Release Note

Release Date : April. 2026

Product Ver. : FEA NX 2026 (v1.1)

ADVANCED NONLINEAR AND DETAIL ANALYSIS

New Paradigm in Advanced Structural Analysis

Enhancements

Analysis

1.1 Optimization Analysis

1.2 NX Interact - Automated Soil Structure Interaction between FEA NX and CIVIL NX

Pre/Post Processing

2.1 Wave Absorbent Surface for Sloshing Fluid Medium

2.2 Addition of Normal Stiffness vs Depth for Pile Interface

2.3 Automatic Update of Interface Parameters

2.4 Enhancement in Virtual Beam Generation

2.5 Stagewise Relative Displacement

2.6 Structural Design of Elements via CIVIL NX/GEN NX (SAR Format)

2.7 Report Options: New Tunnel Analysis Report Type

2.8 Report Options: Improved Orbit Review Functionality

2.9 Improved Application of Observation-point Images and Type Verification

2.10 Addition of a New Double I-Section Property

2.11 Fixed Geometry Set for 'Intersect' Function

2.12 Enhancement in Load Table Import

2.13 Enhancement of Sweep-Ortho Function

2.14 Excel Compatibility (Copy & Paste) in Train Dynamic Load Table

2.15 Enhancement in Selection of Function Data for Modification

2.16 Damping constants in Elastic Link

2.17 Radiation Condition

2.18 Temperature-Dependent Unit Volume Weight

2.19 Elastic Modulus Function: Addition of JSCE (2022) Standards

2.20 Heat Source Function as per JSCE-2022

2.21 Steel Section DB Addition – Pacific (SI) (AU/NZ Standards)

2.22 Direct Acceleration Excitation

2.23 Soil Test Optimization

2.24 Jointed Rock Mass SRM

1.1 Optimization Analysis

- **Size optimization** based on approximate/surrogate modeling: extract design of experimental points (DOEs) → build a surrogate/approximate model → solve for optimization → derive solution.
- Optimization algorithm: genetic algorithm / direct method provided
- Possible design variables (4 types): elastic modulus (E), Poisson's ratio (v), cohesion (C), and angle of internal friction (Φ) → suited for model types elastic/Mohr-Coulomb (model type and design variables will be expanded).
- Linear and nonlinear stress analysis (including construction stage analysis).
- Automatic derivation of optimal design variables within user-specified input conditions (constraints).
- Provides correlation analysis results of design variables for each soil layer.

Definition of Optimal Design

Deriving the best design plan (design variables) that satisfies the original goal by using available means for optimal design.

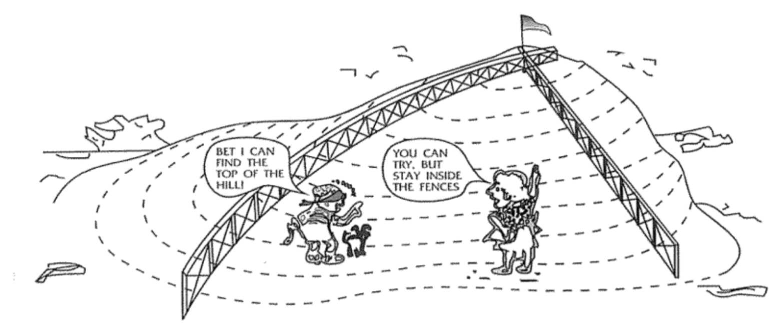
Design Response

Optimization Elements

Design Variable — Value to be Modified Arbitrarily

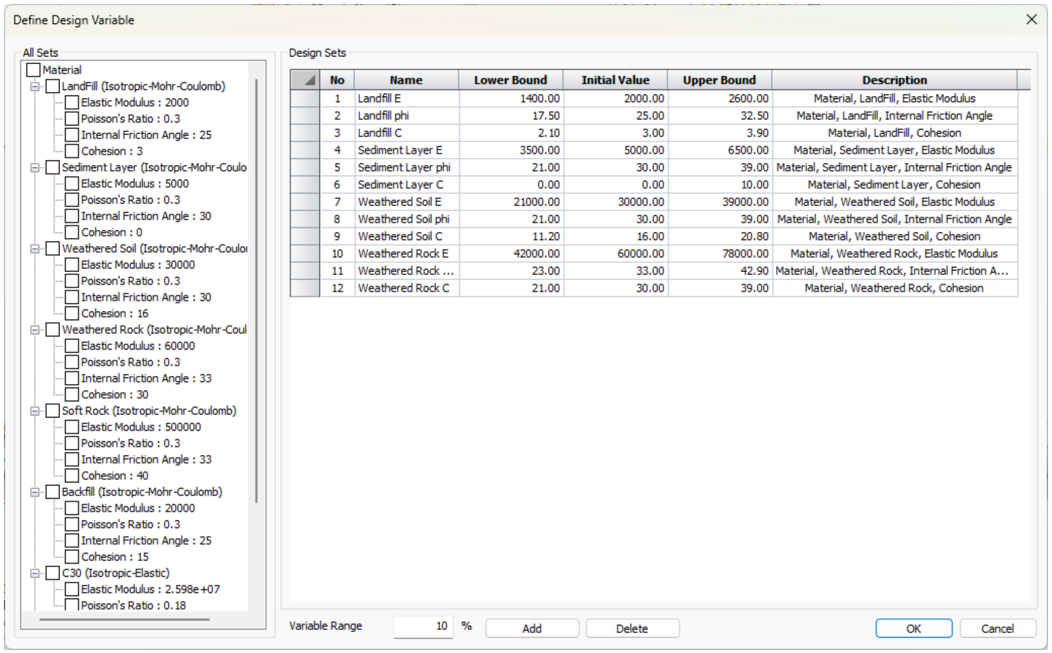
Constraints — The Value You Want to Satisfy

Objectives — Values to be Reduced or Increased



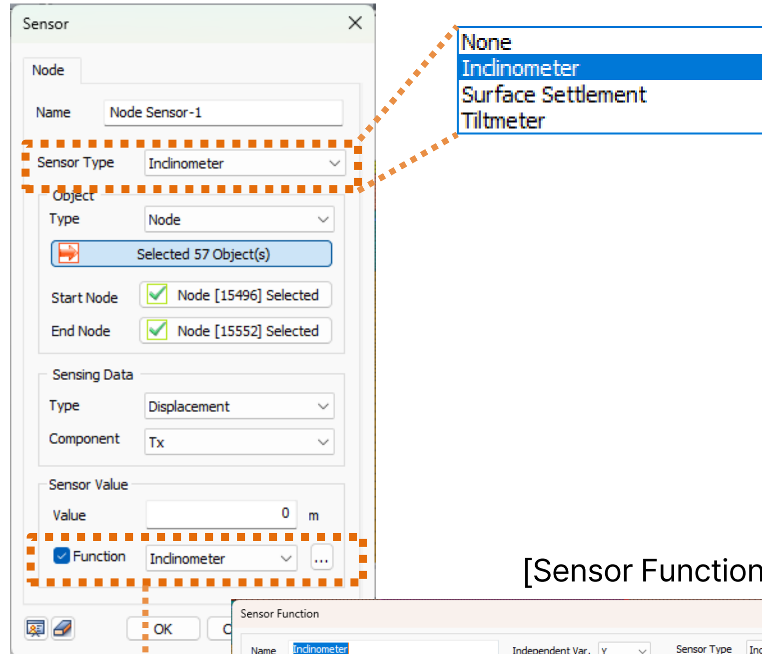
1.1 Optimization Options

Analysis > Optimize > Design Variables

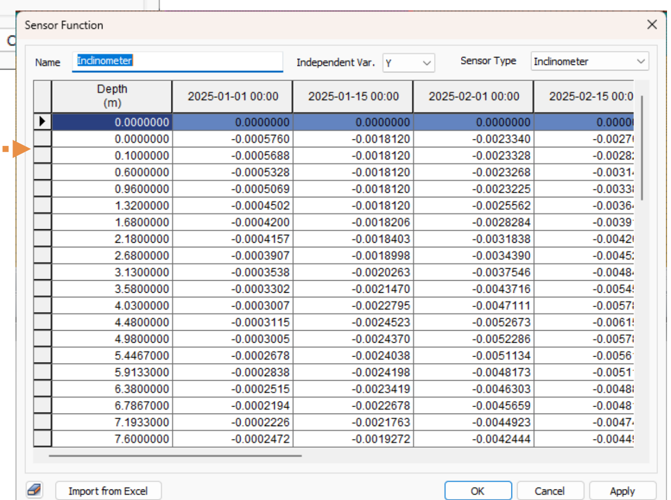


[Design Variable Definition]

Analysis > Optimize > Define Sensor



[Sensor Function]



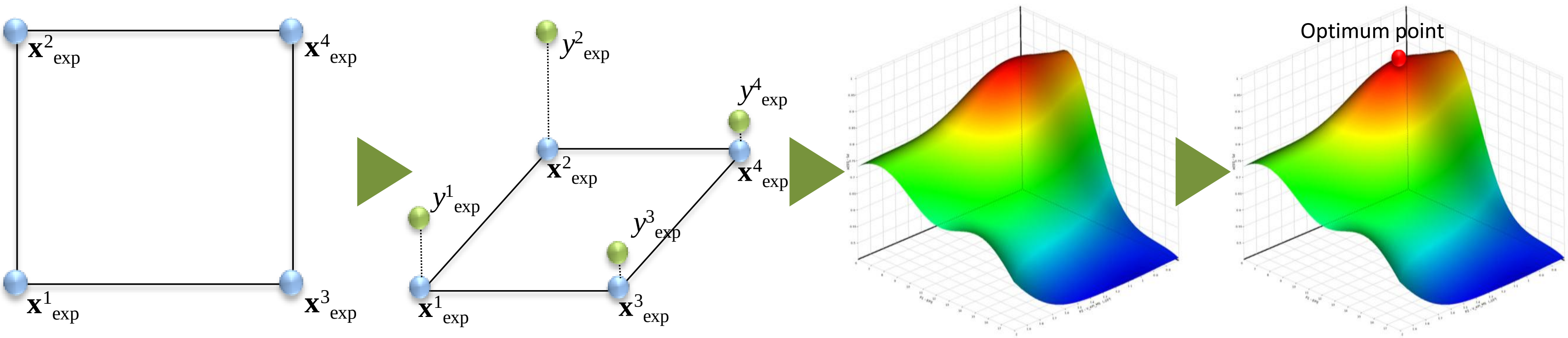
- Original Input Property used in the Analysis – Initial Value.
- It is necessary to define the minimum/maximum values of the design variables.

(Example: Design variable decreases → Displacement result increases and Vice Versa)

1.1 Optimization Options

Surrogate-based Size Optimization

- An optimization technique that replaces the actual model with a surrogate model
- Allows efficient use of analysis with fewer simulations.
- The accuracy of the surrogate model equals the accuracy of the optimum → a precise model is essential.

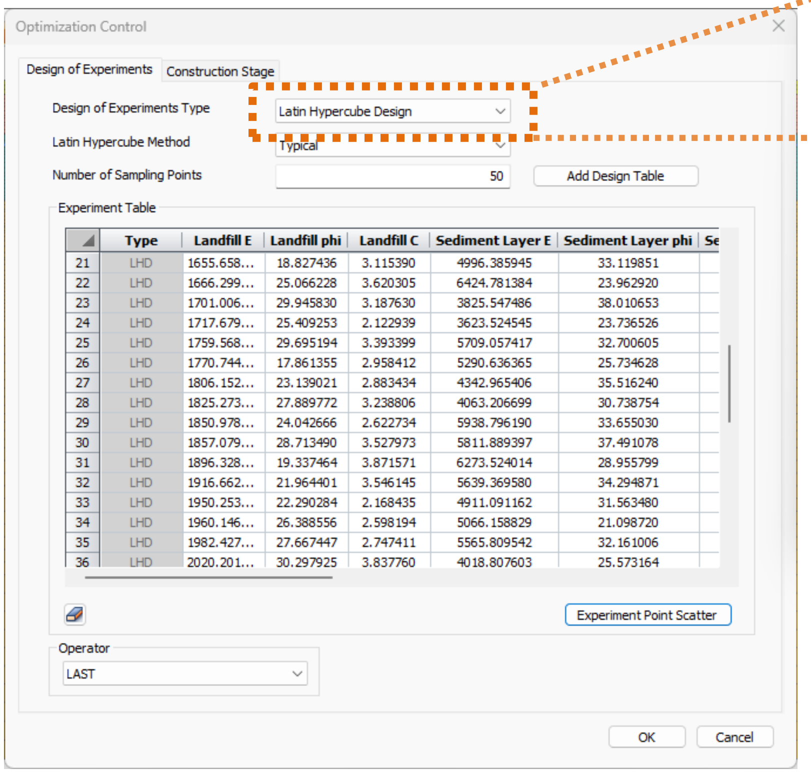


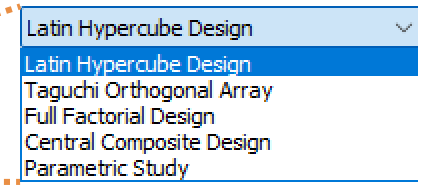
[Design of Experiments (DOE)] [Interpretation] [Build a Surrogate/Approximate Model] [Optimum Point Search]

1.1 Optimization Options

Design of Experiments (DOE)

- Multiple methodologies for determining the combination of design variables.
- The influence of design variables on the response can be evaluated. (For nonlinear analysis: minimum – 10 design variables, recommended – 50 design variables)





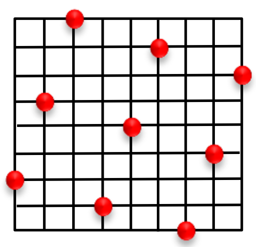
[Optimization Control – Extract Experimental Points]

[Check Experiments Scatter]

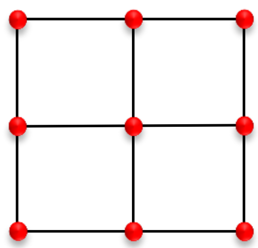
1.1 Optimization Options

Types of Design of Experiments (DOE)

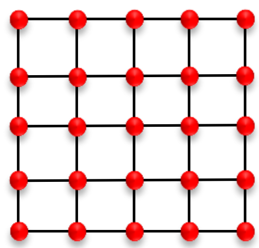
1. Latin Hypercube Design: The design space is divided uniformly, and design points are assigned randomly. It has good space-filling properties, making it widely used for Kriging surrogate models.
2. Taguchi Orthogonal Array: A fractional factorial design method that satisfies the orthogonality of design variables.
3. Full Factorial Design: Explores all possible combinations of the levels of design variables.
4. Central Composite Design: Combines a 2-level fractional factorial design with center points and star points for evaluation.
5. One-dimensional parameter study: Vary one design variable while keeping others fixed to assess its sensitivity.



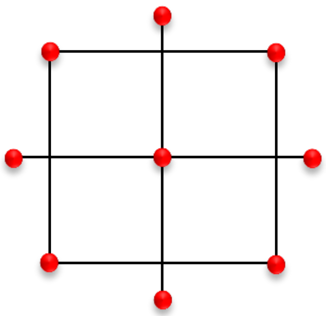
[Latin Hypercube Design (LHD)]



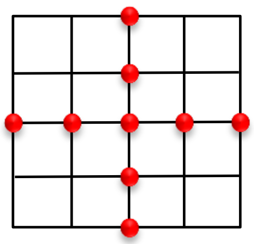
[Taguchi Orthogonal Array]



[Full Factorial Design]



[Central Composite Design]

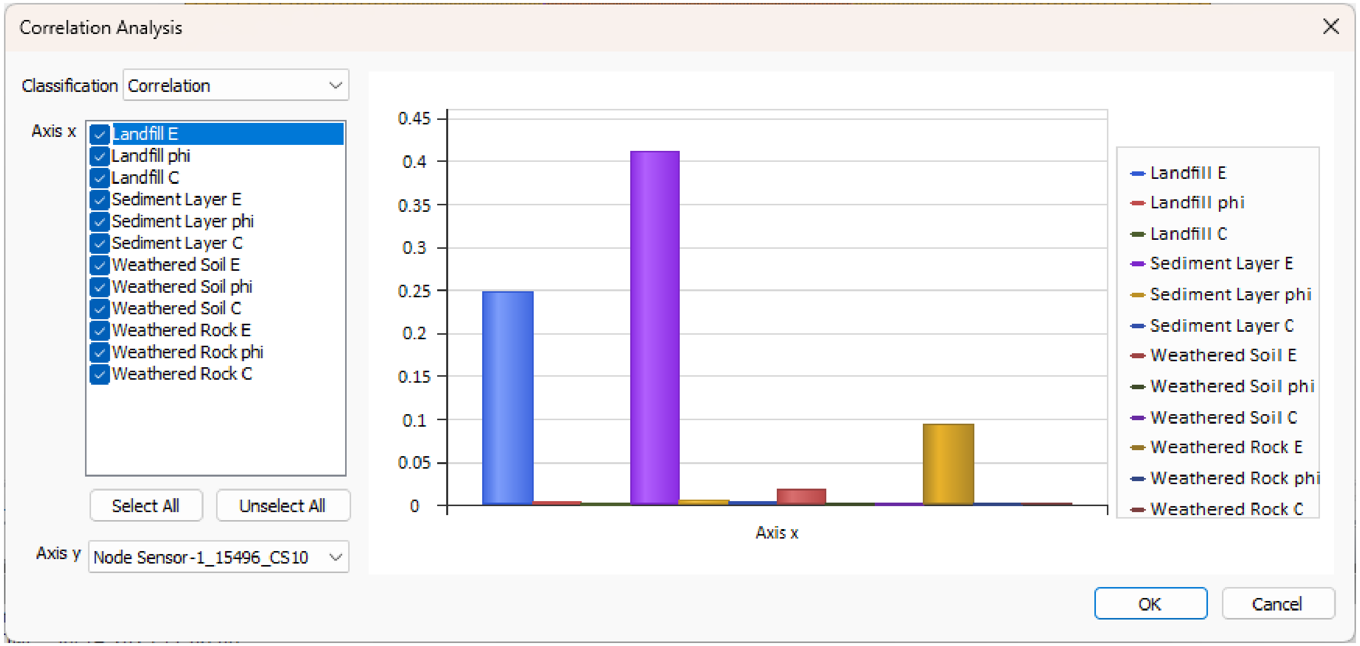


[Parametric Study]

1.1 Optimization Options

Design of Experiments (DOE) – Post Processing

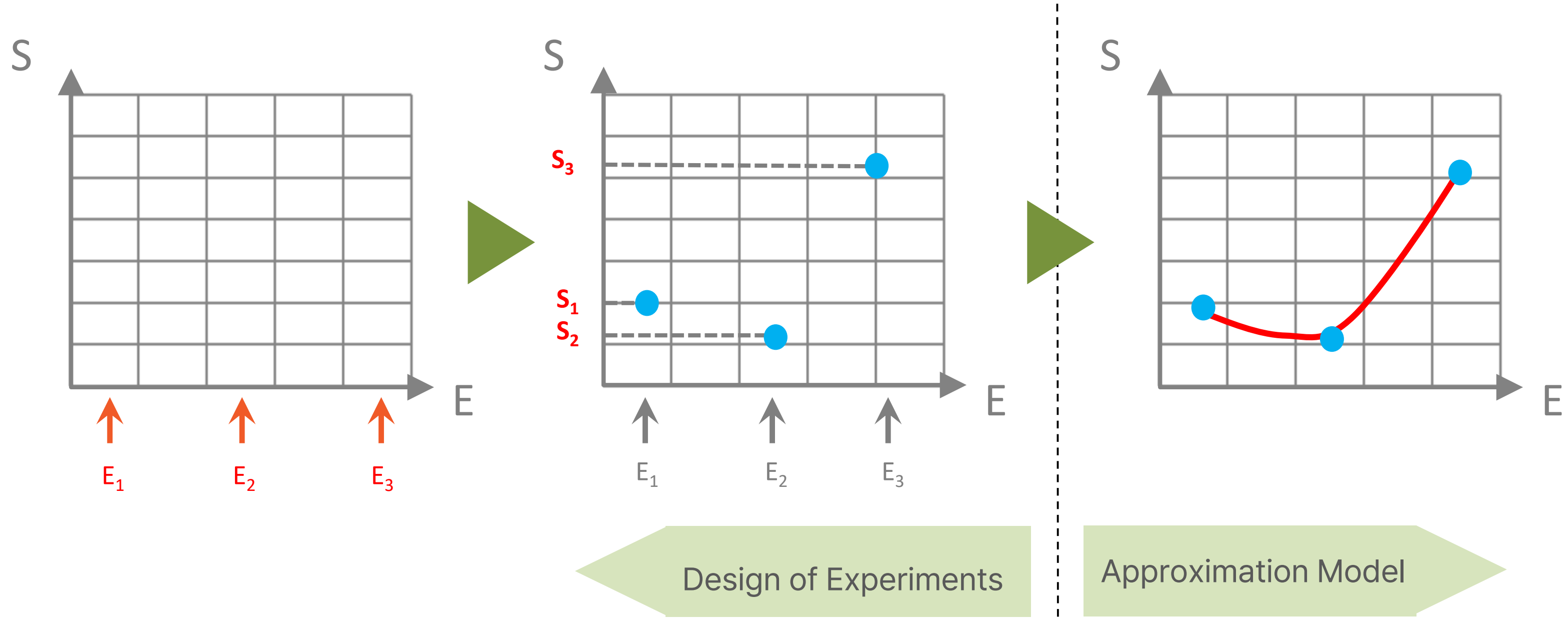
- Correlation Analysis: Determines the importance coefficient of each design variable. Used to assess how each design variable affects the accuracy of the surrogate model
- When there are many design variables, those with correlation values close to 0 can be excluded from the design problem.
- It is possible to identify which design variables have a significant impact on each design response.



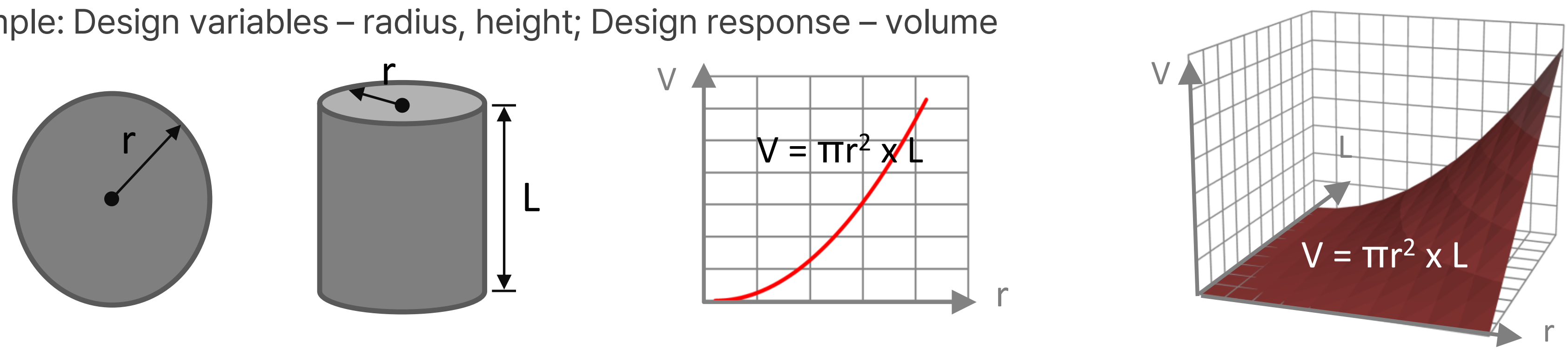
[Correlation Analysis Results]

1.1 Optimization Options

Surrogate/Approximate Model



Example: Design variables – radius, height; Design response – volume



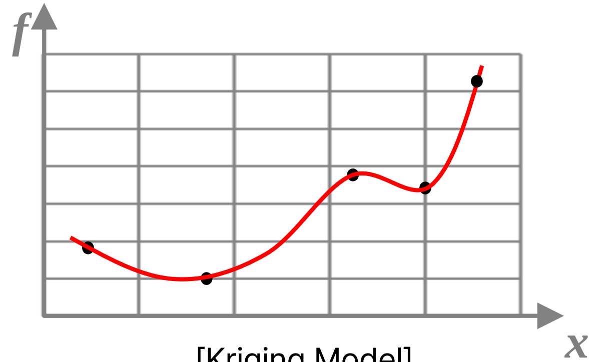
1.1 Optimization Options

Approximate Model Types

1. Kriging Model
   - An interpolation model that passes exactly through the experimental points.
   - Combines a global regression model with local residuals.

     $$y = \mathbf{f}^T(\mathbf{x})\boldsymbol{\beta} + \mathbf{r}^T(\mathbf{x})\mathbf{R}^{-1}(\mathbf{yexp} - \mathbf{F}\boldsymbol{\beta})$$

   - Since the most probable points need to be explored, the numerical cost of generating the surrogate model is high



[Kriging Model]

2. Polynomial Regression Model (Linear / Pure Quadratic / Full Quadratic / Pure Cubic)
   - Surrogate equation that fits the experimental points as closely as possible
   - Linear :

     $$y = \beta_0 + \beta_1 x_1 + \beta_2 x_2$$

   - Pure Quadratic:

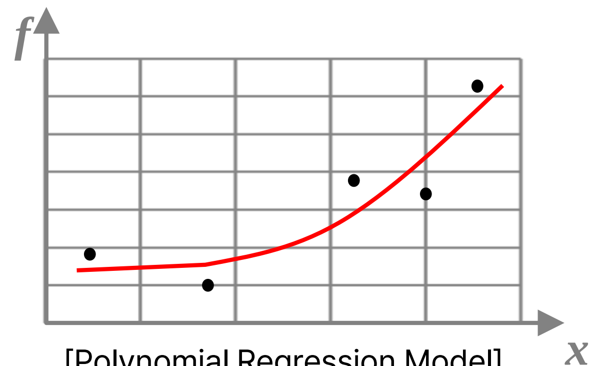
     $$y = \beta_0 + \beta_1 x_1 + \beta_2 x_2 + \beta_3 x_1^2 + \beta_4 x_2^2$$

   - Full Quadratic:

     $$y = \beta_0 + \beta_1 x_1 + \beta_2 x_2 + \beta_3 x_1^2 + \beta_4 x_1 x_2 + \beta_5 x_2^2$$

   - Pure Cubic:

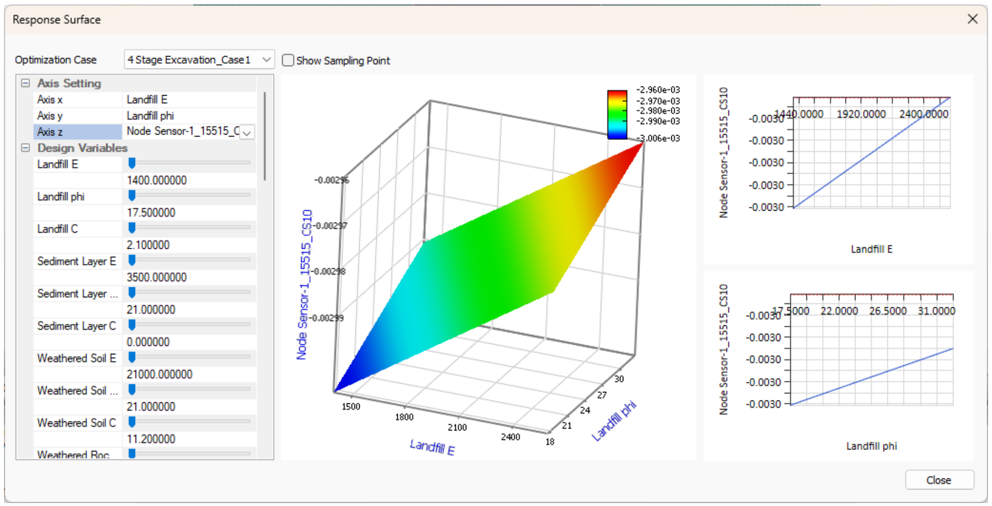
     $$y = \beta_0 + \beta_1 x_1 + \beta_2 x_2 + \beta_3 x_1^2 + \beta_4 x_1 x_2 + \beta_5 x_2^2 + \beta_6 x_1^3 + \beta_7 x_2^3$$



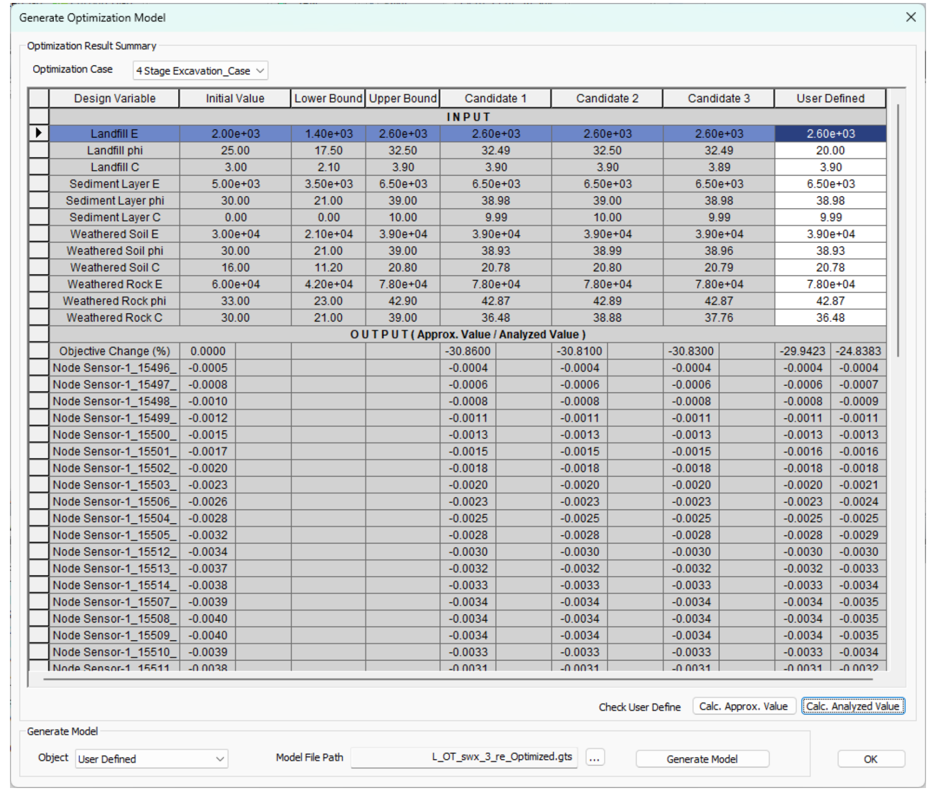
[Polynomial Regression Model]

1.1 Optimization Options

Approximate Model Post Processing



[Approximate Model Shape]

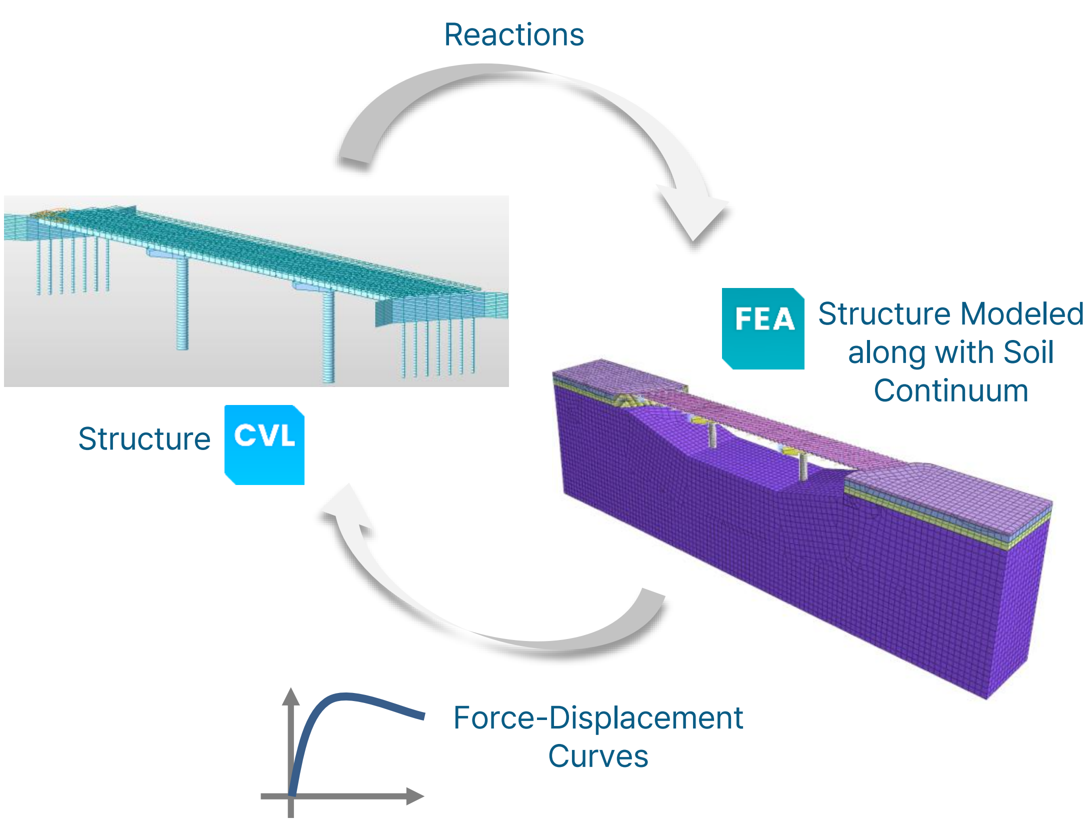


[Optimization Results Summary]

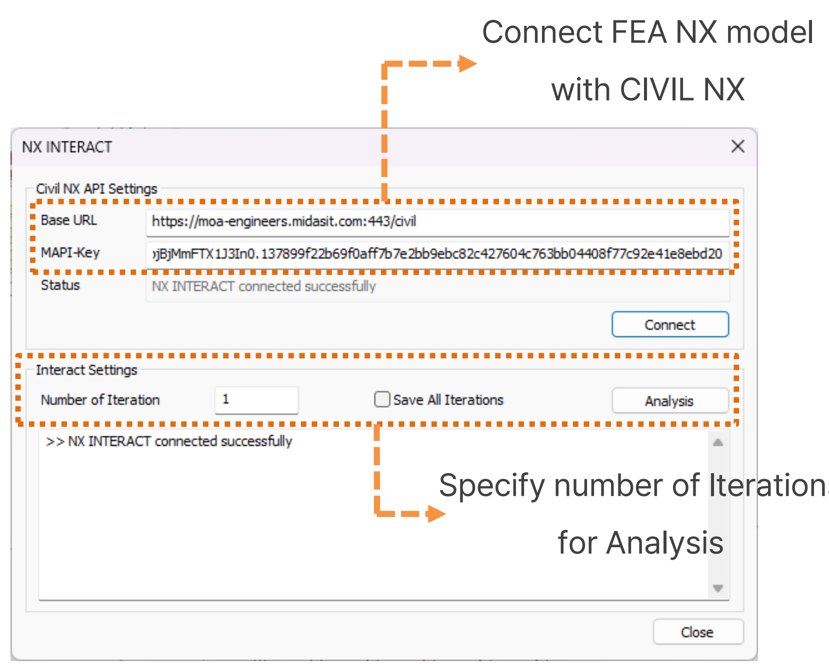
1.2 NX Interact - Automated Soil Structure Interaction between FEA NX and CIVIL NX

- Detailed **soil-structure interaction (SSI)** studies have become essential for stability assessments of critical and heavy structures, such as high-rise buildings, historic structures, and bridges. These studies require seamless interoperability between structural and geotechnical software.
- The **NX Interact** feature provides full interoperability between FEA NX and CIVIL NX/GEN NX. Load combination reactions from CIVIL NX/GEN NX can be imported into FEA NX along with the structural model. A soil continuum is then generated in FEA NX, the required analyses are performed, and the resulting spring data can be exported back to CIVIL NX/GEN NX for further studies (iteration).
- The number of iterations can be specified, and NX Interact automatically runs the analyses, exports loads to FEA NX, and imports the updated soil springs into CIVIL NX/GEN NX, streamlining the iterative process

SSI Iterative Analysis



Tools > Options > NX Interact



[NX Interact Function Window]

2.1 Wave Absorbent Surface for Sloshing Fluid Medium

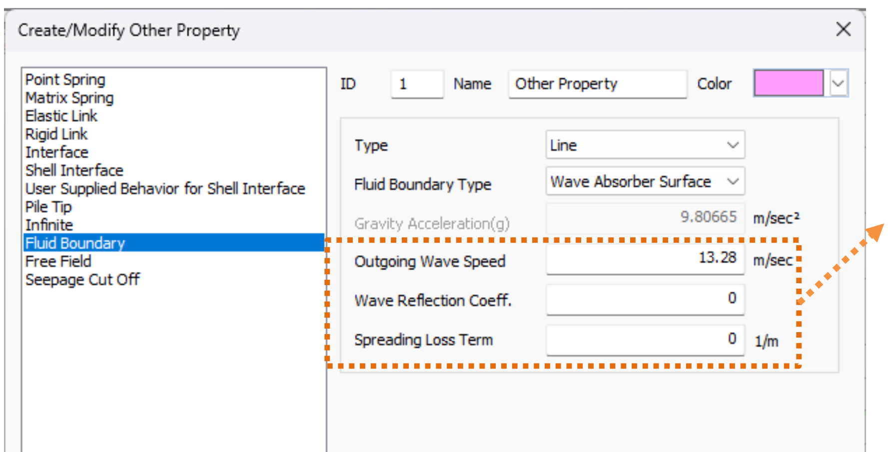
- A **wave-absorbent fluid boundary** has been introduced as an additional boundary element for the sloshing fluid medium.
- In the case of Dam-Reservoir interaction during a seismic event, this element can be assigned at the far end of the reservoir. This can result in the simulation of the infinite reservoir by blocking and absorbing the reflection of waves.
- This is essentially the Sommerfeld radiation condition, ensuring outgoing waves don't reflect into the domain.

Mesh > Prop./Csys./Func. > Property > Create : Other > Fluid Boundary > Wave Absorber Surface

In the case of an infinite boundary, an absorbing boundary, or under the Sommerfeld radiation condition, the pressure boundary condition is defined through attenuation and loss, as expressed below:

$$-\mathbf{n}\cdot\nabla p = \frac{1-\alpha_r}{c_i(1+\alpha_r)}\frac{\partial p}{\partial t} + \beta_l p$$

- Ci: Wave speed of interest
- αr: Wave reflection coefficient
- βl: Wave spreading/attenuation loss

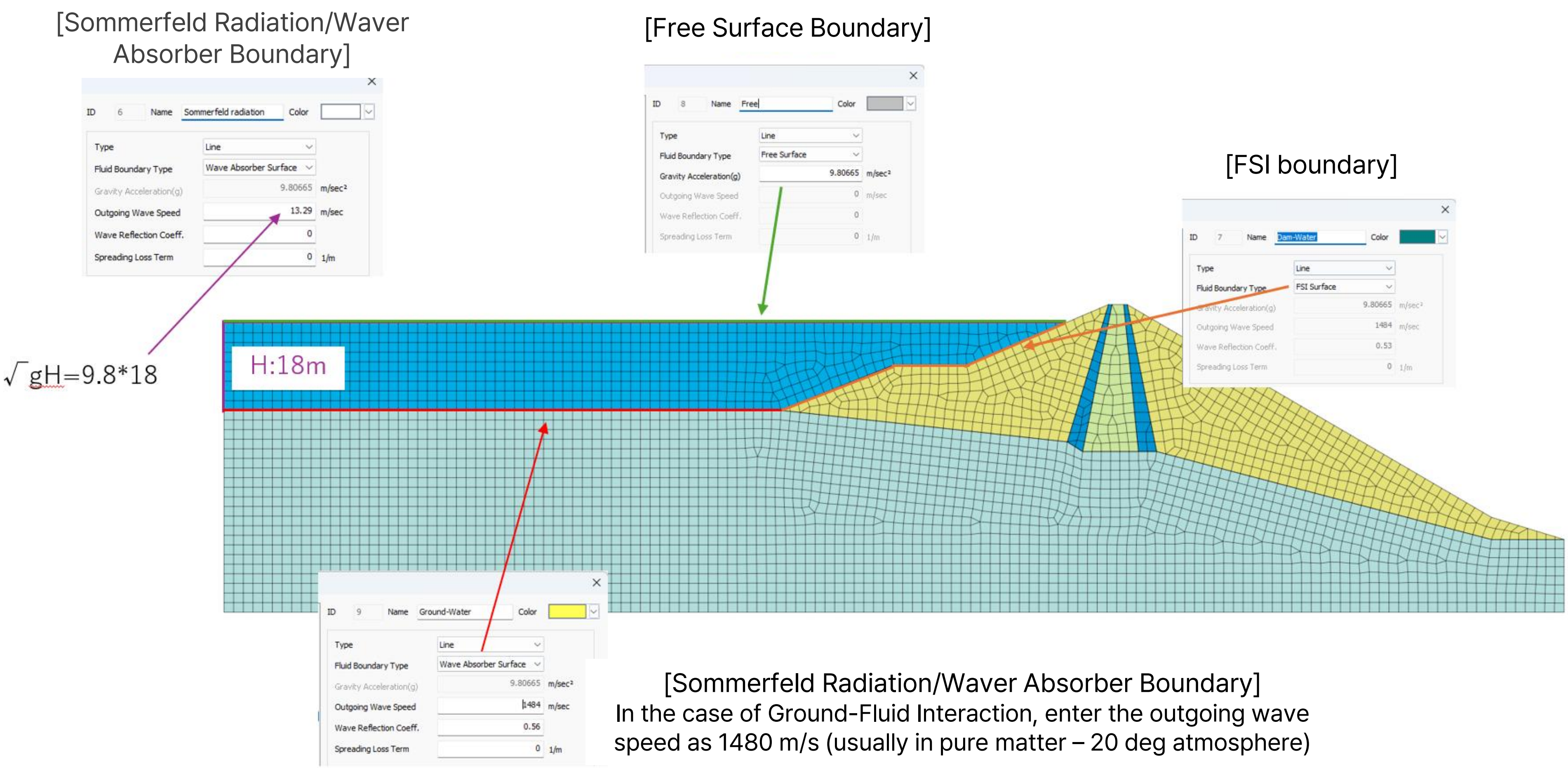


[Fluid Boundary Definition Window]

- **Outgoing Wave Speed:** Enter the wave speed of interest. For the outer condition consisting only of the Sloshing element, you can enter the Sommerfeld radiation condition as the Sqrt(g*H).
- **Wave Reflection Coefficient:** Enter the amplitude ratio of the wave reflected from the other media boundary. If it is 1 with a value of 0~1, it means full reflection.
- **Spreading Loss Term:** Enter the loss value in which the wave spreads in the sloshing medium and the energy decreases.

2.1 Wave Absorbent Surface for Sloshing Fluid Medium

Fluid Structure Interaction in the Dam-Reservoir System during an earthquake event could be as follows,



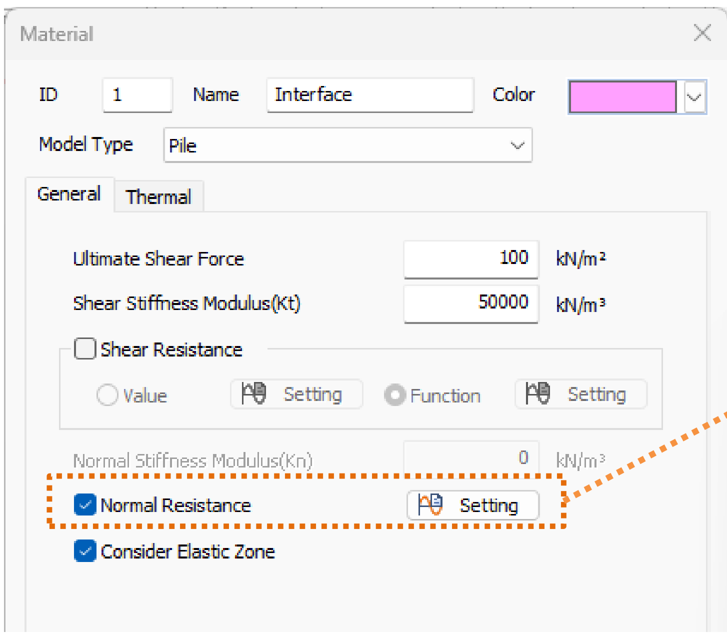
[Sommerfeld Radiation/Waver Absorber Boundary]
In the case of Ground-Fluid Interaction, enter the outgoing wave speed as 1480 m/s (usually in pure matter – 20 deg atmosphere)

2.2 Addition of Normal Stiffness vs Depth for Pile Interface

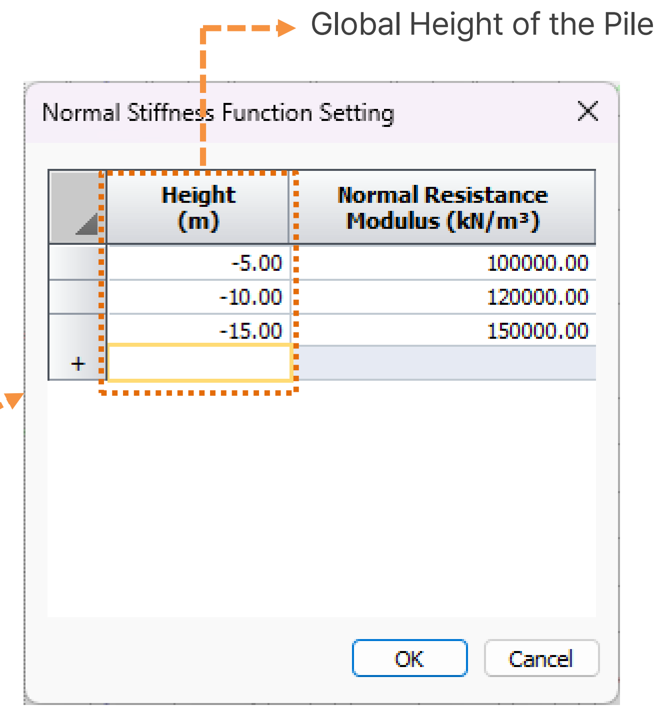
Now, defining Normal Stiffness Modulus vs. Depth for the Pile Interface is simpler. Users can directly input the global pile depth and corresponding Normal Stiffness Modulus. Previously, individual pile interfaces for each layer were required. This update offers two methods for defining Nomal Stiffness of the Pile Interface:

1. Direct definition of Normal Stiffness Modulus for the entire pile. 2. Normal Stiffness Modulus vs. Depth.

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



[Pile Interface Definition Window]

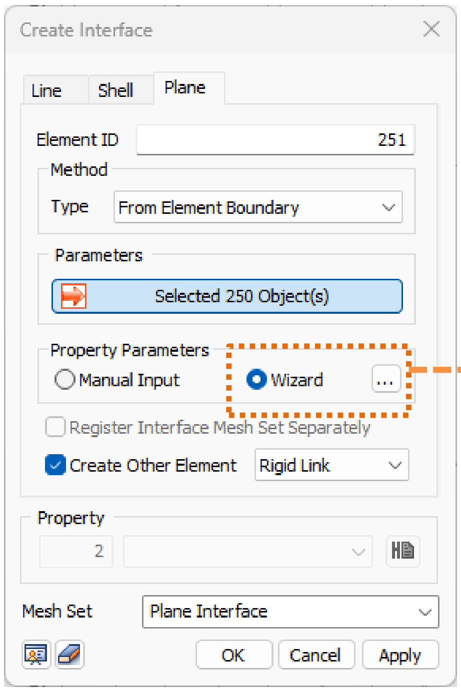


[Normal Stiffness vs Depth]

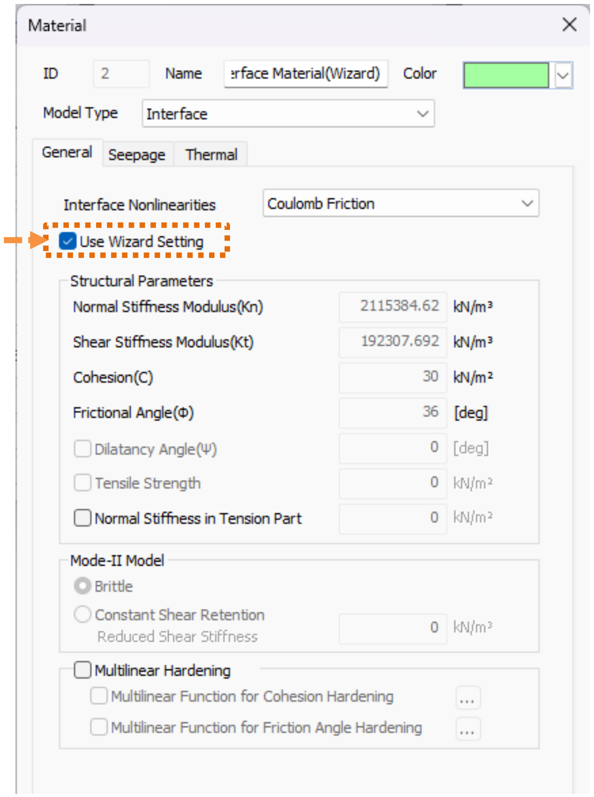
2.3 Automatic Update of Interface Parameters

- When creating interface elements using the wizard, a new feature has been added so that interface parameters are automatically updated whenever the material properties of adjacent elements are modified.
- However, for interface properties created in versions prior to 2026v1.1, the interface properties must be regenerated through the Interface Wizard in order for the auto-update function to be activated

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



[Interface Elements Creation]



[Interface Material]

[Interface Wizard Calculation Method]

- Normal Stiffness Modulus Kn= Eoed,I / tv
- Shear Stiffness Modulus Kt= Gi / tv
- Cohesion of Interface Ci = R x Csoil

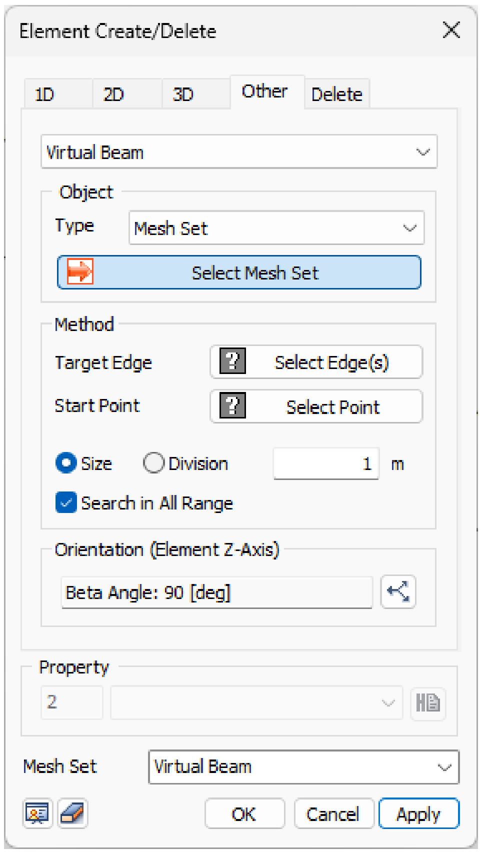
Where, Eoed,I = 2 x Gi x (1 - vi)/(1 - 2 x vi)

- vi= Poisson's ratio for the interface = 0.45(The interface simulates incompressible frictional behavior. To prevent numerical errors, a fixed value of 0.45 is used for automatic calculation.)
- tv= Virtual thickness(Typically in the range of 0.01–0.1; smaller values are recommended when the stiffness difference between adjacent elements is large.)
- Gi= R^2 x Gsoil (Gsoil = E/2(1+vsoil)), R= Strength Reduction Factor

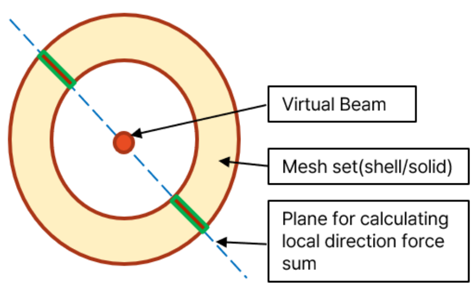
2.4 Enhancement in Virtual Beam Generation

- Previously, a virtual beam was created as a straight line connecting two random points, but the function has been improved so that you can select a virtual beam by selecting a line or wire. As a result, the forces such as Bending Moment, Shear Force and Axial Force of the virtual beam can be calculated according to the diameter of the pipe in a cross-section such as a pipe.
- Previously, the size of the virtual beam was forcibly determined internally at intervals of 1 meter, but it has been changed so that users can directly input the size or spacing.
- It has also been improved to provide table information for virtual elements so that users can change the Beta Angle value to suit their purpose.

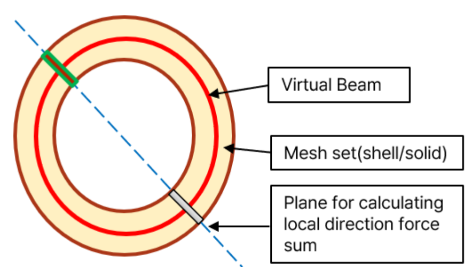
Mesh >Element > Create > Other > Virtual Beam



- **Type:** Select Elements or Mesh Set.
- **Target Edge:** Select the line or wire to create the virtual beam.
- **Start Point:** Select the starting point at which you want to create the virtual beam.
- **Size/Division:** Specifies the element size or division interval of the virtual beam.
- **Search in All Range:** Specifies the method for the elements that can be used as a reference when creating a virtual beam. It is calculated based on the history of the elements across the cross-section drawn in the normal direction of the virtual beam that selects the entire number, and if unchecked, it is calculated based on the history of adjacent elements. For example, if you set the virtual beam in the shape of the diameter of the pipe in the pipe section, you must uncheck it, and if the virtual beam is located only in the center of the pipe, you must check the corresponding option.



[Check Option]

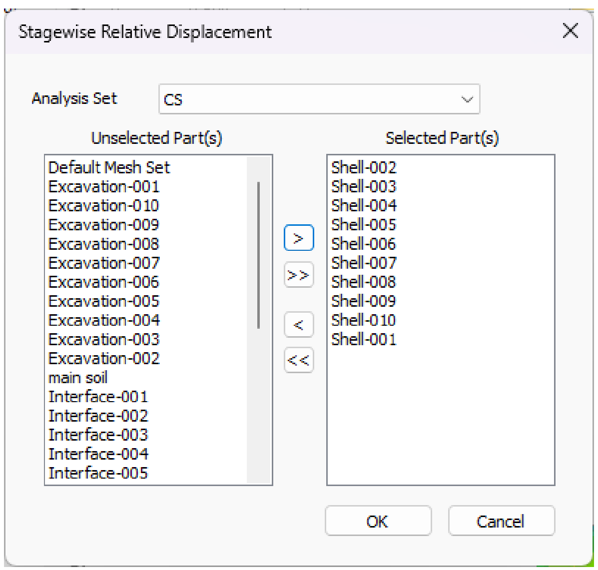


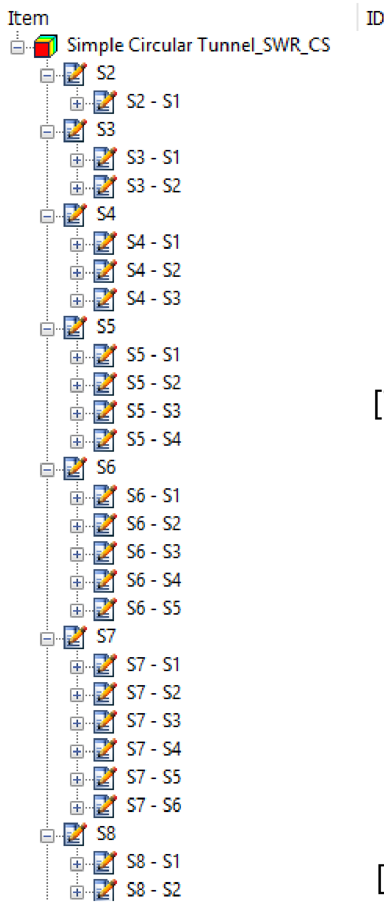
[Uncheck Option]

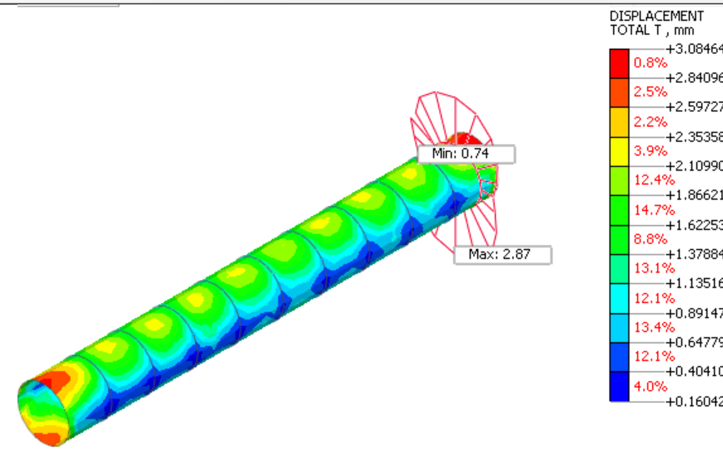
2.5 Stagewise Relative Displacement

- Stage-by-stage relative displacement results can now be output based on user-selected mesh sets for each construction stage.
- With the result-combination function, users can quickly review and output results without the need to manually generate data for dozens or even hundreds of construction stages.
- For example, this function can be much helpful in determining the displacement of the 'tunnel lining only' rather than finding the displacement of the tunnel walls nodes (common nodes of the tunnel lining) which may include the displacements that were present before the activation of the tunnel lining elements such as relaxation or volume loss.

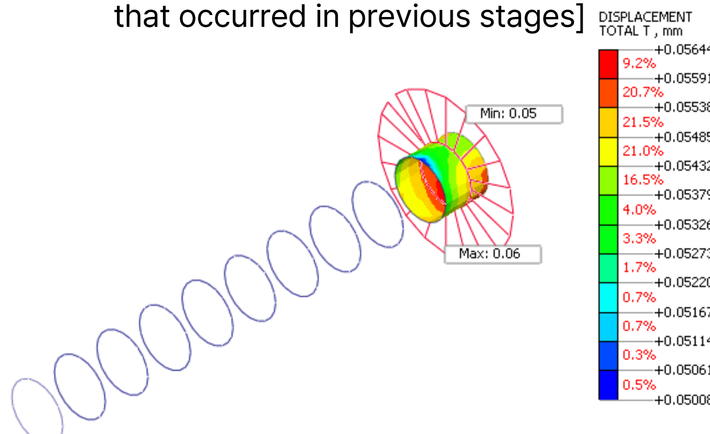
Result > Advanced > Others > Stagewise Relative Displacement







[Total Displacement of the Tunnel including the Displacement that occurred in previous stages]

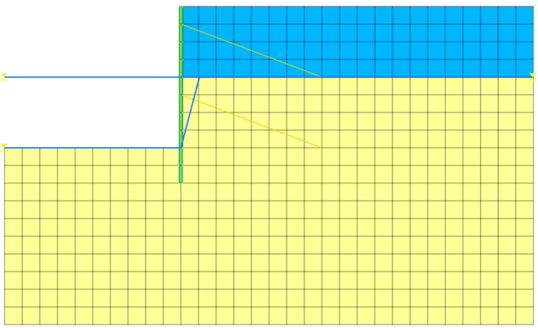


[Total Displacement of the Tunnel Lining only using Stagewise Relative Displacements]

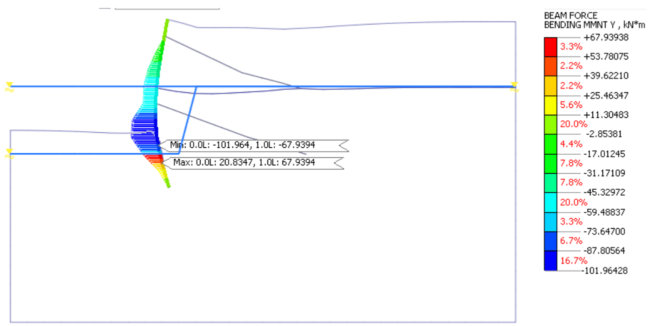
2.6 Structural Design of Elements via CIVIL NX/GEN NX (SAR Format)

- The 'SAR' (Standard Analysis Result) format can link the Analysis tool like FEA NX with the Structural Design tools like 'CIVIL NX' & GEN NX'. This way, we can now export the model file along with the results such as Bending Moments, Shear Forces, and Axial forces obtained from FEA NX to Civil NX/GEN NX for detailed structural design.
- This format can only support Truss, beam, Plane Stress, Shell, and Gauging Shell elements.

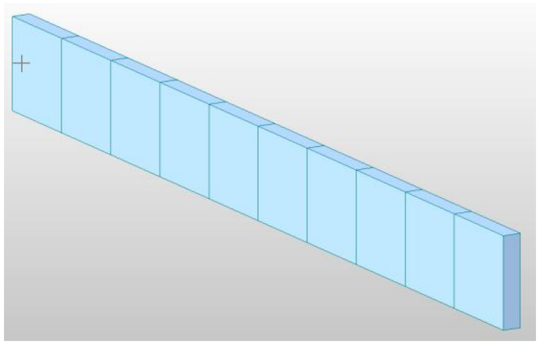
FEA NX ICON > Export > Export SAR File (for Structural Design)



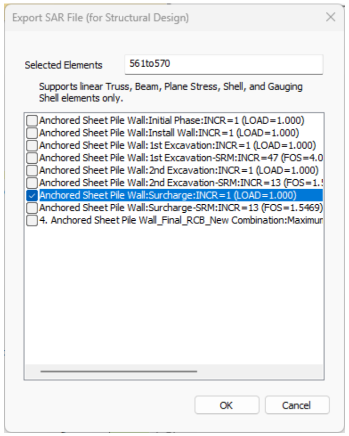
1. Run the required analysis in FEA NX



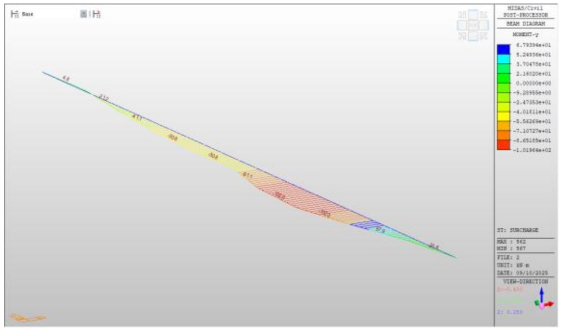
2. Check the Results needed for Structural Design



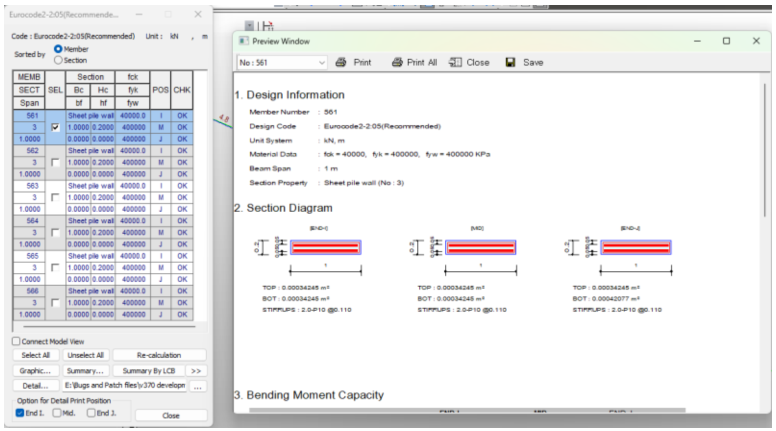
3. Export the Model from FEA NX to CIVIL NX/GEN NX via .mxt format



4. Select the Stage/Step for Result Export



5. Import the Results into CIVIL NX/GEN NX via 'SAR'

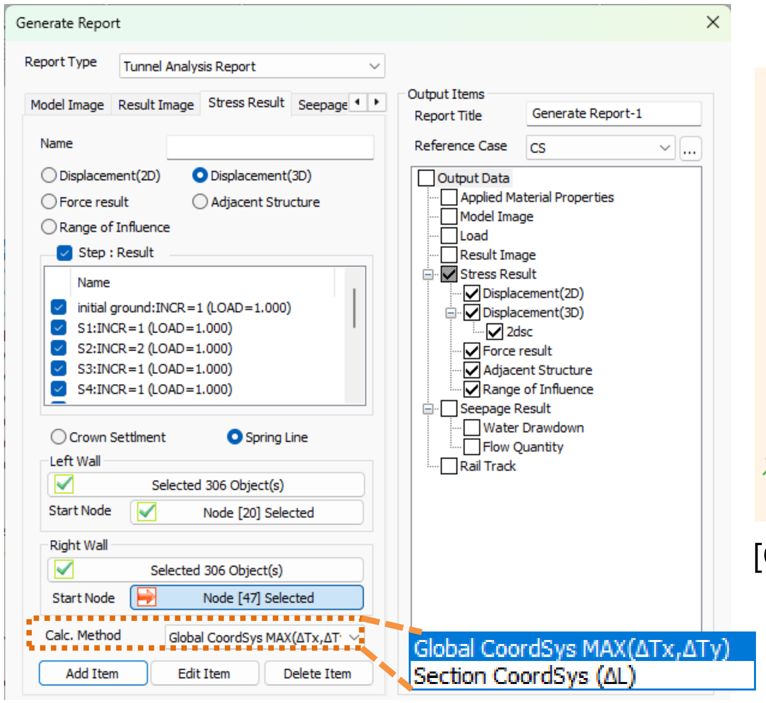


6. Select the Code and perform Structural Design in CIVIL NX/GEN NX

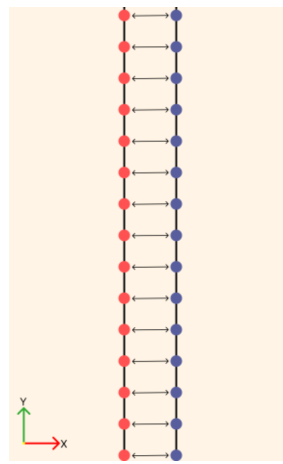
2.7 Report Options: New Tunnel Analysis Report Type

- 2D/3D crown displacement and internal displacement results are now available.
- For 3D internal displacement calculations, results can be provided based on both the ‘Global Coordinate’ system and the ‘Section Coordinate (Shortest Distance)’ system, enabling more accurate evaluation of actual deformation influenced by curvature.

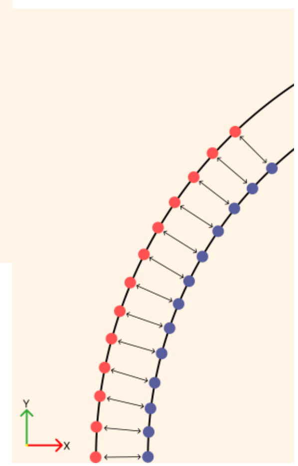
Tools > Export > Generate Report



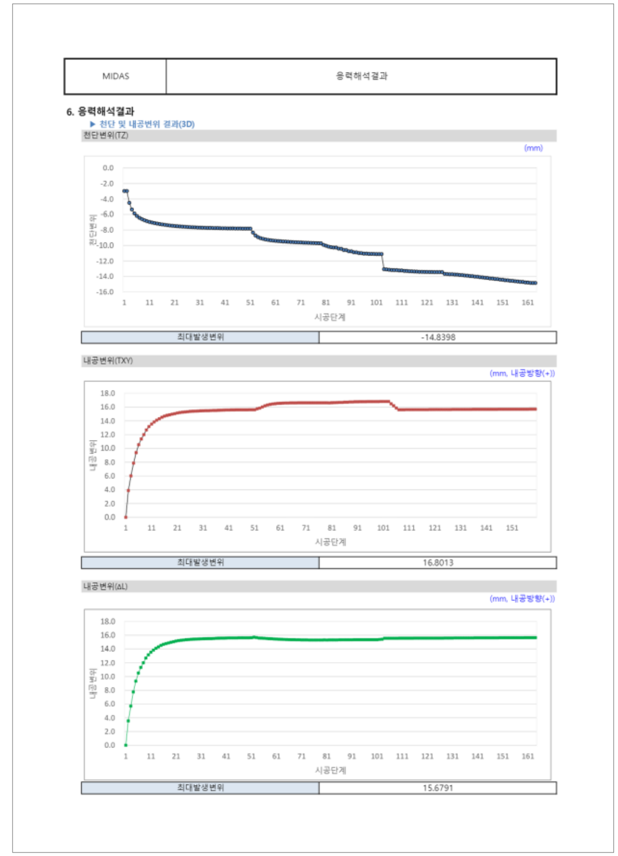
[Tunnel Analysis Report – 3D Internal Displacement]



[Global Coordinate System]



[Section Coordinate System] (Shortest Distance Basis)



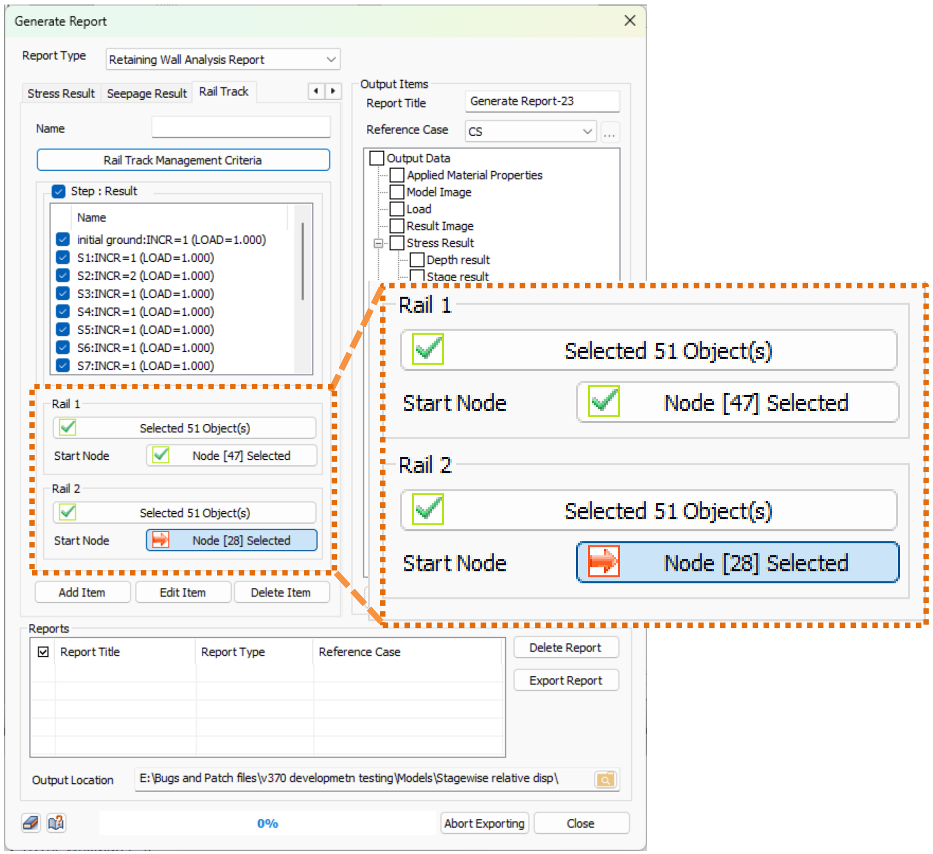
[3D Displacement Result Graph]

2.8 Report Options: Improved Orbit Review Functionality

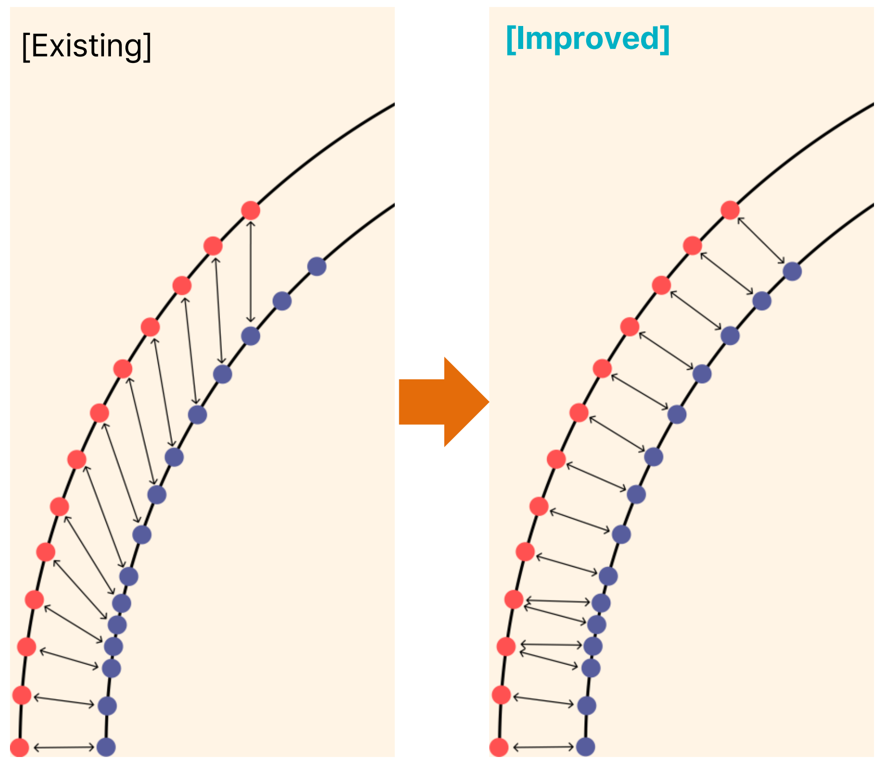
Improved Algorithm for Railway Track Evaluation

- [Previous] In 3D track evaluation, the process was carried out sequentially from the starting nodes of Rail 1 and Rail 2.
- **[Improved]** The calculation method has been updated to use the rail with the greater number of nodes as the reference, finding corresponding nodes based on the shortest orthogonal distance.

Tools > Export > Generate Report



[Rail Track Definitions]



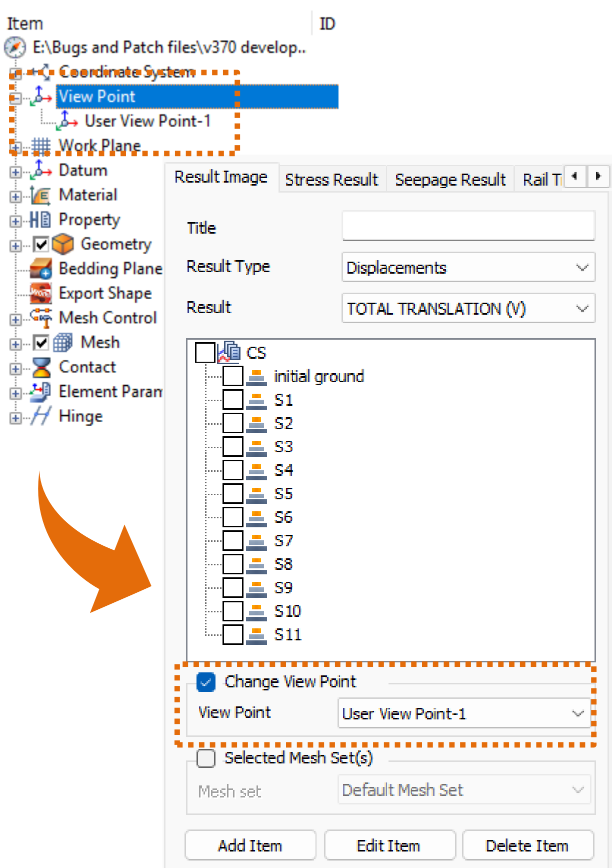
[Improved Track Node Matching Algorithm]

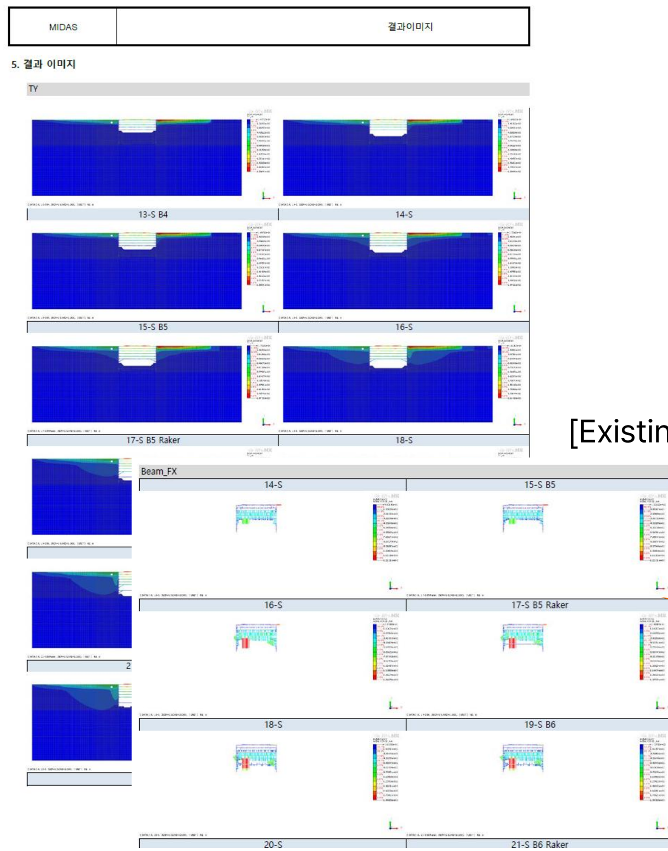
2.9 Improved Application of Observation-point Images and Type Verification

- When outputting Model View/Result View, the observation point can be adjusted to generate images.
- Image viewpoints can be set separately for the overall model and for individual key structures, allowing resolution adjustments as needed.

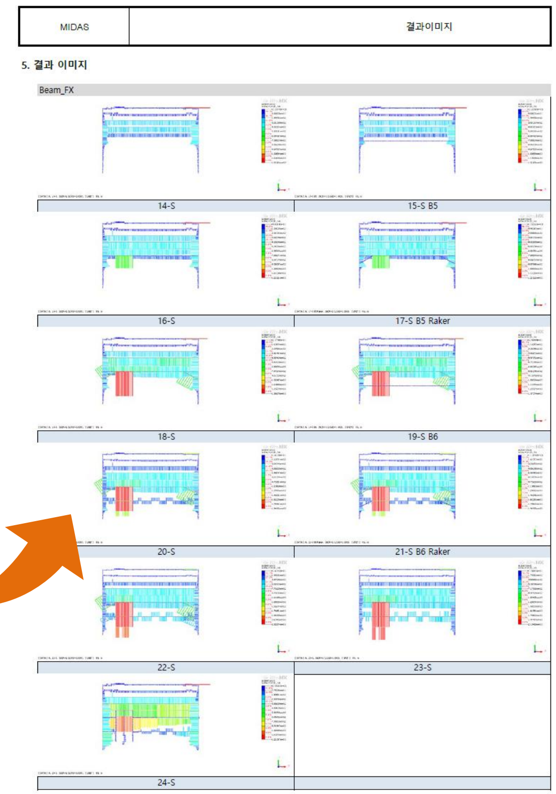
Tools > Export > Generate Report

[Improved] Viewpoint for each output item can be applied





[Existing]

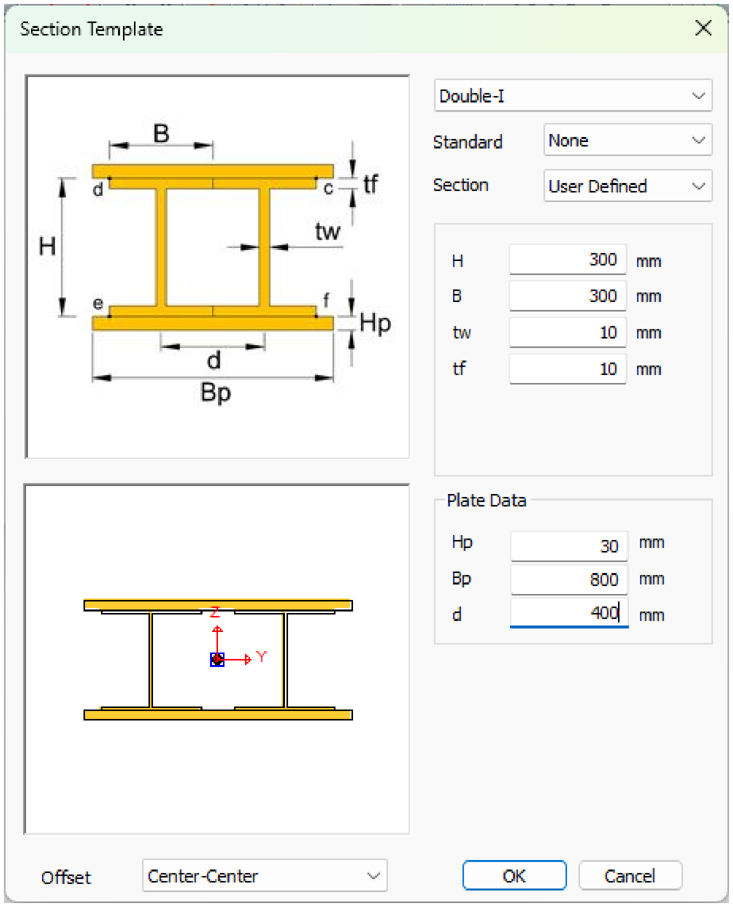


[Improved Image Capture –viewpoint Customization]

2.10 Addition of a New Double I-Section Property

A new cross-section property, a double I beam with plates on top and below the two H-beams is added for 1D elements.

Mesh > Prop./Csys/Func. > Property > 1D > Section



- Hp: Plate thickness (Hp ≥ 0)
- Bp: Plate width (Bp ≥ 0)
- d: Distance between H-beam flanges (B ≤ d ≤ Bp)

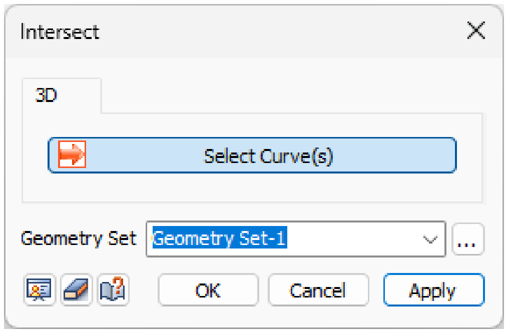
2.11 Fixed Geometry Set for 'Intersect' Function

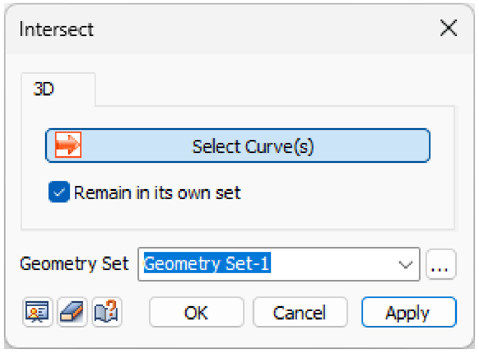
When curves are intersected using the Intersect function, each line is registered in its own geometry set if the 'Remain in Own Set' option is enabled.

2.12 Enhancement in Load Table Import

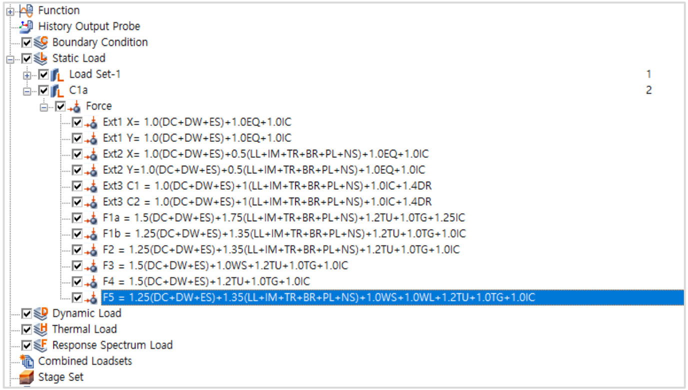
In FEA NX, frequently used load types can be defined and imported from Excel files or exported after definition. Previously, when load names were too long, duplication occurred, and the loads could not be retrieved correctly. This issue has been resolved by increasing the allowable string length

Geometry > Point & Curve > Intersect





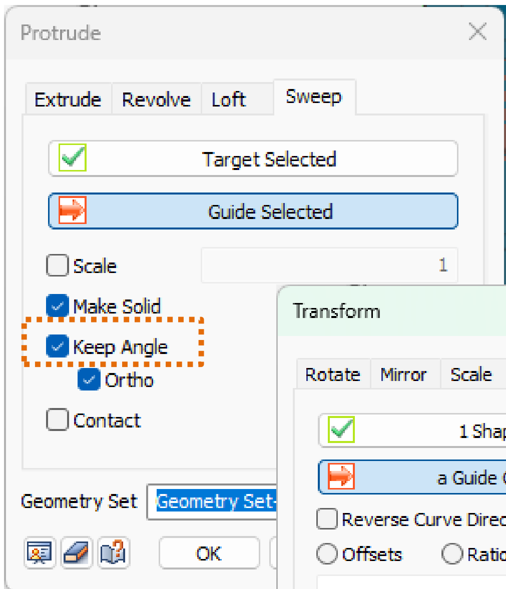
Static/Slope Analysis > Load > Table > Load Table Import



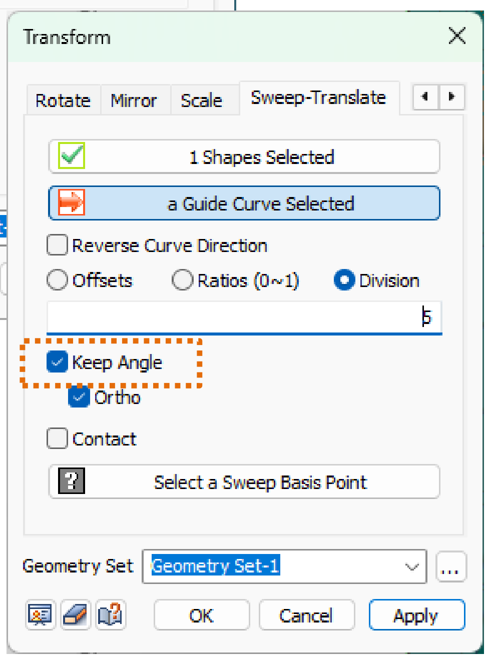
2.13 Enhancement of Sweep-Ortho Function

- Added option to maintain the orthogonal angle of guidelines when extracting or moving geometric sweeps

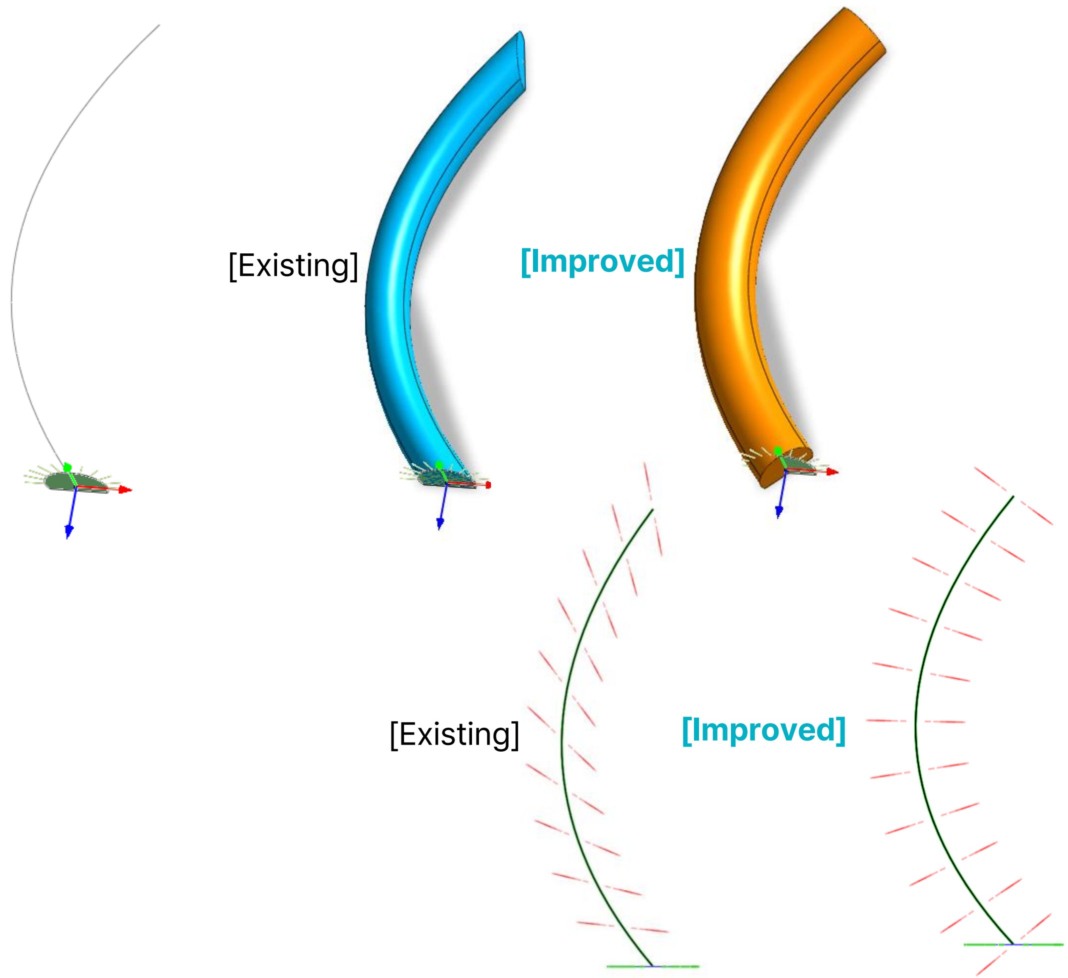
Geometry > Protrude > Sweep, Geometry > Transform > Sweep-Translate



[Sweep]



[Sweep - Translate]



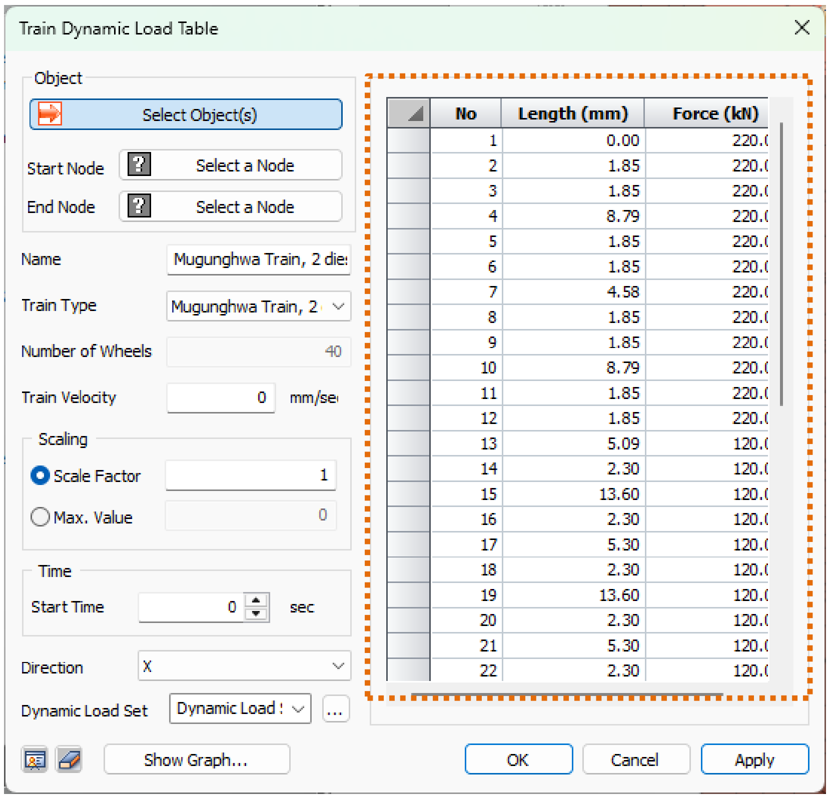
2.14 Excel Compatibility (Copy & Paste) in Train Dynamic Load Table

The method of inputting user-defined train moving load has been improved for direct input in a table format or by copy & paste from excel sheet.

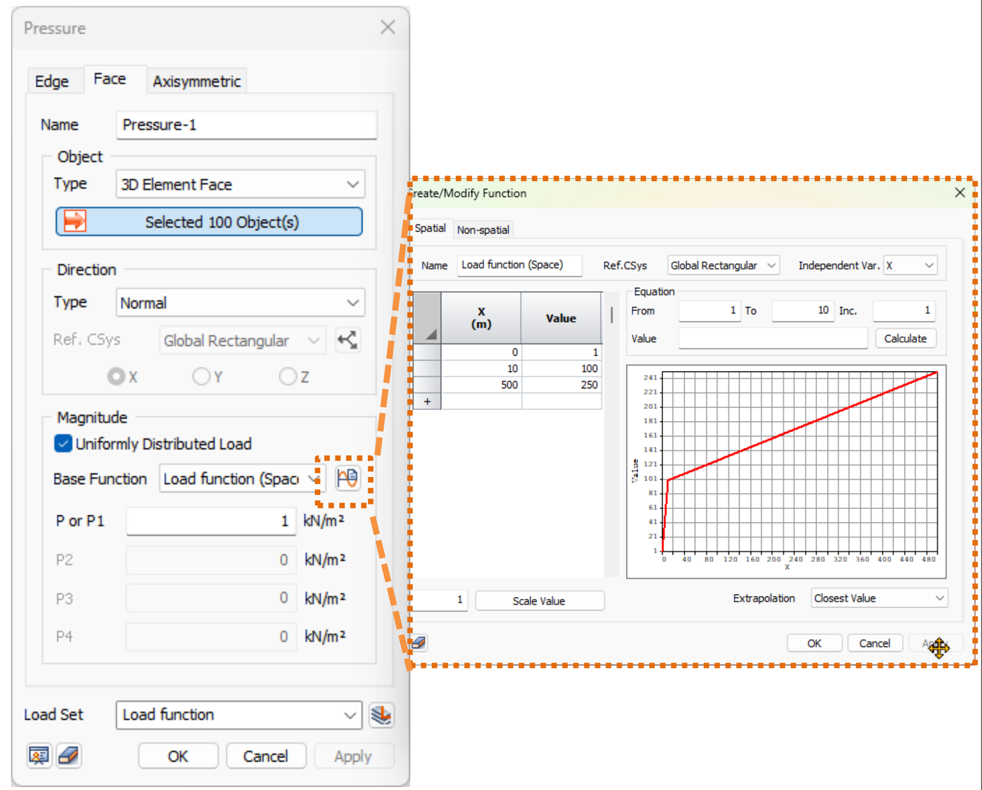
2.15 Enhancement in Selection of Function Data for Modification

Until FEA NX 2025v1.1, editing Function data required navigating to the Analysis Workstree. Starting from FEA NX 2026v1.2, you can simply click on existing data (e.g., Load, Material, etc.) to view or modify the linked function directly

Dynamic Analysis > Load > Train Dynamic Load Table



Function Linked to the Existing Data



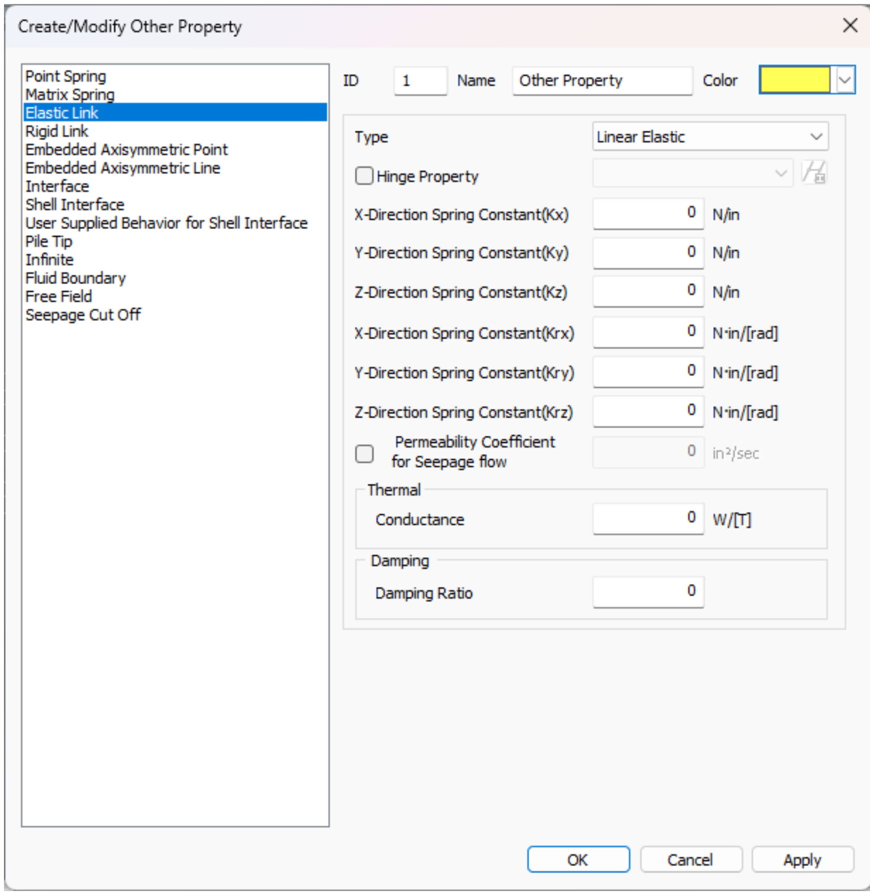
2.16 Damping Constants in Elastic Link

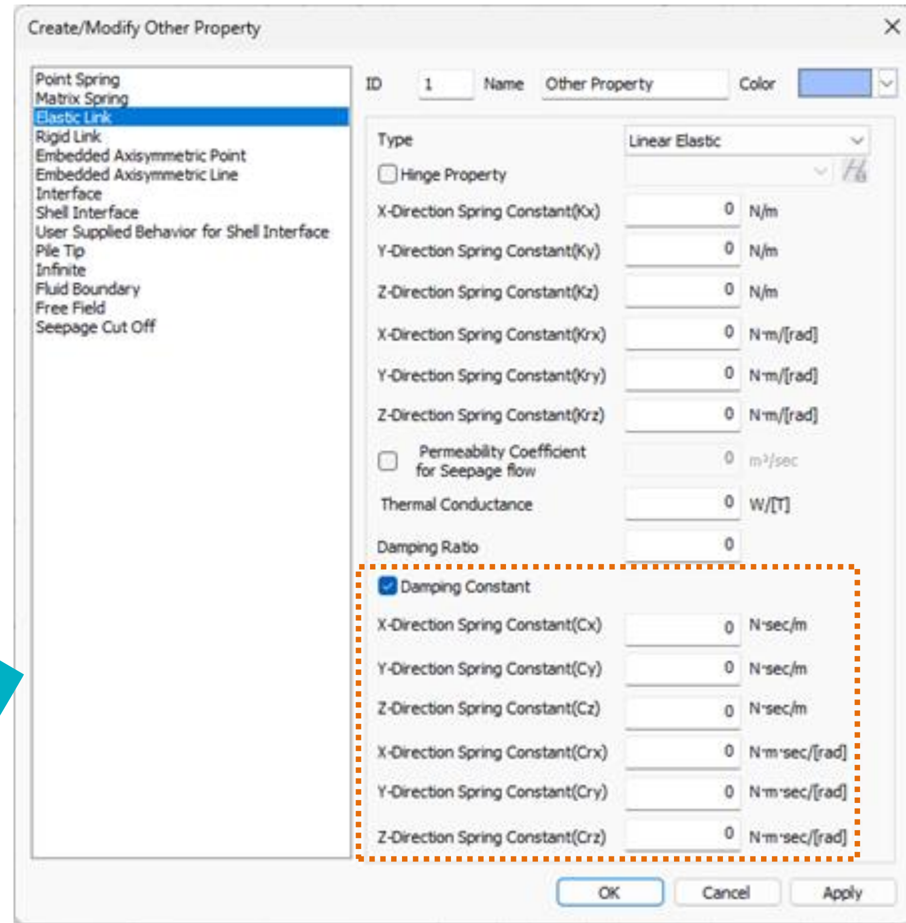
- The function has been enhanced to allow input of a damping constant in the elastic links.
- The Linear Elastic link can now act as an attenuator, absorbing and dissipating energy in addition to its spring.

Mesh > Prop./Csys./Func. > Property > Create : Other > Elastic Link

[Improved] Damping constant

[Existing]

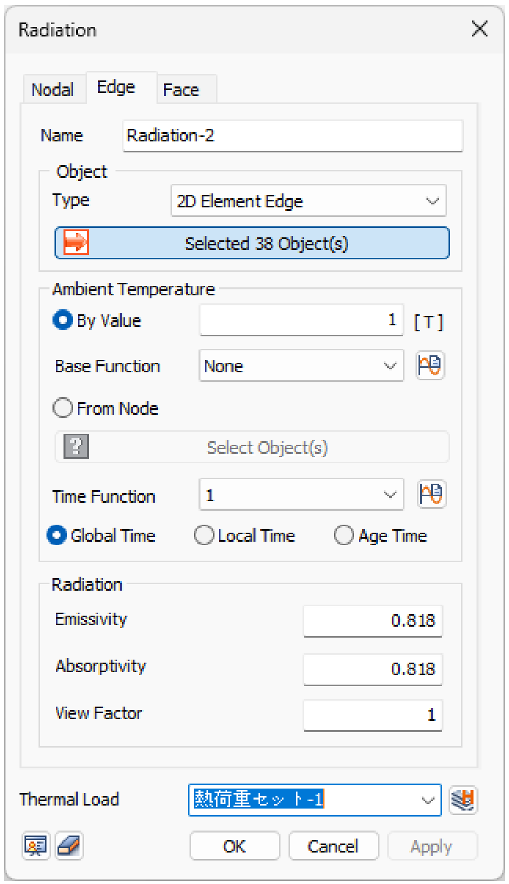




2.17 Radiation Condition

- A new function has been introduced to define radiation conditions based on the temperature difference between the ambient environment and the surface.
- Radiation condition can be assigned to nodes, lines (interfaces), and element faces within the model.
- This function accounts for heat transfer through electromagnetic radiation, where objects above absolute zero emit thermal energy.

Mesh >Element > Create > Other > Virtual Beam

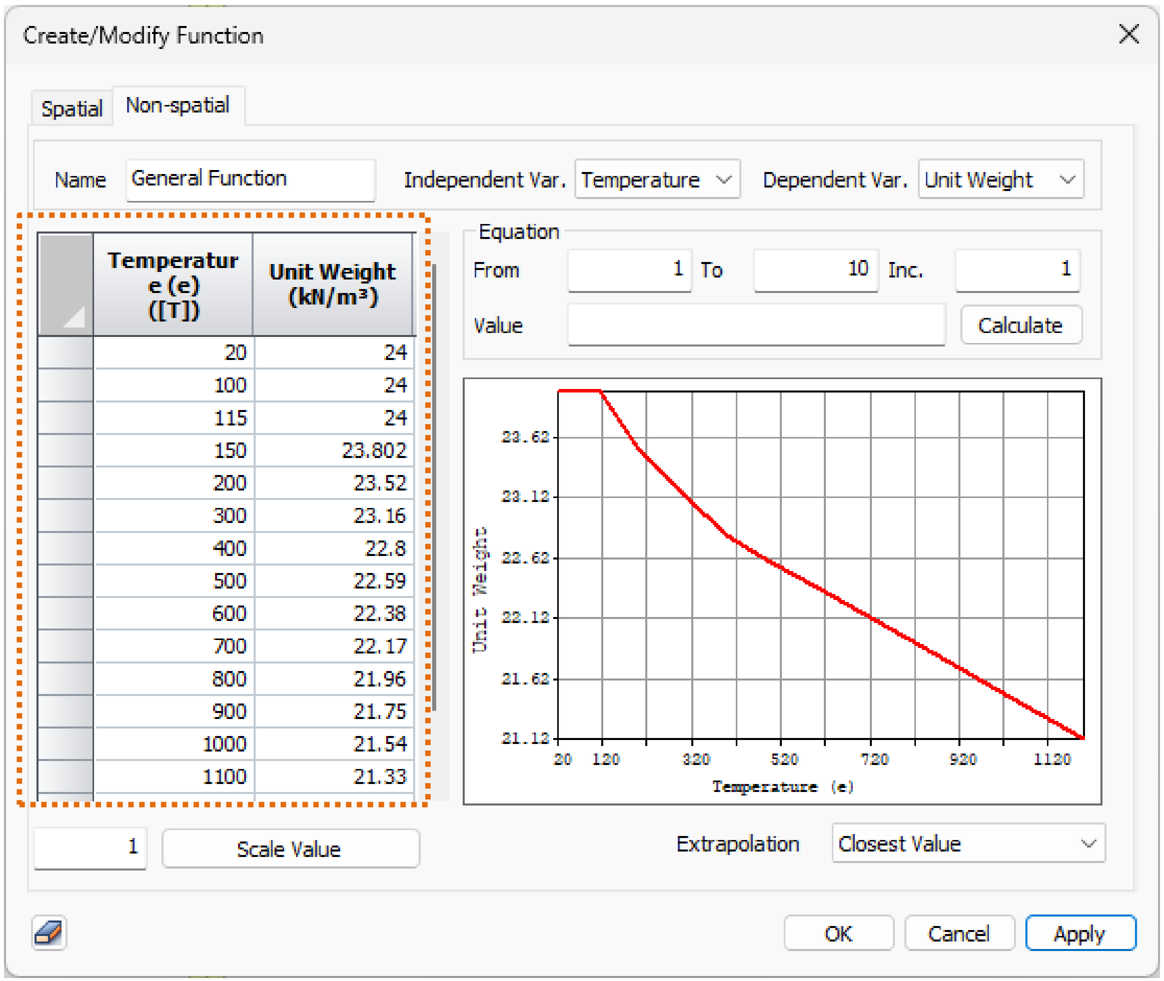


- **Time Function:** Select Full time or Subcase time. Full Time considers the total analysis duration, while Subcase Time defines the function based on the construction stage duration.
- **Emissivity:** The ability of a material to emit thermal radiation compared to an ideal black body. It ranges from 0 to 1, where a black body has an emissivity of 1.
- **Absorptivity:** Defines the fraction of incident radiant energy absorbed by a surface. It ranges from 0 to 1, with higher values indicating greater absorption.
- **View Factor:** Indicates the geometric relationship between two surfaces in terms of radiation exchange. It ranges from 0 to 1, where 1 represents full exposure and 0 represents no exposure.

2.18 Temperature-Dependent Unit Volume Weight

- A new function has been introduced to define unit volume weight as a function of temperature.
- This allows users to account for variations in material density under changing thermal conditions.

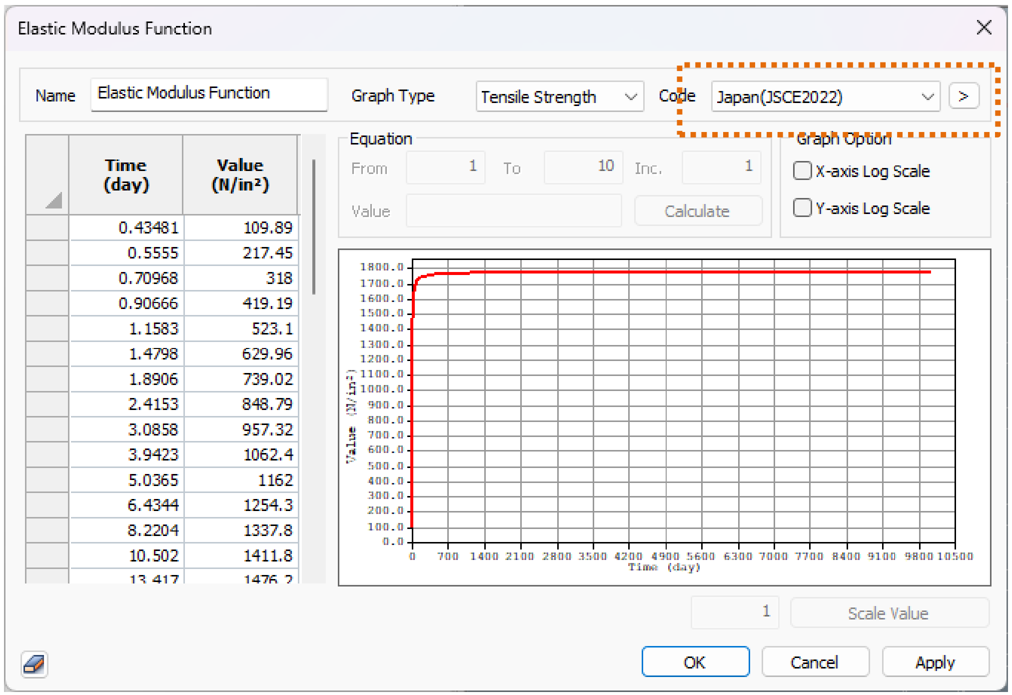
Mesh > Prop./Csys./Func. > Property > Function > General

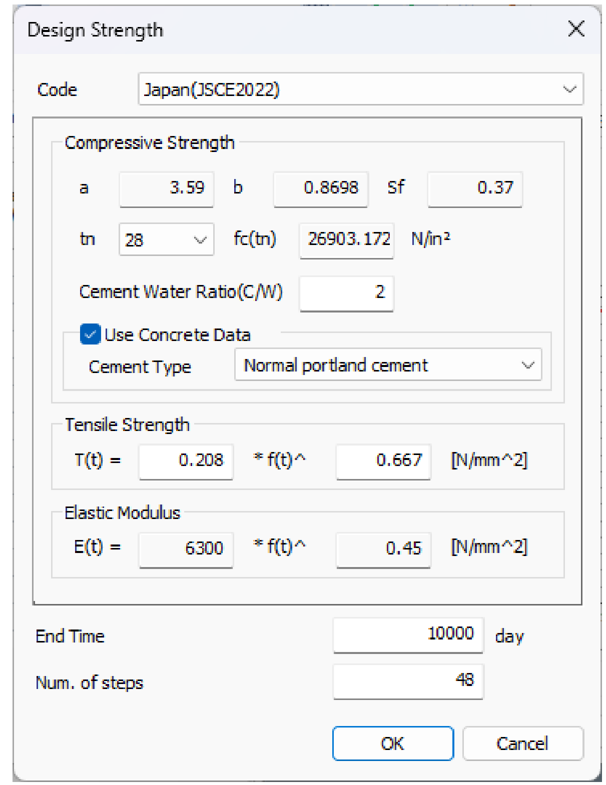


2.19 Elastic Modulus Function: Addition of JSCE (2022) Standards

- The modulus of elasticity variation over time can now be defined based on JSCE (2022) provisions.
- The heat generation (exothermic) function over time has also been updated in accordance with JSCE (2022).
- The design strength formula has also been updated as per the JSCE 2022.

Mesh > Prop./Csys./Func. > Elastic Modulus Function

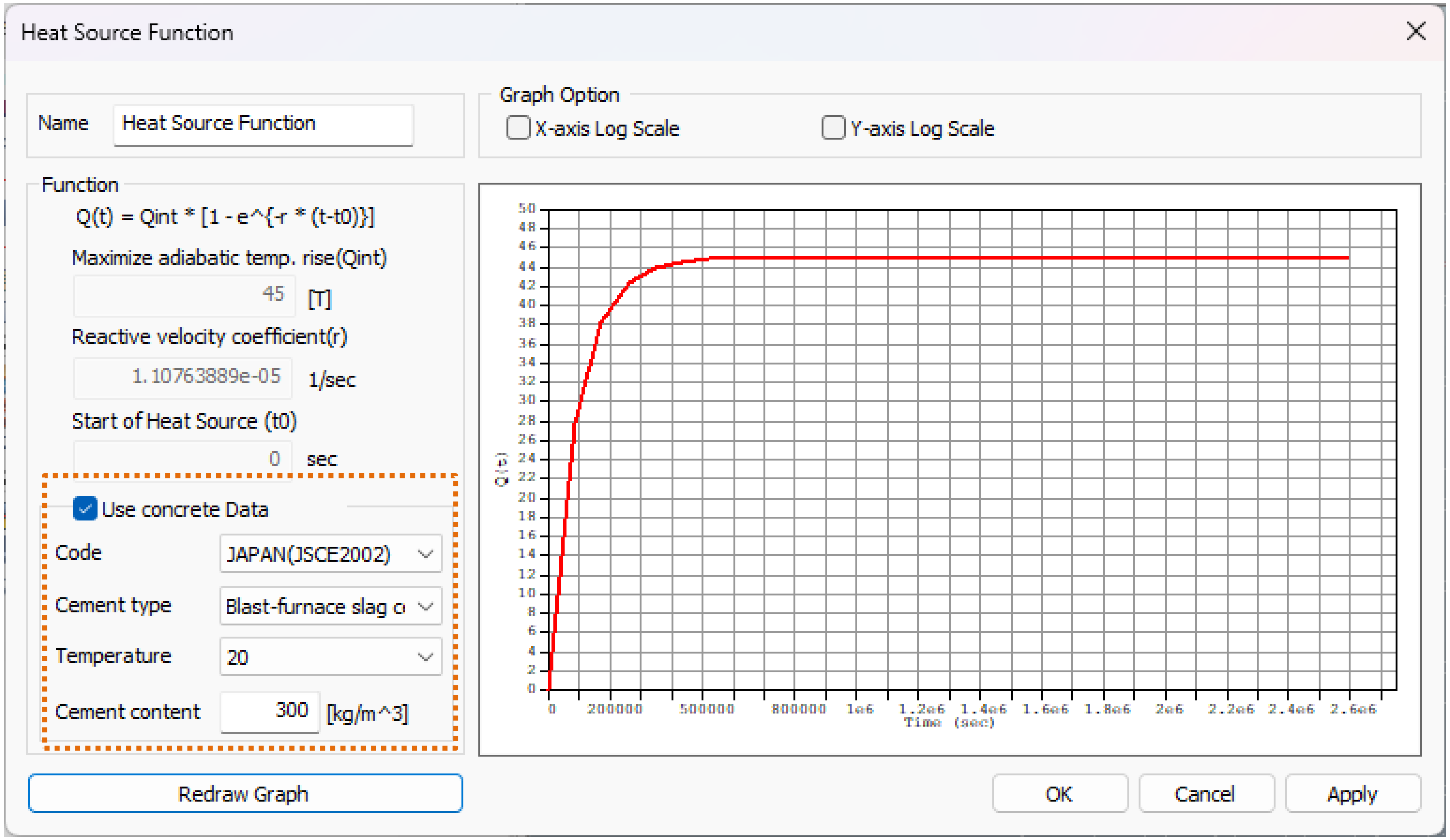




2.20 Heat Source Function as per JSCE-2022

- The heat source function has been updated in accordance with JSCE (2022) standards.
- A new C-type factor for blast furnace cement has been added (C-type cement density: 300-500 kg/m²).

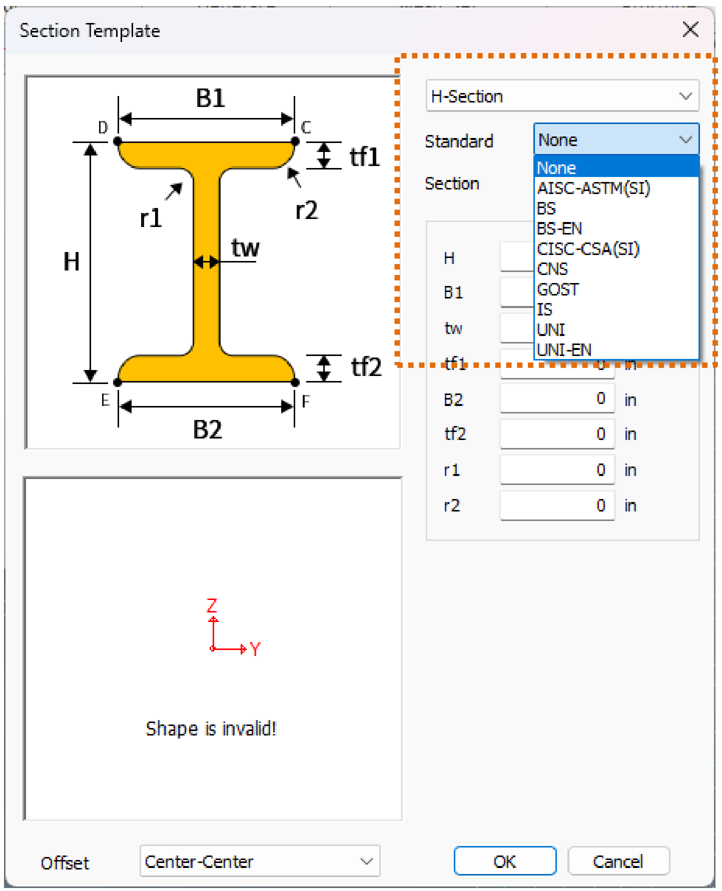
Mesh > Prop./Csys./Func. > Function > Heat Source Function

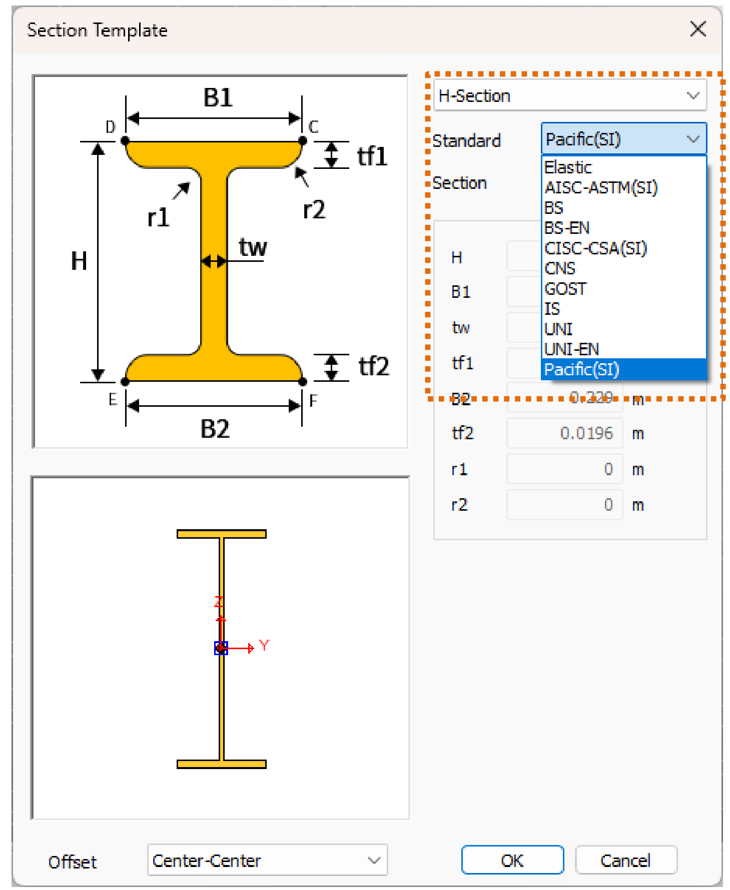


2.21 Steel Section DB Addition – Pacific (SI) (AU/NZ Standards)

- A new steel section database for Pacific (SI) has been added. This includes H-beam sections based on Australian and New Zealand standards.

Mesh > Prop./Csys./Func. > Property > Cread: 1D > Section > H-Section

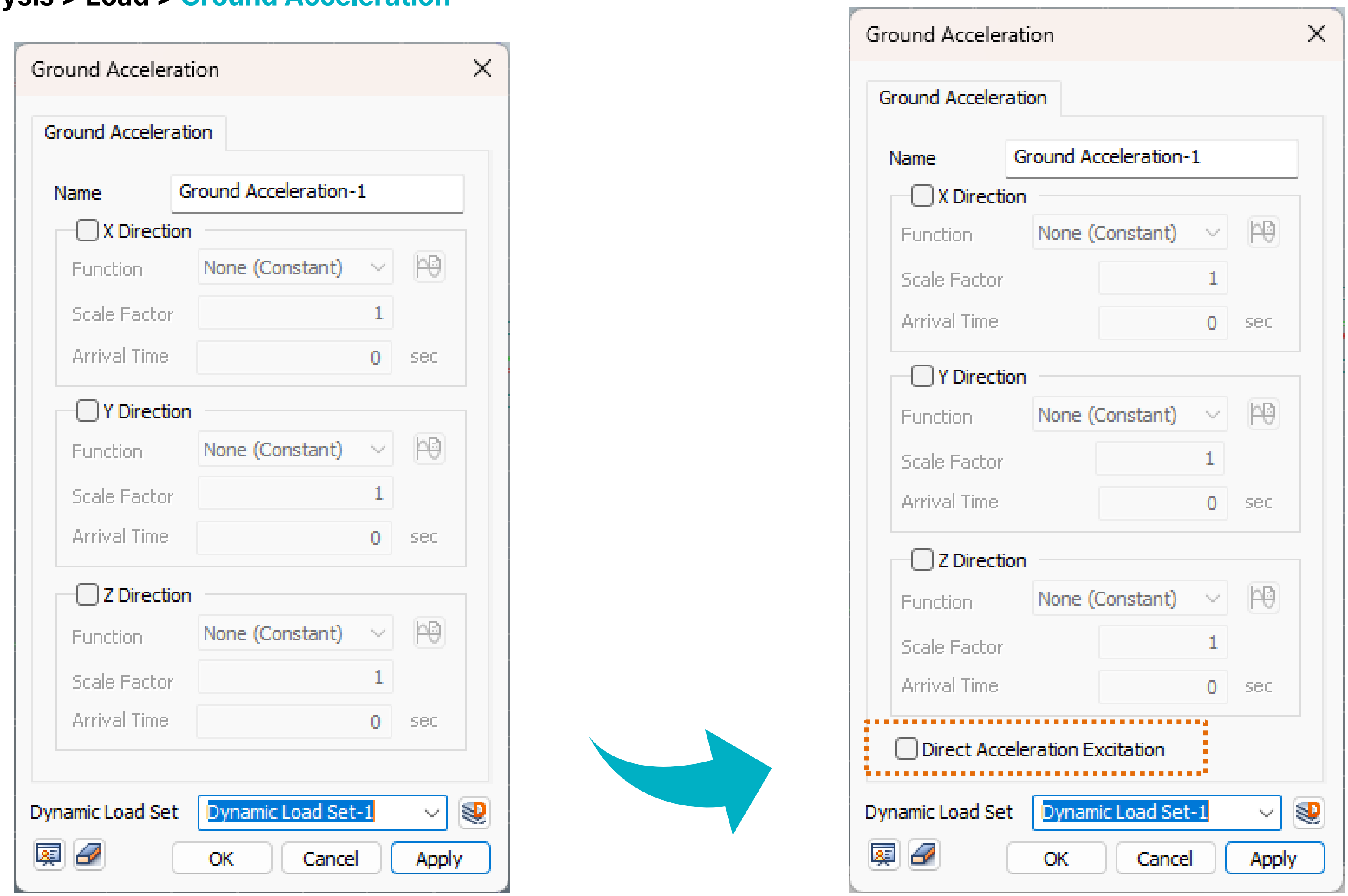




2.22 Direct Acceleration Excitation

- A new Direct Acceleration function has been added for dynamic analysis.
- In the conventional ground acceleration method, only constrained nodes (typically at the base) are excited. With the Direct Acceleration option, acceleration can now be applied to the entire model.

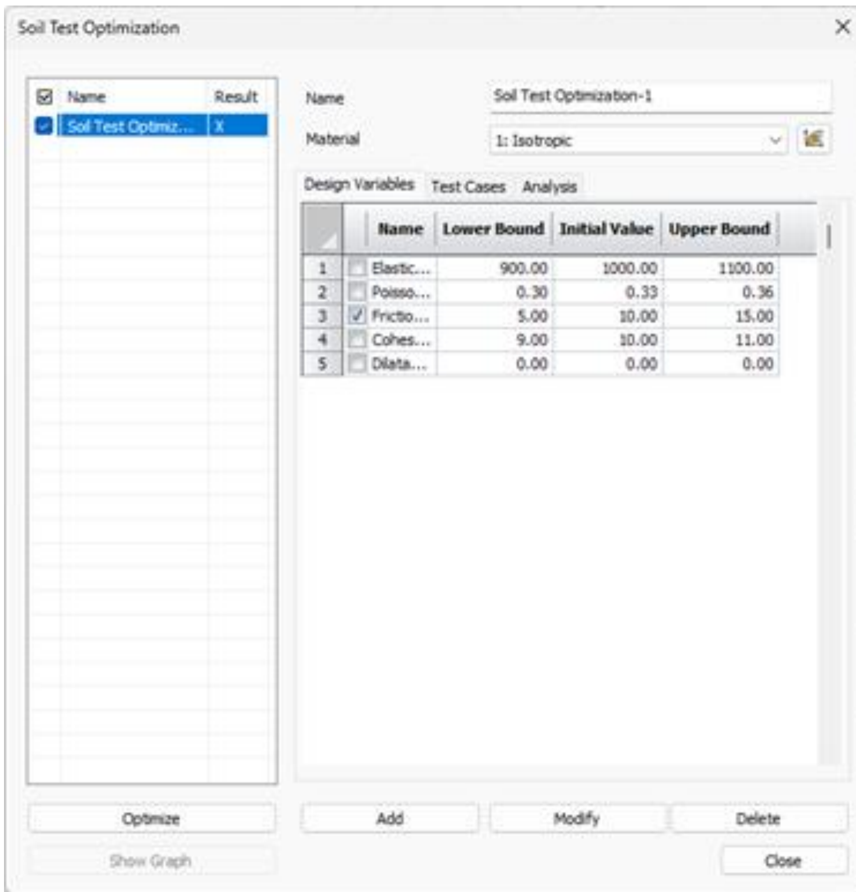
Dynamic Analysis > Load > Ground Acceleration

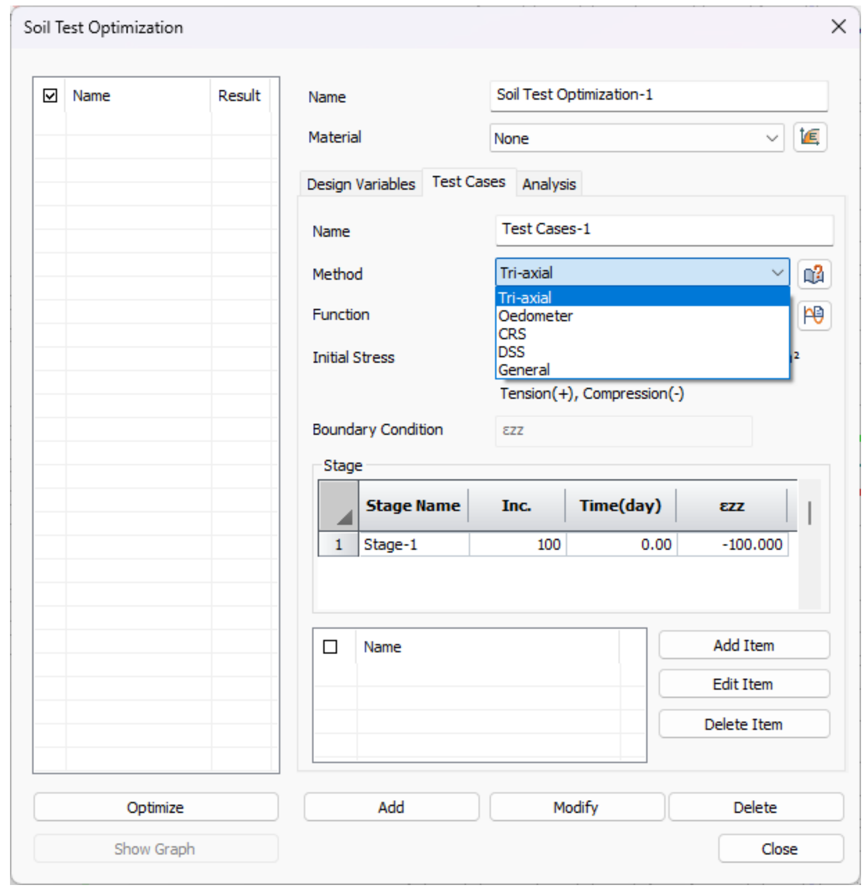


2.23 Soil Test Optimization

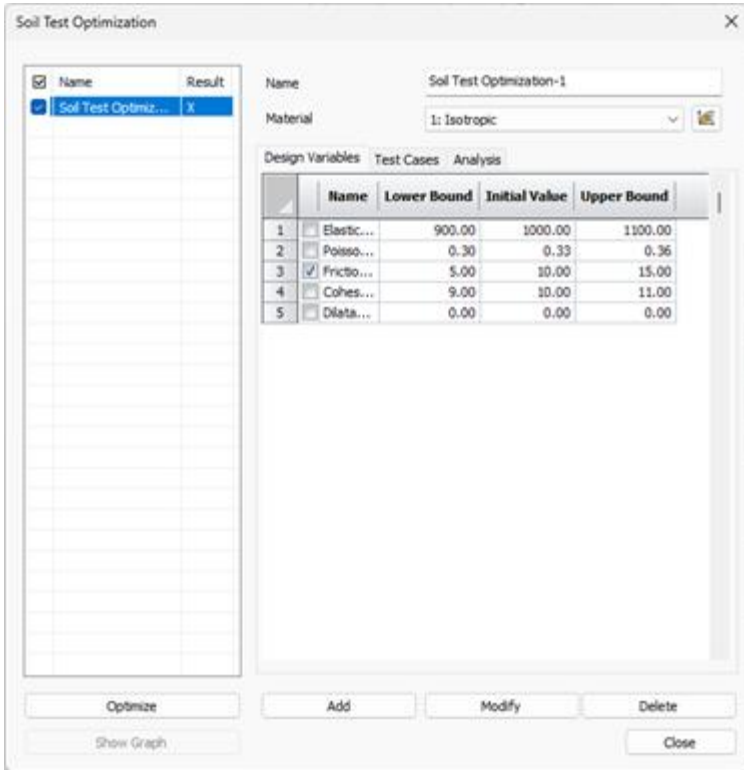
- A new Soil Test Optimization function has been introduced. The feature supports MC, HS(S), and SS(C) material models.
- The program compares the target curve obtained from laboratory tests (for example: triaxial compression test) with the defined material model behavior.
- Based on this comparison, the soil properties are automatically adjusted to achieve the best possible match between the curves.

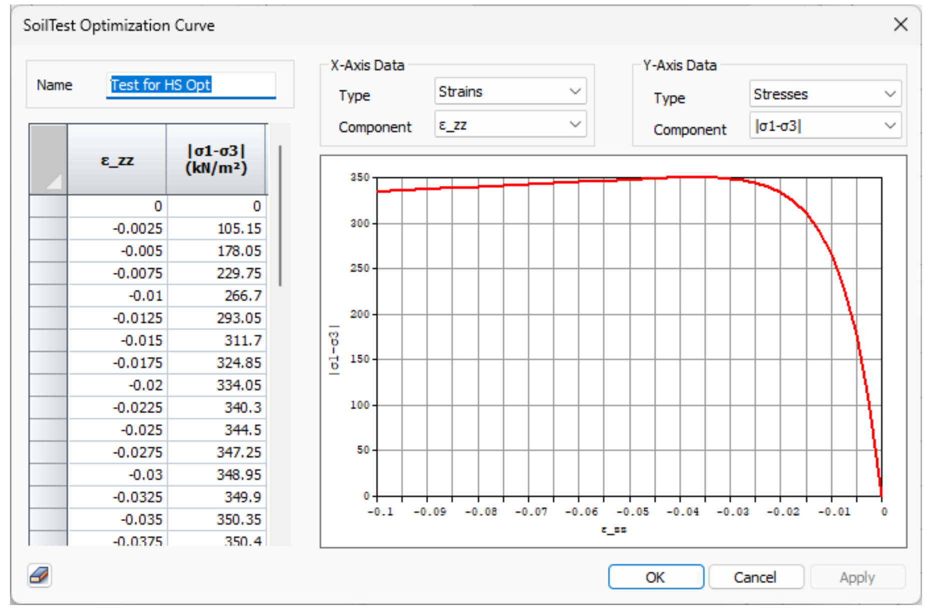
Static Analysis > Wizard > Soil Test > Soil Test Optimization





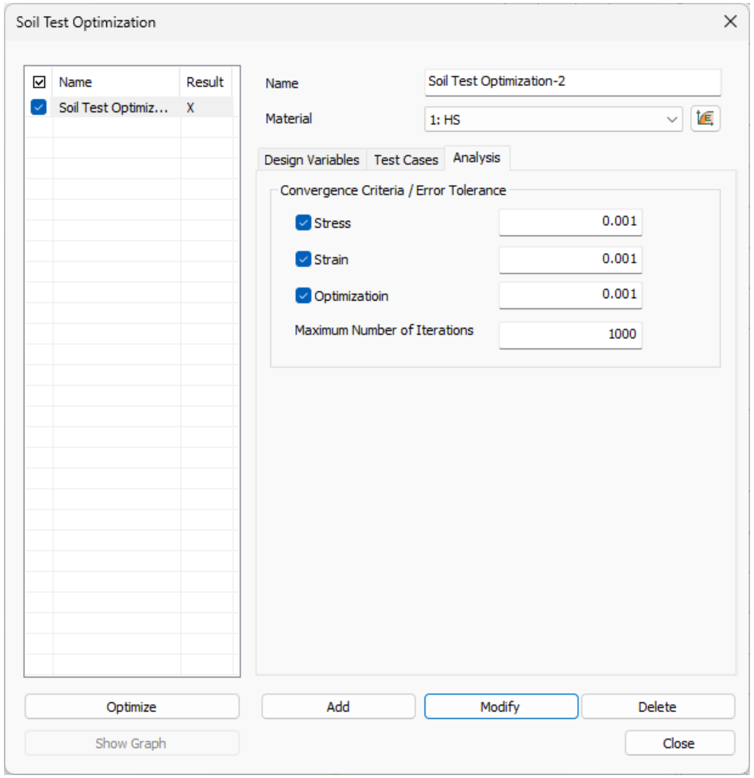
2.23 Soil Test Optimization

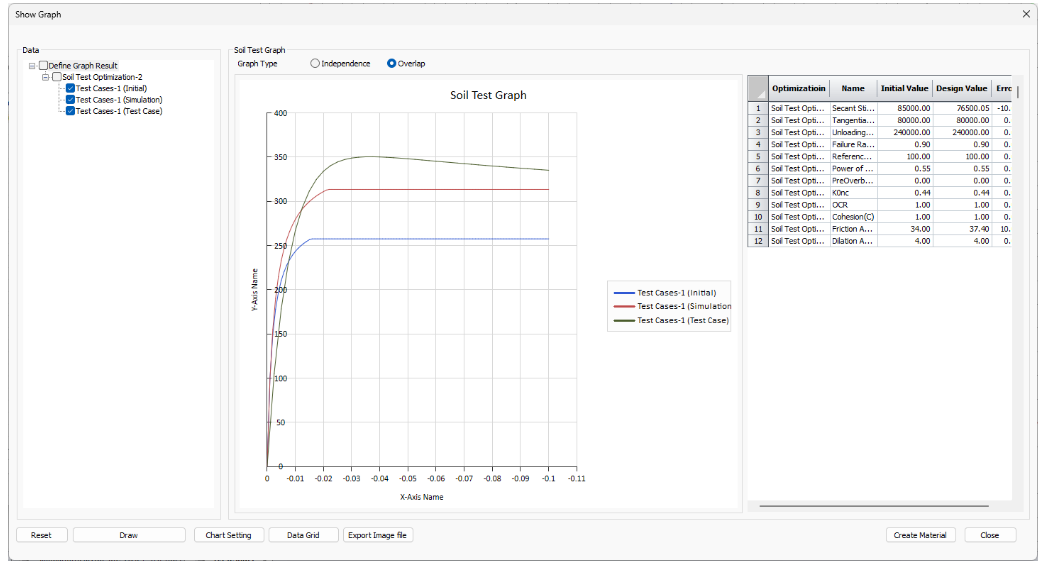


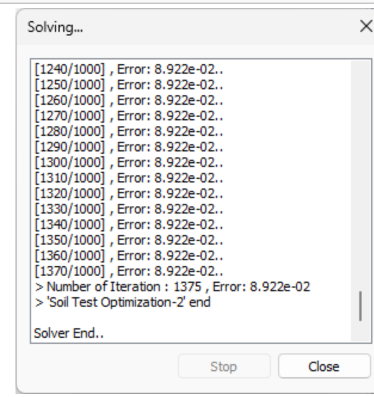


- Design Variable Definition: Users can define the minimum and maximum ranges for material model parameters and select the variables to be included in the optimization process.
- Test Cases: Experimental data inputs can be defined by specifying the test method and corresponding data (function values) obtained from laboratory tests

2.23 Soil Test Optimization





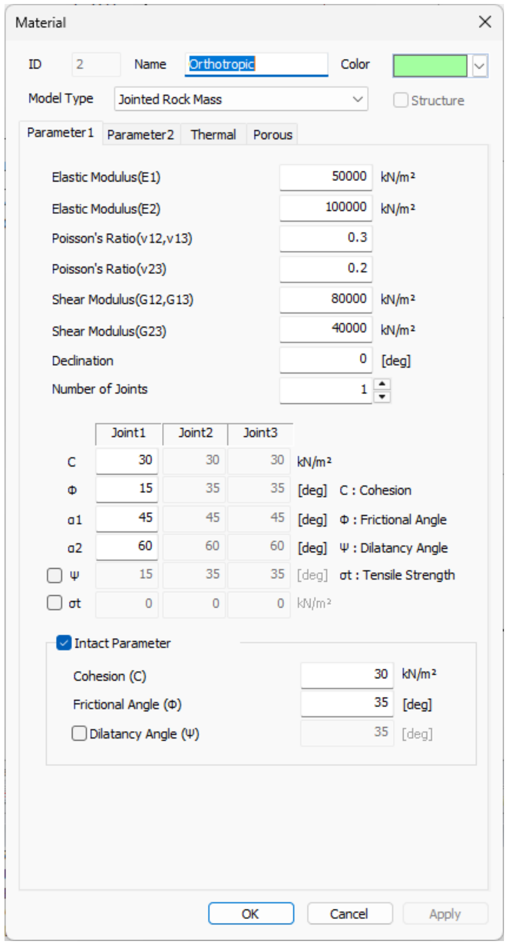


- Analysis: Convergence criteria can be specified to determine the optimization solution.
- Multiple analysis cases can be added, and the Optimize function can be used to obtain parameter values that best match the experimental data within the defined ranges.
- Users can also review comparison graphs, including test data and simulated results based on initial and optimized variables.
- Finally, we can create a new material with optimized inputs.

2.24 Jointed Rock Mass SRM

- The Strength Reduction Method (SRM) has been introduced to support the Jointed Rock Mass model, which represents an orthotropic material and can now factor in strength reduction during analysis.
- Previously, SRM was limited to MC, Drucker–Prager, MMC, Generalized Hoek-Brown, and HS models; it is now extended to include the Jointed Rock Mass model.

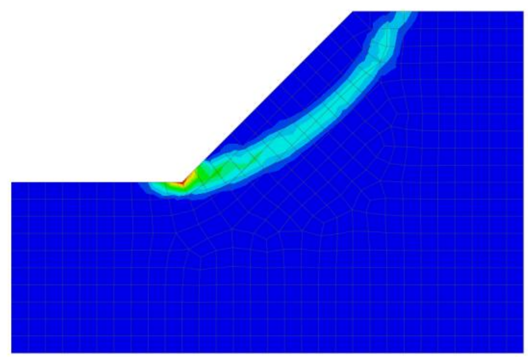
Mesh > Prop./Csys./Func. > Material > Create : Orthotropic > Jointed Rock Mass



$$c_j^{trial} = \frac{1}{F^{trial}} c_j$$

$$\phi_j^{trial} = \arctan\left(\frac{1}{F^{trial}} \tan\phi_j\right)$$

$$\sigma_j^{t(trial)} = \frac{1}{F^{trial}} \sigma_j^{t}$$

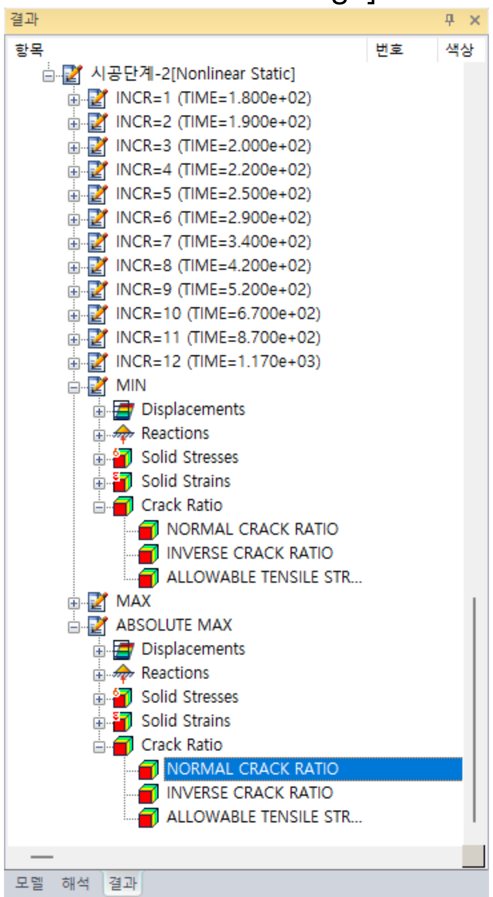


2.25 Min/Max/Abs Max of Nonlinear Static Results in Heat of Hydration Analysis

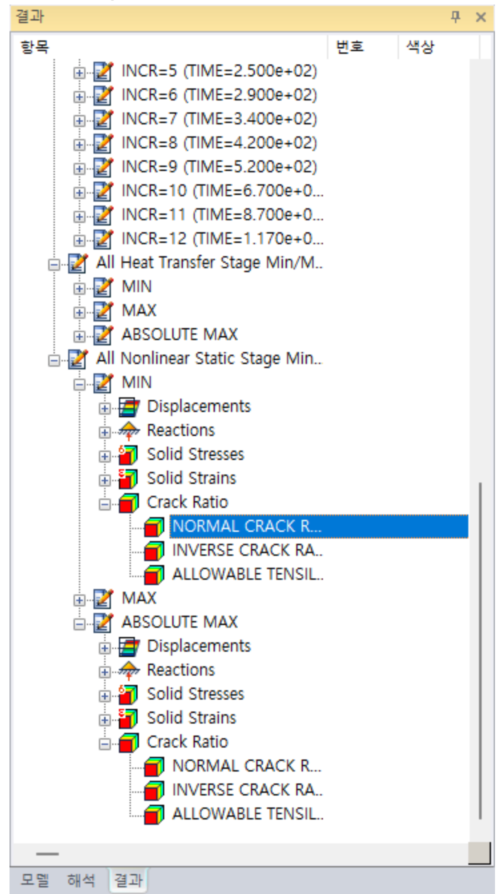
- Nonlinear Static Min/Max/Abs in Hydration Heat Analysis. A function has been added so that you can check the Max value directly in the result tree.
- Min/Max/Abs of Crack Ratio over time in hydration heat analysis is easy to figure out.
- Analysis case > Output control > You can decide whether to output for each construction stage or for the entire stage according to the method of minimum/maximum/absolute results.

Workstree > Results

[Output in each construction stage]



[All nonlinear static stage Min/Max/ABS Max]



Happy Modeling!!