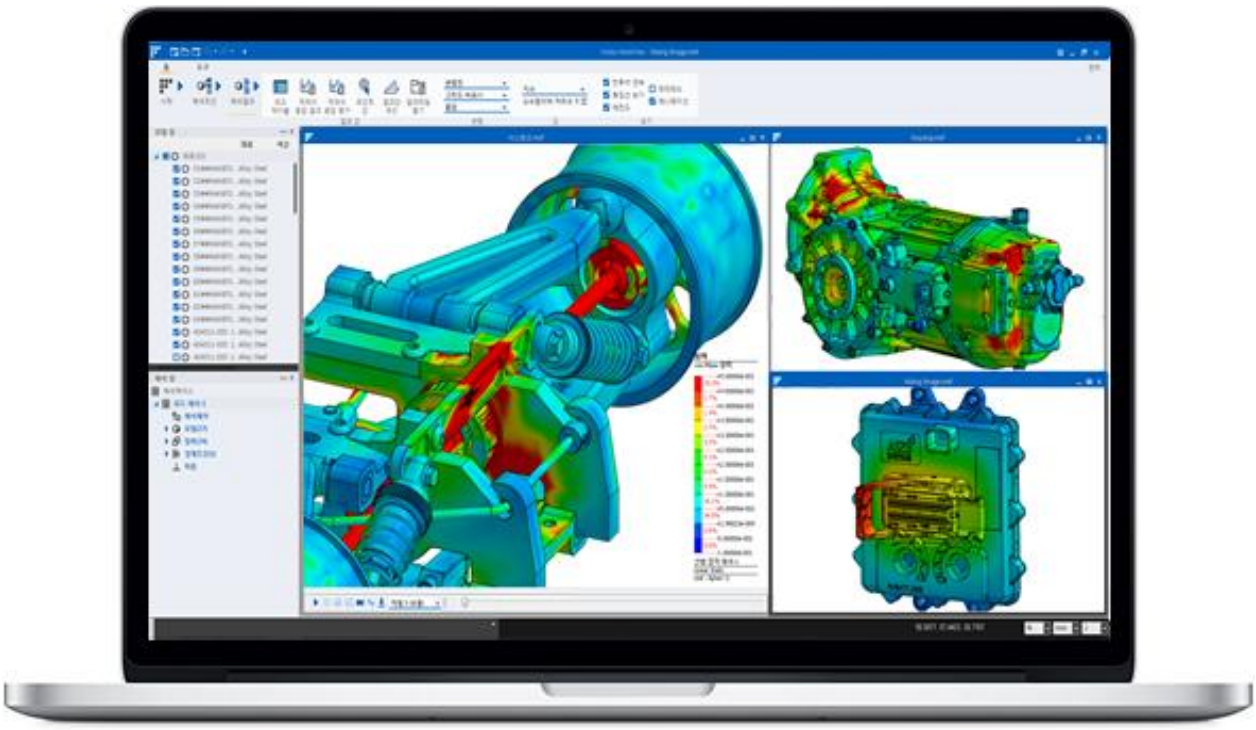
MIDASIT



MeshFree 2026 Release Note

Connecting to a New World

In April 2017, MeshFree 1.0 was released, introducing a small but meaningful change to conventional CAE technology. By moving beyond traditional mesh generation, this innovation overcame the technical limitations of FEM-based CAE, which had dominated the industry for more than half a century. MeshFree enables design engineers to directly utilize the original geometry created during the design process. This allows engineers to perform simulations themselves, verify product performance, and carry out design optimization—driving a transformation in the CAE workflow within the design stage. With MeshFree 2025, further improvements have been made to enhance usability and strengthen its role as a reliable partner for engineers. Continuous development efforts have also been made to improve system reliability and user experience.

CAD Interface Update

Type	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

Support for the latest CAD versions in the CAD Interface may be delayed depending on updates from partner vendors. If the latest version is not supported, we recommend converting the file to a neutral format such as Parasolid before importing it. We will continue to make efforts to support the latest CAD versions as quickly as possible.

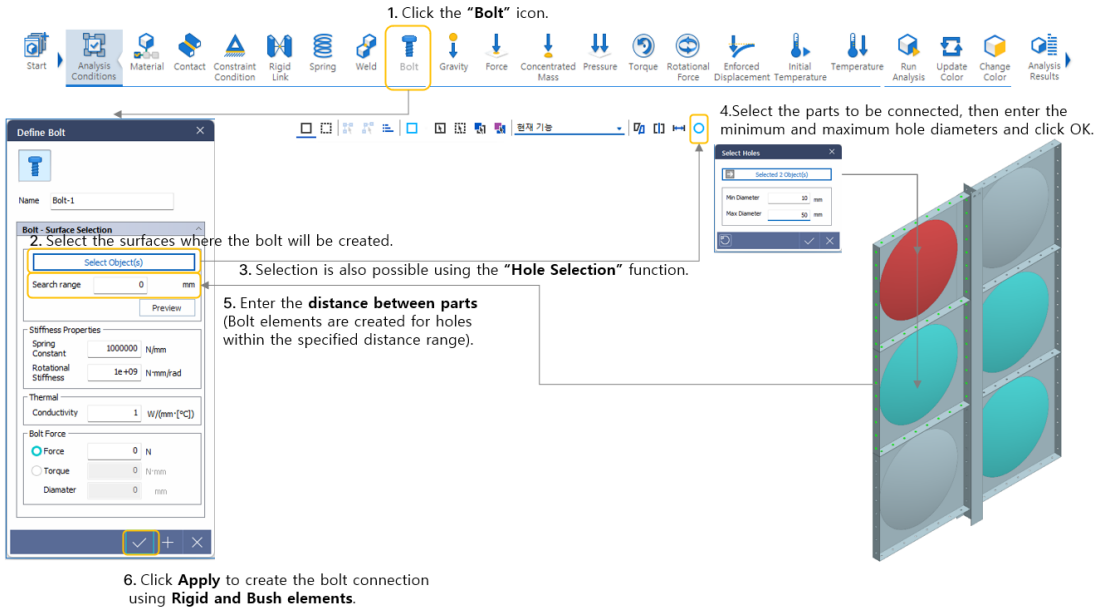
Bolt Connection Feature Added

1. Overview

Mechanical structures are typically assemblies composed of multiple components rather than a single part. This update introduces a feature that automates the bolt connection process, allowing users to calculate both loads acting on bolts and thermal flow through bolt connections.

2. Key New Features (Intelligent Automatic Hole Detection)

- When the user enters the bolt diameter, the software automatically scans the entire model and identifies appropriate holes for bolt connections.
- **Operating Principle :**
    1. **Hole Detection :** Detects circular holes in components that can be used for bolt connections.
    2. **Pairing :** Identifies holes whose center-to-center distance between two components falls within the allowable range
    3. **Creation :** Automatically generates connection elements between the two detected hole centers.



Bolt Modeling (RBE2 + Bush Element)

Instead of representing a bolt as a simple line element, the model uses a combination of two element types to more accurately represent the physical behavior of a real bolt.

- **Rigid Element (RBE2) :** Connects the nodes around the hole perimeter to the center node. It acts as a rigid constraint to prevent deformation of the hole region.
- **Bush Element :** Represents the body of the bolt and models the stiffness characteristics of the bolt. Through the Bush element, forces and shear loads acting on the bolt can be calculated.

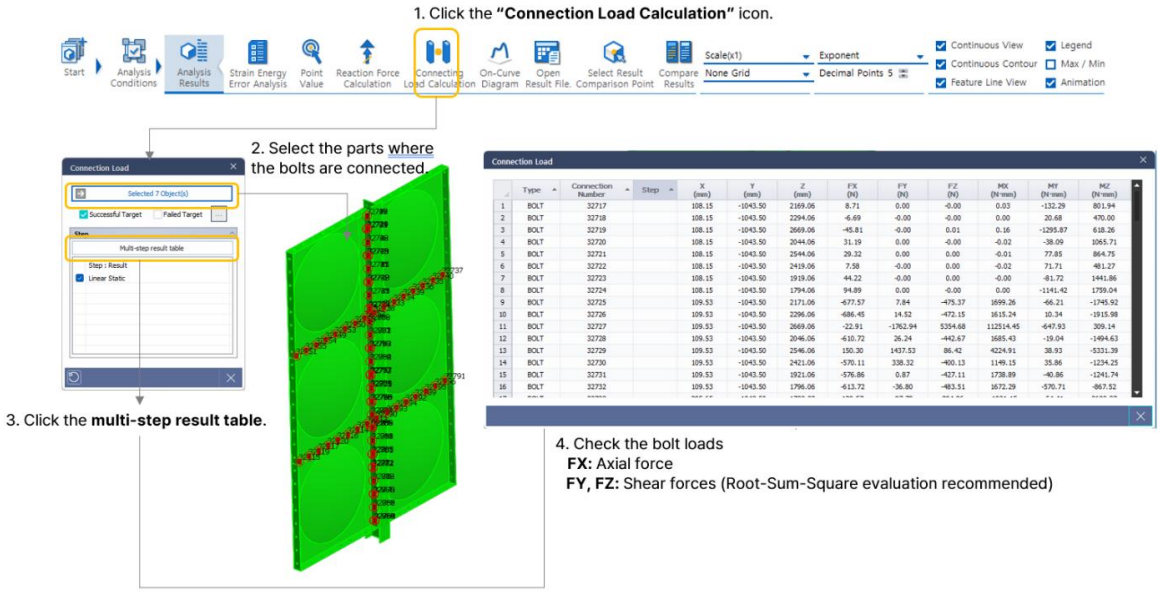
3. Bolt Load and Post-Processing Functions

Bolt Load Application and Thermal Analysis Capability

- **Bolt Load :** Pretension acting on the bolt can be applied. Both force and torque inputs are supported.
- **Thermal Conductivity:** In thermal analysis, heat transfer through the bolt can be considered. Thermal conductivity can be defined to evaluate the heat transfer behavior.

Result Visualization (Post-processing)

The forces and shear forces acting on the bolt can be reviewed using the Connection Load function.



Added Transient Thermal Stress Analysis Capability

1. Overview

Previously, the system supported **only steady-state thermal analysis** and linear thermal stress analysis. With this update, **a transient thermal stress analysis** capability has been added, enabling evaluation of thermal stress under time-dependent temperature variations as well as nonlinear structural behavior caused by temperature changes.

2. Key Features and Concept

In previous versions of MeshFree, thermal stress analysis was typically performed by first conducting a steady-state heat transfer analysis and then applying the resulting temperature distribution as a load condition in a linear structural analysis. With this update, time-dependent temperature results obtained from transient heat transfer analysis can be directly applied as thermal loads in nonlinear static analysis, allowing more accurate evaluation of thermal deformation and thermal stress.

1) Transient Heat Transfer Analysis

- **Steady-state :** A condition in which the temperature no longer changes over time.
- **Transient :** A condition in which temperature varies with time.

2) Transient Thermal Stress Analysis Based on Nonlinear Static Analysis

Temperature changes over time, and the structural response also varies accordingly.
In particular, when temperatures become very high, material properties may change, and nonlinear structural behavior may occur.

- **Material Nonlinearity :** Considers changes in material properties as well as structural deformation due to applied loads.
- **Geometric Nonlinearity :** Accounts for large deformation caused by thermal expansion.

- **Contact Nonlinearity** : Evaluates whether contact between parts occurs due to thermal deformation.

3. Update Details

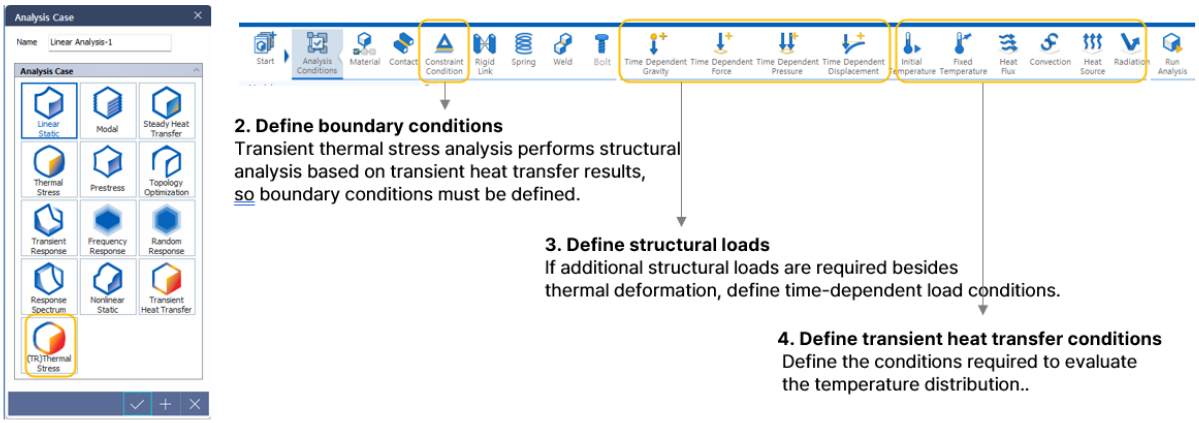
With this feature addition, users can perform the following advanced analysis workflow.

Step	Analysis Type	Main Calculation Description
Step 1	Transient HeatTransfer Analysis	Generate time-dependent temperature distribution data
Step 2	Data Mapping	Transfer calculated temperature results as load conditions for structural analysis
Step 3	Nonlinear Static Analysis	Calculate stress and deformation considering material changes at each time step

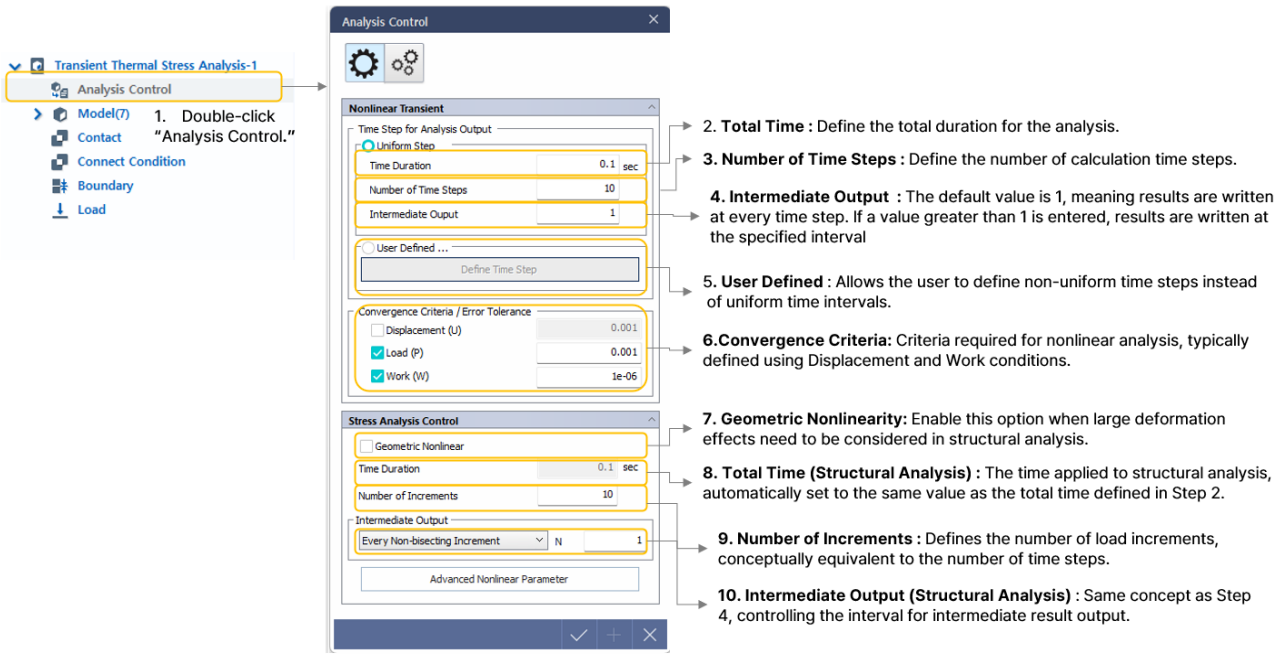
4. How to Use

1) To perform nonlinear transient thermal stress analysis, first select Transient Thermal Stress in the analysis case. The key items to consider when defining the analysis conditions are as follows.

- **Material Definition :** Since this analysis involves both thermal and structural behavior, material properties required for both analyses must be defined. In addition to thermal conductivity and specific heat, structural properties such as elastic modulus, Poisson's ratio, mass density, and thermal expansion coefficient must also be specified.
- **Boundary Condition Definition**: While conventional heat transfer analysis does not require structural boundary conditions, they are necessary for thermal stress analysis.
- **Structural Load Definition:** If only deformation caused by thermal expansion is considered, structural loads are not required.
  Structural loads can be defined if additional mechanical loads are applied together with thermal loading.
- **Transient Thermal Definition:** This is a mandatory setting for performing transient thermal stress analysis.Temperature or heat transfer conditions must be defined as time-dependent boundary conditions.



2) The analysis settings can be defined in Analysis Control of the Transient Thermal Stress analysis case, where the total analysis time and convergence criteria are specified.



AutoGrid Preview

The key feature of this update is the ability to preview the grid that determines analysis quality before running the analysis.

1. Background:

Previously, MeshFree used the AutoGrid method, which automatically generated an optimal grid based on the computer memory size specified by the user.

- **Limitations of the previous approach :** Users could only verify the grid density after the analysis was completed by checking the results window.
  If the grid was too coarse or too large, users had to modify the settings and restart the analysis from the beginning, which caused unnecessary re-analysis.

2. Key Update

Users can now preview the generated grid in real time during the modeling stage, before running the analysis.

- **Memory-based estimation :** The system calculates the grid density based on available memory and geometric complexity, and displays the expected grid distribution.
- **Geometry detail verification:** Users can verify in advance whether thin plates or small holes are properly reflected in the grid.
- **Real-time preview :** When memory settings are modified, the grid size changes immediately, allowing users to check the updated grid configuration in real time.

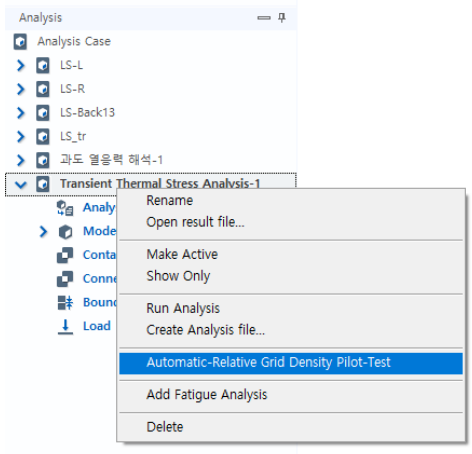
3. Expected Benefits

Item	Before Improvement	After Improvement (AutoGrid Preview)
Usability	Grid quality could only be verified after analysis results were generated	Immediate visual verification after configuration
Analysis Reliability	Risk of errors due to insufficient grid density	Improved reliability by ensuring appropriate grid density in advance
Time Efficiency	High probability of re-analysis (Re-run)	Reduced trial-and-error and shorter total analysis time

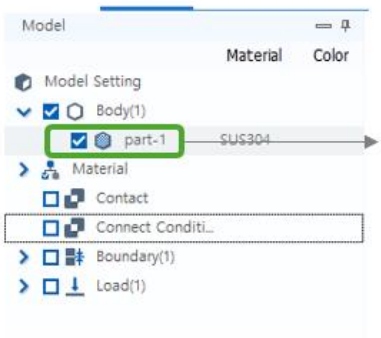
4. How to Use

To preview AutoGrid, the analysis case and analysis conditions (material definition, boundary conditions, and load definitions) must be defined in advance.

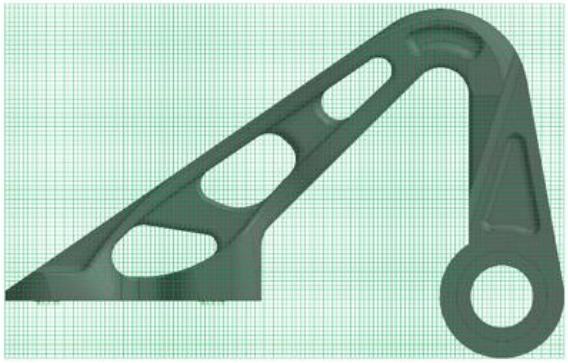
- **Automatic Relative and Density Pilot-Test :** This function is located in the analysis case. As shown in the example below, the function can be accessed by right-clicking the analysis case. When executed, the system performs AutoGrid calculation and grid generation.



- **AutoGrid Preview:** The generated grid can be checked directly in the modeling window.
  By double-clicking the target part, the generated grid will be displayed in green, allowing users to visually confirm the grid distribution.



Double-click the target part to preview the grid, as shown below.



Color Change Feature

MeshFree includes a powerful feature called **color-based load update**. Even when the CAD geometry is slightly modified and the model is reloaded, loads or boundary conditions assigned to specific colors can be **automatically updated**.

This update allows users to **directly control and fully manage this automation feature**.

1. Background

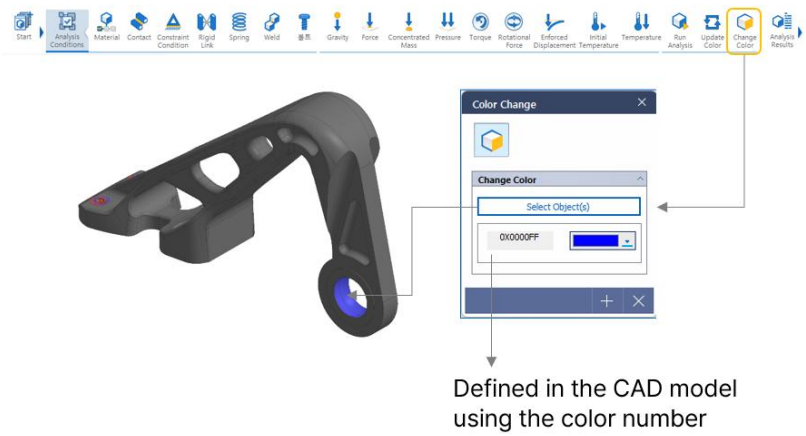
In CAE analysis, loads and boundary conditions can be assigned **based on colors**.

- **Previous inconvenience:** When multiple load surfaces or boundary condition regions existed, each surface had to be selected individually.
  If the design model changed and the analysis had to be rerun with the same load or boundary conditions, users had to repeat the same setup process.
- **Improvement :** In MeshFree, users can modify colors within the software and define analysis conditions based on those colors. By using color codes, the same color-based automatic update feature can be applied even when the CAD model changes.

2. Key Updates

Flexible color control at the Part and Face level

- **Convenience :** With a simple click, users can change the color of the entire part or a specific face.
- **Color number (RGB) specification:** A unique RGB color code is displayed so that the same color values used in the CAD model can be maintained. This is essential for ensuring data consistency between the CAD model and the simulation model.



Color-Based Load / Boundary Condition Management

- Colors can be used as tags. For example, rules such as “Red face = Load 500 N” or “Blue face = Fully Fixed” can be defined and managed directly in MeshFree.

Task	Existing Process	Updated Process
Design Change	CAD modification → color assignment → reload in MeshFree	Direct color modification and automatic mapping in MeshFree
Load Condition Management	Assign loads by manually selecting each face	Automatic load assignment and update based on color codes

Display Mode & Transparency Feature

“The engineer’s eye for discovering hidden truths inside complex assemblies.”

Mechanical systems such as engines, gearboxes, and pumps consist of hundreds of assembled components. From the outside alone, it is difficult to understand what is happening inside the model. This update allows users to freely inspect the interior of complex models.

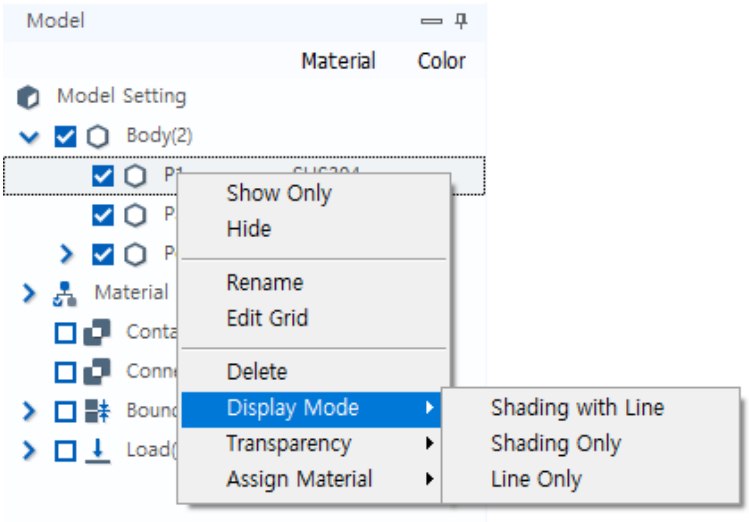
1. Background

Previously, MeshFree provided a shading mode that displayed model surfaces as opaque objects by default.

- **Limitations of the previous approach:**
  For example, when checking whether the load applied to a shaft inside a housing was correctly defined, users had to hide the surrounding casing to inspect it.
  However, hiding the casing made it difficult to verify how the shaft was aligned with the casing.

2. Key Update: Various Visualization Options

In addition to the shading mode, several display modes are now supported, providing better visualization options for complex assemblies. After selecting a target part and right-clicking, users can choose Display Mode or Transparency.



New Display Modes

- **Shaded with Line :** Displays both the shaded surface and the model lines.
  This mode highlights important edges where curvature changes, making it useful for understanding geometric features.
- **Shading Only :** Displays only the shaded appearance of the model without showing the surface edges.
- **Line Only** : Displays only the external edges of the model without shading.
  This mode is useful for identifying internal structural features.
- **Transparency Control :** The transparency of parts can be freely adjusted from Level 1 to Level 9. Level 1 is closest to transparent, while Level 9 is nearly opaque.
  The black edges remain clearly visible, allowing users to accurately identify the positions of internal components.