Femtet Ver.2025.0　What’s New

This document lists the new features added in Femtet2024.1 (only Japanese Version has been released) and Femtet2025.0 in order.

Femtet2025.0 English Version adds all these new features to Femtet2024.0 English Version.

Murata software Co., Ltd.

CAE Software

Femtet Ver.2024.1　What's New

Murata software Co., Ltd.

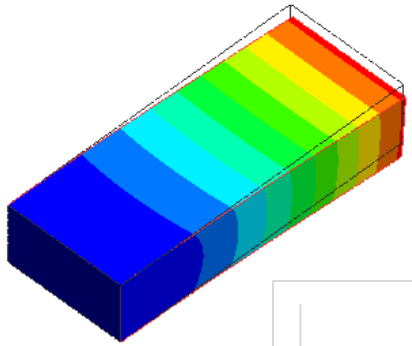
CAE Software

Table of Contents

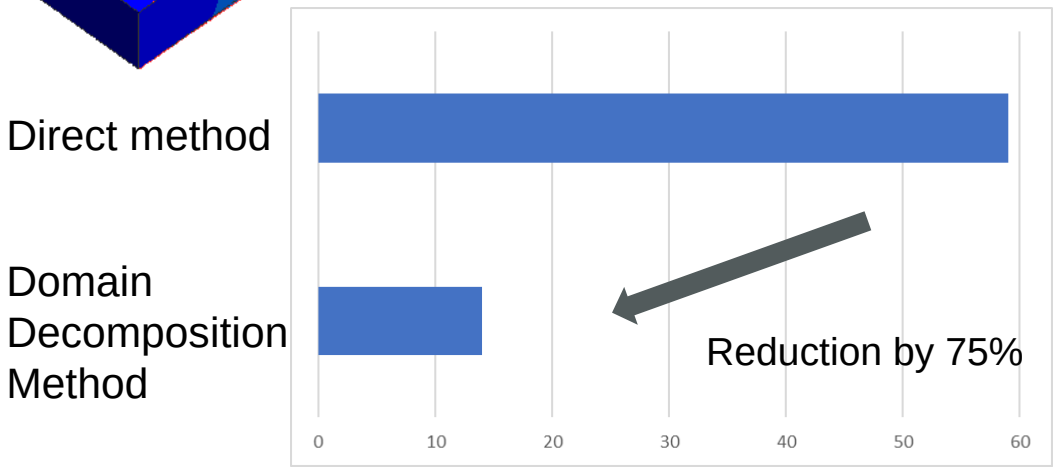
Table of Contents

Solver – Stress Analysis : Domain Decomposition Method execution conditions

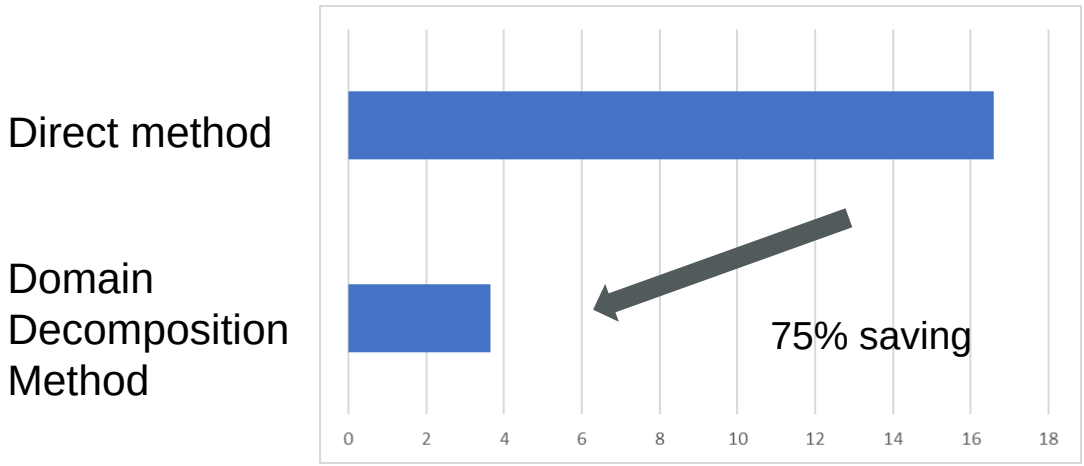
In static stress analysis, the domain decomposition method for faster computation used to require both the Advanced mechanical option and the Accelerator option.
As of version 2024.1, only the Accelerator option is needed.



Stress Analysis example-1
200k meshes



Simulation time(s)

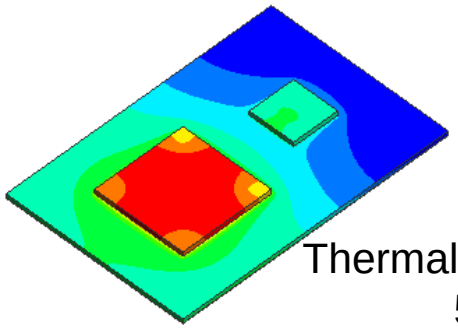


Memory usage(GB)

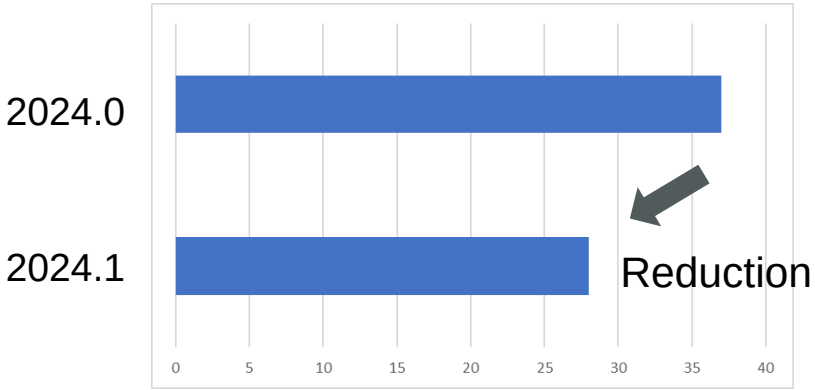
* CPU : i7014700k

Solver – Thermal Analysis/Stress Analysis
Speed up by the algebraic multigrid method

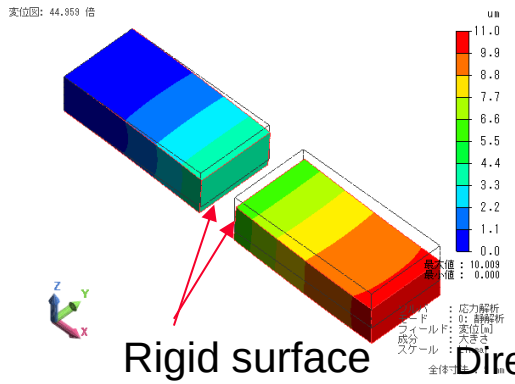
-The algebraic multigrid method, which is an iterative method, has been updated for faster calculations.
-In static stress analysis, the algebraic multigrid method has been enhanced to operate properly under boundary conditions that include rigid surfaces or bonded interfaces.



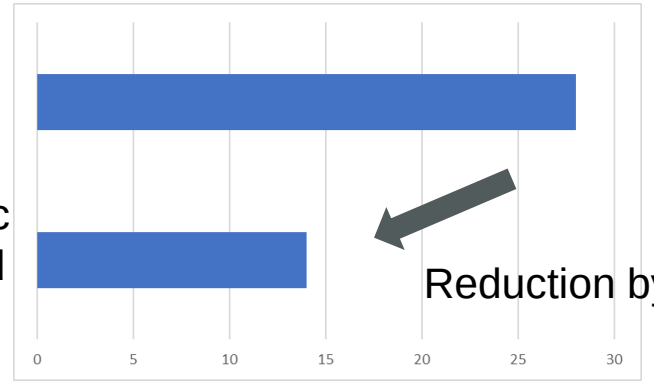
Thermal analysis example-16
500k meshes



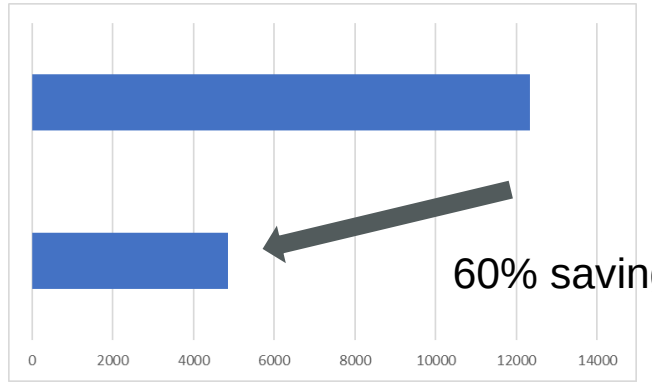
Simulation time(s)



Rigid surface
170k meshes



Simulation time(s)



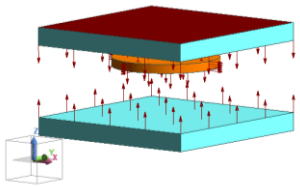
Memory usage (MB)

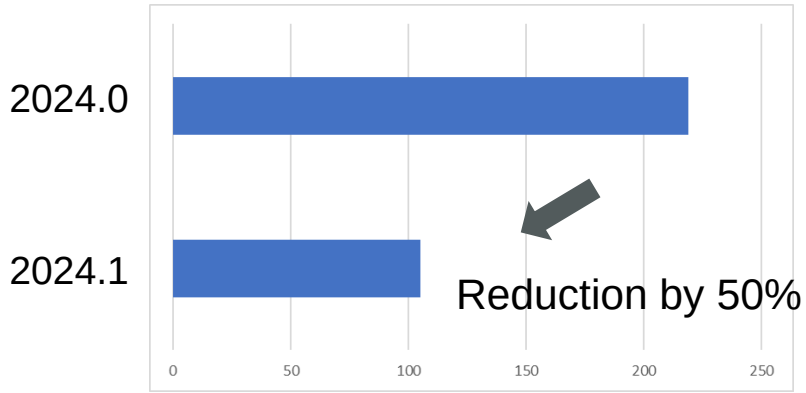
*CPU：i7-14700k

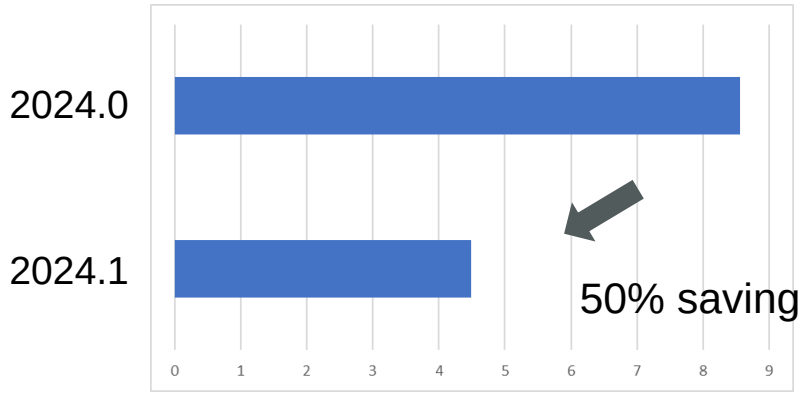
Solver – Thermal Analysis
Faster and less memory by improved thermal radiation

The thermal radiation algorithm has been improved, resulting in faster computation and less memory usage.

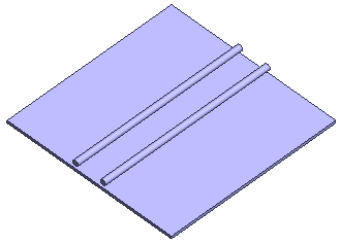
Example-16
680k meshes
iterative method

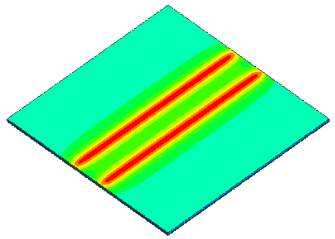


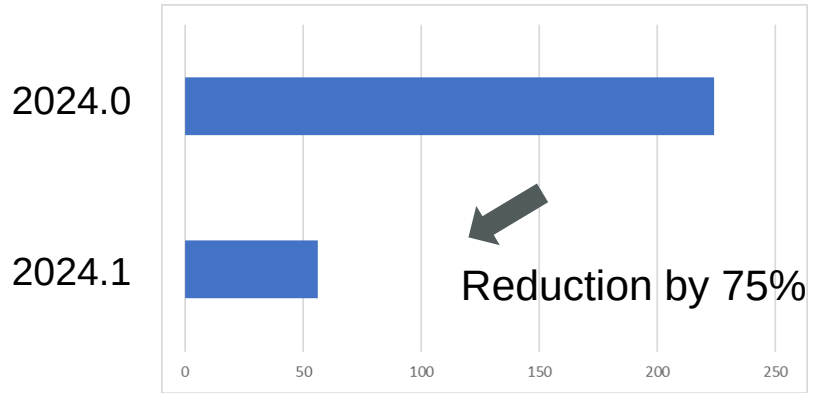


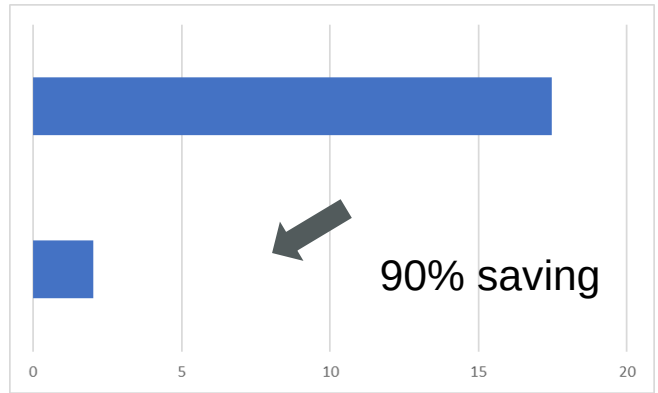


Heater model
350k meshes







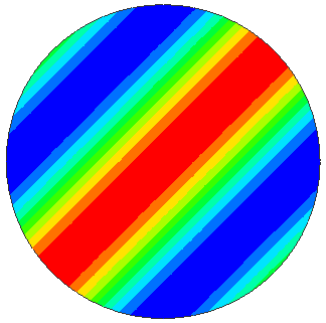


*CPU : i7-14700k

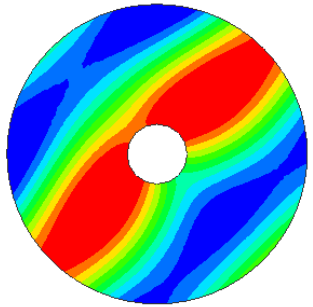
Solver - Acoustic Analysis: A plane wave incidence feature

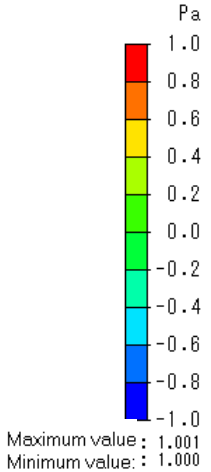
A plane wave incidence feature has been implemented.

Plane wave

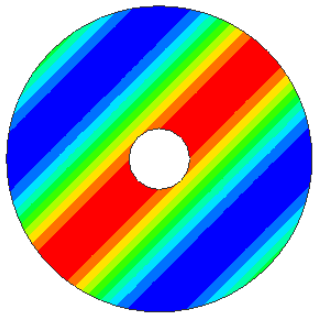


Incident wave + Reflected wave

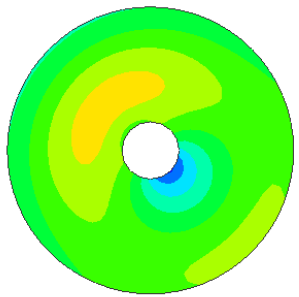




Incident wave

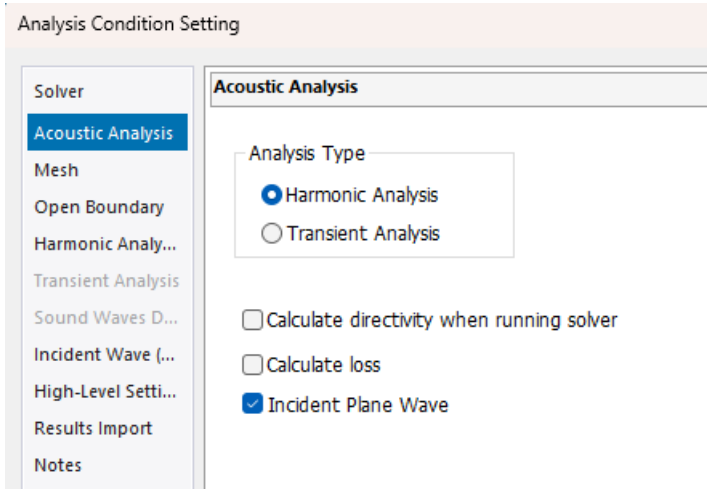


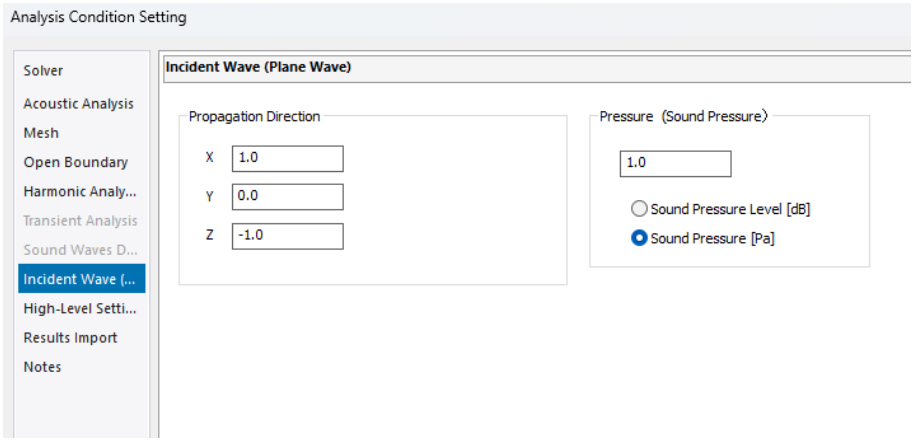
Reflected wave



* This feature support Harmonic Analysis only.

Setting for plane wave incidence feature

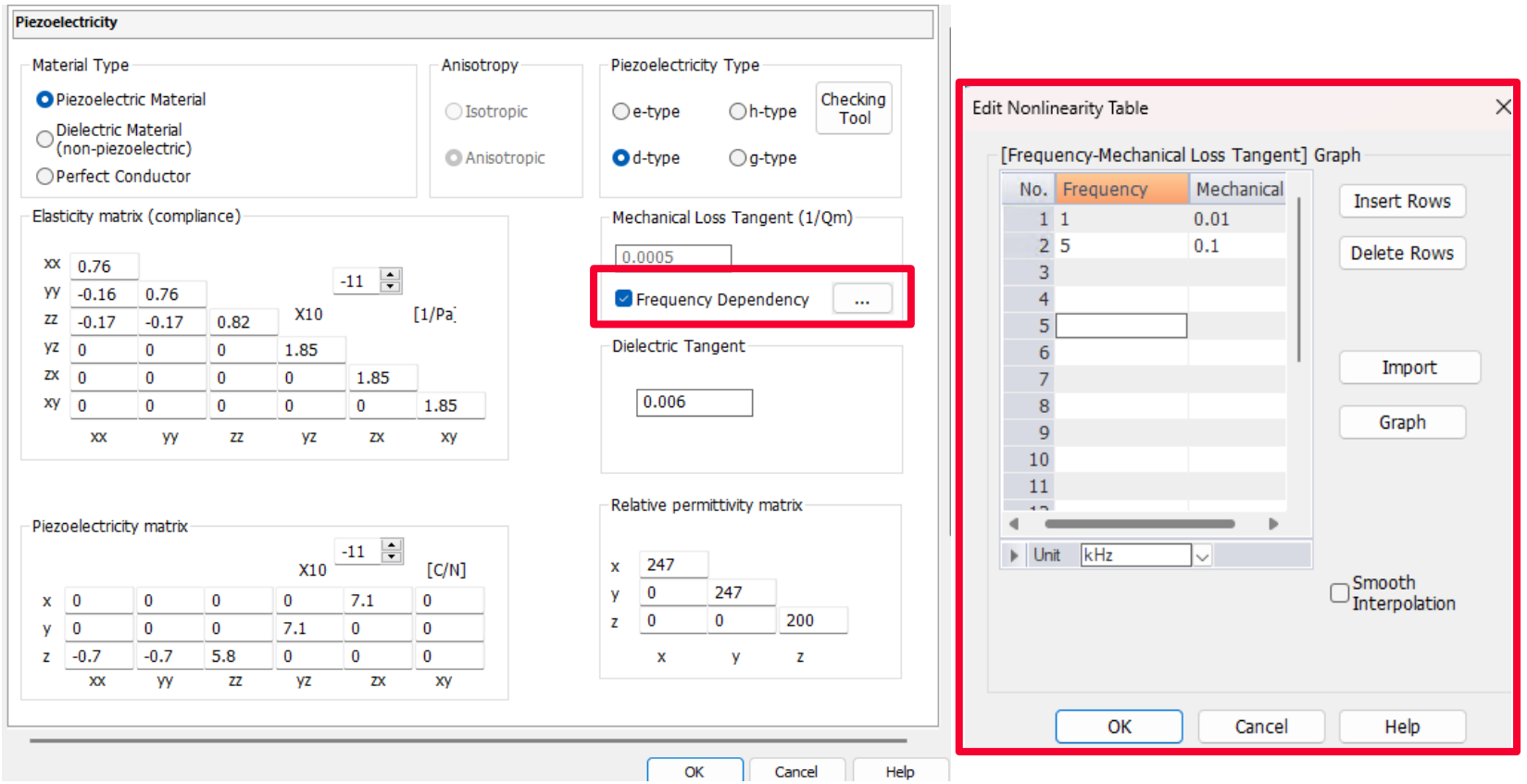




Solver – Piezoelectric Analysis:
The reciprocal of Qm (mechanical quality factor)

The reciprocal of Qm (mechanical quality factor) is supported

Selectable when the frequency sweep is discrete sweep or parallel discrete sweep in the piezoelectric-harmonic analysis



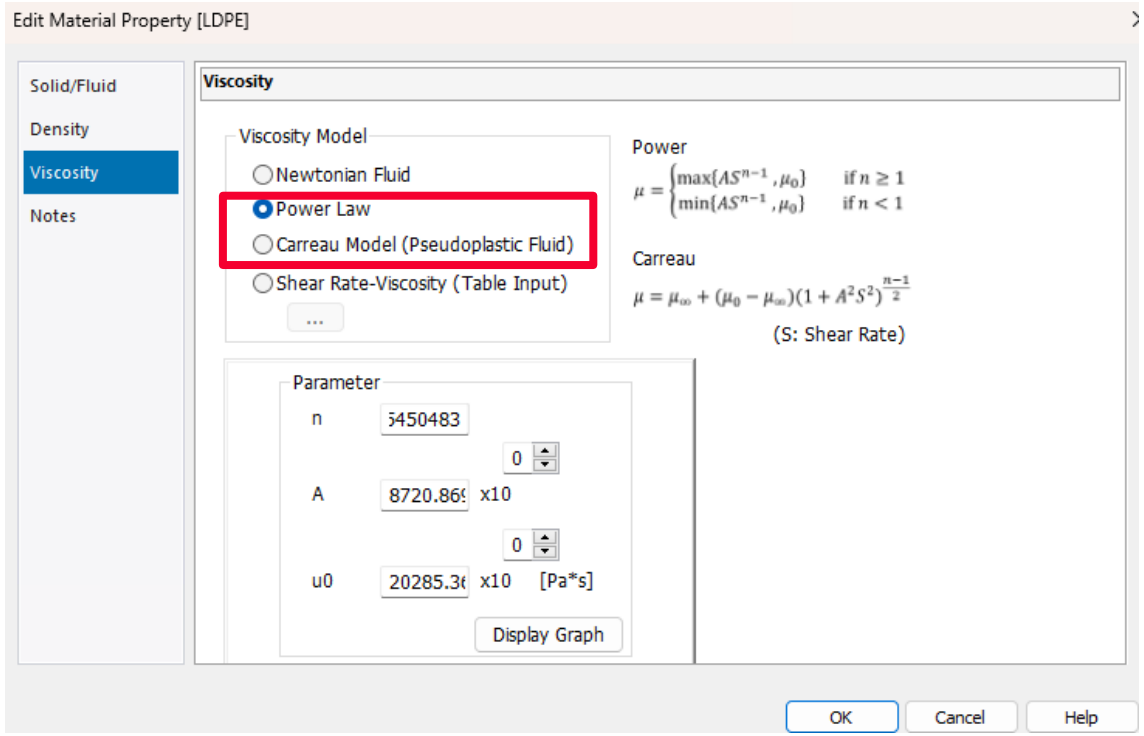
Solver – Fluid Analysis: Non-newtonian fluid

Non-newtonian fluid is supported.

> Parameter setting

NEW

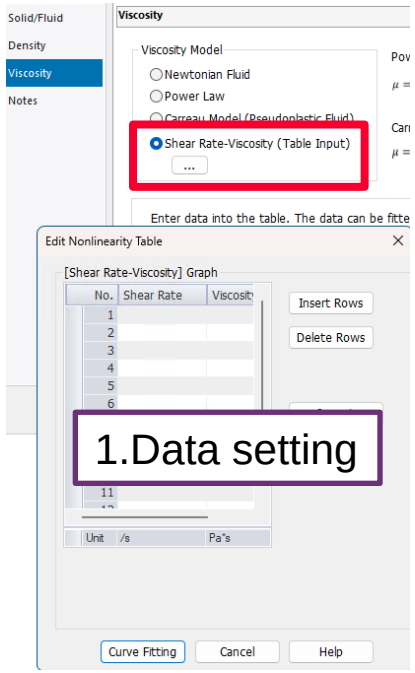
Power law mode and Carreau model are supported for non-Newtonian fluid.

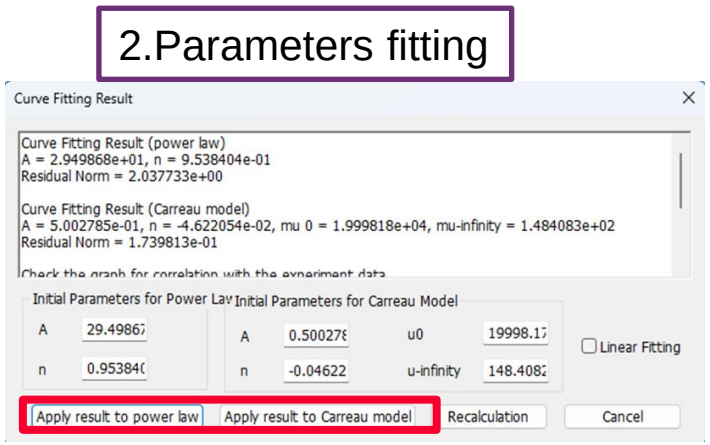


> Parameter fitting

NEW

1. **Enter the measurement data** from a rotational viscosimeter (rheometer)
2. **The parameters of the power law** or **Carreau model** can be obtained through fitting.

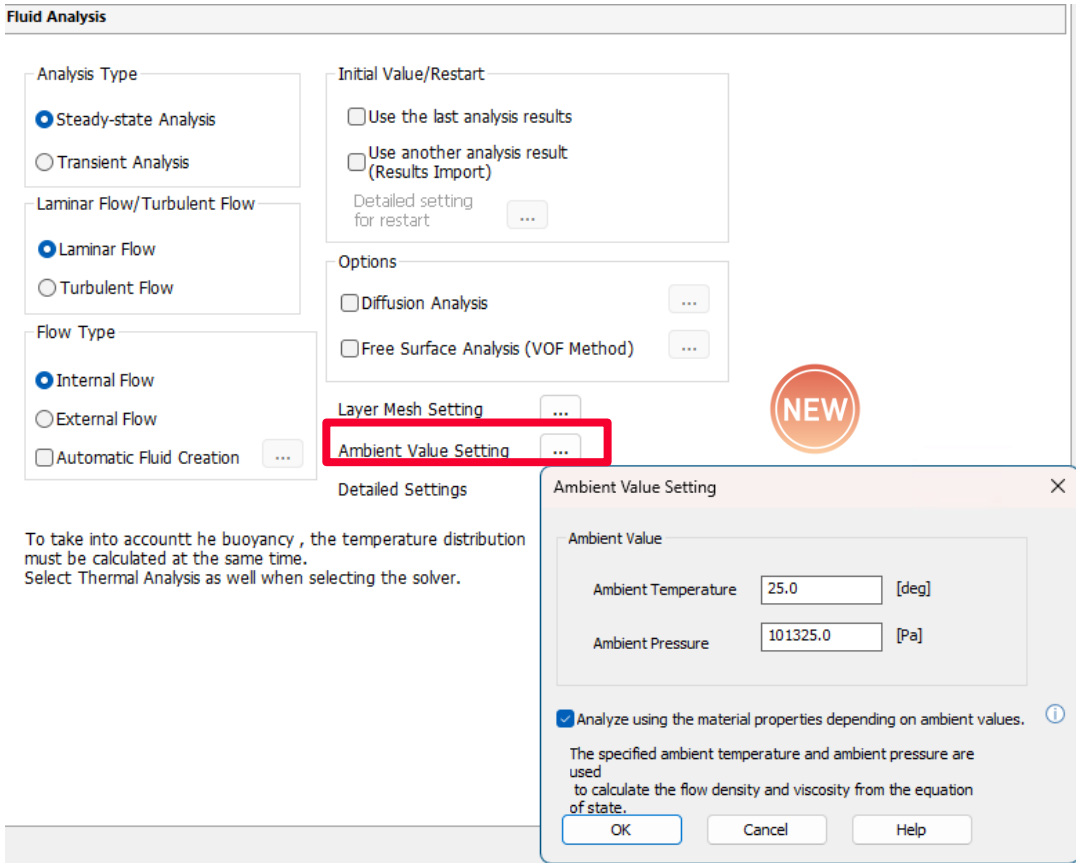




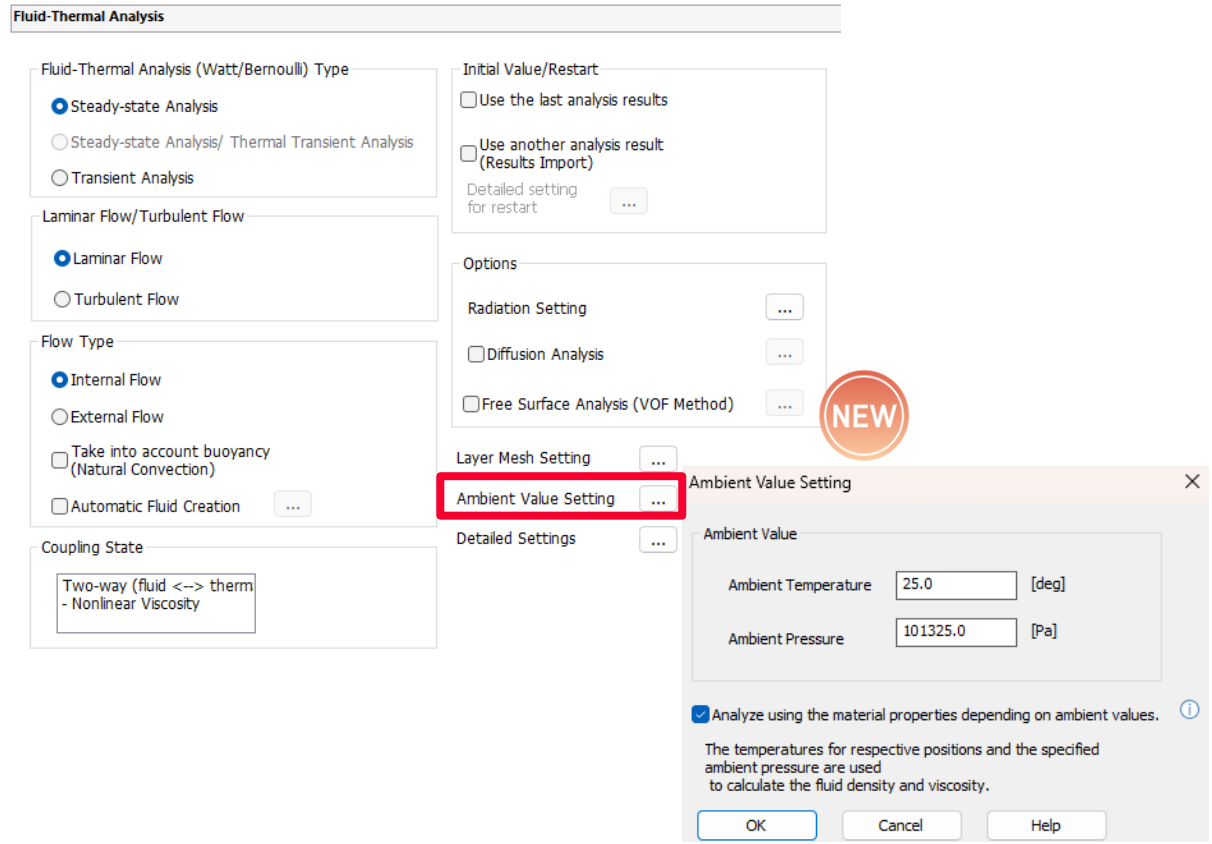
Solver – Fluid Analysis: Analysis using the material properties depending on ambient values

Analysis using the material properties depending on ambient temperature and ambient pressure.

> Fluid Analysis



> Fluid-Thermal Analysis

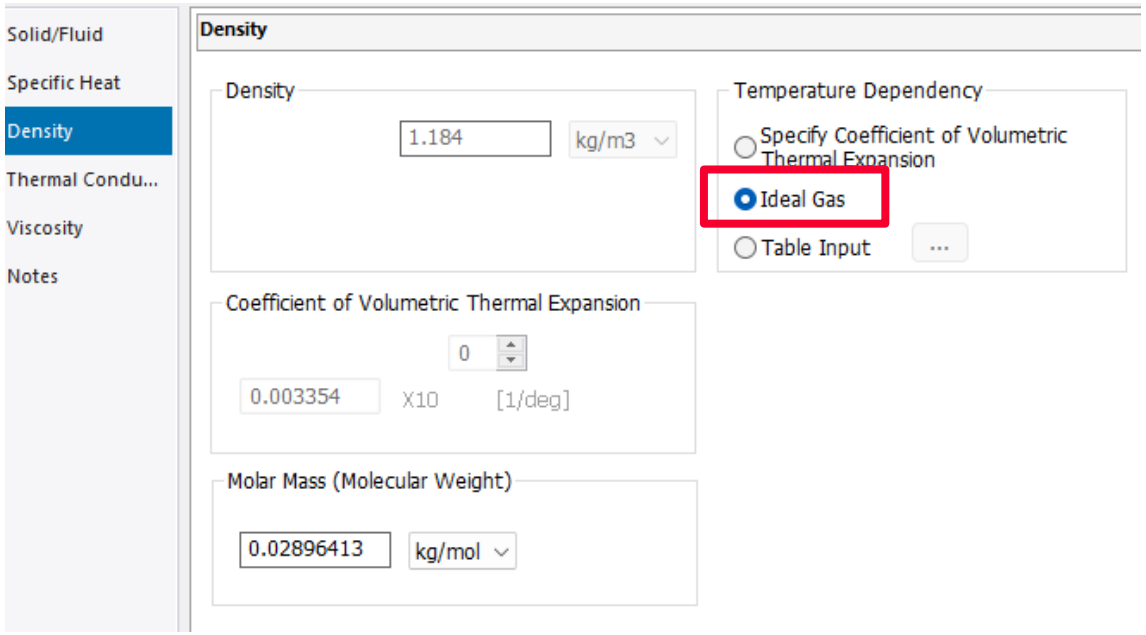


The option [Analyze using the material properties depending on ambient values] is can be selected.

Solver – Fluid Analysis: Change the methods to calculate the density and buoyancy of ideal gas for better accuracy

Change the methods to calculate the density and buoyancy of ideal gas for better accuracy.

>The Molar Mass M_w is used to calculate ideal gas.
>Reference density P_{ref} will be calculated from ambient temperature and ambient pressure.
>When the option [Analyze using the material properties depending on ambient values] is selected, the density of fluid will be calculated as ideal gas.



Density formula $\rho = \dfrac{M_w P_{ref}}{R T_{abs}}$ NEW

(P_{ref} : ambient pressure R : specific gas constant)

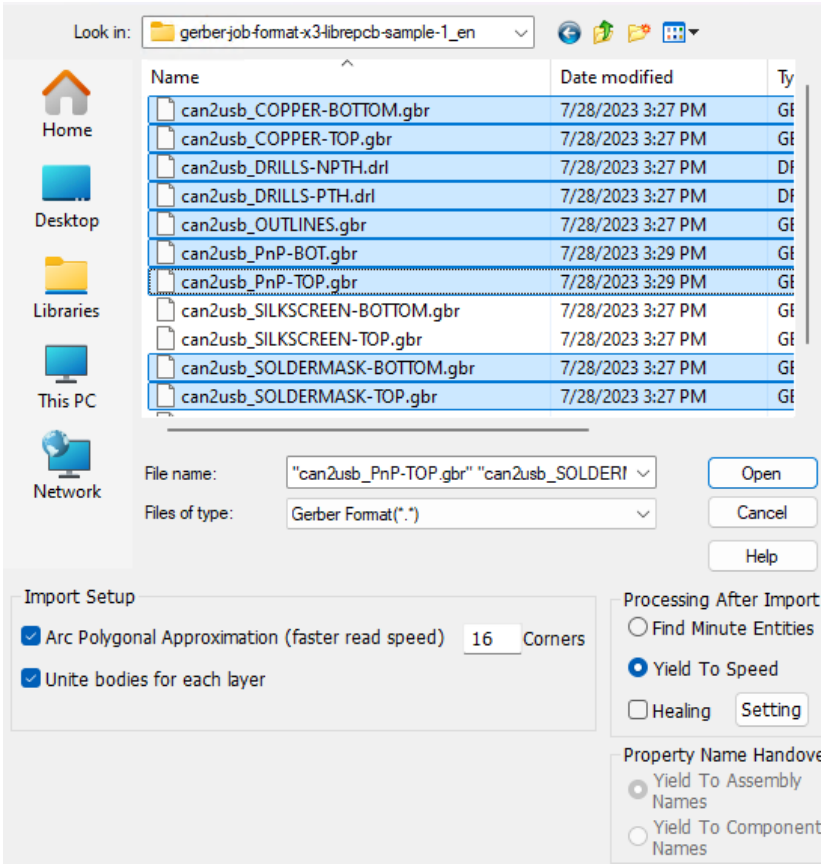
Buoyancy formula $f_b = (\rho - \rho_{ref})g$ NEW

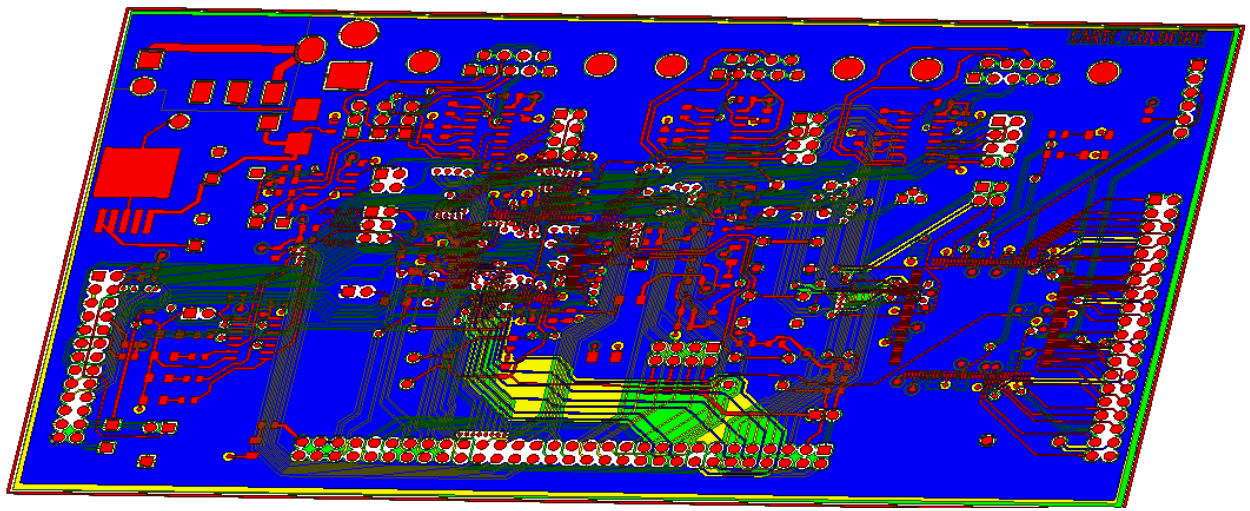
As a result of this modification, the accuracy of calculations has been improved in cases where the temperature difference between the environment and the heating element is large.

Model - Model import function of Gerber data (RS274X) and drill data

Gerber data (RS274X) and drill data (Excellon format) are supported.

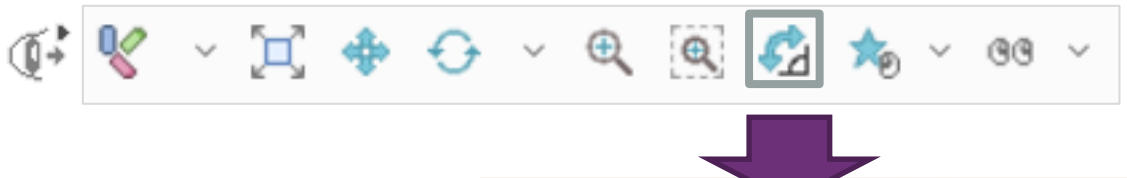
Femtet can import Gerber data (RS274X) and drill data.

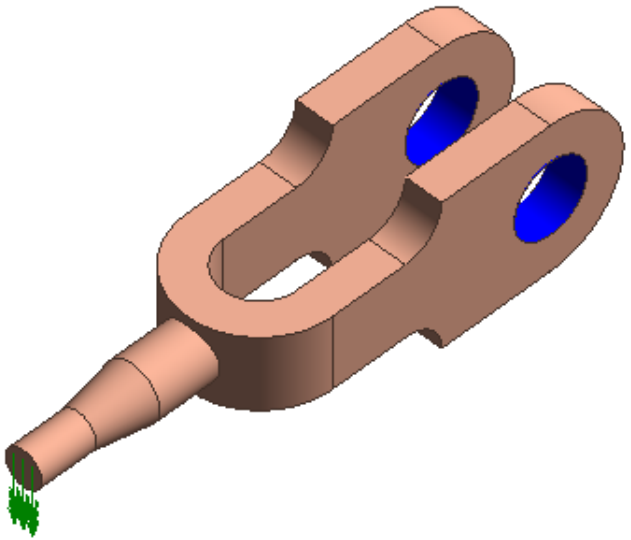


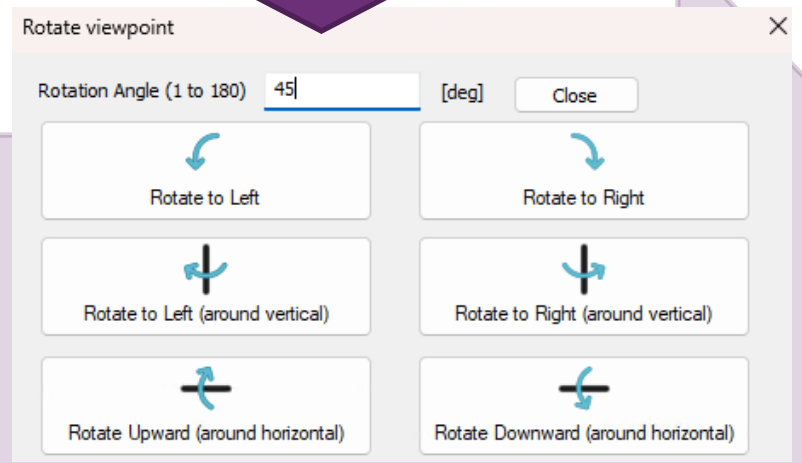


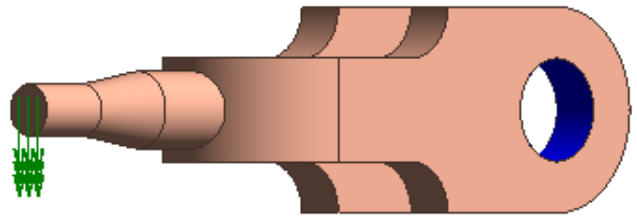
UI – New function of [Rotate viewpoint by any angle]

The [Rotate viewpoint by any angle] function is available.









Isometric view

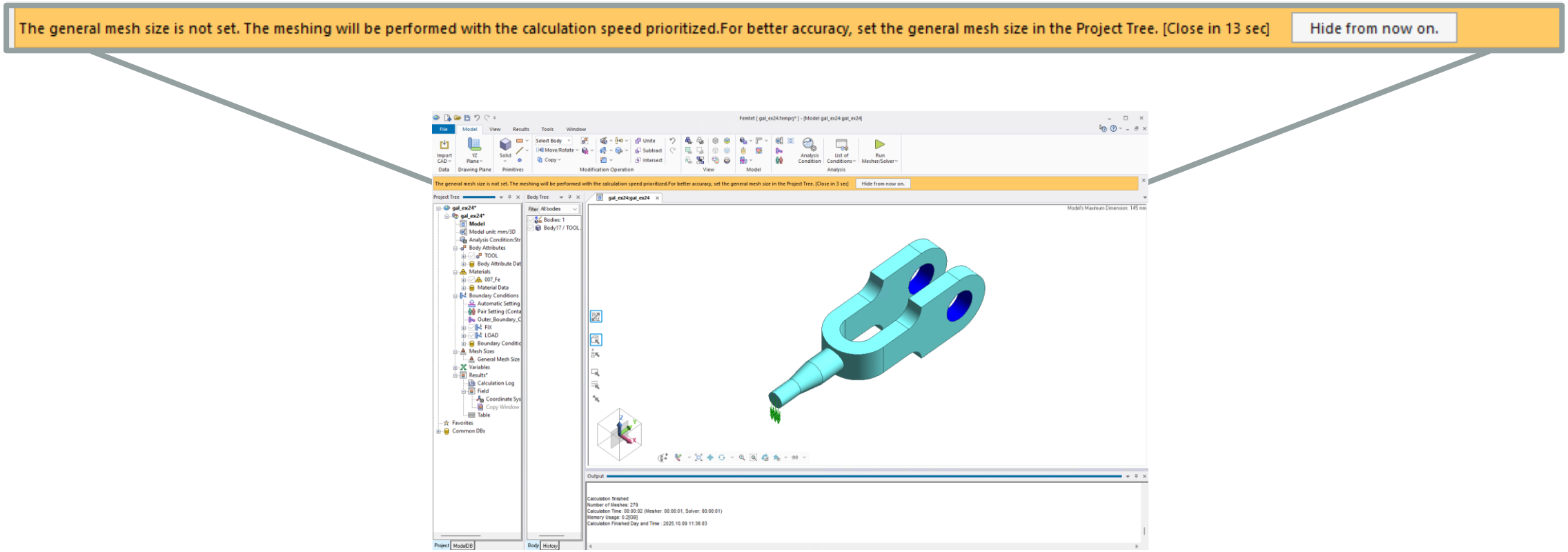
45-degree upward rotated view

View the model from different angles to easily check its shape and structure.

UI - Warning messages are displayed in a message bar

Warning messages are displayed in a message bar.

The warning message display has been changed from balloon tips to the message bar.
This improves message visibility and reduces interference with operation.



Macro - Added CFemtet.Exit function

Added the CFemtet.Exit function, which allows Femtet to be closed from a macro.

Home / CFemtet Class / CFemtet / Method / Exit

Exit Method

Syntax

Exit(bForce As Boolean) As Boolean

Return Value

True	Success
False	Failure (While unsaved files exist, If Femtet does not terminate, such as the cancel button is selected in the [Save as] dialog box.)

Parameters

bForce	In case unsaved projects exist, set whether to exit without saving all of them (yes [True], no [False]).

Femtet can now be exited via command,
enabling the development of more flexible automation processes.

Femtet Ver.2025.0　What’s New

Murata software Co., Ltd.

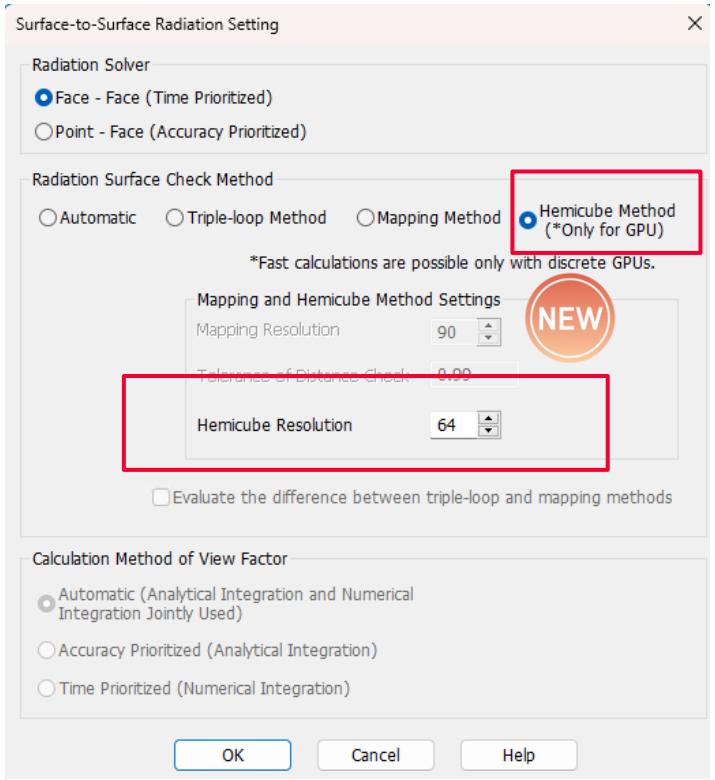
CAE Software

Table of Contents

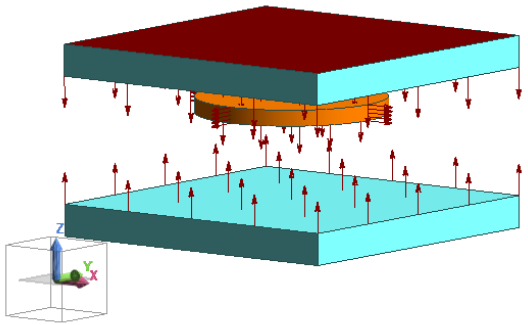
Table of Contents

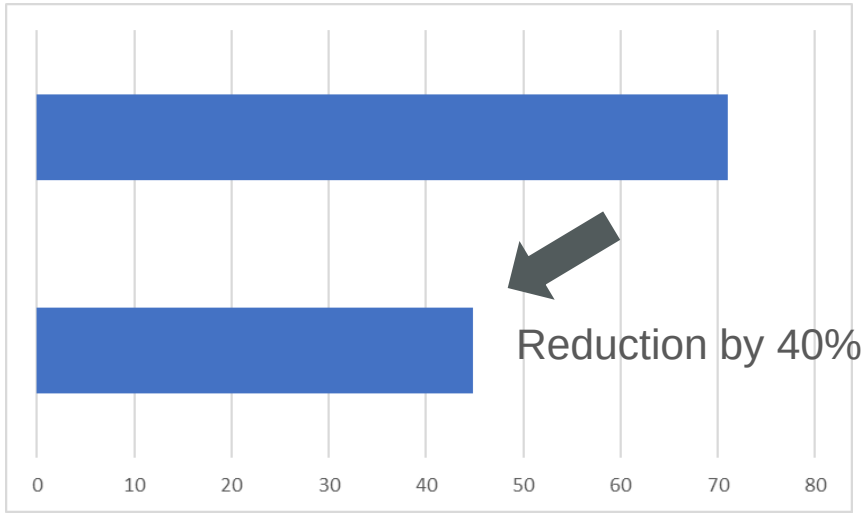
Solver – Thermal Analysis:
Speed up for surface-surface radiation by using hardware GPU

Hemicube method makes speed up for surface-surface radiation by using hardware GPU



Thermal Analysis Example 16
1200k meshes



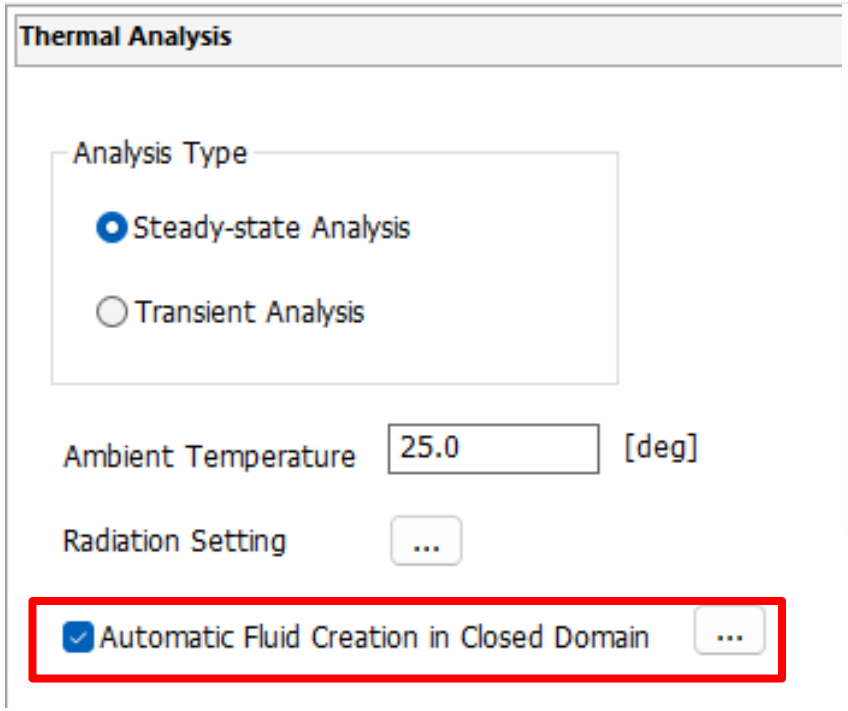


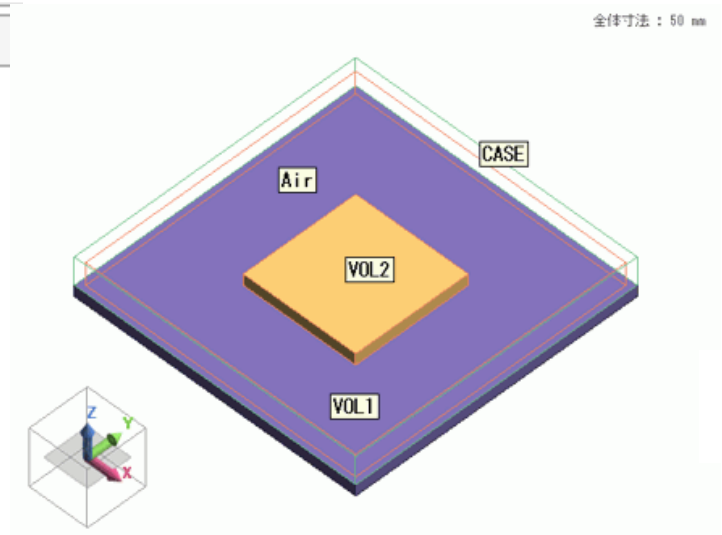
*Simulation condition
Use same resolution (90)
CPU: i7-12700
Mapping method: CPU8 cores parallel
Hemicube method: GPU T400 + CPU8 cores parallel

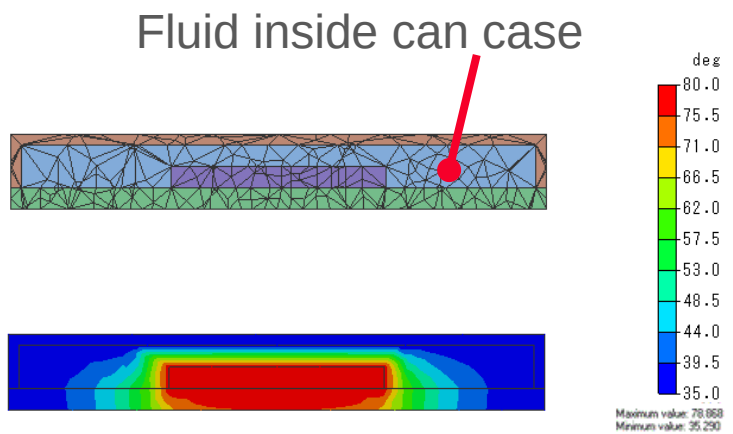
Implemented the hemicube method to accelerate radiative surface checking through GPU-based rendering.

Solver – Thermal Analysis: Automatic internal fluid creation is applied

A fluid domain is created inside the solid domain.







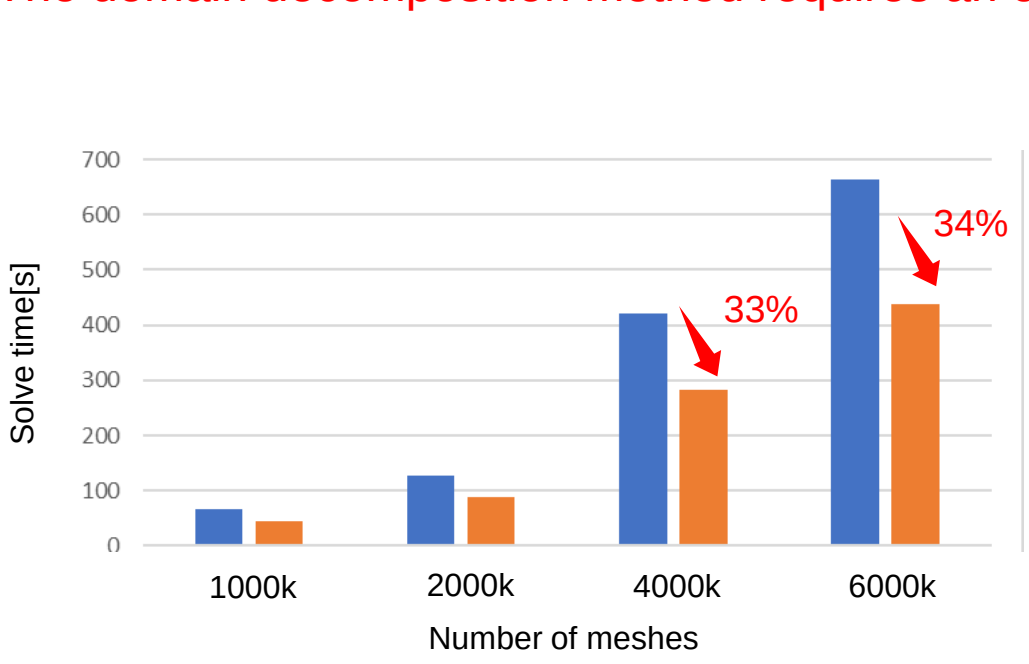
Thermal Analysis example 30:
Heat Dissipation of IC inside a Case by Natural Convection

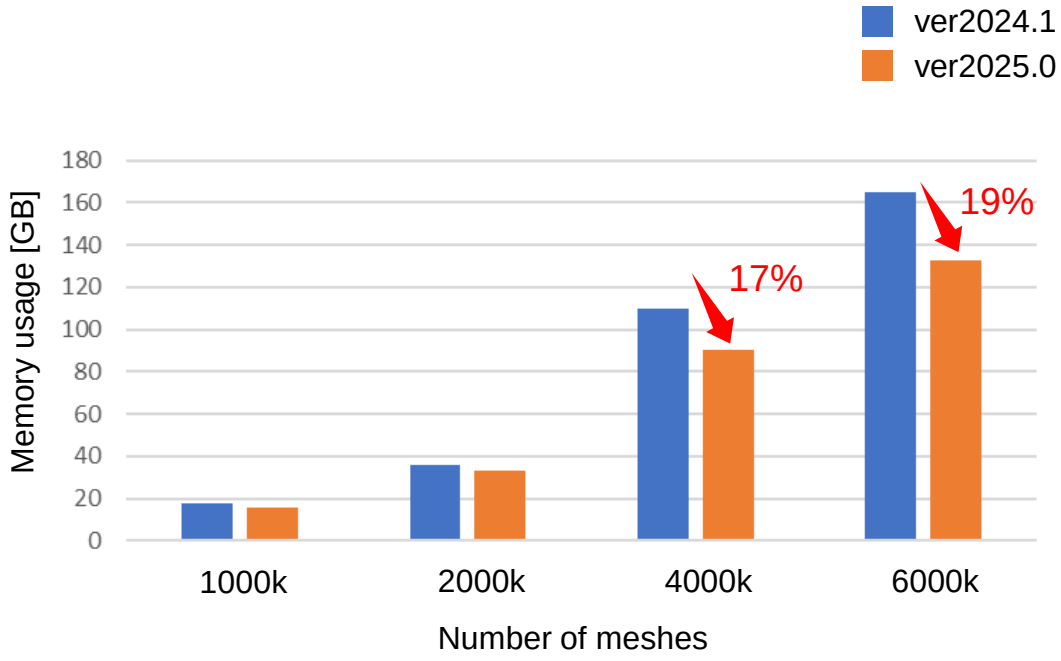
-A fluid domain is automatically created in the domain surrounded by solids (closed domain).
-The automatic setting allows you to specify the dimensions, mesh size, and fluid material.
-Heat transfer by thermal conduction in the fluid domain is taken into account (convection is not).

Solver – Stress Analysis: Fast Calculation in Static Analysis

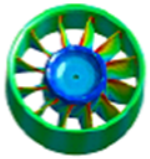
The improved domain decomposition method allows for faster calculation and less memory usage.

*The domain decomposition method requires an optional license.





Model



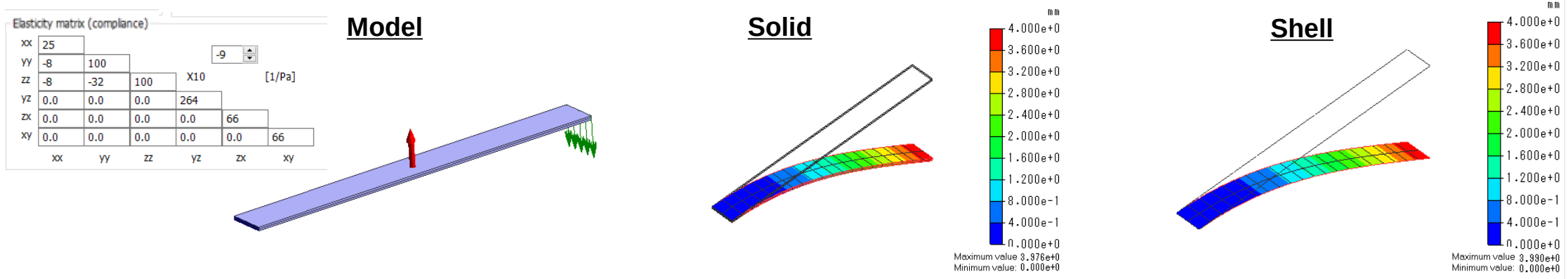
CPU : Xeon W-3375 (38 core)

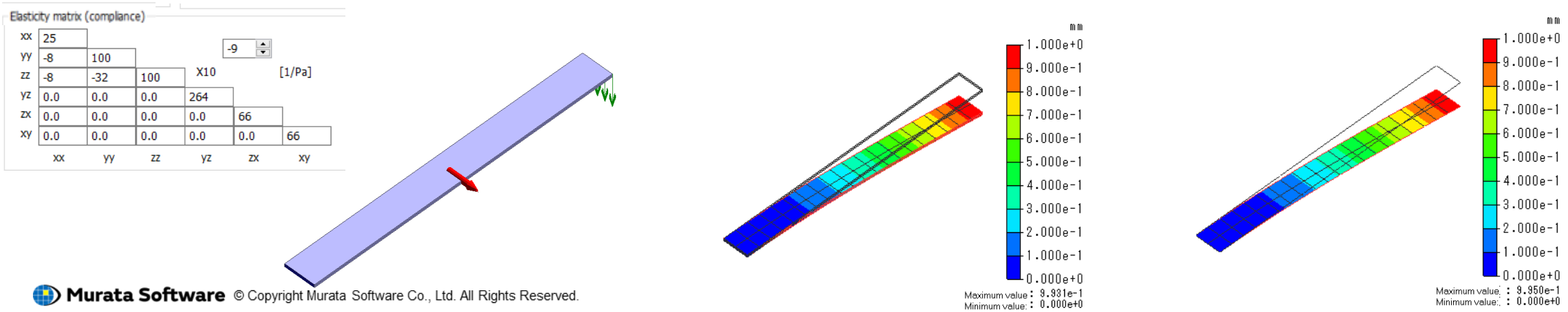
Solver – Stress Analysis: Shell element of Orthotropic Material

A shell element made of orthotropic material can be calculated.

Stress solver supports shell elements made of orthotropic materials.

The diagrams below show the calculated results of displacement when the direction of orthotropic material is changed.





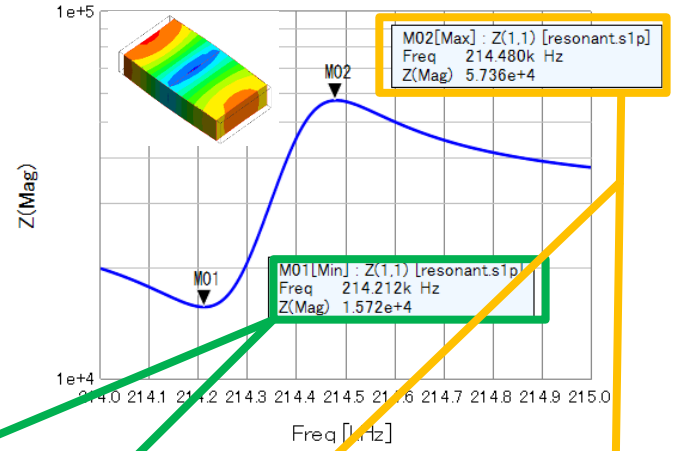
Solver – Piezoelectric Analysis: Impedance Graph is Added to Result Table

The [Impedance Graph (Resonant Analysis)] tab is added to the result table for the resonant analysis.

The result table can display the values close to the values on the impedance graph, for resonant frequency, anti-resonant frequency, and their impedances.

The values are acquired in the parametric analysis as well.

The analysis result shown here has been acquired closely based on the conditions set in Example 15: External Resistor (Piezoelectric transformer).
The accuracy of the spread between resonant and antiresonant frequencies has improved.



Table

Floating electrode potential [V]	Resonant frequency [Hz]	Convergence Judgment	Damping capacitance [F]	Free capacitance [F]	Spread between resonant and antiresonant frequencies [Hz]
Coupling coefficient[%]	Resonant resistance [ohm]	Equivalent capacitance [F]	Equivalent inductance [H]	FEM Info	Impedance Graph (Resonant analysis)

	Frequency [...	Resonant Frequency (f0) [...	R(f0)[ohm]	X(f0)[ohm]	abs(Z(f0))[o...	Antiresonant Frequency (fa) [...	R(fa)[ohm]	X(fa)[ohm]	abs(Z(fa))[o...
0: 1.768964e+05Hz	1.76896350e+5	1.76850204e+5	7.67556689e+2	-3.57313205e+4	3.57370 e+4	1.76944 e+5	6.60017515e+2	-3.69898290e+4	3.6995 169e+4
1: 2.142689e+05Hz	2.14268885e+5	2.14211051e+5	1.96947835e+3	-1.29062403e+4	1.57169520e+4	2.14480865e+5	1.27333931e+4	-4.71002613e+4	5.73577339e+4

Comparison with the previous method

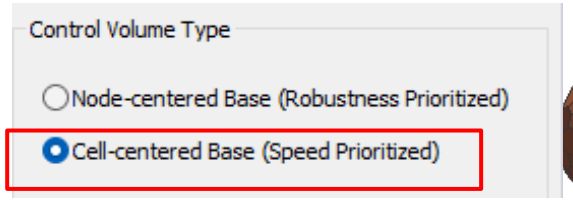
-For the analysis result above, the graph shows the spread between resonant and antiresonant frequencies as 214480 214212=268 [Hz].
-The previous result table displays a value of 154 [Hz] on the [Spread between Resonant and Antiresonatn Frequencies] tab, which differs from the values on the graph. The updated method has calculated the value to be 270 [Hz], which is closer to the values on the graph.
-Note that the previous method works well for smaller losses or the fundamental mode.

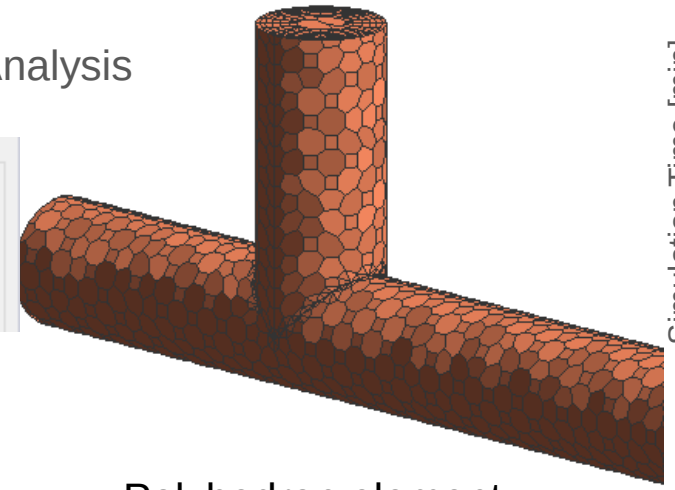
Solver – Fluid Analysis: Cell-centered Solve Is Available in Fluid/Fluid-Thermal Analysis

Cell-centered solver is available in fluid analysis for faster calculation.

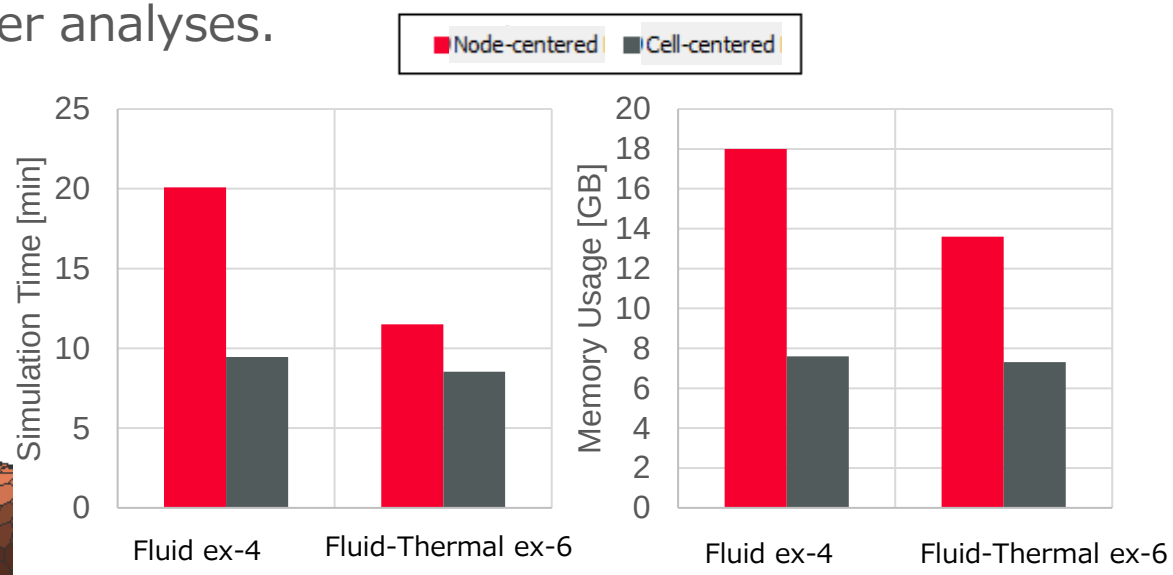
* In the version earlier than 2025.0, cell-centered solver is available only in the VOF method analysis. In version 2025.0, it is available in other analyses.

Use [Detailed setting] of Fluid Analysis





Polyhedron element



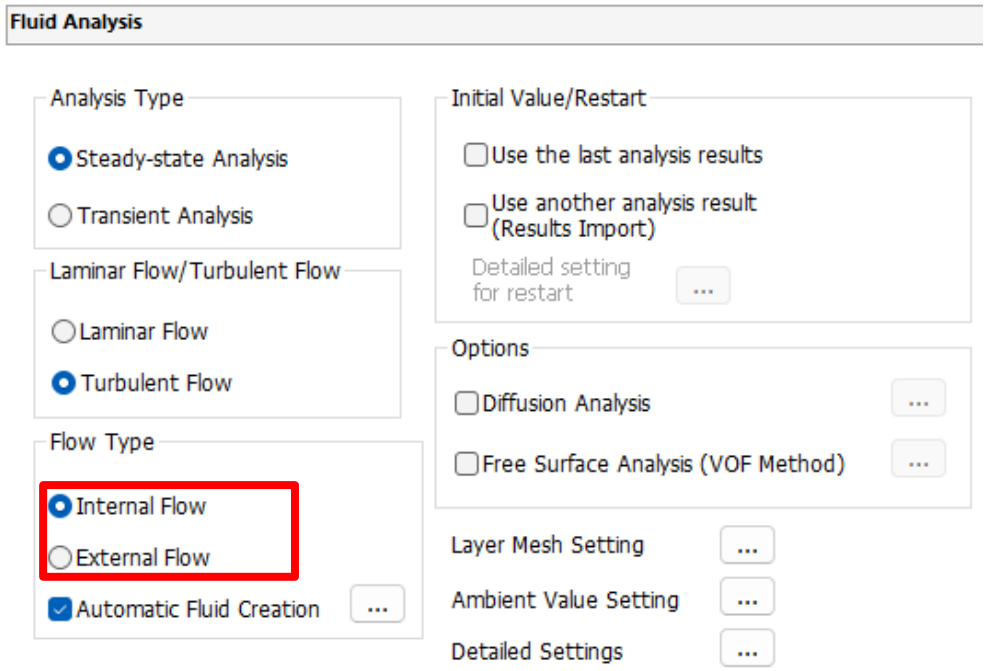
Time Memory

Compares between node-centered and cell-centered for 1000k elements

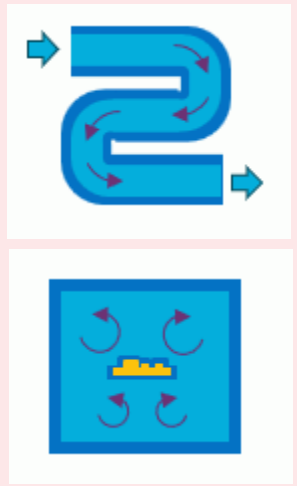
-Advantages: Faster calculation and less memory usage than node-centered solver
-Disadvantages: Less robustness than node-centered solver (calculations may not converge).
-Triangular or tetrahedral free mesh is automatically changed to a polygon or polyhedron for analysis.
-More robustness is achieved in the VOF method analysis.

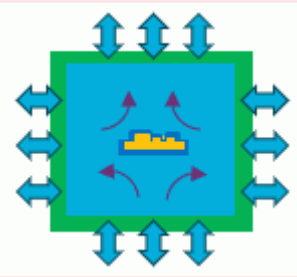
Solver – Fluid Analysis: Automatic Setting of Outer Boundary Condition

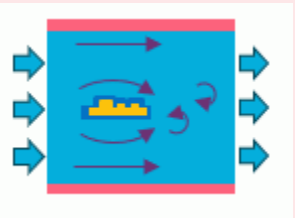
Outer boundary condition is automatically set based on the specified flow type.



Internal flow	External flow [normal]	External flow [One-way Forced Convection]
Solid wall	Inlet/Outlet	Slip wall





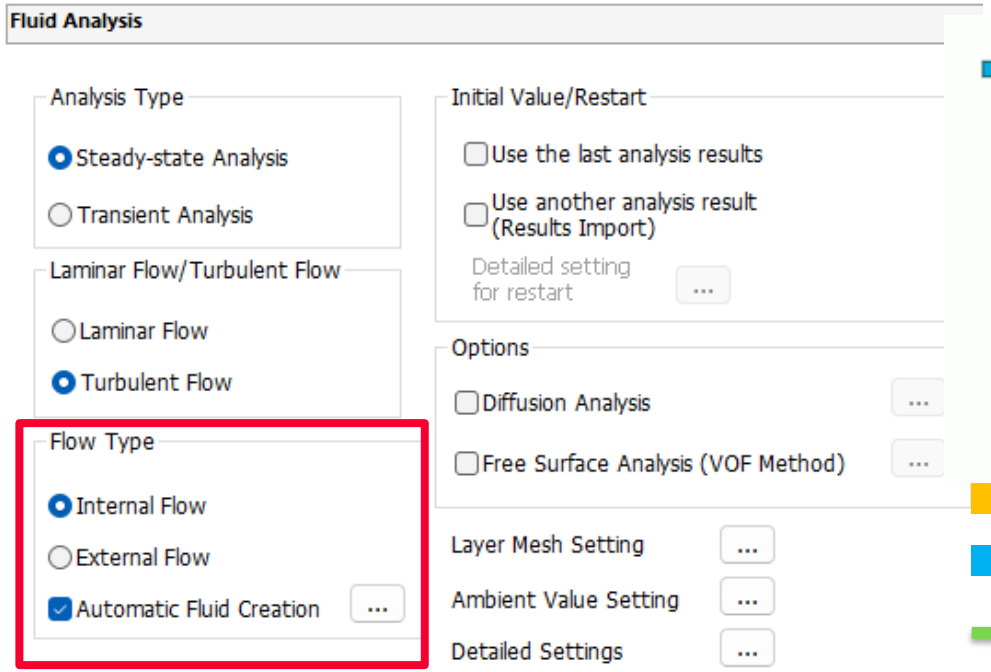


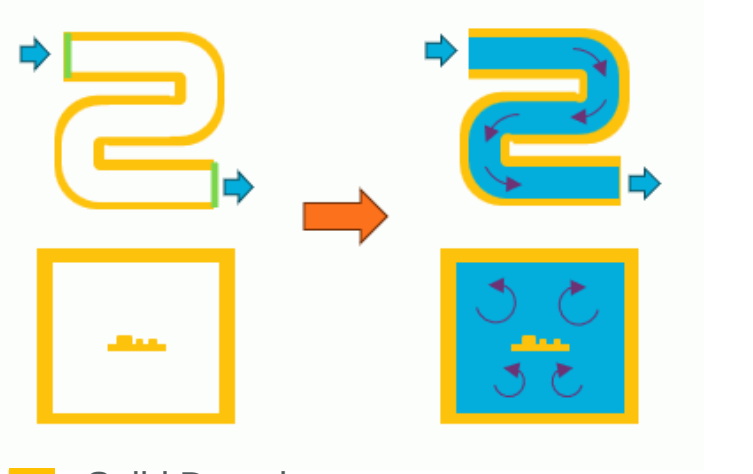
Solid Fluid

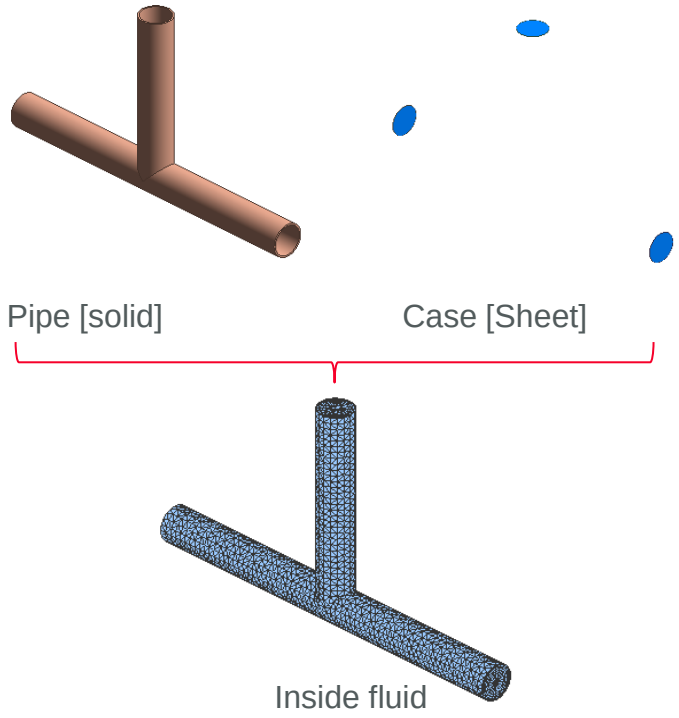
-Opening and editing the outer boundary condition set-up menu can be skipped.
-Only when the flow type is external and the boundary condition is one-way forced convection, the outer boundary condition is changed to [Slip Wall].

Solver – Fluid Analysis: Automatic Fluid Domain Creation (using Internal Flow)

Outer boundary condition is automatically set based on the specified flow type.





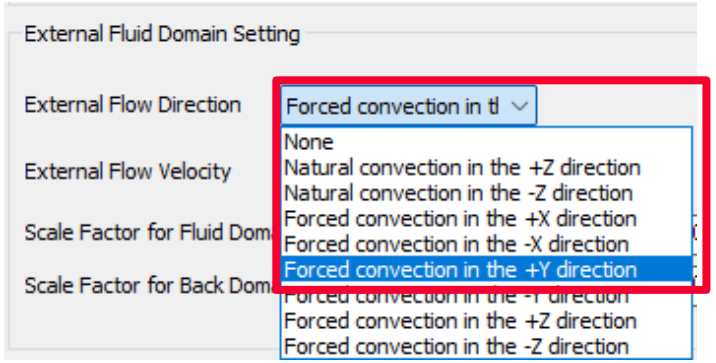


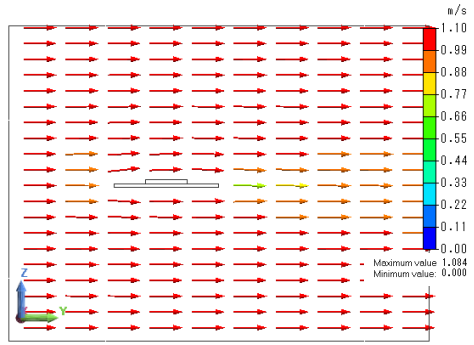
EX: Tutorial example-2

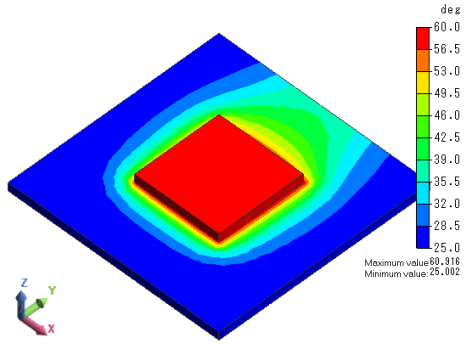
-A fluid domain is automatically created based on whether the internal or external flow is selected.
-A fluid domain is automatically created in the domain surrounded by solids.
-Even if there is an opening, a lid is created as a sheet body, and a fluid boundary condition is set on it to form the fluid domain.
-To create a lid, it is recommended to select an edge of the opening and then select the conversion option from the right-click menu (a new feature of the modeler).
-The automatic setting allows you to specify the dimensions, mesh size, and fluid material.

Solver – Fluid Analysis: Automatic Fluid Domain Creation (using External Flow)

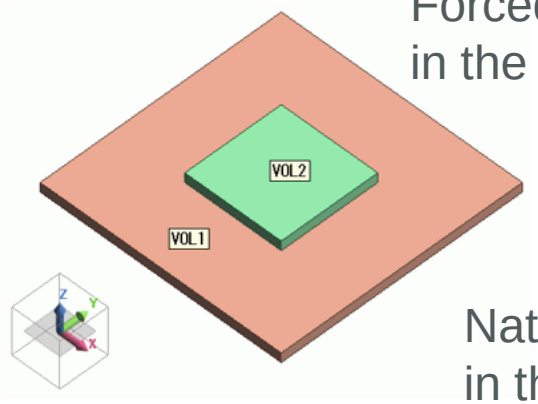
Outer boundary condition is automatically set based on the specified flow type.







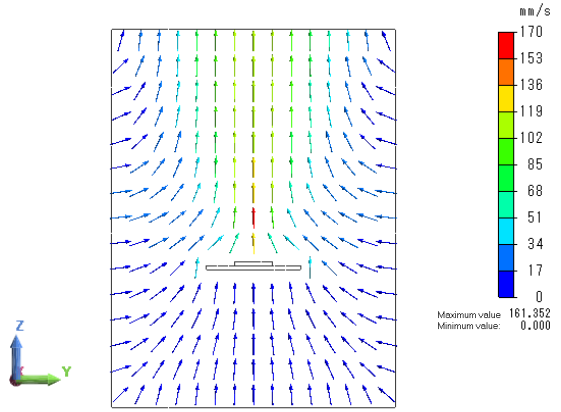
Fluid-Thermal EX-3：Forced convection in the +Y direction 1m/s

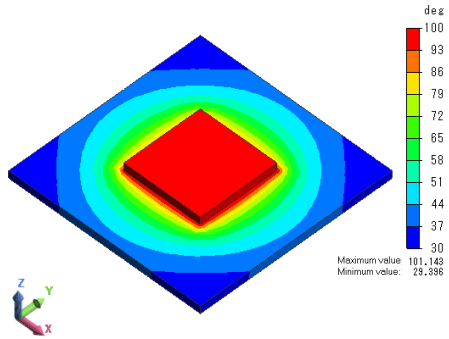


Forced convection in the +Y

Natural conv in the +X

EX: Substrate + IC





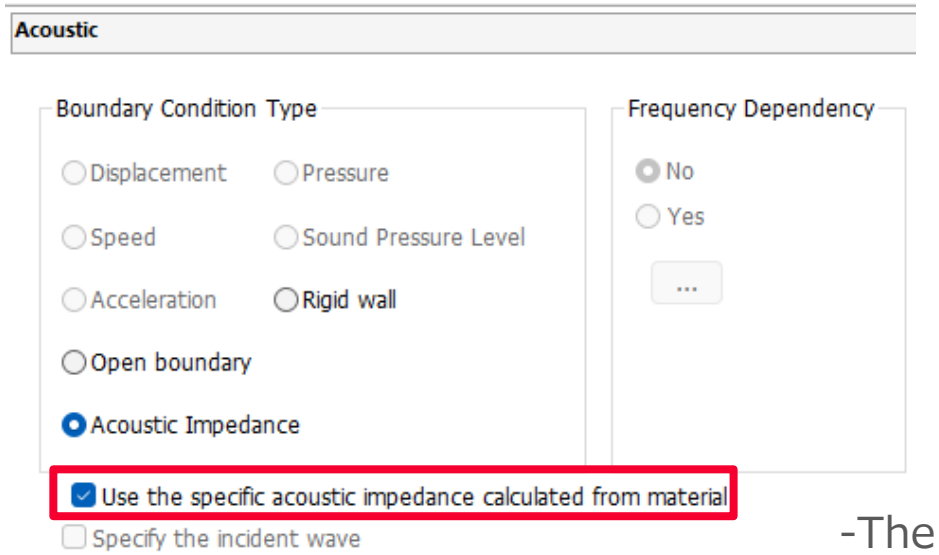
Fluid-Thermal EX-6: Normal convection in the +Z direction

-A fluid domain is created surrounding the solid.
-The automatic setting allows you to specify the dimensions, mesh size, fluid material, external flow direction, and velocity of forced convection.

Solver – Acoustic Analysis: Specific Acoustic Impedance Boundary Condition

The specific acoustic impedance of the medium can be set as a boundary condition.

The specific acoustic impedance can be set by selecting an option, without entering numeric values.



-The specific acoustic impedance, also called characteristic impedance, is the ratio of the sound pressure of a plane wave to the particle velocity.

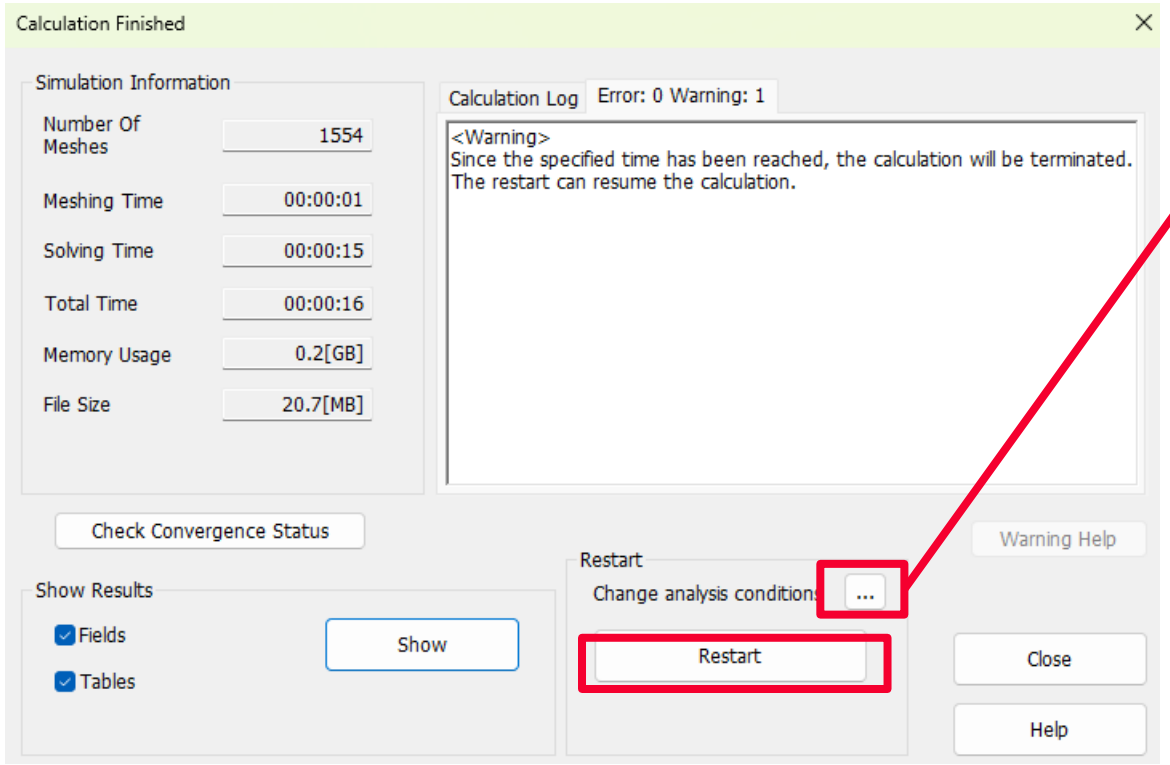
-It is given from the equation below using the material properties of medium:

Specific acoustic impedance = Density * Sound speed

* The harmonic analysis is supported, but the transient analysis is not.

Solver – Simple Restart

Restarting is improved for simple use.



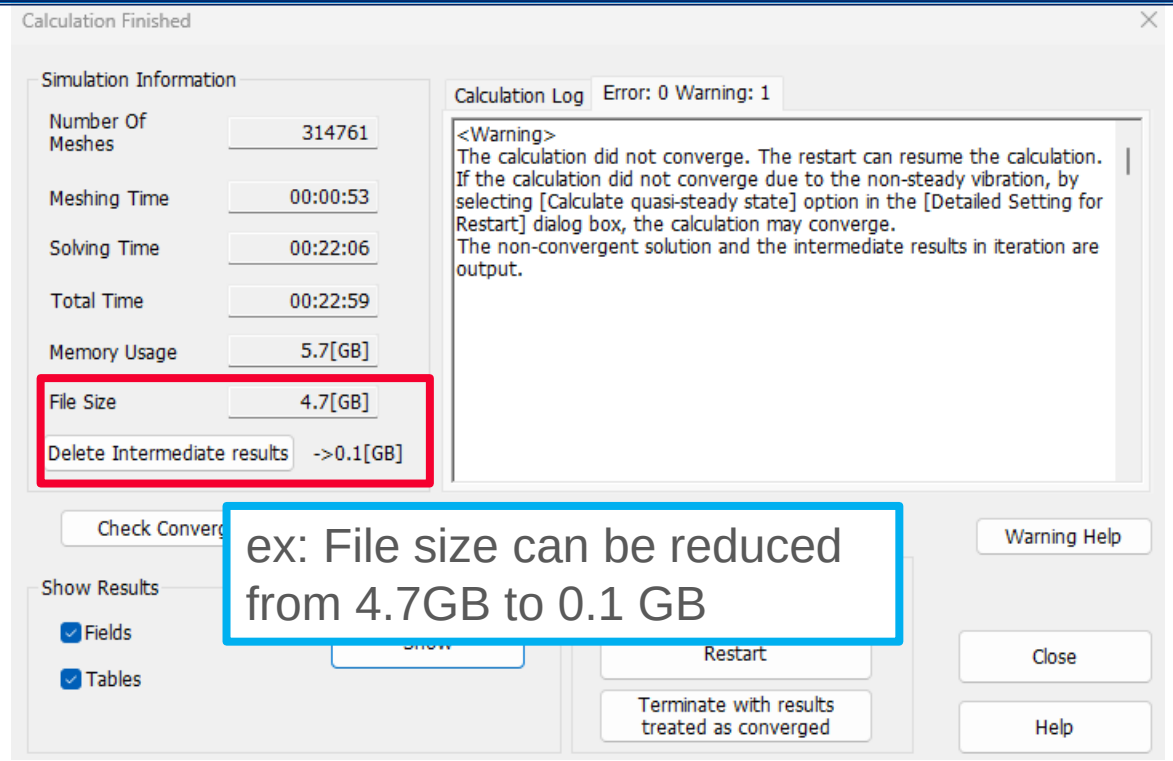
Before restarting, clicking this button allows you to change the analysis conditions.

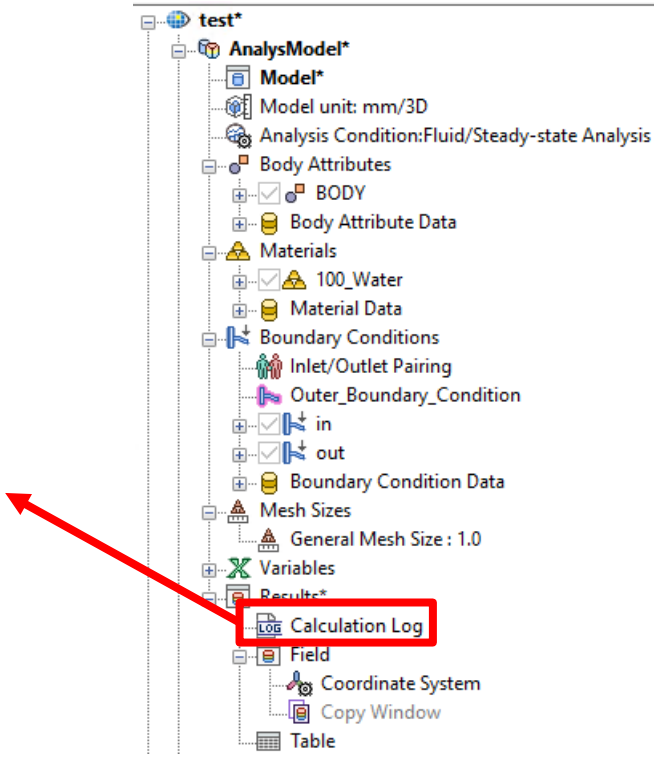
Examples:
Convergence judgment
Relaxing factors in fluid analysis
Suspended step in stress analysis
Timestep in transient analysis

-Analysis can be restarted in the [Calculation Finished] dialog box.
-Analysis can be restarted after such analysis conditions as the relaxing factor in the fluid analysis and the convergence judgement are changed.
-Applicable in transient analysis, fluid steady-state analysis, multiple steps/multi-step thermal load analysis.

Solver - Delete Intermediate Results

To solve the problem with a larger file size due to intermediate results, a new function can easily delete the results.



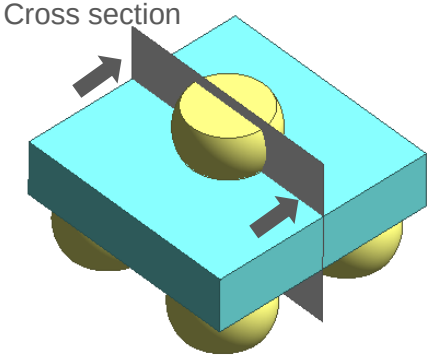


-In fluid analysis or stress analysis, intermediate results are saved for analyzing the cause in case of non-convergence/divergence.
-Since the size of the result file can be too large for calculation, you can delete the intermediate results after the analysis is complete
-After the result is saved, you can open the [Calculation Finished] dialog from [Calculation Log] on the project tree and delete the intermediate results.

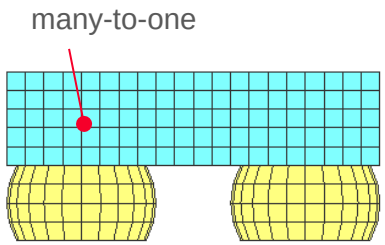
Mesher: Sweep Mesh Expanded to Many-to-Many

Sweep meshing now supports many-to-many models.

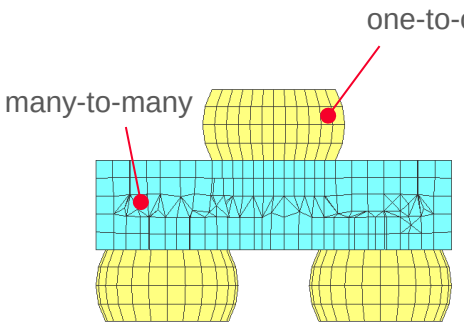
Cross section



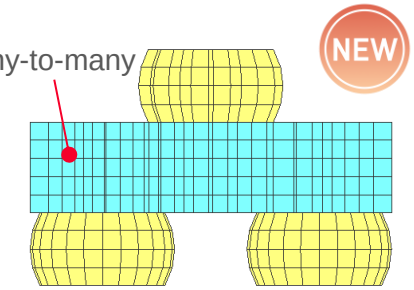
-Sweep mesh divides a face into tetrahedral elements and sweeps the face in parallel or rotationally to create hexahedral elements.
-Sweep mesh can create well-shaped hexahedral elements, but it is applicable only for shapes that can sweep.



If there is no body above, a conventional many-to-one relationship is formed, allowing the use of sweep meshing.

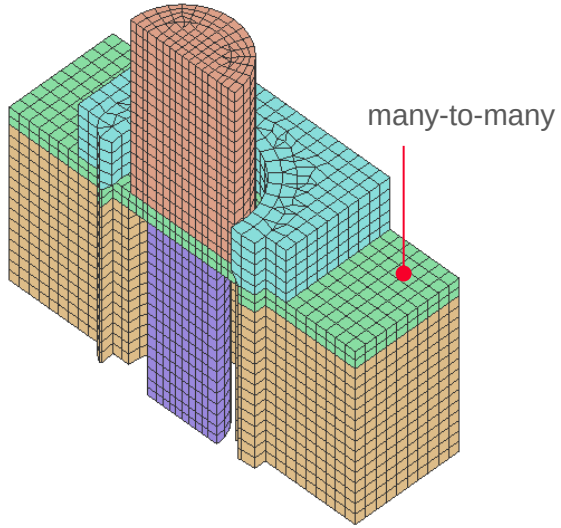


If there is a body above, a many-to-many relationship is formed, which prevents the application of sweep meshing and results in poor free meshes.



From Ver. 2025.0, sweep meshing is applicable even when a many-to-many relationship is present

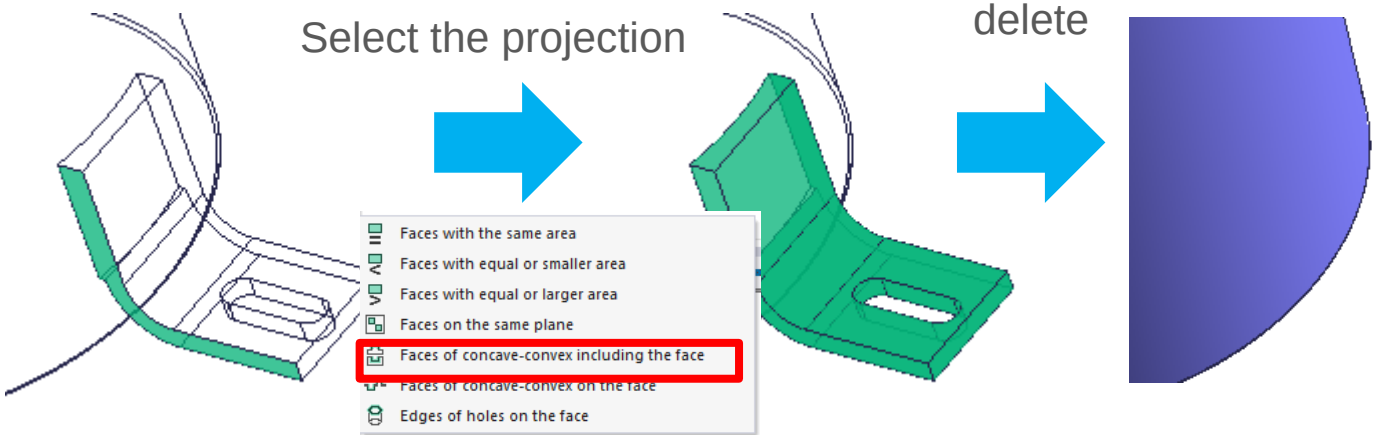
Example: Punching Die Mode

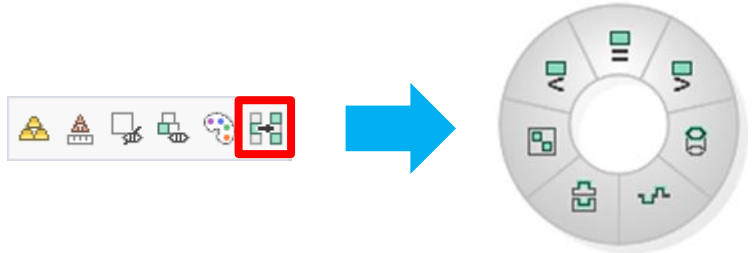


Model: Selecting function has been extended

Apply for extending the objects to be selected according to conditions.

Procedure example





Body

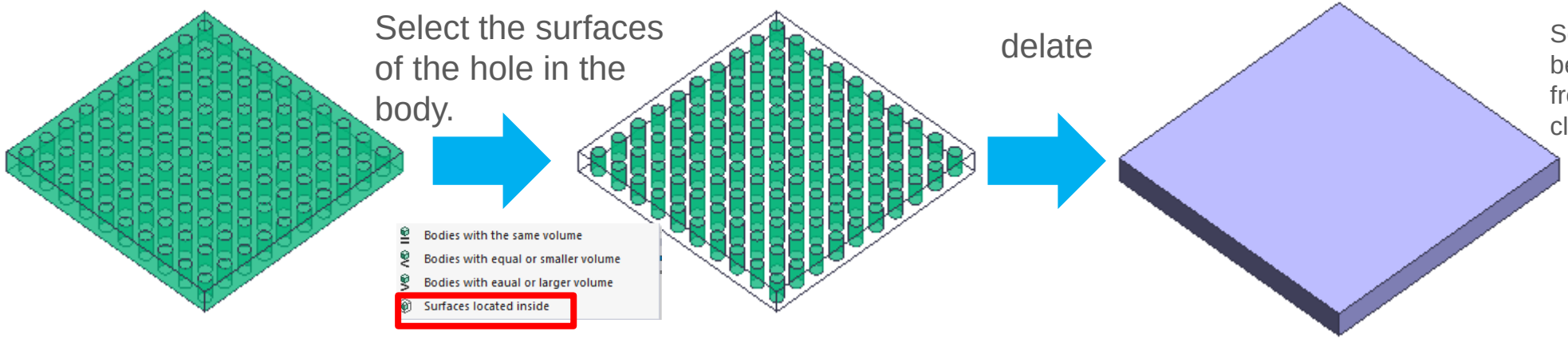
Bodies with the same volume
Bodies with equal or smaller volume
Bodies with eaual or larger volume
Surfaces located inside

Surface

Faces with the same area
Faces with equal or smaller area
Faces with equal or larger area
Faces on the same plane
Faces of concave-convex including the face
Faces of concave-convex on the face
Edges of holes on the face

Edge

Edges with the same length
Equal or shorter edges
Equal or longer edges
Edges included in a loop on the plane



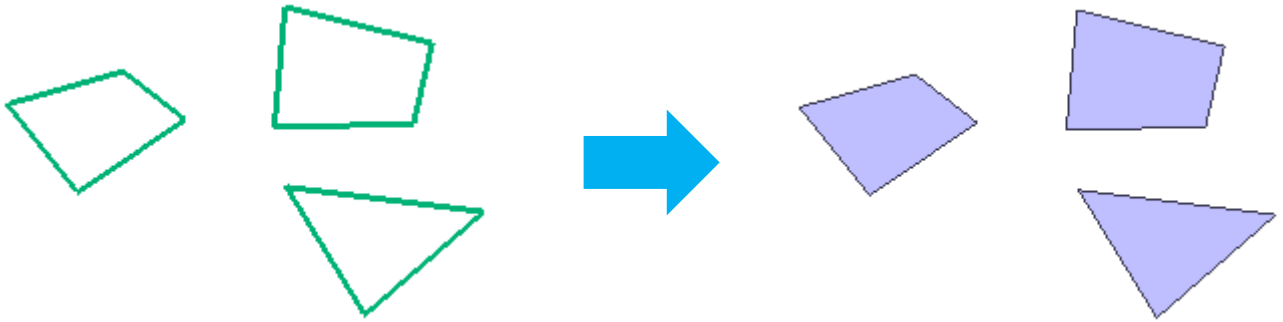
Select a body, face, or edge before selecting the functions from the mini toolbar or right-click menu.

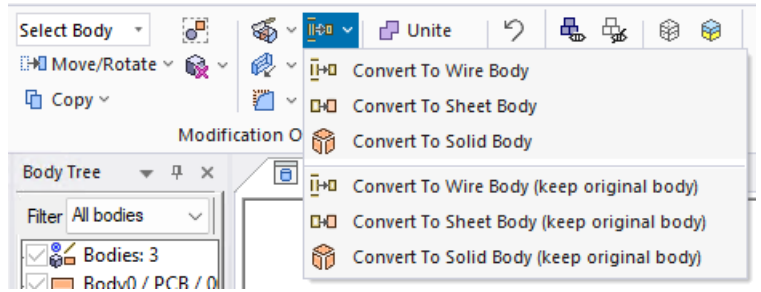
Model: Convert Multiple Bodies Collectively

Wire/Sheet/Solid body converting has been applied to multiple bodies.

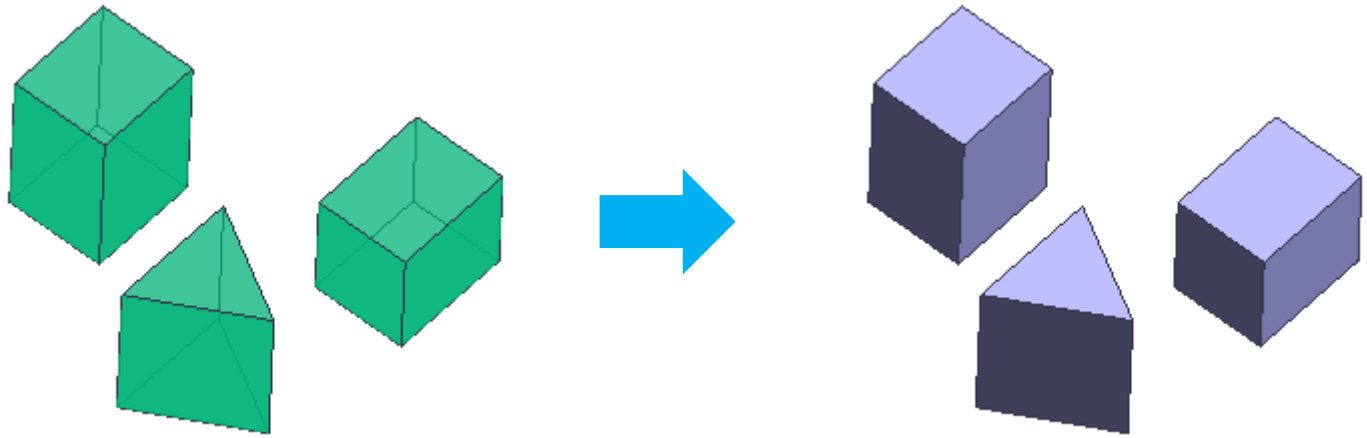
Multiple bodies can be converted collectively, or individually as previously done.

Conversion to Sheet Bodies





Conversion to Solid Bodies

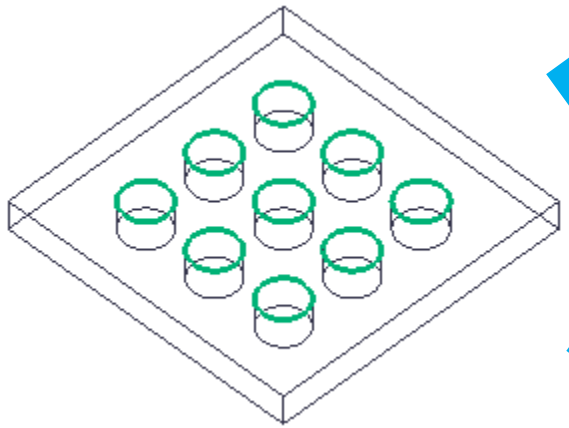


Model: [Convert] menu for topology to convert automatically.

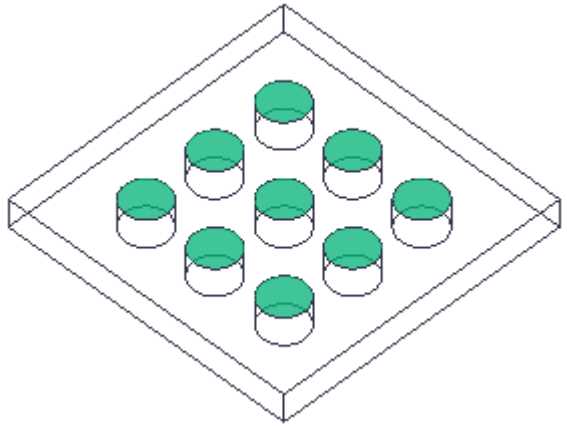
Added Convert to the right-click menu, allowing automatic duplication and conversion of topology.

Example of Surface Conversion

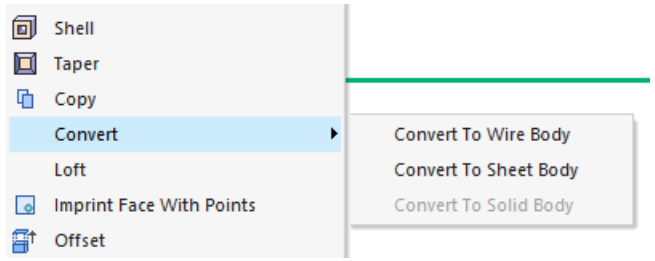
Femtet2025.0 procedure



Select Edges

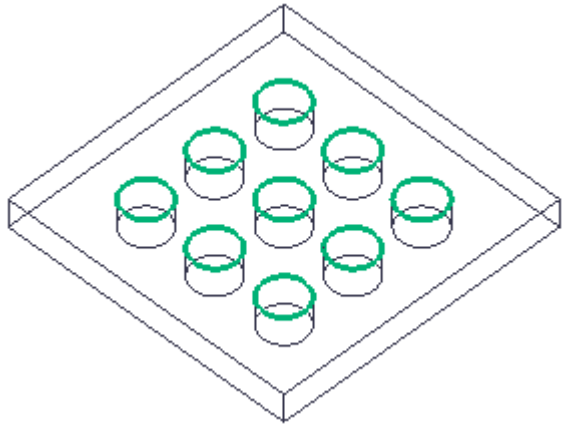


Convert to Sheet bodies

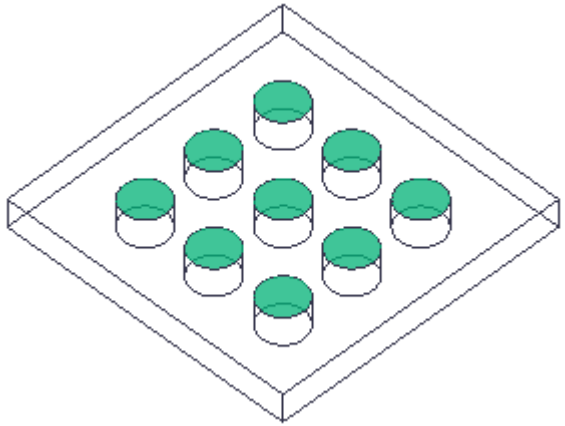


In Femtet 2025.0, Convert can be executed from the right-click menu when a topology is selected.
Note: Execution from the ribbon menu is not supported.

Previous procedure



Duplicate edges -> Create wire bodies



Convert to Sheet bodies

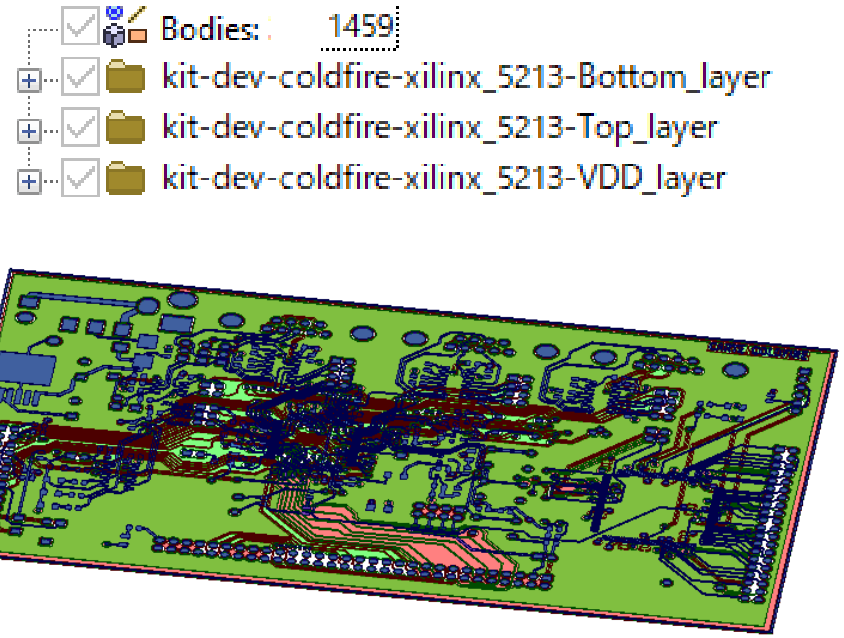
Model: Model export function of Gerber data (RS274X)

Added model export function of Gerber data (RS274X)

Femtet can export Gerber data (RS274X).

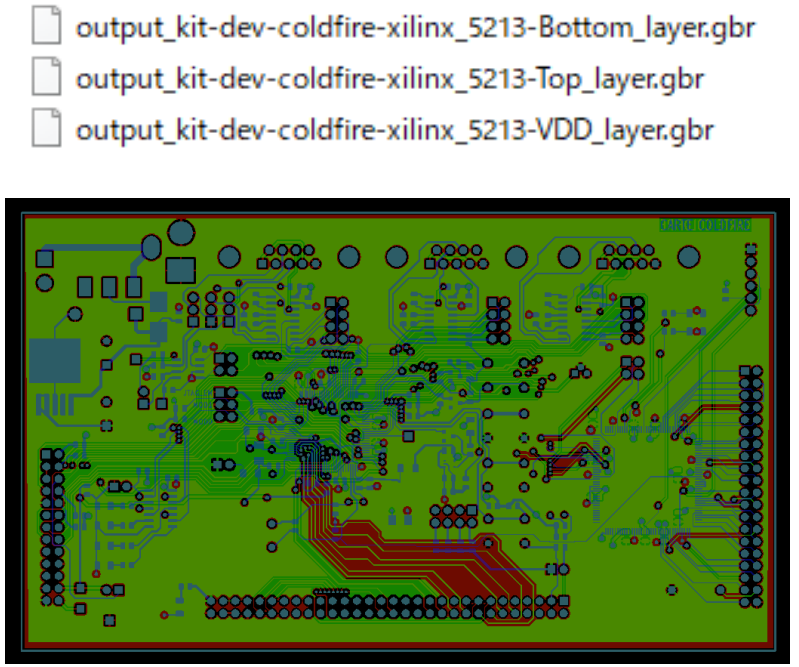
Classifying bodies into groups causes each group to be recognized as a layer, with the Gerber data split and output accordingly.

Femtet Model



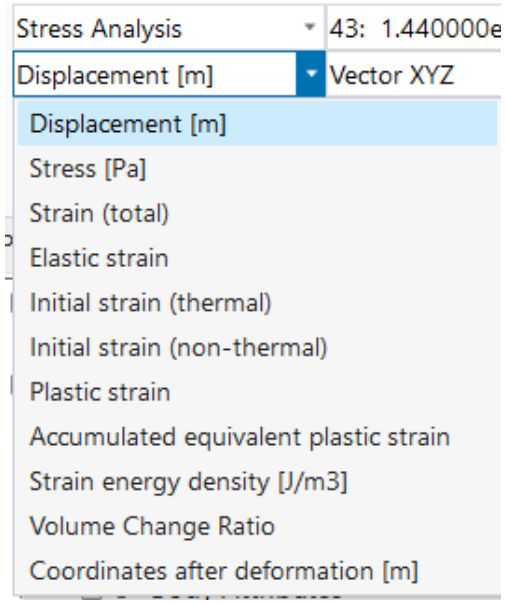
Export

Gerber data files

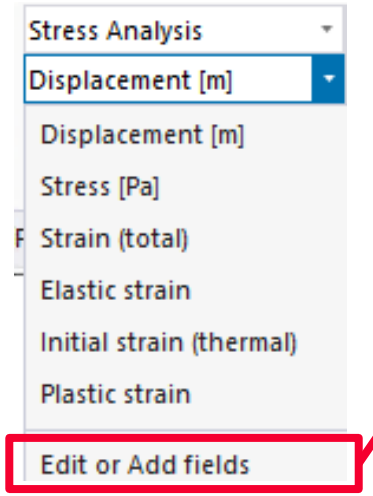


Result display: Edit or Add Fields

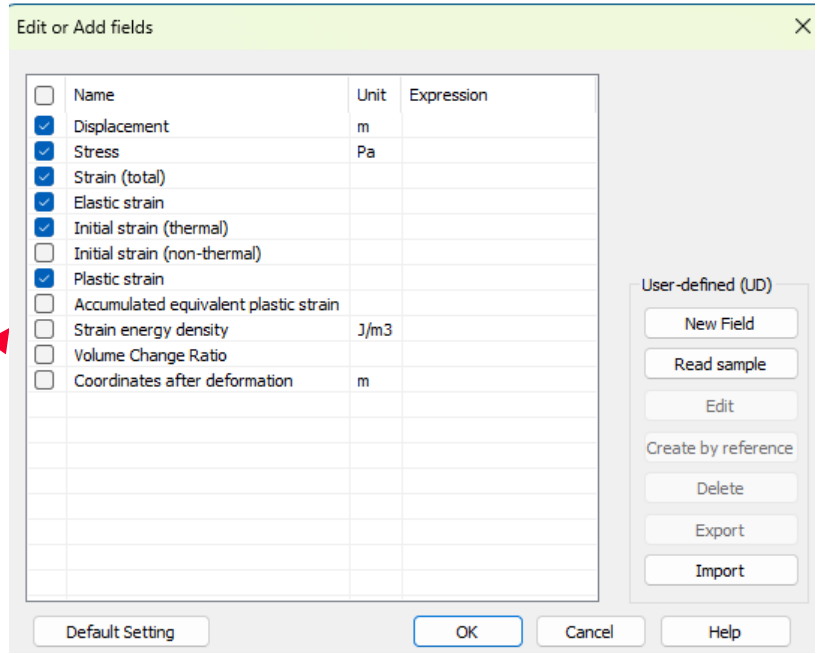
Allow for editing or adding fields to be displayed.



Initial Display
in Ver 2024

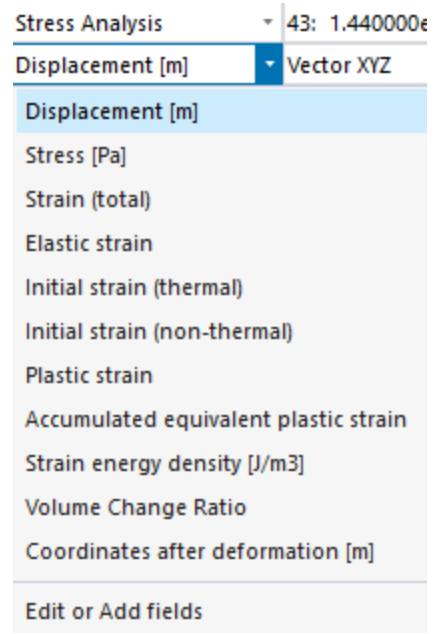


Initial Display
in Ver 2025



Field Editing or adding dialog box

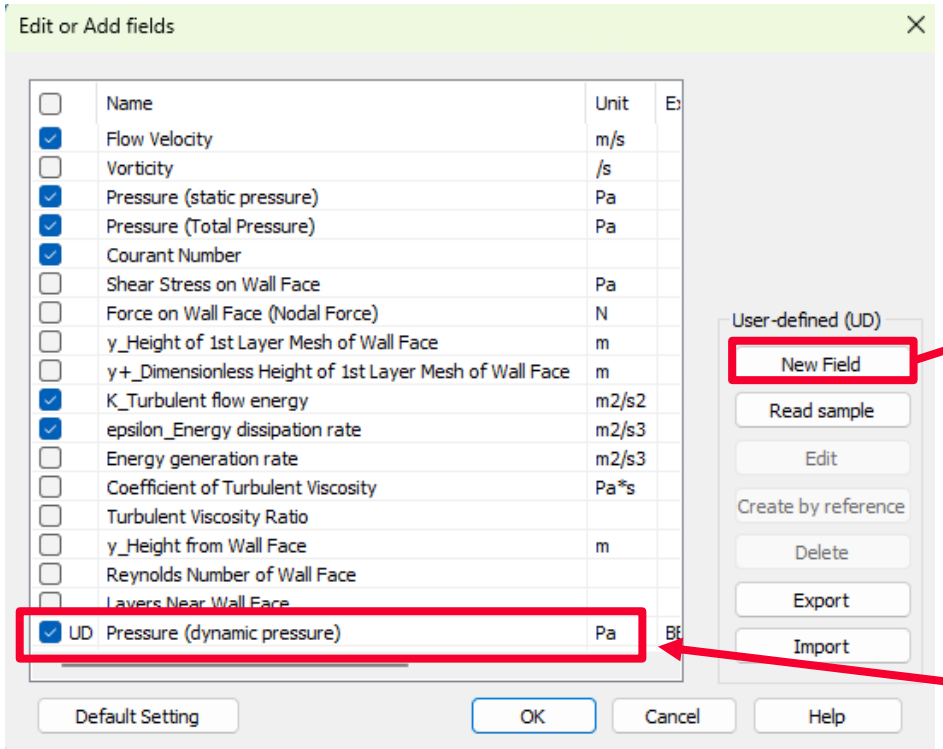
Show all



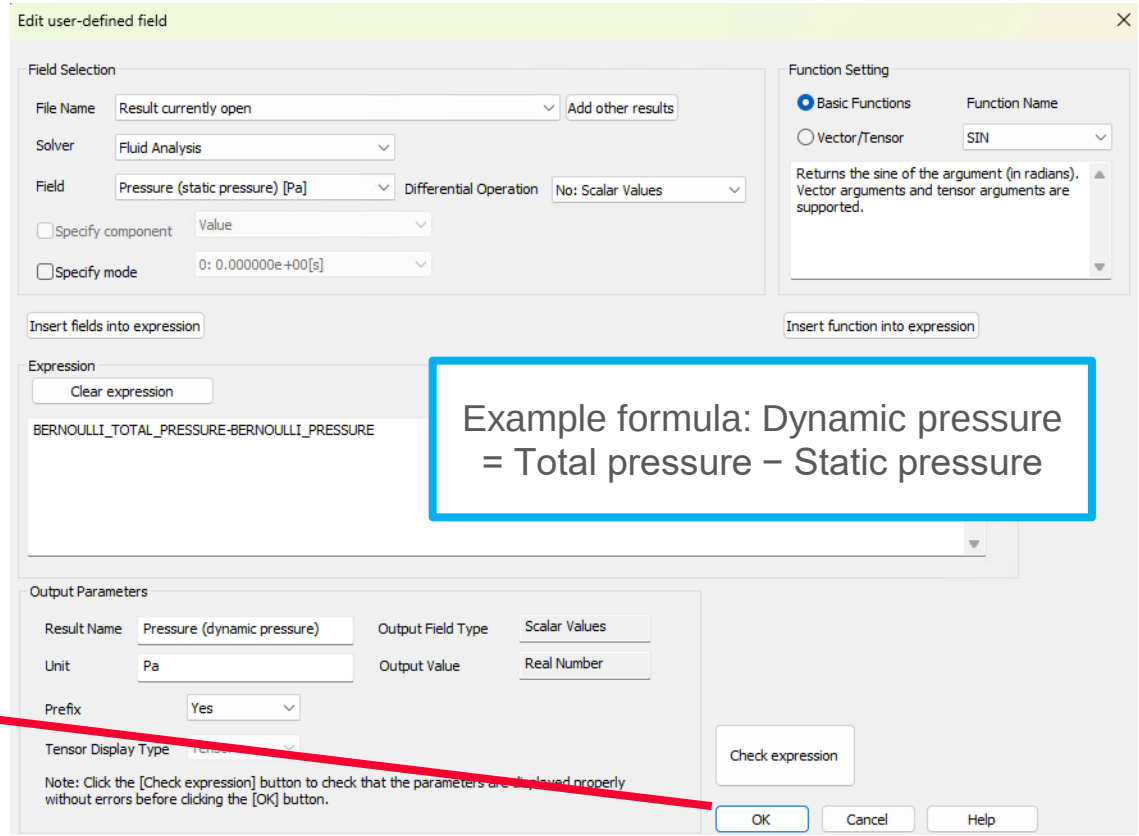
-Select [Edit or Add Fields] in the combo box to change fields to be displayed.
-Fewer fields are initially displayed for easy selection.
-Show or Hide is switched by the check box in the field editing or adding dialog box.
-New fields can be created by user definition and displayed.

Result display: Edit or Add Fields

[New Field] button is added.



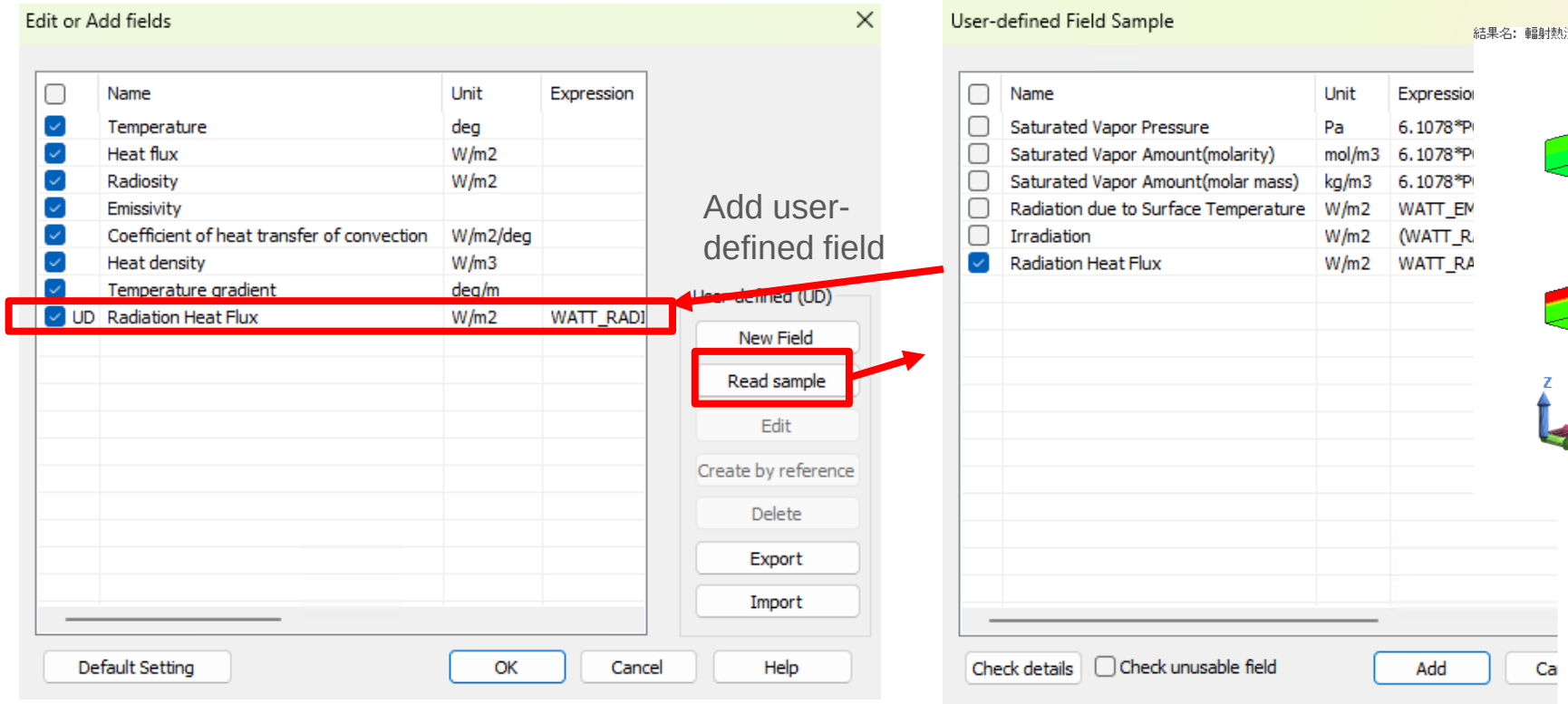
Add user-defined field

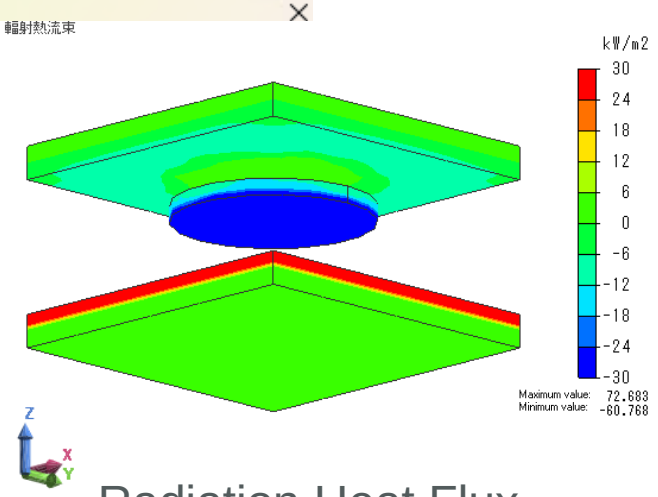


-A new field is created by combining existing fields and functions.
-Operations of vectors or tensors and such differential operations of fields as gradient, divergence, and rotation are applicable.
-It is possible to select from sample lists and to import or export settings.

Result display: Edit or Add Fields

[Read sample] button is added.





Radiation Heat Flux

$$Q = R - \frac{(R - \varepsilon\sigma\theta^4)}{1 - \varepsilon}$$

R: Radiance、ε: Emissivity、
θ: Absolute temperature
σ : Stefan–Boltzmann constant

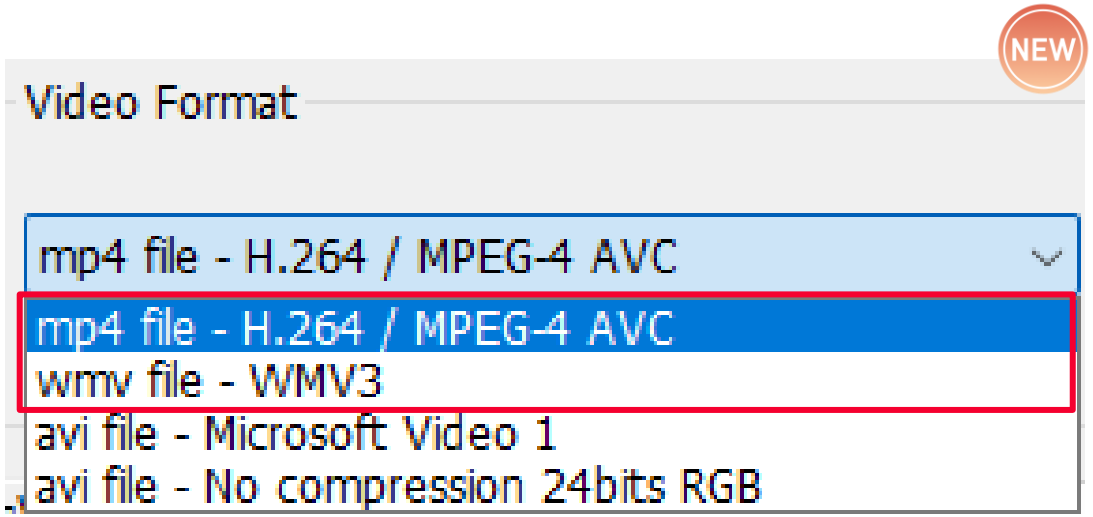
Several sample calculation formulas are provided by default
-Formula for calculating radiative heat flux from radiance, emissivity, and temperature.
-Formula for calculating the saturated water vapor amount from temperature.
-Formula for calculating engineering strain / Green–Lagrange strain / logarithmic strain from displacement.
-Formula for calculating the magnitude of shear stress acting on a plane perpendicular to the Z-axis.

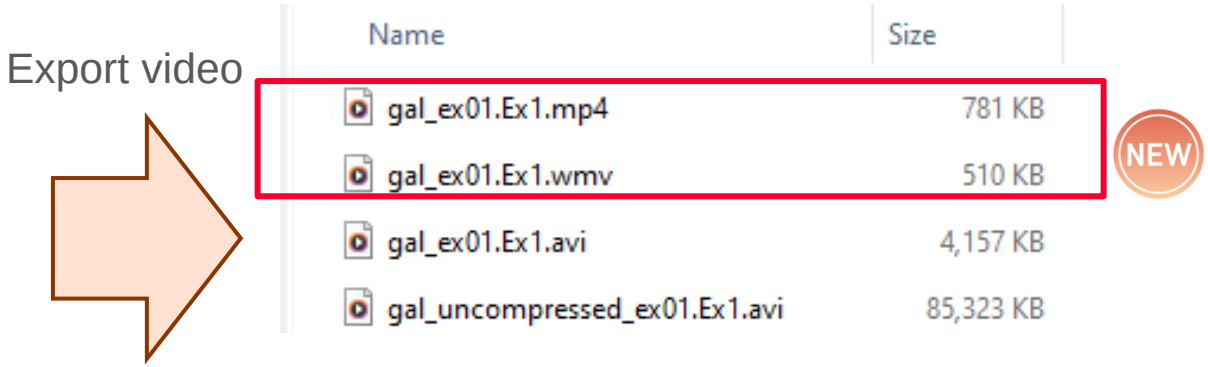
Display result: Additional animation video export formats

Additional animation video formats are supported.

Added support for the .mp4 (H.264) and .wmv (WMV3) video formats in animation output. These formats produce smaller file sizes and can be played on most devices and platforms.

The default video save format is now .mp4.

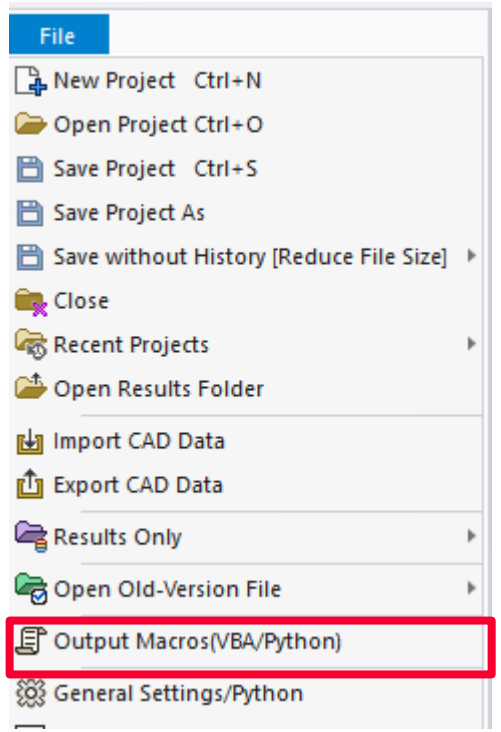


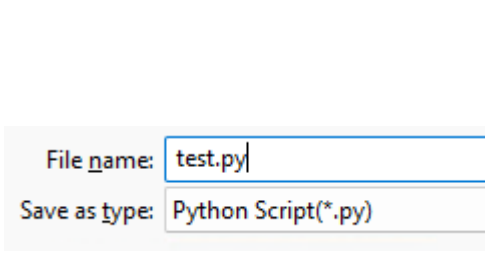


Compared to the traditional AVI file, the high compression allows for smaller file sizes -about 80% reduction.

Macro: Export Python script file

The macro file output feature now supports generating Python script files.






```
# Main function
def FemtetMain():
    Global Femtet   # Femtet variables are specified as global variables.
    global Gaudi

    # Python Femtet utility packages are required.
    util.auto_execute_femtet()  # Femtet auto start.

    Femtet = Dispatch(const.CFemtet)    # Cfemtet class instance is defined..

    if Femtet.OpenNewProject()==False:
        print(Femtet.LastErrorMsg)
        sys.exit()
    # ------- Variables definition -------
    InitVariables()
    # ------- Database setting -------
    AnalysisSetUp()
    BodyAttributeSetUp()
    MaterialSetUp()
    BoundarySetUp()
    # ------- Model creation -------
    Gaudi = Femtet.Gaudi
    MakeModel()
    # ------- Set up general mesh size -------
    # <<<<<<< Set to -1 for automatic calculation >>>>>>>
    Gaudi.MeshSize = 1.0
```

It can be utilized for a wide range of purposes, such as learning Femtet macro operations or getting started with automation processes.
Before outputting a Python script, make sure to enable the “Macro” feature in advance.
Execution requires the Python Femtet Utility Package.

Python library - PyFemtet :
Using Femtet Parametric Analysis Results as an Objective Function

The results of Femtet's parametric analysis can be directly set as the objective function.

Previously, in optimization programming using PyFemtet, it was necessary to define the objective function with Femtet macros.

```python
from pyfemtet.opt import FEMOpt, FemtetInterface

def mises(Femtet):
    # ミーゼス応力計算
    Gogh = Femtet.Gogh
    Gogh.Galileo.Potential = constants.GALILEO_VON_MISES_C
    succeed, (x, y, z), mises = Gogh.Galileo.GetMAXPotentialPoint_py(constants.CMPX_REAL_C)

    # エラーチェック
    if not succeed:
        raise PostProcessError('ミーゼス応力の取得に失敗しました。')

    return mises

if __name__ == '__main__':

    # Femtetとの接続
    fem = FemtetInterface()
    femopt = FEMOpt(fem=fem)
    femopt.add_parameter('in_radius', 10, 5, 10)
    femopt.add_parameter('out_radius', 20, 20, 25)
    femopt.add_objective('mises', mises, 20, 20, 25)
    femopt.set_random_seed(42)  # 乱数シードの固定
    femopt.optimize(n_trials=20)
```

Previous PyFemtet Optimization Code

With the new feature, the definition of optimization objective functions can be replaced by Femtet's standard functionality.

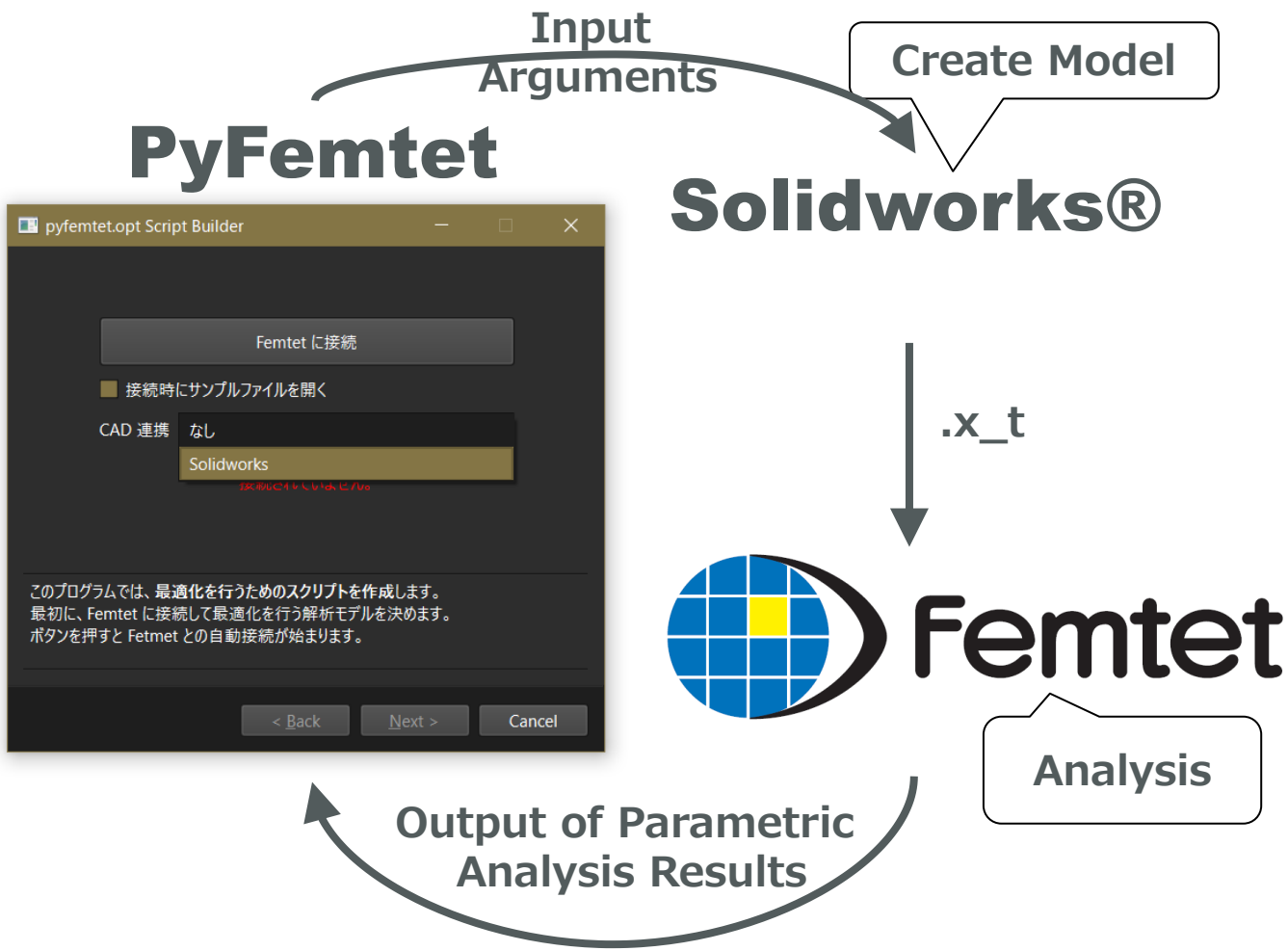
As a result, the amount of coding required for optimization can be significantly reduced.

```python
from pyfemtet.opt import FEMOpt, FemtetInterface

if __name__ == '__main__':

    # Femtetとの接続
    fem = FemtetInterface()

    # ミーゼス応力の出力を目的関数として使用（Femtet のパラメトリック解析結果出力に設定しておく）
    fem.use_parametric_output_as_objective(number=1, direction='minimize')

    femopt = FEMOpt(fem=fem)
    femopt.add_parameter('in_radius', 10, 5, 10)
    femopt.add_parameter('out_radius', 20, 20, 25)
    femopt.set_random_seed(42)  # 乱数シードの固定
    femopt.optimize(n_trials=20)
```

PyFemtet Optimization Code Using the New Feature

Python library - PyFemtet : Improved Script Builder User Interface

The functionality of the PyFemtet Script Builder has been enhanced.



Scripts for utilizing Femtet's parametric analysis can be created easily.

Scripts for integration with SolidWorks® are now supported.

Python library - PyFemtet : Surrogate Model Construction Feature Added

Surrogate model construction using Femtet is now supported.

Training Data and Trained Models Created with PyFemtet

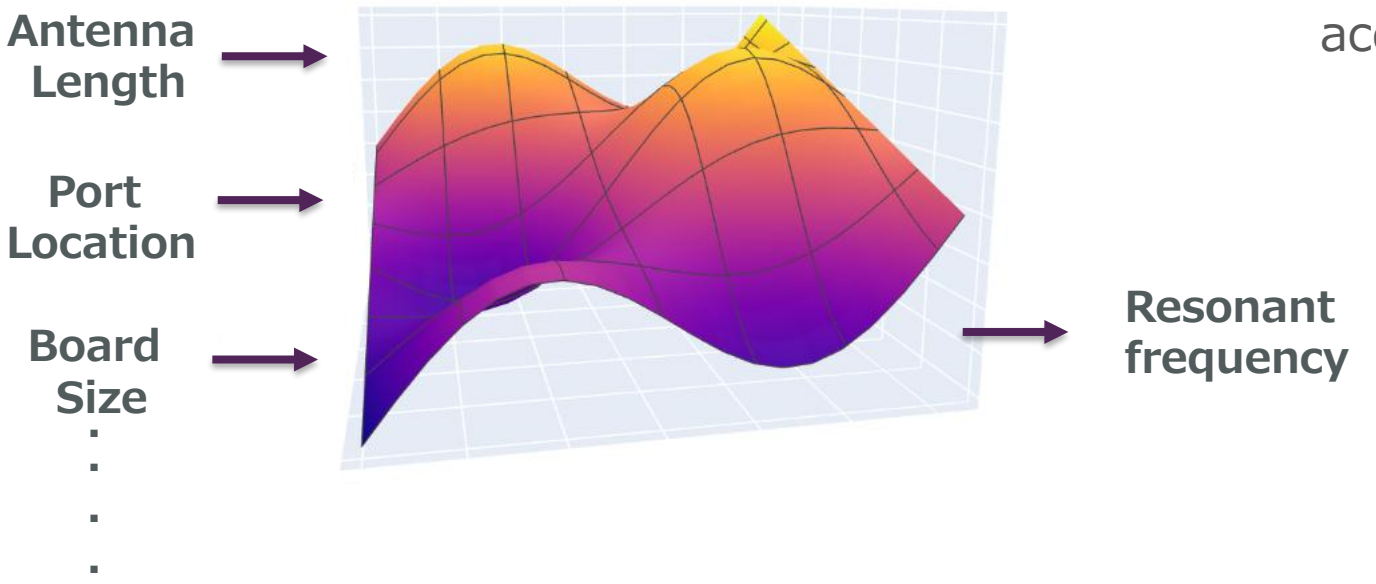


Image of Output for Estimated Results (Objective Function) Using a Surrogate Model

Surrogate models can be constructed from analysis results obtained via PyFemtet.

The use of surrogate models enables fast acquisition of estimated results.

Femtet

Murata Software Co., Ltd.

muRata