



Multidisciplinary Integrated Analysis Solution for Design Optimization

# midas NFX 2026 Feature Overview



# midas NFX

## New Features

### Recommended Key Updates(Release Version : 2026.03.03)

- CAD Interface update
- Added rotor dynamics analysis capability
- Added Slope Factor option to improve stability of nonlinear surface contact
- Expanded analysis capabilities for fluid and particle results
- Optimized workflow through expanded Drag & Drop functionality in analysis settings
- Added Value-based load visualization for more intuitive load representation
- Supported integrated material modeling for thermo-nonlinear analysis
- Optimized fatigue life evaluation workflow for Result Combination cases
- Improved system stability and engineering workflow in midas NFX

midas NFX provides a fully integrated simulation environment for structural, fluid, and optimization analysis using a single model within a unified workspace.

With its Windows-based GUI and MIDAS customer support ecosystem, the software enables engineers to perform simulation efficiently while providing a structured environment for technical training and support.

## 1. CAD Interface Update

The CAD Interface has been updated in accordance with the CAD version updates. The CAD Interface will continue to be updated in line with the updates of supported CAD systems. If compatibility issues occur with the latest CAD version, we recommend converting the file to Parasolid format before importing it. We will continue working to ensure faster support for the latest CAD versions.

Category	File Extension	Applicable Version
Parasolid	x_t, xmt_txt, x_b, xmt_bin	9.0 ~ 38.0.x
ACIS	sat, sab, asat, asab	R1 ~ 2026.1.0
STEP	stp, step	AP203, AP214, AP242
IGES	igs, iges	Up to 5.3
Pro-E / Creo	prt, prt.*, asm, asm.*	16 ~ Creo 12.0
SolidWorks	sldprt, sldasm, slddrw	98 ~ 2026
CATIA V4	model, exp, session	4.1.9 ~ 4.2.4
CATIA V5	CATPart, CATProduct	V5 R8 ~ V5-6R2026
Unigraphics	prt	11 ~ NX2506
Inventor Part	ipt	V6 ~ V2026
Inventor Assembly	iam	V11 ~ V2026
SolidEdge	par, asm, psm	V18 ~ SE2026

## 2. Added Rotor Dynamics Analysis Capability

The Rotor Dynamics analysis feature has been added, enabling users to evaluate the stability of rotating machinery systems.

With this feature, vibration phenomena occurring in rotating systems such as engines, turbines, and motors can be predicted more accurately within the analysis environment.

### 1. Vibration behavior depending on rotational speed : Complex Eigenvalue Analysis

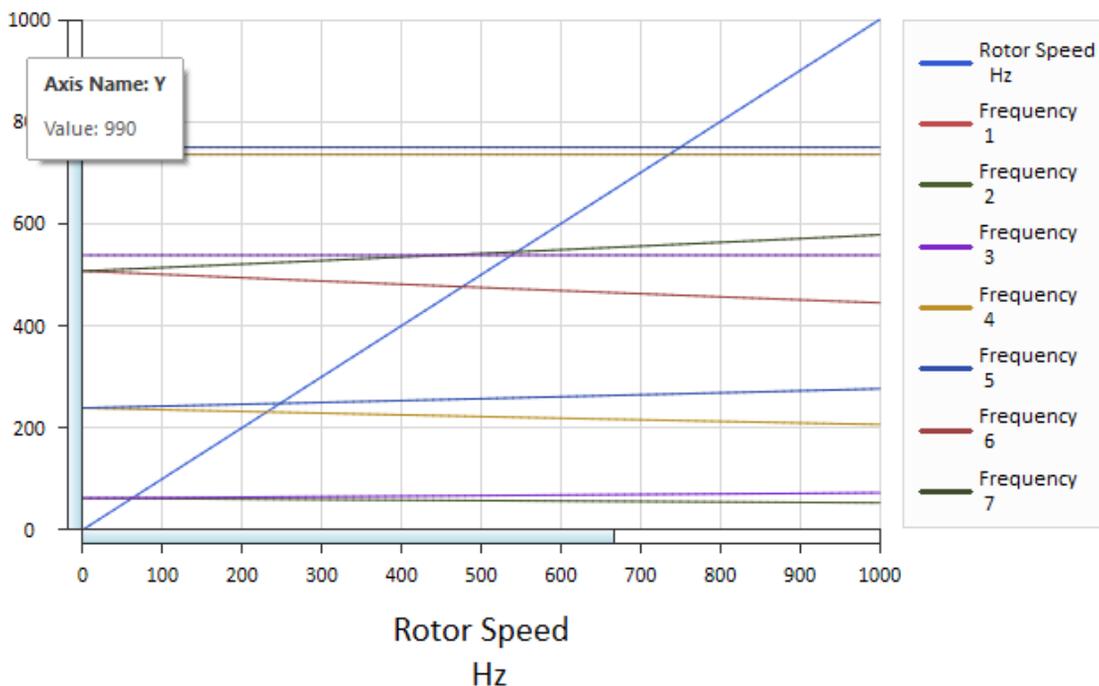
All structures have natural frequencies. However, in rotating systems, vibration characteristics vary depending on the rotational speed.

- Reference Rotating Speed : Users can define the actual operating rotational speed of the system in the analysis model, allowing the physical characteristics of rotating systems to be reflected in the analysis.
- Complex Eigenvalue Analysis : This analysis method considers rotational damping and gyroscopic effects of rotating bodies.

Through this approach, users can numerically evaluate whether the system becomes stable or unstable at specific rotational speeds.

### 2. Stability at a Glance : Support for Campbell Diagrams

Provides a feature to output Campbell Diagrams that visualize the relationship between a rotor's rotational speed and its natural frequencies. Users can easily grasp the trend of frequency changes relative to rotational speed. By immediately identifying critical speeds, potential failure risks due to resonance can be preemptively blocked during the design phase.



## 2. Added Rotor Dynamics Analysis Capability(Continue)

### 3. Efficient Modeling: Utilization of Fixed Reference Frames

In this update, analysis efficiency has been enhanced by adopting a Fixed Reference Frame approach. Since the analysis is performed from the perspective of observing the rotating body from the outside, the following modeling principles apply:

- Rotating Region (Rotor): Elements defined as rotors must have a perfectly axisymmetric shape to ensure physical consistency.
- Stationary Region (Stator): Fixed components such as bearings or housings that support the rotor can be modeled freely without shape constraints, allowing the stiffness of the entire system to be reflected.

### 4. Reflecting Precise Physical Phenomena: Consideration of Gyroscopic and Circulatory Matrices

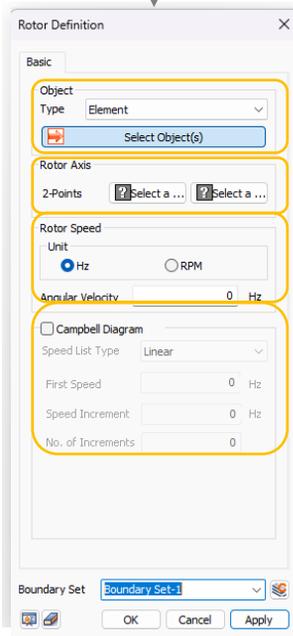
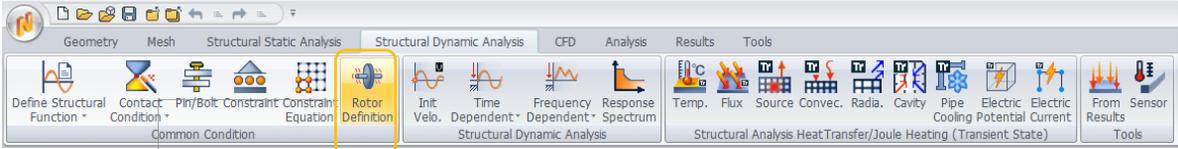
For elements defined as rotating regions, special physical phenomena occurring during rotation are incorporated into the calculations through the following matrices, in addition to simple mass and stiffness:

- Gyroscopic Matrix: Accurately calculates the inertial effects arising from the rotor's tendency to maintain its axis of rotation.
- Circulatory Matrix: Accounts for instability components that may arise from energy circulation due to internal damping or friction within the system.

## 2. Added Rotor Dynamics Analysis Capability(Continue)

### 5. Rotor Definition Method

In the Structural Dynamics common tab, the analysis conditions are defined using the Rotor Definition function, and the analysis is executed as a Complex Mode Analysis.



Select Node 1 of the rotation axis.

Select the model to be analyzed.

Select the nodes that define the rotation axis.

Enter the rotational speed of the shaft.

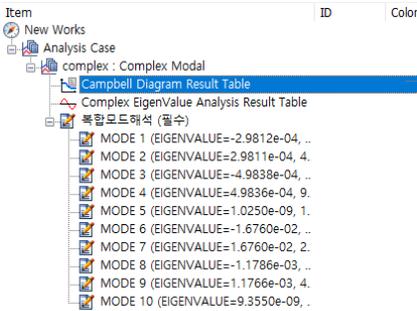
Enter the parameters required to generate the Campbell diagram.

Enter the parameters required to generate the Campbell diagram, including the starting speed, speed increment, and number of increments.

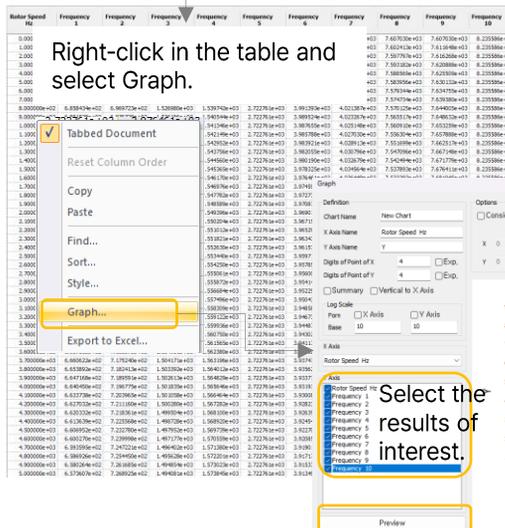
In this example, the speed range is defined from 0 Hz to 5,000 Hz with 100 Hz increments (50 increments total).

Select Node 1 of the rotation axis.

### 6. Result Analysis

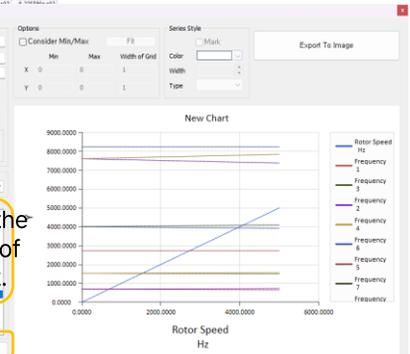


Double-click Campbell Diagram Result Table.



Right-click in the table and select Graph.

Select the results of interest.



### 3. Enhanced Slope Factor for Improved Nonlinear Contact Convergence

To address numerical instability that may occur at the moment of initial contact in nonlinear analysis, the contact stiffness control capability has been enhanced. This feature is particularly effective for large assembly models where component alignment is not perfectly matched, helping resolve initial contact detection issues and significantly reducing unnecessary computational cost.

#### 1. Technical Background and Necessity

Conventional nonlinear contact models assume that the change in contact force is proportional to the penetration depth, meaning the slope begins from zero. However, from a numerical analysis perspective, when the initial slope is zero, it becomes difficult to clearly determine whether sliding contact occurs at the moment of contact. As a result, numerical instability during contact calculations may occur, often causing convergence failures. To address this issue, users previously had to create multiple analysis stages to gradually apply contact conditions and stabilize the initial contact state. This approach increased both analysis time and computational cost.

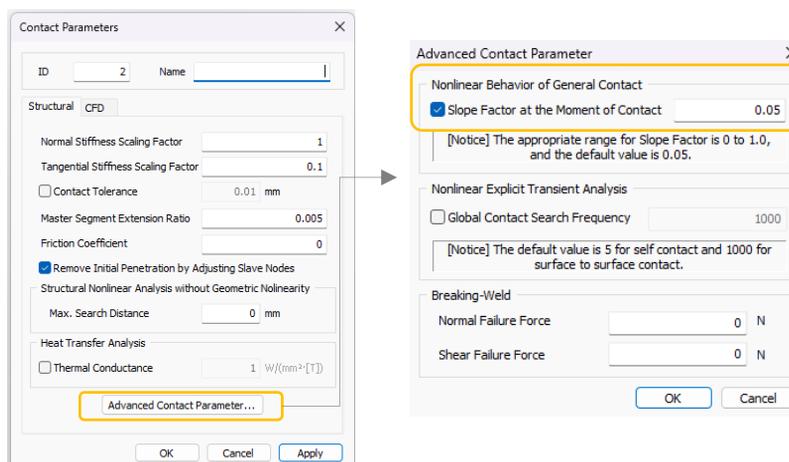
#### 2. Initial Contact Stiffness Control

The optimized Slope Factor parameter ensures a minimum level of contact stiffness at the moment contact occurs, improving the stability of initial contact. As a result, contact stability is improved even at the early stage of analysis. In particular, issues such as gap overlap or repeated unstable contact behavior are significantly reduced.

#### 3. User Configuration

The feature can be defined in the Contact Parameter Settings, allowing users to configure it directly within the Contact Parameter Options..

- Function location: Contact Parameters → Advanced Contact Parameter
- Setting option: Slope Factor at the Moment of Contact
- Valid range and recommended value: The recommended range is 0.0–1.0, and the default value 0.05 is suitable for most general contact analysis cases.
- Process efficiency: By preventing failures in the initial contact detection stage, unnecessary preliminary analysis steps can be eliminated, simplifying the overall nonlinear analysis workflow.



## 4. Expanded Arbitrary Surface/Solid Summary Function

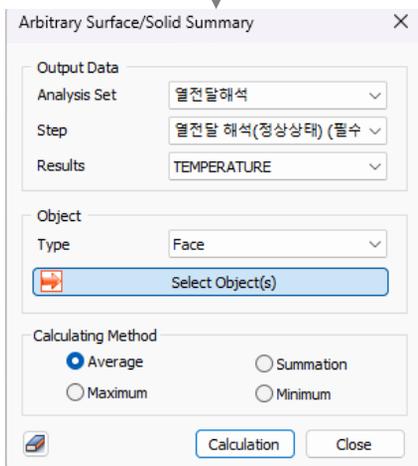
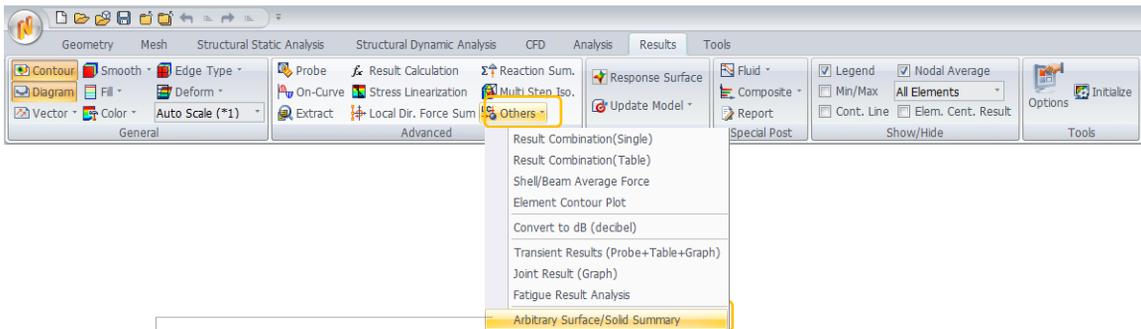
To evaluate the total heat transfer (Heat Rate, W) through a specific surface in heat transfer analysis, users previously needed to extract nodal results and perform manual calculations. With this update, midas NFX allows users to directly obtain Sum, Maximum, Minimum, and Average values for surfaces and volumes with a single click.

### 1. Technical Background and Necessity

In heat transfer analysis, the primary result obtained is Heat Flux ( $W/m^2$ ), which represents the amount of heat flowing through a unit area. However, engineers often need to know the actual total heat transfer (W) passing through a surface. Previously, users had to export the results of the region of interest, create a table, and perform calculations using external tools such as Excel. This update enables users to select a surface or volume and immediately obtain statistical values such as Sum, Max, Min, and Average, eliminating the need for additional post-processing steps.

### 2. User Workflow

The feature is available after the analysis is completed in the Result Analysis tab → Other Functions. Using the Surface/Volume Summary function, users can review aggregated results not only for heat transfer analysis but also for all analysis cases supported in midas NFX.



#### Analysis Set:

- Select the analysis set for which the results will be reviewed.
- All analysis set results provided in NFX can be selected.

#### Step :

- Select the step of interest.
- If the analysis is a transient or nonlinear analysis with multiple steps, the desired step result can be selected.

#### Result :

- Results generated for each analysis case can be used.

#### Object :

- A Face or Solid model can be selected.
- This function is available only when automatic mesh generation has been performed for the selected Face or Solid.

#### Calculating Method

- You can select Average, Summation, Maximum, or Minimum.

**Node :** Results are extracted from the selected nodes in the target object. The arithmetic average and arithmetic summation are calculated.

**Element :** Results are extracted from the selected elements, and the representative value of each element is used.

## 5. Expanded Drag & Drop Functionality for Analysis Conditions

To maximize user convenience in complex analysis model setup processes, the method for moving Load and Boundary Condition items within the tree menu has been significantly improved. With this update, users can quickly reorganize a large number of analysis conditions with a simple drag-and-drop operation.

### 1. Background and Purpose : Eliminating repetitive engineering tasks

Previously, Geometry or Mesh sets could be freely moved between sets using drag functionality, but this capability was limited for Load and Boundary Condition items.

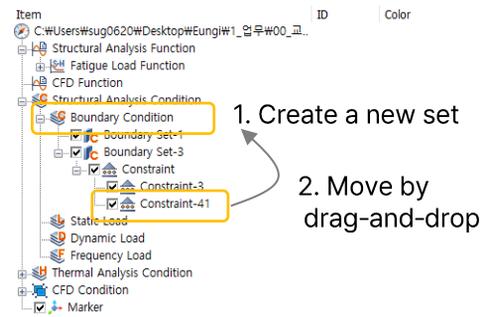
- Previous Method : When conditions were assigned to an incorrect set, users had to return to the Set window (Property window) and manually reassign the target set. When managing a large number of load items, this process became repetitive and inefficient.
- Improved Method : Similar to moving files in a file explorer, users can now drag and drop conditions directly in the tree menu, allowing multiple conditions to be moved instantly to a new set.

Target Category	Applicable Scope (Tree Menu Sub-items)
Structural Analysis Conditions	Boundary Condition, Static Load
Thermal Analysis Conditions	Heat Flux, Transient Heat Flux, Electric Load
Fluid Analysis Conditions	Fluid Boundary Condition, Pressure Distribution

### 2. Detailed Workflow

Users can organize the model with a simple two-step operation.

- Create a new set : Right-click in the Tree Menu and create a new condition set.
- Drag & Drop : Select the existing conditions to move and drag them into the newly created set.



### 3. Expected Benefits

- **Significant reduction in preprocessing time and improved productivity** : Grouping and reorganizing hundreds of load conditions can reduce preprocessing time by more than 90% compared to the previous workflow.
- **Prevention of user errors** : When modifying sets in the property window, selection mistakes may occur. Drag-and-drop operations provide immediate visual feedback, reducing the risk of such errors.
- **Improved model management efficiency** : When performing Load Case Study tasks that require different load combinations for each analysis case, the drag-and-drop workflow significantly improves data management efficiency.

## 6. Improved Visualization through Added Load Value Display

To allow users to verify the validity of load conditions immediately during the preprocessing stage and improve the readability of analysis reports, a load value display feature within the model view has been added.

### 1. Background

In the existing NFX environment, loads were displayed only as directional arrows, making it difficult to intuitively recognize the magnitude of the applied load.

- Inconvenience : To verify whether load values were entered correctly, users had to check the properties of the load set or open separate tables.
- Documentation limitation : When preparing analysis reports, users often had to manually insert additional text annotations to describe the load conditions

### 2. Core Function

The feature provides real-time numerical values in the modeling workspace for the most frequently used load types: Force and Pressure.

Load Type	Display Method and Characteristics
<b>Force</b>	The magnitude of the load entered at a node is displayed as text together with the directional arrow. Positive/negative values are indicated according to the sign convention.
<b>Pressure</b>	The applied pressure is converted to nodal loads and displayed with direction and numerical values at the element centers. The sign determined by the input value and summation is preserved.

Note : This feature currently applies primarily to loads directly assigned to Node and Element, and is also applicable to Static Load cases.

### 3. User Customization Options

Users can adjust how numerical values are displayed through the Options menu.

- Display Value :

The numerical value display can be turned on or off using a checkbox.

- Size & Color:

The value size (Value Size) and value color (Value Color) can be adjusted depending on model size or background color to improve visual clarity.

- Menu Path:

Options → Loads/B.C. → Display

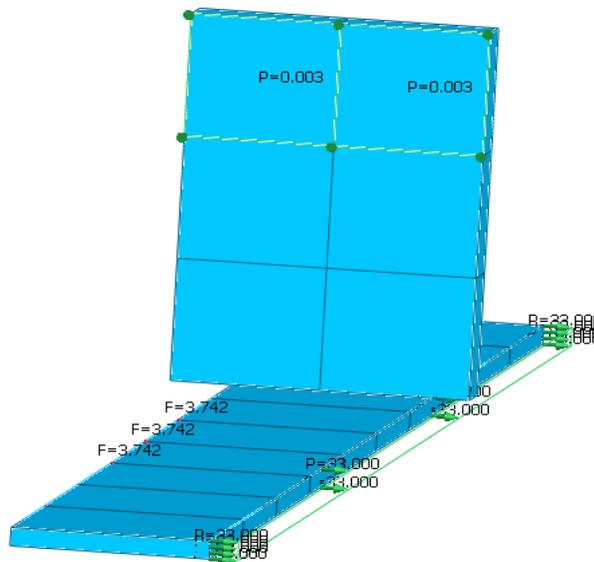
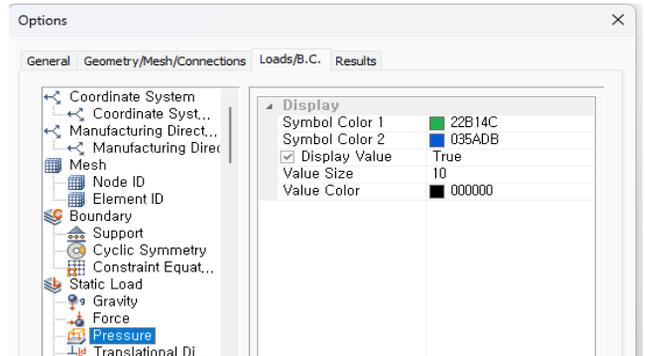
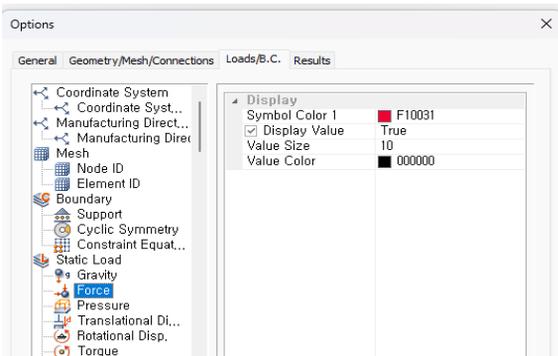
## 6. Improved Visualization through Added Load Value Display(Continue)

### 4. Expected Benefits and Use Cases

- Error Prevention : Human errors such as incorrect unit input or mistakes in decimal placement can be visually identified immediately.
- Improved Report Clarity : Load conditions are clearly displayed directly in the model image, allowing them to be included in reports without additional annotation editing, improving communication with collaborators and clients.

[ Force ]

[ Pressure ]



## 7. Integrated Material Modeling Support for Thermal-Nonlinear Analysis

This update addresses numerical constraints in material definitions that may occur during transient thermal analysis, enabling more accurate prediction of permanent structural deformation under high-temperature environments.

### 1. Technical Background : Ensuring Data Consistency in Coupled Analyses

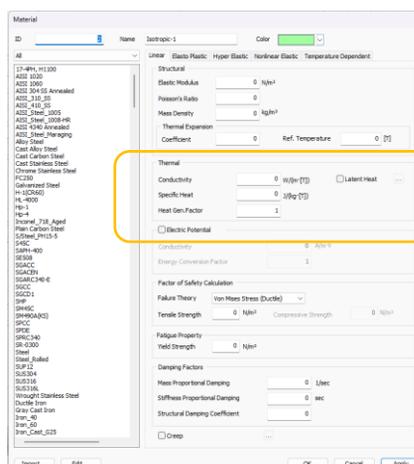
This update addresses numerical constraints in material definitions that may occur during transient thermal analysis, enabling more accurate prediction of permanent structural deformation under high-temperature environments.

- Technical Background : In real-world physical systems, thermal and structural fields interact with each other. In particular, transient thermal analysis involves time-dependent heat transfer, while nonlinear structural analysis involves material nonlinearities and geometric nonlinearities, making the interaction between the two analyses critical.
- Limitations of the previous system : Previously, nonlinear material models were primarily designed for structural behavior such as elastic-plastic deformation, which sometimes caused numerical inconsistencies when thermal properties were defined separately.
- Improvement introduced in this update : Metallic materials operating at high temperatures experience both mechanical deformation and heat energy diffusion simultaneously. This update improves the material definition framework so that these thermo-mechanical coupled behaviors can be defined within a single material model.

### 2. Key Enhancement : Integration of Thermal Properties in Nonlinear Material Definitions

The material definition framework has been improved so that thermal properties can be defined directly within nonlinear material models.

By allowing both nonlinear mechanical properties and thermal properties to be defined together, transient thermal analysis can now be performed more efficiently and consistently.



Key thermal properties such as thermal conductivity and specific heat can now be defined within the material model.

## 8. Optimized Fatigue Life Evaluation Process for Result Combination Cases

For Result Combination analysis cases based on the Linear Superposition Principle, a fatigue analysis linkage feature has been officially implemented to evaluate structural durability and reliability. This enhancement significantly simplifies the fatigue life evaluation workflow for complex loading scenarios.

### 1. Technical Background : Relationship between Result Combination and Fatigue Analysis

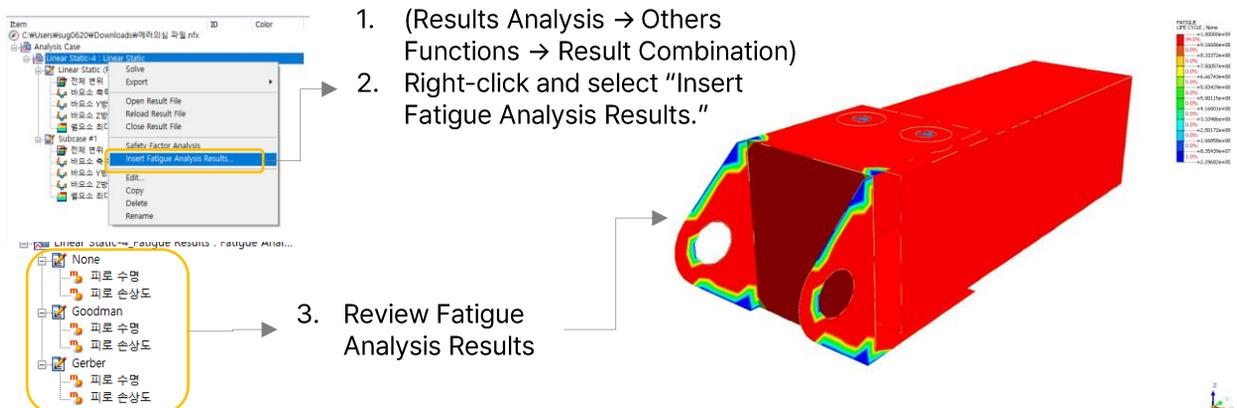
In real structures, loads are rarely applied individually. Instead, structures are typically subjected to multiple loads acting simultaneously in different directions.

- Role of Result Combination : Linear static analysis results of individual Unit Loads can be mathematically combined to obtain the structural response under specific operating conditions.
- Connection to Fatigue Analysis : Fatigue failure occurs due to repeated stress amplitudes. Therefore, combined result cases serve as base load cases for fatigue life evaluation.

### 2. Key Update : Integration of Fatigue Evaluation with Result Combination

Previously, fatigue evaluation required additional steps to transfer results from the Result Combination analysis case to the fatigue analysis environment. With this update, the workflow has been directly integrated.

- Context Menu Support (Right-click) : The context menu has been expanded so that users can execute Add Fatigue Result directly by right-clicking a Result Combination case in the tree menu.
- Compatibility with Linear Static Cases : The data mapping logic has been enhanced so that Result Combination cases, which have a similar data structure to linear static results for a single step, can be directly recognized by the Fatigue Solver.



## 8. Optimized Fatigue Life Evaluation Process for Result Combination Cases

### 3. Numerical Analysis Process and Workflow

- **Reduced Analysis Resources:** There is no need to individually analyze multiple complex loading conditions. Various fatigue cycles can be generated through combinations of base loads.
- **Accurate Life Prediction:** Using the combined stress tensor, the system compares the results with the S-N curve (Stress–Life curve) to estimate the expected fatigue failure location of the component.

### 4. Expected Benefits and Engineering Recommendations

This feature is particularly useful for durability design of mechanical systems subjected to repetitive vibration or irregular load histories (Load History).

- Since Result Combination assumes linearity, fatigue evaluation using this method is recommended when the model does not include significant material nonlinearities or large deformations.
- If nonlinear behavior is significant, it is recommended to perform fatigue evaluation using results obtained from step-by-step nonlinear analysis.

## 9. Improved midas NFX System Stability and Engineering Workflow

This update improves numerical stability in the CFD solver, enhances the computational performance for large structural models, and increases usability during preprocessing and post-processing stages.

### 1. Improved CFD Solver Convergence and Numerical Stability

CFD analysis often involves strong nonlinear behavior, which can lead to numerical divergence. This update reduces such risks and improves solver robustness.

- **Restored stability settings for 1D CFD** : Solver parameters used to detect numerical singularities have been optimized and restored to improve convergence stability and prevent solution oscillations.
- **Error fixes for large models and boundary condition conflicts** : Errors occurring during Out-of-Core calculations when handling large-scale CFD models have been resolved to ensure stable computation in large projects.
- **Improved memory management for advanced turbulence models** : Memory leaks that occurred during calculations using the 7-equation Reynolds Stress turbulence model have been resolved, preventing resource exhaustion during long simulations.
- **Improved physical boundary condition interpretation** : Errors caused by incorrect interpretation of inlet flow directions when using reference or local coordinate systems have been corrected.
- **Clarified transient analysis time-step output** : The solver message and output time information have been synchronized to eliminate confusion during steady-state calculations.
- **Enhanced overset mesh convergence and stability** : Overset mesh techniques are commonly used when structures move within narrow passages or internal flow regions. Previously, rapid pressure changes in narrow regions caused convergence delays and solver warnings such as "Trying other solver." The numerical treatment of overset mesh boundaries has been improved to ensure stable convergence in complex moving boundary problems.

### 2. Improved Reliability of Structural and Fatigue Analysis

Structural durability prediction and fatigue analysis performance have been enhanced for improved accuracy and efficiency.

- **Structural analysis performance improvement** : Numerical processing of singular states has been improved, significantly reducing computation time for large-scale models that previously required hours of calculation.
- **Improved fatigue stress calculation accuracy** : The Exact Nodal Average method has been applied to better represent stress concentrations at nodes. In addition, issues related to file path conflicts during analysis execution have been resolved.
- **Resolved Gap element issues in nonlinear dynamic analysis** : Errors in Gap elements used in nonlinear dynamic analysis have been corrected to ensure stable operation.
- **Improved data consistency for nonlinear Bush elements** : An issue where 3D table data for nonlinear Bush elements caused ID mismatches during solver transfer has been resolved, preventing result discrepancies in practical models.

## 9. Improved midas NFX System Stability and Engineering Workflow

### 3. Pre-processing and UI Usability Improvements

These updates focus on improving the user interface to reduce the time engineers spend on modeling tasks.

- **Element Control Menu Position Adjustment** : For improved usability during element alignment tasks, the Element Control and Element Parameter menus are now fixed at the top of the interface.
- **Improved 2D → 3D Mesh Generation Performance**: An issue where the program could freeze during internal mesh generation for complex models containing intersecting elements has been resolved.
- **Improved File Generation Speed for Large Thin Wall Models** : A visualization issue caused by shared rendering buffers—where color boundaries became unclear when elements with different material properties were mixed—has been corrected.
- **Element Set Grouping Function Added** : A new function allows elements of specific interest within large assembly models to be grouped and managed separately. This enables engineers to quickly filter model regions during result analysis and efficiently extract data for reporting.

### 4. Post-processing and Fluid Analysis Enhancements

- **Improved Voxel Resolution for Flow Path Analysis** : The voxel resolution has been optimized to prevent fluid flow paths from leaving the mesh, improving the clarity of result visualization.
- **Fix for Radiation Heat Transfer Restart Error** : An issue where the final temperature from the previous step affected results during radiation heat transfer restart analysis has been resolved, improving the reliability of sequential analyses.
- **Fix for Termination Error in Partition Boundary Conditions** : An issue where simulation termination errors occurred when partition boundary conditions were included in partition-based analyses has been corrected.
- **Extended Input Length for Fluid Function Expressions** : The character limit for fluid analysis function expressions has been increased from 128 characters to 1024 characters.