Multidisciplinary Integrated Analysis Solution for Optimal Design

midas NFX 2025 Introduction

midas NFX 2025

Introduction for update function

Major update list (Release Version : 2025.02.07)

- CAD Interface update
- Graphic User Interface improvement (Ribbon menu, Tree menu)
- Added Abaqus file (*.inp) import feature (for linear materials and structural elements)
- Updated 3D Auto Mesh Generation Library
- Added 3D Layered Mesh generation feature
- Enhanced 3D Auto Mesh internal surface functionality
- Added automatic 2D fillet welding generation feature
- Added automatic Fluid-to-Structure model generation feature
- Improved rendering speed in preprocessing environment
- Enhanced usability for frequency/random response analysis
- Added memory extension for structural analysis solver
- Other improvements

midas NFX provides fully integrated and linked analysis for structure, thermal, fluid, and optimization within a single working environment using a single model. It features a Windows-based GUI localized in Korean and offers a user-friendly environment with structured training and technical support through the MIDAS customer value system.

CAD Interface Update

The CAD Interface has been updated according to the CAD version update. Support for the latest versions may be delayed depending on the update environment of partner companies. If the latest version is not supported, please convert the file to Parasolid format for use. We will do our best to quickly incorporate the latest CAD versions.

Type	File format	Applied version
Parasolid	x_t, xmt_txt, x_b, xmt_bin	9.0 ~ 37.0.118
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 11.0
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 ~ NX2406
Inventor Part	ipt	V6 ~ V2025
Inventor Assembly	iam	V11 ~ V2025
SolidEdge	par, asm, psm	V18 ~ SE2024

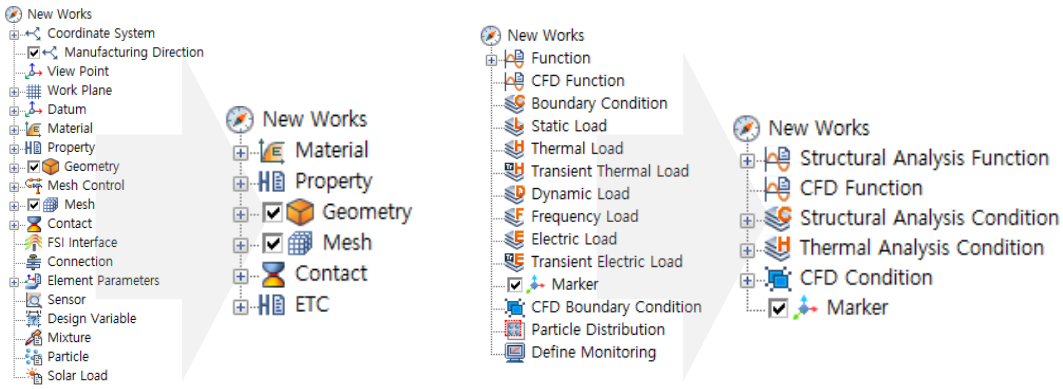
Graphic User Interface improvement

<Development Purpose and Usage>

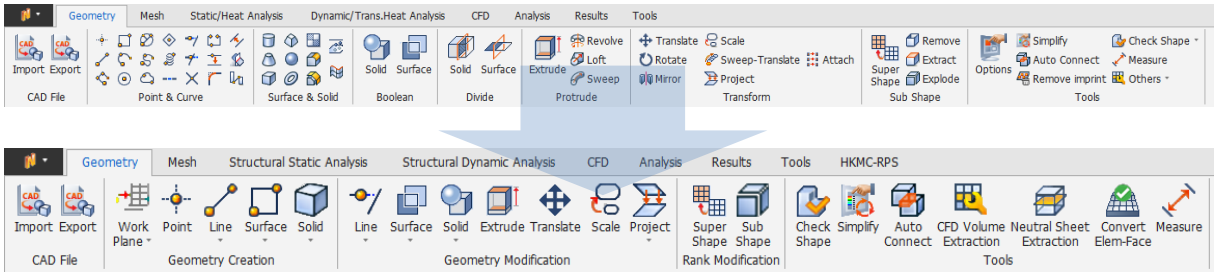
The GUI (Graphic User Interface) provided by NFX has **been completely redesigned to make the entire process of geometry, meshing, and analysis more intuitive and convenient**. Instead of displaying all available icons at once, they have been grouped and categorized based on user workflows, allowing for easier identification and application according to the required analysis.

The interface is structured **to follow the user's workflow step by step, progressing from left to right**. This ensures that only the necessary functions are displayed at each stage, enabling users to complete each step efficiently. Additionally, the interface supports multiple languages, including Korean, English, and Chinese, allowing users to switch as needed.

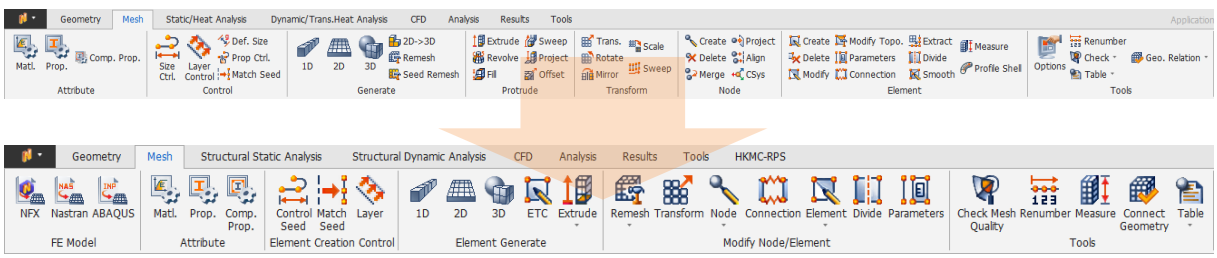
- Tree menu simplification: **Quickly access only the necessary items!**



- Shape tab icon reorganization: **Only the necessary items based on the workflow!**
  CAD / Creation / Modification / Dimension Change / Tools



- Element tab icon reorganization: **Only the necessary items based on the workflow!**
  FE Model / Material & Properties / Generation Control / Creation / Modification / Tools



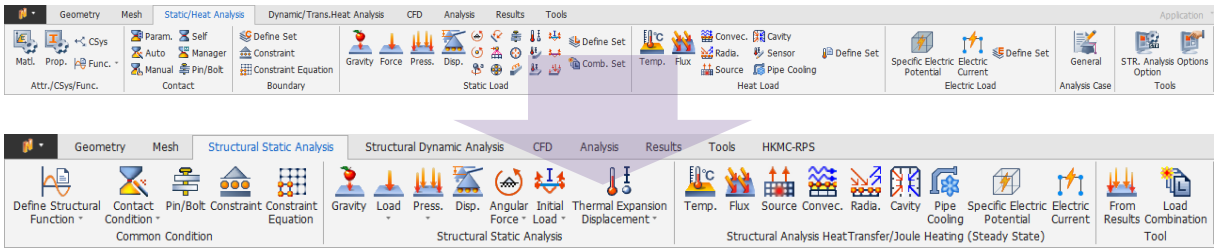
Graphic User Interface improvement

<Development Purpose and Usage>

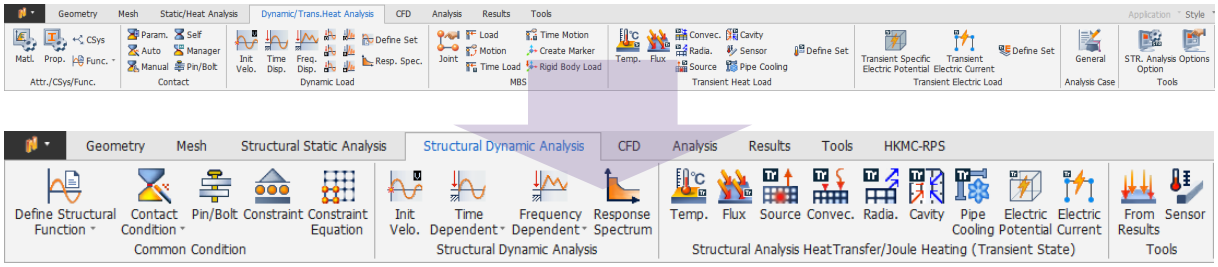
The GUI (Graphic User Interface) provided by NFX has **been completely redesigned to make the entire process of geometry, meshing, and analysis more intuitive and convenient**. Instead of displaying all available icons at once, they have been grouped and categorized based on user workflows, allowing for easier identification and application according to the required analysis.

The interface is structured **to follow the user's workflow step by step, progressing from left to right**. This ensures that only the necessary functions are displayed at each stage, enabling users to complete each step efficiently. Additionally, the interface supports multiple languages, including Korean, English, and Chinese, allowing users to switch as needed.

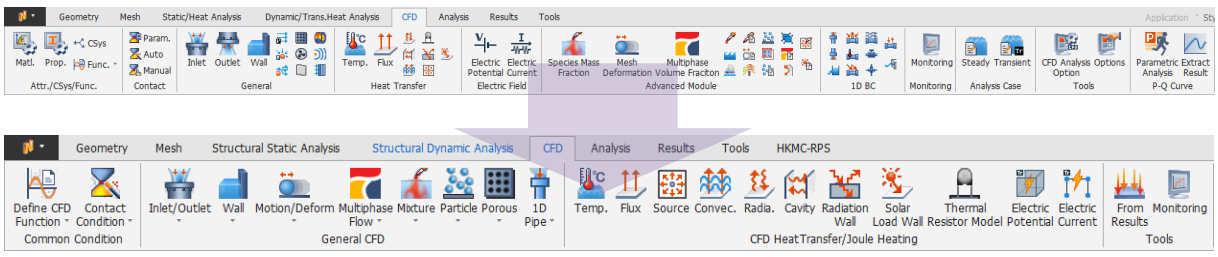
• STR / Static Load : **More intuitive classification based on types of static loads!**



• STR / Dynamic Load : **More intuitive classification based on vibration and transient analysis types!**



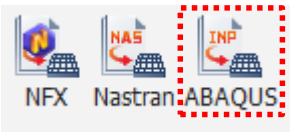
• CFD Analysis : **More intuitive classification based on fluid analysis modules!**

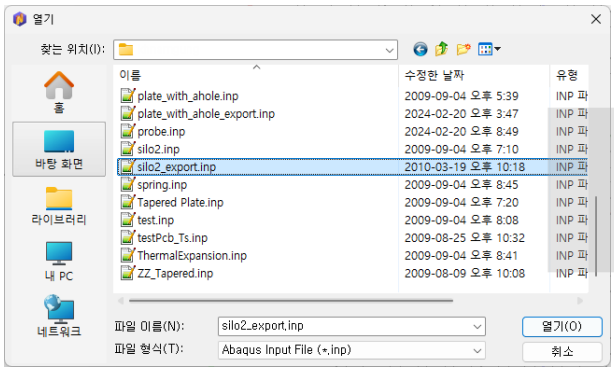


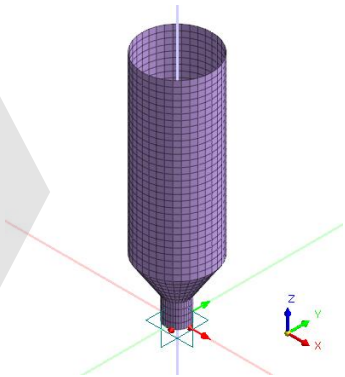
Added Abaqus file (*.inp) import feature

<Development Purpose and Usage>

Support for Abaqus Input File (*.inp) Import has been expanded beyond the existing NFX and Nastran Standard Format. This extension allows users to import and work with Abaqus files. The development includes key materials, properties, 1D/2D/3D elements, connections, and special elements. Detailed importable items can be checked using keywords below.







keyword	
*ASSEMBLY *ENDASSEMBLY	*FASTENER
*BEAM GENERAL SECTION	*FASTENER PROPERTY
*BEAM SECTION	*INCLUDE
*BOUNDARY	*INSTANCE *ENDINSTANCE
*CLOAD	*KINEMATIC
*COHESIVE SECTION	*MATERIAL
*CONDUCTIVITY	*NODE
*CONNECTOR SECTION	*NSET
*CONNECTOR BEHAVIOR	*ORIENTATION
*CONNECTOR ELASTICITY	*PART *ENDPART
*COUPLING	*PLASTIC
*DENSITY	*RIGID BODY
*DISTRIBUTING	*SHELL GENERAL SECTION
*DLOAD	*SHELL SECTION
*DSLOAD	*SYSTEM
*ELASTIC	*SPECIFIC HEAT
*ELEMENT	*SURFACE
*ELSET	*SYSTEM
*EXPANSION	*TRANSFORM

*FASTENER : Connetor
*ELASTIC : ISOTROPIC, ENGINEERING, LAMINA
*CONDUCTIVITY : ISOTROPIC, ORTHOTROPIC

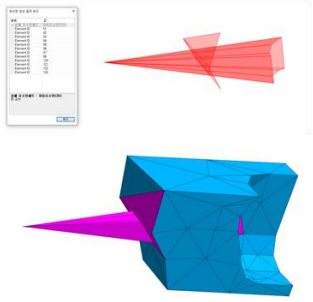
3D Auto Mesh Generation Library Update

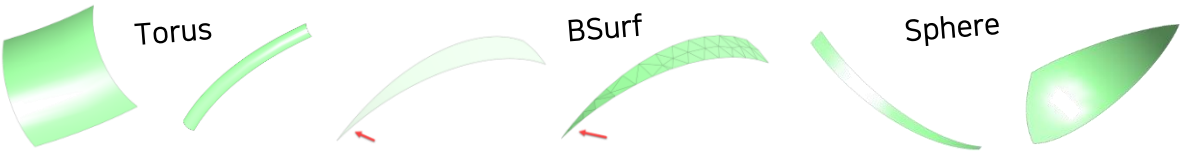
<Development Purpose and Usage>

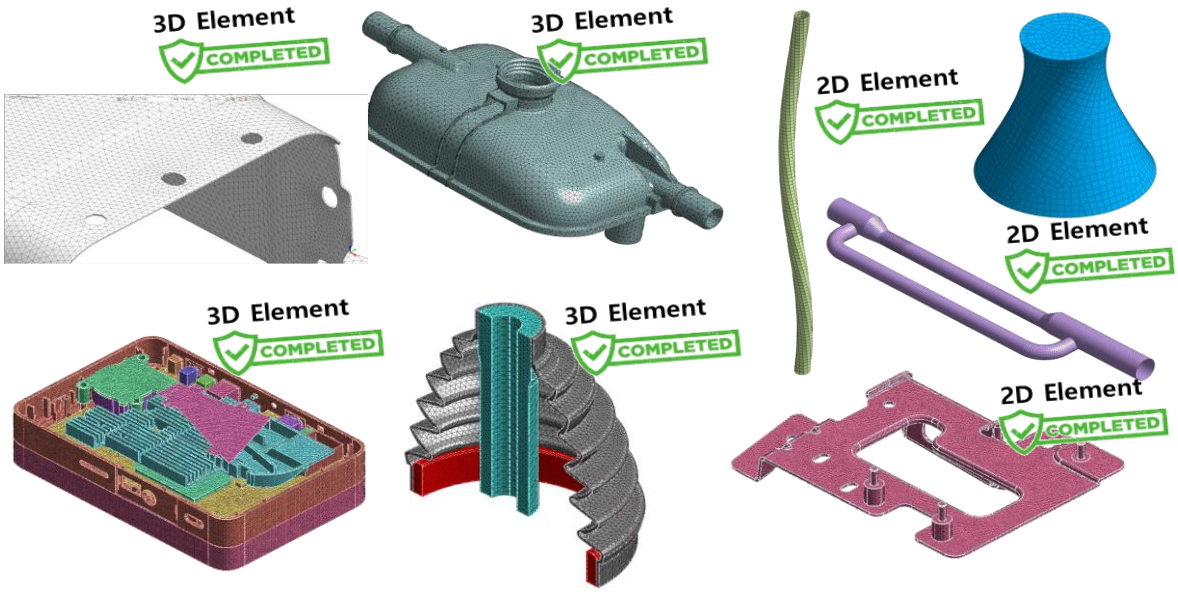
To improve the success rate of mesh generation on curved surfaces in 3D Auto-Mesh creation based on the Parasolid Kernel, enhancements have been made to address singularities and correct errors in commonly occurring geometries such as Torus, Bsurf, and Sphere. Additionally, failed mesh element sets during generation are now separated and marked for clear identification.

Ongoing refinements and corrections are being applied to the Parasolid Kernel and Mesh Generation Library to further enhance the stability and reliability of automatic mesh generation.

- Added a reinforcement routine for UVBox parameters on **Torus surfaces**.
- Error handling for **BSurf surfaces** with sharp edges.
- Special error handling for **Sphere surfaces** containing poles.
- Applied modifications to library return error codes → **Proper error messages are now displayed.**
- Added a function to check problematic elements **when AutoMesh 2D to 3D conversion fails.**



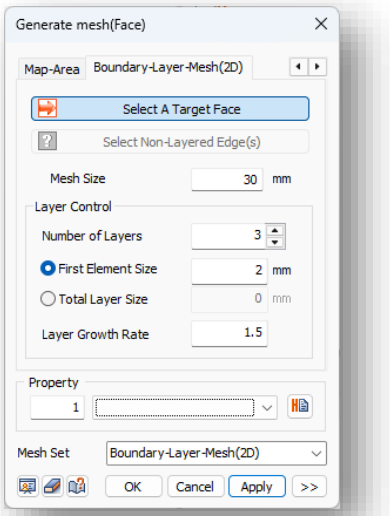


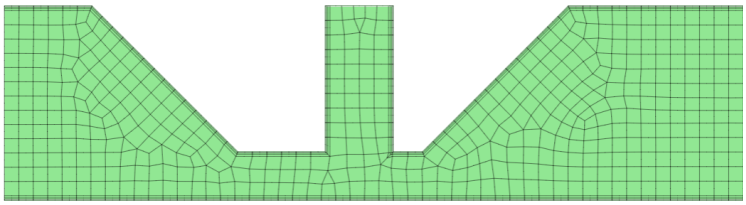


Added Layered Mesh generation feature

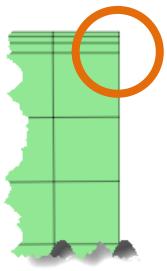
<Development Purpose and Usage>

The wall function option is commonly used to simulate velocity changes in the viscous sublayer region near the wall. Even when using wall functions, an element mesh that accurately represents the boundary layer characteristics is essential for precisely capturing velocity variations with respect to the wall distance. Now, with the boundary layer meshing function, you can easily generate multiple layers of fine mesh in regions with high physical variable gradients.

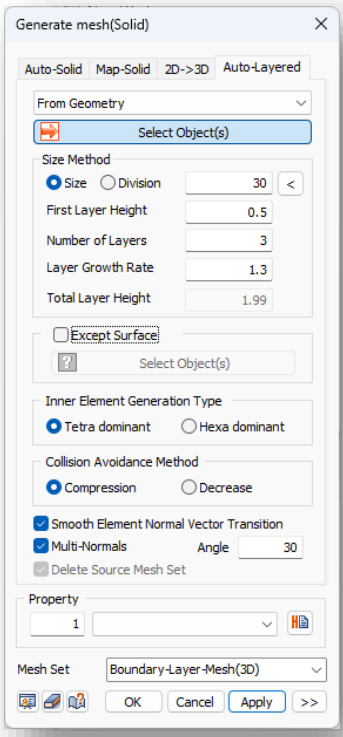


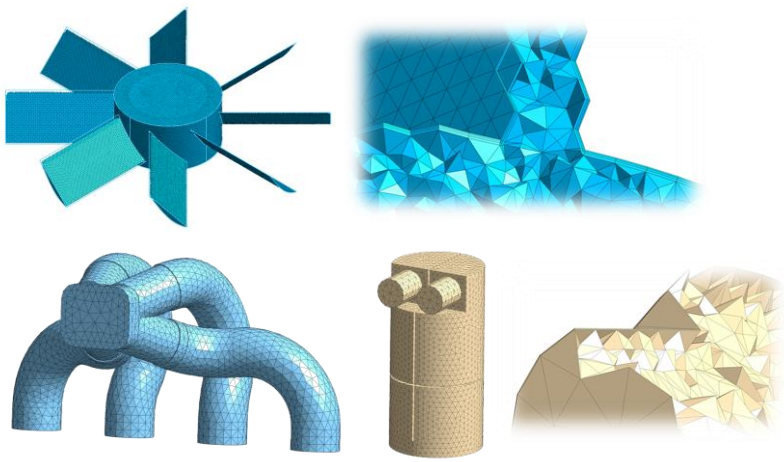


- **Two layer specification options** are provided (first element size, total layer size).
- Boundary layer **exclusion edge selection** function is available (maintains boundary layer thickness at the edge)
- Includes geometry-to-mesh relationship settings.

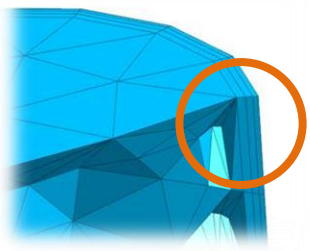


[2D layered mesh generation]





- **Two layer specification options** are provided (first element size, total layer size).
- Boundary layer **exclusion face selection** function is available (maintains boundary layer thickness at the edge)
- Includes geometry-to-mesh relationship settings.



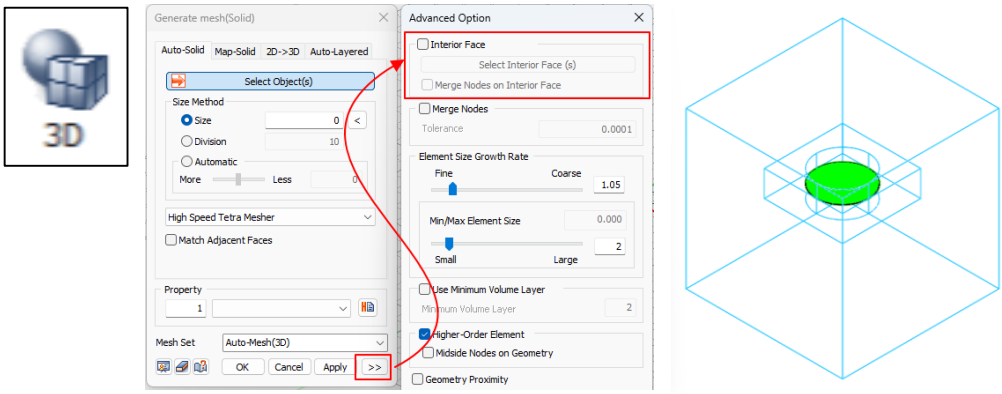
[3D layered mesh generation]

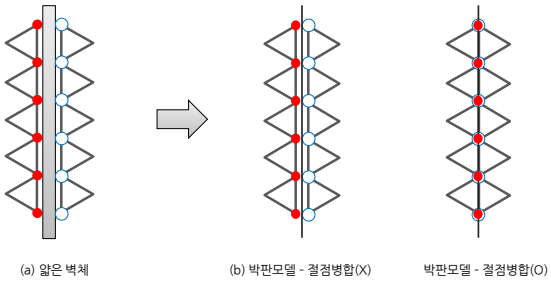
Enhanced 3D Auto Mesh internal surface functionality

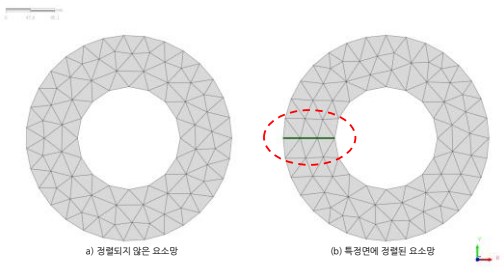
<Development Purpose and Usage>

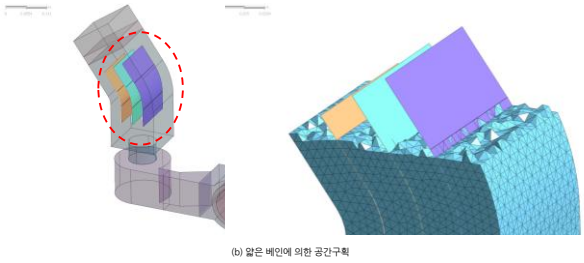
The internal face function, one of the 3D Auto Mesh features, is widely used in fluid analysis. Errors were identified in cases where the geometry was split, and improvements were made by extending the functionality to include single cutting, orthogonal division, and multi-face splitting.

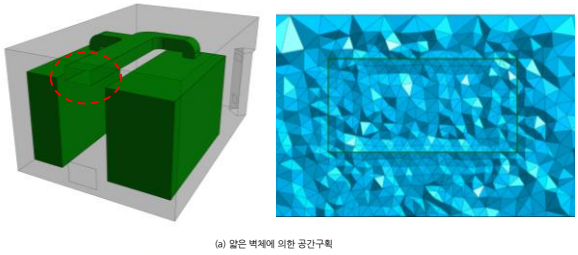
It can be used for thin plates that may degrade element quality, guide vanes affecting fluid flow, and internal fan conditions requiring pressure jumps. Previously, it was limited to single face (Face) types, but now it has been expanded to include shell (Shell) type assemblies.

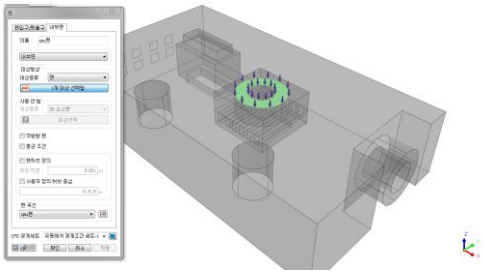


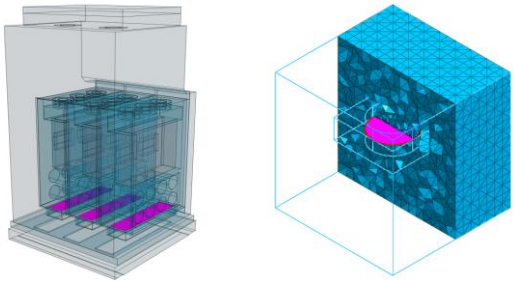










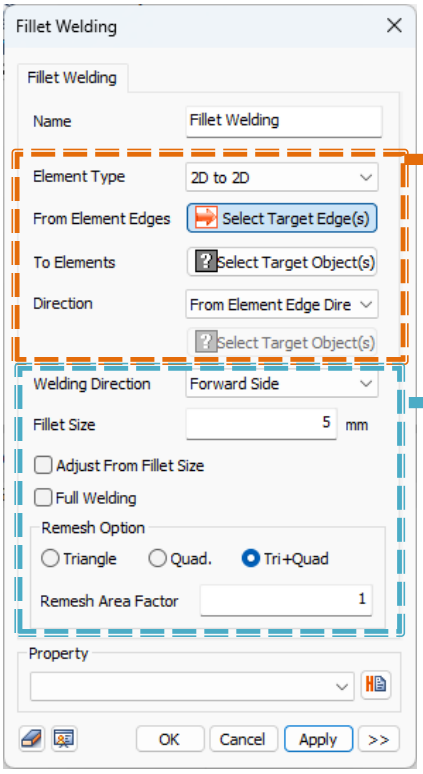


Added automatic 2D fillet welding generation feature

<Development Purpose and Usage>

Various welding functions are used across many industries. Following the implementation of spot welds, a 2D fillet weld feature has been added for application to 2D shell elements. Based on the user-specified 2D element edge (From), the system automatically performs remeshing on the target element (To) while considering the fillet size and type.

Users can preview the placement based on the From element or custom-defined values, allowing them to check the automatic alignment and connection results.

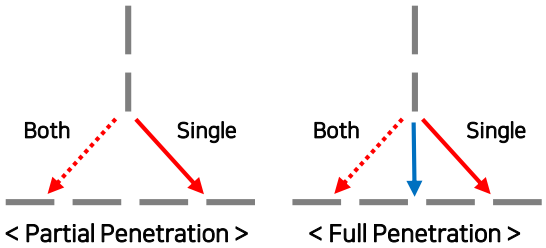


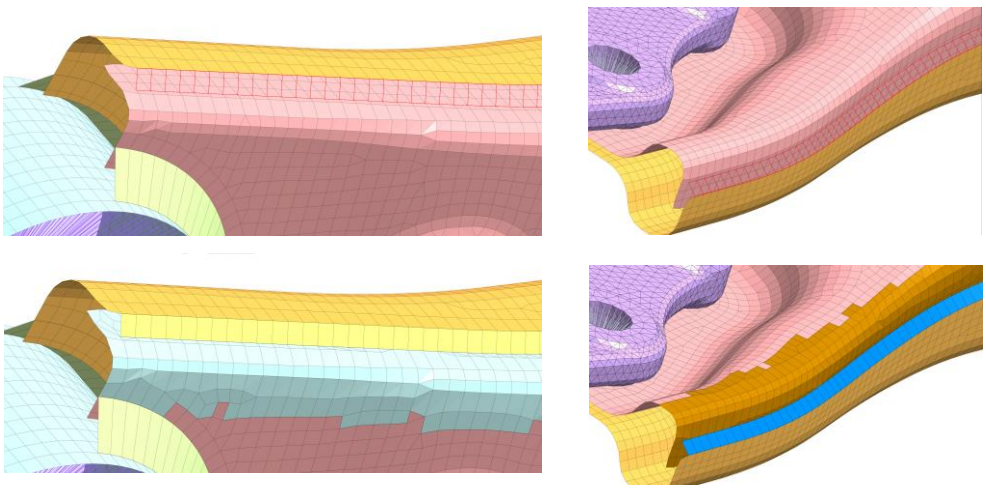
[Fillet Weld Connection Element Selection]

Automatically generate fillet weld elements from the starting edge (From) to the target element (To).

[Fillet Weld Size Specification]

- Specify the size and direction of the fillet weld along the short edge.
- Define the welding direction (Partial (default), Full).
- Additional element quality options for remeshing targets.





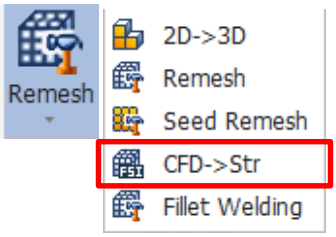
Added automatic Fluid-to-Structure model

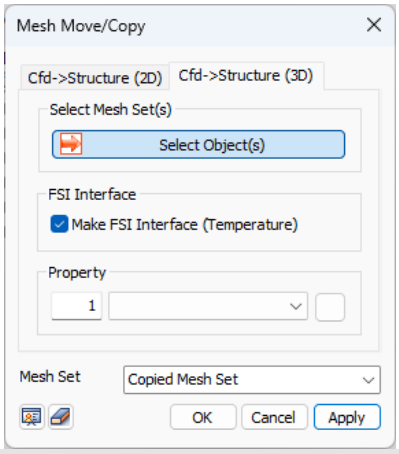
<Development Purpose and Usage>

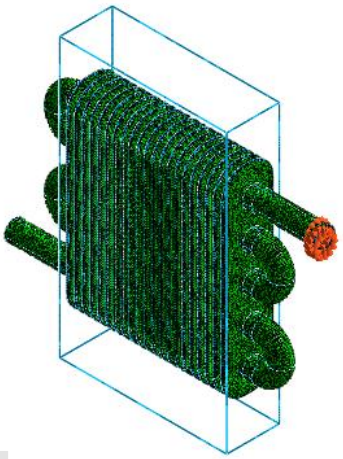
In real-world scenarios, most cooling phenomena occur in fluids, except for electrical effects like Peltier. One of the primary purposes of CFD analysis in industries is to evaluate cooling efficiency and temperature distribution in key components.

Previously, FSI INTERFACE and coupled analysis cases were provided, but defining mesh and conditions for overlapping regions was somewhat inconvenient. To streamline the process and improve usability, a new feature has been added that automatically defines a structural element model sharing the same nodes and a volume-based FSI INTERFACE using user-defined structural material and property values.

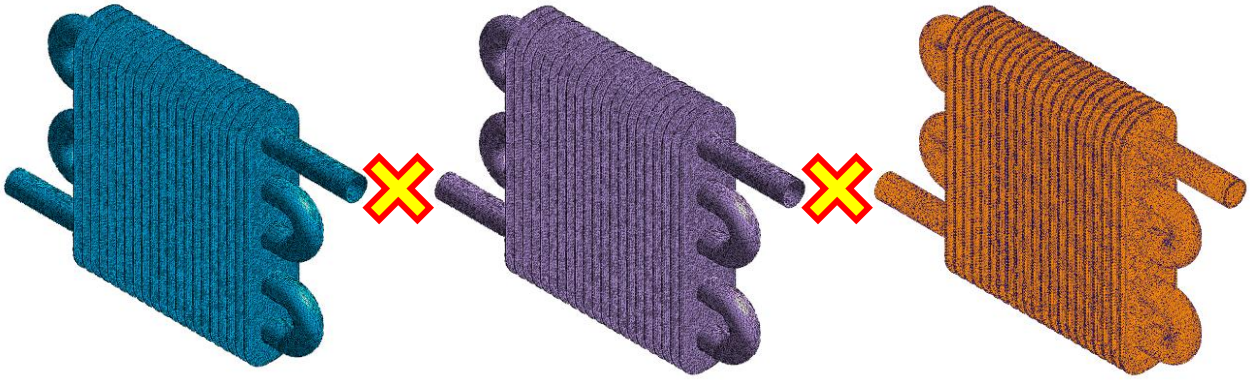
Even when performing the analysis manually, users can easily map node-based data using the general result conversion function.







- Auto generate Structural model
- Auto generate FSI interface(volume)



[CFD Model] [Structural Model] [FSI INTERFACE - Volume]

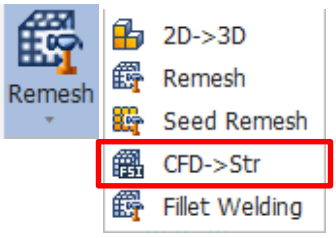
Added automatic Fluid-to-Structure model

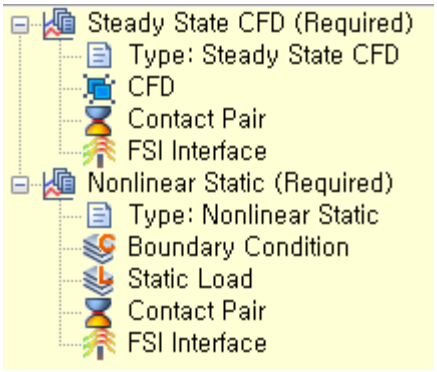
<Development Purpose and Usage>

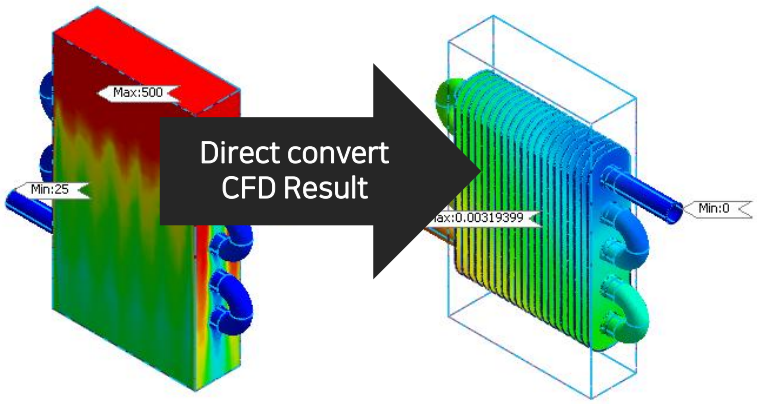
In real-world scenarios, most cooling phenomena occur in fluids, except for electrical effects like Peltier. One of the primary purposes of CFD analysis in industries is to evaluate cooling efficiency and temperature distribution in key components.

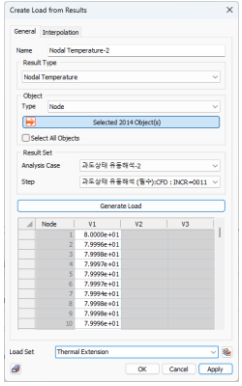
Previously, FSI INTERFACE and coupled analysis cases were provided, but defining mesh and conditions for overlapping regions was somewhat inconvenient. To streamline the process and improve usability, a new feature has been added that automatically defines a structural element model sharing the same nodes and a volume-based FSI INTERFACE using user-defined structural material and property values.

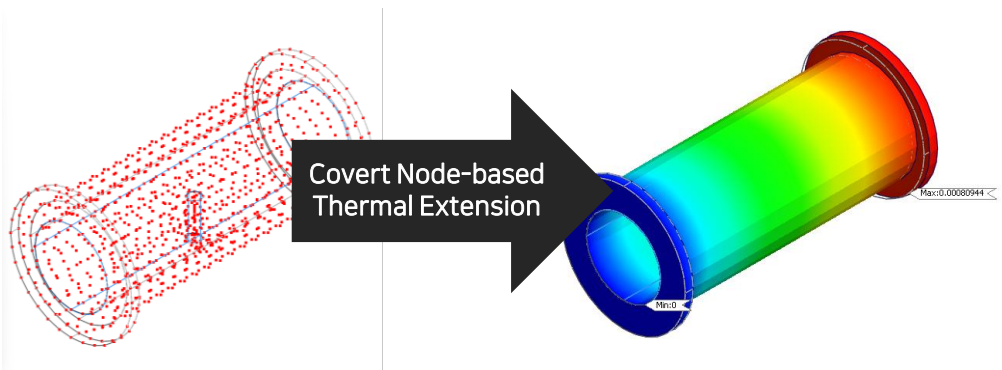
Even when performing the analysis manually, users can easily map node-based data using the general result conversion function.











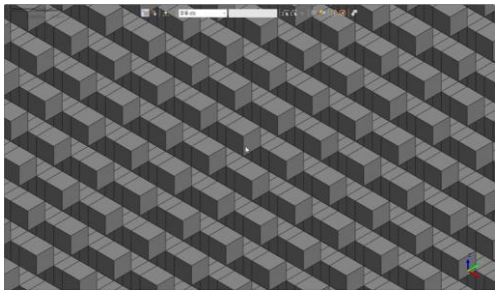
Improved rendering speed in pre-processing

<Development Purpose and Usage>

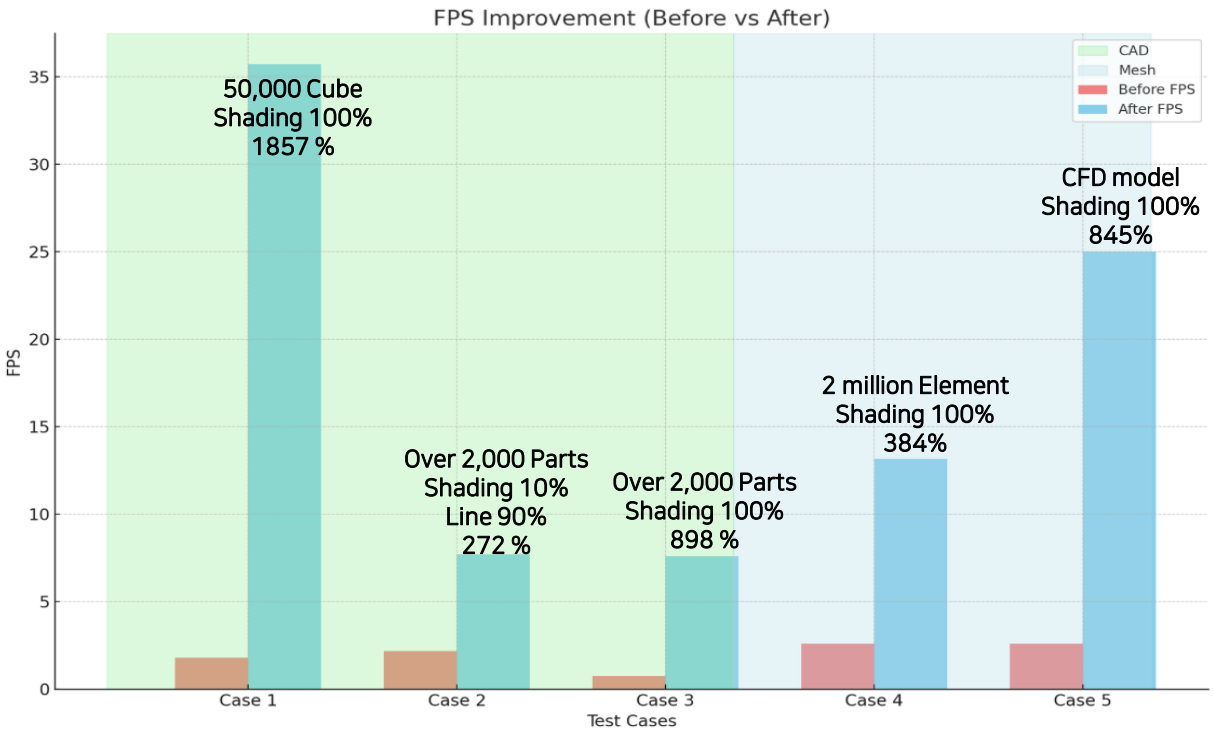
As large-scale models have become more common, increased use of complex geometries and mesh elements has led to a decline in rendering speed when manipulating models. To address this, the body information processing method in the existing rendering system has been restructured to allow parallel processing.

Validation was conducted on various cases, including repetitive geometric shapes, large mesh structures, automotive assemblies with over 2,000 parts, element models with over 2 million elements, and large-scale CFD models. The results showed an average rendering speed improvement of over 400%.

- Case 1 : 50,000 Cubes (600,000 Elements)
  : 548 ms (1.8 fps) → **28 ms (35.7 fps)** ··· 1,857% Improve



- Case 5 : CFD model
  : 378 ms (2.6 fps) → 40 **ms (25.0 fps)** ··· 845% Improve



* Case 2/3/4 Due to security concerns, real-world models cannot be disclosed.
(Over 2,000 Parts, Over 2 million elements model)

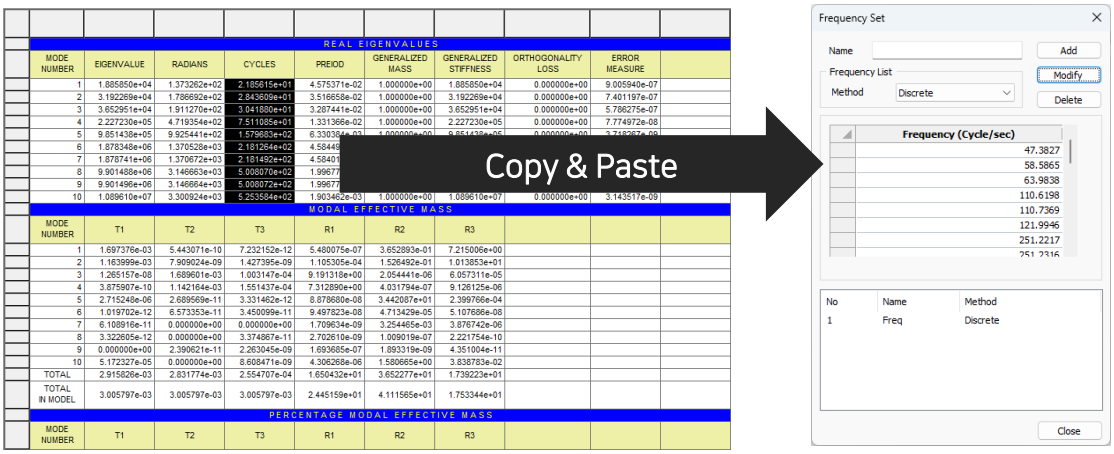
Enhanced usability for frequency/random response analysis

<Development Purpose and Usage>

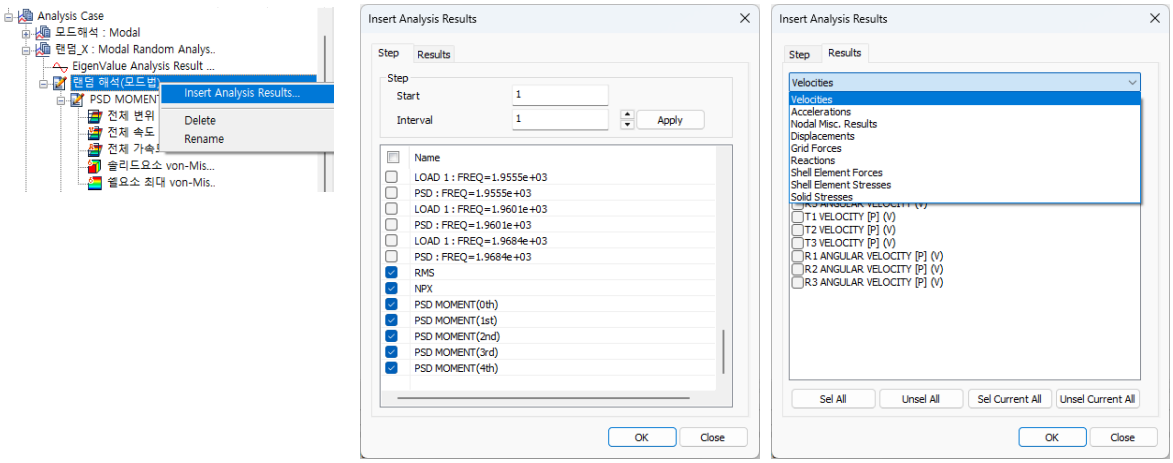
Recently, as the complexity and scale of frequency/random response analysis have increased in industries such as ESS, defense, and automotive, there has been a growing need for cluster-based methods and definitions for irregular forms. Previously, in NFX, users had to manually input irregular frequency sets, but this has now been **improved to a table format that allows data to be copied and pasted from Excel.** This enables quicker verification and application of necessary data alongside mode table results.

Additionally, in random response analysis, structural safety and vibration fatigue evaluation are typically conducted using moment, RMS, and NPX values, reducing the need for single-response outputs. As a result, single-response data is now **hidden by default, and users can selectively output results for desired frequency sets through subcase analysis settings.**

▶ Changed the definition method for discontinuous frequency sets (Copy & Paste enabled).



▶ Modified the result addition method in random response analysis (Default: Moment, RMS, NPX output).
→ Users can right-click on the subcase > Add Analysis Results

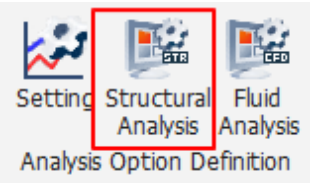


Added memory extension for structural analysis solver

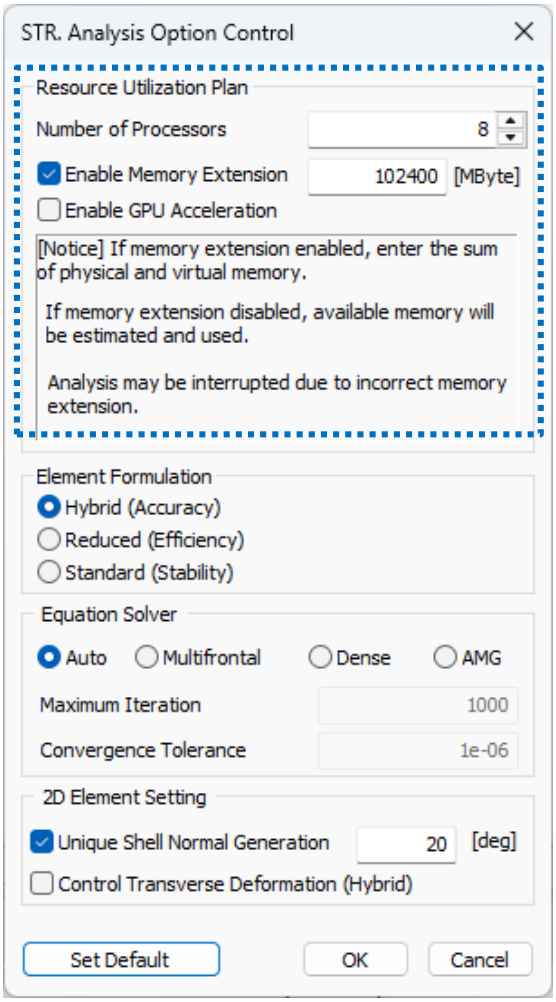
<Development Purpose and Usage>

Added Memory Extension option for structural analysis solver to allow users to manually adjust memory allocation. If physical memory is insufficient, users can enable virtual memory using the hard disk to prevent slowdowns. Recommendation: Ensure sufficient disk space before enabling virtual memory, as excessive memory usage may overload the model.

If the hard disk has enough free space (e.g., 100 GB), the software will automatically allocate the required memory for analysis. Helps compensate for memory shortages and improve analysis speed.



Test model (2D Shell Element + RBE2, Weld)
Elements : 1,891,593
Nodes : 1,941,628
DOFs : 11,340,769
Equations : 11,297,653



Case	Available Memory (GB)	Memory Extension (GB)	Usage Memory (GB)	Working Time (sec)	CPU Clock (GHz)
A	32	0	20.33	1585.28	2.5
B	32	32	26.99	1819.17	2.5
C	32	100	49.02	786.00	2.5
D	128	0	49.08	499.77	3.6

A. 8 CPU Threads (No Memory Extension) : 1585.28 sec

```
[SYSTEM INFO]
NUMBER OF THREADS    : 8
MAXIMUM MEMORY USAGE : 11217 MB(MAIN), 20762 MB(MFSEXE.EXE)
AVAILABLE MEMORY     : 20826 MB
TOTAL CPU TIME       : 2387.56 sec(MAIN), 110.234 sec(MFSEXE.EXE)
WALL CLOCK TIME      : 1585.28 sec
TOTAL WARNINGS : 17
```

B. 8 CPU Threads + Memory Extension 32GB : 1819.17 sec (114.7%)

```
[SYSTEM INFO]
NUMBER OF THREADS    : 8
MAXIMUM MEMORY USAGE : 11228 MB(MAIN), 27558 MB(MFSEXE.EXE)
AVAILABLE MEMORY     : 27633 MB
TOTAL CPU TIME       : 2246.5 sec(MAIN), 89.422 sec(MFSEXE.EXE)
WALL CLOCK TIME      : 1819.17 sec
TOTAL WARNINGS : 17
```

C. 8 CPU Threads + Memory Extension 100GB : 786.03 sec (49.6%)

```
[SYSTEM INFO]
NUMBER OF THREADS    : 8
MAXIMUM MEMORY USAGE : 50196 MB
AVAILABLE MEMORY     : 96377 MB
TOTAL CPU TIME       : 742.562 sec
WALL CLOCK TIME      : 786.003 sec
TOTAL WARNINGS : 18
```

D. 8 CPU Threads (No Memory Extension) : 499.77 sec

```
[SYSTEM INFO]
NUMBER OF THREADS    : 8
MAXIMUM MEMORY USAGE : 50253 MB
AVAILABLE MEMORY     : 98716 MB
TOTAL CPU TIME       : 1290.66 sec
WALL CLOCK TIME      : 499.767 sec
TOTAL WARNINGS : 17
```

Other improvements

<Improvement in Geometry-Based CFD Monitoring Definition>

During CFD monitoring, an issue was identified where monitoring conditions remained even after the monitored surface was modified or removed, leading to abnormal termination. To address this, unused monitoring conditions are now automatically deleted, and a comment is logged in the output window. Additionally, previously inaccessible files due to this issue can now be opened properly.

<Improvement in Heat Flux (Convection) Handling in Mixture Flow Analysis Module>

When performing mixture flow analysis with flux-type loads, it was found that heat flux information was being ignored as the solver received both mixture flux and heat flux data. This issue has been fixed to ensure that heat flux (convection) is correctly applied even in mixture flow conditions.

<Modification of Fatigue Analysis Formula in Random Vibration Analysis>

An error in the calculation of certain parameters in the Steinberg method for fatigue analysis in random vibration analysis was identified and corrected. For existing analysis files, re-analysis of the fatigue analysis is required if verification using the updated formula is necessary.

<Unit Conversion Error in NFX CAD Export>

An issue was identified where certain models were being converted to 1/100 of their original size during CAD export using the Parasolid Kernel provided by NFX. This has been corrected to ensure that the models are saved according to the current working unit system.

<Correction of Missing Temperature Data in CFD .res Result Files>

An error was identified where temperature values were missing in the *.res result files of CFD analysis when used for load generation. This issue has been resolved by enabling the addition and conversion of node-based temperature data, allowing for faster data mapping in fluid-structure transformation functions.

<Expansion Function Enhancement for Element Visibility>

Previously, the Expand function in element visibility allowed checking only one adjacent target. The new "Expand All" feature enables full expansion, and welds and contacts are now included as expansion targets. This helps verify model connectivity, and the inverse function allows users to identify unconnected parts in advance.

<Improvement in Energy (W) Definition for Heat Flux and Heat Generation in Structural Analysis>

In structural analysis, heat flux and heat generation conditions were previously applied as secondary calculated values in W/m^2 and W/m^3, causing inconvenience for users. A new feature has been added to allow direct definition of total energy (W), which is automatically applied by the solver based on the selected target.

Other improvements

<Addition of Multi-Node Connection and Flow Coupling Grouping in 1D Flow Analysis>

In 1D flow analysis, conditions are now inherited when copying elements. Additionally, flow coupling has been expanded to allow batch selection and application of 3D volume surfaces and 1D nodes in manifold structures, enabling faster multi-definition workflows.

<Fix for Graphic Distortion Issue When Checking Minimal Geometries in Simplification Feature>

An issue was identified where extremely small CAD items would cause graphic distortion or disappear when double-clicked for repositioning due to a scaling error. This problem, which resulted in a reduced screen view, has been identified and corrected.

<Improved Program Uninstallation Speed>

Previously, CAD libraries and font files from third-party applications were added to the system during installation, causing delays in the uninstallation process. To resolve this, the system no longer installs fonts. Depending on the user's environment, the previous uninstallation process could take up to 5 minutes. With the updated version, the process now starts immediately and completes within 30 seconds.