Multi-disciplinary integrated analysis solution for optimal design

midas NFX **2024 Enhancements**

midas NFX Release Note 2024

Major Updates

- CAD Interface update
- Additional methods included for response spectrum analysis: Rigid response Gupta, Lindley-Yow method and Missing mass method.
- Added functionality for load combination in analysis result table (simple sum, SRSS, envelop).
- Enhanced processing of time history data in structural analysis (final value, 1st interpolation, 2nd interpolation, specific value, cyclic repetition).
- Added capability to copy loads/boundary conditions when duplicating elements.
- Improved support for high-resolution GUI in Windows.
- Added operations for faces (set operations: difference, intersection, embedding).
- Added import functionality for NFX files (*.nfx).
- Added automatic bolt definition feature based on faces and 2D elements.
- Added automatic sweep generation feature based on lines and cross-sectional shapes.
- 2D, 3D Layered Mesh Generation Techniques Development
- Other improvements

midas NFX provides complete integration/linked analysis of structure/heat/fluid/optimization using a single model in a single work environment and a familiar environment for designers and technology through a windows-based GUI.

CAD Interface Update

The CAD Interface was updated according to the CAD Version update. Support for the latest version of CAD Interface may be delayed depending on the supplier's update environment. If the newest version is not supported, please convert it to Parasolid or STEP file. We will do our best to reflect on the latest version of CAD quickly.

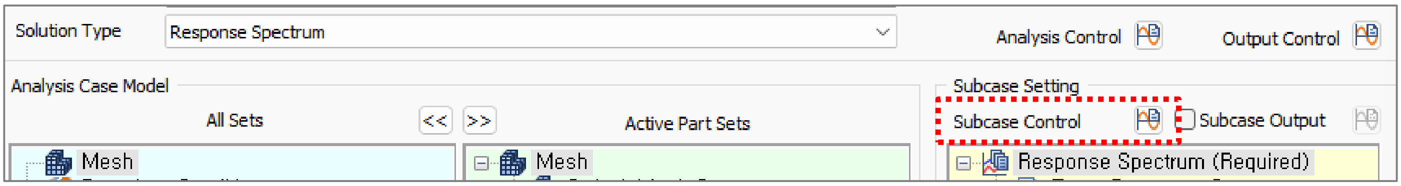
Type	Extention	Version
Parasolid	x_t, xmt_txt, x_b, xmt_bin	9.0 ~ 36.0
ACIS	sat, sab, asat, asab	R1 ~ 2024.1.0
STEP	stp, step	AP203, AP214, AP242
IGES	igs, iges	Up to 5.3
Pro-E / Creo	prt, prt.*, asm, asm.*	16 ~ Creo 10.0
SolidWorks	sldprt, sldasm, slddrw	98 ~ 2024
CATIA V4	model, exp, session	4.1.9 ~ 4.2.4
CATIA V5	CATPart, CATProduct	V5 R8 ~ V5-6R2024
Unigraphics	prt	11 ~ NX2306
Inventor Part	ipt	V6 ~ V2024
Inventor Assembly	iam	V11 ~ V2024
SolidEdge	par, asm, psm	V18 ~ SE2024

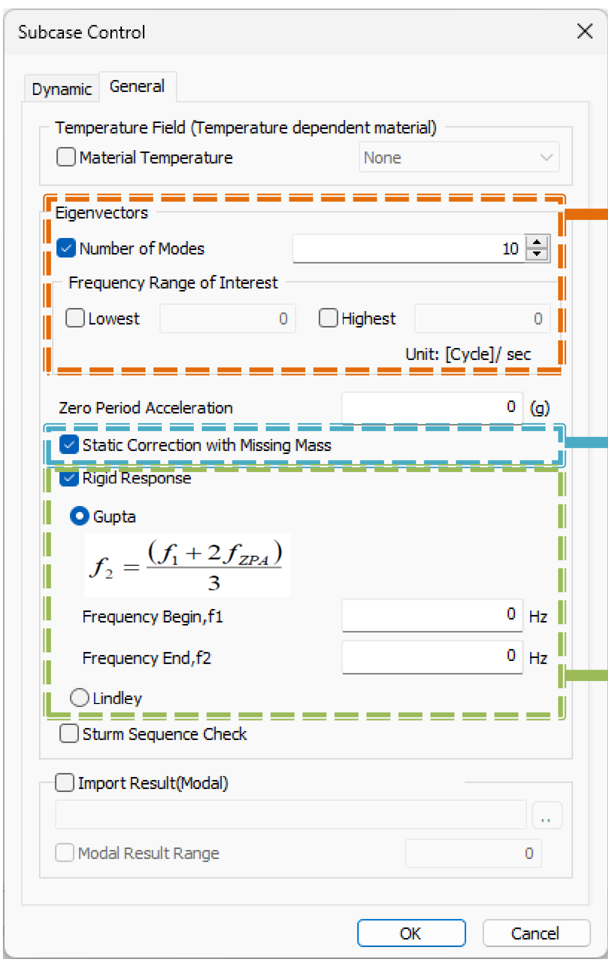
Enhancement of Response Spectrum Functionality

< Purpose and Usage>

The U.S. Nuclear Regulatory Commission (U.S.NRC) has provided analytical approaches for both variability and stiffness mode sections to enable complete response spectrum analysis for all segments. Development and integration of analytical capabilities for each have been completed.

For the variability segment, the Gupta, Lindley-Yow method has been developed in accordance with the guidelines presented in Reg. Guide 1.92. Additionally, the Missing mass method (Static ZPA) has been developed to address stiffness modes omitted from the mass participation ratio after ZPA. With the functionality to specify mode counts and frequency ranges from existing Eigenvalue analyses, comprehensive response spectrum analysis is achievable. Detailed usage within the program is outlined below.





[Specifying Frequency Calculation Range]

- It is now possible to add a limit on the number of modes within the user-defined range (terminate if there are insufficient modes).
- When activated, only Eigenvalue modes within the specified frequency range will be computed.

[Missing Mass method]

- When activated, computations will be performed using the Static ZPA method for the missing mass participation ratios.

[Rigid Response method]

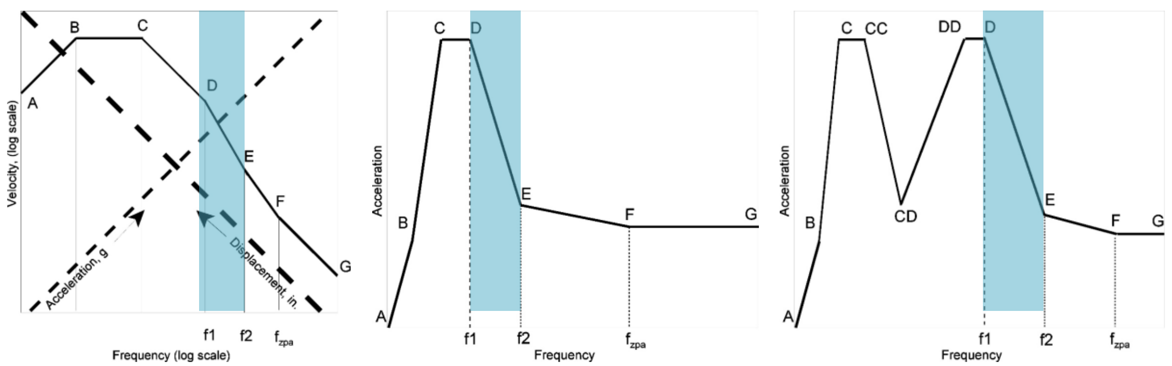
- When activated, the rigid response coefficient α_i will be calculated and incorporated for the variability segment.
- Depending on the selection of the Gupta, Linley-Yow method, the calculation method will be distinguished and applied accordingly.

Addition of Rigid Response Method to Response Spectrum Analysis

〈 Purpose and Usage〉

During seismic safety evaluations, the widely used response spectrum is divided into AB, BC, and CD segments, each exhibiting amplification ranges for displacement, velocity, and acceleration respectively. The DE segment is characterized by amplification and variability ranges of stiffness modes, while the segment following F (ZPA) is dominated by stiffness modes.

The U.S. Nuclear Regulatory Commission (U.S.NRC) has provided analytical approaches for both variability and stiffness mode segments to enable complete response spectrum analysis for all segments. Development and integration of analytical capabilities for the variability segment have been completed, specifically with regards to the Gupta, Lindley-low method, to consider the variability segment.



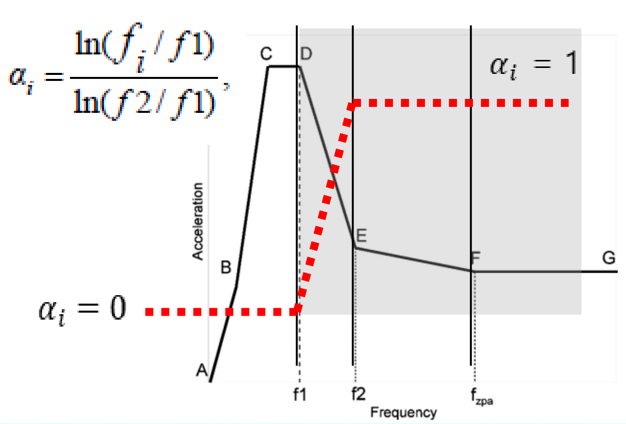
Regions of a broad-banded | A narrow banded | A multiple narrow banded

U.S.NRC Regulatory Guide 1.92 / Response Spectrum Type

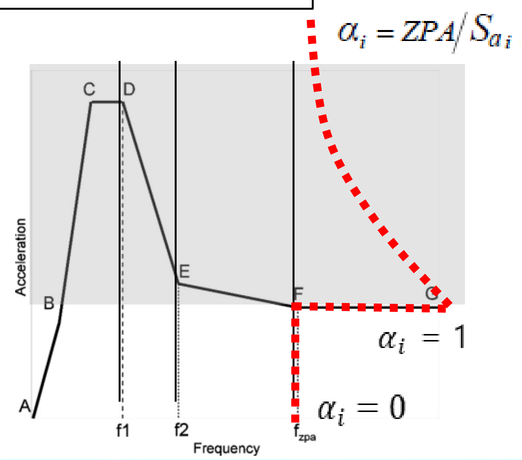
▶ rigid response coefficient α_i

1) $\alpha_i = 0$: Periodic response
2) $0 < \alpha_i < 1$: Periodic + Rigid response
3) $\alpha_i = 1$: Rigid response

Gupta method



Lindley-Yowmethod

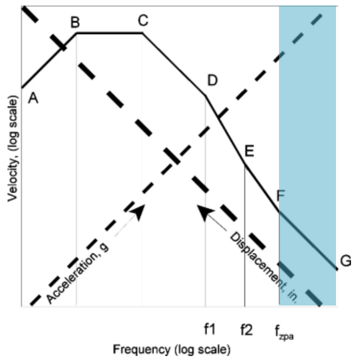


Addition of Missing Mass Method to Response Spectrum Analysis

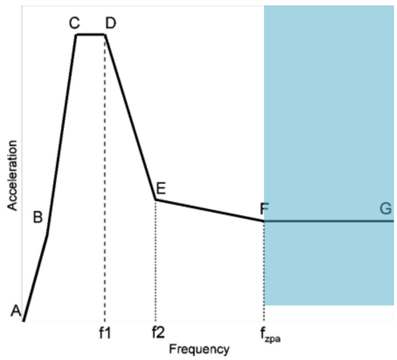
〈 Purpose and Usage〉

During seismic safety evaluations, the commonly used response spectrum is divided into AB, BC, and CD segments, each exhibiting amplification ranges for displacement, velocity, and acceleration respectively. The DE segment is characterized by amplification and variability ranges of stiffness modes, while the segment following F (ZPA) is dominated by stiffness modes.

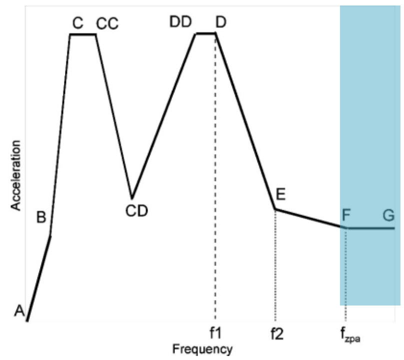
The U.S. Nuclear Regulatory Commission (U.S.NRC) has provided analytical approaches for both variability and stiffness mode segments to enable complete response spectrum analysis for all segments. Development and integration of analytical capabilities for addressing missing stiffness modes through the Missing mass method (Static ZPA method) has been completed to supplement this analysis.



Regions of a broad-banded



A narrow banded



A multiple narrow banded

U.S.NRC Regulatory Guide 1.92 / Response Spectrum Type

Percentage Modal Effective Mass

MODE NUMBER	T1	T2	T3	R1	R2	R3
1	18.17%	0.00%	0.00%	0.00%	28.60%	57.92%
2	16.22%	0.00%	0.00%	0.00%	11.28%	1.56%
⋮						
16	2.36%	3.08%	36.94%	0.05%	1.91%	0.67%
17	15.54%	0.43%	5.25%	0.01%	13.61%	5.48%
18	0.00%	8.48%	20.10%	13.29%	0.00%	0.00%
TOTAL	84.08%	81.73%	87.01%	69.19%	64.80%	84.02%
Residual mass	**15.02%**	**18.27%**	**12.99%**	**30.81%**	**35.20%**	**15.98%**

Static ZPA : **Static Analysis = Residual mass percentage x ZPA (Zero Period Acceleration)**

▶ Complete Combination Method (Gupta, Lindley-Yow, Missing mass)

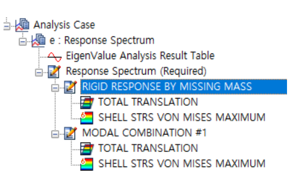
$$R_{p_i} = \left[1-\alpha_i^2\right]^{1/2} R_i$$

$$R_{r_i} = \alpha_i R_i$$ → Rigid Response method (Gupta, Lindley-Yow)

$$R_{pI} = \left[\sum_{i=1}^{n}\sum_{j=1}^{n} \varepsilon_{ij} R_{p_i} R_{p_j}\right]^{1/2}$$, where n = number of modes below f_{zpa}

$$R_{rI} = \sum_{i=1}^{n} R_{r_i} + R_{MissingMassI}$$ → Missing Mass method (Static ZPA)

$$R_I = \left[R_{rI}^2 + R_{pI}^2\right]^{1/2}$$



Missing Mass method Analysis Result

Complete Combination method Analysis Result

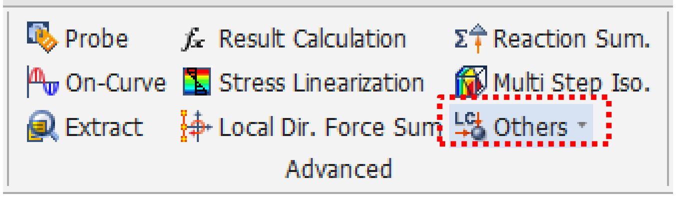
Added functionality for load combination in analysis result table

〈 Purpose and Usage〉

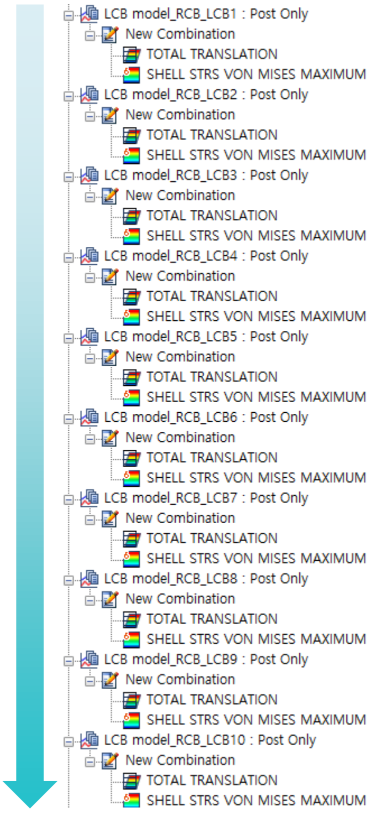
In structural analysis, various loads such as live loads, wind loads, fluid loads, pressure loads, seismic loads, etc., exist alongside the self-weight of the structure. Ensuring safety under different conditions such as installation, operation, and extreme conditions requires verifying against various load combinations according to standards.

Load combinations can be generated either by combining loads themselves or by using analyzed results during the post-processing stage. While a single load combination feature was available regardless of the analysis type in the existing NFX, it was inconvenient for simultaneous generation of multiple loads.

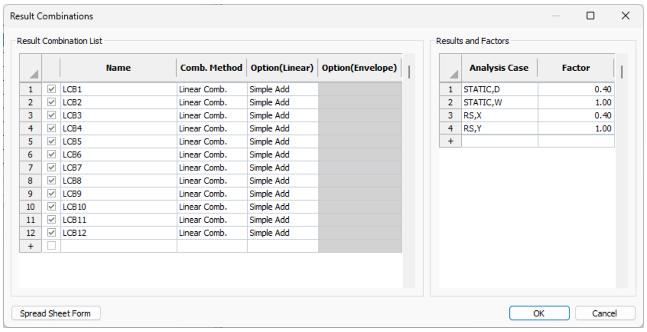
To facilitate the easy creation of multiple load combinations, additional functionality in the form of a table is provided. This allows for the batch creation of combinations with various conditions such as SRSS, simple summation, enveloping, etc., in a single table or editing data using Excel for compatibility. The regeneration is determined based on the check marks within the feature.



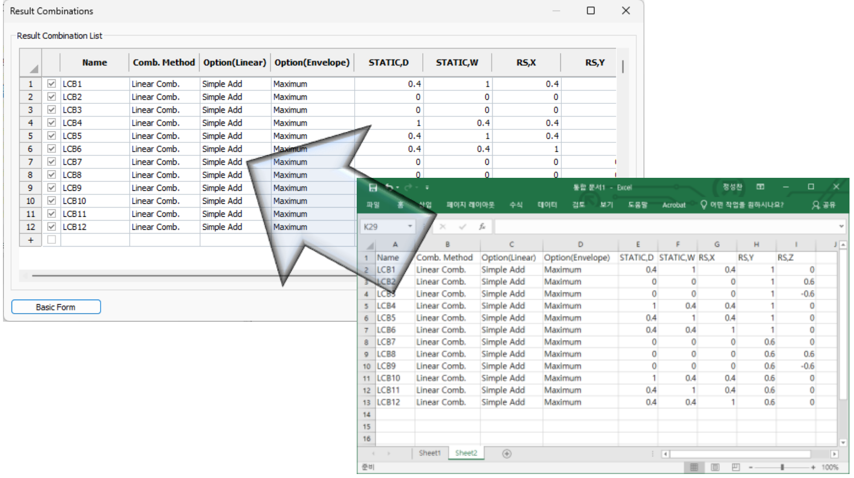
Load combination results are automatically generated sequentially.



▶ **Default Form of Result Combination (Table): Presented in a menu format, allowing for the inspection of information for each combination.**



▶ **Spreadsheet Form of Result Combination (Table): Represented in a table format, compatible with Excel data, facilitating easy editing and manipulation.**

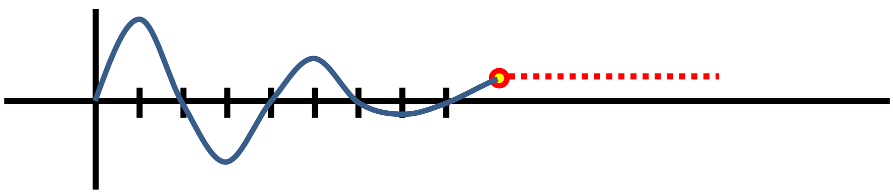
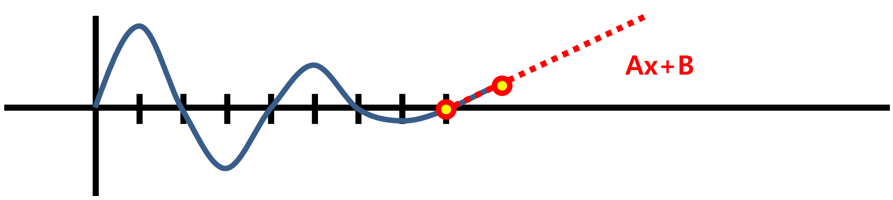
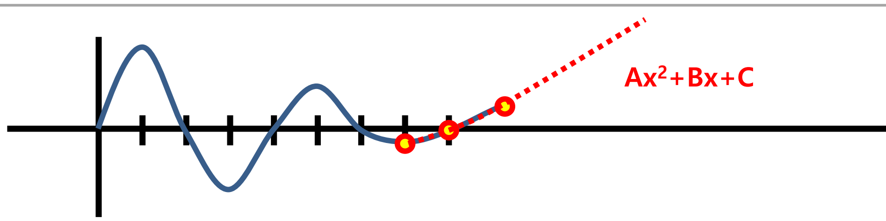
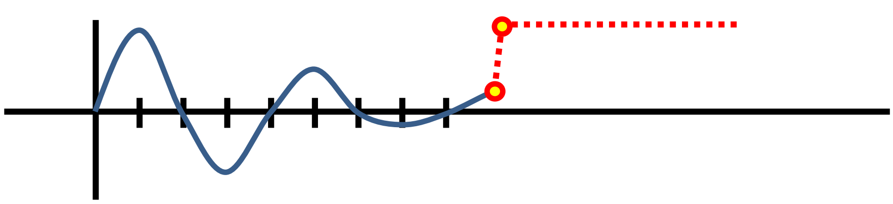
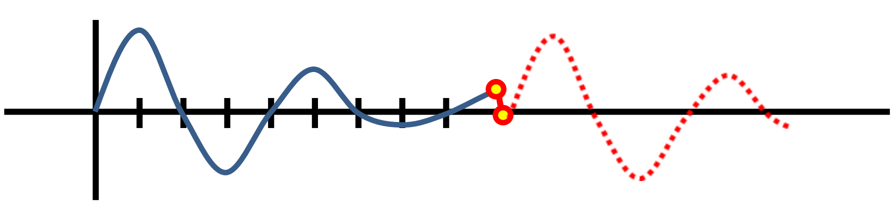


Additional Processing Items for Time Function Data in Structural Analysis

〈 Purpose and Usage 〉

In the existing time function for structural analysis, linear extrapolation was performed based on the final two data points for items outside the data. However, as the complexity or scope of analysis expanded, there was inconvenience in increasing difficulty of definition. In particular, to simulate repetitive situations, it is necessary to define directly for all analysis areas.

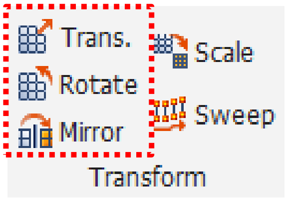
For processing items outside the data in structural analysis time functions, they are provided in five categories. Beyond a certain value, automatic definition is applied to the final value or a specific value. For recurring operating conditions, the range specified for automatic recognition and repetition is set to infinite time as a periodic condition, and additional support is provided for second-order nonlinear extrapolation, including existing first-order linear extrapolation. Refer to the figure below for details on data processing methods for each item.

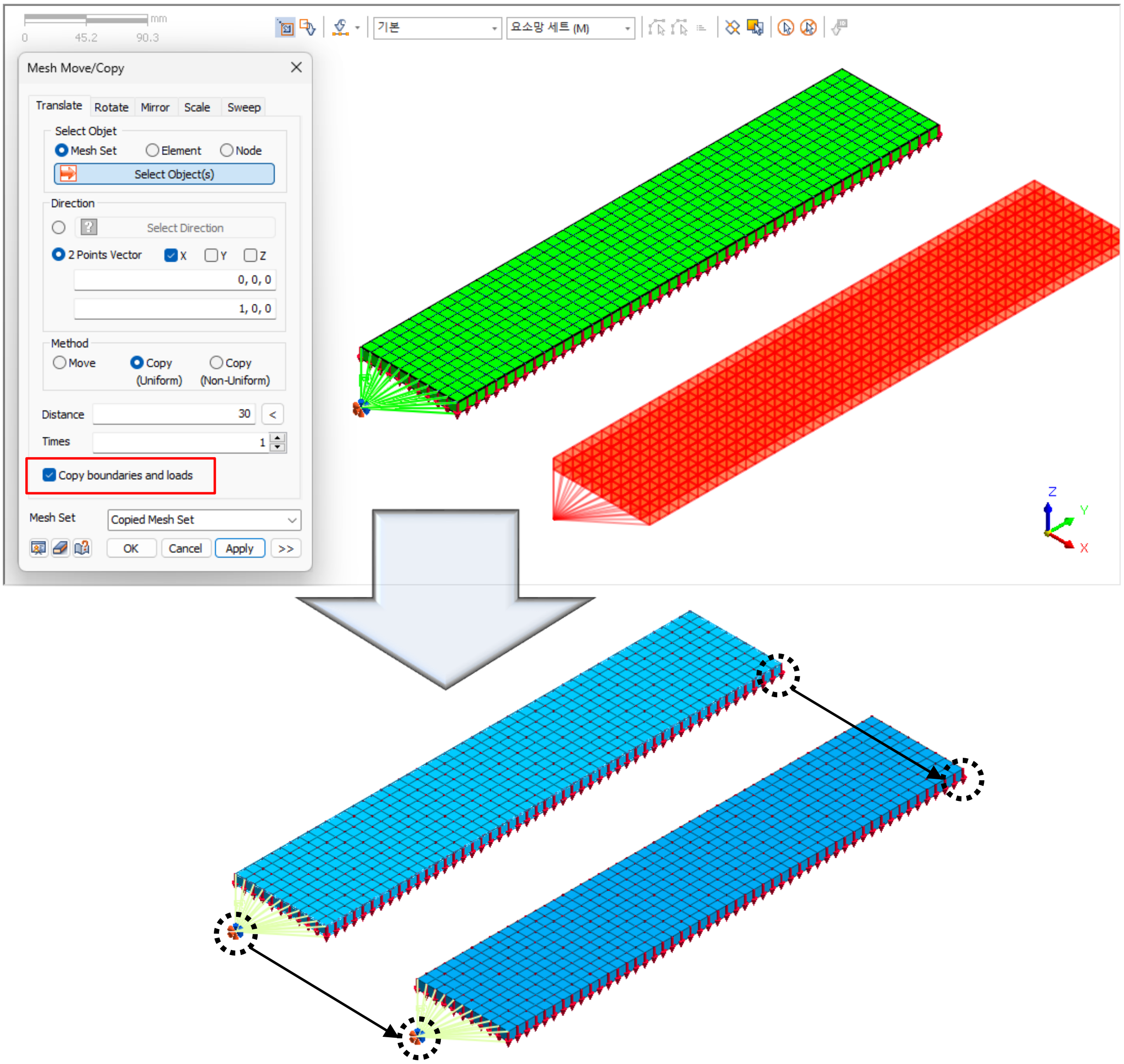
Last value	
First-order interpolation	
Second-order interpolation	
Specific value	
Cyclic reputation	

Addition of Load/Boundary Condition Copy Functionality during Element Copying

< Purpose and Usage >

In various situations such as repetitive arrays, symmetric shapes, and sensitivity checks of elements, elements are often copied for use. However, in the existing NFX, conditions were not inherited, causing inconvenience of needing to add them anew when the same conditions were required. To address this, improvements have been made to enable the inheritance of load and boundary condition data during the copying process of elements based on direction, such as linear, rotational, and symmetrical copying.



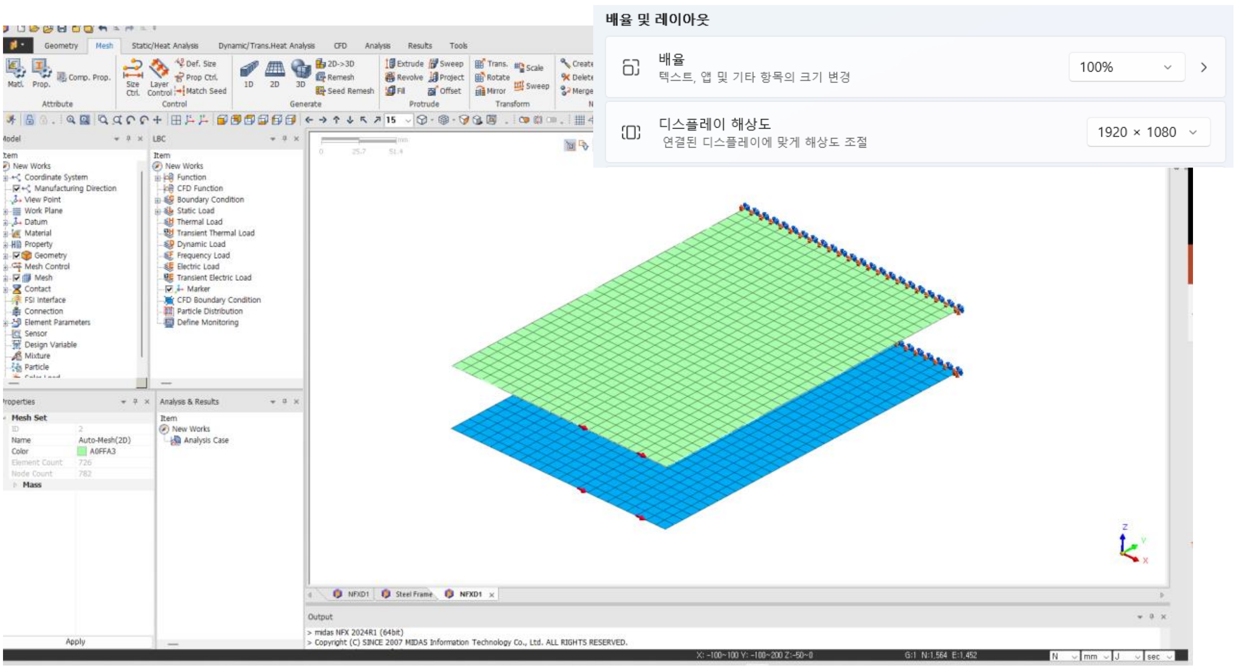


Improved support for high-resolution GUI in Windows.

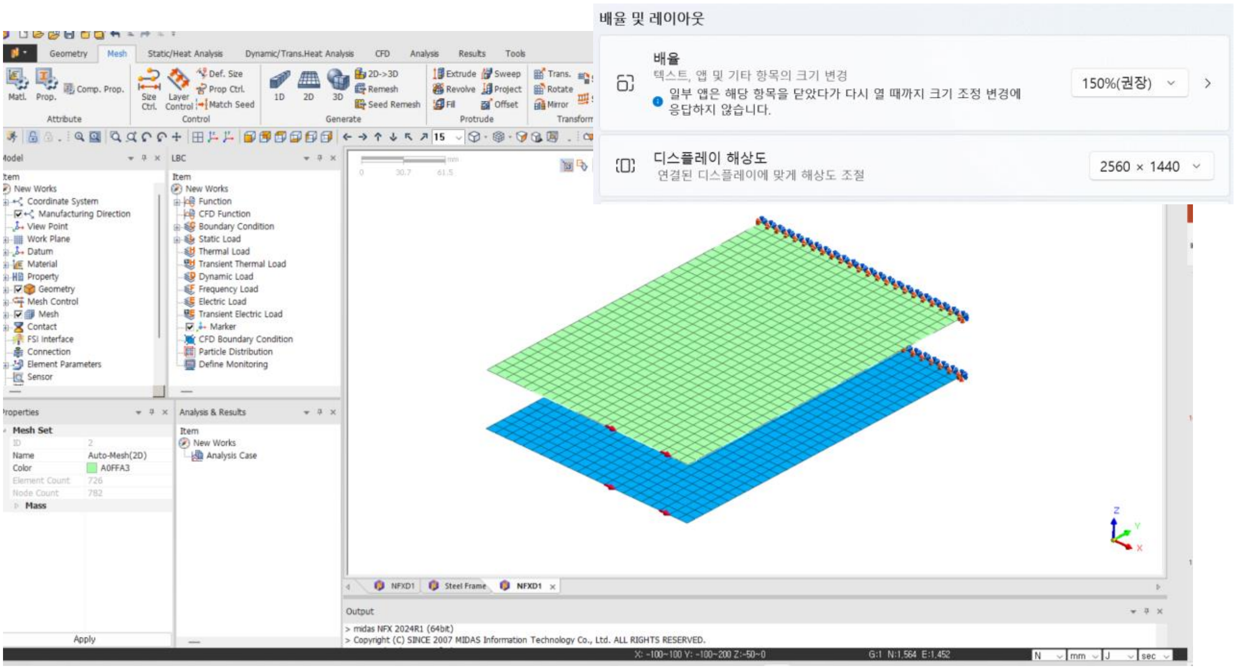
〈 Purpose and Usage 〉

The NFX GUI, previously optimized for FHD (1920x1080 pixels), has been expanded to support up to 4K (3840x2160 pixels), allowing the interface, function icons, text, etc., to be displayed according to the Windows user scale. This enhancement ensures compatibility with different display resolutions. It is automatically applied upon installation of NFX 2024 R1, requiring no additional installations.

▶ **NFX GUI : 1920x1080 pixel resolution / 100% magnification**



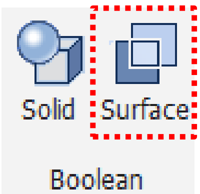
▶ **NFX GUI : 2560x1440 pixel resolution / 150% magnification**



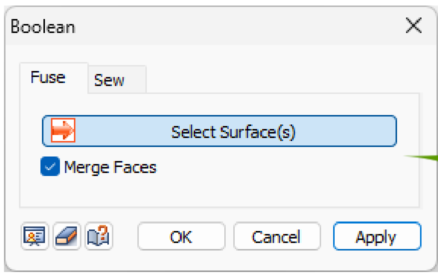
Added operations for faces (set operations: difference, intersection, embedding).

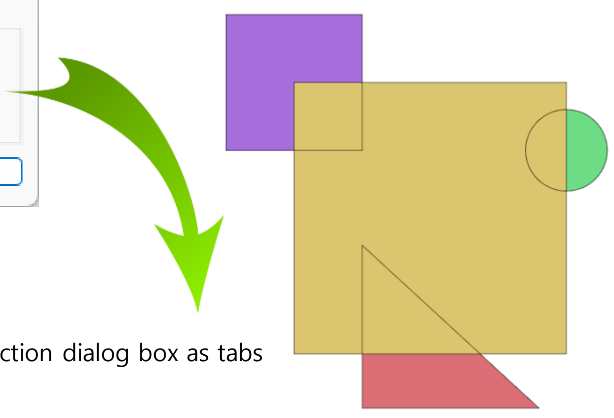
〈 Purpose and Usage 〉

In the existing NFX, only merging and splitting functions were provided for two-dimensional surface shapes, which necessitated additional steps such as creating a separate shape, dividing it, or deleting it, resulting in a process requiring 2 to 3 steps. To address this issue, functionality for complement, intersection, and embedding, similar to those available for three-dimensional solid shapes, has been developed to enable easier operations



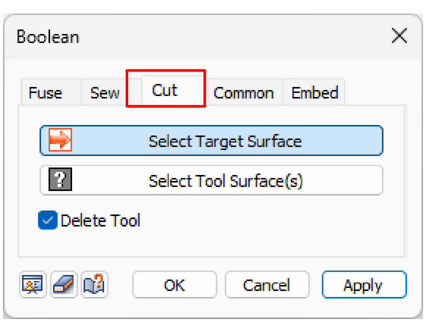
[Existing] Only Fuse and Sew functions are available for surface shapes.

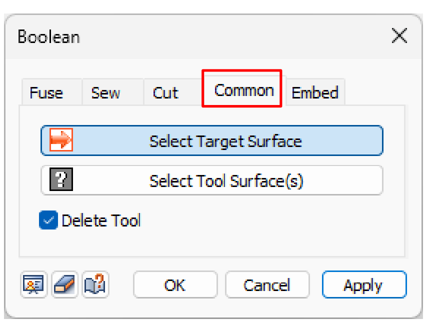


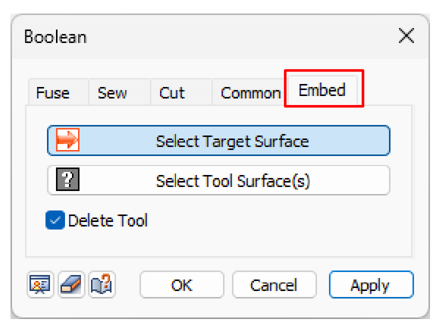


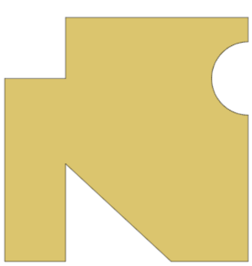
[Surface Intersection Operations]

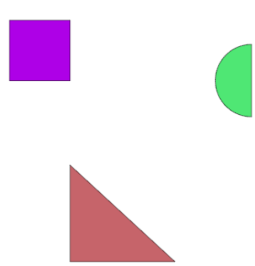
- Addition of Cut, Common, and Embed functions
- Functionality added to the existing surface intersection dialog box as tabs
- Default ON option for tool shape deletion
- Error message displayed if the operation is performed on shapes rather than surfaces of the same type

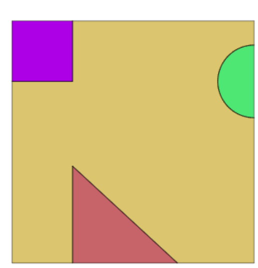










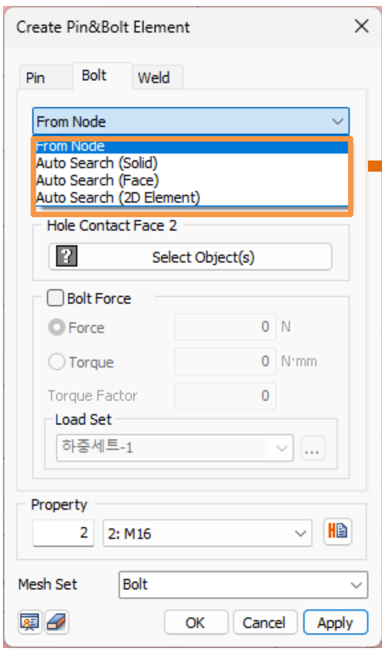


Added automatic bolt definition feature based on faces and 2D elements.

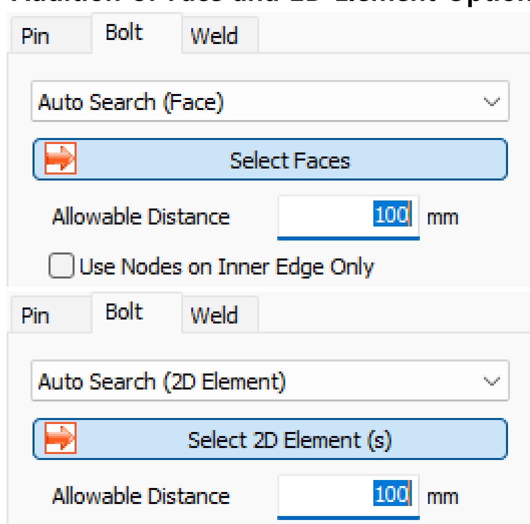
〈 Development Purpose and Usage Method 〉

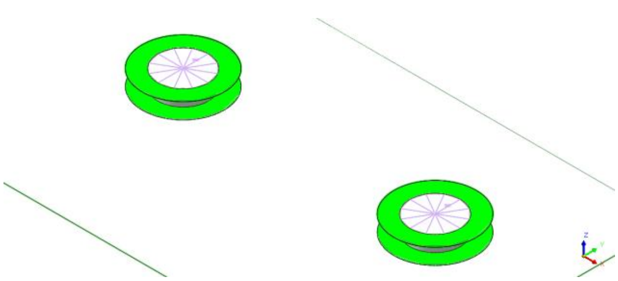
Generally used bolts calculate member forces and assess safety using rigid bodies and beams (1D Bar elements). In the existing NFX, to set bolts (rigid body + 1D bar), each must be defined individually, or elements must be copied manually for repetitive arrays. In large-scale models with dozens or hundreds of bolts, this results in significant work time and numerous user errors.

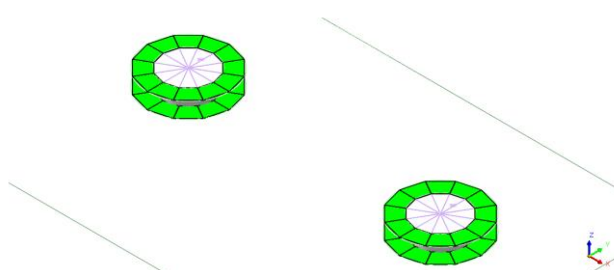
The automatic search function for surfaces and 2D elements performs tasks by automatically searching for and creating all areas located within the allowable distance specified by the user, without any limit on the number. Preload can be defined if necessary for batch creation.



Addition of Face and 2D Element Options







[Automatic Search (Face)]

- Select the surface shape corresponding to the washer
- Automatically search for the closest pair within the allowable distance
- If only nodes within the internal hole are needed, use the option to connect only the nodes of the hole's edge

[Automatic Search (2D Element)]

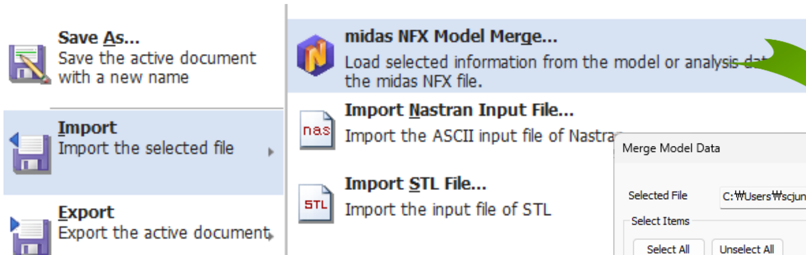
- Select the surface elements corresponding to the washer.
- Automatically search for the closest pair within the allowable distance.

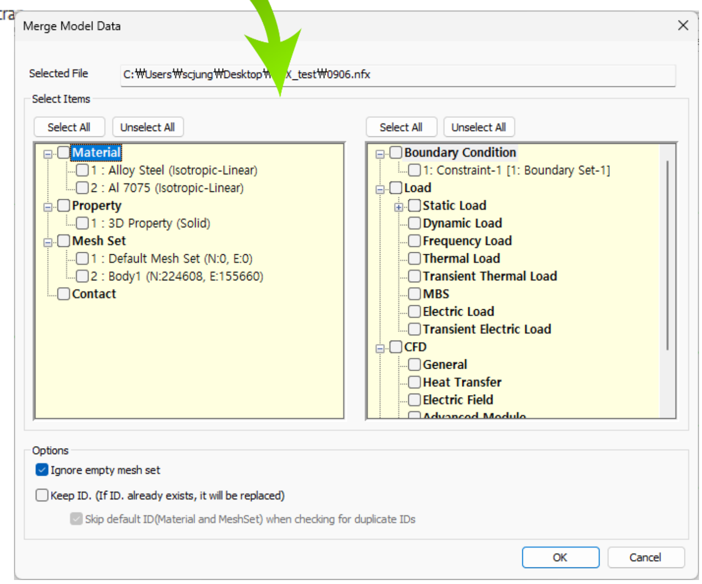
Add NFX file (*.nfx) import feature

〈 Development Purpose and Usage Method〉

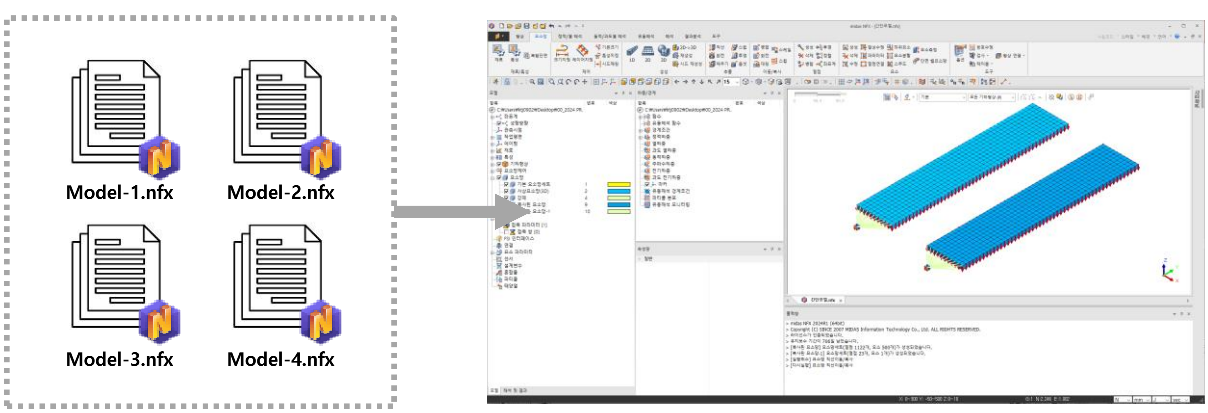
In the existing NFX, quick compatibility between setting files (*.nfx) was not possible, requiring step-by-step processes such as using Nastran import/export. The newly developed NFX file (*.nfx) import feature allows for rapid loading of all input data from the preprocessing stages, excluding the geometry, such as materials, properties, meshes, boundary conditions, and loads.

This feature enables various tasks, such as merging files that were worked on separately for one structure, importing and using different conditions for a single model, or segmenting large-scale models to focus on necessary areas.





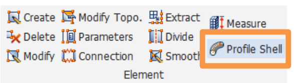
- Located in the submenu of Load Menu
- Separated into two groups: Model Group and LBC Group
  - Material/Property/Mesh Set/Contact
  - Load/Boundary/Flow
- Due to the relationship between 'Material < Property < Mesh Set', selecting a higher-level entity automatically loads the lower-level entities (Auto-load Material when selecting Property, auto-load Material and Property when selecting Mesh Set)
- Ability to select types of Load/Boundary conditions
- Assumes the geometry remains the same (Limits the degree of freedom when loading geometry with assigned indices)
- Adds an option to Keep ID.(If ID, already exists, it will be replaced)
  - Recommend keeping this option ON if using a specific number model
  - Default OFF to avoid user confusion

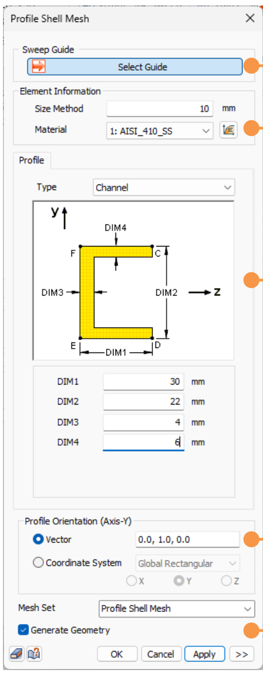


Add functionality to create 2D cross-sectional elements.

< Purpose of Development and How to Use >

Users can easily create 2D elements using custom cross-sections and guidelines. This functionality is applicable to piping systems, frame structures, and more. Even with different thickness inputs, it automatically generates property values. The geometric shape used for creation can be retained, and the element density can be adjusted at desired locations using the seed regeneration feature.





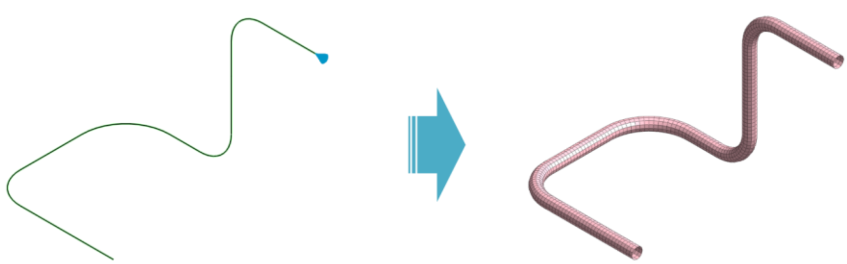
Select Wire or Edge as Selection Filter

Material selection instead of properties, automatically generate properties based on each cross-section thickness

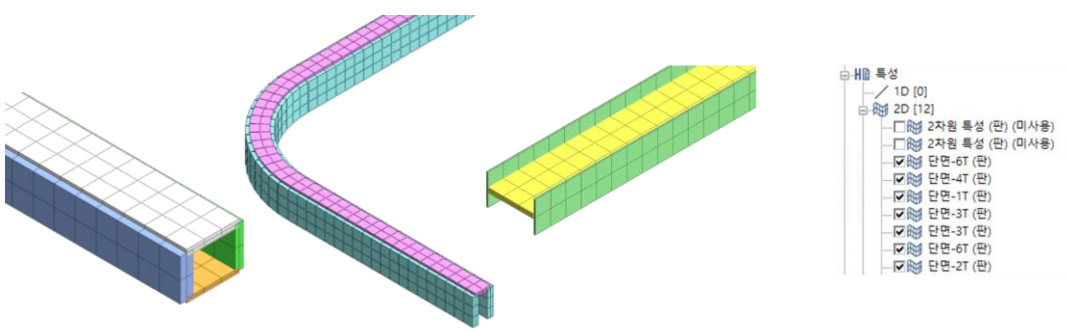
5 different cross-sections (parameters defining 1D property creation are the same)
Utilize Mid-line using user-defined values for each cross-section

Prioritize setting cross-section orientation direction with vector input, possibility to use coordinate system

Option to include geometric shape in final result



[Display Start Position when Selecting Guidelines for Cross-Section Orientation]

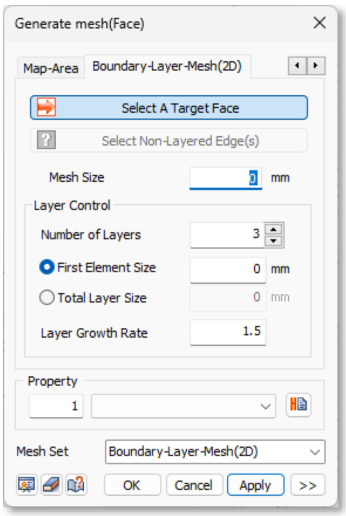


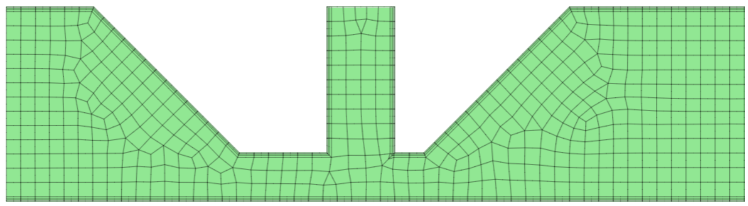
[Automatically Generate Properties Based on Thickness Input]

2D, 3D Layered Mesh Generation Techniques Development

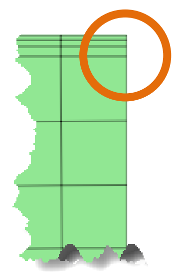
〈 Purpose of Development and Usage 〉

The wall function option is commonly used to simulate the velocity changes in the viscous sublayer region very close to the wall. Even when using wall functions, an element mesh that can accurately represent the characteristics of the boundary layer is required to better simulate the velocity changes according to the distance from the wall. By using the boundary layer mesh function, you can now easily generate dense, multi-layered element meshes in regions with large gradients of physical variables.

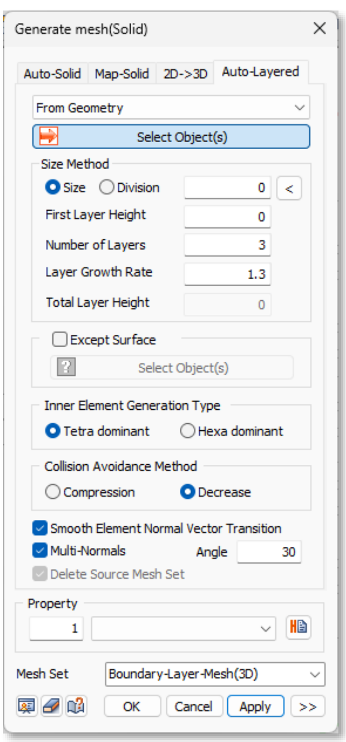


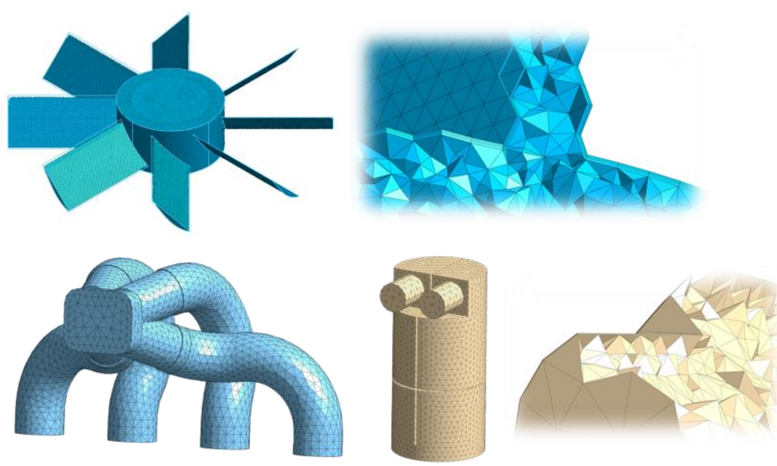


- **Two options for specifying layers are provided (first element size, total layer thickness).**
- **Feature to exclude boundary layers by selecting lines (maintain boundary layer thickness at edge lines).**
- **Includes setting the relationship between geometry and mesh.**

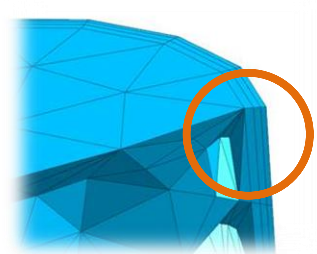


[2D Boundary Layer Mesh Generation]





- **Two options for specifying layers are provided (first element size, total layer thickness).**
- **Feature to exclude boundary layers by selecting surfaces (maintain boundary layer thickness at edge lines).**
- **Includes setting the relationship between geometry and mesh.**



[3D Boundary Layer Mesh Generation]

Other improvements

〈 Development of Weld Connector (C-Weld) and Cohesive Element 〉

We have expanded our existing Area Contact Model (ACM) welding capabilities to include the development and functionalization of Weld Connect Elements (C-Weld). These elements are automatically implemented in RBE3 + Beam connection format based on user-defined search distance, weld diameter, and number of layers either through direct input at welded locations or by batch applying coordinate data from files. Additionally, elements and properties have been developed and integrated to automatically apply to Cohesive elements imported in Nastran (bdf) format.

〈 Improvement of Errors in Bolt Load Preload Reinterpretation 〉

To better reflect the effects of fastening when applying bolt loads, we now perform a separate Preload sub-case before the first sub-case in all applicable analysis cases. This allows us to review the remaining loads after considering the preload. Previously, when reinterpreting with changed conditions, the existing Preload sub-case remained, causing inconvenience as users had to manually delete it for reinterpretation. This has been improved to streamline the process.

〈 Improvement of Issue Ignoring Initial Temperature Conditions in Excessive Joule Heating Analysis 〉

We identified an issue where, when applying electrical loads in excessive Joule heating analysis, the initial temperature was being ignored and instead, the material temperature was applied. This issue has been addressed, and now specific scalar value input and initial temperature settings via analysis sets are both functioning properly to set initial temperatures point-wise.

〈 Improvement of Unit Conversion Error in Table Representation within Result Extraction Function 〉

We have identified an issue where abnormal conversions occur when applying the result extraction function in the results analysis, depending on user unit system changes (e.g., N, kgf, tonf). This issue has been addressed, and now results are correctly represented in all user unit systems.

〈 Fixing Operational Error in Force Type Sensor Category 〉

An error was identified in the category and export of force type sensors, which has been rectified within the GUI. Additionally, a malfunction was observed where if element results are defined on nodes within sensor results, it should yield nodal average sensors, but this was not functioning properly. This has been addressed to ensure correct output within the solver.

〈 Adding Turbulence Variables to Fluid Analysis Norm Graph 〉

The Norm Graph displays the residuals of the independent variables being solved in the governing equations. In fluid analysis, when turbulence models are applied, turbulence variables are additionally solved along with velocity and pressure. While turbulence variables were already being solved, they were not displayed on the graph to keep the interface less cluttered. However, for improved intuitiveness, the Norm Graph has been modified to consistently display the independent variables being solved in the current analysis.