

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.

CAE Software



muRata

Murata software Co., Ltd.

CAE



Femtet Ver.2024.1 What's New

CAE Software



muRata

Murata software Co., Ltd.

CAE



Table of Contents

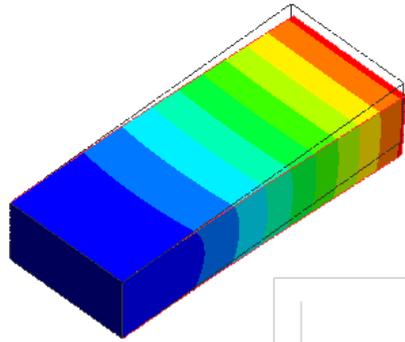
Functionality	What's New
Solver	<ul style="list-style-type: none">• <u>Stress Analysis: Domain Decomposition Method execution conditions</u>• <u>Thermal Analysis/Stress Analysis: Speed up by the algebraic multigrid method</u>• <u>Thermal Analysis: Faster and less memory by improved thermal radiation</u>• <u>Acoustic Analysis: A plane wave incidence feature</u>• <u>Piezoelectric Analysis: The reciprocal of Q_m (mechanical quality factor)</u>• <u>Fluid Analysis: Non-newtonian fluid</u>• <u>Fluid Analysis: Analysis using the material properties depending on ambient values</u>• <u>Fluid Analysis: Change the methods to calculate the density and buoyancy of ideal gas</u>

Table of Contents

Functionality	What's New
Model	<ul style="list-style-type: none">• Model – Model import function of Gerber data and drill data
UI	<ul style="list-style-type: none">• UI - New function of [Rotate viewpoint by any angle]• UI - Warning messages are displayed in a message bar
Macro	<ul style="list-style-type: none">• Macro - Added CFemtet.Exit function

Solver – Stress Analysis : Domain Decomposition Method execution conditions

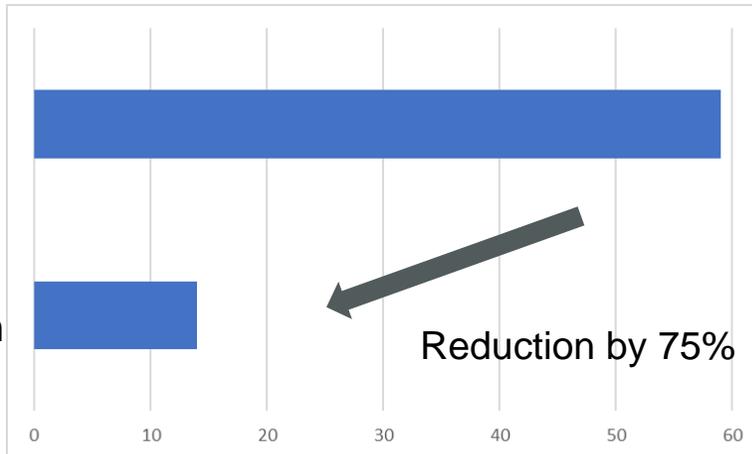
In static stress analysis, the domain decomposition method for faster computation used to require both the Advanced mechanical option and the Accelerator option. As of version 2024.1, only the Accelerator option is needed.



Stress Analysis example-1
200k meshes

Direct method

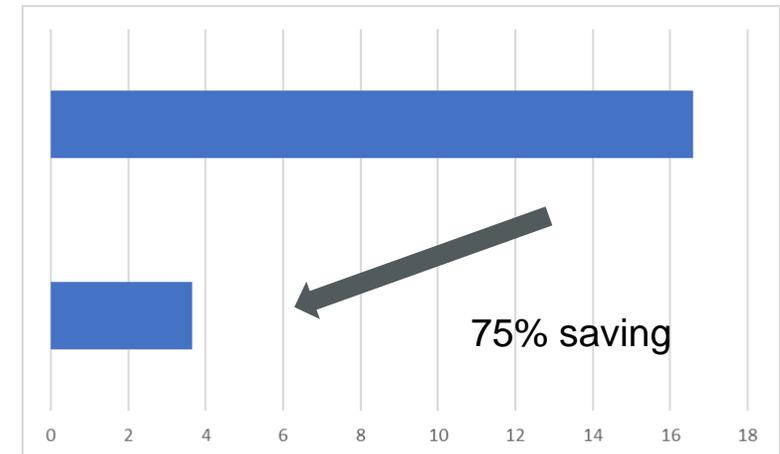
Domain
Decomposition
Method



Simulation time(s)

Direct method

Domain
Decomposition
Method



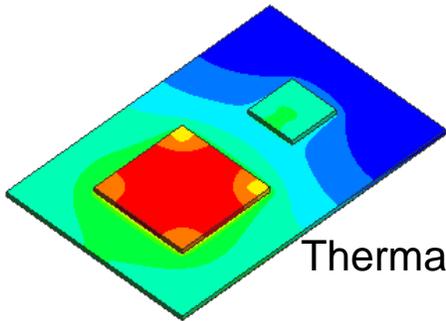
Memory usage(GB)

* CPU : i7014700k

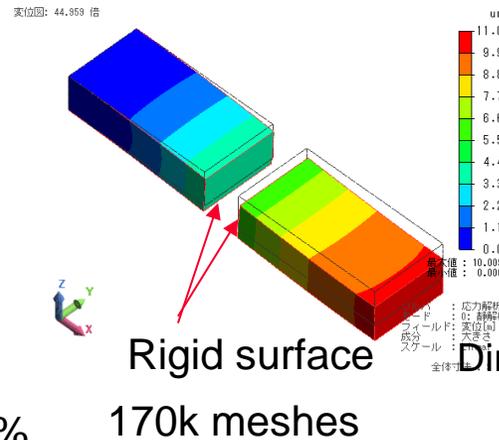
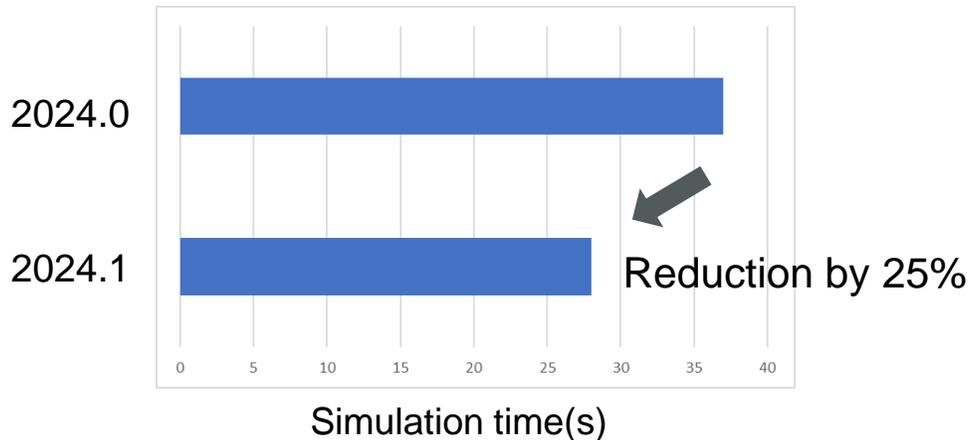
Solver – Thermal Analysis/Stress Analysis

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.

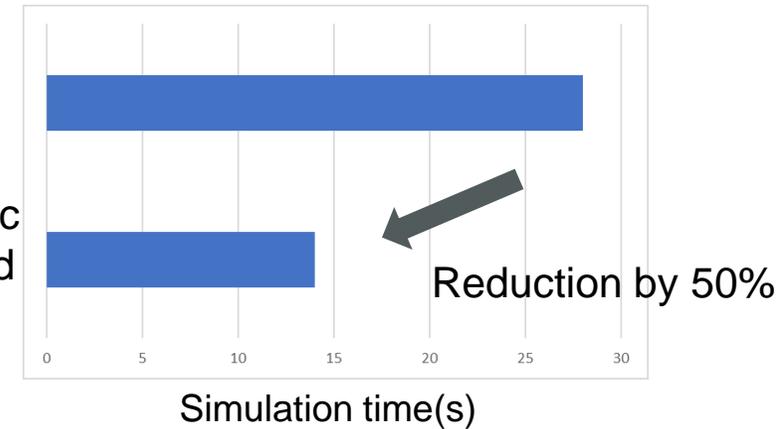


Thermal analysis example-16
500k meshes



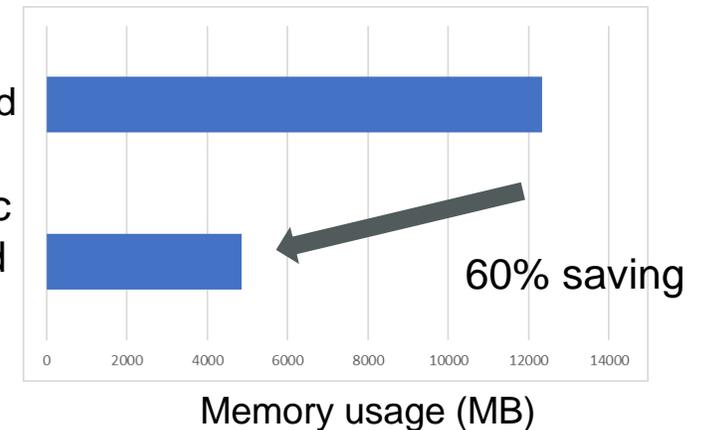
Direct method

Algebraic multigrid method



Direct method

Algebraic multigrid method



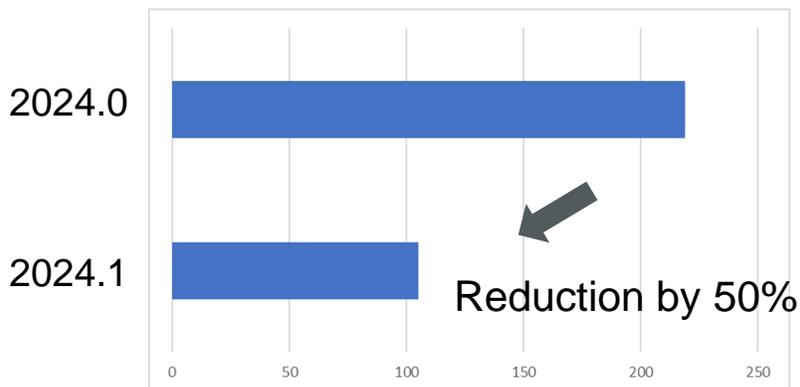
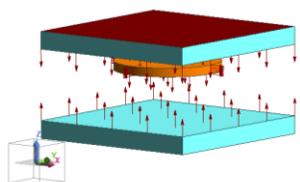
*CPU : i7-14700k

Solver – Thermal Analysis

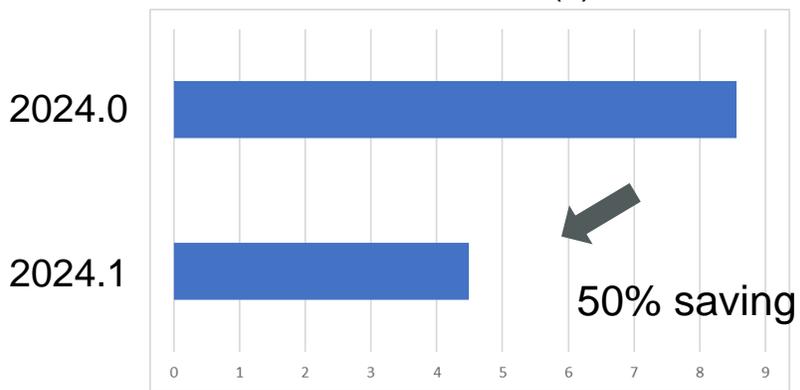
Faster and less memory by improved thermal radiation

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

Example-16
680k meshes
iterative method

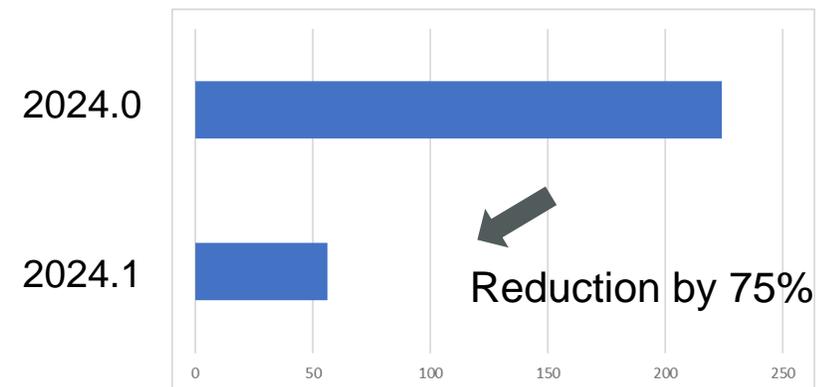
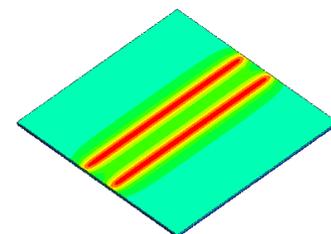
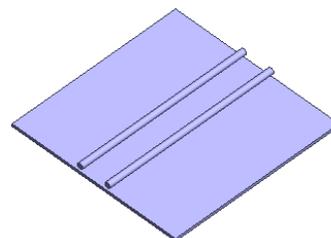


Simulation time(s)

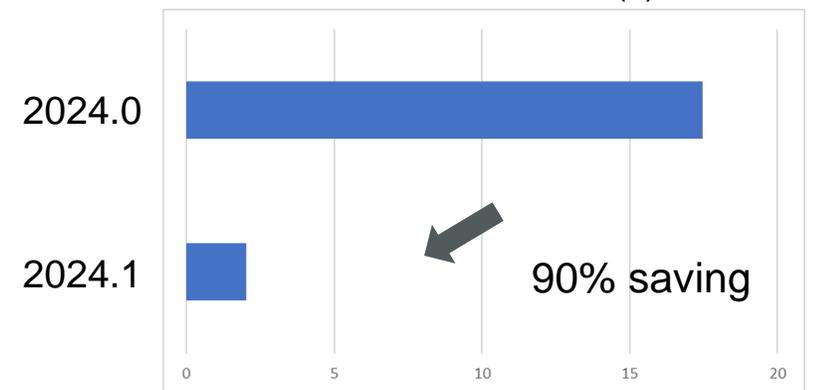


Memory usage(GB)

Heater model
350k meshes



Simulation time(s)



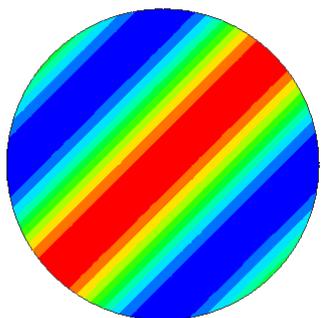
Memory usage(GB)

*CPU : i7-14700k

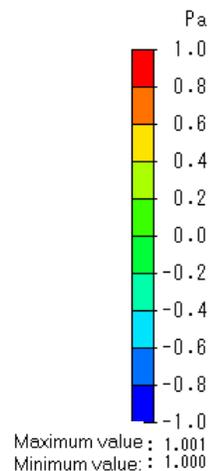
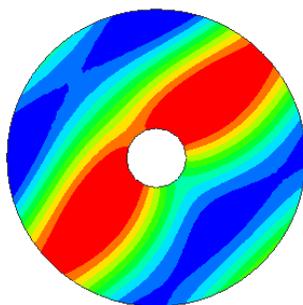
Solver - Acoustic Analysis: A plane wave incidence feature

A plane wave incidence feature has been implemented.

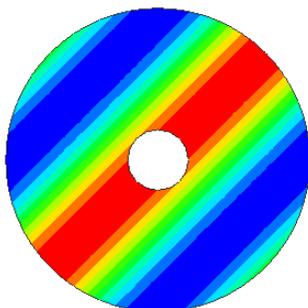
Plane wave



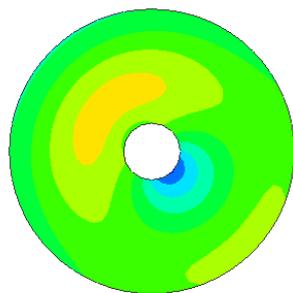
Incident wave +
Reflected wave



Incident wave



Reflected wave



Setting for plane wave incidence feature

Analysis Condition Setting

Solver

Acoustic Analysis

Mesh

Open Boundary

Harmonic Analy...

Transient Analysis

Sound Waves D...

Incident Wave (...)

High-Level Setti...

Results Import

Notes

Acoustic Analysis

Analysis Type

Harmonic Analysis

Transient Analysis

Calculate directivity when running solver

Calculate loss

Incident Plane Wave

Analysis Condition Setting

Solver

Acoustic Analysis

Mesh

Open Boundary

Harmonic Analy...

Transient Analysis

Sound Waves D...

Incident Wave (...)

High-Level Setti...

Results Import

Notes

Incident Wave (Plane Wave)

Propagation Direction

X

Y

Z

Pressure (Sound Pressure)

Sound Pressure Level [dB]

Sound Pressure [Pa]

* This feature support Harmonic Analysis only.

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

The image shows two overlapping dialog boxes from a software interface. The background dialog is titled "Piezoelectricity" and contains several sections: "Material Type" with radio buttons for "Piezoelectric Material", "Dielectric Material (non-piezoelectric)", and "Perfect Conductor"; "Anisotropy" with radio buttons for "Isotropic" and "Anisotropic"; "Piezoelectricity Type" with radio buttons for "e-type", "h-type", "d-type", and "g-type"; "Elasticity matrix (compliance)" with a 6x6 grid of input fields; "Piezoelectricity matrix" with a 3x6 grid of input fields; "Mechanical Loss Tangent (1/Qm)" with a text input field containing "0.0005" and a checked "Frequency Dependency" checkbox; "Dielectric Tangent" with a text input field containing "0.006"; and "Relative permittivity matrix" with a 3x3 grid of input fields. The foreground dialog is titled "Edit Nonlinearity Table" and contains a table with columns "No.", "Frequency", and "Mechanical". The table has 11 rows, with the first two rows containing data: (1, 1, 0.01) and (2, 5, 0.1). The dialog also includes buttons for "Insert Rows", "Delete Rows", "Import", "Graph", "Unit" (set to kHz), "Smooth Interpolation", "OK", "Cancel", and "Help".

Piezoelectricity Dialog:

- Material Type: Piezoelectric Material
- Anisotropy: Isotropic, Anisotropic
- Piezoelectricity Type: e-type, h-type, d-type, g-type
- Mechanical Loss Tangent (1/Qm): 0.0005
- Frequency Dependency:

Edit Nonlinearity Table Dialog:

No.	Frequency	Mechanical
1	1	0.01
2	5	0.1
3		
4		
5		
6		
7		
8		
9		
10		
11		

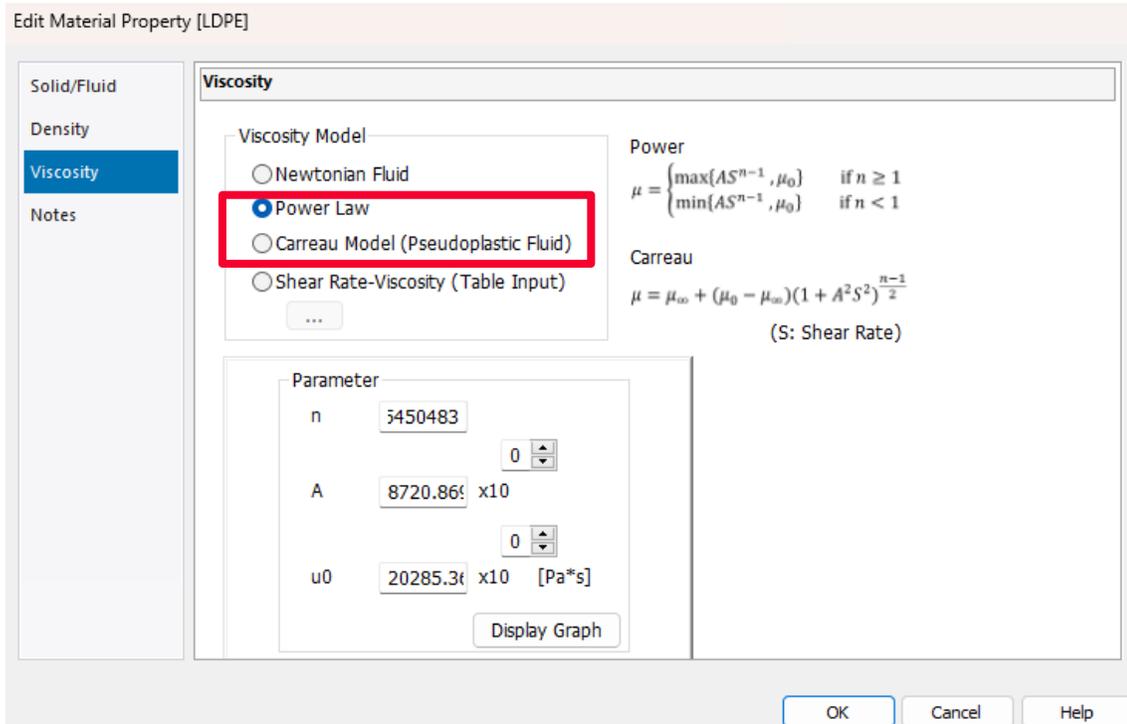
Solver – Fluid Analysis: Non-newtonian fluid

Non-newtonian fluid is supported.

> Parameter setting



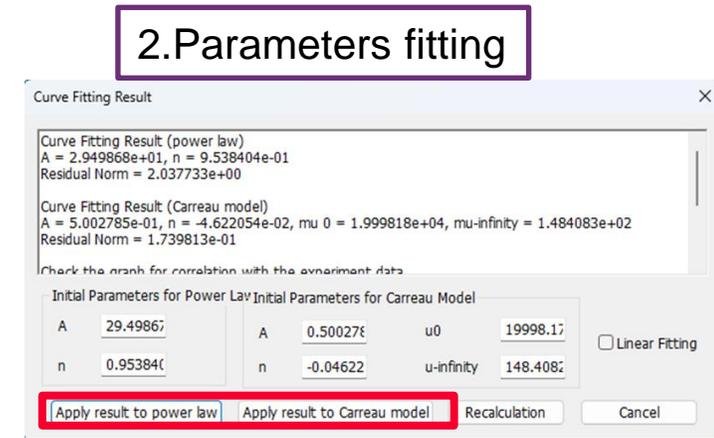
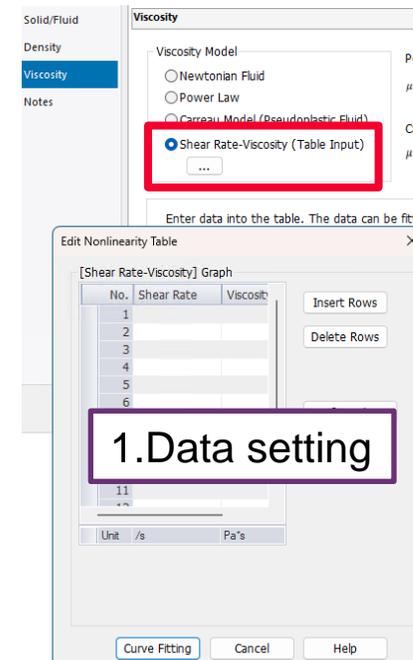
Power law mode and Carreau model are supported for non-Newtonian fluid.



> Parameter fitting



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

The screenshot shows the 'Fluid Analysis' dialog box. On the left, 'Steady-state Analysis' is selected under 'Analysis Type', 'Laminar Flow' is selected under 'Laminar Flow/Turbulent Flow', and 'Internal Flow' is selected under 'Flow Type'. The 'Ambient Value Setting' option is highlighted with a red box. A 'NEW' badge is visible. The 'Ambient Value Setting' sub-dialog is open, showing 'Ambient Temperature' set to 25.0 [deg] and 'Ambient Pressure' set to 101325.0 [Pa]. The checkbox 'Analyze using the material properties depending on ambient values.' is checked. A note at the bottom states: 'The specified ambient temperature and ambient pressure are used to calculate the flow density and viscosity from the equation of state.'

> Fluid-Thermal Analysis

The screenshot shows the 'Fluid-Thermal Analysis' dialog box. On the left, 'Steady-state Analysis' is selected under 'Fluid-Thermal Analysis (Watt/Bernoulli) Type', 'Laminar Flow' is selected under 'Laminar Flow/Turbulent Flow', and 'Internal Flow' is selected under 'Flow Type'. The 'Ambient Value Setting' option is highlighted with a red box. A 'NEW' badge is visible. The 'Ambient Value Setting' sub-dialog is open, showing 'Ambient Temperature' set to 25.0 [deg] and 'Ambient Pressure' set to 101325.0 [Pa]. The checkbox 'Analyze using the material properties depending on ambient values.' is checked. A note at the bottom states: 'The temperatures for respective positions and the specified ambient pressure are used to calculate the fluid density and viscosity.'

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.

The screenshot shows the 'Density' settings panel in a software application. The left sidebar lists various material properties: Solid/Fluid, Specific Heat, Density (highlighted), Thermal Condu..., Viscosity, and Notes. The main panel is titled 'Density' and contains several input fields and options. The 'Density' field is set to 1.184 kg/m3. The 'Temperature Dependency' section has three radio button options: 'Specify Coefficient of Volumetric Thermal Expansion', 'Ideal Gas' (which is selected and highlighted with a red box), and 'Table Input'. Below this, the 'Coefficient of Volumetric Thermal Expansion' is set to 0.003354 X10 [1/deg]. The 'Molar Mass (Molecular Weight)' is set to 0.02896413 kg/mol.

Density formula $\rho = \frac{M_w P_{ref}}{RT_{abs}}$ 

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

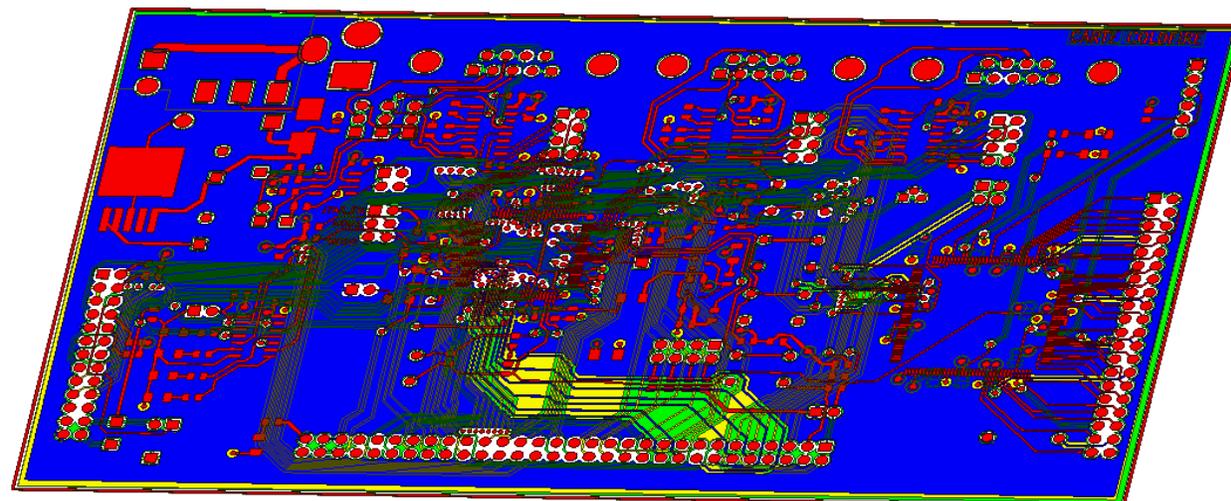
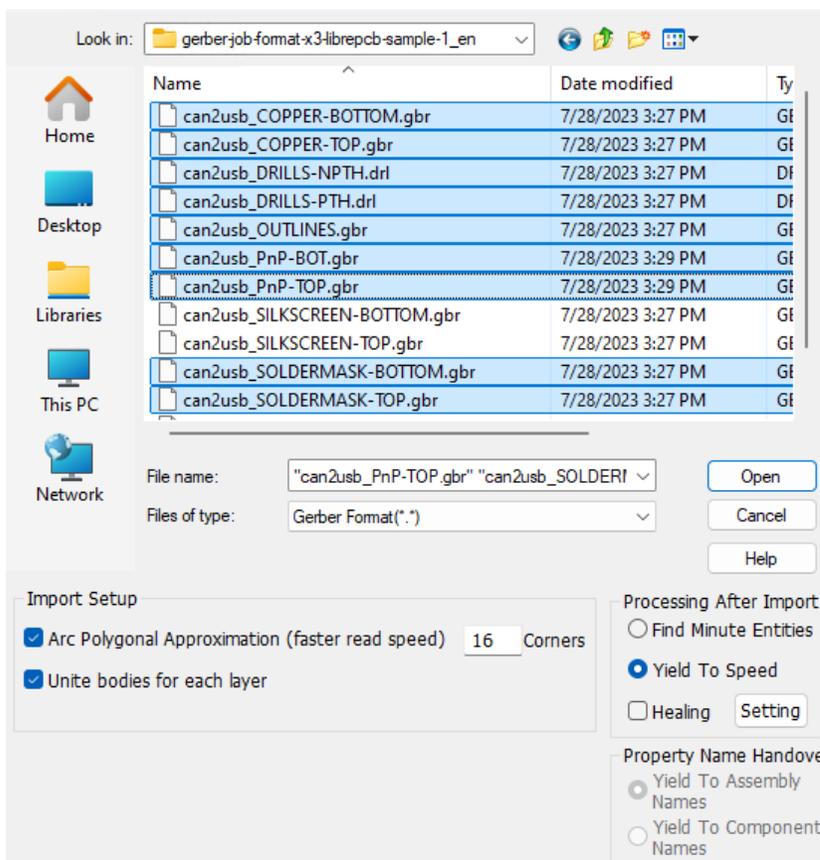
Buoyancy formula $f_b = (\rho - \rho_{ref})g$ 

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

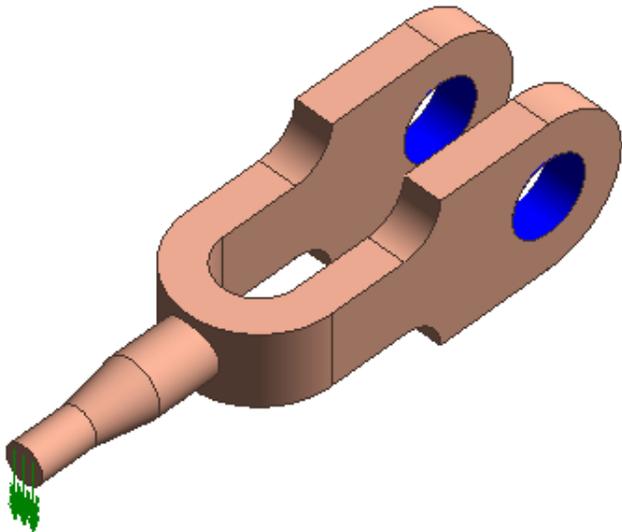
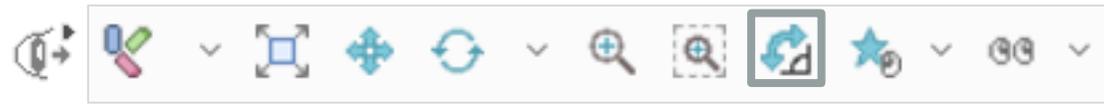
Gerber data (RS274X) and drill data (Excellon format) are supported.

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

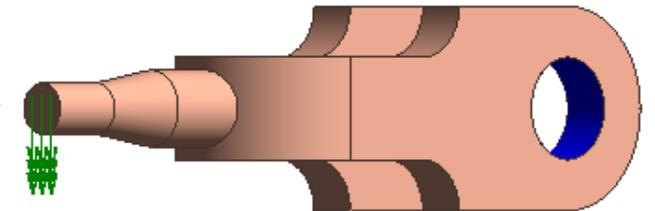
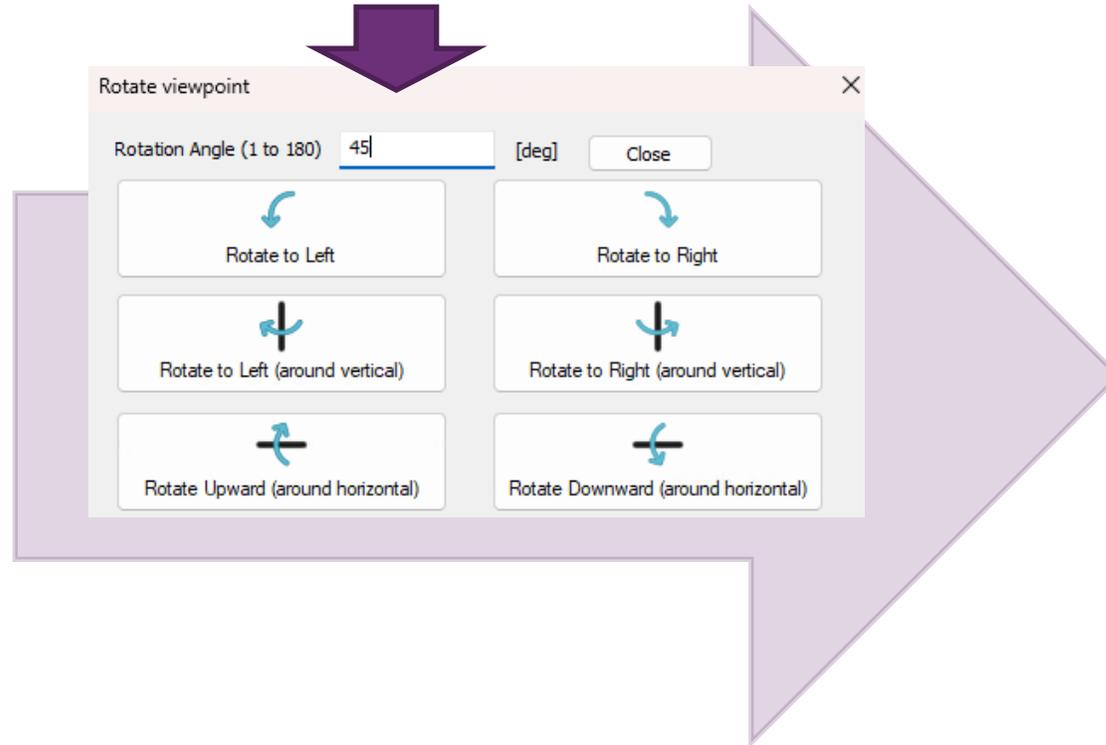


UI – New function of [Rotate viewpoint by any angle]

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.

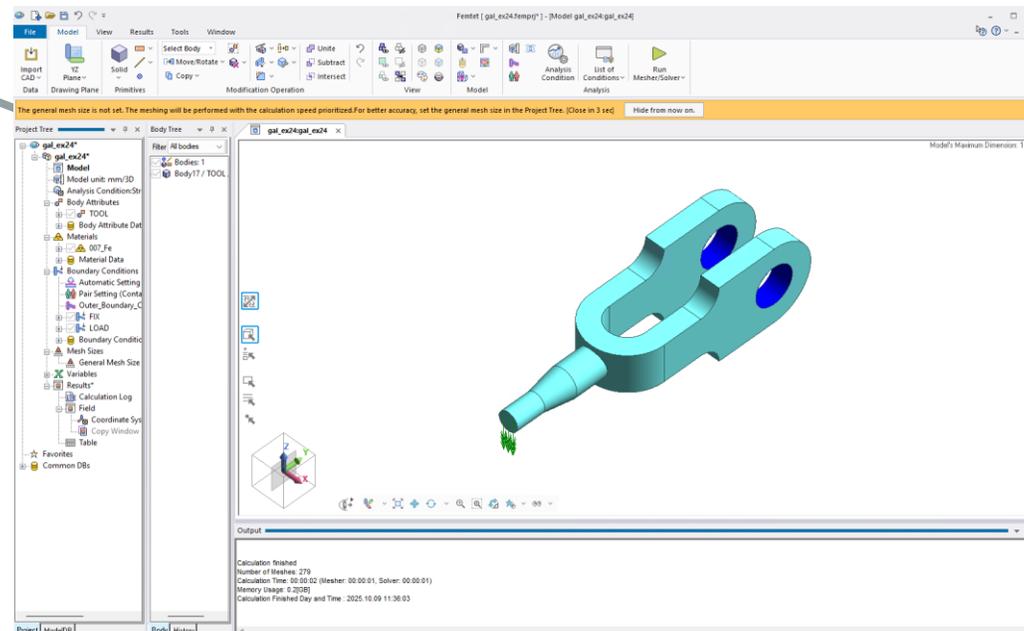
UI - Warning messages are displayed in a message bar

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.

The general mesh size is not set. The meshing will be performed with the calculation speed prioritized. For better accuracy, set the general mesh size in the Project Tree. [Close in 13 sec]

Hide from now on.



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

CAE Software



muRata

Murata software Co., Ltd.

CAE



Table of Contents

Functionality	What's New
Solver	<ul style="list-style-type: none">• Thermal Analysis: Speed up for surface-surface radiation by using hardware GPU• Thermal Analysis: Automatic internal fluid creation is applied• Stress Analysis: Fast Calculation in Static Analysis• Stress Analysis: Shell element of Orthotropic Material• Piezoelectric Analysis: Impedance Graph is Added to Result Table• Fluid Analysis - Cell-centered Solve Is Available in Fluid/Fluid-Thermal Analysis• Fluid Analysis: Automatic Setting of Outer Boundary Condition• Fluid Analysis: Automatic Fluid Domain Creation (using Internal Flow)• Fluid Analysis: Automatic Fluid Domain Creation (using External Flow)• Acoustic Analysis: Specific Acoustic Impedance Boundary Condition• Solver - Simple Restart• Solver - Delete Intermediate Results

Table of Contents

Functionality	What's New
Mesher	<ul style="list-style-type: none">• Mesher: Sweep Mesh Expanded to Many-to-Many
Model	<ul style="list-style-type: none">• Model: Selecting function has been extended• Model: Convert Multiple Bodies Collectively• Model: [Convert] menu for topology to convert automatically.• Model: Model export function of Gerber data (RS274X)
Result Display	<ul style="list-style-type: none">• Result display: Edit or Add Fields• Display result: Additional animation video export formats
Macro	<ul style="list-style-type: none">• Macro: Export Python script file
Python Library PyFemtet	<ul style="list-style-type: none">• Using Femtet Parametric Analysis Results as an Objective Function• Improved Script Builder User Interface• Surrogate Model Construction Feature Added

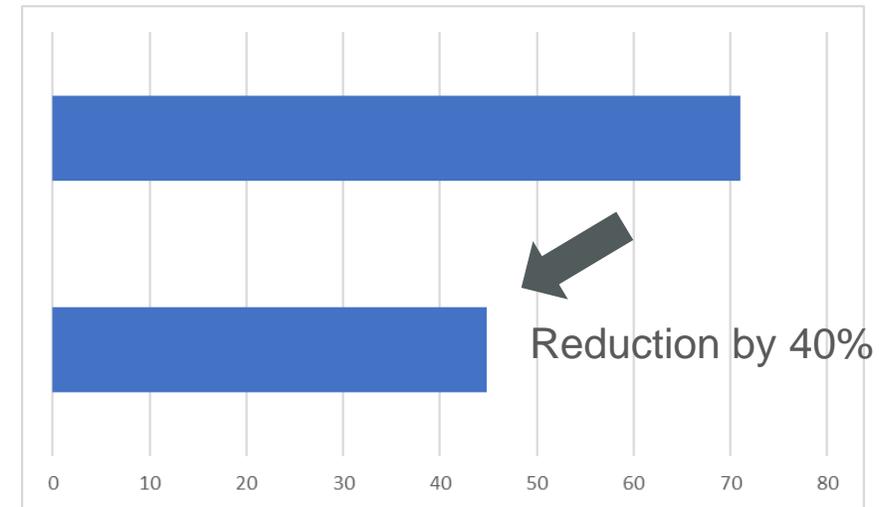
Solver – Thermal Analysis: Speed up for surface-surface radiation by using hardware GPU

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

Thermal Analysis Example 16
1200k meshes

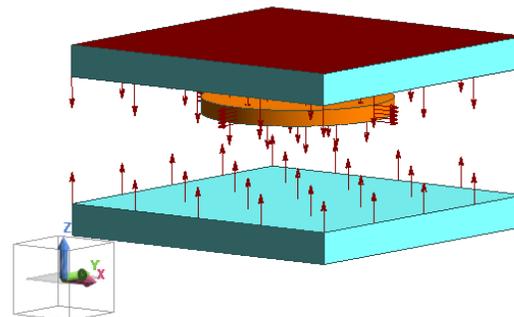
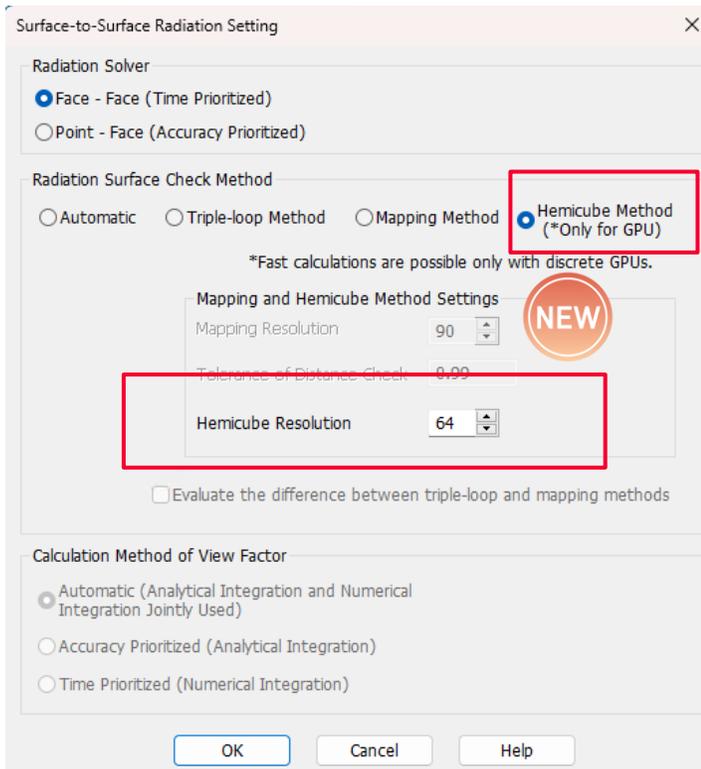
Ver2024.1
Mapping method

Ver2025.0
Hemicube method



Checking time for radiation(s)*

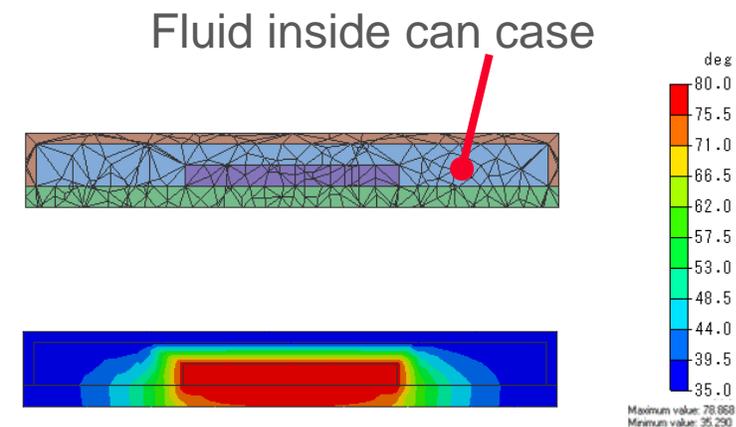
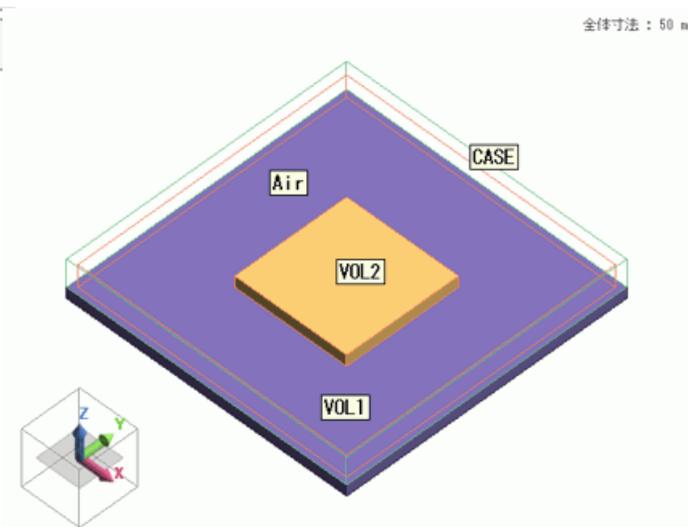
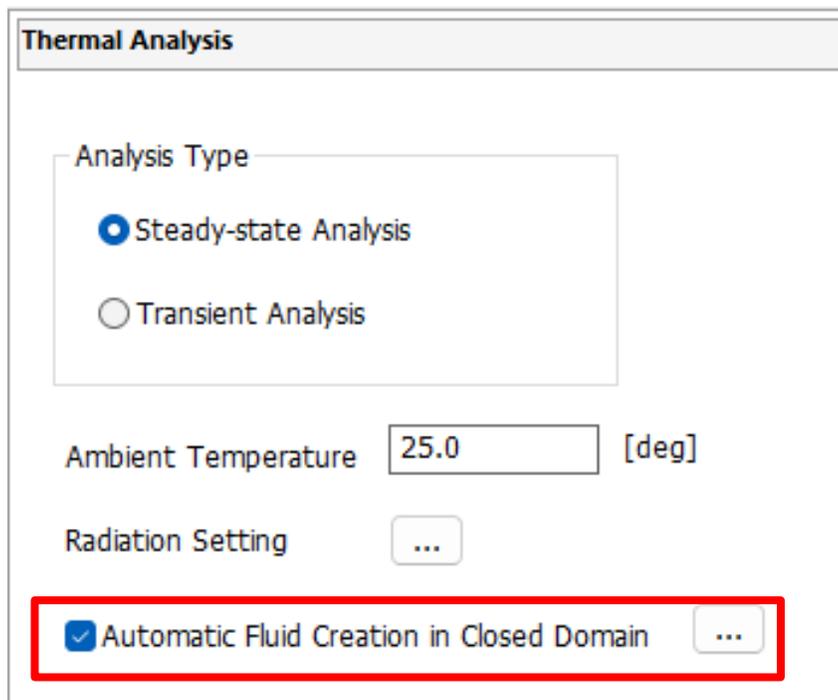
*Simulation condition
Use same resolution (90)
CPU: i7-12700
Mapping method: CPU8 cores parallel
Hemicube method: GPU T400 + CPU8 cores parallel



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.



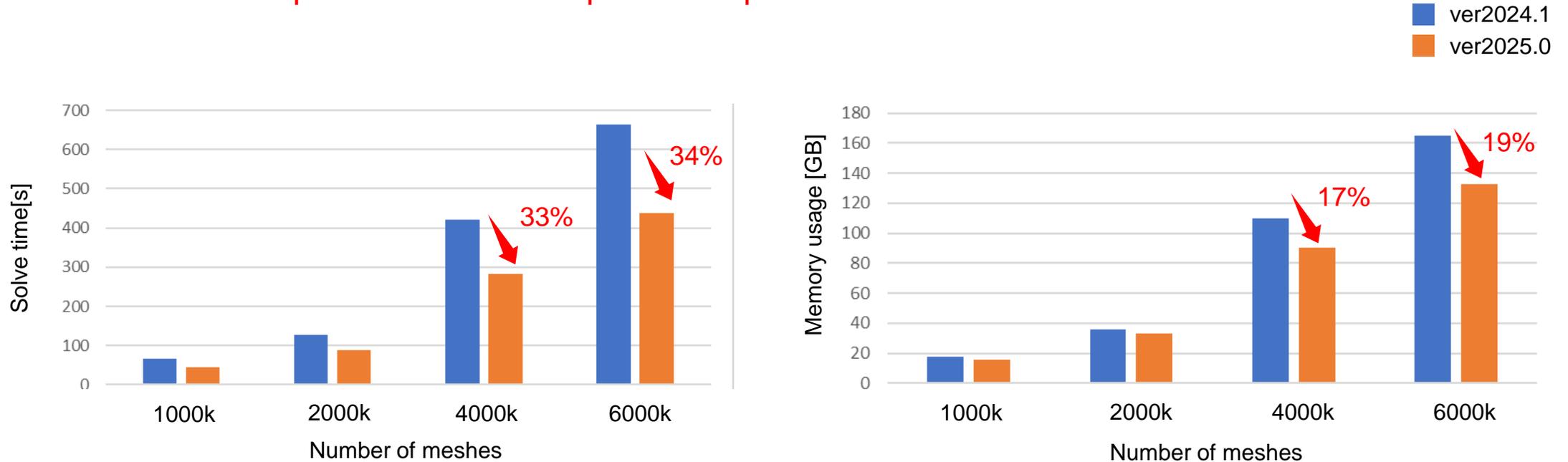
Thermal Analysis example 30:
Heat Dissipation of IC inside a Case by Natural Convection

- 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.



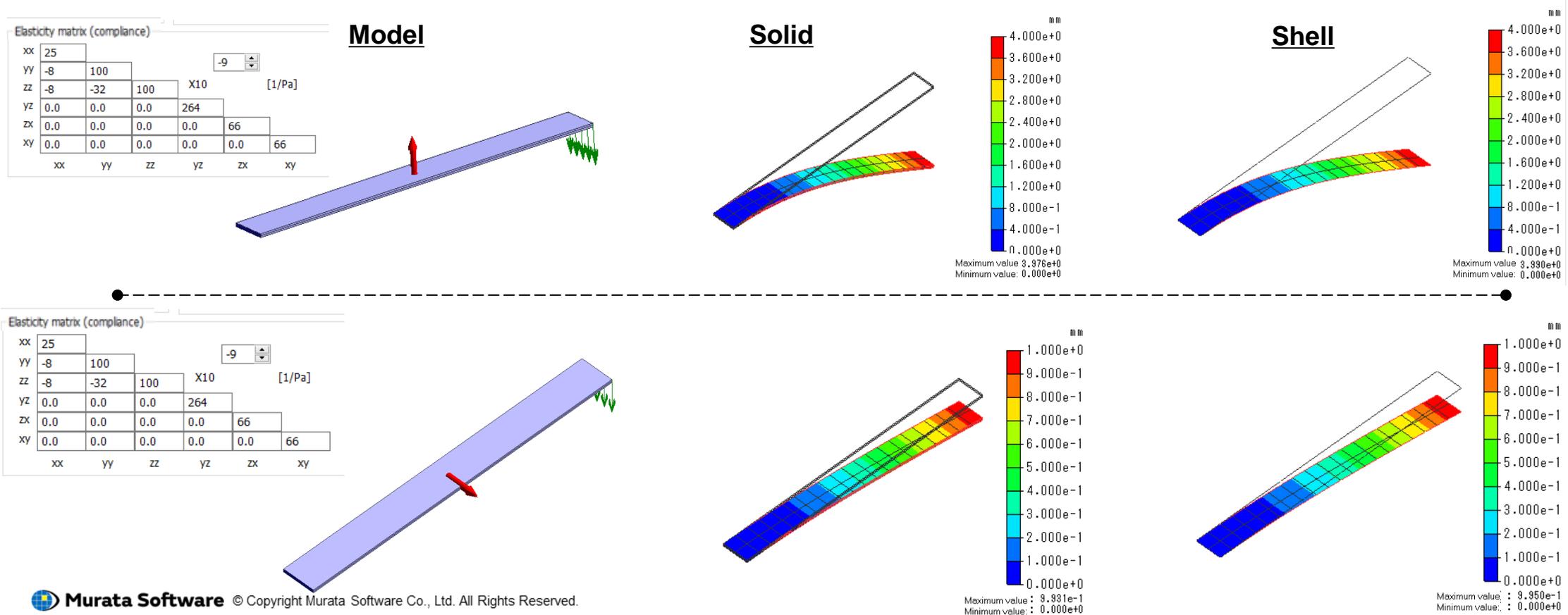
CPU : Xeon W-3375 (38 core)

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.

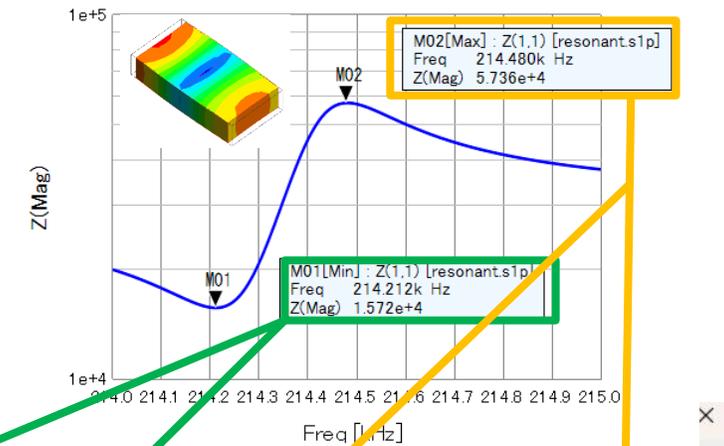


Table	Floating electrode potential [V]	Resonant frequency [Hz]	Convergence Judgment	Damping capacitance [F]	Free capacitance [F]	Spread between resonant and antiresonant frequencies [Hz]	Coupling coefficient [%]	Resonant resistance [ohm]	Equivalent capacitance [pF]	Equivalent inductance [H]	FEM Info	Impedance Graph (Resonant analysis)
	Frequency [...]	Resonant Frequency (f0) [...]	R(f0)[ohm]	X(f0)[ohm]	abs(Z(f0))[ohm]	Antiresonant Frequency (fa) [...]	R(fa)[ohm]	X(fa)[ohm]	abs(Z(fa))[ohm]			
0:	1.768964e+05Hz	1.76896350e+5	7556689e+2	-3.57313205e+4	3.573700e+4	1.769447e+5	6.60017515e+2	-3.69898290e+4	3.6995169e+4			
1:	2.142689e+05Hz	2.14268885e+5	2.14211051e+5	96947835e+3	-1.29062403e+4	1.57169520e+4	2.14480865e+5	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 Antiresonant 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.

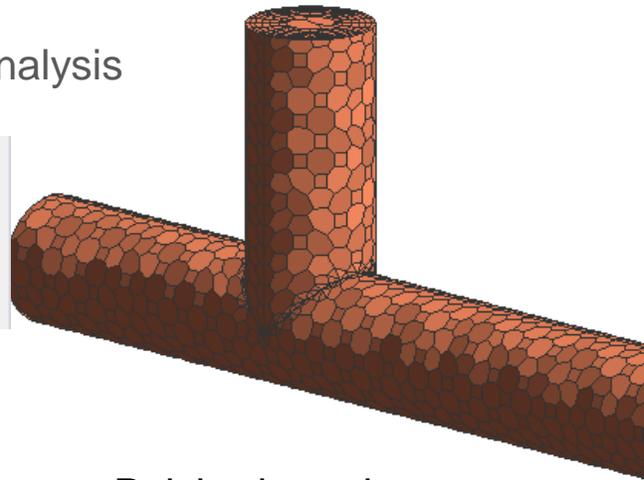
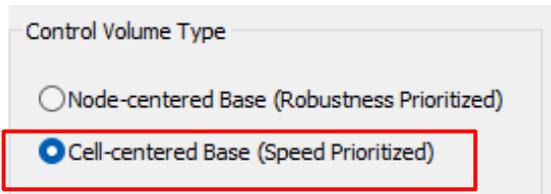
Solver – Fluid Analysis:

Cell-centered Solve Is Available in Fluid/Fluid-Thermal Analysis

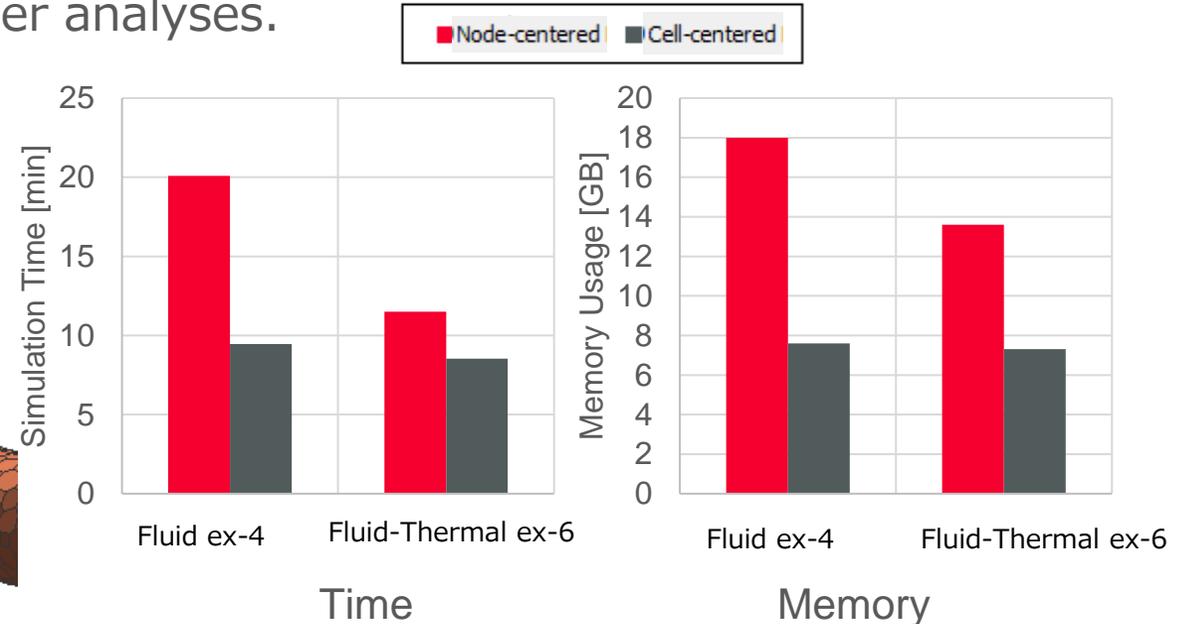
Cell-centered solver is available in fluid analysis for faster calculation.

* 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

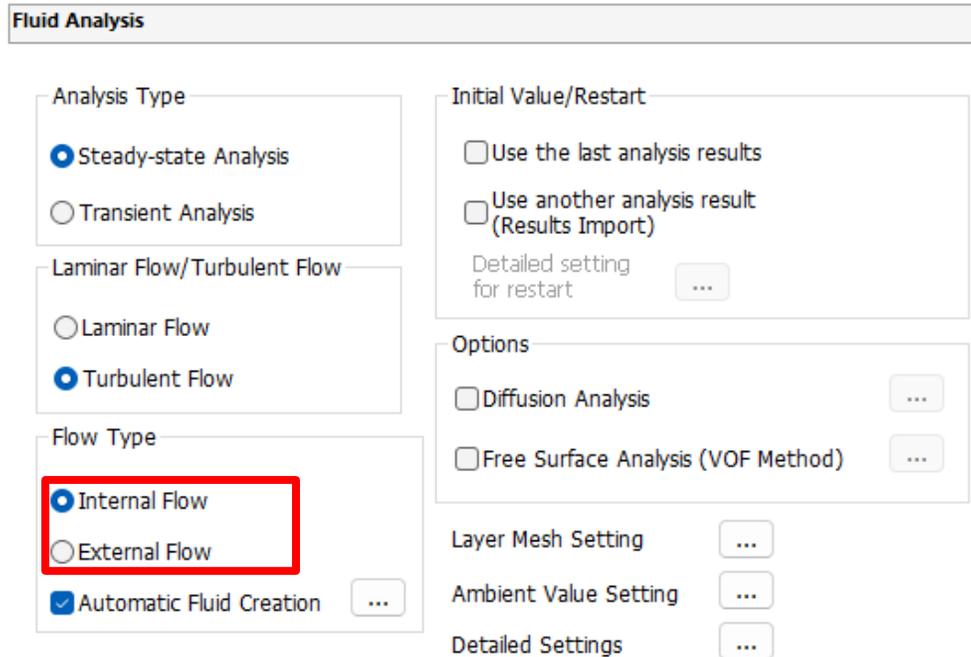


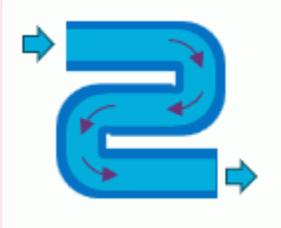
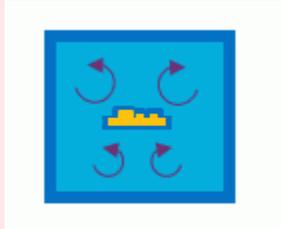
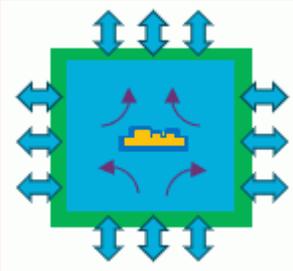
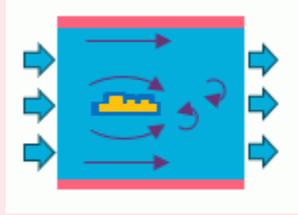
Compares between node-centered and cell-centered for 1000k elements

- 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.

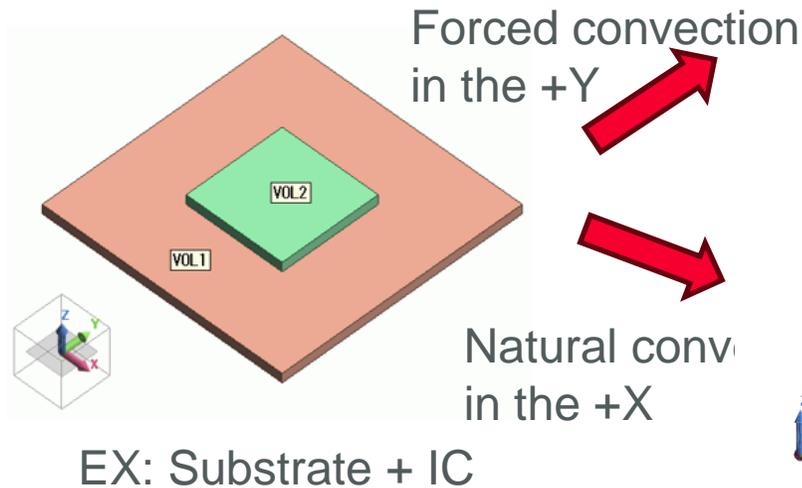
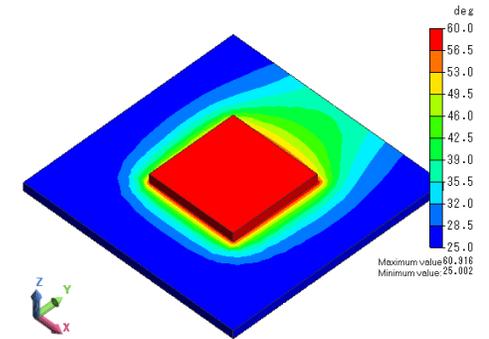
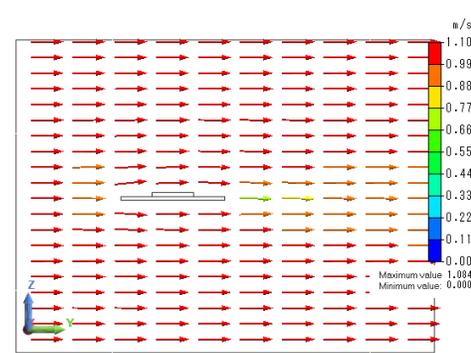
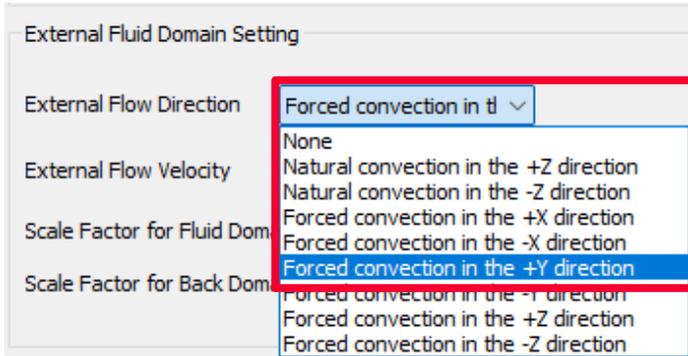
The screenshot shows the 'Fluid Analysis' settings panel. Under 'Flow Type', 'Automatic Fluid Creation' is checked. The diagram illustrates the automatic creation of a fluid domain from a solid pipe. The legend identifies the yellow area as the 'Solid Domain', the blue area as the 'Automatic Created Fluid Domain', and the green line as the 'Sheet body with boundary conditions set'. The 3D model shows the 'Pipe [solid]' and 'Case [Sheet]' components, with the 'Inside fluid' domain meshed in blue.

- 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.

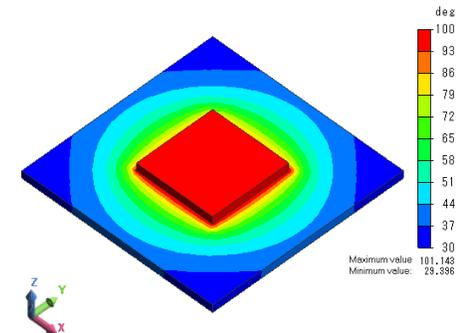
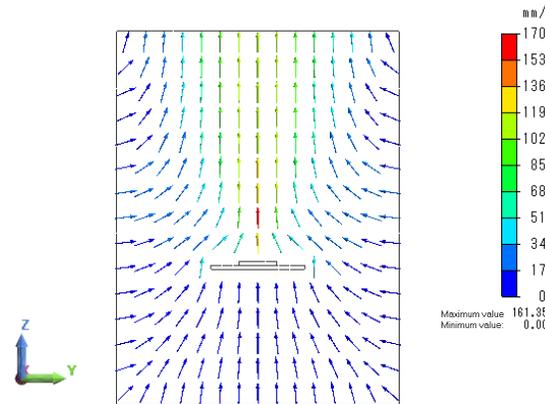
EX: Tutorial example-2

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



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.

Acoustic

Boundary Condition Type

Displacement Pressure

Speed Sound Pressure Level

Acceleration Rigid wall

Open boundary

Acoustic Impedance

Frequency Dependency

No

Yes

...

Use the specific acoustic impedance calculated from material

Specify the incident wave

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

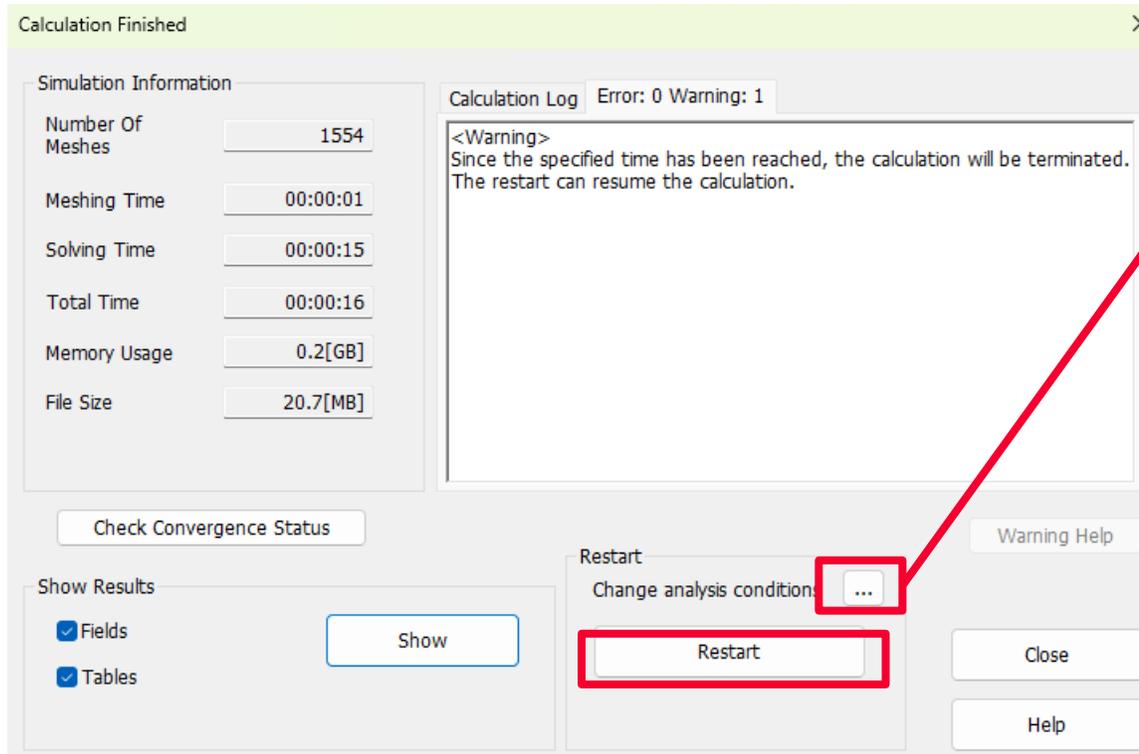
-It is given from the equation below using the material properties of medium:

$$\text{Specific acoustic impedance} = \text{Density} * \text{Sound speed}$$

* 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

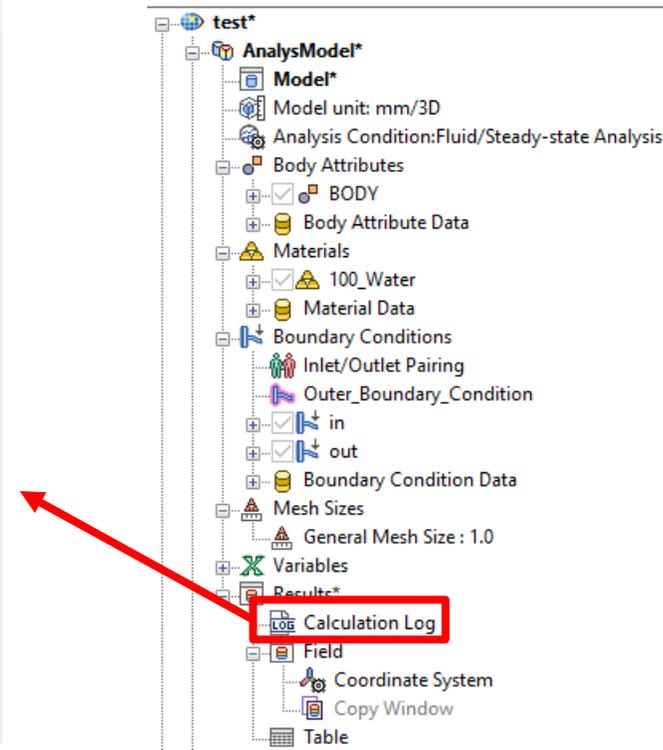
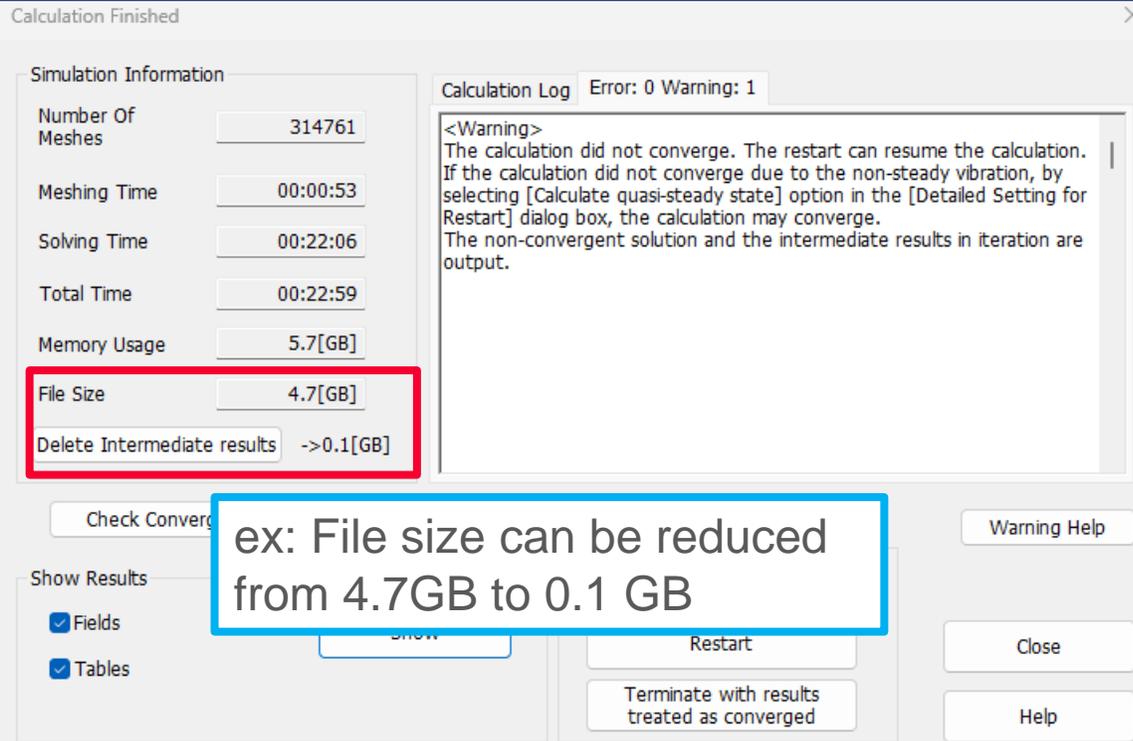
Suspended step in stress analysis

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.

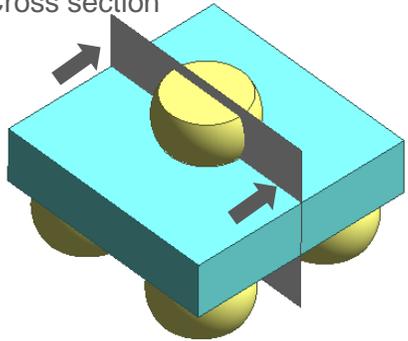


- In fluid analysis or stress analysis, intermediate results are saved for analyzing the cause in case of non-convergence/divergence.
- Since the size of the result file can be too large for calculation, you can delete the intermediate results after the analysis is complete
- After the result is saved, 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

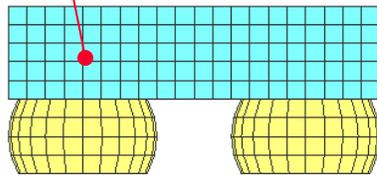
Sweep meshing now supports many-to-many models.

Cross section



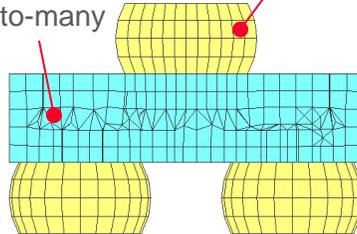
- Sweep mesh divides a face into tetrahedral elements and sweeps the face in parallel or rotationally to create hexahedral elements.
- Sweep mesh can create well-shaped hexahedral elements, but it is applicable only for shapes that can sweep.

many-to-one



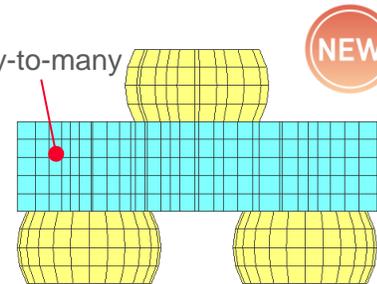
If there is no body above, a conventional many-to-one relationship is formed, allowing the use of sweep meshing.

many-to-many



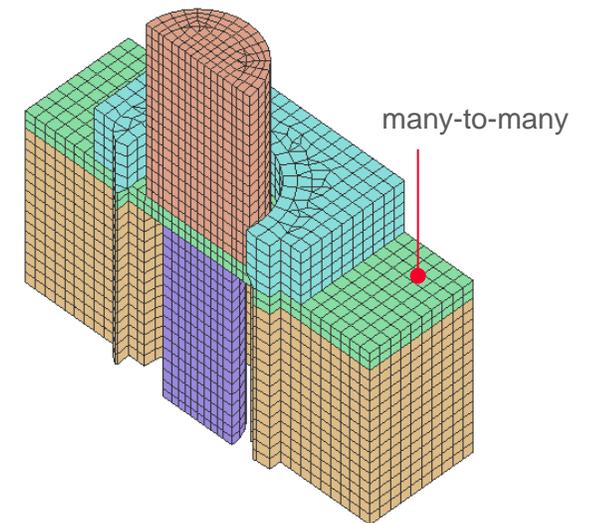
If there is a body above, a many-to-many relationship is formed, which prevents the application of sweep meshing and results in poor free meshes.

many-to-many



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

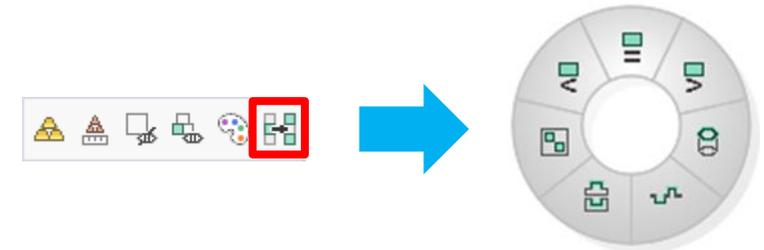
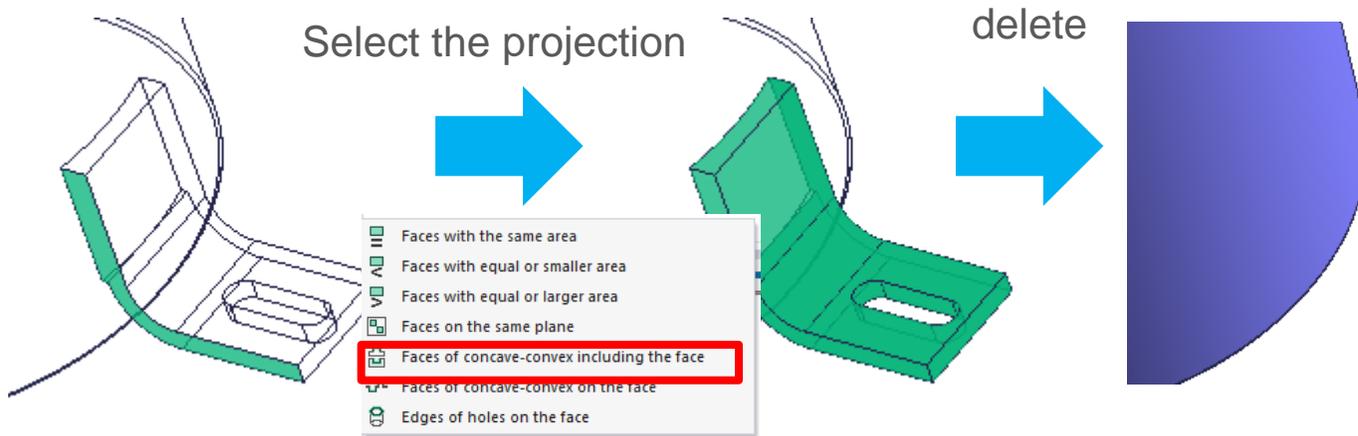
Example: Punching Die Mode



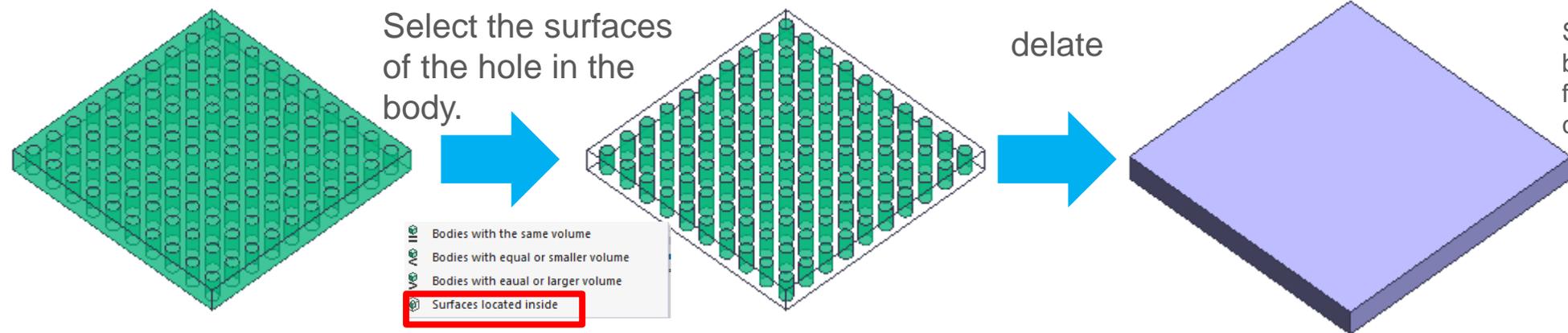
Model: Selecting function has been extended

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

Procedure example



Body	Surface	Edge
<ul style="list-style-type: none">Bodies with the same volumeBodies with equal or smaller volumeBodies with equal or larger volumeSurfaces located inside	<ul style="list-style-type: none">Faces with the same areaFaces with equal or smaller areaFaces with equal or larger areaFaces on the same planeFaces of concave-convex including the faceFaces of concave-convex on the faceEdges of holes on the face	<ul style="list-style-type: none">Edges with the same lengthEqual or shorter edgesEqual or longer edgesEdges included in a loop on the plane



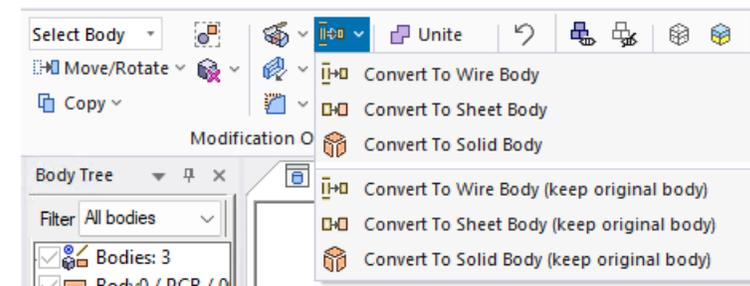
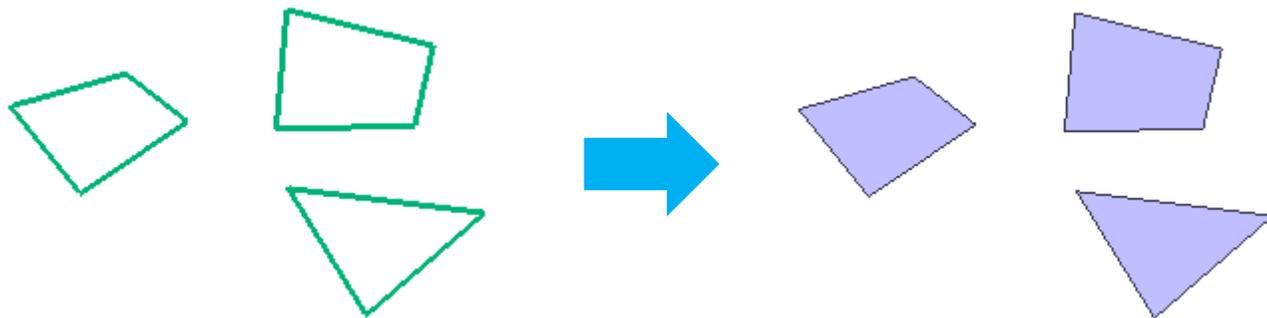
Select a body, face, or edge before selecting the functions from the mini toolbar or right-click menu.

Model: Convert Multiple Bodies Collectively

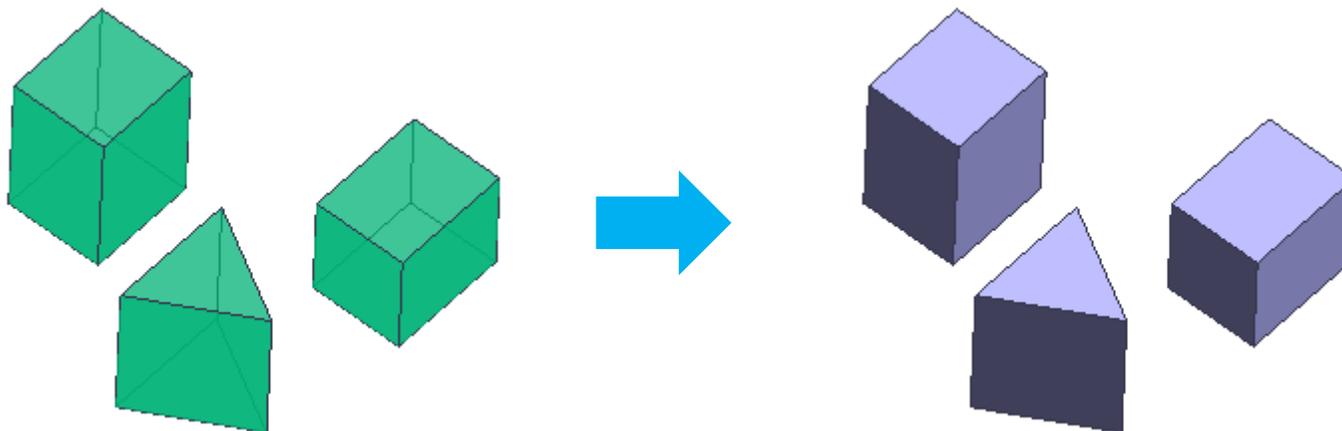
Wire/Sheet/Solid body converting has been applied to multiple bodies.

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

Conversion to Sheet Bodies



Conversion to Solid Bodies

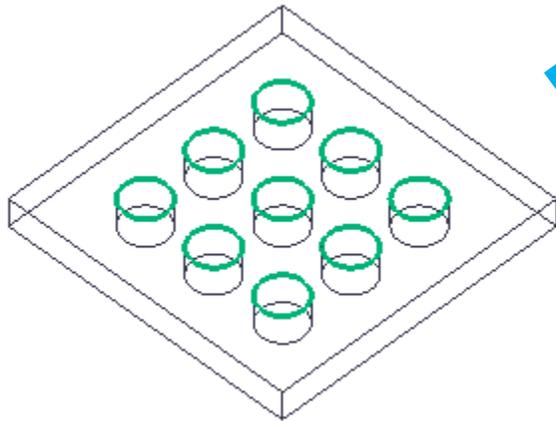


Model: [Convert] menu for topology to convert automatically.

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

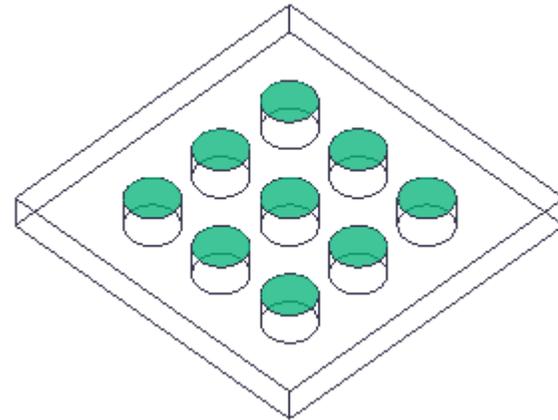
Example of Surface Conversion

Femtet2025.0
procedure

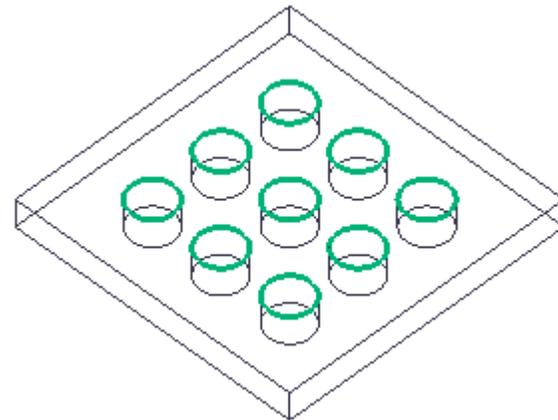


Select
Edges

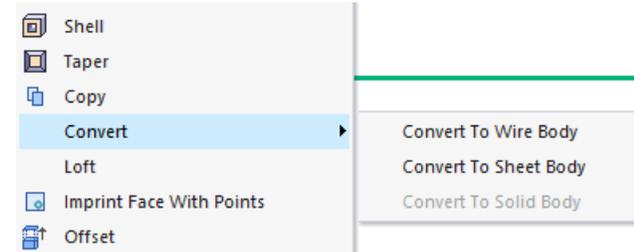
Previous
procedure



Convert to Sheet bodies



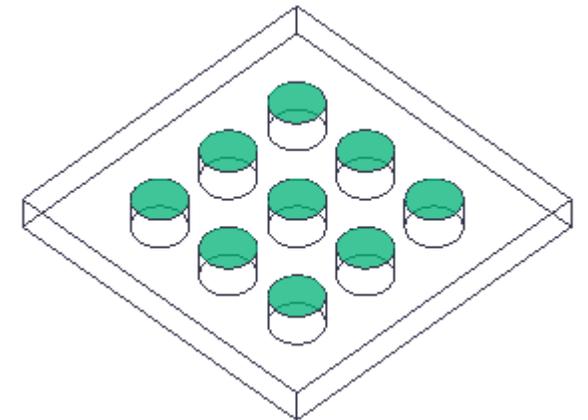
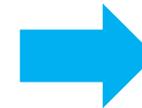
Duplicate edges -> Create wire bodies



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.

- Body11
- Body12
- Body13
- Body14
- Body15
- Body16
- Body17
- Body18
- Body19



Convert to Sheet bodies

Model: Model export function of Gerber data (RS274X)

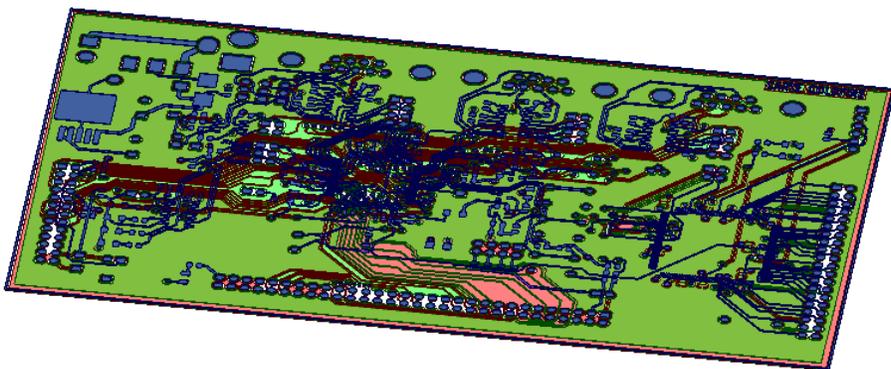
Added model export function of Gerber data (RS274X)

Femtet can export Gerber data (RS274X).

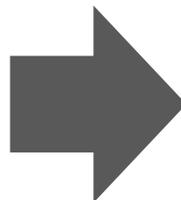
Classifying bodies into groups causes each group to be recognized as a layer, with the Gerber data split and output accordingly.

Femtet Model

- Bodies: 1459
- kit-dev-coldfire-xilinx_5213-Bottom_layer
- kit-dev-coldfire-xilinx_5213-Top_layer
- kit-dev-coldfire-xilinx_5213-VDD_layer

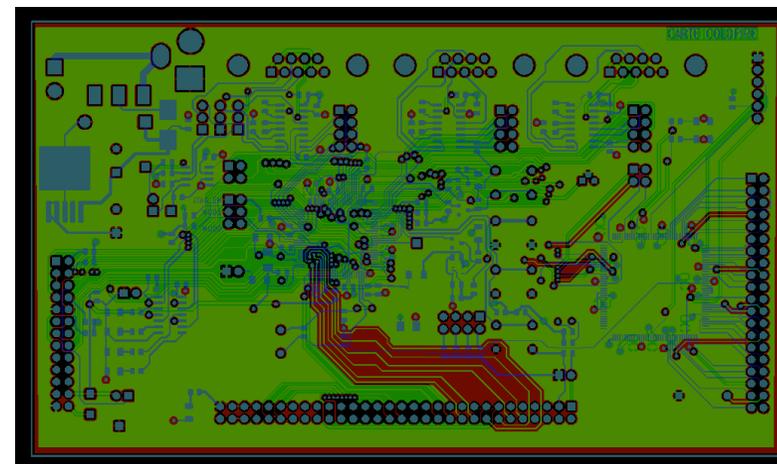


Export



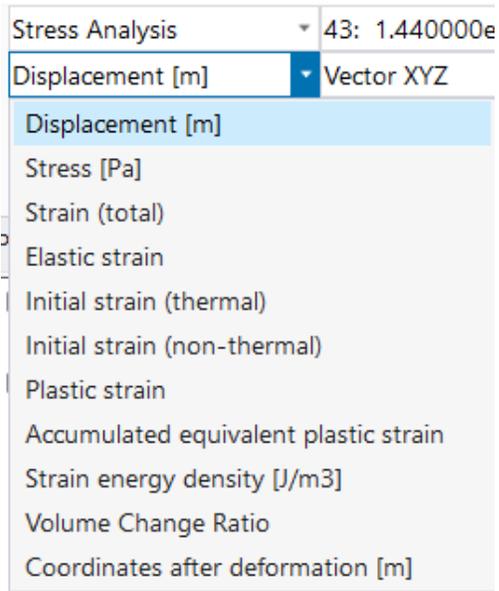
Gerber data files

-  output_kit-dev-coldfire-xilinx_5213-Bottom_layer.gbr
-  output_kit-dev-coldfire-xilinx_5213-Top_layer.gbr
-  output_kit-dev-coldfire-xilinx_5213-VDD_layer.gbr

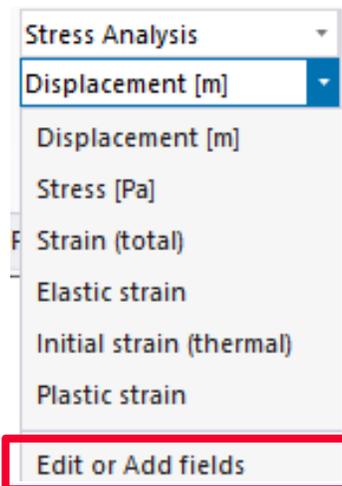


Result display: Edit or Add Fields

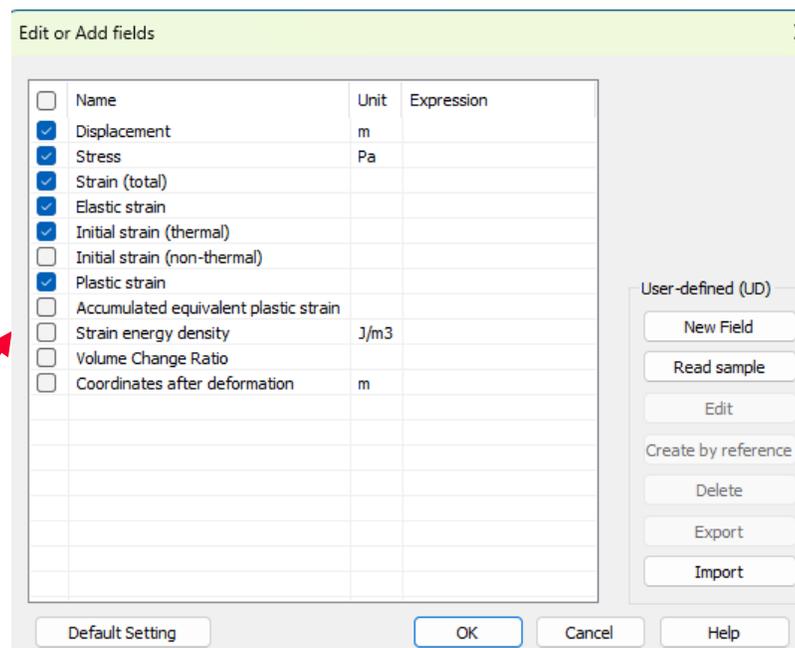
Allow for editing or adding fields to be displayed.



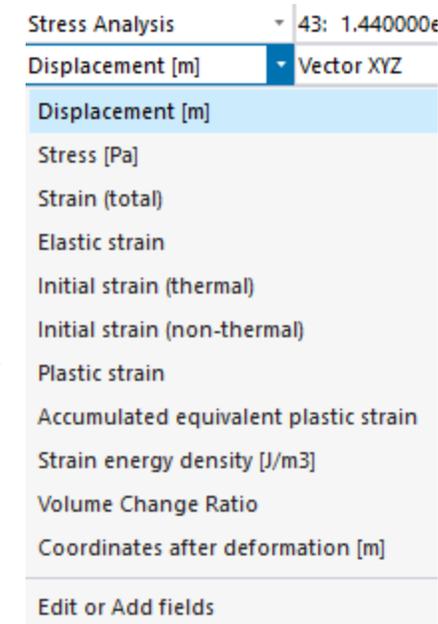
Initial Display in Ver 2024



Initial Display in Ver 2025



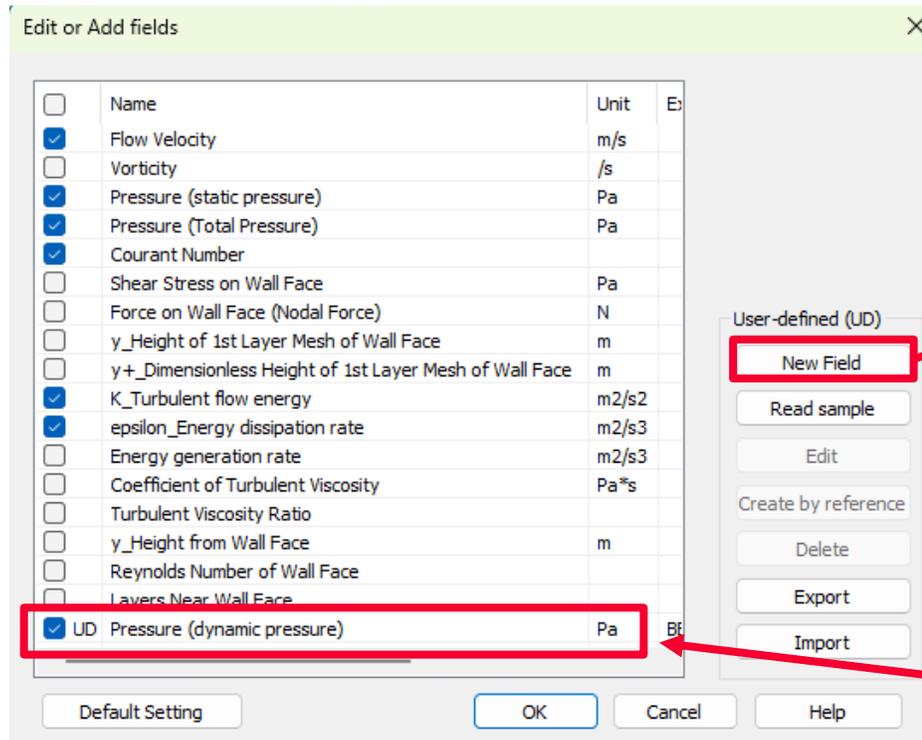
Field Editing or adding dialog box



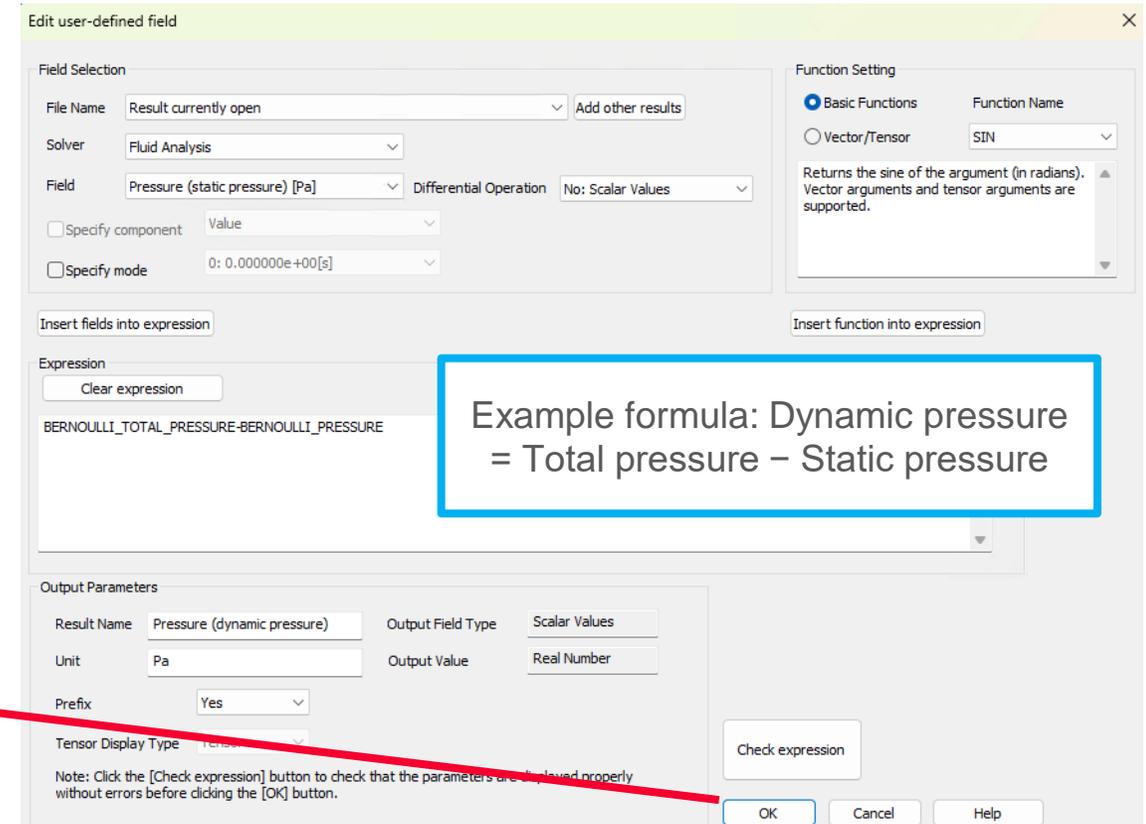
- Select [Edit or Add Fields] in the combo box to change fields to be displayed.
- Fewer fields are initially displayed for easy selection.
- Show or Hide is switched by the check box in the field editing or adding dialog box.
- New fields can be created by user definition and displayed.

Result display: Edit or Add Fields

[New Field] button is added.



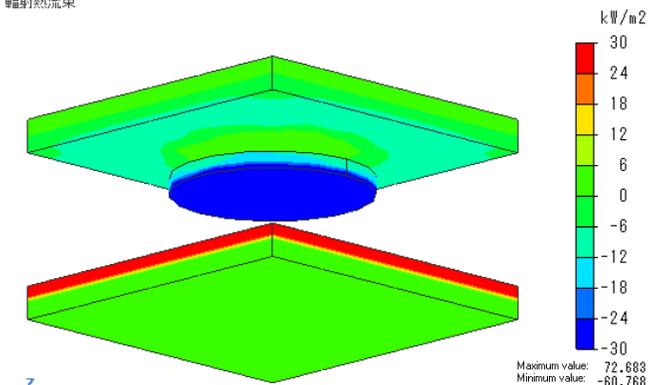
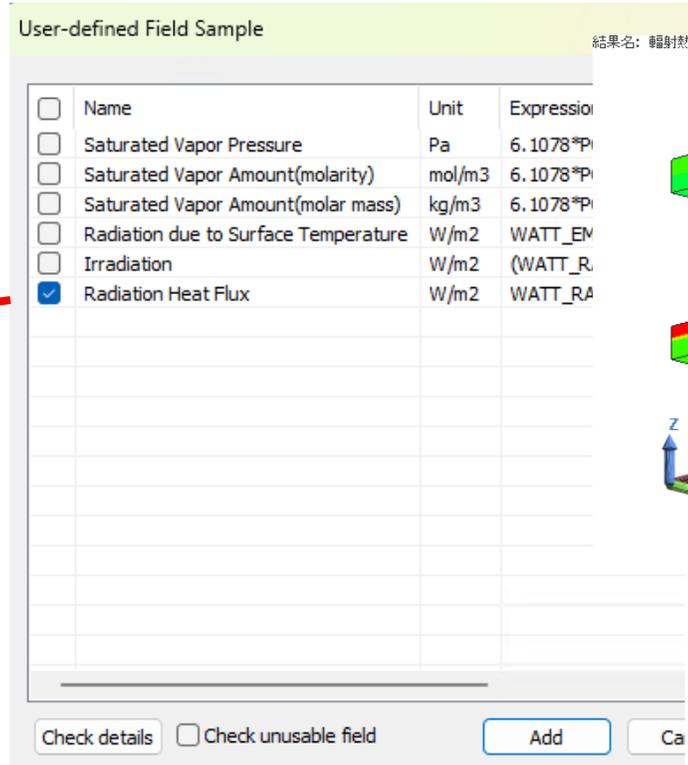
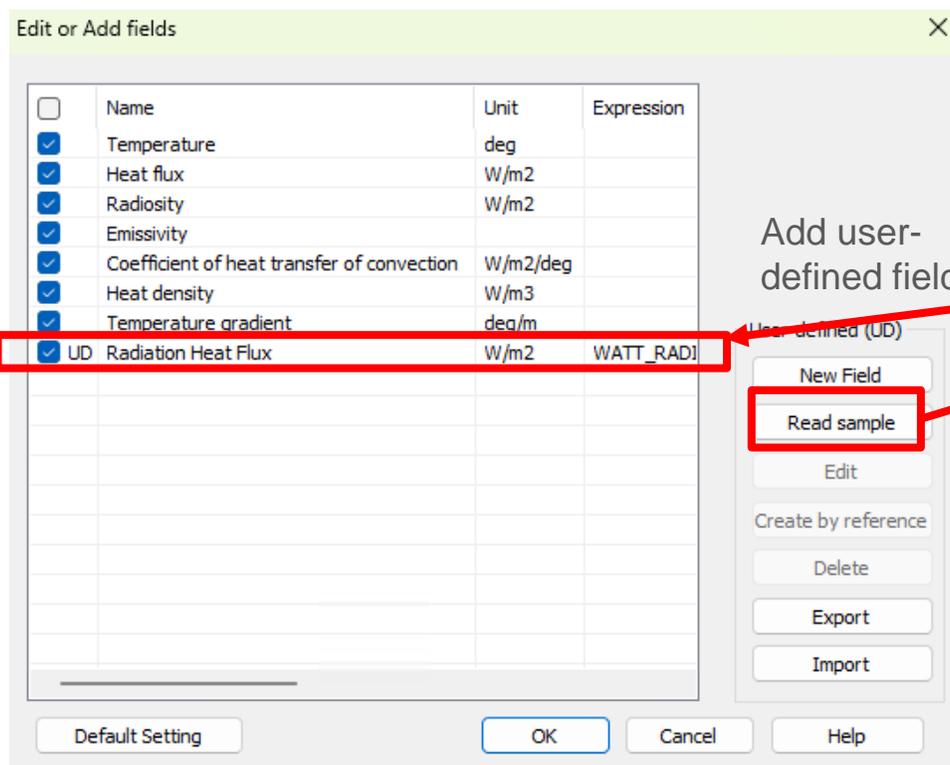
Add user-defined field



- 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

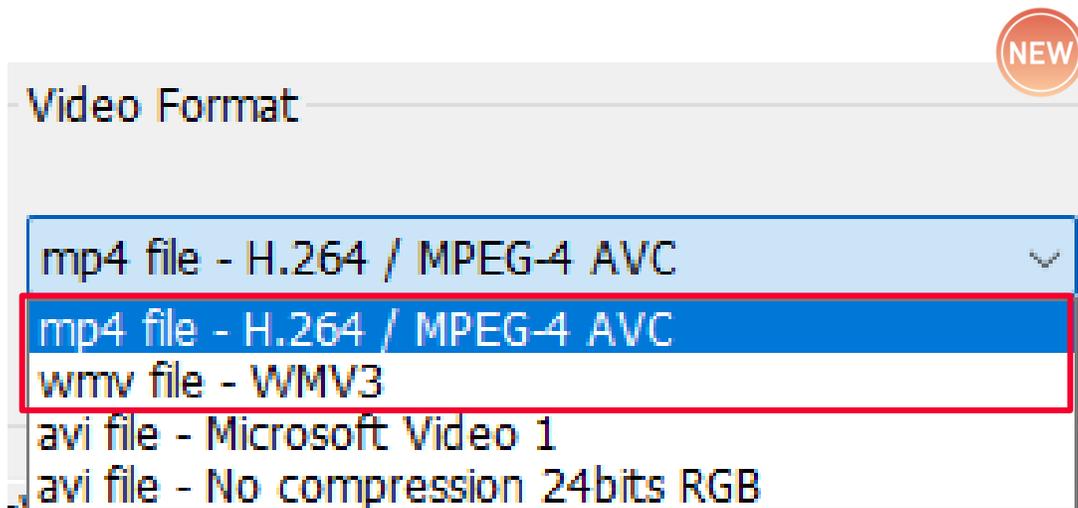
- Formula for calculating radiative heat flux from radiance, emissivity, and temperature.
- Formula for calculating the saturated water vapor amount from temperature.
- Formula for calculating engineering strain / Green-Lagrange strain / logarithmic strain from displacement.
- Formula for calculating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

Display result: Additional animation video export formats

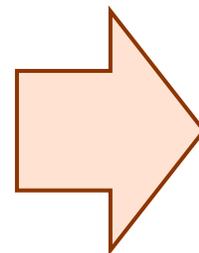
Additional animation video formats are supported.

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.



Export video



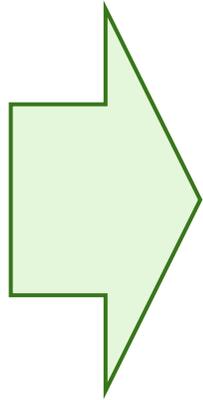
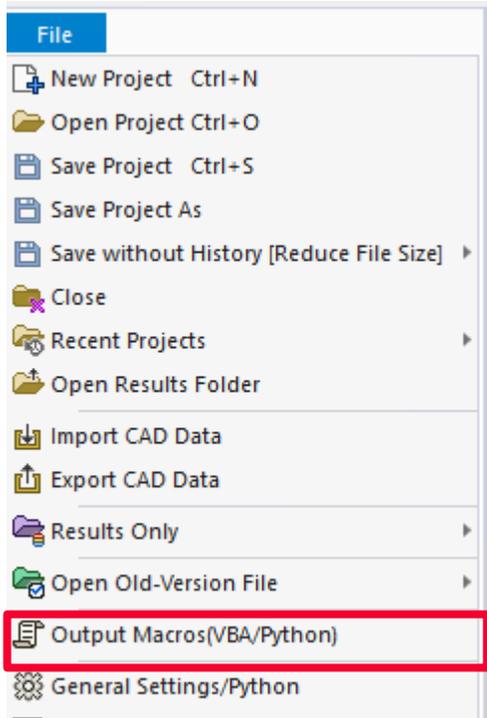
Name	Size
gal_ex01.Ex1.mp4	781 KB
gal_ex01.Ex1.wmv	510 KB
gal_ex01.Ex1.avi	4,157 KB
gal_uncompressed_ex01.Ex1.avi	85,323 KB

NEW

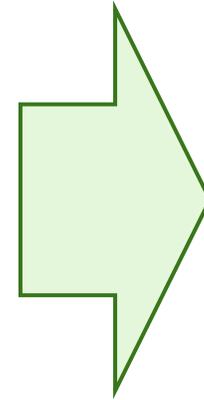
Compared to the traditional AVI file, the high compression allows for smaller file sizes -about 80% reduction.

Macro: Export Python script file

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



File name: test.py
Save as type: Python Script(*.py)



```
# 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.

```
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.

```
from pyfemtet.opt import FEMOpt, FemtetInterface

if __name__ == '__main__':
    # Femtetとの接続
    fem = FemtetInterface()

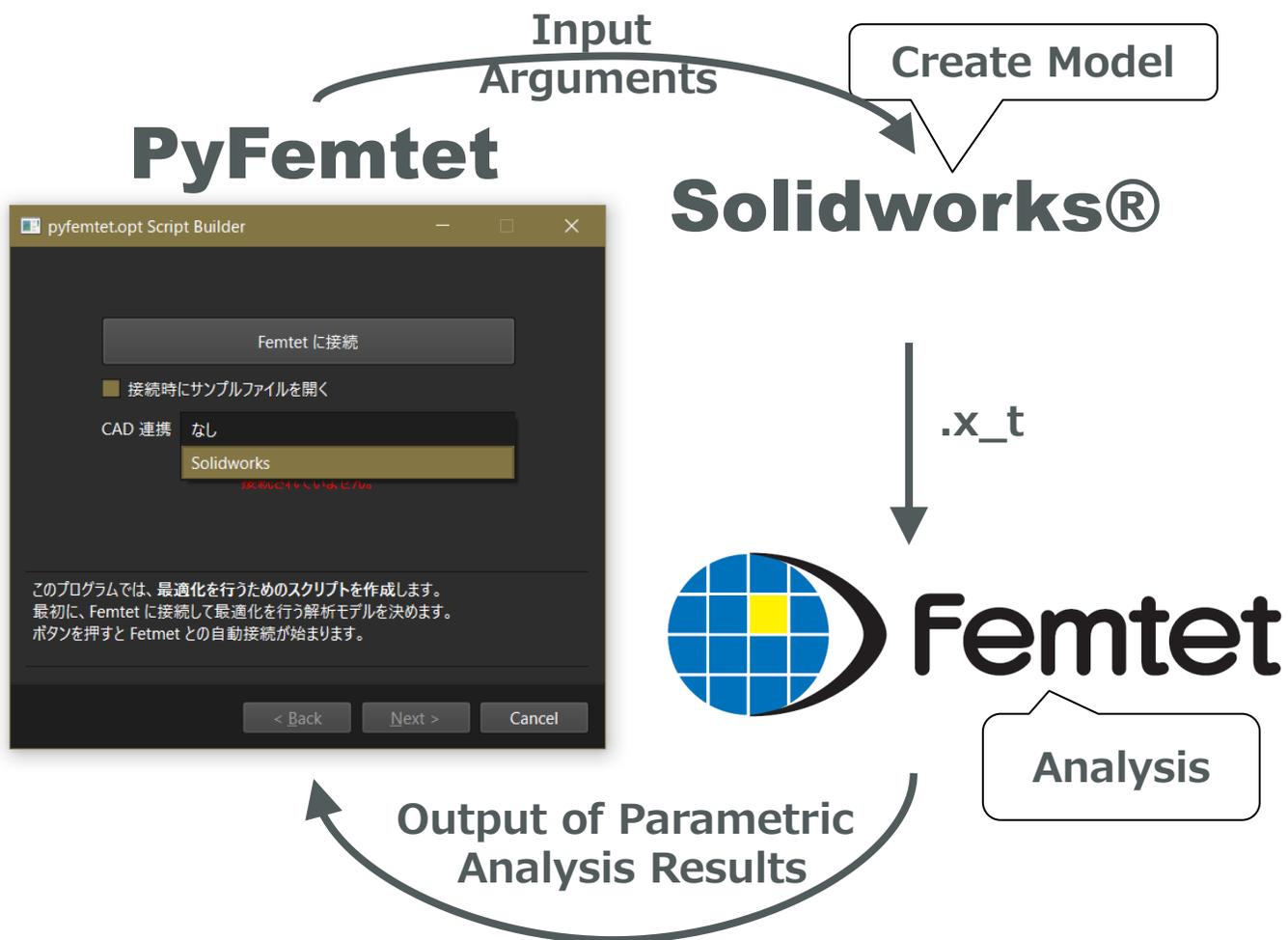
    # ミーゼス応力の出力を目的関数として使用 (Femtet のパラメトリック解析結果出力に設定しておく)
    fem.use_parametric_output_as_objective(number=1, direction='minimize')

    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.



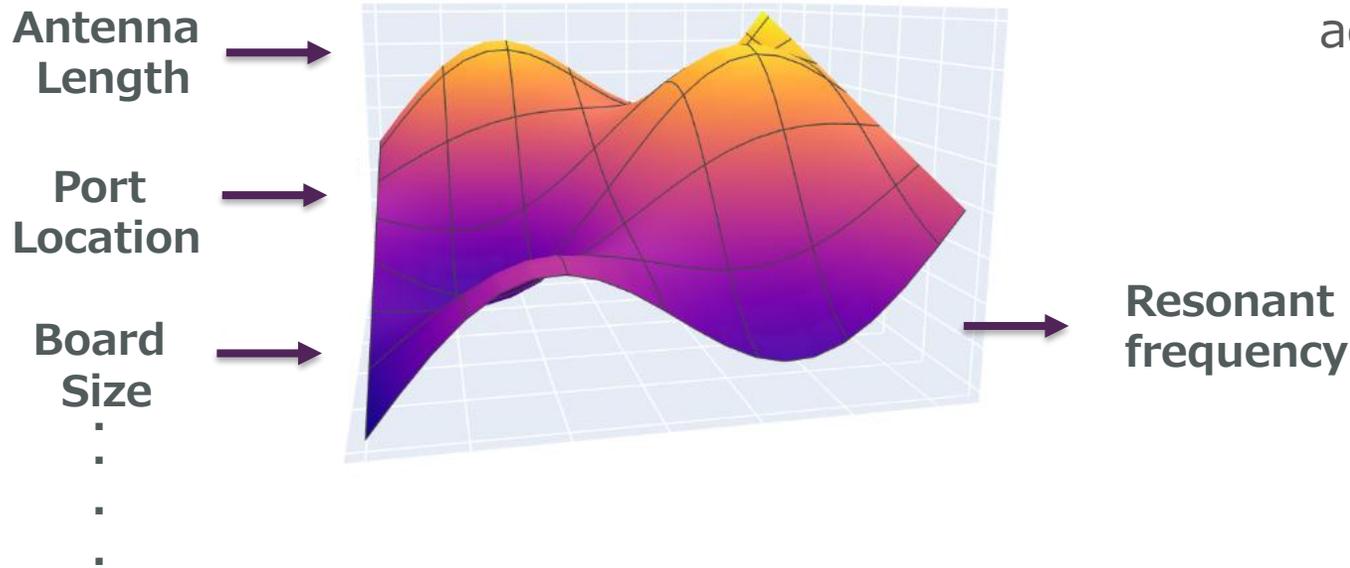
Scripts for utilizing Femtet's parametric analysis can be created easily.

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



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

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

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



Murata Software Co., Ltd.

muRata

