Release Note

Release Date : March 2025

Product Ver. : FEA NX 2025 (v1.1)

ADVANCED NONLINEAR AND DETAIL ANALYSIS

New Paradigm in Advanced Structural Analysis

Enhancements

Analysis

1-1 Coupled Stress, Seepage and Nonlinear Time History Analysis

1-2 Saturated and Unsaturated Soil Properties

1-3 Improved GHE-S Model Parameter Fitting

1-4 Material Evaluator (Bowl Model)

Pre-post

2-1 Skin Friction vs Depth in Pile Interface

2-2 Plastic Status Contour Improvement

2-3 Geometry and Mesh Connection (Geo-Relation)

2-4 Body forces

2-5 Improved Load-to-Mass Conversion

2-6 Enhanced Load Combination Sets

2-7 Construction Stage Wizard Function Improvements

2-8 Multiple Copy Objects Relative to Base Point

2-9 Improved Report Generation

2-10 Automatic Gravity Settings

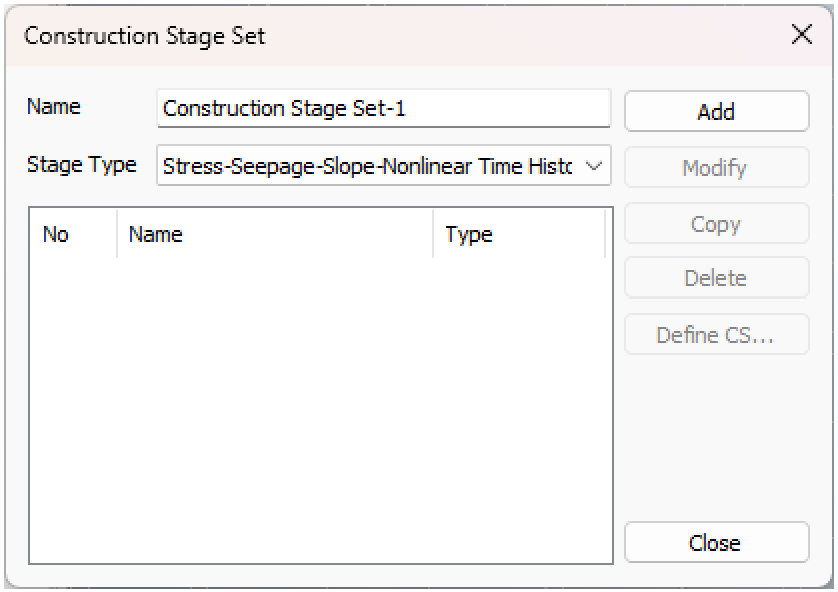
2-11 High-Resolution GUI Support

2-12 Area of Interface Elements

1-1 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.

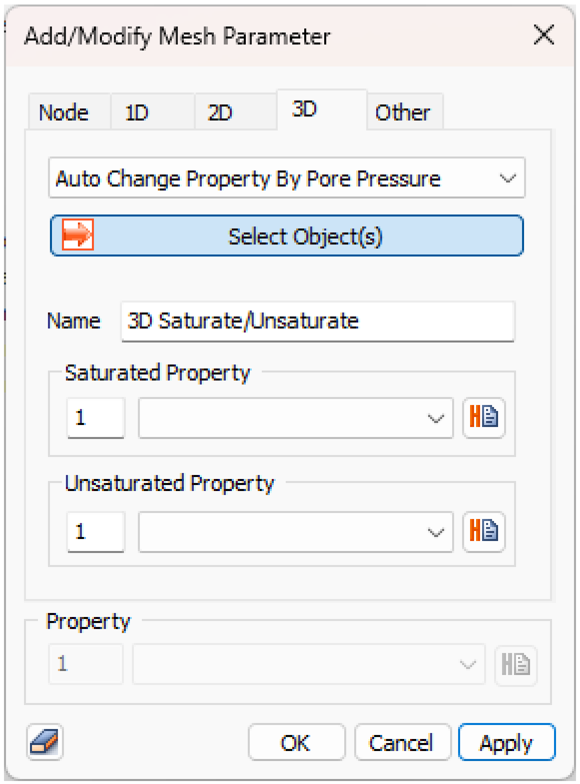
Static Analysis > Construction Stage > Stage Set > Stress-Seepage-Slope-Nonlinear Time History



1-2 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.

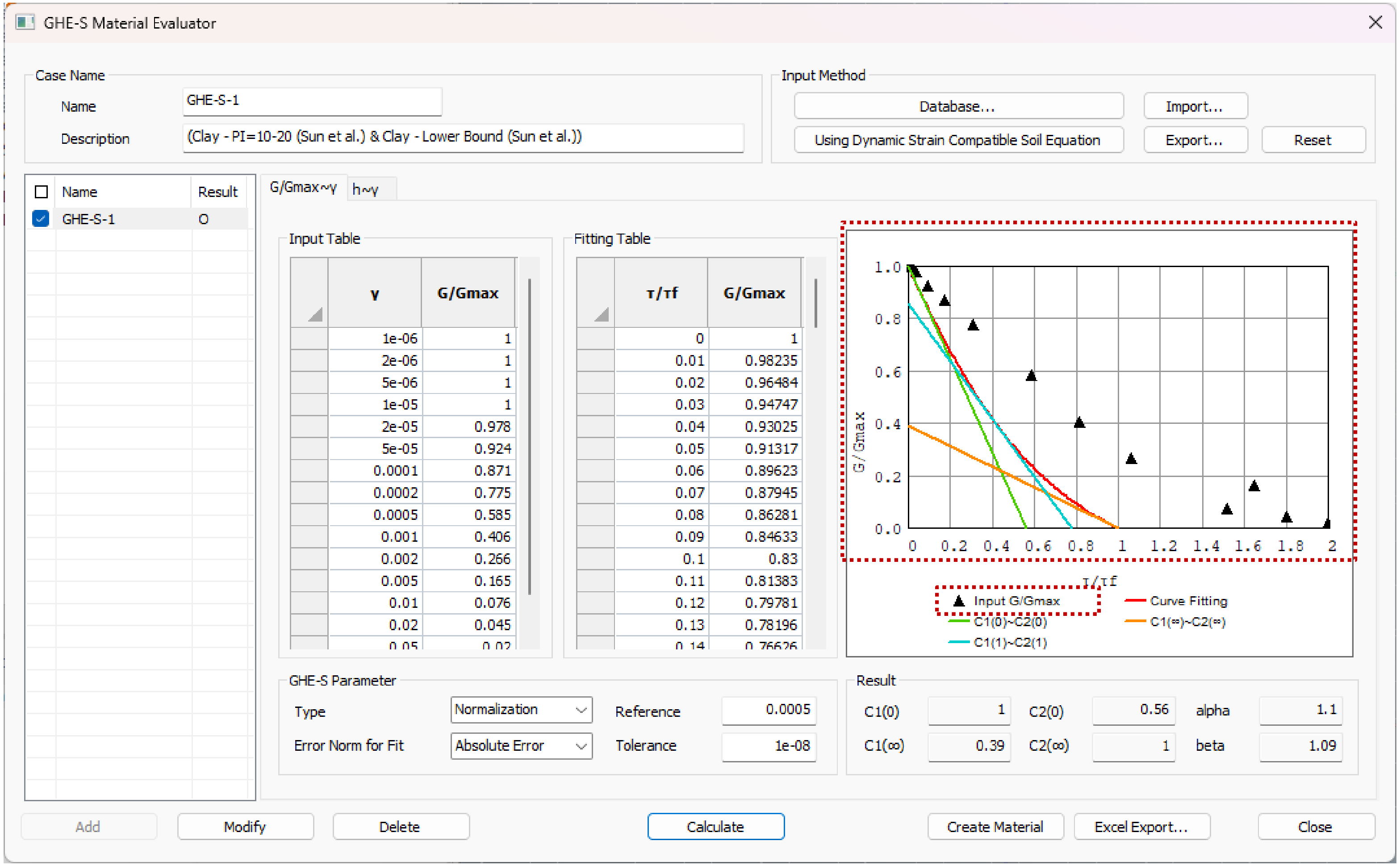
Mesh > Element > Parameters > 2D/3D > Auto Change Property by Pore Pressure



1-3 Improved GHE-S Model Parameter Fitting

The user can plot input data points for GHE-S model parameter determination.

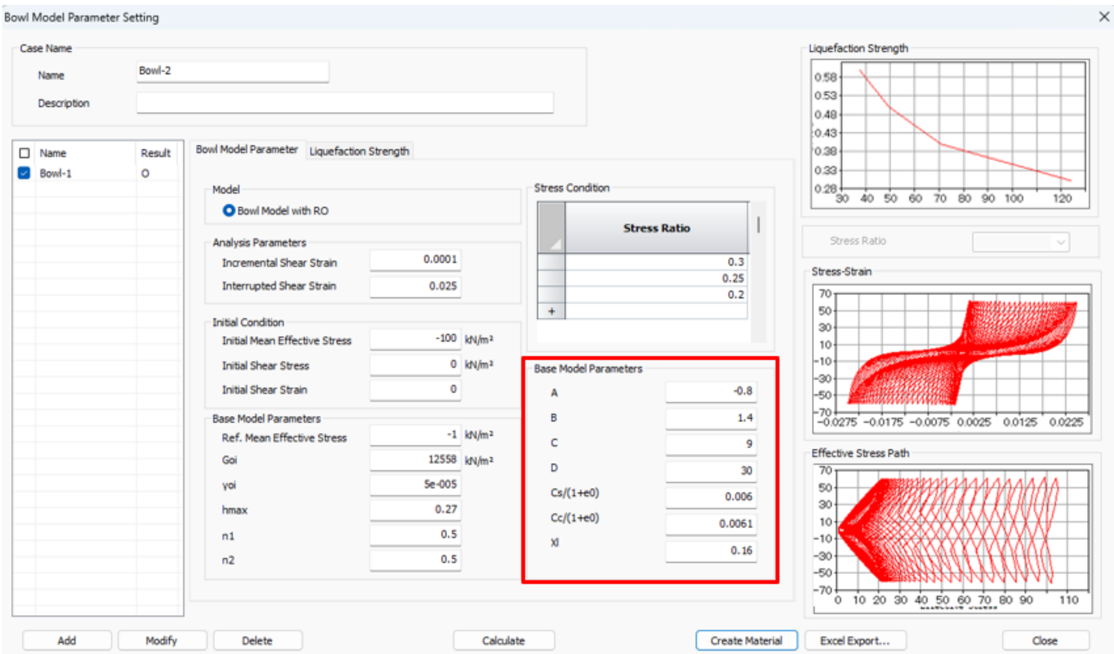
Dynamic Analysis > Tools > Material Evaluator



1-4 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]

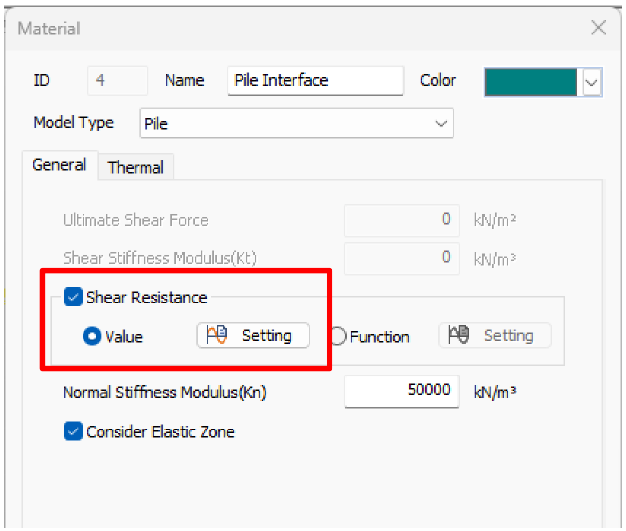
	Description
A, B	Represents the swelling component within dilatancy. The larger the magnitude of |A|, the more pronounced the dilatancy.
C, D	Indicates the compressive component in the dilatancy. 1/C is the slope of the dilatancy with respect to the early shear, 1/D is a tangent of a hyperbola
Cs/(1+e0)	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*,

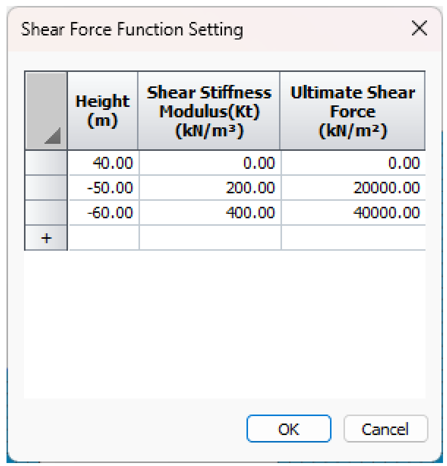
[Bowl Parameters]

2-1 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'

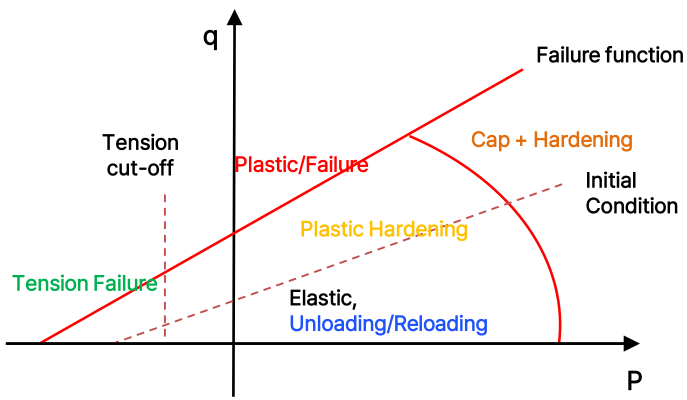
Height :

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

2-2 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.

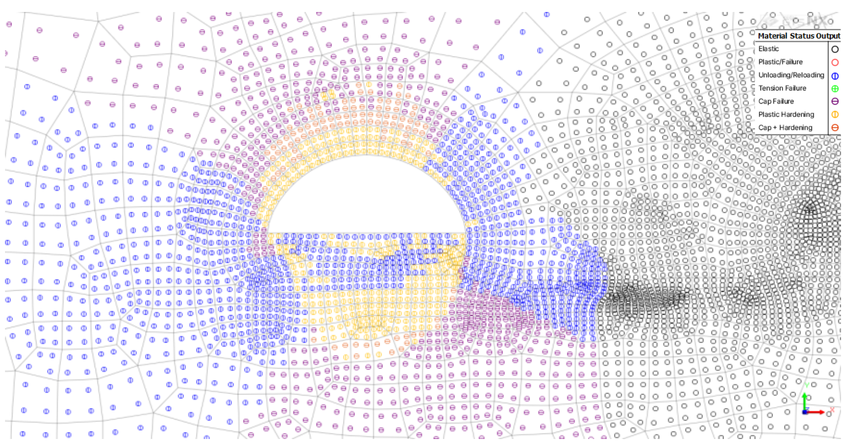
Results Tree > Plane Strain Stress/Solid Stress > Plastic State

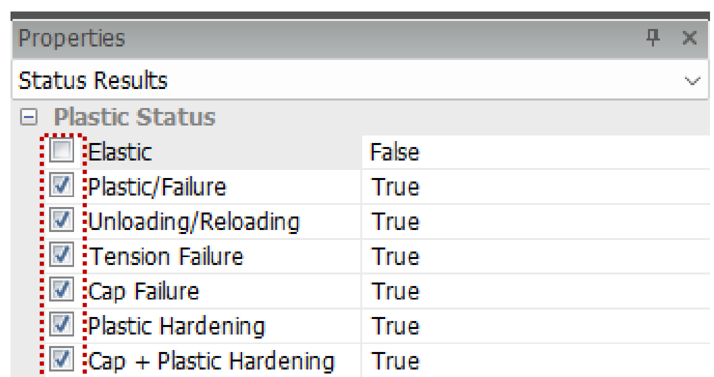


Material Status Output	
Elastic	○
Plastic/Failure	○
Unloading/Reloading	⦶
Tension Failure	⊕
Cap Failure	⊖
Plastic Hardening	⦶
Cap + Hardening	⊖

- 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 window > Plastic State

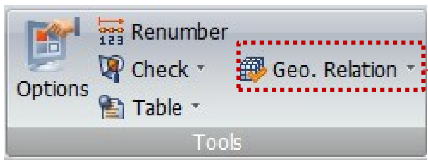




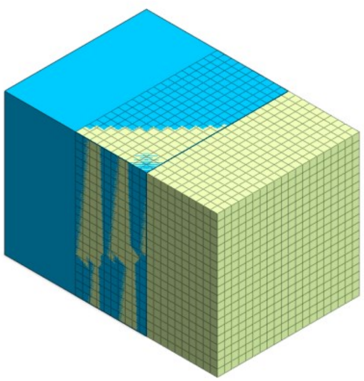
2-3 Geometry and Mesh Connection (Geo-Relation)

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 FEA NX 2025v1.1, users can automatically reconnect using manual editing or tolerance ranges. This enhancement streamlines tasks like load assignment and element extraction.

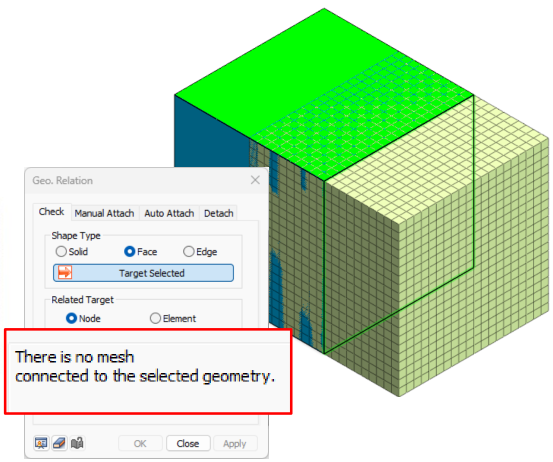
Mesh > Tools > Geo-Relation



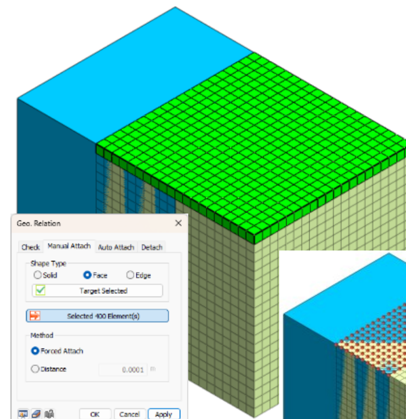
- Check : Verify Geometry-Mesh Connectivity
- Manual Attach : Manually Connect the Target Geometry
- Auto Attach : Automatically connect the target shape (including sub shapes) within a tolerance
- Detach : Disconnect geometry mesh connection



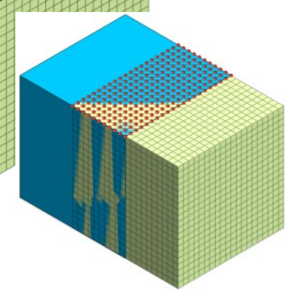
Geometry and Mesh are Over Lapping



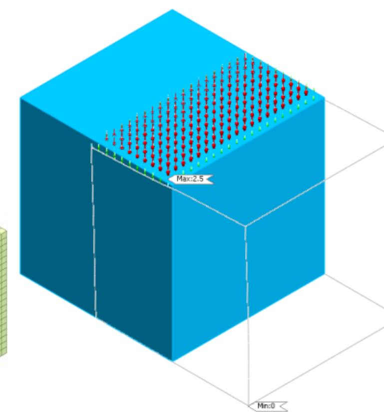
Check if the Geometry is in connection with Mesh or using 'Check'



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



Check the connectivity



Applied Load in the Analysis

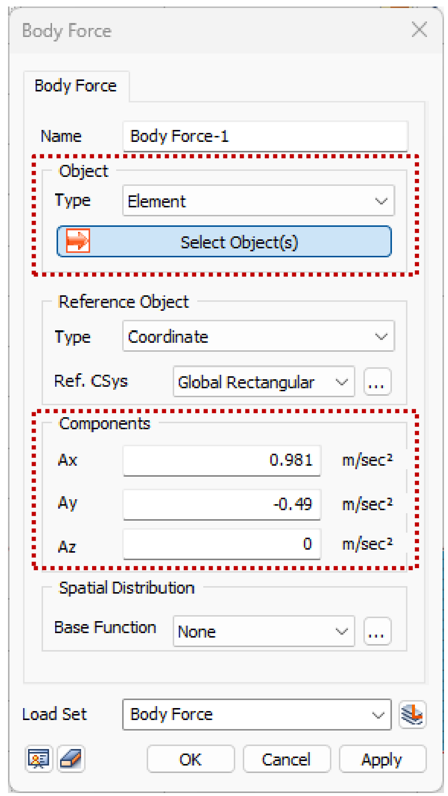
2-4 Body Forces

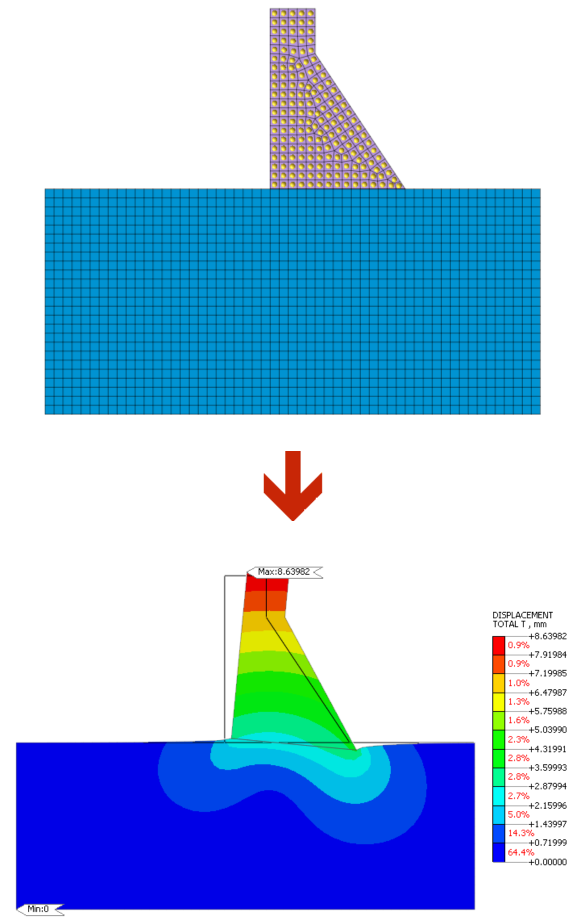
A new load set is introduced to assign the accelerations (pseudo static loads) for respective Elements/Mesh Sets.
In the case of Pseudo Static Loads, user needs to input the Accelerations directly (seismic coefficients*acceleration due to gravity) in the body force definition.

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

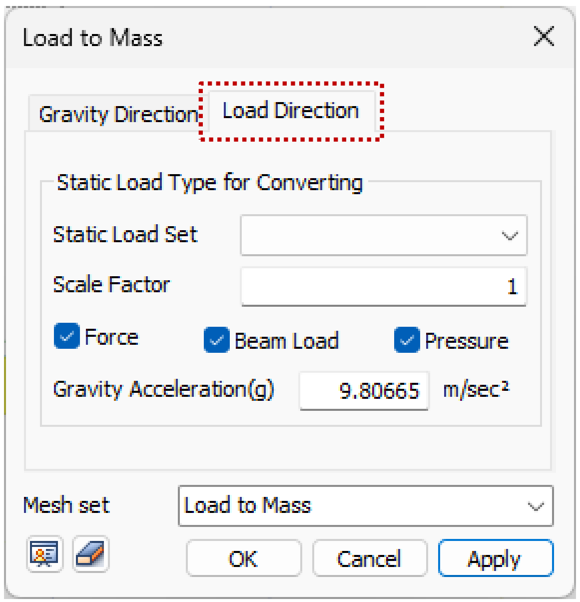


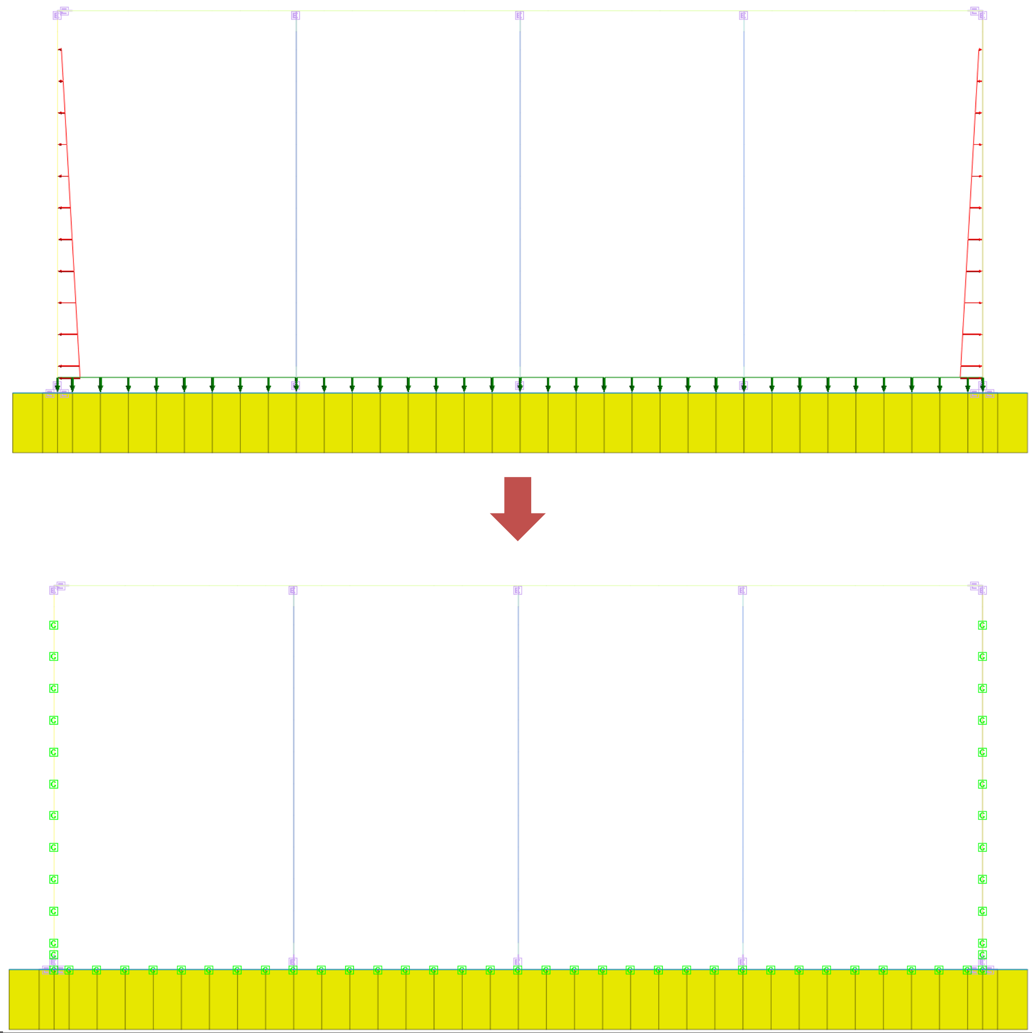


2-5 Improved Load-to-Mass Conversion

Enhancements allow conversion of loads (not limited to gravity direction) into equivalent masses.

Dynamic Analysis > Load > Load to Mass > Load Direction

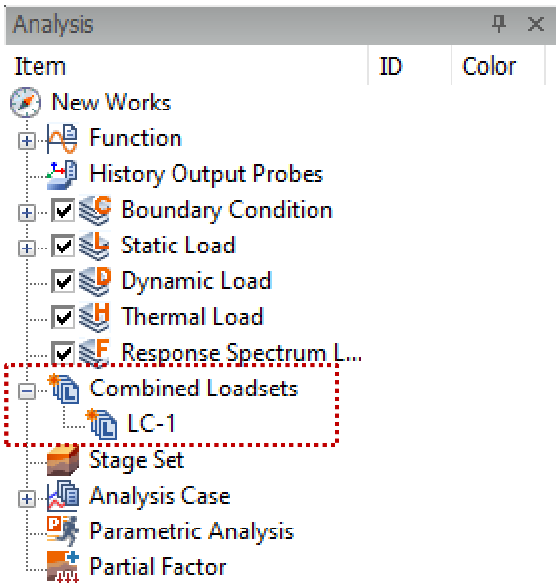




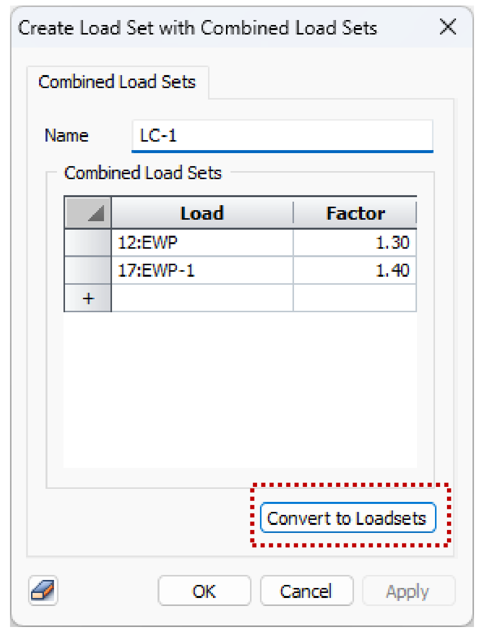
2-6 Accessing the Load Combination & Convert to Loadsets

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

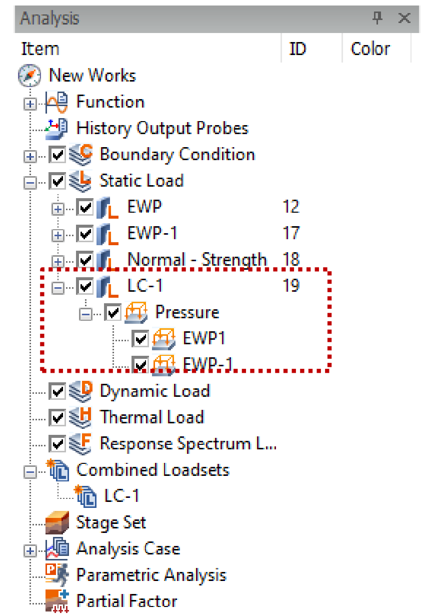
Analysis Workstree > Combined Loadsets



[Accessing the defined Load Combination]



[Converting to Loadsets]



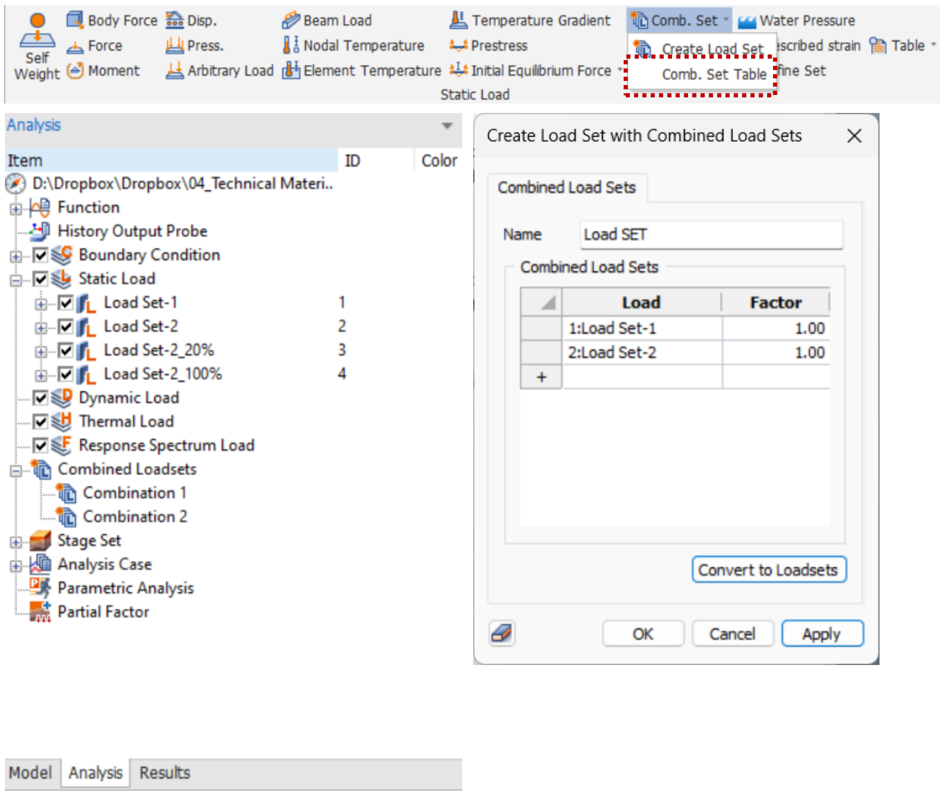
[Converted Loadsets]

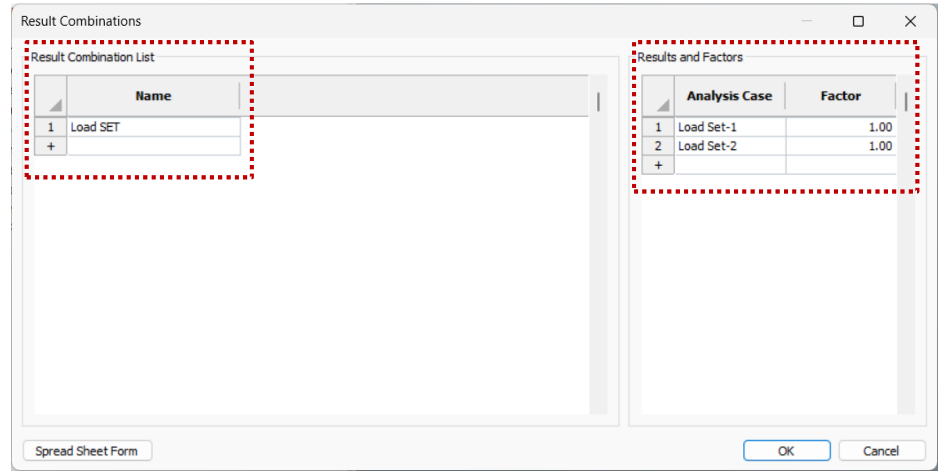
※ 'Load Combination Set' which is not converted to a 'Load Set' can also be used as a 'Load Set' in the Analysis

2-6 Enhanced Load Combination Sets

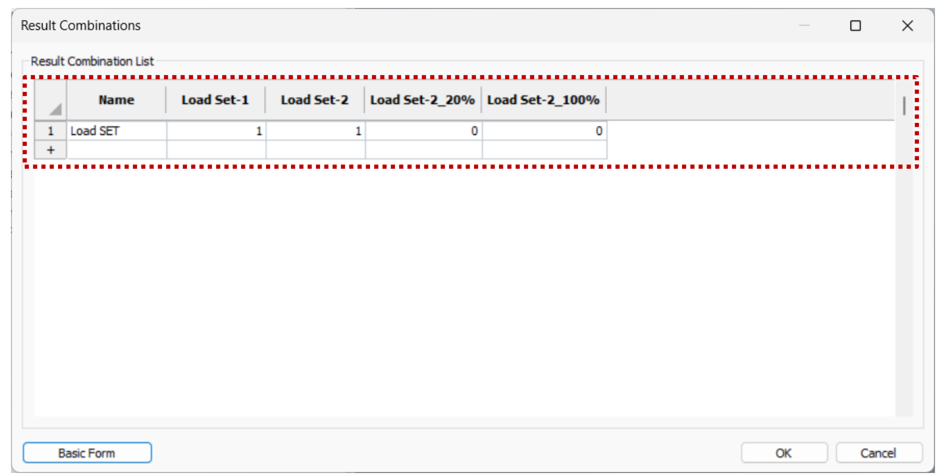
FEA NX 2025 comes with a load combination set table, which can be used to create combination load sets. The spreadsheet format also supports copying and pasting data edited in Excel.

Static Analysis > Static Load > Comb. Set > Comb. Set Table





[Basic format]

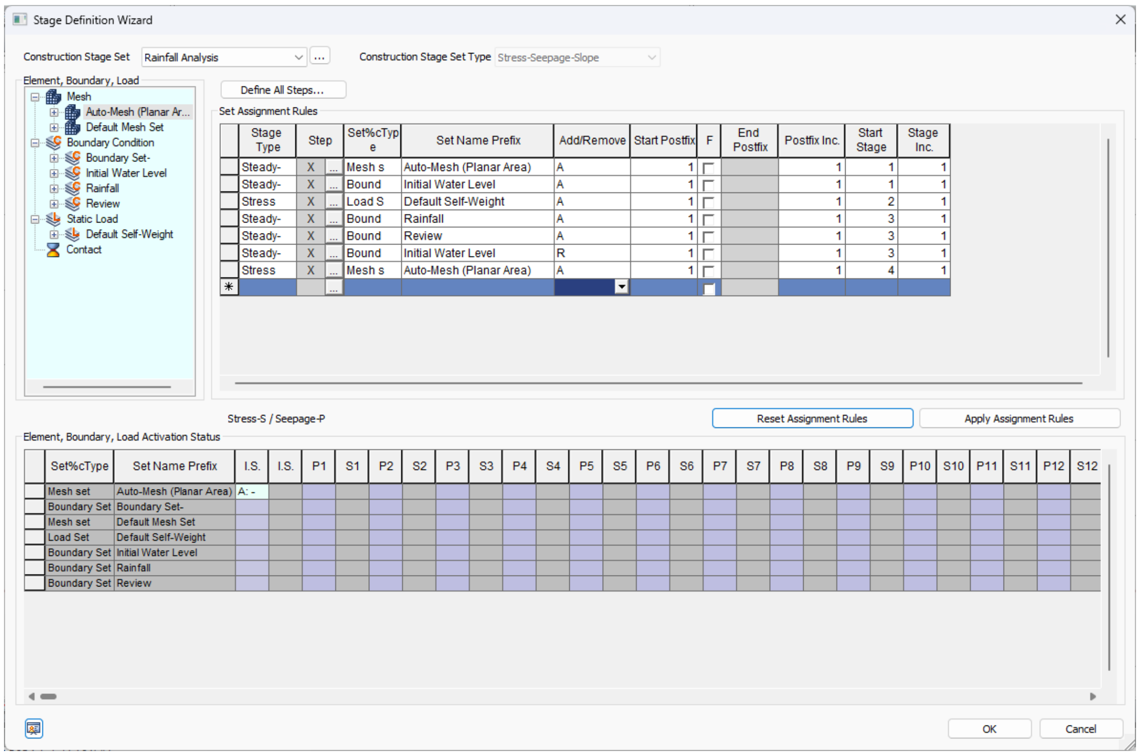


[Spreadsheet format]

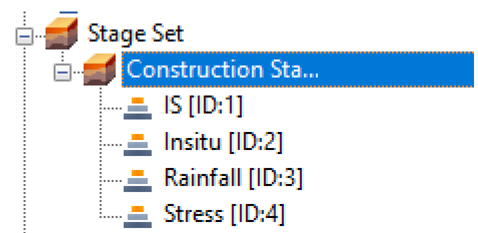
2-7 Construction Stage Wizard Function Improvements

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/Dynamic/Geotechnical Analysis > Construction Stage > Stage Wizard



[Construction Stage Wizard]

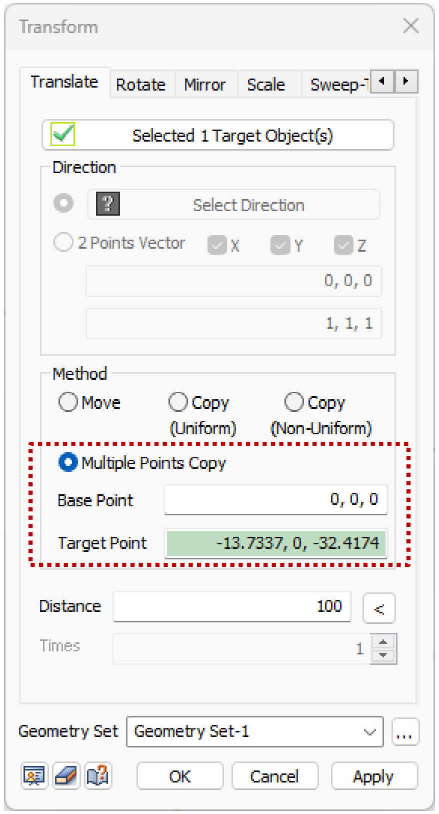


[Construction Stage Set]

2-8 Multiple Copy Objects Relative to Base Point

Enables copying of geometries to multiple locations by manual coordinate input or cursor-based selection.

Geometry > Transform > Translate > Multiple Points Copy

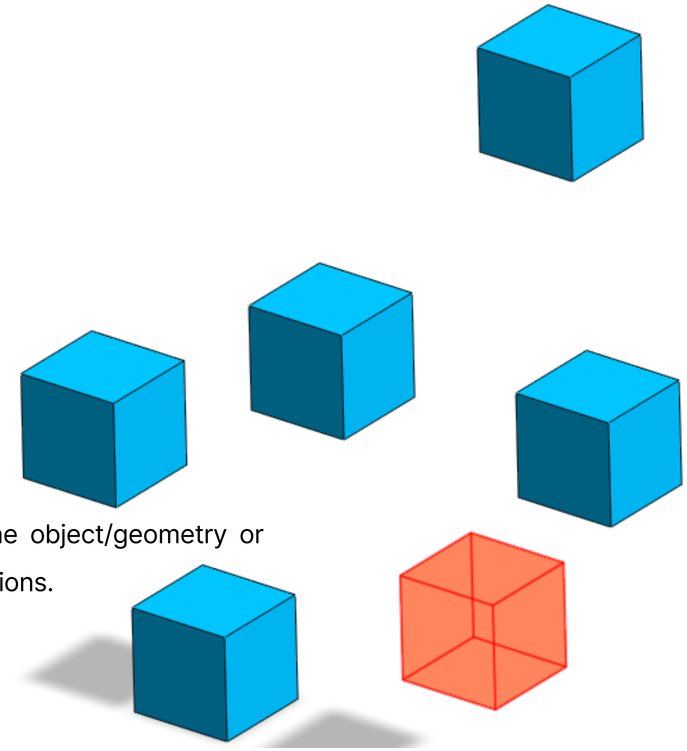


Base Point:

Select the Base Point to copy the object

Target Point:

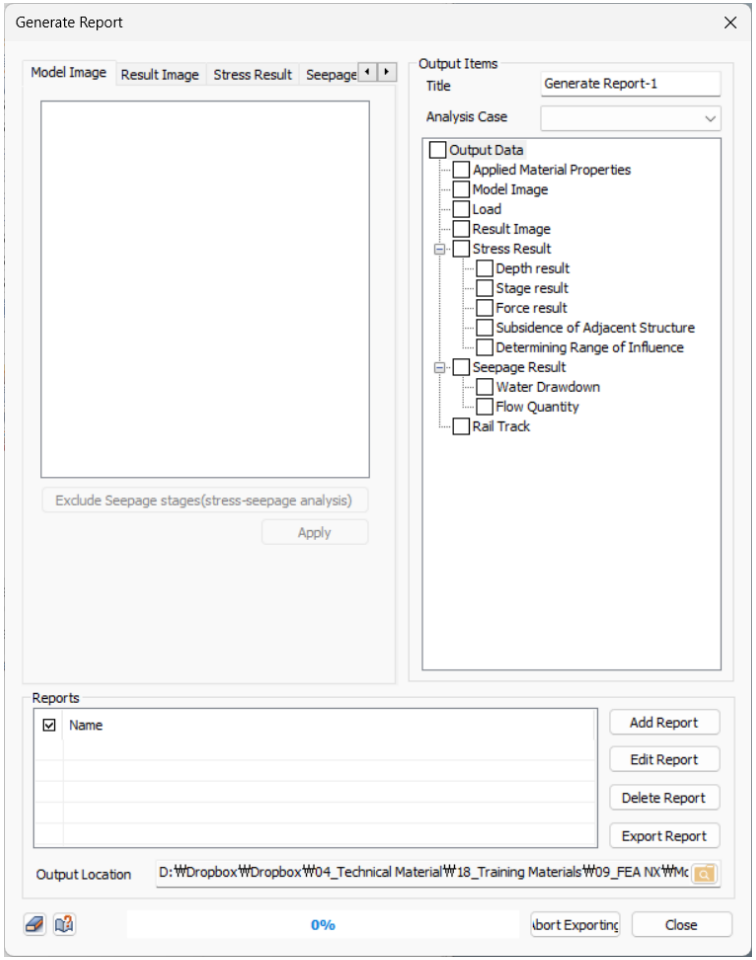
Either manually input the coordinate to copy the object/geometry or use the cursor to copy the object to multiple locations.



2-9 Improved Report Generation

New auto-save functionality reduces human error. Users can resume interrupted report outputs, and some naming conventions have been updated for clarity.

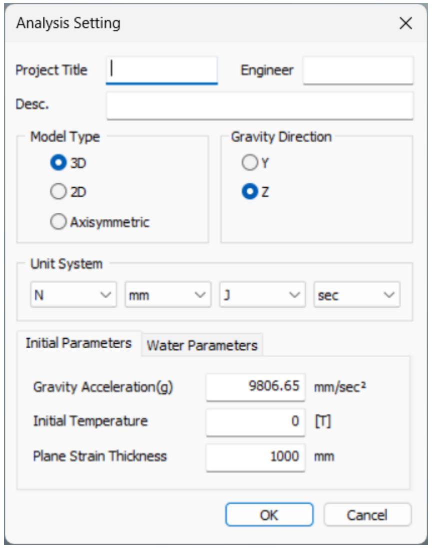
Tools > Export > Generate Report



2-10 Automatic Gravity Settings

Default gravity settings now auto-configure based on 2D/3D analysis type for new models.

Analysis > Analysis Case > Setting



2-11 High-Resolution GUI Support

Enhanced support for 4K resolution (3840x2160 pixels) with auto-scaling of interface elements, icons, and text according to Windows scaling settings.

2-12 Area of Interface Elements

The area of 2D Interface Elements is now auto-calculated, allowing users to seamlessly extract Normal and Sliding Forces. By multiplying the Normal and Tangential Stresses with the computed a rea, users can directly obtain Normal and Shear/Sliding forces. This enhancement simplifies the calculation of the Factor ofSafety (FoS) against sliding, making it more efficient for applications suc h as retaining walls, dams, and other structures.

Mesh > Tools > Table > Element Table

No.	Type	Property	Node1	Node2	Node3	Node4	Node5	Node6	Node7	Node8	Area (m²)
1635	Quadrilateral	2:Interface Property(Wizard)	404	421	391	409	847	848	849	850	1.077280
1636	Quadrilateral	2:Interface Property(Wizard)	406	395	398	415	851	852	853	854	0.987311
1637	Quadrilateral	2:Interface Property(Wizard)	392	404	409	416	855	847	850	856	0.992679
1638	Quadrilateral	2:Interface Property(Wizard)	422	387	401	396	857	858	859	860	1.048936
1639	Quadrilateral	2:Interface Property(Wizard)	395	389	410	398	852	861	862	853	1.016140
1640	Quadrilateral	2:Interface Property(Wizard)	399	405	416	414	863	864	856	865	0.766063
1641	Quadrilateral	2:Interface Property(Wizard)	408	385	417	410	866	867	868	862	1.127401
1642	Quadrilateral	2:Interface Property(Wizard)	405	423	392	416	864	869	855	856	0.969127
1643	Quadrilateral	2:Interface Property(Wizard)	397	403	393	386	870	871	872	873	0.948060
1644	Quadrilateral	2:Interface Property(Wizard)	387	406	415	401	858	851	854	859	1.084856
1645	Quadrilateral	2:Interface Property(Wizard)	400	402	413	411	874	875	876	877	1.048146
1646	Quadrilateral	2:Interface Property(Wizard)	393	400	411	386	872	874	877	873	0.903385
1647	Quadrilateral	2:Interface Property(Wizard)	421	397	386	391	848	870	873	849	0.978912
1648	Quadrilateral	2:Interface Property(Wizard)	389	418	408	410	861	878	866	862	1.042463
1649	Quadrilateral	2:Interface Property(Wizard)	402	396	401	413	875	860	859	876	1.096466
1650	Quadrilateral	2:Interface Property(Wizard)	391	386	411	394	849	873	877	879	0.735343
1651	Quadrilateral	2:Interface Property(Wizard)	394	390	412	419	879	880	881	882	0.368580
1652	Quadrilateral	2:Interface Property(Wizard)	420	413	401	415	883	876	859	854	1.211228
1653	Quadrilateral	2:Interface Property(Wizard)	417	414	407	412	868	865	884	881	0.583008
1654	Quadrilateral	2:Interface Property(Wizard)	394	420	388	390	879	883	885	880	0.504190
1655	Quadrilateral	2:Interface Property(Wizard)	417	412	390	388	868	881	880	885	0.635852
1656	Quadrilateral	2:Interface Property(Wizard)	399	414	417	385	863	865	868	867	0.876809
1657	Quadrilateral	2:Interface Property(Wizard)	407	414	416	409	884	865	856	850	0.629694
1658	Quadrilateral	2:Interface Property(Wizard)	420	415	398	388	883	854	853	885	0.882558
1659	Quadrilateral	2:Interface Property(Wizard)	412	407	409	419	881	884	850	882	0.575199
1660	Quadrilateral	2:Interface Property(Wizard)	394	419	409	391	879	882	850	849	0.771589
1661	Quadrilateral	2:Interface Property(Wizard)	394	411	413	420	879	877	876	883	1.114703
1662	Quadrilateral	2:Interface Property(Wizard)	417	388	398	410	868	885	853	862	1.024023

1D 2D 3D Spring Rigid Link Elastic Link Interpolation Interface Embedded Axisymmetric Line Embedded Axisymmetric Point Gauging Shutoff Wall Infinite

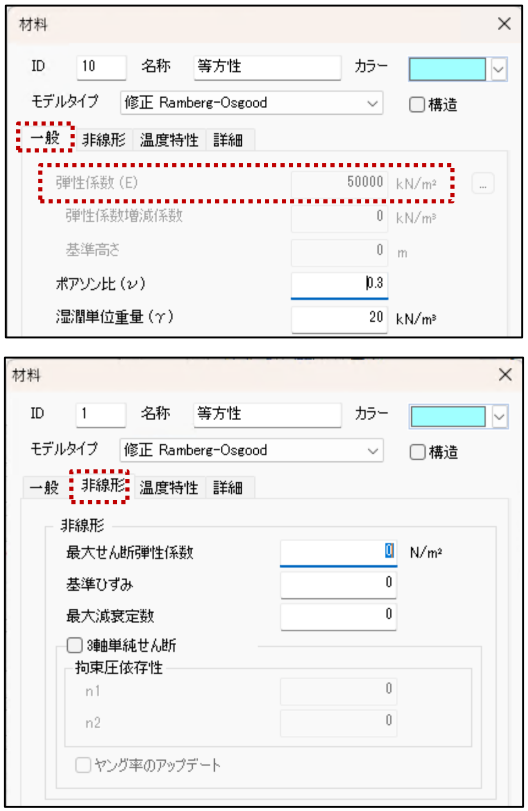
Specification Changes

Pre-post function change

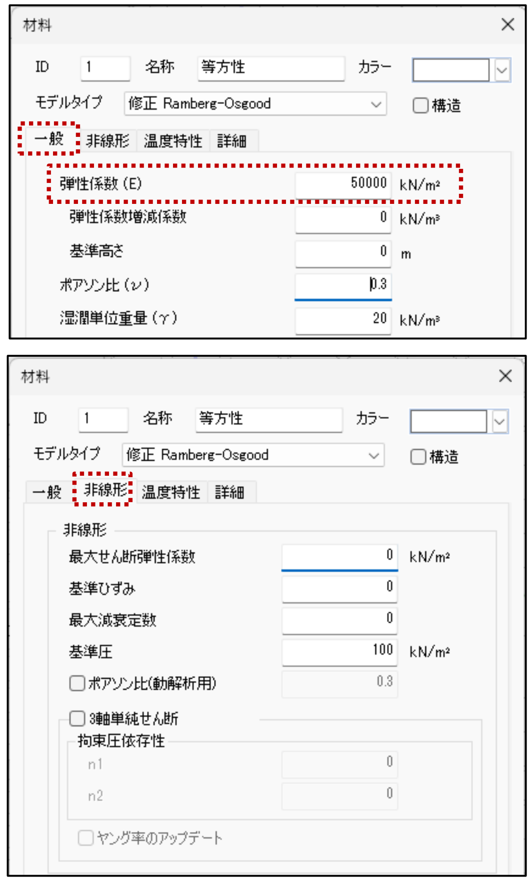
3-1. Changes in the usage of the modified RO model, modified HD model, GHE-S model, and bowl model

3-1. Changes in the usage of the modified RO model, modified HD model, GHE-S model, and bowl model

In previous versions, it was not possible to set the [Elastic Modulus] in the General tab when using the Modified RO Model, Modified HD Model, GHE-S Model, or Bowl Model.
In addition, when analyzing in the [Static to Dynamic] construction stage, the [Maximum Shear Modulus] in the Nonlinear tab was used, and the initial stress in the [Static] step was calculated using the value back-calculated with E=2(1+ν)G.
From FEA NX 2025 (v1.1) onwards, the [Static Constants] and [Dynamic Constants] have been modified so that they can be used separately, and the specifications have been changed so that [Elastic Modulus] and [Poisson's Ratio] in the General tab are used in the static stage, and [Maximum Shear Modulus] and [Poisson's Ratio (for Dynamic Analysis)] in the Nonlinear tab are used in the dynamic stage.



Previous version



FEA NX 2025 (v1.1)

*If you analyze previously created data with FEA NX 2025 (v1.1), you may get different results from the previous results. The reason is that the [Elastic Modulus] used in the calculation is different. You need to enter E calculated using the formula E=2(1+ν)G in the [General] tab and run the analysis again.