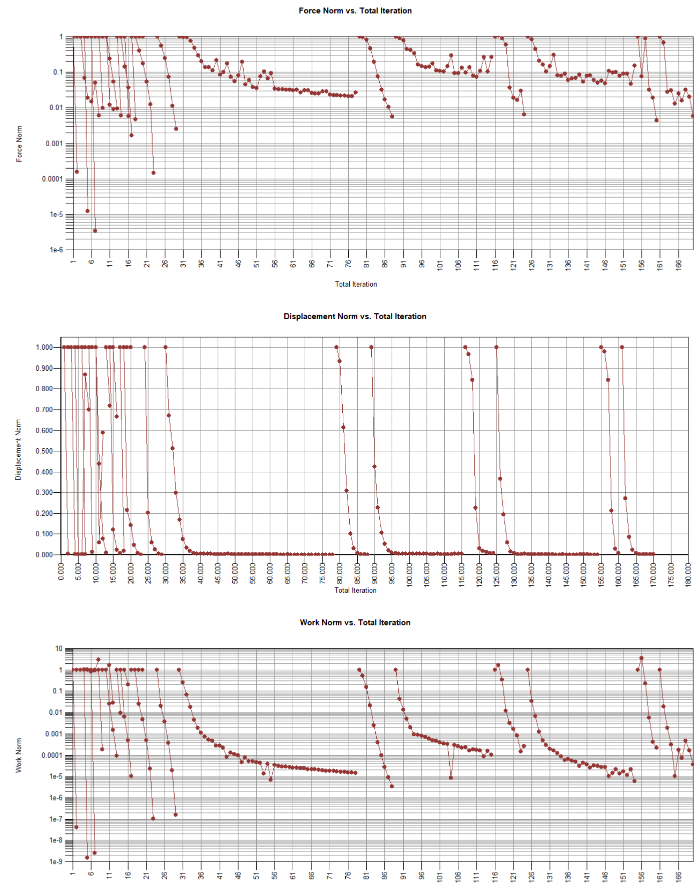
2.19 Analysis Log Visualization

In this version, user can be able to visualize the Work/Load/Displacement Norm vs Iteration graphically to better understand the convergence and divergence in the solution during the analysis.

Tools > Options > Tools > Results > Results > Graph > Show Norm Graph



[Options]



[Norm vs Total Iteration]

2.20 Nastran file Export

In the new version, a function has been added to export the GTS NX Model into a Nastran Input File.

File > Export > Export Nastran Input File

2.21 Default Self-Weight

When creating a new model, the system has been updated to automatically register self-weight according to the analysis settings (2D/3D).

Analysis Works Tree > Static Load > Default Self Weight

2.22 High Resolution Support

The previously optimized GUI for FHD (1920x1080 pixels) has been enhanced to support 4K (3840x2160 pixels) resolution. The interface, function icons, and text now scale according to the Windows user scaling settings.

Thank you for being a part of our journey.
Let's achieve more together!