Femtet
Computer Aided Engineering System
Murata Software Co., Ltd.

Femtet Ver.2024.0 What's New

This document lists the new features added in Femtet2023.1 (only Japanese Version has been released) and Femtet2024.0 in order.
Femtet2024.0 English Version adds all these new features to Femtet2023.0 English Version.

Femtet Ver2023.1 What's New

Table of Contents

Functionality	What's New
Solver	

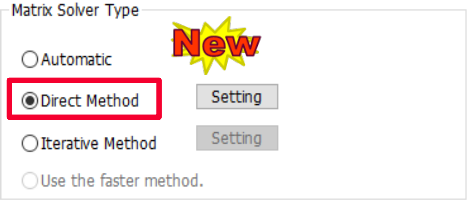
Table of Contents

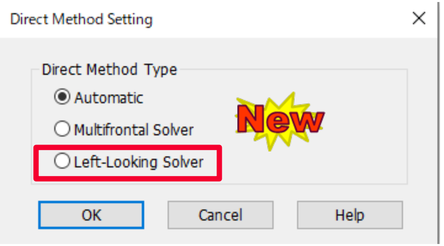
Murata Software

Functionality	What's New
Mesher	
Macro	

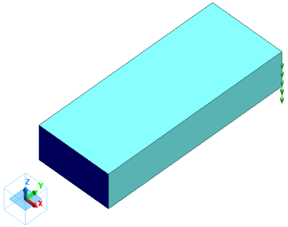
Solver in General: New Direct Solver

New direct solver enables faster analysis on high-performance PCs.

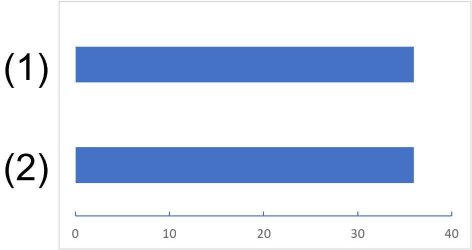




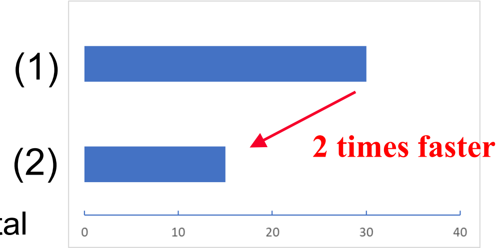
Example 1 in the Stress Analysis
110,000 elements



CPU:6core
(Xeon E-2176G)



CPU:38core
(Xeon W-3375)



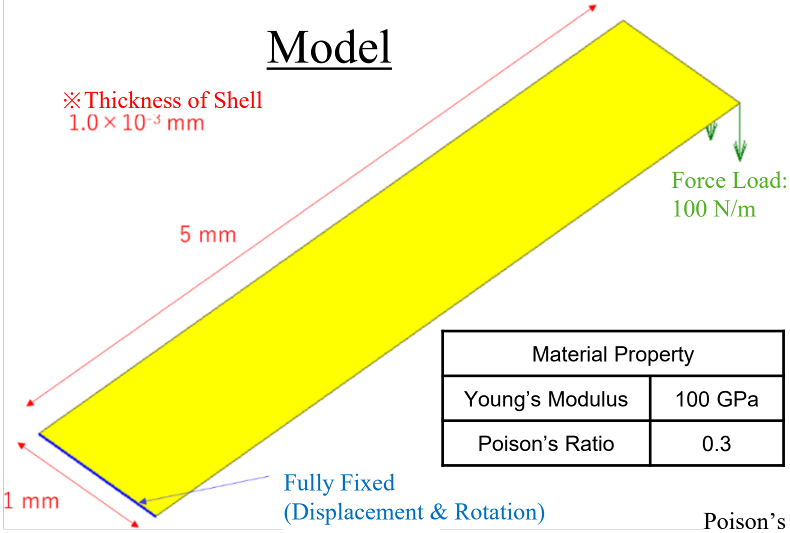
(1)Multifrontal
(2)Left-Looking

Solving Time (s)

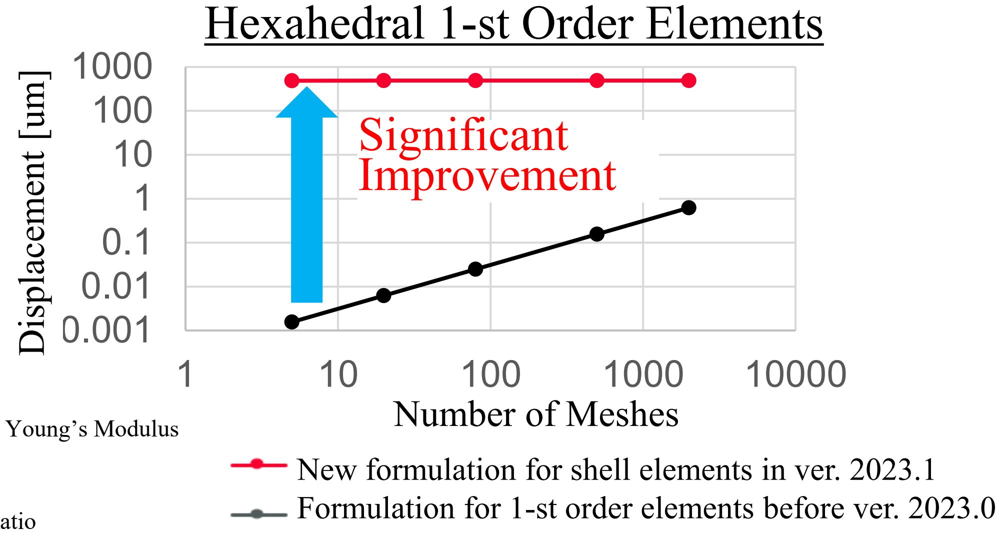
- In addition to the Multifrontal solver, which is the existing direct solver, the Left-Looking solver has been added.
- The Left-Looking solver with high parallel performance enables fast analysis on a high-performance PC with many cores.
- If a disk is used in the direct method, the Multifrontal is faster than the Left-Looking solver.
- Now the Multifrontal solver is automatically selected. In the future, the function to automatically set a proper solver for faster analysis will be available.

Solver: Stress
Analysis Accuracy for Shell Elements & Moment Load Boundary Condition

Analysis accuracy for shell elements has increased



Material Property	
Young’s Modulus	100 GPa
Poison’s Ratio	0.3

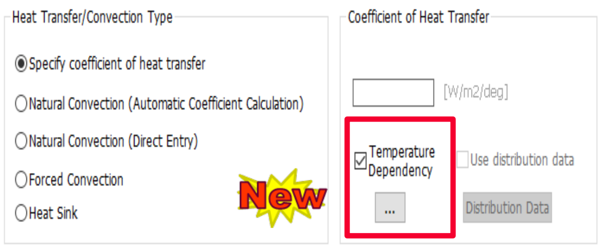


- Before ver. 2023.0, when the conventional formulation method was used for shell elements, due to shear locking, the analysis accuracy was lower with coarser meshes.
- In ver. 2023.1, the new formulation method has been introduced for shell elements, resulting in the improvement of analysis accuracy, in particular, for 1-st order elements.

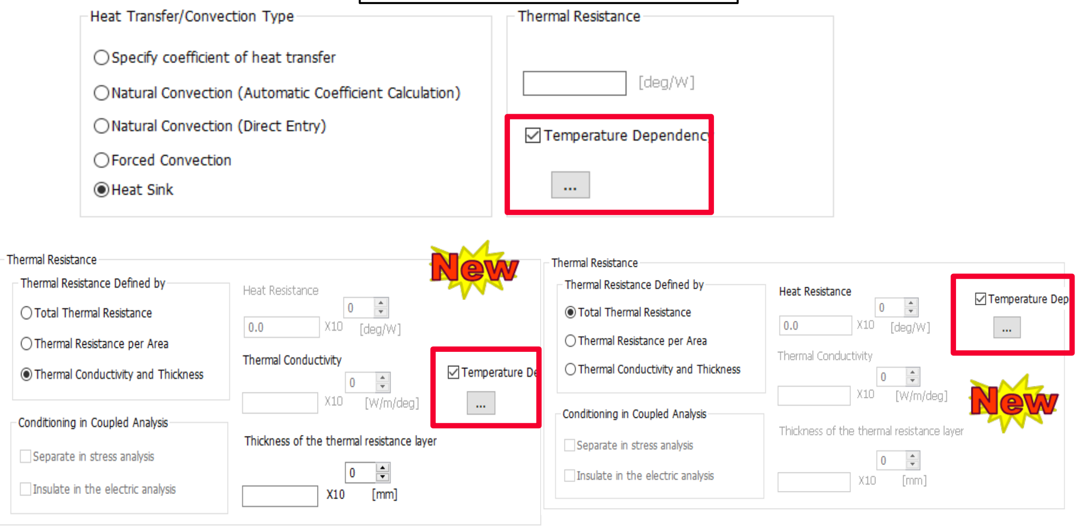
Solver: Thermal
Temperature Dependency for Coefficient of Heat Transfer and Thermal Resistance

Temperature dependency is applicable to the coefficient of heat transfer and thermal resistance boundary conditions.

Coefficient of Heat Transfer



Thermal Resistance



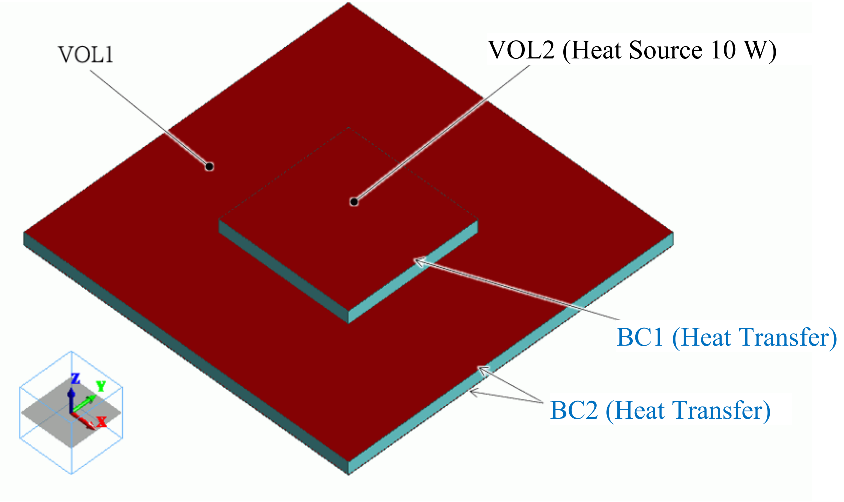
- Heat Transfer/Convection->Specify coefficient of heat transfer, then you can give temperature dependency to the coefficient of heat transfer.
- Heat Transfer/Convection->Heat Sink or Thermal Resistance->(Total Thermal Resistance, Thermal Resistance per Area, or Thermal Conductivity and Thickness), then you can give temperature dependency to thermal resistance.

Murata Software

New example in the thermal Analysis with temperature dependency.

Example 29: Water Cooling

- How to analyze water cooling is provided.
- Enter the coefficient of heat transfer that is calculated from the correction factor for water convection and has temperature dependency, in the table.

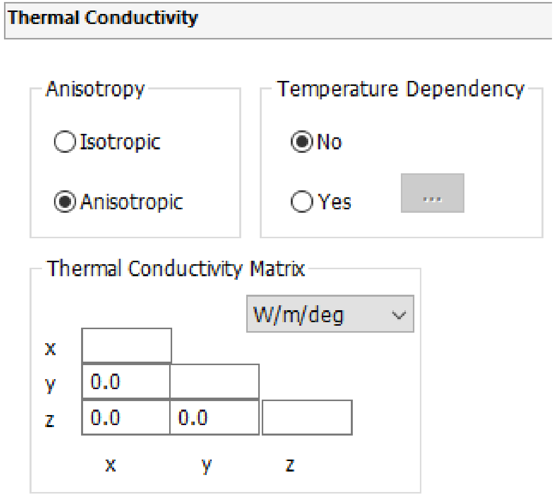


Solver: Thermal
Updated UI for Anisotropic Thermal Conductivity

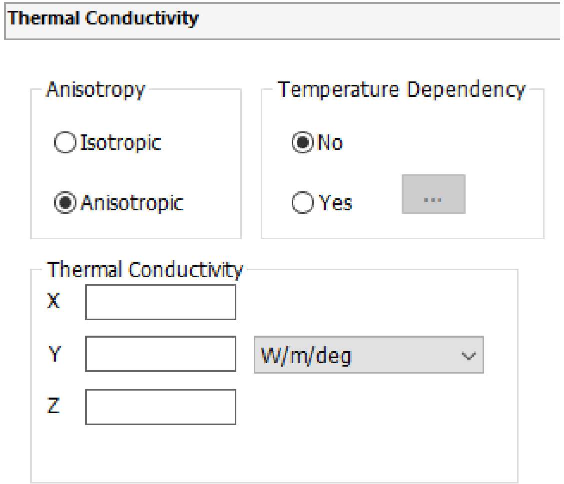
Murata Software

The format of the input UI for anisotropic thermal conductivity has changed from matrix to vector.

to Ver.2023.0



Ver.2023.1

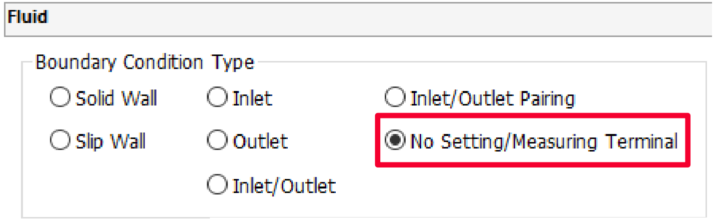


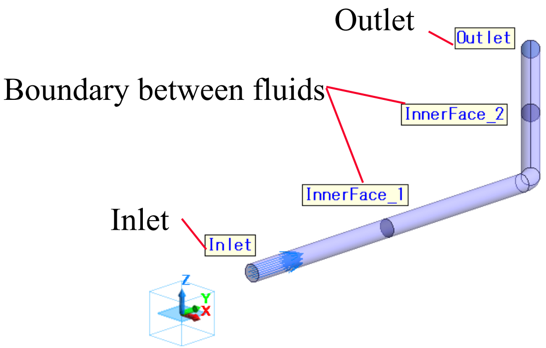
muRata

Solver: Fluid
More physical Propertied Saved in Table

Murata Software

Allows you to save flow rate and pressure at the [No Setting/Measuring Termina] boundary





Volumetric Flow Rate Output

New

Convergence status | FEM Info | Force on Wall Face [N] | Volumetric Flow Rate [m3/s]

	Value
Inlet	3.777e-2
Outlet	-3.777e-2
InnerFace_1	3.777e-2
InnerFace_2	3.777e-2

Pressure Output

New

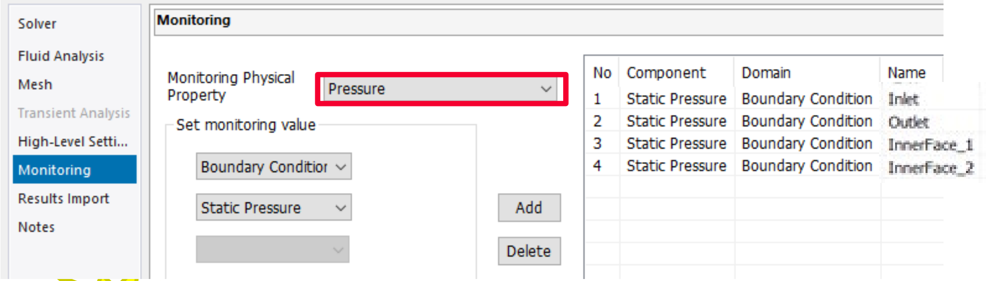
Convergence status | FEM Info | Force on Wall Face [N] | Volumetric Flow Rate [m3/s] | Pressure [Pa]

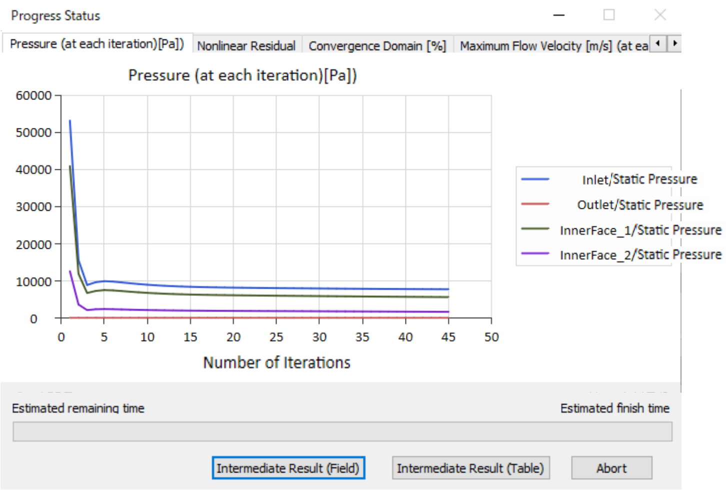
	Static Pressure [Pa]	Total Pressure [Pa]
Inlet	8.476e+3	2.094e+4
Outlet	0.000e+0	1.413e+4
InnerFace_1	6.100e+3	1.944e+4
InnerFace_2	2.060e+3	1.495e+4

- Previously, physical properties such as pressure were saved only at the inlet/outlet boundary.
- By setting the condition at the boundary between fluids, you can obtain the flow rate and pressure there.
- You can also obtain the pressure at portions that have flow boundary conditions.

Solver: Fluid
More Physical Properties Monitored

Additional physical properties can be acquired by monitoring function.





Monitored Pressure Value with Respect to Number of Iterations

New

- Velocity (X, Y, Z Components, Magnitude)
- Pressure (Total Pressure, Static Pressure)
- Force on Wall Face (X, Y, Z Components, Magnitude)
- Torque on Wall Face of Moving Wall
- Diffusion Value in Diffusion Analysis

- Flow rate, pressure, temperature, and diffusion value can be monitored at any point.
- In addition to temperature, some physical properties can be used to determine convergence when the values monitored have converged.

Solver: Electromagnetic/Acoustic Radiation Characteristics Saved in Result Table

Radiation characteristics such as maximum radiation gain of electromagnetic waves and maximum directivity of sound waves can be saved in the result table.

Electromagnetic Analysis: Efficiency, Maximum Gain, Average Gain, etc.
Acoustic Analysis: Maximum Value of Directivity, Average Value of Directivity.

Will be saved if all the following conditions are satisfied:

- Harmonic Analysis is selected in the electromagnetic analysis or acoustic analysis.
- Discrete Sweep or Parallel Discrete Sweep is selected.
- If Electromagnetic analysis, Select [Options] in the electromagnetic analysis dialog box and set the settings for directivity according to Fig. 1.
  If Acoustic analysis, Select [Calculate directivity when running solver] in the acoustic analysis dialog box.

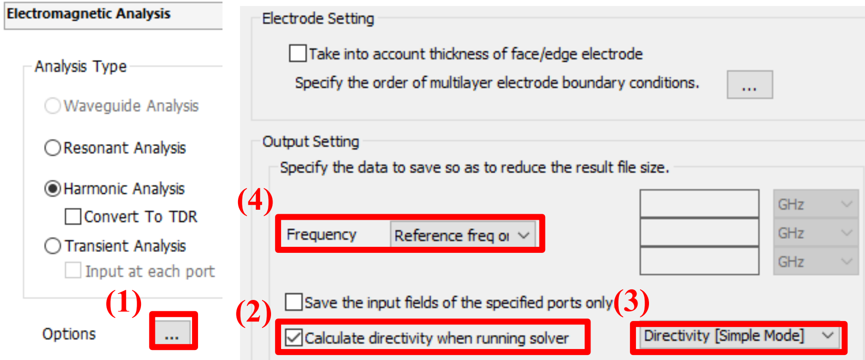
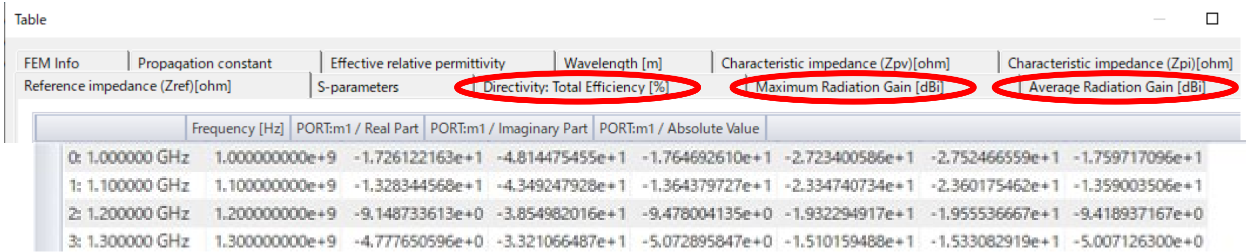


Fig. 1 The [Options] button on the Electromagnetic Analysis tab and the [Options for Electromagnetic Analysis] dialog box.



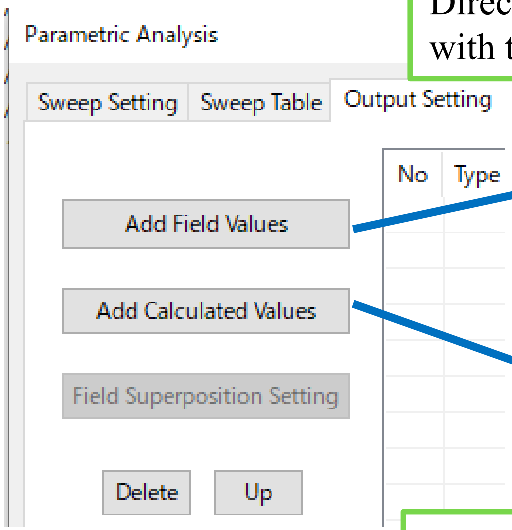
	Frequency [Hz]	PORT:m1 / Real Part	PORT:m1 / Imaginary Part	PORT:m1 / Absolute Value			
0: 1.000000 GHz	1.000000000e+9	-1.726122163e+1	-4.814475455e+1	-1.764692610e+1	-2.723400586e+1	-2.752466559e+1	-1.759717096e+1
1: 1.100000 GHz	1.100000000e+9	-1.328344568e+1	-4.349247928e+1	-1.364379727e+1	-2.334740734e+1	-2.360175462e+1	-1.359003506e+1
2: 1.200000 GHz	1.200000000e+9	-9.148733613e+0	-3.854982016e+1	-9.478004135e+0	-1.932294917e+1	-1.955536667e+1	-9.418937167e+0
3: 1.300000 GHz	1.300000000e+9	-4.777650596e+0	-3.321066487e+1	-5.072895847e+0	-1.510159488e+1	-1.533082919e+1	-5.007126300e+0

Fig.2 Result Table

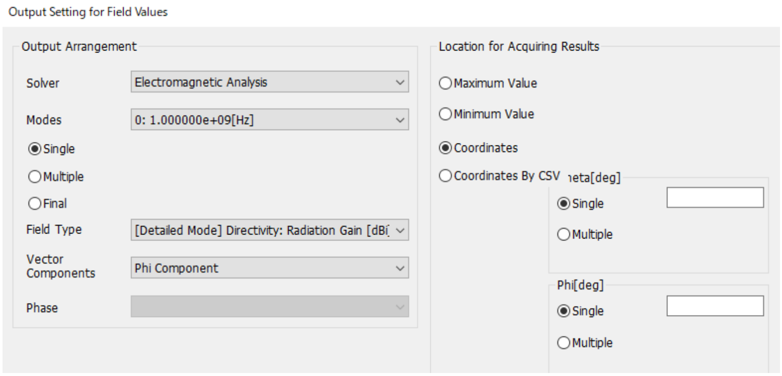
Solver: Electromagnetic / Acoustic
Radiation Characteristics Have Been Supported by Parametric Analysis

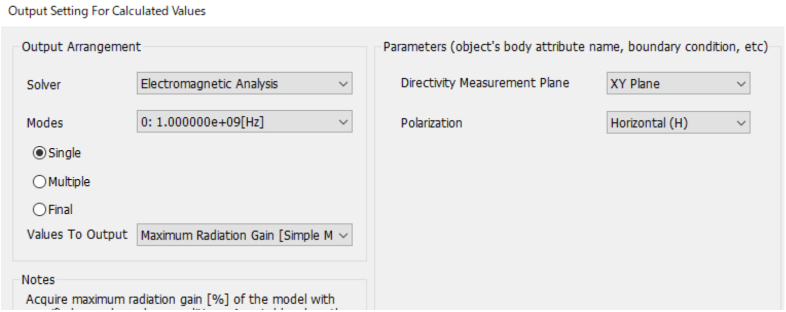
Murata Software

Radiation characteristics can be specified in the output setting of the parametric analysis.



Directivity can be obtained with the direction specified.





The values that are saved in the result table can be obtained in the parametric analysis.

muRata

Solver: Electromagnetic/Acoustic
Calculate Directivity When Running Solver

Murata Software

Allows you to graph radiation characteristics without opening the directivity calculation dialog box after calculation.

1. Starting from the Electromagnetic Analysis tab, set the parameters for directivity according to Fig. 1
2. Set the parameters for directivity according to Fig. 2 on the [Directivity [Detailed Mode]] tab.
3. Perform the analysis.
4. A file of the directivity graph is saved in the results folder as shown in Fig. 3.
5. Open the file of the directivity graph with Femtet's graphic app as shown in Fig. 4.

In addition to [Directivity [Detailed Mode]], you can operate as shown above for surrounding electromagnetic field and sound wave directivity. Not applicable for [Directivity [Simple Mode]].

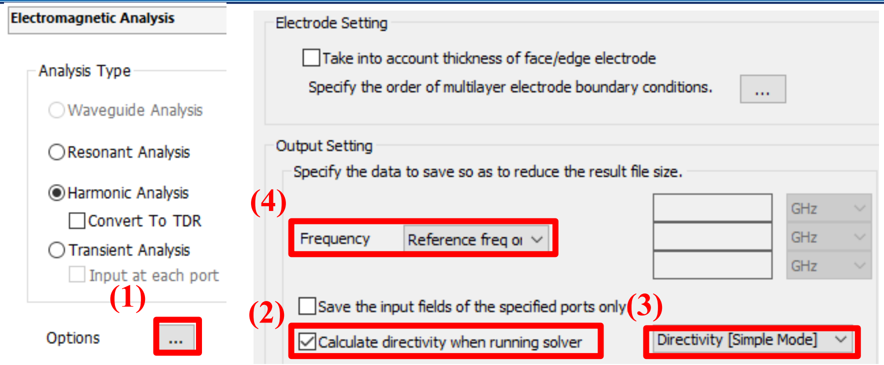


Fig. 1 The [Options] button on the Electromagnetic Analysis tab and the [Options for Electromagnetic Analysis] dialog box.

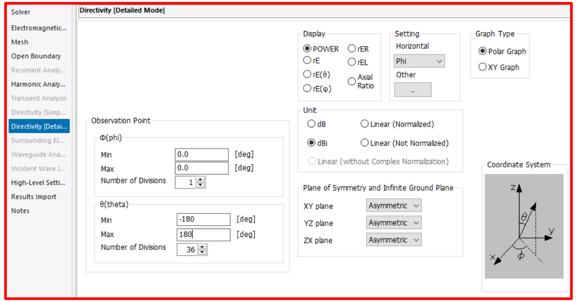


Fig.2 Setting on [Directivity [Simple Mode]] tab

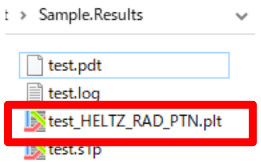


Fig. 3 Check the file
In the <Project_name>.Results folder, the directivity graph is saved in the file of <Analysis_mode>_HELTZ_RAD_PTN.plt.

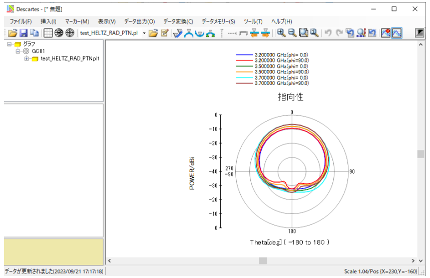
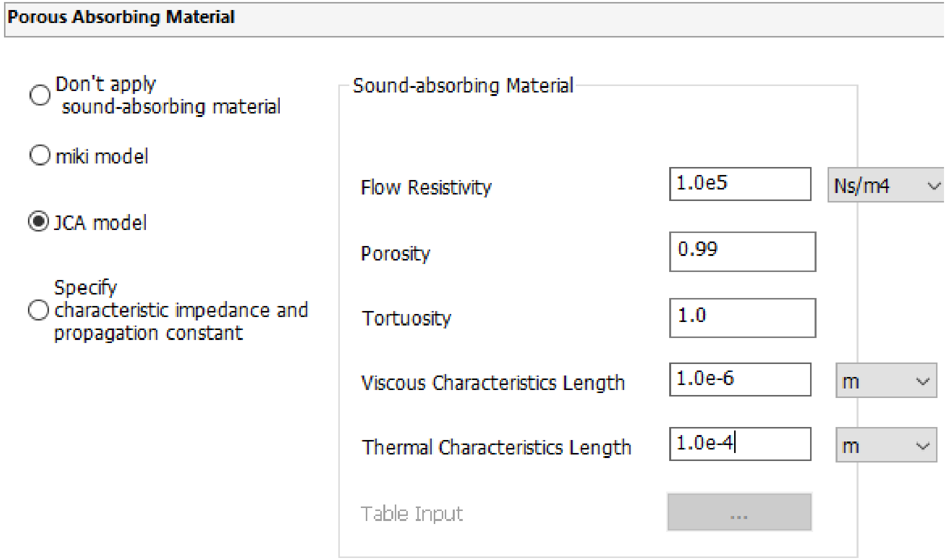


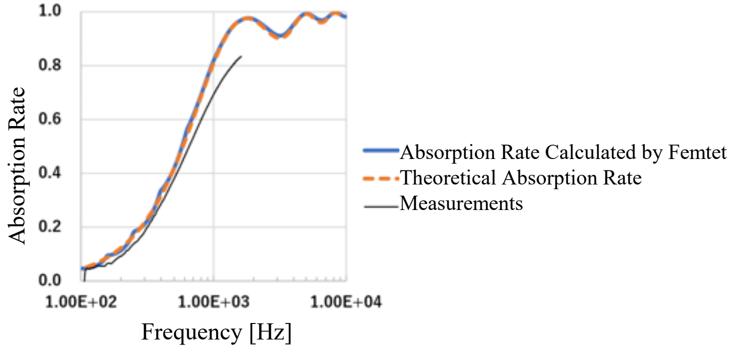
Fig.4 Directivity Graph Save in Result Folder

Solver: Acoustic
New Analysis Models for Porous Sound-Absorbing Materials

Murata Software

Allows you to use the miki and JCA (Johnson-Champoux-Allard) models.



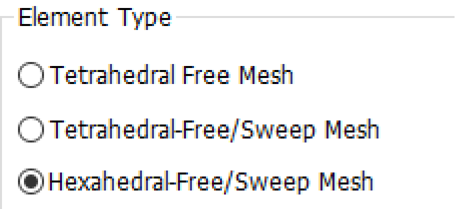


Result using the miki model

Refer to Example 14: Sound-absorbing Material (miki model) in the acoustic analysis.

Solver: Piezoelectric
Double Periodic Boundary Condition is Supported

Allows you to apply double periodic boundary conditions to sweep meshes.

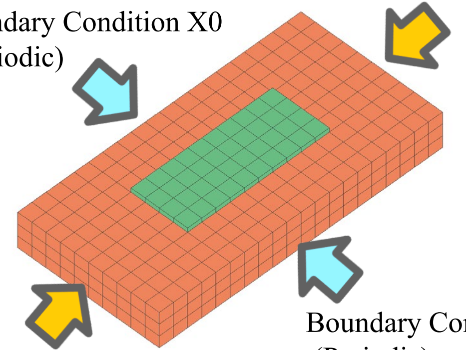


Boundary Condition: X0
Boundary Condition: X1
Phase Difference: 180 degrees

Boundary Condition: Y0
Boundary Condition: Y1
Phase Difference: 180 degrees

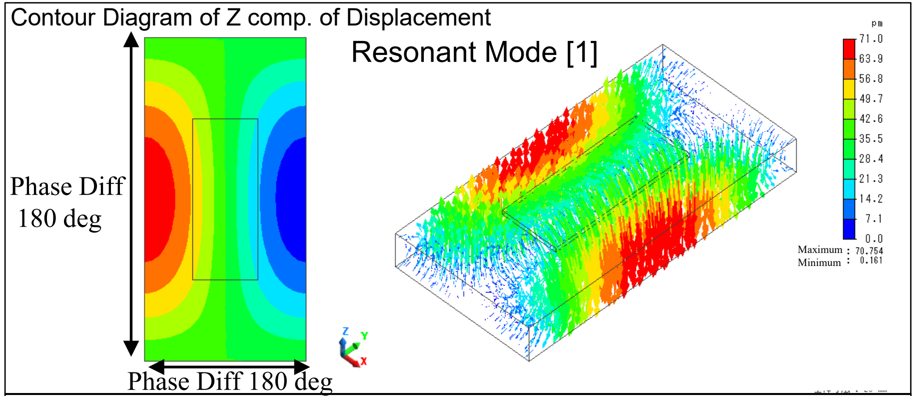
Boundary Condition X0 (Periodic)

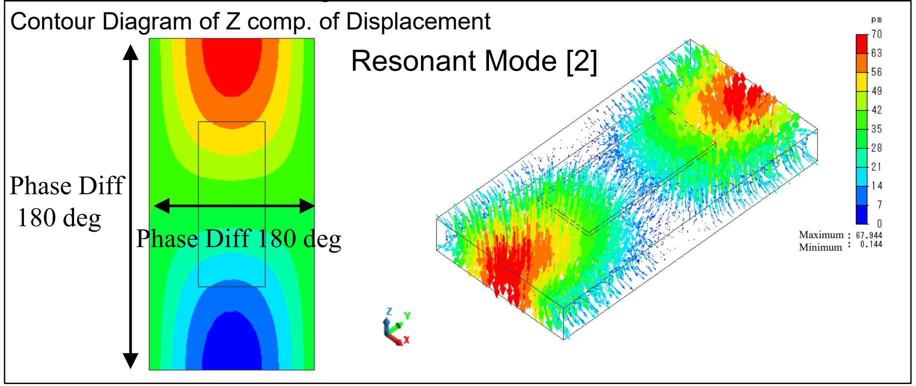
Boundary Condition Y1 (Periodic)



Boundary Condition Y0 (Periodic)

Boundary Condition X1 (Periodic)

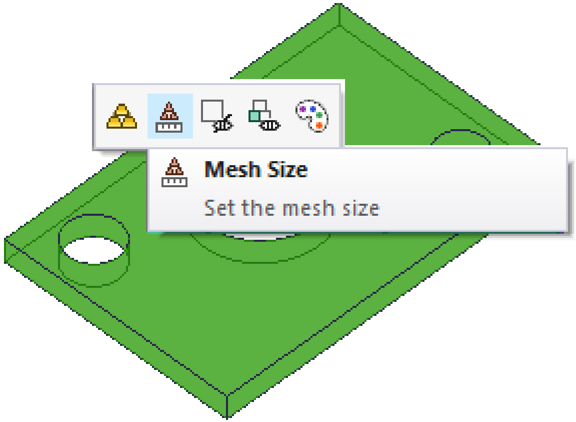


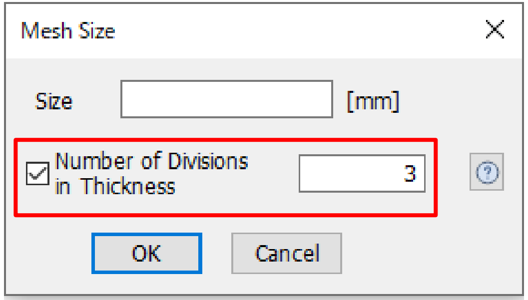


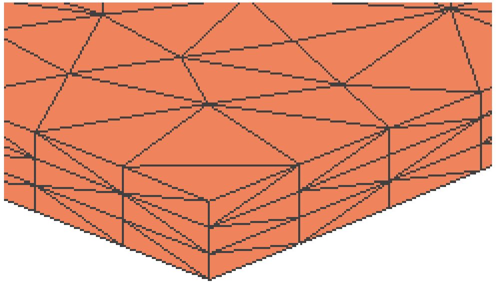
The phase difference between a pair of periodic boundary conditions is limited to 0 or 180 degrees, as usual.
Please refer to Femtet help for more information on sweep mesh.

Mesher:
Setting for Number of Divisions in Thickness Direction

Allows you to set the number of mesh divisions in the thickness direction of a thin plate.





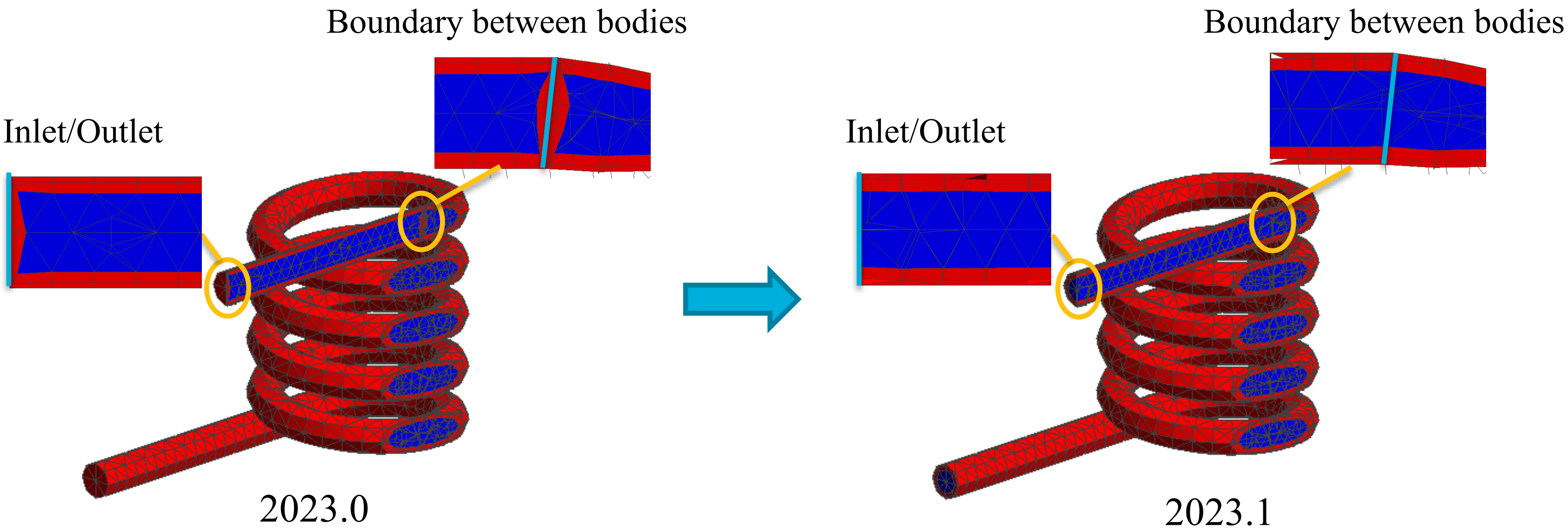


Select a body → Set the number of divisions in the mesh size dialog box. → Example Meshes (trisection)

- Divisible in the thickness direction without exceeding the necessary divisions.
- Applicable only to bodies in which the thickness direction can be determined.

Mesher:
Improved Skin Meshing

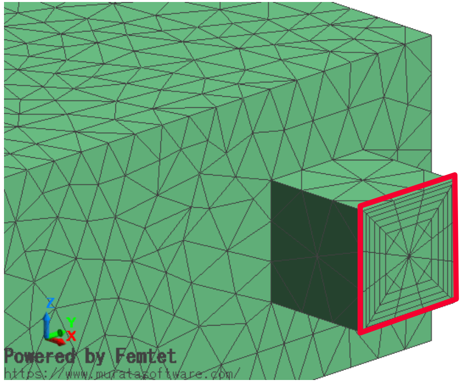
The skin meshing has improved.



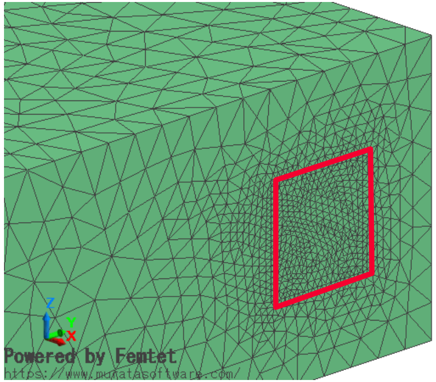
- No skin meshes have been created around either inlet/outlet or the boundary between bodies. (with some exceptions)
- The success rate of meshing and the quality of mesh have improved.

Mesher:
Layer Meshing Has Improved

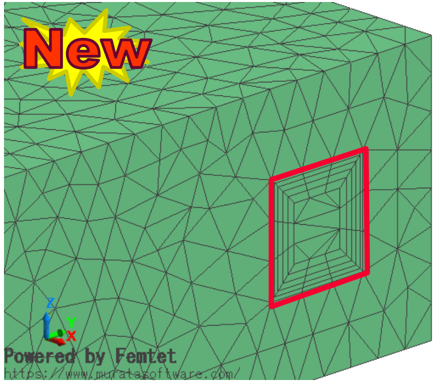
Mesh shapes on the not-convex inflow/outflow face Improved.



Inflow/Outflow Face on Convex Face



Inflow/Outflow Face on Flat Face.
Meshing with the option in ver. 2023.0



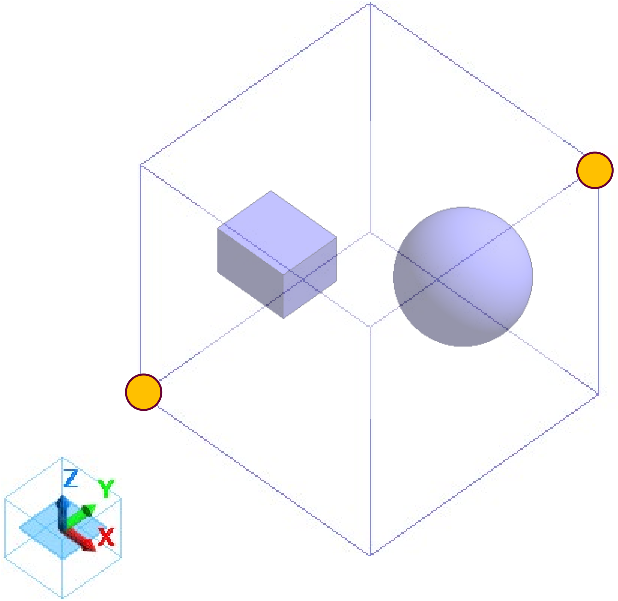
Inflow/Outflow Face on Flat Face.
Meshing in ver. 2023.1

- When the inflow/outflow face was not on the convex face, we tended to create coarse meshes on the inflow/outflow face. To avoid it, we used the option to refine meshes according to the size of faces.
- Due to the new version, the shapes of meshes obtained on the not-convex inflow/outflow face have been similar to those obtained on the convex inflow/outflow face. According to this improvement, the option has been eliminated.

Macro:
Method to Obtain Body Topologies within Size-Specified Box.

Murata Software

The method allows you to specify the box (size box) defined by two diagonal coordinates and search for body topologies within that range



- Specifys the two points ◯ by coordinates and calls the method, allowing you to search for body topologies in the specified range.
- You can search for them in either the modeling or results window.

[In the modelling window]

CGaudi.FindXXXAllByBox method

[In the results window]

CGogh.FindXXXAllByBox method

※XXX can be any of the following words:

Body, Topology, Vertex,

Edge, Face

Example: Search for edge topologies in the results window.

CGogh.FindEdgeAllByBox method

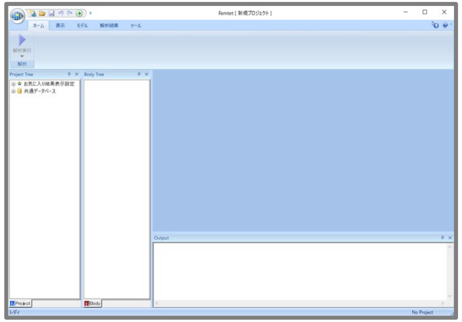
Macro:
Femtet Utility Package by Python

Python script can use the method to start or terminate Femtet.

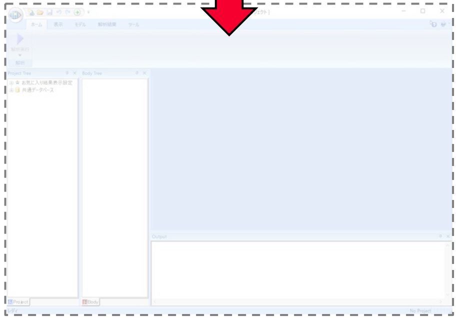
Python Script

```
import sys
from win32com.client import Dispatch,
constants
from femtetutils import util, const

util.auto_execute_femtet()


Femtet = Dispatch(const.CFemtet)
if Femtet.OpenNewProject() == False:
    print(Femtet.LastErrorMsg)
    sys.exit()


util.close_femtet(Femtet.hWnd)
```

Start up Femtet



Terminate Femtet with macros



Macro:
Femtet Utility Package by Python

Murata Software

The definition of classes for Femet interface has been moved to the utility package.

to Ver.2023.0

1. Download a sample script from the macro help.
2. Copy the attached file, FemetClassConst.py, to the same folder as the folder where your intended script will be stored.
3. Import the file into your intended script for use.

```
from FemtetClassConst import FemtetClassName as const
Femtet = Dispatch(const.CFemtet)
```

Ver.2023.1

1. Download the installer of the utility package from the macro help to install the utility package. (First time only)
2. Import the package into your intended script for use.

```
from femtetutils import util, const
from win32com.client import Dispatch, constants

util.auto_execute_femtet()
Femtet = Dispatch(const.CFemtet)
```

muRata

Macro:
Femtet Utility Package by Python

Murata Software

Miscellaneous

- When the utility package is installed, if the pywin32 package, required for running Femtet macro, has not been installed in the phyton environment, that package will be installed automatically.
- The macro help originally describes how to activate the makepy tool from the OS's startup menu to prepare the operating environment for executing Femtet macro from Python.
- The way to activate the makepy tool has changed to entering the command: python -m win32.com.client.makepy FemtetMacro.

Femtet Ver2024.0 What's New

Murata Software

Table of Contents

Functionality	What's New
Solver	

muRata

Table of Contents

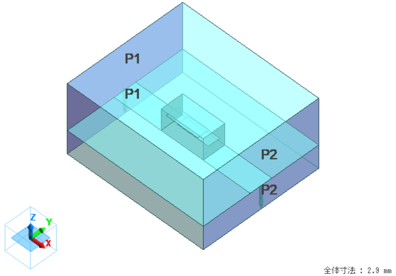
Murata Software

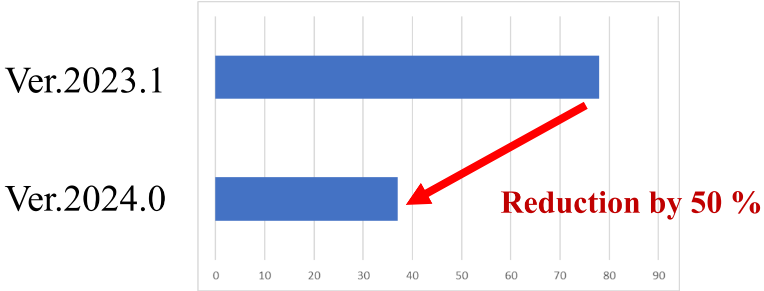
Functionality	What's New
Solver	• Acoustic: New Option [Specify the incident wave] for Speed Boundary Condition • Thermal & Simple Fluid: Algebraic Multigrid Method Applied Widely for fast Calculation • Thermal: New Function to Automatically Calculate Correction Factor for Convection
Mesher	• Faster Calculation • Improved Layer Meshing
UI	• Icons of File Menu, Ribbons, and Tree Changed in Design and Coloring. • Improved User Interface in Working Windows • New Table Format of CSV Files from Parametric Analysis Results • Improved Output Setting for Results of Parametric Analysis
Results Display	• Stress: Improved Display of Analysis Results for Shell Elements

Solver in General: Faster Default Direct Solver

Allows for fast calculation with the default direct solver with high parallel performance

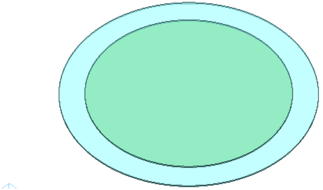
Example 18 in Electromagnetic Analysis
Apply adaptive meshing

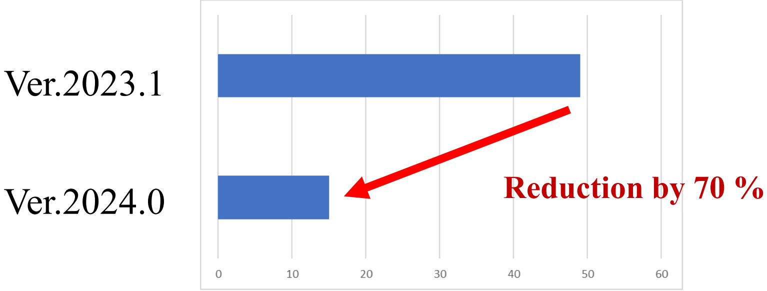




Total Simulation Time (s)

Example 12 in Piezoelectric Analysis
Discrete Sweep





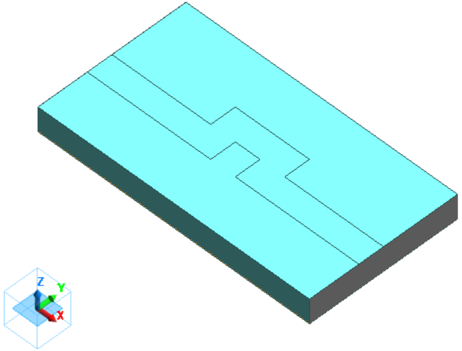
Solving Time (s)

Calculate with CPU: Xeon W-3375 (38core)

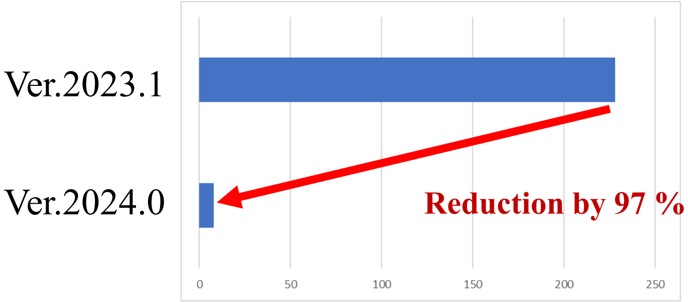
Solver in General: Faster Process to Import Results with Different Mesh Sizes

The process to import the result file having different mesh sizes is improved and accelerated.

Example 14 in Thermal Analysis



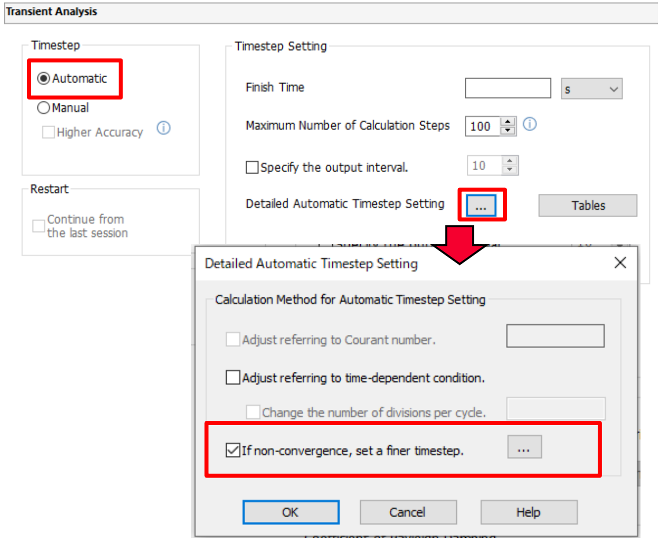
Import from 370000 mesh model to 270000 mesh model



Import Time (s)

Solver in General:
Automatic Timestep Setting Based on Time-Dependent Conditions

Automatic timestep setting based on a time-dependent condition is available.



For each solver, the time-dependent conditions can be applied?

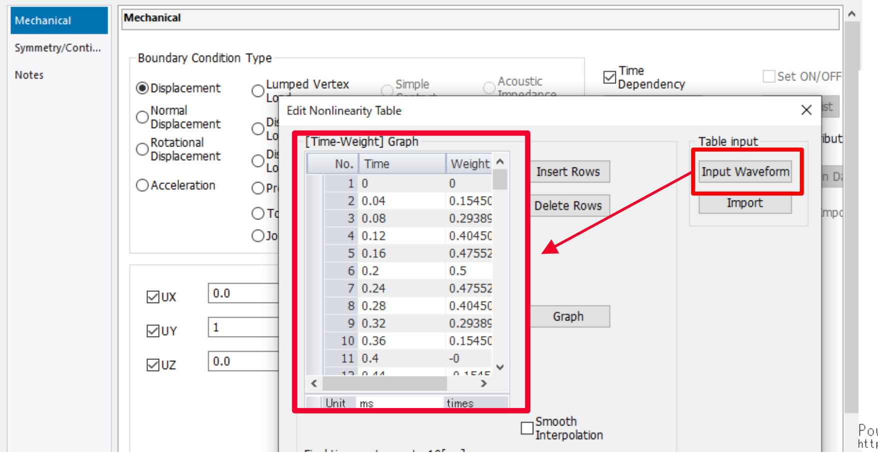
	Time-dependent Condition	
	Weight Function	Number of Divisions in Waveform
Stress Piezoelectric (without resonant mode)	Yes	No
Piezoelectric (with resonant mode)	Yes	Yes
Thermal	Yes	No
Electric-Thermal Coupled	Yes	Yes
Magnetic-Transient	No	Yes
Acoustic (Implicit method)	Yes	Yes
Fluid	Yes	No
Electromagnetic	No	Yes

- If a weight function table is used to specify time dependency, the timestep is automatically set based on the table, to perform calculations.
- If a waveform to be input is used to specify time dependency, the time step is defined by equally dividing a cycle, to perform calculations.

Solver in General: Automatic Timestep Setting Based on Time-Dependent Conditions

Automatic timestep setting based on a time-dependent condition is available.

Calculate Example 75: Transient Analysis of Standing Bars in the stress analysis.

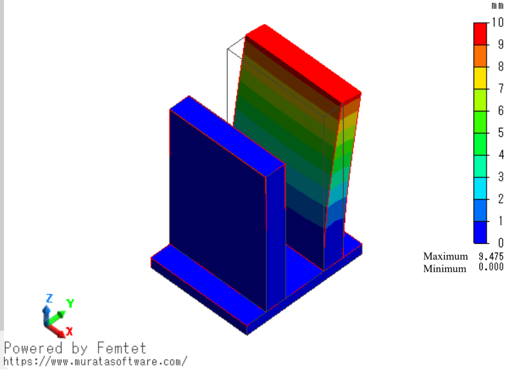


Specify the weight function with the [Waveform Input] auxiliary function.

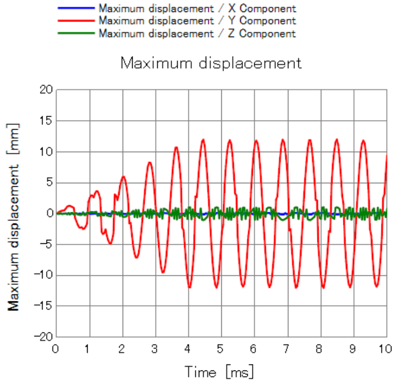
Resonant Frequency: 1247 Hz

Sampling Frequency: 1250 Hz

Number of Samples: 1250 x 5 (5: Number of Repetitions)



The higher bar largely vibrates at the resonant frequency.

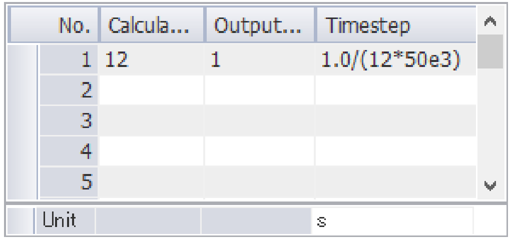


The displacement is increasing due to resonance.

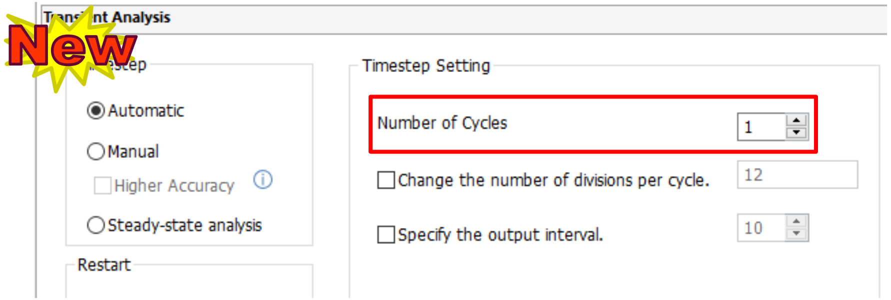
Solver in General: Automatic Timestep Setting Based on Time-Dependent Conditions

Allows you to set the timestep easily when using periodically vibrating waveform in the magnetic-transient analysis.

Analyze Example 8: Iron Loss of Coil Wound on Core in the magnetic-transient analysis.
（Analysis for one cycle of a 50 kHz AC input）



No.	Calcula...	Output...	Timestep
1	12	1	1.0/(12*50e3)
2			
3			
4			
5			
Unit			s



Using a frequency of 50 kHz and the number of divisions per cycle of 12, the timestep and number of steps are manually defined.

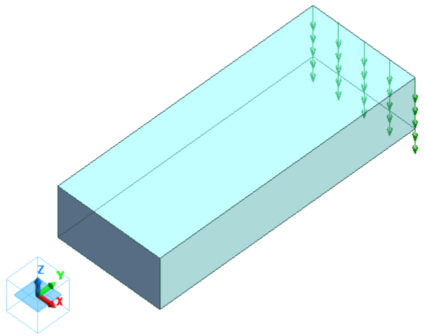
By specifying the number of cycles, the timestep is determined.

- Available when the input waveform is specified in the analyses other than the rotating and translation machine analyses.
- You can set the number of cycles.
- You can set the number of divisions per cycle.

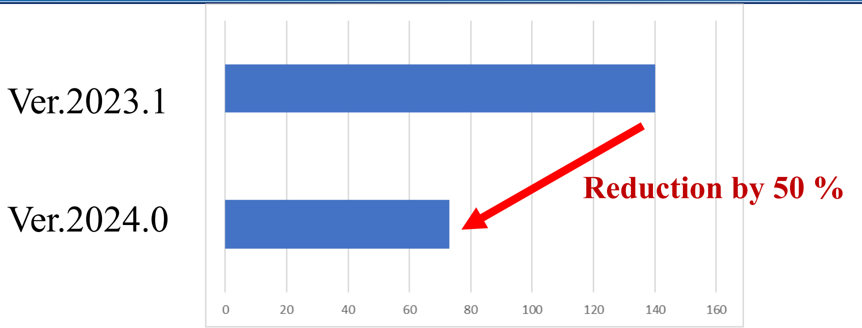
Solver: Stress
Fast computation and reduction in memory usage in the stress-static analysis

The improved iterative method in the stress-static analysis has achieved fast computation and reduction in memory usage.

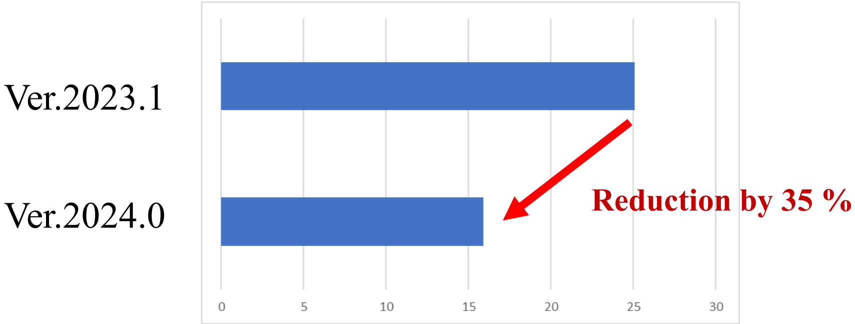
Example 1 in the stress analysis



Calculate with CPU: Xeon W-3375 (38core)



Solving Time (s)

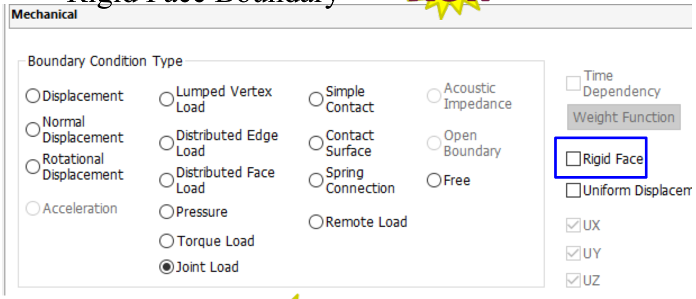


Memory Usage (GB)

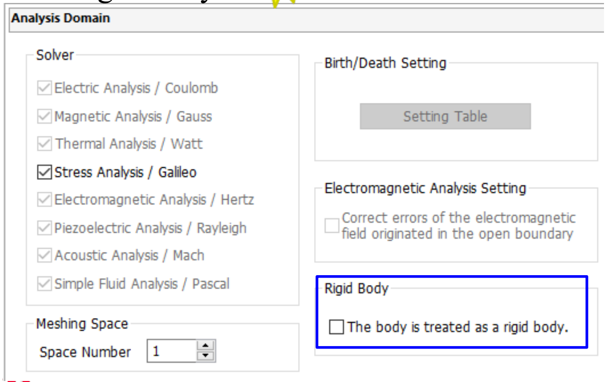
Solver: Stress
Rigid Face Boundary and Rigid Body

A rigid attribute can be set to boundaries or bodies by treating the specified topology as rigid.

■ Rigid Face Boundary New

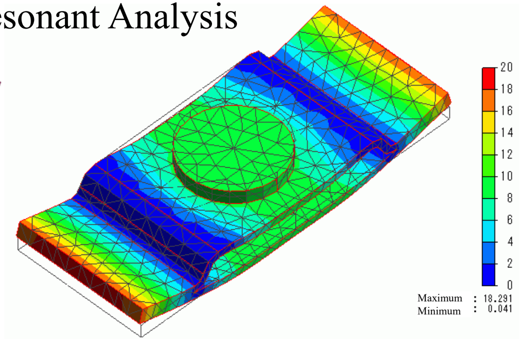


■ Rigid Body New

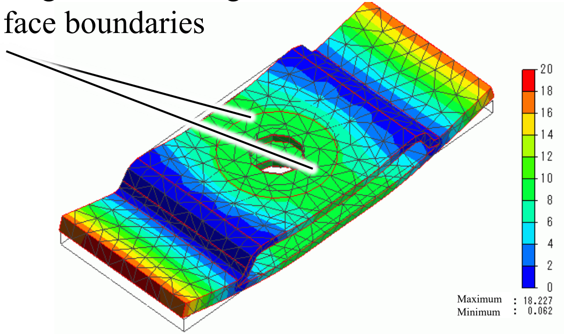


Example 73: Resonant Analysis

New



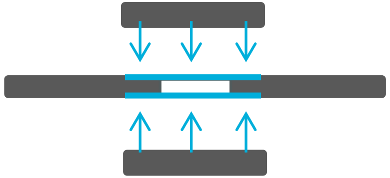
Reproduces the bolt-tightened seating surface using rigid face boundaries



Solver: Stress
Additional [Constrain except axial direction] Option on Joint Load Tab

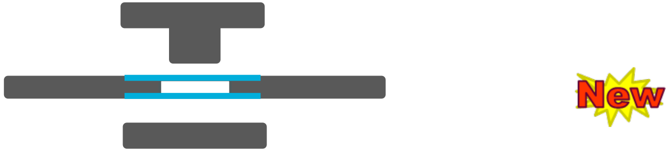
The joint load constrained in non-axial directions allows you to simulate and analyze the actual bolt tightening.

- Ver.2023.1 Joint Load Only

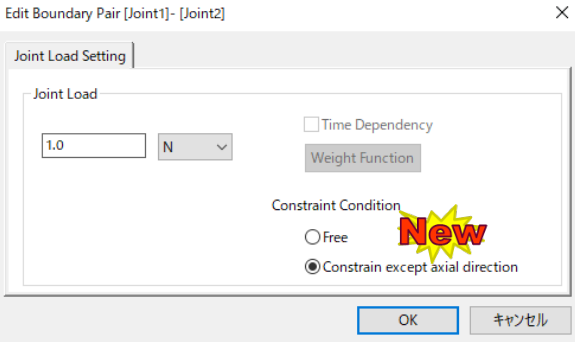


Applys load from above and below

- Ver.2024.0 Joint Load + Constraints in non-axial directions



Applys load from above and below + Set the relative displacement in non-axial directions to zero
(Same as the fixation by the shaft of a bolt)



Selectable on the Edit Boundary Pair dialog box

Solver: Stress
Shell Elements Applicable to Transient and Nonlinear Analyses

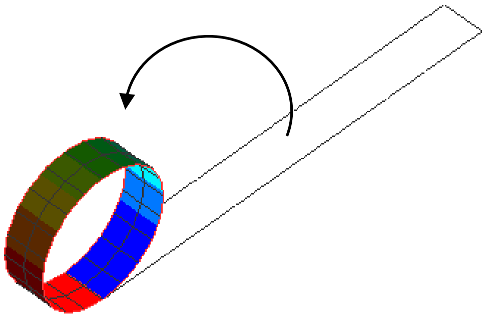
Murata Software

Shell elements can be used in the geometrically nonlinear, materially nonlinear, and transient analyses.

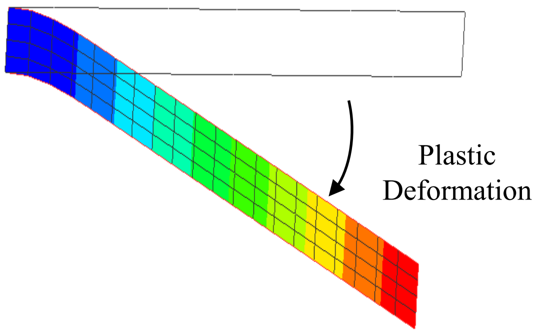
The function of shell elements, which achieves the accurate and fast analysis of thin plates, has improved significantly.

- Applicable in the geometrically nonlinear analysis for large displacement.
- Applicable in the analysis of nonlinear materials, such as elastic-plastic, viscoelastic, and hyperplastic non-linear materials.
- Applicable in the transient analysis.

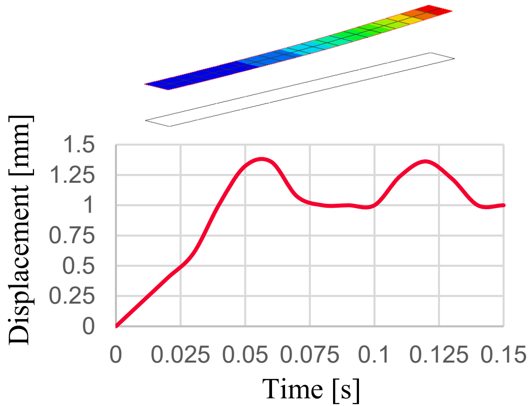
Large Displacement Problem



Elastic-Plastic Material

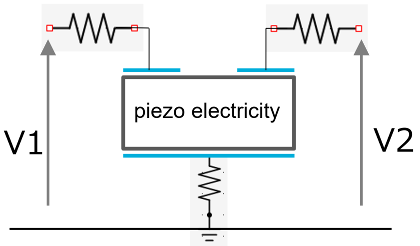


Transient Analysis

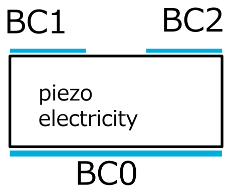


Solver: Piezoelectric
Additional Option for Electric Potential Specified Electric Wall Boundary

The option allows you to include L, C, or R in the electric potential-specified boundary.

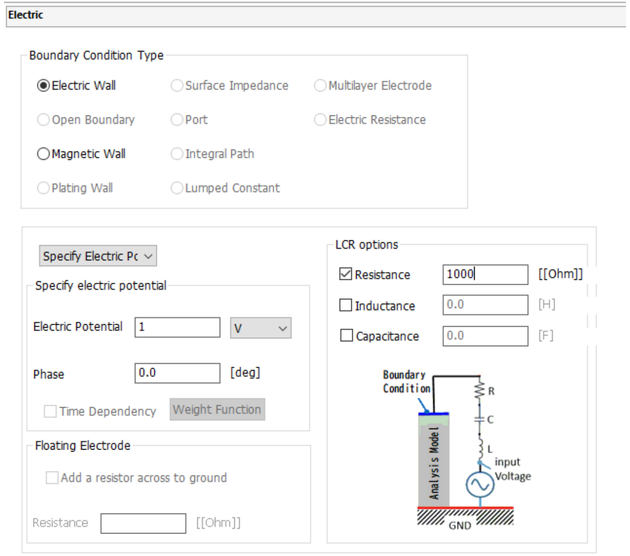


Target Model for Analysis



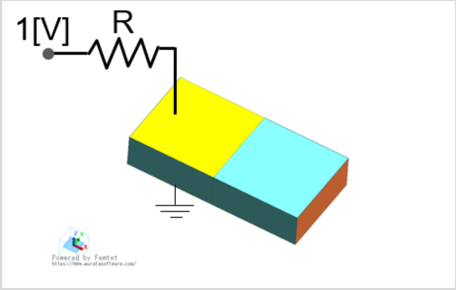
Femtet Model

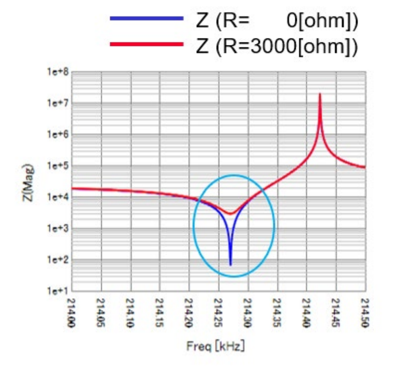
The setting below indicates a resistor of 1000 Ω is included in the electric potential-specified boundary.



Note: This option can be used only in the harmonic analysis.

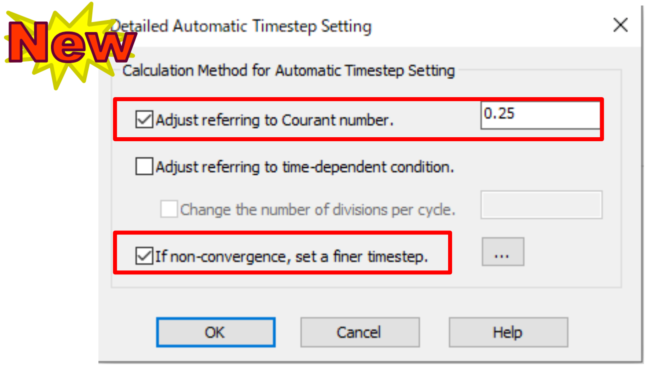
Analysis Example





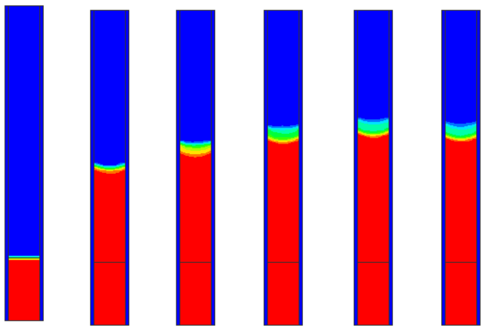
Solver: Fluid
Accuracy of Transient Analysis Improved

The automatic timestep setting with the Courant number or convergence can achieve higher accuracy.



* the Courant number
A quantity that indicates how many meshes the fluid travels across during one timestep.

Example 16: Capillary Action Analysis



	Manual	Automatic
Number of Steps	5000	1660
Analysis Time	8min 17s	6min 05s

Proper timestep can shorten the analysis time.

- Each timestep is shortened for analysis such that the Courant number during the timestep is less than the specified Courant number.
- The free surface analysis (VOF method) makes it difficult to set a proper timestep. By using the Courant number to be 1 or less, the analysis can be calculated with a proper timestep.
- If the calculation does not converge for each timestep, shorten the timestep to perform the analysis.

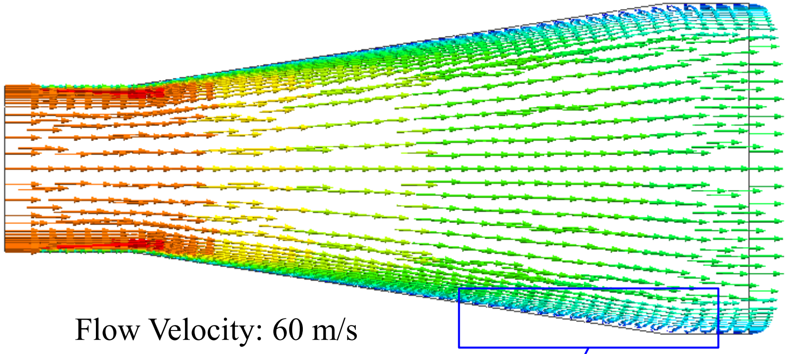
Solver: Fluid
Added SST k-ωTurbulent Model

The added SST k-ω turbulent model has demonstrated good repeatability of fluid separation.

■ Realizable k-ε Model

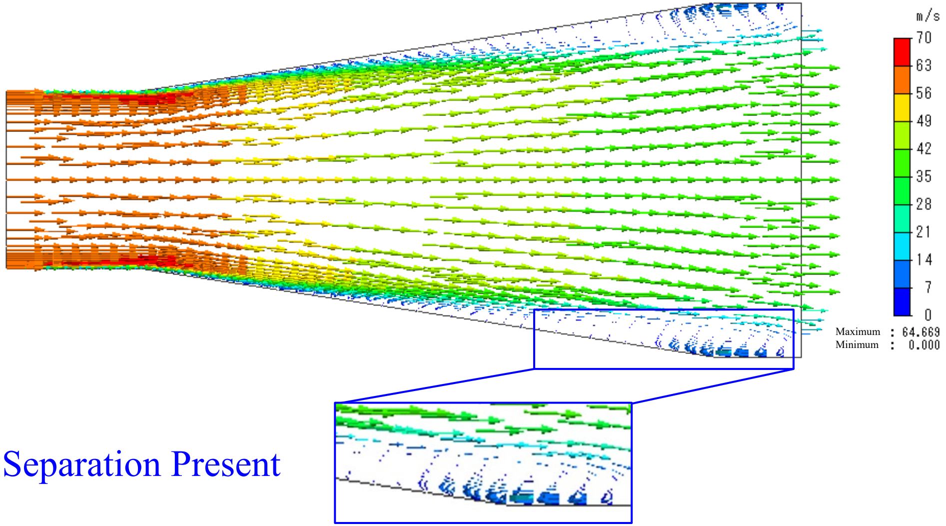
■ SST k-ω Model New

Re: about 10000



Flow Velocity: 60 m/s
Air Flow

No Separation

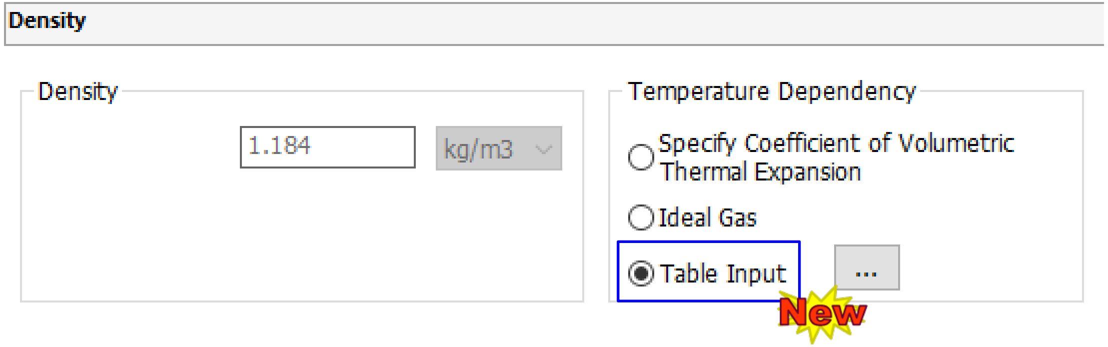


Separation Present

See Example 22 for the differences between the models.

Solver: Thermal-Fluid
Temperature Dependency Table Available in Density Dialog

Density with temperature dependency increases the calculation accuracy of buoyancy.



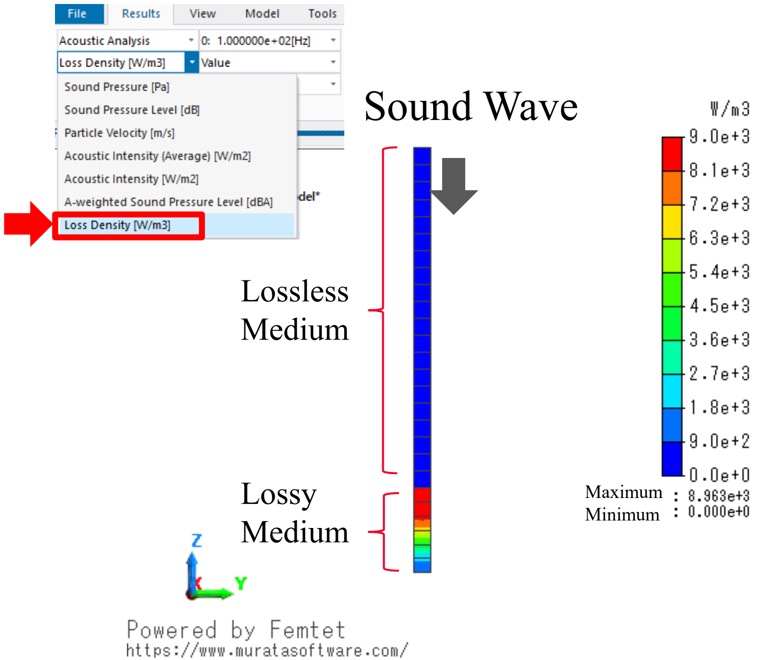
- For air and water, their temperature-density tables are stored in the Femtet material database.
  → If you use their values in the material database, select [Table Input].

- For other materials, select [Table Input] and enter the values manually.

Solver: Acoustic
Loss Density and Loss in Body

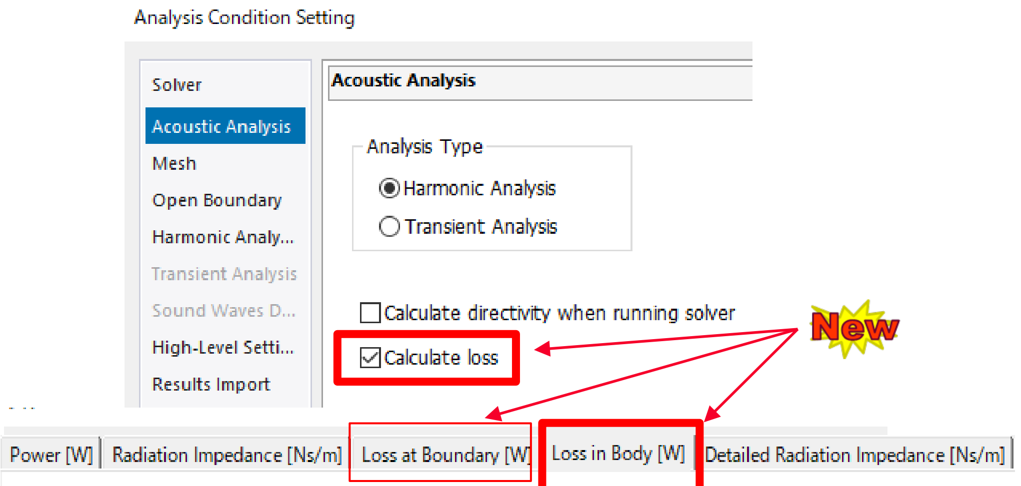
Loss density is added to the result field and loss in body is added to the result table.

- Loss density is added to the result field.



- The loss in each body is added to the result table.

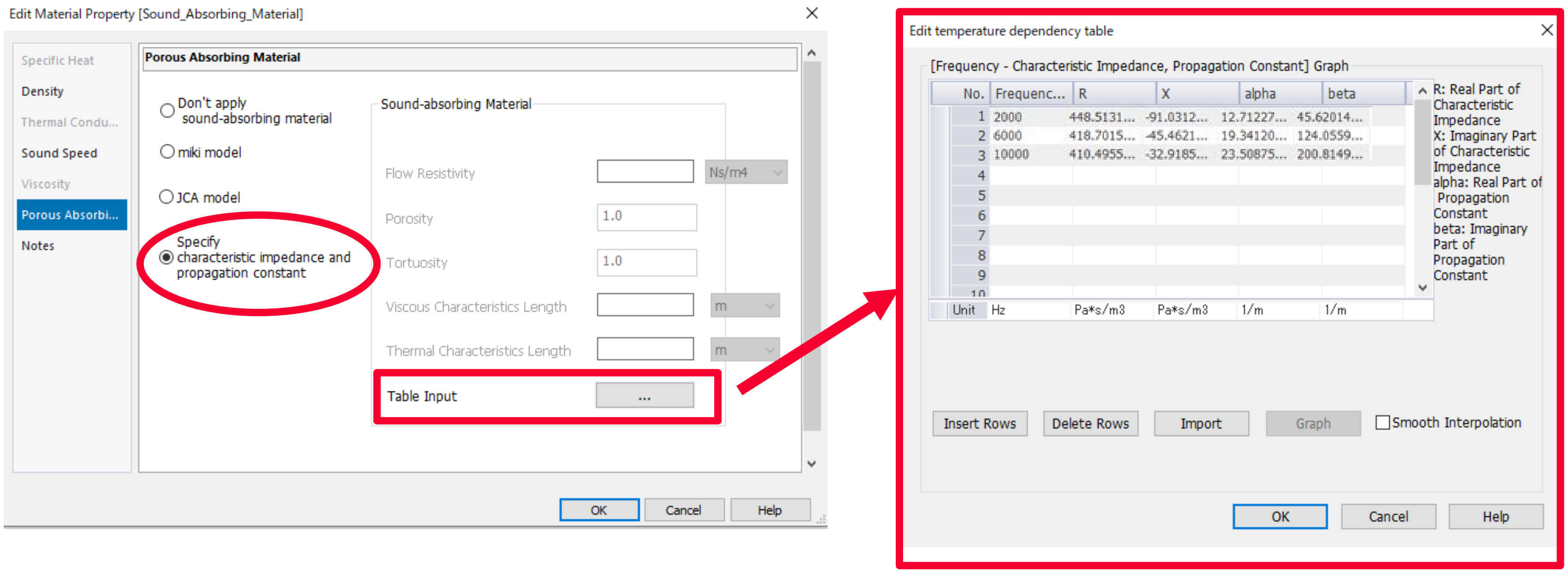
  On the acoustic analysis tab, select [Calculate loss].



	Frequency [Hz]	Body_Attribute_001	Body_Attribute_002	Body_Attribute_003
0: 2.000000e+03[Hz]	2.000000000e+3	0.000000000e+0	2.076277752e-2	8.279125889e-3
1: 6.000000e+03[Hz]	6.000000000e+3	0.000000000e+0	1.855809315e-2	9.303728184e-3
2: 1.000000e+04[Hz]	1.000000000e+4	0.000000000e+0	1.302012308e-2	5.588317317e-3

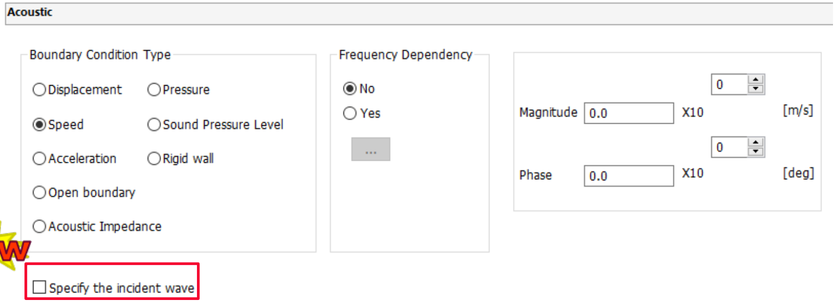
Solver: Acoustic
New Method to Specify Lossy Medium

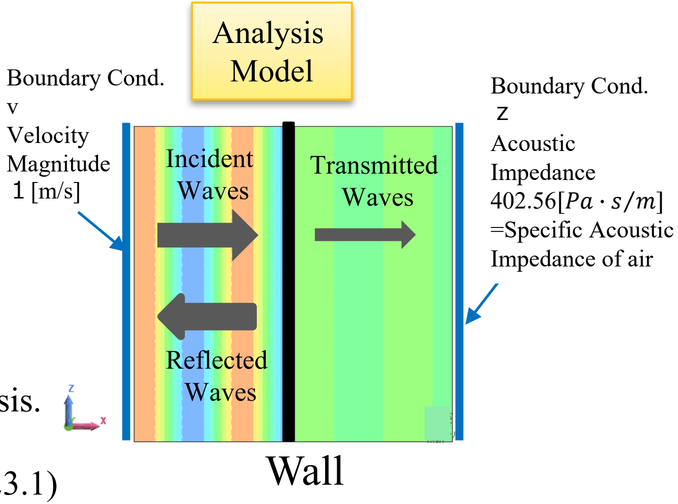
On the Porous Absorbing Material tab, [Specify characteristic impedance and propagation constant] has been selectable.



Solver: Acoustic
New Option [Specify the incident wave] for Speed Boundary Condition

Allows you to easily analyze the analysis model with significant reflections.





New

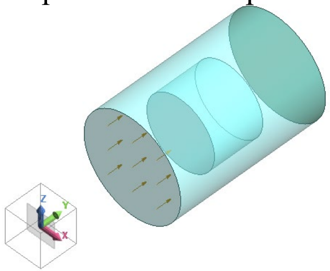
- Available condition
  When selecting the speed boundary condition in the acoustic harmonic analysis.
- [Specify the incident wave] not selected (Analysis in Femtet version up to 2023.1)
  The velocity that results from the interaction between incident waves and reflected waves was specified.
- [Specify the incident wave] selected (Femtet version 2024.0)
  The velocity of incident waves can be specified.

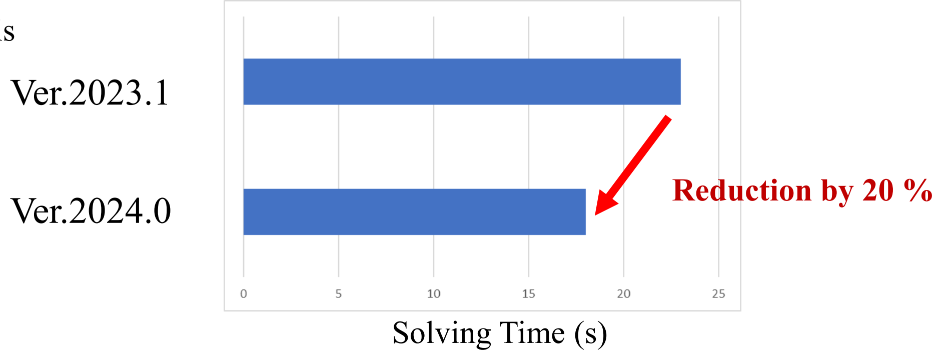
The analysis model with significant reflection, which was difficult to analyze, has been easily analyzed.
Example: Normal Transmittance
Refer to Example 15: Mass Law in the acoustic analysis

Solver: Thermal & Simple Fluid
Widely Applied Algebraic Multigrid Method Accelerates Calculation

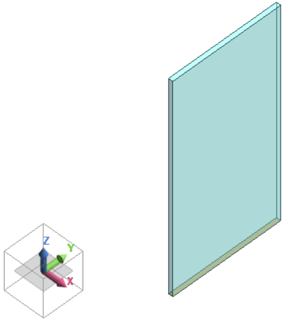
The new iterative method suitable for the thermal and simple fluid analyses has been developed to accelerate computations.

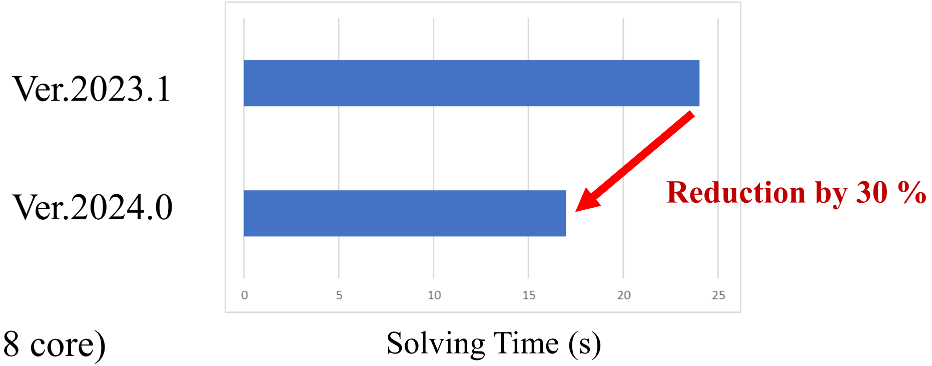
Example 1 in the Simple Fluid Analysis





Example 5 in the Thermal Analysis

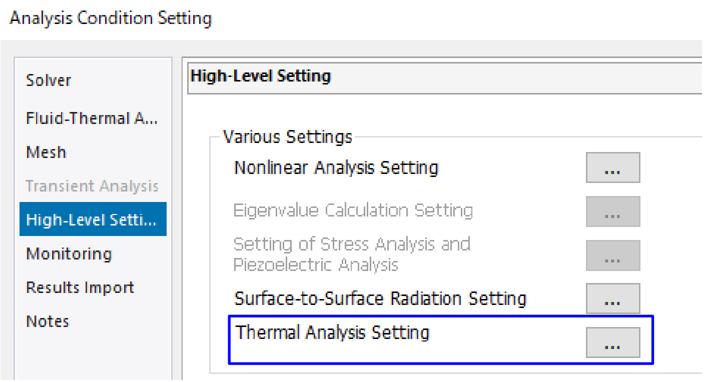




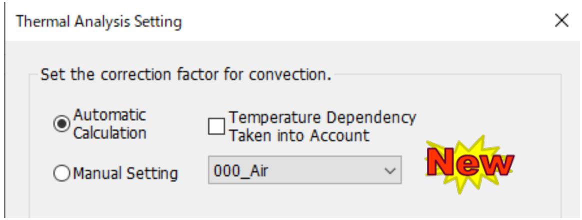
Calculate with CPU: Xeon W-3375 (38 core)

Solver: Thermal
New Function to Automatically Calculate Correction Factor for Convection

- **New function to automatically calculate the correction factor for convection has improved the accuracy of the automatically calculated coefficient of heat transfer.**
- **Allows you to easily calculate the heat dissipation to a fluid other than air.**



Select [Thermal Analysis Setting] on [High-Level Setting] tab



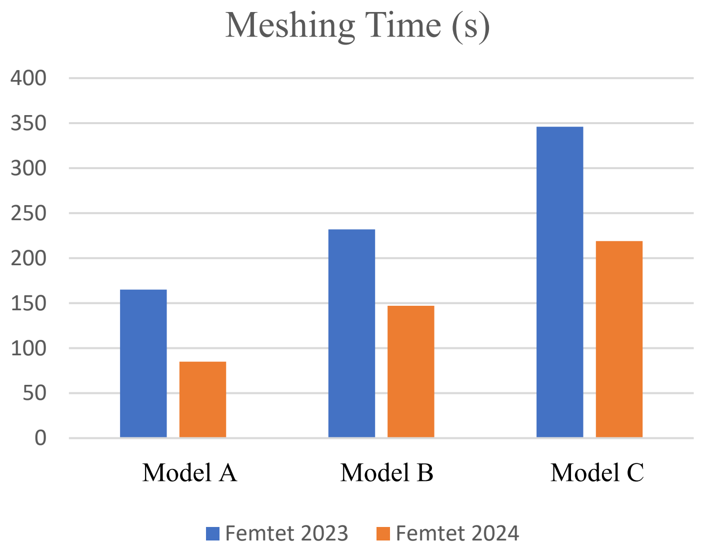
Select the fluid for heat dissipation.
⇒Only in the thermal analysis, heat dissipation to various fluids can be analyzed.

Select [Temperature Dependency Taken into Account]
⇒The changes in material properties with temperature are taken into account.

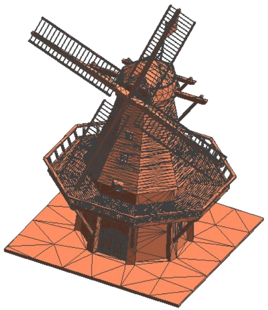
Select [Manual Setting]
⇒The correction factor for convection is entered by the user as before.

Mesher
Faster Calculation

Identify bottlenecks in the analysis of meshing time and then eliminate them for faster meshing calculations.



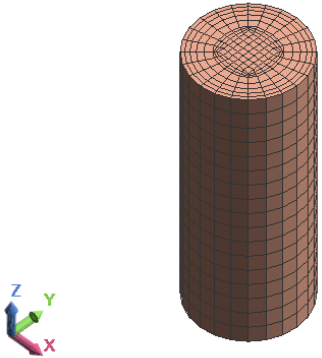
The revised mesher can calculate faster when a large-scale model with many bodies or a complex model with many free-curved surfaces has more than one million meshes.

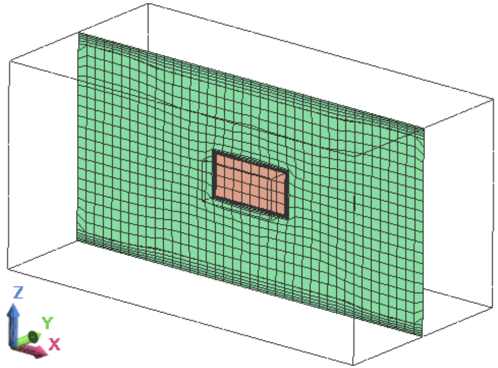


It can calculate up to **about twice as fast as** Femtet version 2023.

Mesher
Improved Layer Meshing

Layer meshes have been able to be created also on the meshes created with rectangular/sweep mesh or hexahedral/sweep mesh.

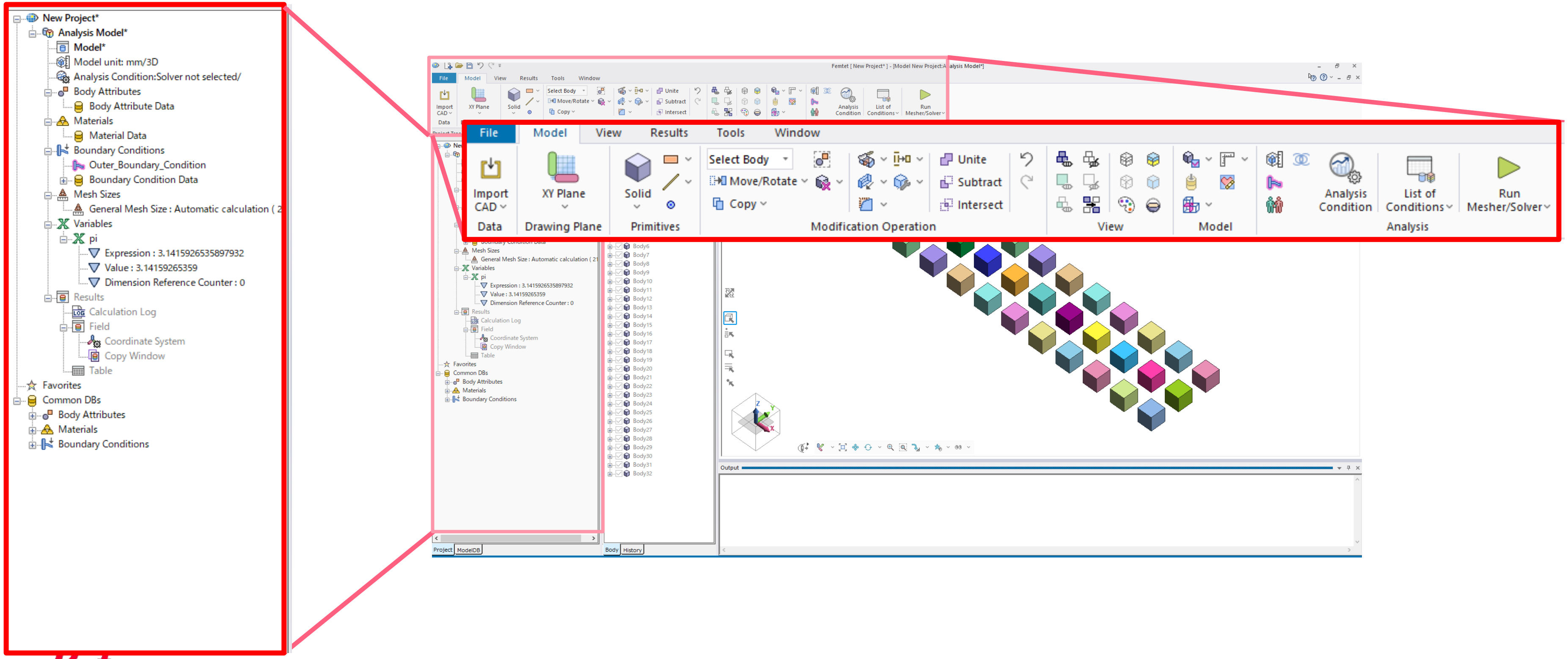




- In the fluid analysis of a simple shape created from sweep meshes or only hexahedrons, the convergence and accuracy of calculations will improve.

Icons of File Menu, Ribbons, and Tree Changed in Design and Coloring

The revised design and coloring have given Femtet an evolved, user-friendly, and attractive appearance.



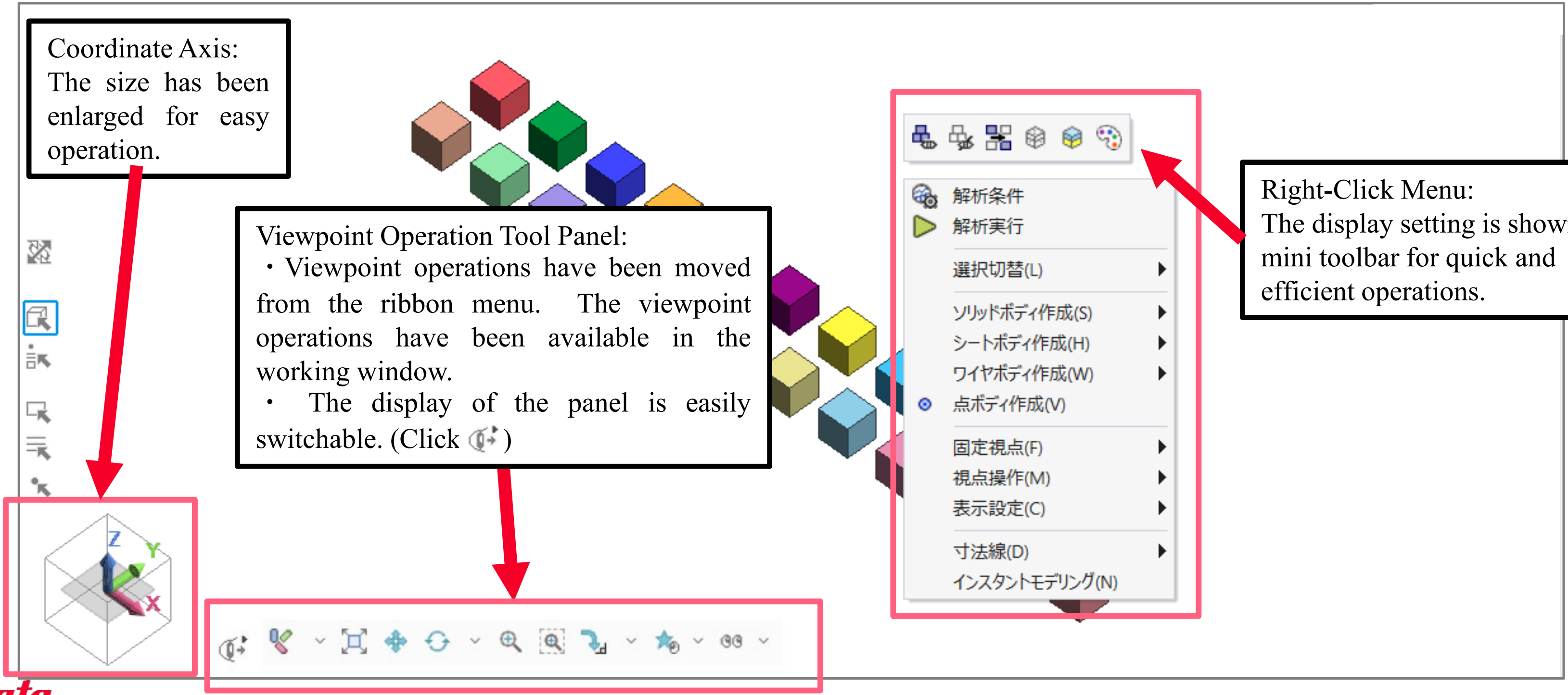
UI
Improved User Interface in Working Windows

Optimizing the UI, such as coordinate axis operations, tool panels, and right-click menus, in working windows has resulted in efficient operations.

Coordinate Axis:
The size has been enlarged for easy operation.

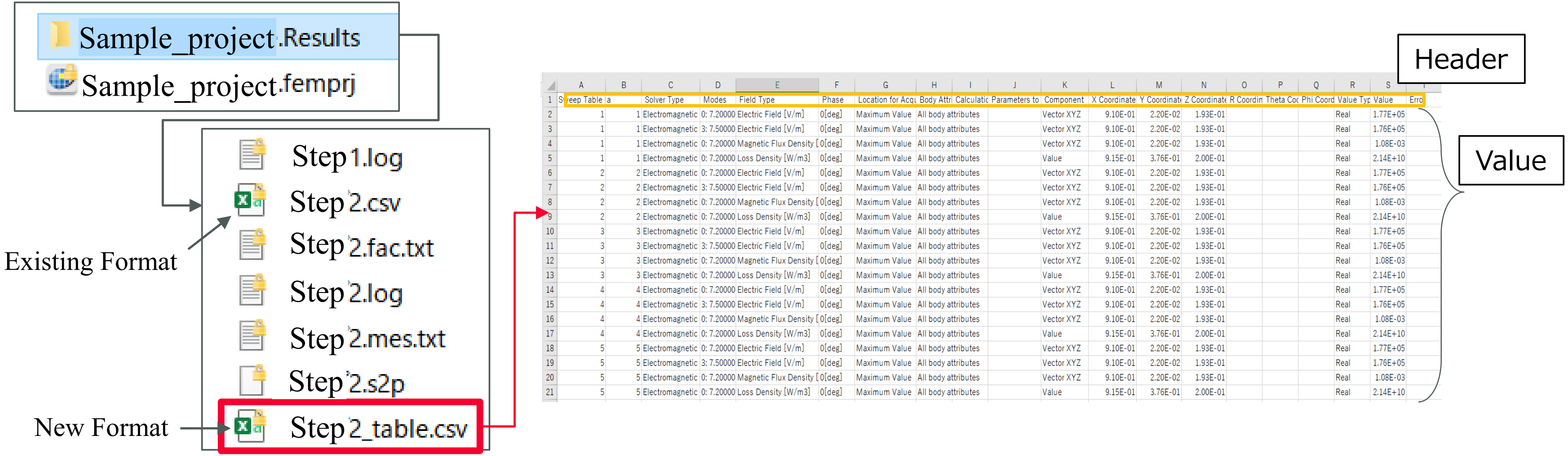
Viewpoint Operation Tool Panel:
・Viewpoint operations have been moved from the ribbon menu. The viewpoint operations have been available in the working window.
・ The display of the panel is easily switchable. (Click)

Right-Click Menu:
The display setting is shown in a mini toolbar for quick and efficient operations.



UI
New Table Format of CSV Files from Parametric Analysis Results

The new output format allows you to analyze with the pivot table function in Excel.

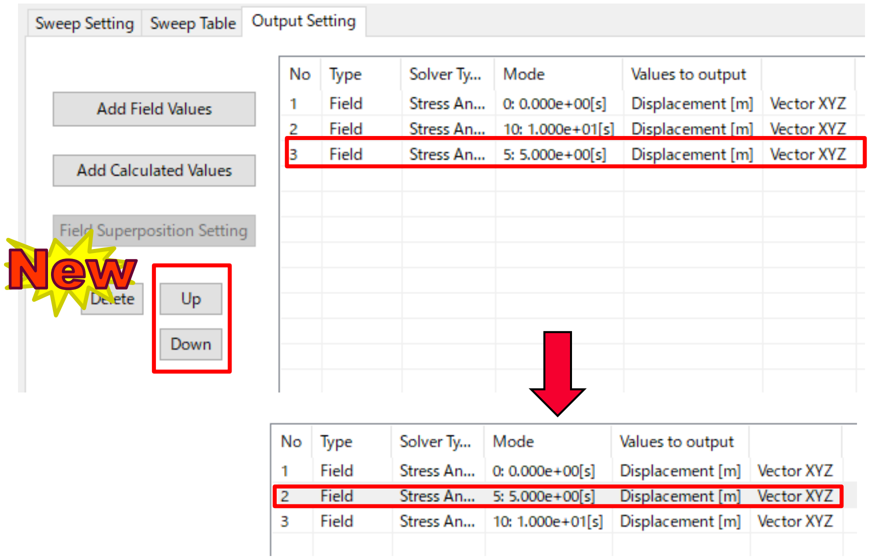


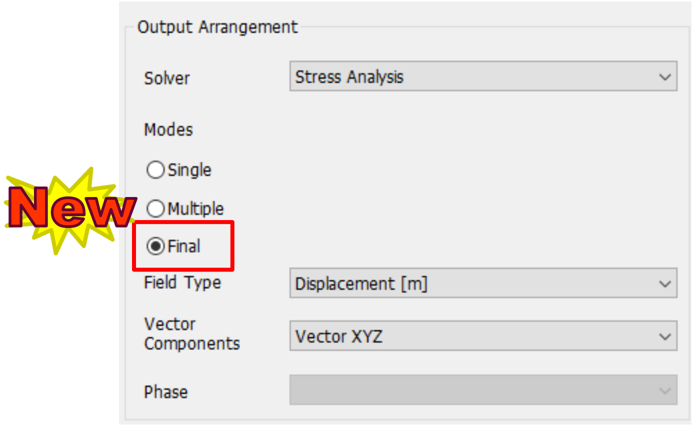
Sweep Table	a	Solver Type	Modes	Field Type	Phase	Location for Acqu	Body Attri	Calculatic	Parameters to	Component	X Coordinate	Y Coordinat	Z Coordinate	R Coordin	Theta Coo	Phi Coord	Value Typ	Value	Erro
1	1	Electromagnetic	0: 7.20000	Electric Field [V/m]	0[deg]	Maximum Value	All body attributes			Vector XYZ	9.10E-01	2.20E-02	1.93E-01				Real	1.77E+05	
1	1	Electromagnetic	3: 7.50000	Electric Field [V/m]	0[deg]	Maximum Value	All body attributes			Vector XYZ	9.10E-01	2.20E-02	1.93E-01				Real	1.76E+05	
1	1	Electromagnetic	0: 7.20000	Magnetic Flux Density [	0[deg]	Maximum Value	All body attributes			Vector XYZ	9.10E-01	2.20E-02	1.93E-01				Real	1.08E-03	
1	1	Electromagnetic	0: 7.20000	Loss Density [W/m3]	0[deg]	Maximum Value	All body attributes			Value	9.15E-01	3.76E-01	2.00E-01				Real	2.14E+10	
2	2	Electromagnetic	0: 7.20000	Electric Field [V/m]	0[deg]	Maximum Value	All body attributes			Vector XYZ	9.10E-01	2.20E-02	1.93E-01				Real	1.77E+05	
2	2	Electromagnetic	3: 7.50000	Electric Field [V/m]	0[deg]	Maximum Value	All body attributes			Vector XYZ	9.10E-01	2.20E-02	1.93E-01				Real	1.76E+05	
2	2	Electromagnetic	0: 7.20000	Magnetic Flux Density [	0[deg]	Maximum Value	All body attributes			Vector XYZ	9.10E-01	2.20E-02	1.93E-01				Real	1.08E-03	
2	2	Electromagnetic	0: 7.20000	Loss Density [W/m3]	0[deg]	Maximum Value	All body attributes			Value	9.15E-01	3.76E-01	2.00E-01				Real	2.14E+10	
3	3	Electromagnetic	0: 7.20000	Electric Field [V/m]	0[deg]	Maximum Value	All body attributes			Vector XYZ	9.10E-01	2.20E-02	1.93E-01				Real	1.77E+05	
3	3	Electromagnetic	3: 7.50000	Electric Field [V/m]	0[deg]	Maximum Value	All body attributes			Vector XYZ	9.10E-01	2.20E-02	1.93E-01				Real	1.76E+05	
3	3	Electromagnetic	0: 7.20000	Magnetic Flux Density [	0[deg]	Maximum Value	All body attributes			Vector XYZ	9.10E-01	2.20E-02	1.93E-01				Real	1.08E-03	
3	3	Electromagnetic	0: 7.20000	Loss Density [W/m3]	0[deg]	Maximum Value	All body attributes			Value	9.15E-01	3.76E-01	2.00E-01				Real	2.14E+10	
4	4	Electromagnetic	0: 7.20000	Electric Field [V/m]	0[deg]	Maximum Value	All body attributes			Vector XYZ	9.10E-01	2.20E-02	1.93E-01				Real	1.77E+05	
4	4	Electromagnetic	3: 7.50000	Electric Field [V/m]	0[deg]	Maximum Value	All body attributes			Vector XYZ	9.10E-01	2.20E-02	1.93E-01				Real	1.76E+05	
4	4	Electromagnetic	0: 7.20000	Magnetic Flux Density [	0[deg]	Maximum Value	All body attributes			Vector XYZ	9.10E-01	2.20E-02	1.93E-01				Real	1.08E-03	
4	4	Electromagnetic	0: 7.20000	Loss Density [W/m3]	0[deg]	Maximum Value	All body attributes			Value	9.15E-01	3.76E-01	2.00E-01				Real	2.14E+10	
5	5	Electromagnetic	0: 7.20000	Electric Field [V/m]	0[deg]	Maximum Value	All body attributes			Vector XYZ	9.10E-01	2.20E-02	1.93E-01				Real	1.77E+05	
5	5	Electromagnetic	3: 7.50000	Electric Field [V/m]	0[deg]	Maximum Value	All body attributes			Vector XYZ	9.10E-01	2.20E-02	1.93E-01				Real	1.76E+05	
5	5	Electromagnetic	0: 7.20000	Magnetic Flux Density [	0[deg]	Maximum Value	All body attributes			Vector XYZ	9.10E-01	2.20E-02	1.93E-01				Real	1.08E-03	
5	5	Electromagnetic	0: 7.20000	Loss Density [W/m3]	0[deg]	Maximum Value	All body attributes			Value	9.15E-01	3.76E-01	2.00E-01				Real	2.14E+10	

- The parametric analysis can automatically output the csv files in the table format, as well as the csv files in an existing format.
- The csv files in the table format are easy to analyze with the pivot table function of Excel or Python’s libraries.
- The data represented by the two csv files, one in the existing format and the other in the table format, equal to each other. You can use a preferable one for your application.

UI
Improved Output Setting for Results of Parametric Analysis

The display order of Output Setting is replaceable.
The final mode allows you to check the result at the last time.





- Regardless of the setting of the timestep, the result of the final time in the transient analysis can be saved.

Result Display: Stress
Improved Display of Analysis Results for Shell Elements

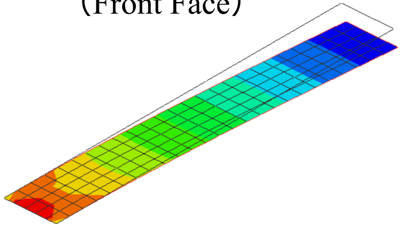
Murata Software

The display of analysis results for shell elements has been improved.

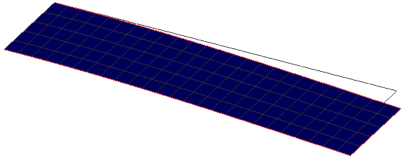
- In 2023.1 version and earlier versions, only the front data of stress and strain results was displayed.
- Staring with the 2024.0 version, the front and back data of stress and strain results can be displayed individually.

Contour Diagram of Maximum Principal Stress

(Front Face)

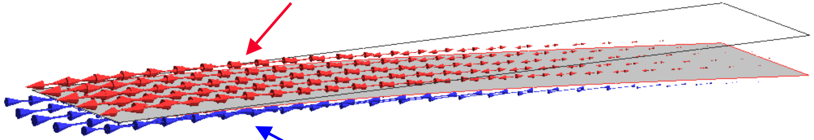


(Back Face)



Tensor Display Diagram of Stress

(Front Face)



(Back Face)

You can easily find on which face compressive or tensile stress is generated.

muRata

Thank you!