Femtet Seminar

Understanding

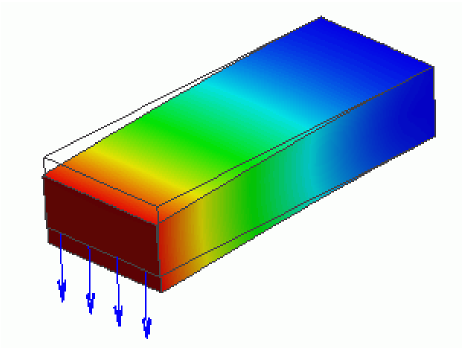
Stress Analysis
&
Thermal Analysis

202009

Table of Contents

Murata Software

1. Stress Analysis [Galileo] Overview



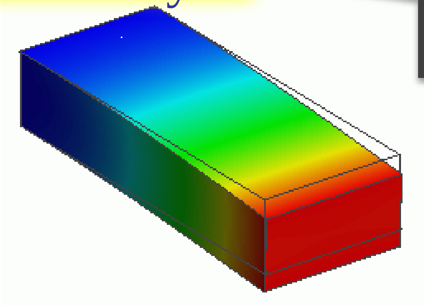
- Type of Stress Analysis
- Analysis Options
- Material Property
- Body Attribute
- Boundary Condition
- Results Display
- Step/Thermal Load Setting

Analysis Type

Murata Software

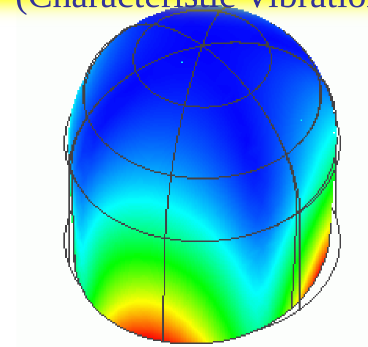
Static Analysis

Solves equation of equilibrium.
[K] {x}= {f}



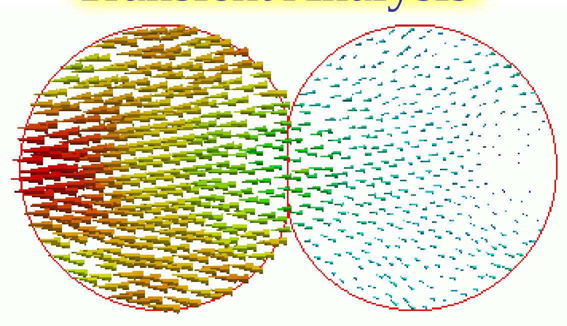
Cantilever under Distributed Load (Example 1)

Resonant Analysis
(Characteristic Vibration)



Resonance of Bell (Example 11)

Transient Analysis

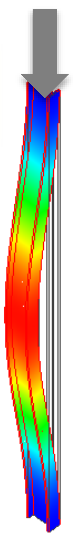


Colliding Balls (Example 32)

Solves equation of motion.
[M]{a} + [C]{v} + [K]{x} = {f}

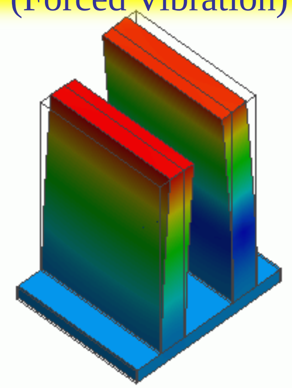
*Option for Advanced Mechanical is required.

Buckling Analysis



H-beam Steel (Example 48)

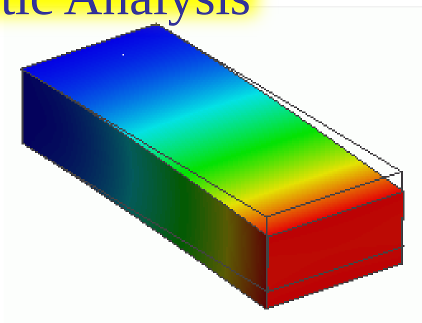
Harmonic Analysis
(Forced Vibration)



Standing Bars (Example 16)

Analysis Type

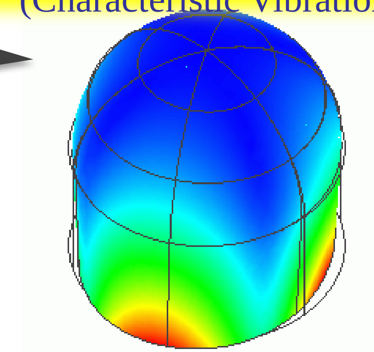
Static Analysis



Cantilever under Distributed Load (Example 1)

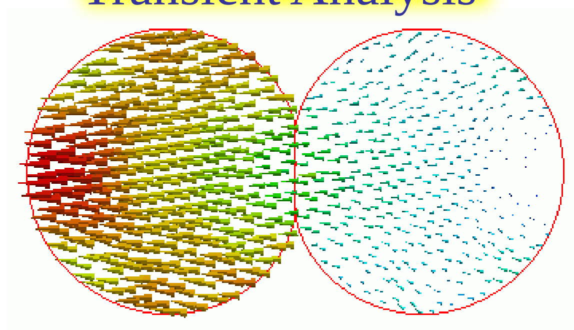
Resonant Analysis
(Characteristic Vibration)

Solves characteristic vibration.
$\{ [K] - \omega^2 [M] \} \{x\} = \{0\}$



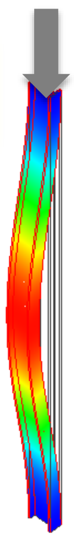
Resonance of Bell (Example 11)

Transient Analysis



Colliding Balls (Example 32)

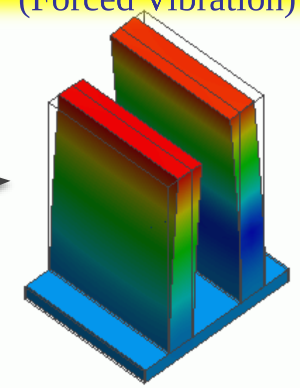
Buckling Analysis



H-beam Steel (Example 48)

Harmonic Analysis
(Forced Vibration)

Solves vibration generated by sinusoidal vibration force.
$\{ [K] - \omega^2 [M] \} \{x\} = \{f\}$

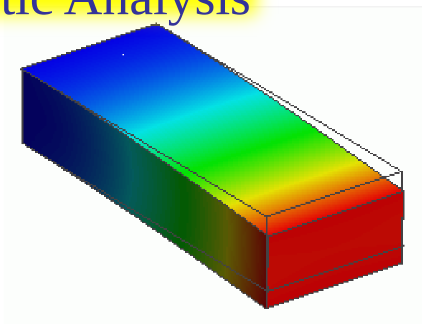


Standing Bars (Example 16)

Analysis Type

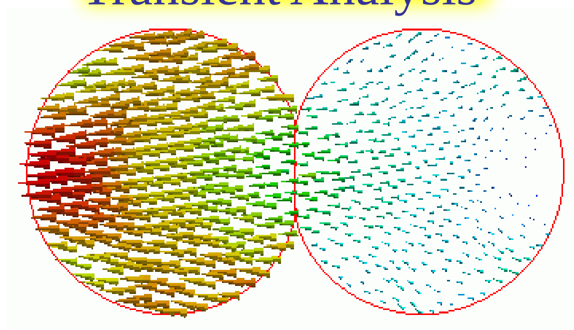
Murata Software

Static Analysis



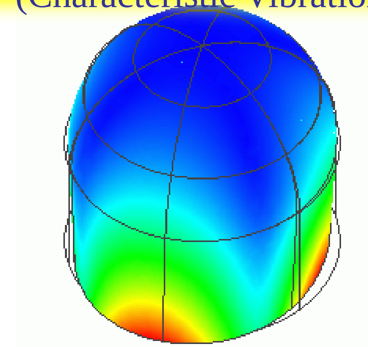
Cantilever under Distributed Load (Example 1)

Transient Analysis



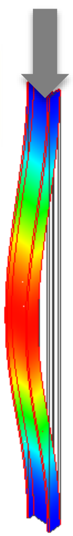
Colliding Balls (Example 32)

Resonant Analysis
(Characteristic Vibration)



Resonance of Bell (Example 11)

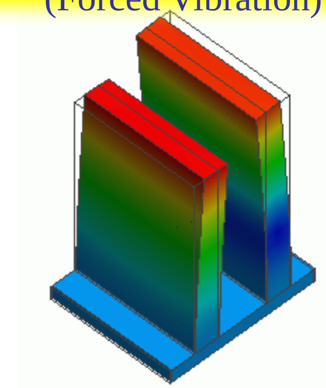
Buckling Analysis



Solves buckling mode generated by a compressive force.

H-beam Steel (Example 48)

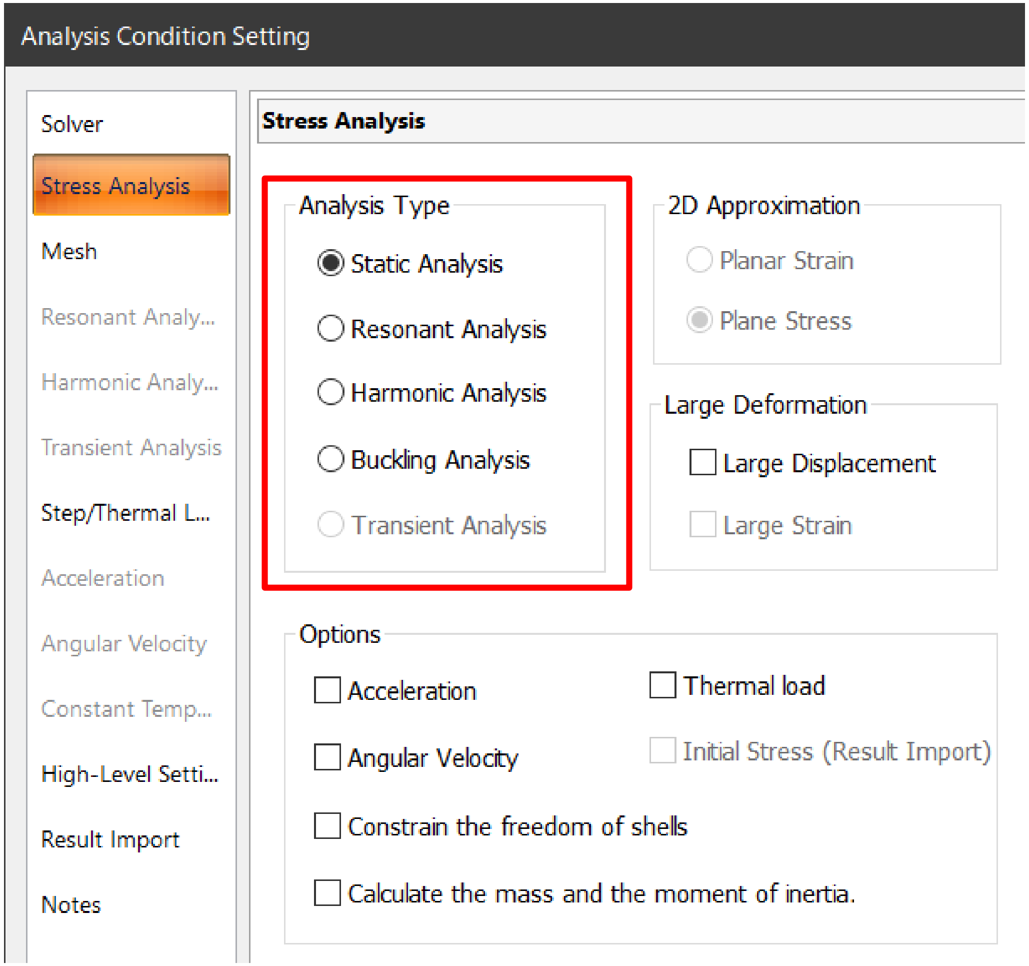
Harmonic Analysis
(Forced Vibration)



Standing Bars (Example 16)

Analysis Type

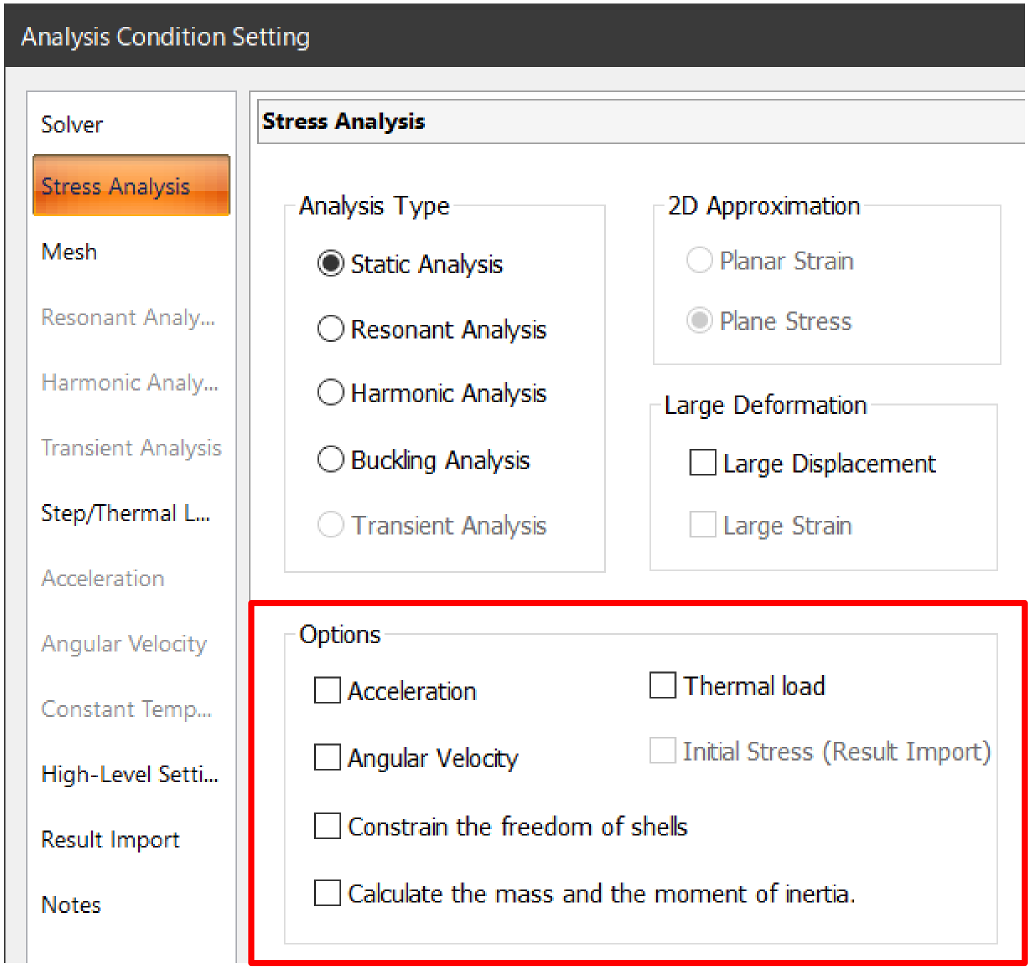
5 analysis types are available.



Analysis Options

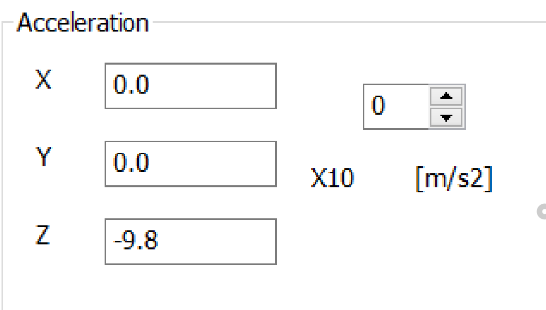
Murata Software

Auxiliary options are available.

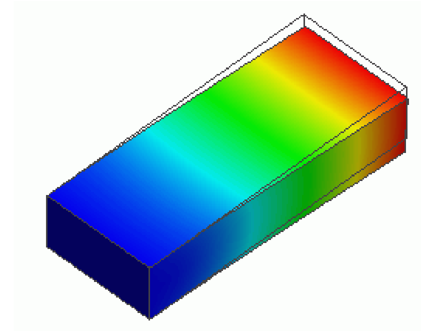


Acceleration

generates inertial force (ma).



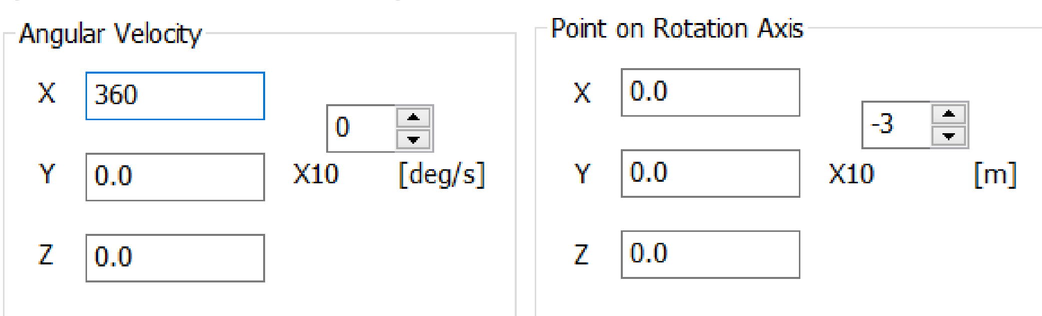
Apply gravity to the model.

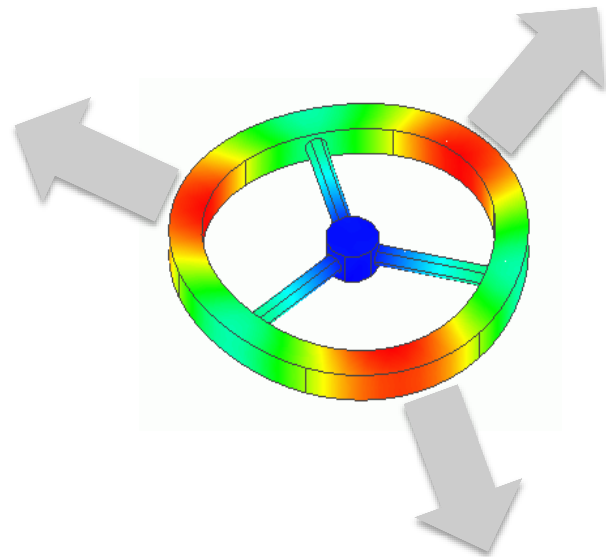


Cantilever under Gravity (Example 4)

Angular Velocity

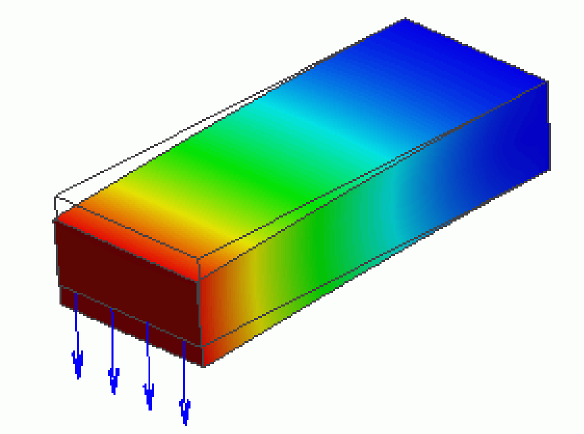
generates centrifugal force ($mr\omega^2$).





Deformation due to Centrifugal Force (Example 23)

Flows of Stress Analysis



① Draw Analysis Model
(can import CAD data)

② Select Analysis Type

③ Auxiliary Options

④ Set Material Property to Each Body

⑤ Set Body Attribute to Each Body

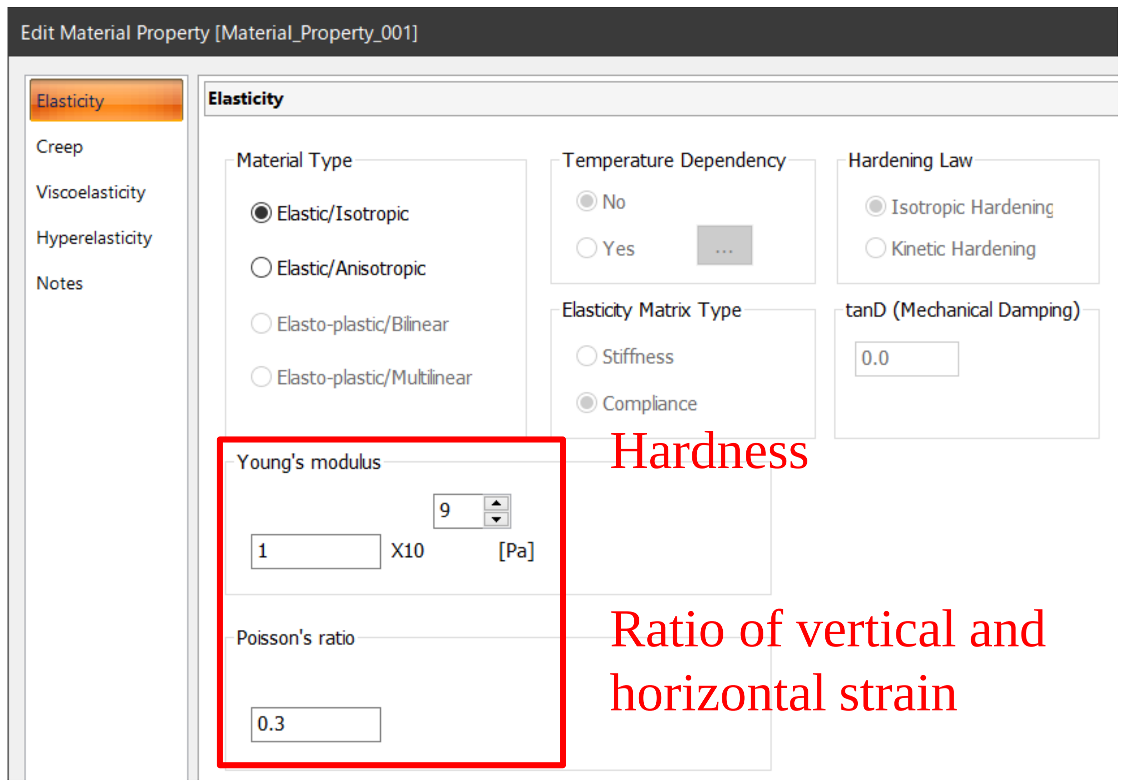
⑥ Set Boundary Condition (such as applied force)

⑦ Run Solver

Material Property

Murata Software

- Elasticity (Young's modulus and Poisson's ratio)
- Density
- Coefficient of thermal expansion

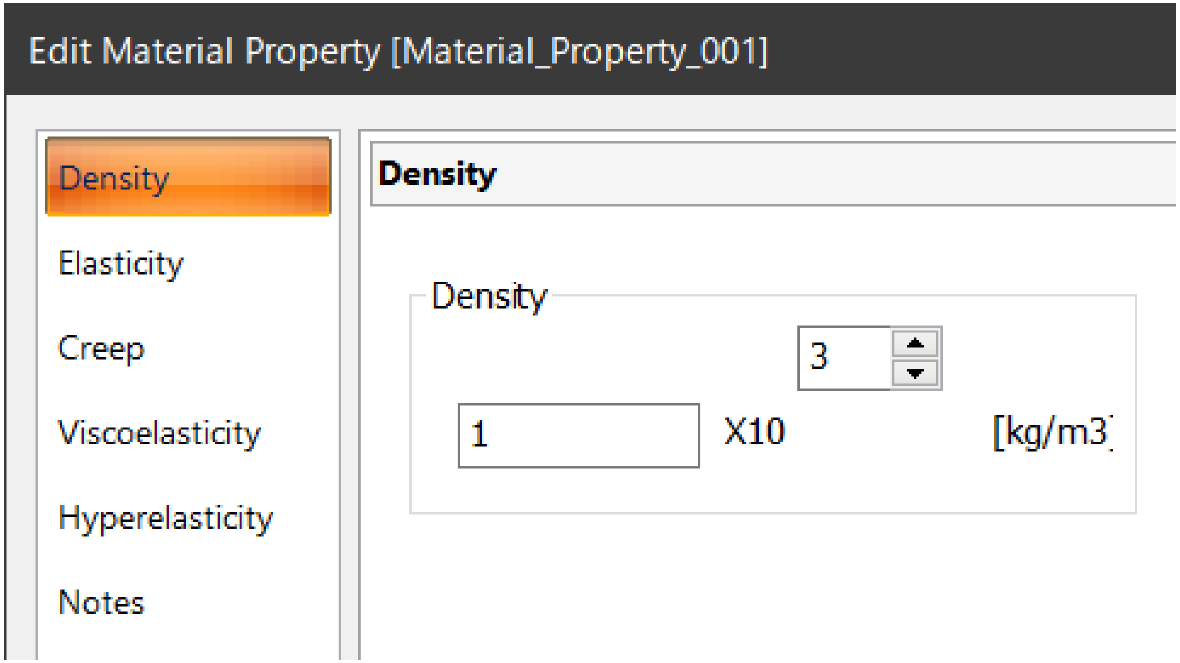


Hardness

Ratio of vertical and horizontal strain

Material Property

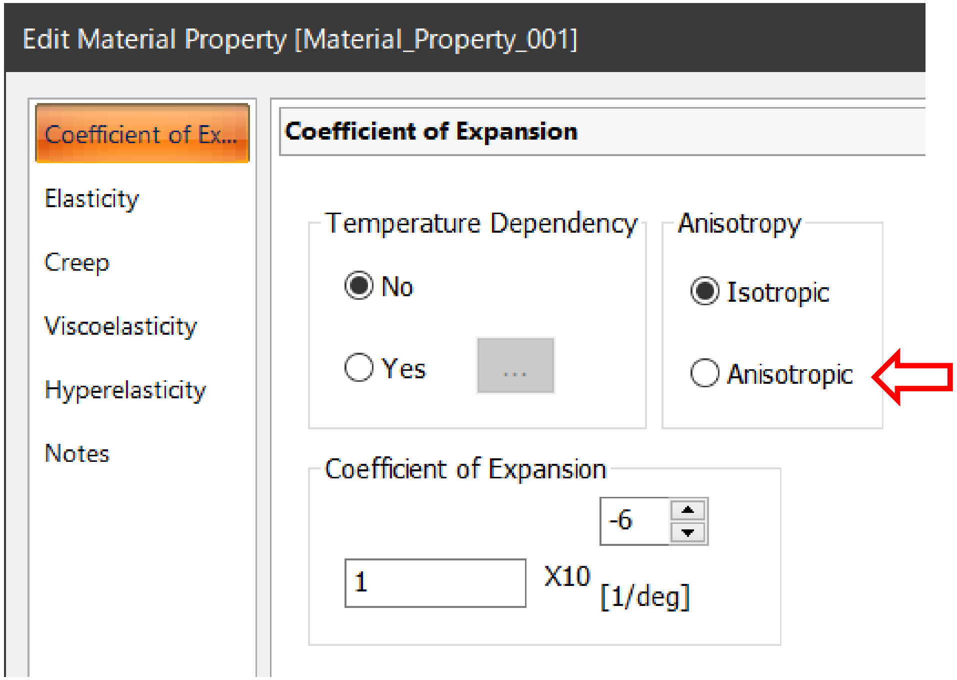
- Elasticity (Young’s modulus and Poisson’s ratio)
- Density
- Coefficient of thermal expansion



Material Property

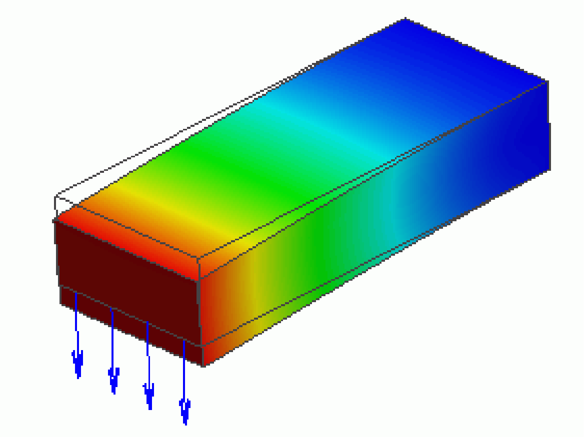
Murata Software

- Elasticity (Young's modulus and Poisson's ratio)
- Density
- Coefficient of thermal expansion



[Anisotropic] is available for Coefficient of thermal expansion and Elasticity.

Flows of Stress Analysis



① Draw Analysis Model
(can import CAD data)

② Select Analysis Type

③ Auxiliary Options

④ Set Material Property to Each Body

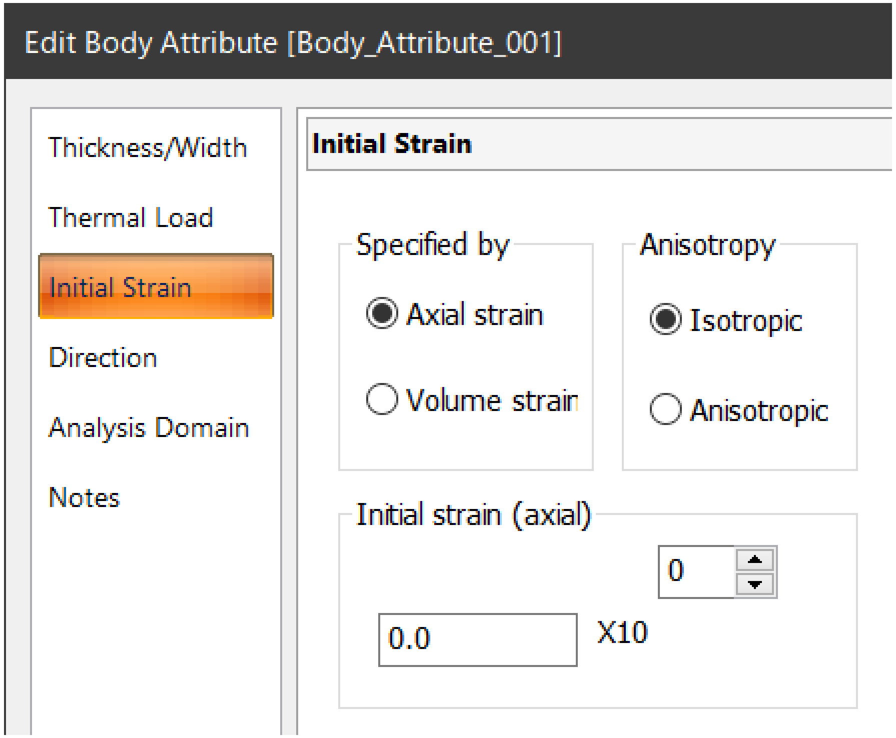
⑤ Set Body Attribute to Each Body

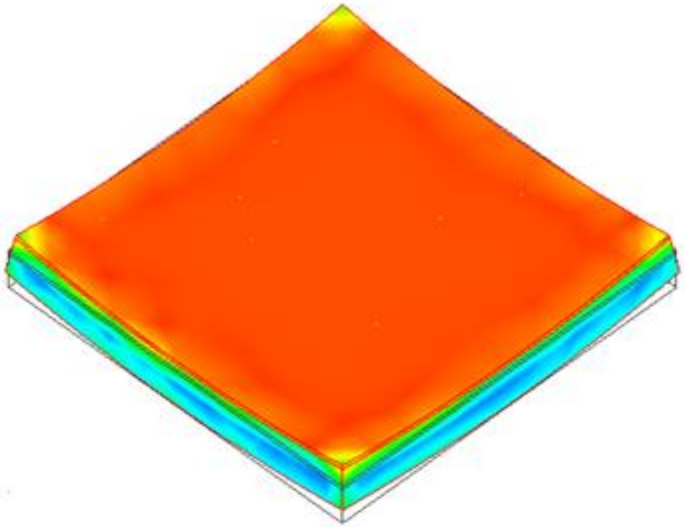
⑥ Set Boundary Condition (such as applied force)

⑦ Run Solver

Body Attribute

Initial stress, such as thin film stress, can be set.

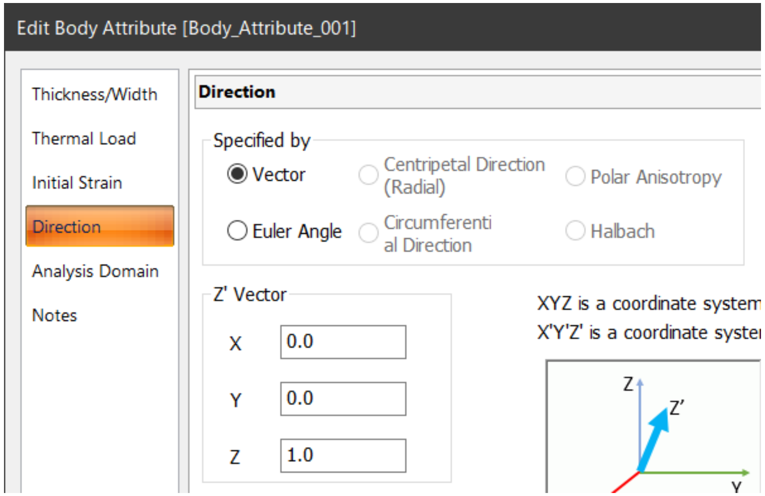




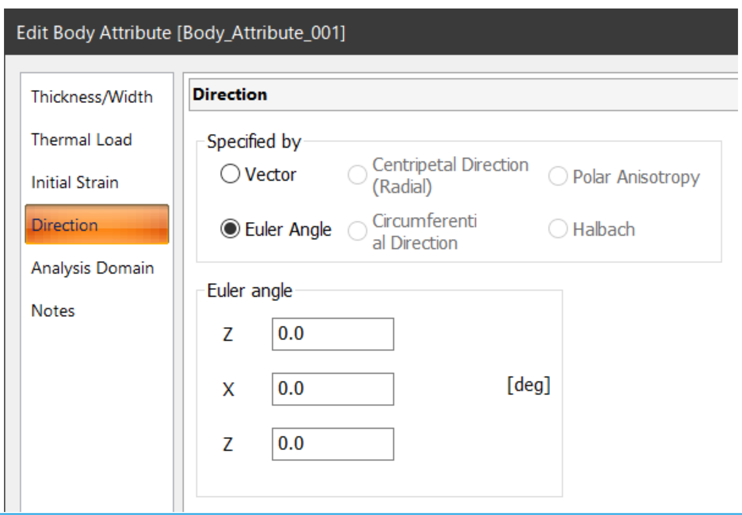
Mechanical Stress of Substrate Coated with Thin Film (Example 41)

Body Attribute

On [Direction] tab, direction of anisotropic material can be set.



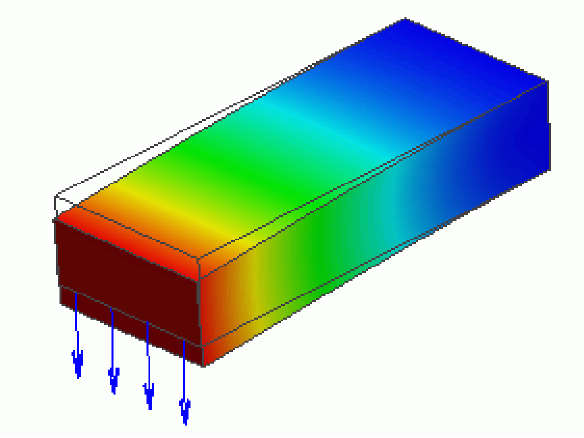
If only the 3rd axis is a special anisotropic material, the material direction can be shown by specifying the vector of the 3rd axis.



Two vectors can be specified.

For details of Euler angle, see Help.

Flows of Stress Analysis



① Draw Analysis Model
(can import CAD data)

② Select Analysis Type

③ Auxiliary Options

④ Set Material Property to Each Body

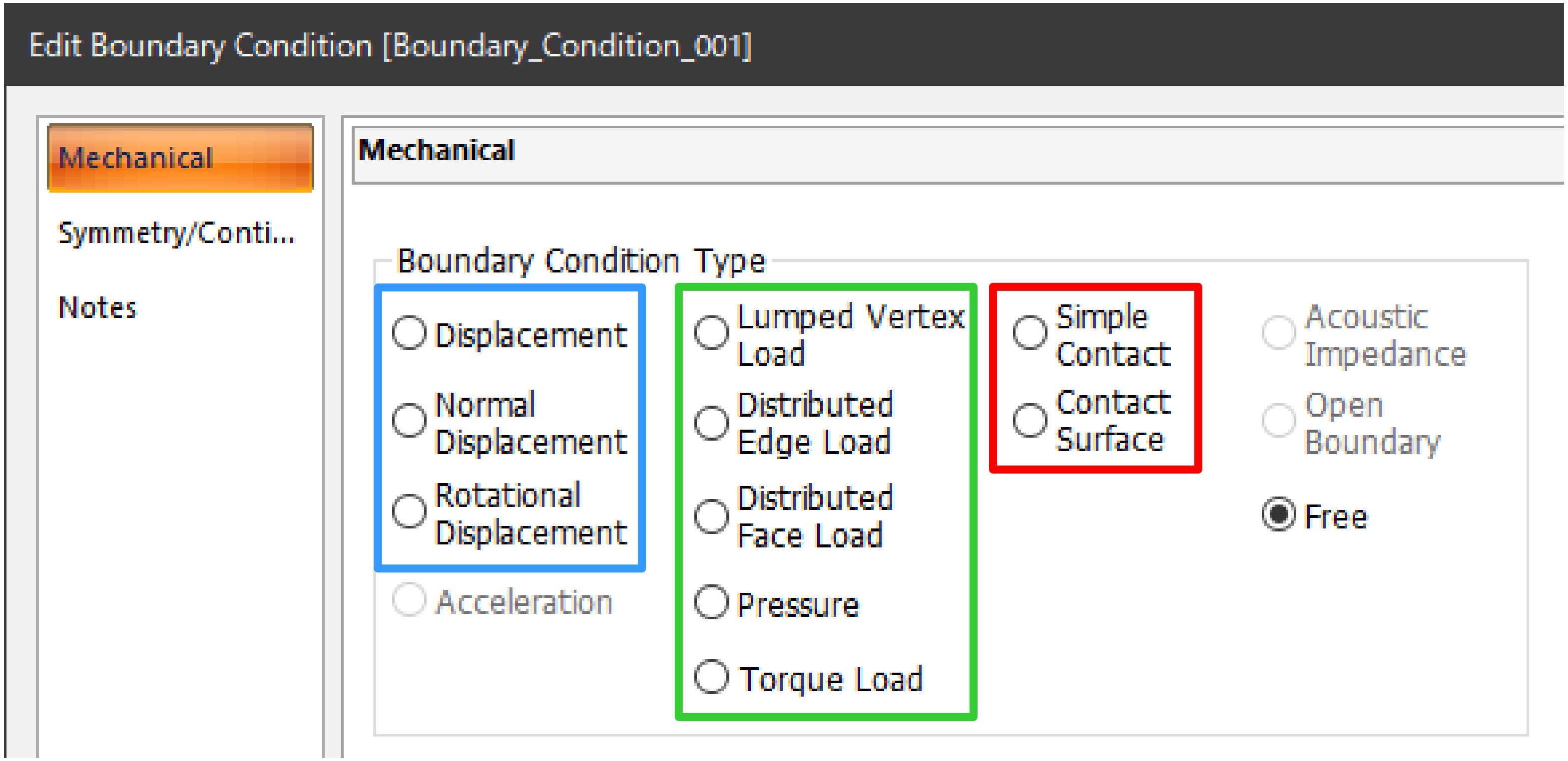
⑤ Set Body Attribute to Each Body

⑥ Set Boundary Condition (such as applied force)

⑦ Run Solver

Boundary Condition

Murata Software

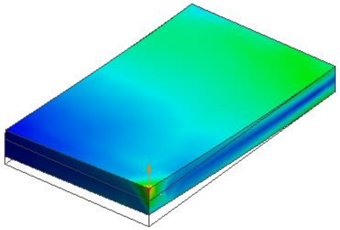


Displacement Force Contact

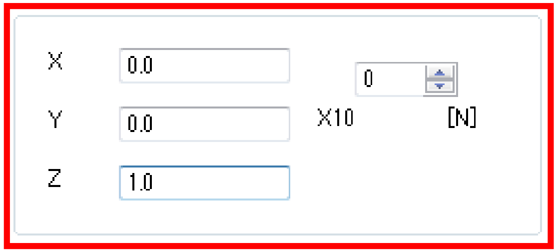
Boundary Condition [Load]

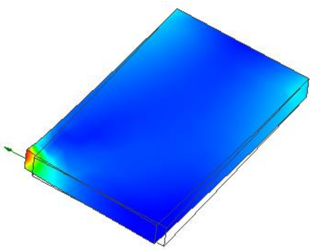
Murata Software

The load type is specified.

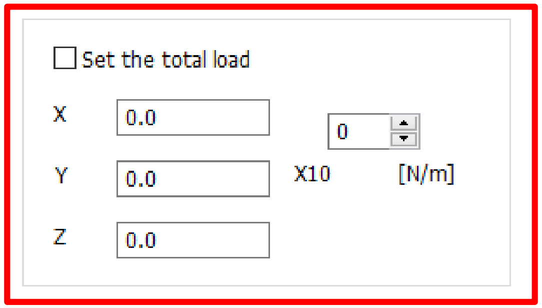


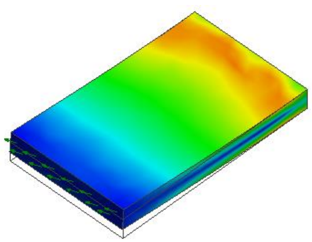
Lumped Vertex Load



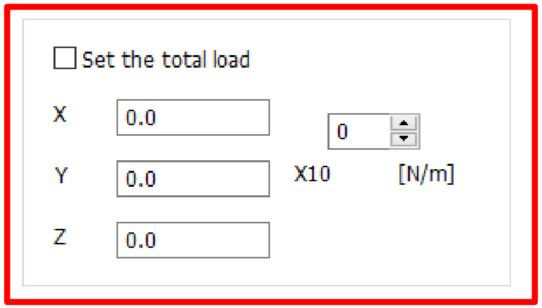


Distributed Edge Load

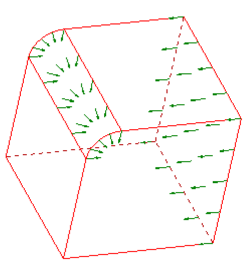




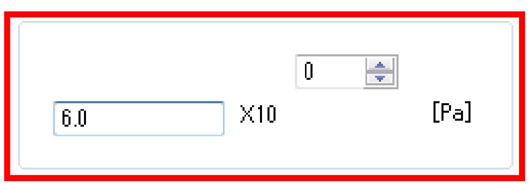
Distributed Face Load



※ Unit for the total load is [N]



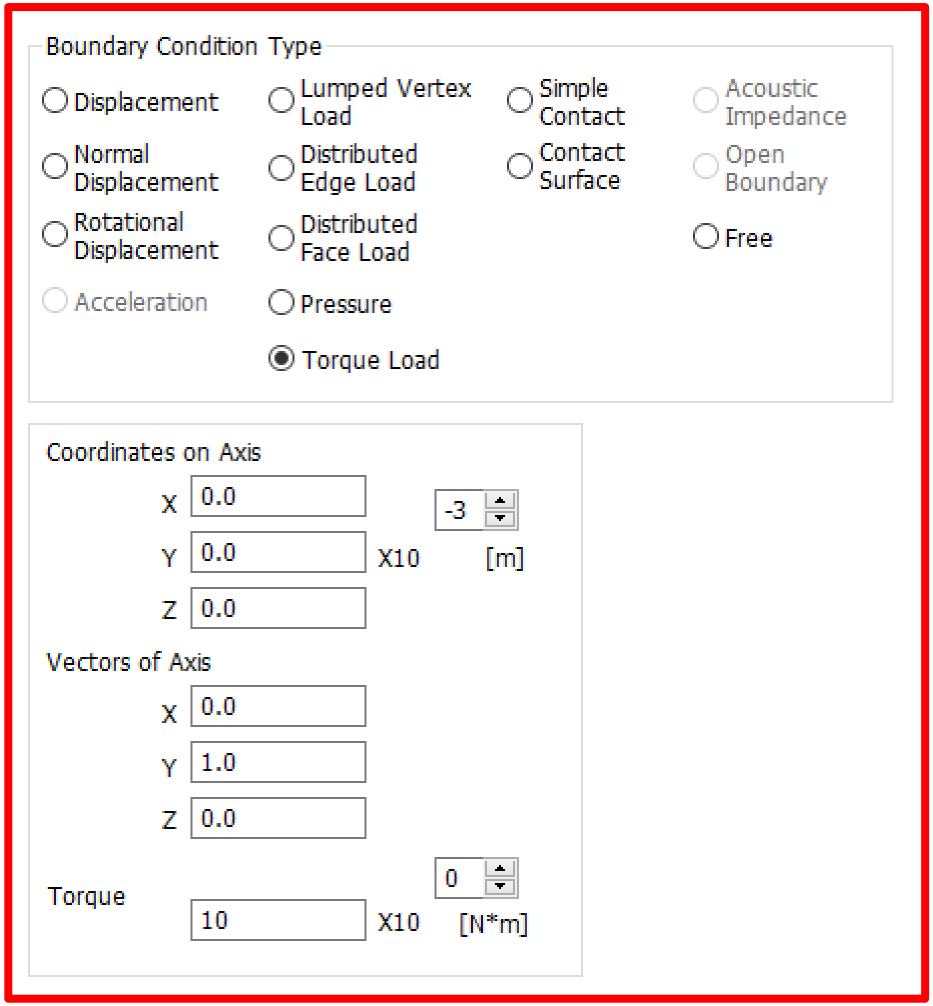
Pressure on Face

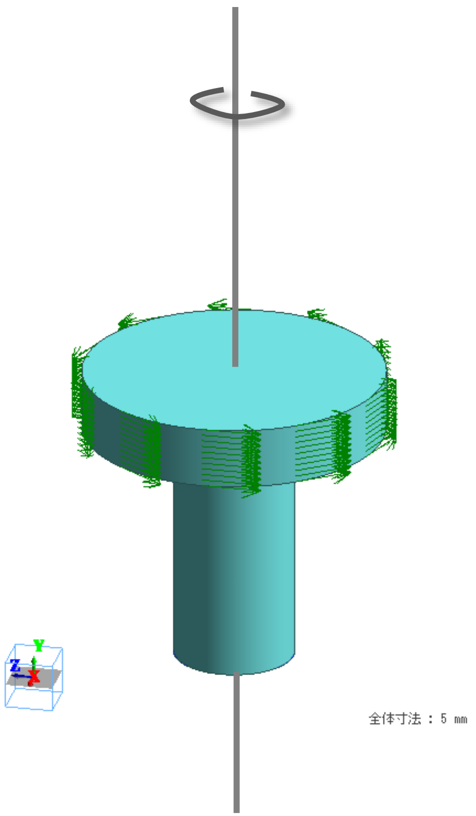


Boundary Condition [Torque]

Murata Software

Torque is specified.

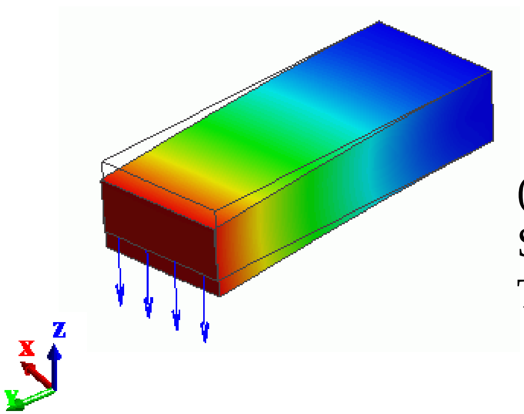




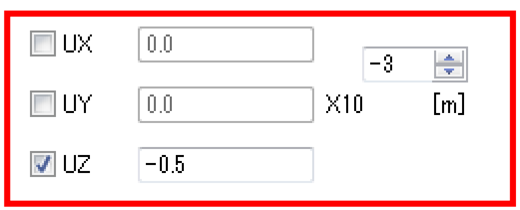
Boundary Condition [Displacement]

Murata Software

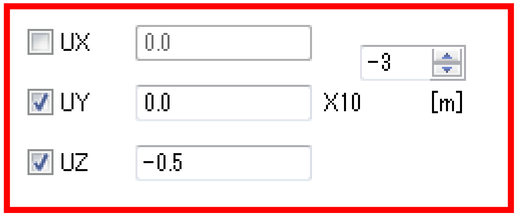
The displacement is specified.



(Example 3)
Specify the displacement at the tip of a bar.
The other end of the bar is fixed.

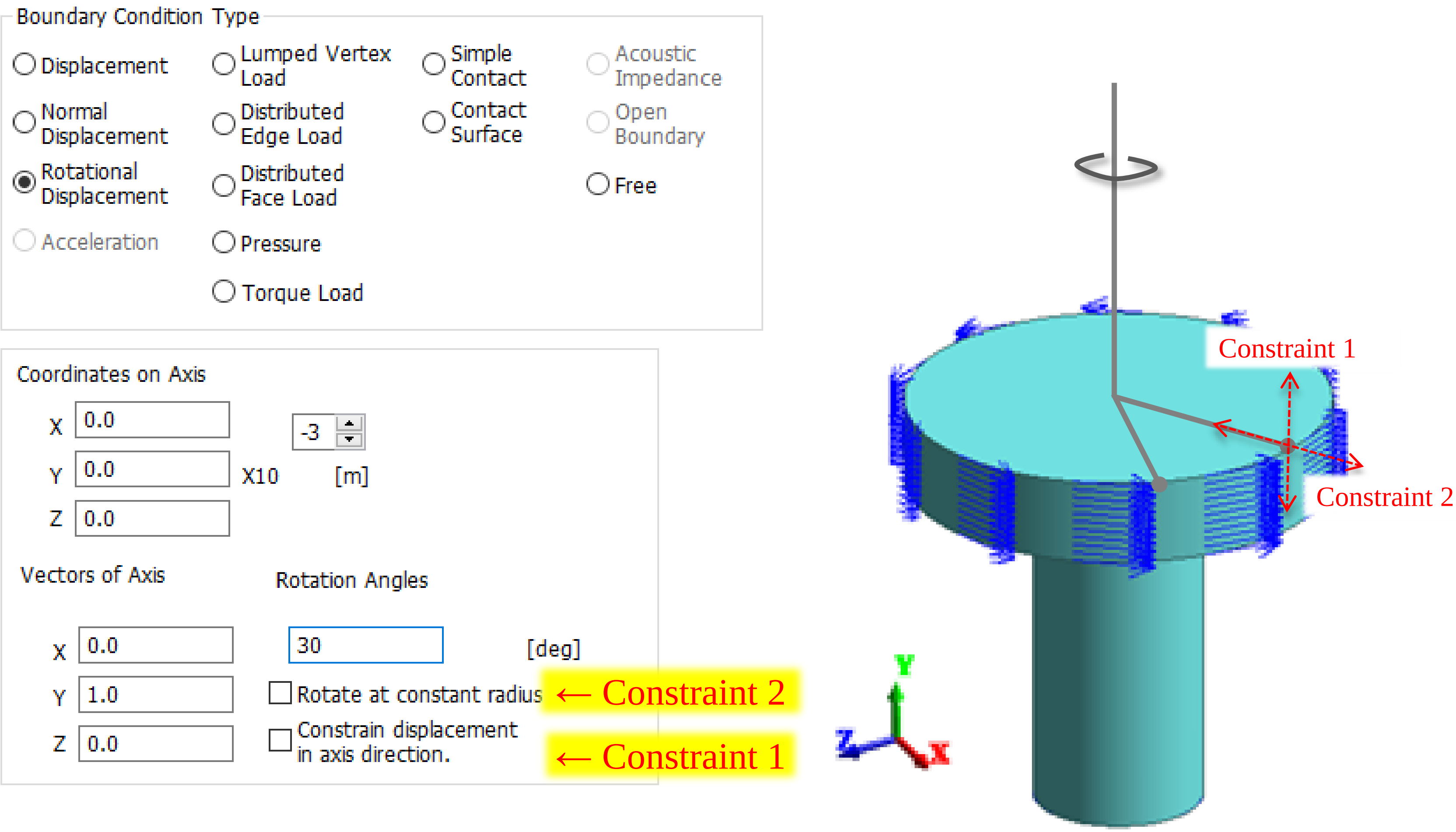


X- and Y-direction are not constrained.



Y-direction is constrained to zero.

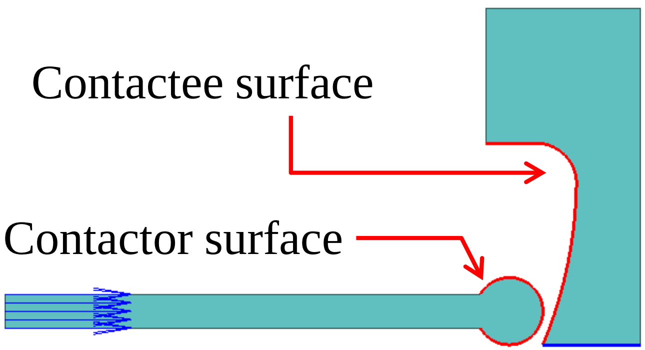
Rotational angle is specified.

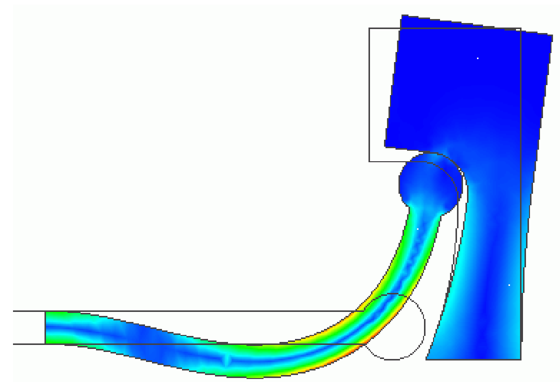


Boundary Condition [Contact]

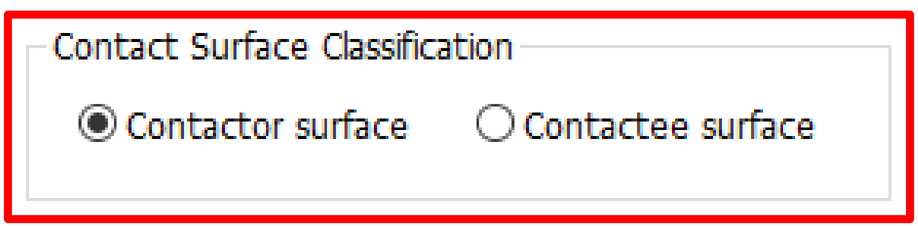
Murata Software

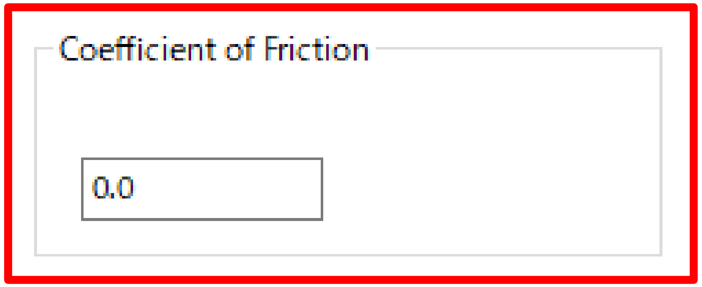
Boundary condition of contactor and contactee surface must be set. Otherwise, the bodies cannot recognize each other.

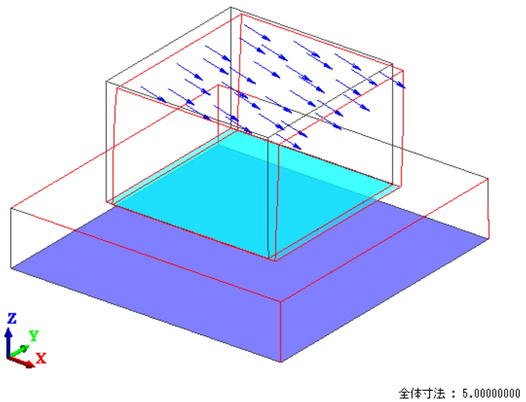




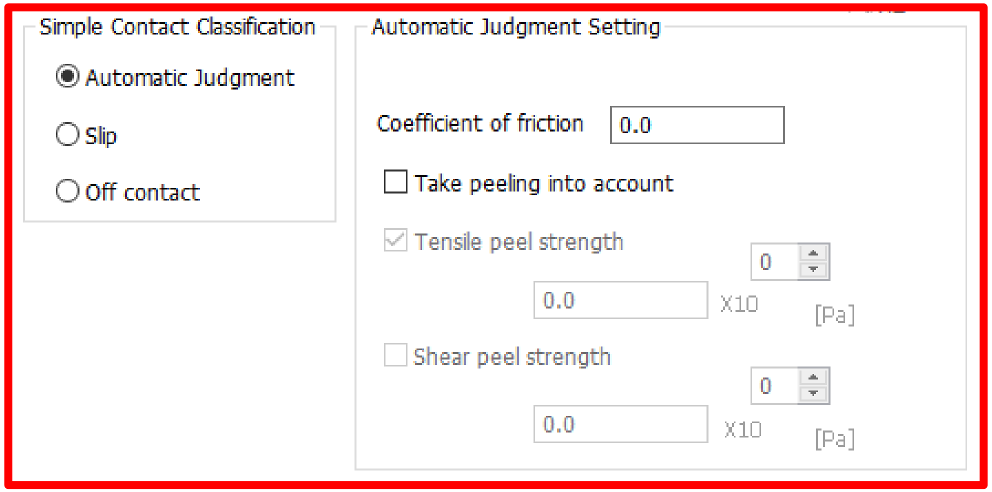
Contact Analysis with Large Deformation (Example 28)







Frictional Contact
(Example 43)



If the faces are contacting, simple contact boundary condition can be applied.

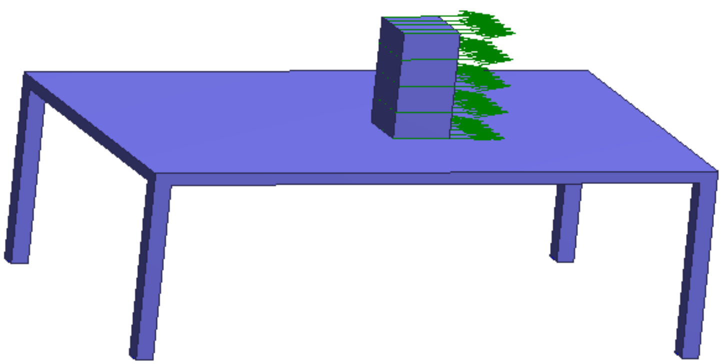
Advantages:
- Shorter calculation time than the usual contact setting.
- Peeling can be taken into account.

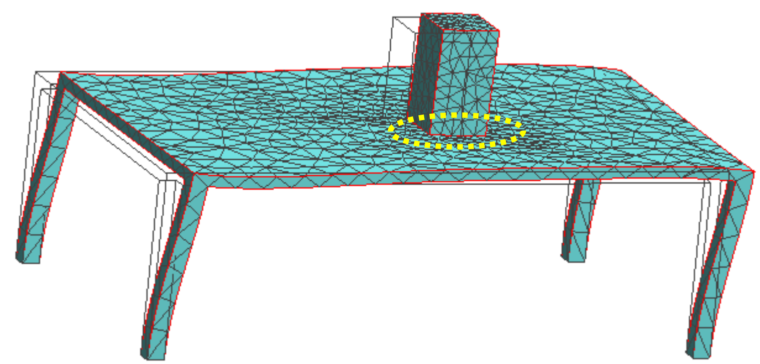
Disadvantages:
- If usual contact and simple contact coexist, calculation will become longer.
- Calculation accuracy is deteriorated if a slip larger than one mesh occurs in the model.

Contact & Bond

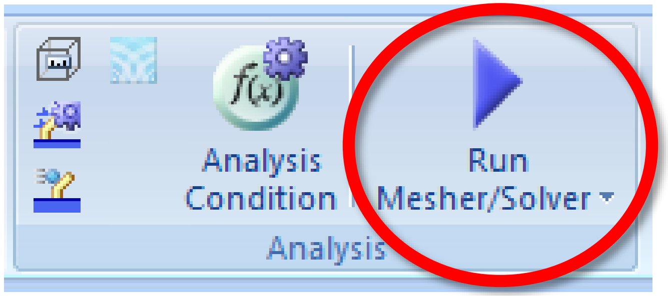
When drawing a model, if two bodies contact each other graphically, and no boundary conditions are set on the boundary, they are considered to be *bonded*.

If you don’t want them bonded or want to take slip into account, set a boundary condition of either simple contact or contactor/contactee face.





Flows of Stress Analysis



Meshes are created automatically

① Draw Analysis Model
(can import CAD data)

② Select Analysis Type

③ Auxiliary Options

④ Set Material Property to Each Body

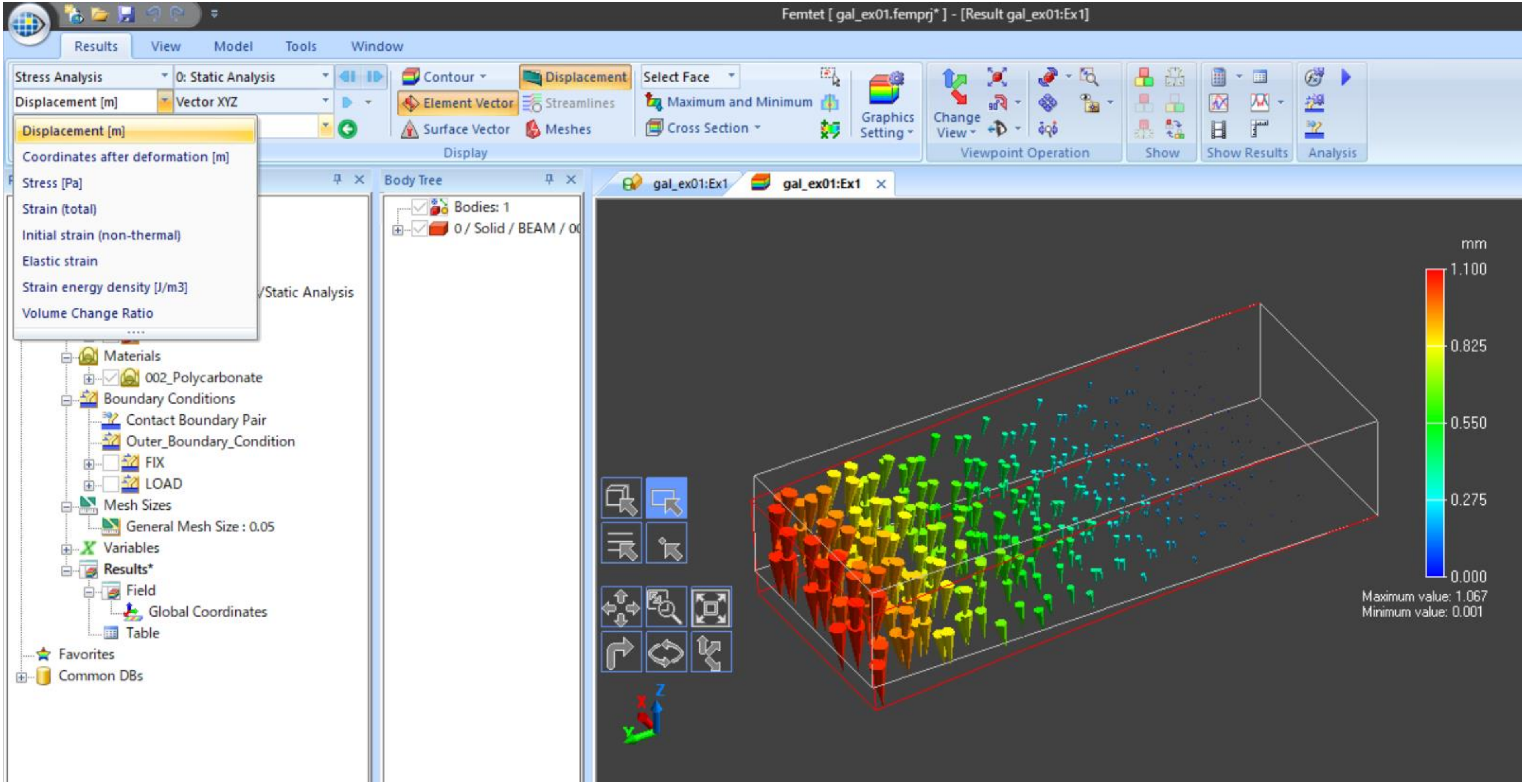
⑤ Set Body Attribute to Each Body

⑥ Set Boundary Condition (such as applied force)

⑦ Run Solver

Results Display

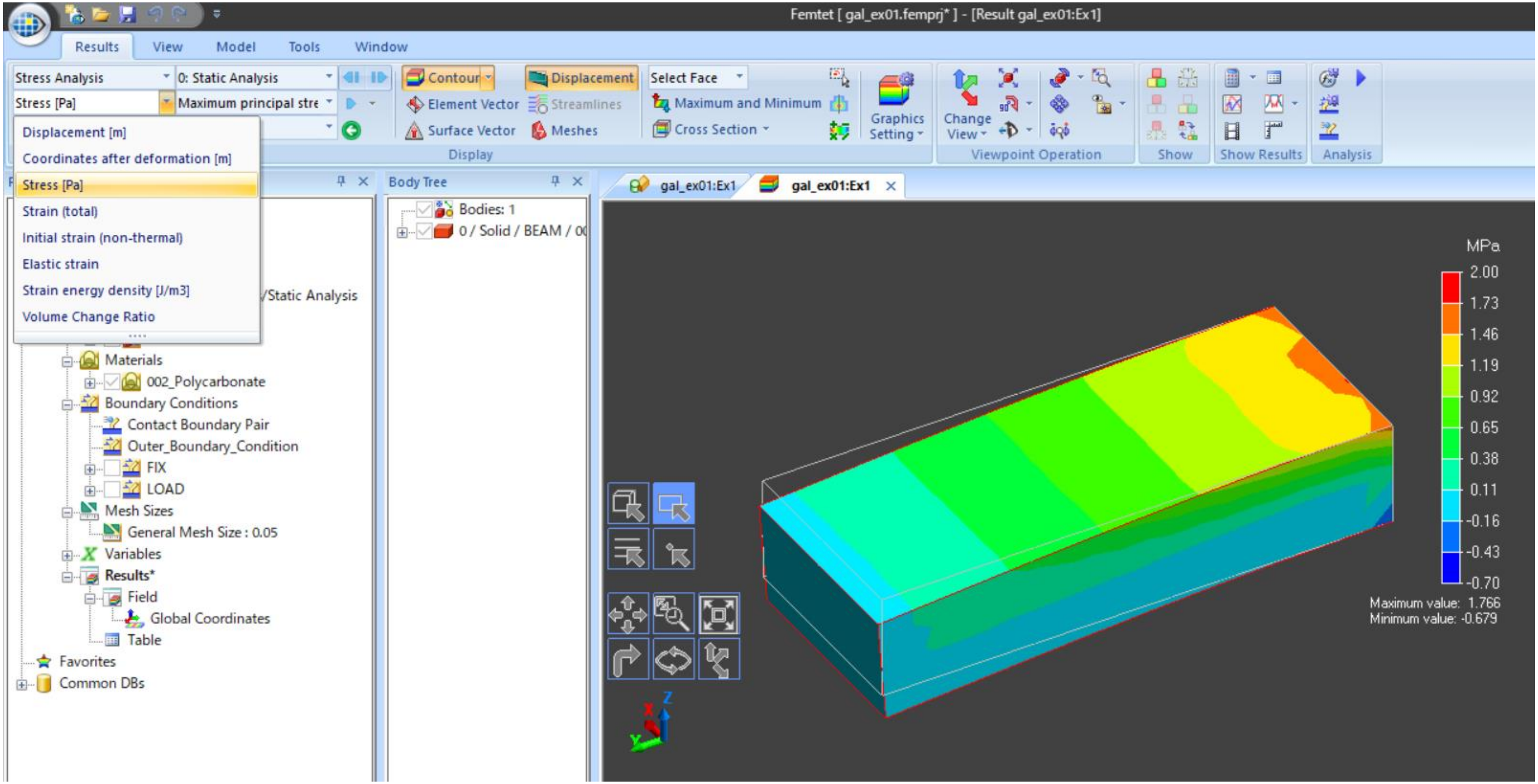
Murata Software



Vector Display

Results Display

Murata Software



Color Contour

Results Table

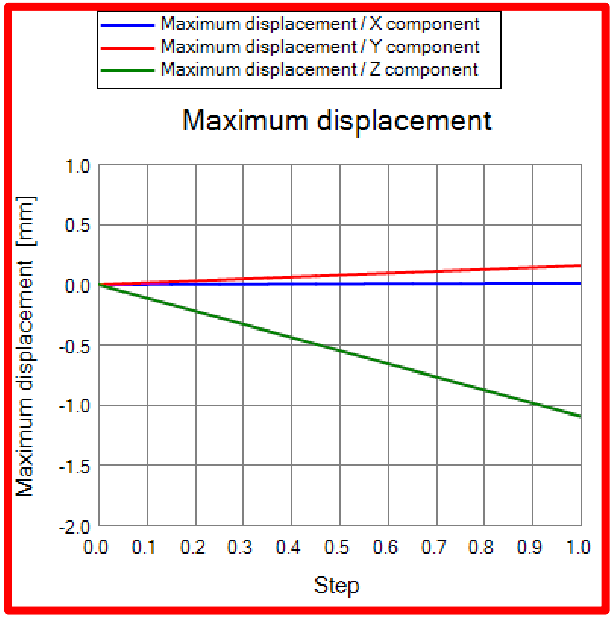
The results are output to the numerical summary table.
Graphing is possible from the table.

Table

Convergence judgment | External/Reactive Force [N] | Strain energy [J] | Maximum displacement [m] | Maximum stress [Pa] | FEM Info

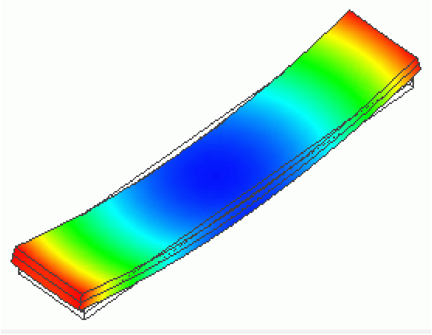
	Step	Maximum displacement / X component	Maximum displacement / Y component	Maximum displacement / Z component
0: 0.05[Step]	5.000e-2	6.865e-7	8.063e-6	-5.464e-5
1: 0.1[Step]	1.000e-1	1.373e-6	1.613e-5	-1.093e-4
2: 0.15[Step]	1.500e-1	2.060e-6	2.419e-5	-1.639e-4
3: 0.2[Step]	2.000e-1	2.746e-6	3.225e-5	-2.185e-4
4: 0.25[Step]	2.500e-1	3.433e-6	4.031e-5	-2.732e-4
5: 0.3[Step]	3.000e-1	4.119e-6	4.838e-5	-3.278e-4
6: 0.35[Step]	3.500e-1	4.806e-6	5.644e-5	-3.825e-4
7: 0.4[Step]	4.000e-1	5.492e-6	6.450e-5	-4.371e-4
8: 0.45[Step]	4.500e-1	6.179e-6	7.256e-5	-4.917e-4
9: 0.5[Step]	5.000e-1	6.865e-6	8.063e-5	-5.464e-4
10: 0.55[Step]	5.500e-1	7.552e-6	8.869e-5	-6.010e-4
11: 0.6[Step]	6.000e-1	8.239e-6	9.675e-5	-6.556e-4
12: 0.65[Step]	6.500e-1	8.925e-6	1.048e-4	-7.103e-4
13: 0.7[Step]	7.000e-1	9.612e-6	1.129e-4	-7.649e-4
14: 0.75[Step]	7.500e-1	1.030e-5	1.209e-4	-8.196e-4
15: 0.8[Step]	8.000e-1	1.098e-5	1.290e-4	-8.742e-4
16: 0.85[Step]	8.500e-1	1.167e-5	1.371e-4	-9.288e-4
17: 0.9[Step]	9.000e-1	1.236e-5	1.451e-4	-9.835e-4
18: 0.95[Step]	9.500e-1	1.304e-5	1.532e-4	-1.038e-3
19: 1[Step]	1.000e+0	1.373e-5	1.613e-4	-1.093e-3

Stress Analysis | Show all results summary | Display Options | Graph | Export



Mechanical Stress Analysis: Appendix

Step/Thermal Load Setting

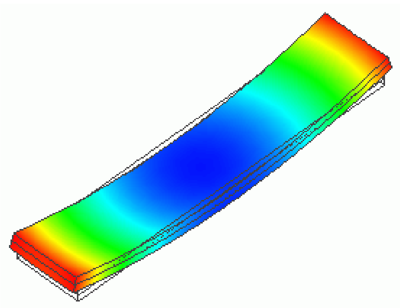


Bimetal under Thermal Load
(Example 7)

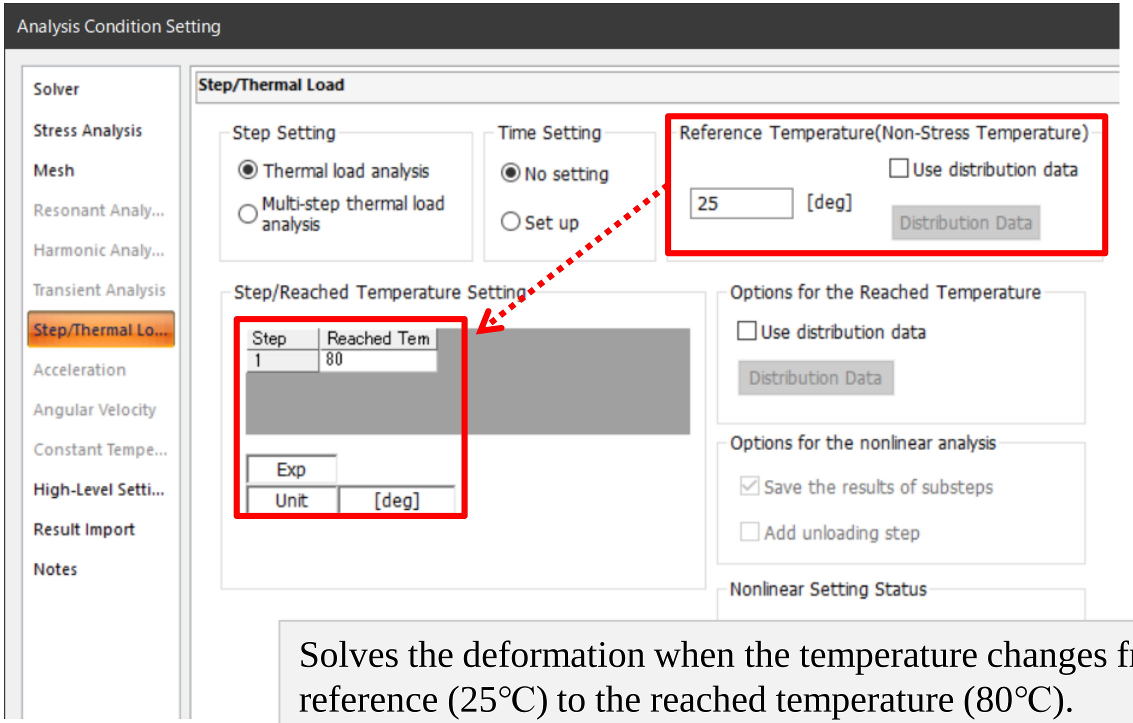
Thermal Load

Murata Software

Deformation generated by expansion or shrinkage due to the temperature change is solved.



[Step/Thermal Load] tab for the setup.

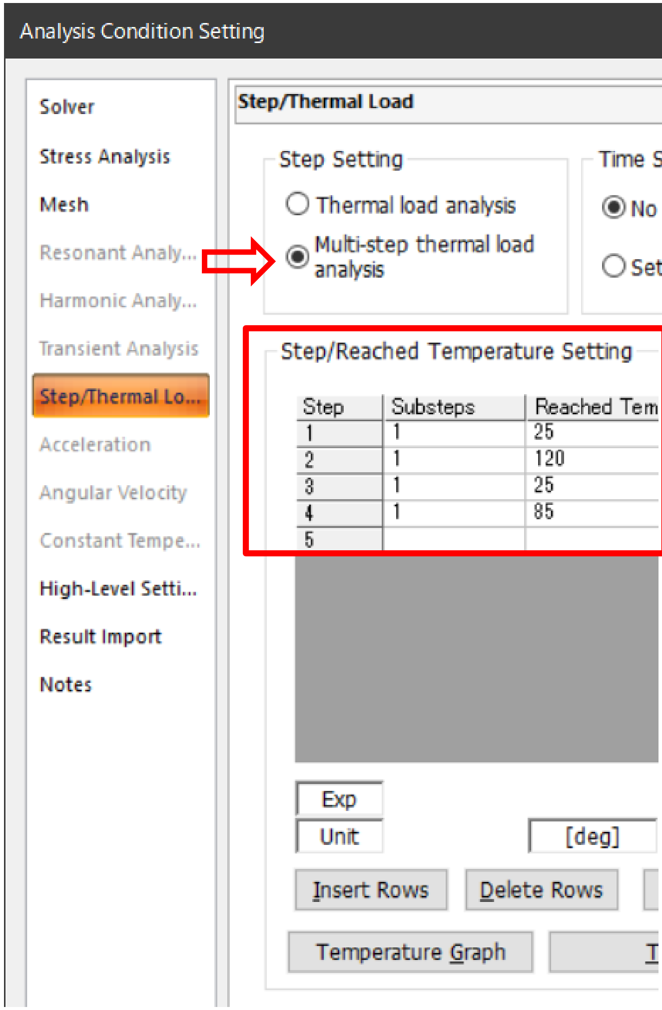


Solves the deformation when the temperature changes from the reference (25°C) to the reached temperature (80°C).

Multi-Step Thermal Load

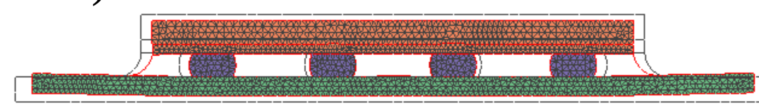
Murata Software

Temperature change over the multiple steps is solved.

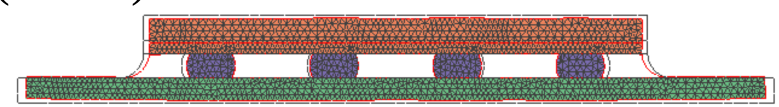


Example 10: Soldering Process of IC

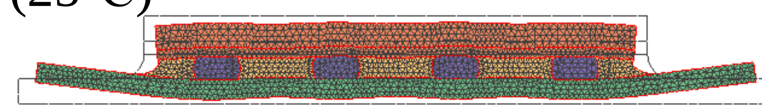
Step 1(25°C)



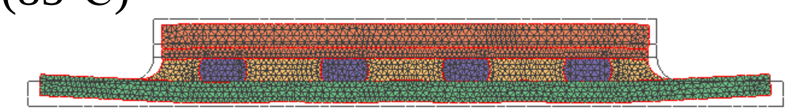
Step 2 (120°C)



Step 3 (25°C)



Step 4 (85°C)

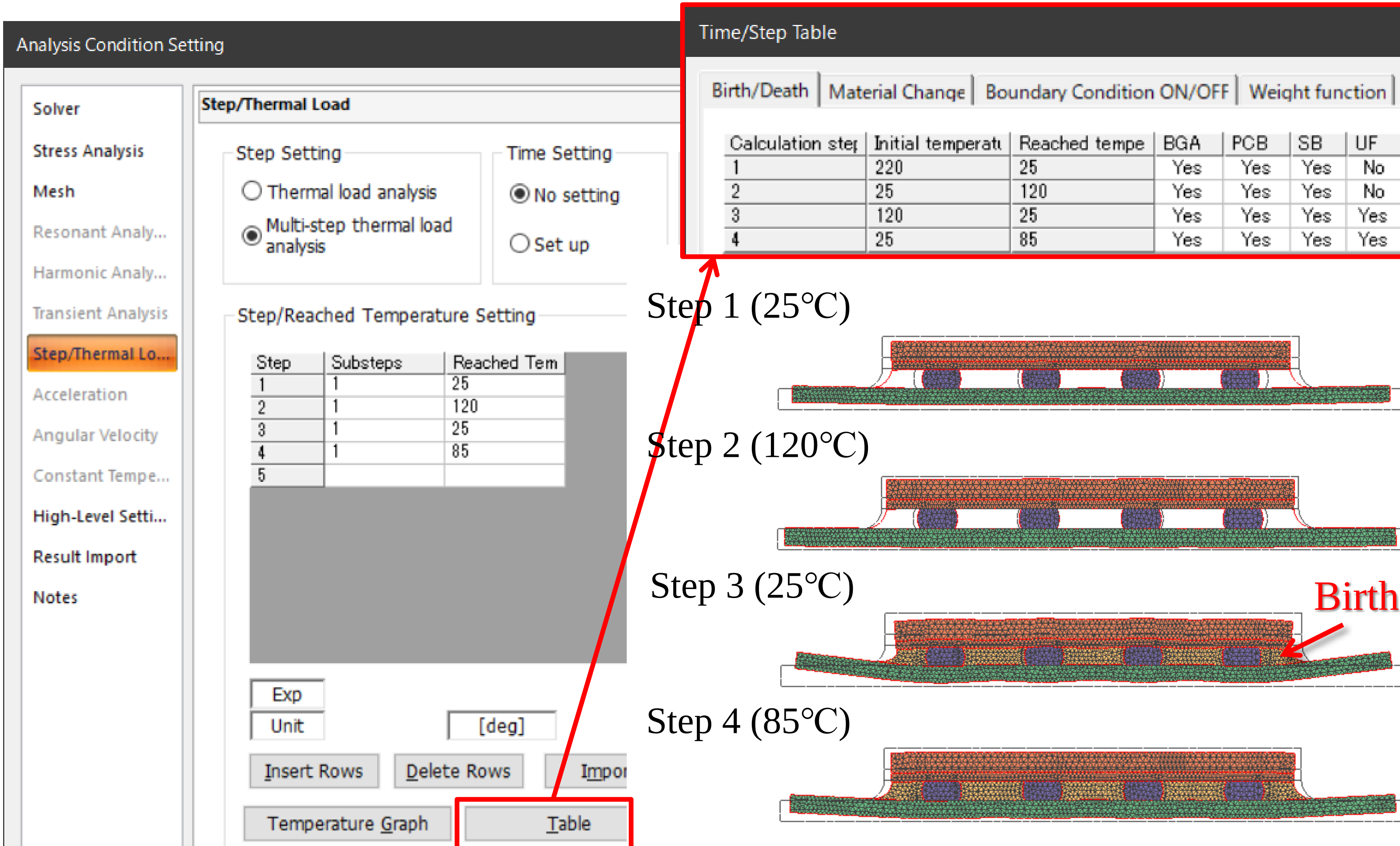


Birth/Death Setting

Murata Software

Birth and death can be set in the Time/Step Table

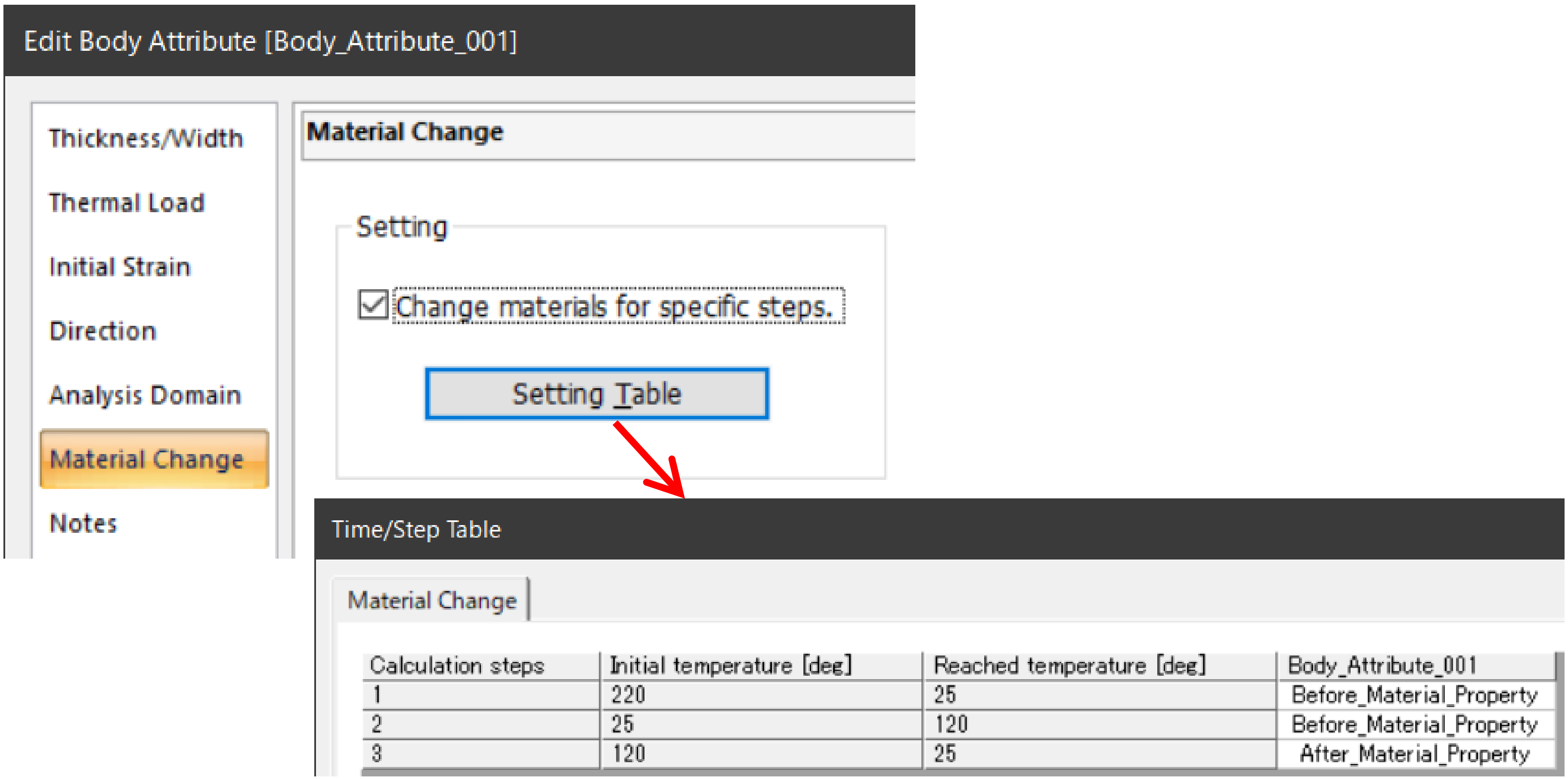
* Can be set on the [Analysis Doman] tab in the body attribute setting for each body as well.



Material Property Change

Murata Software

Material property can be changed for each calculation step.
The setting is done on the [Material Change] tab of the body attribute setting.

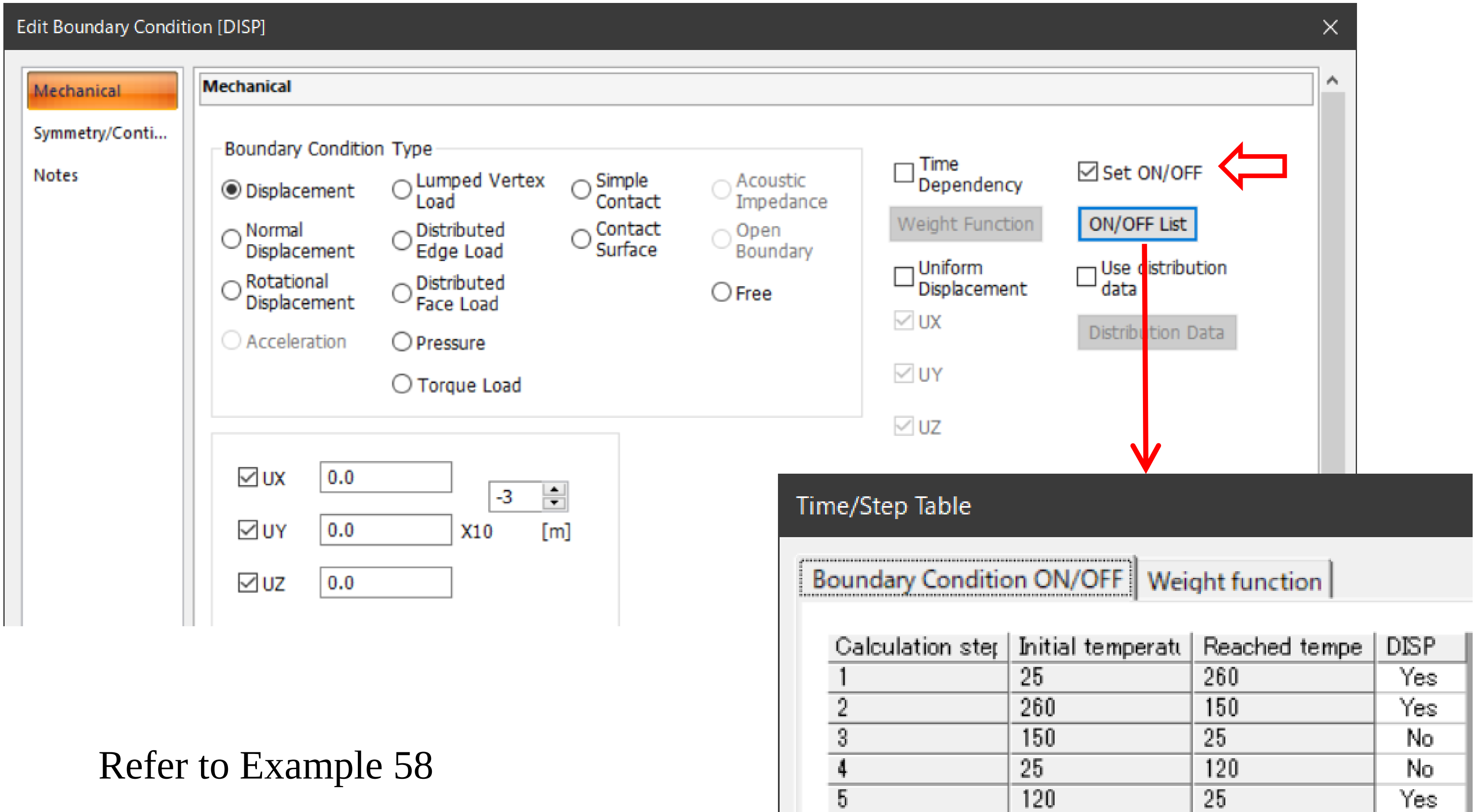


Boundary Condition ON/OFF

Murata Software

Boundary condition can be set ON or OFF for each step.

*Can be set on the [Step/Thermal Load] table as well.



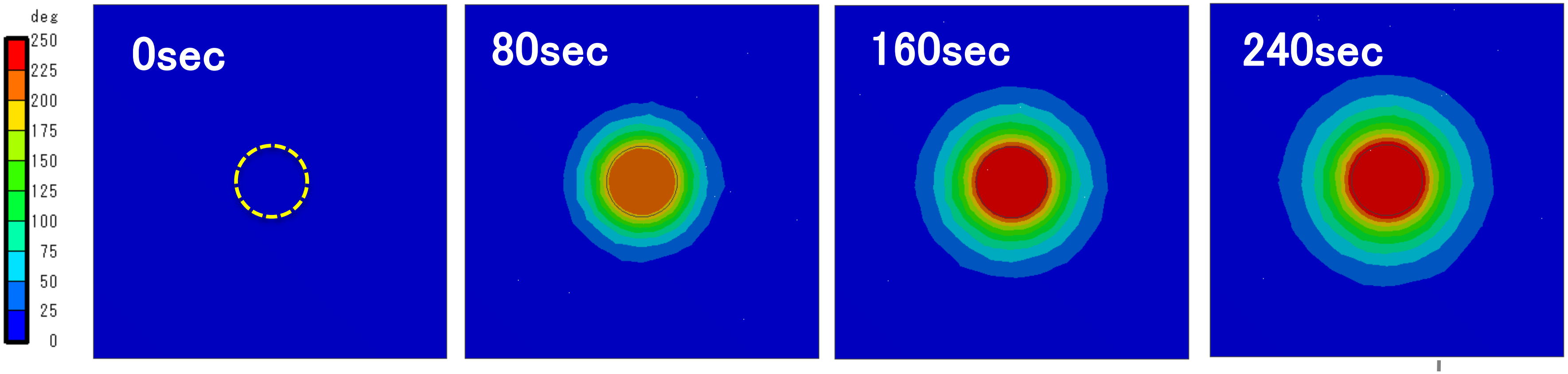
Refer to Example 58

2. Thermal Analysis [Watt] Overview

- Analysis Condition
- Material Property
- Body Attribute
- Boundary Condition
- Results Display

Analysis Type

Murata Software



A Heating Element of 10W in the center with Temperature of 0°C around Four Edges (Example 4)

Transient Analysis

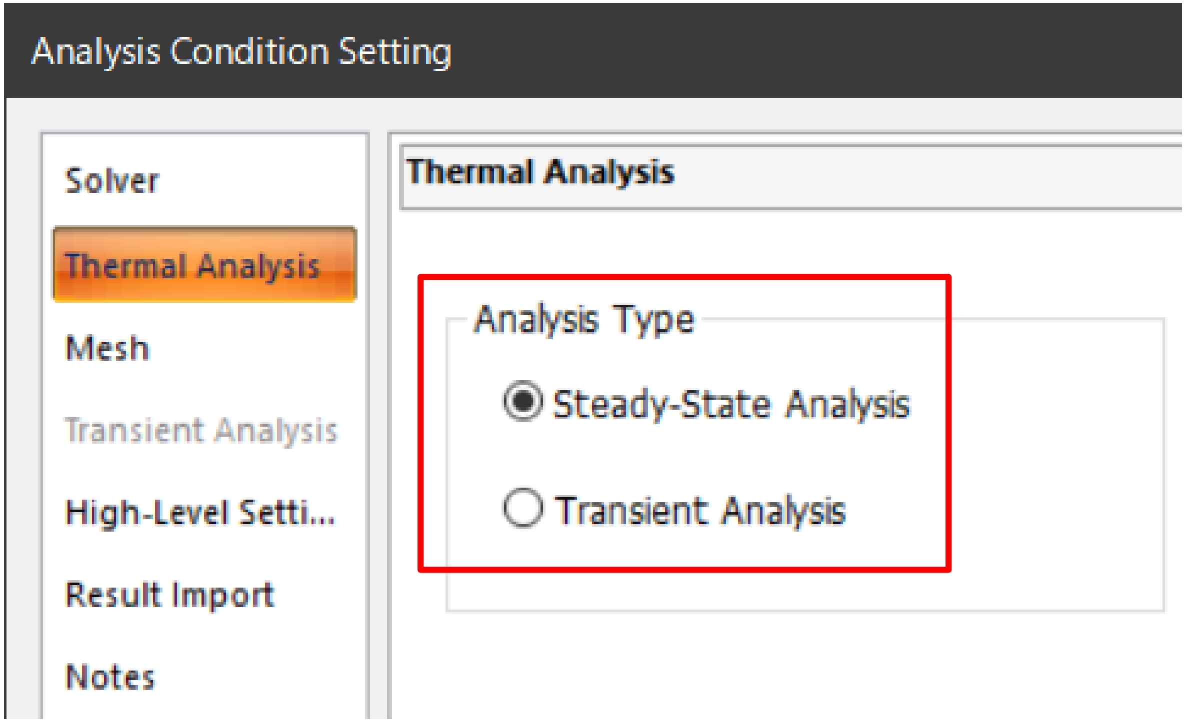
Solves the thermal equation over time steps from the initial temperature.
Temperature change over time can be observed.

Steady-state Analysis

Solves a reached state after long time elapsed.
The steady state can be calculated straight away by solving the thermal equation assuming that the temperature does not change over time.

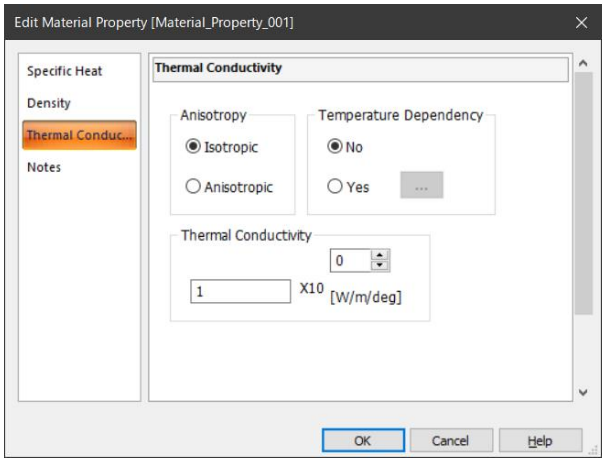
*Steady-state analysis cannot solve the problem where the steady state does not exist.
(example: where the temperature continues to go up permanently)

Analysis Type

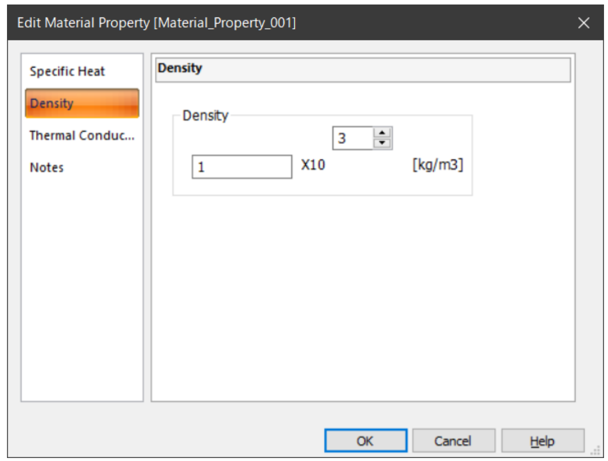


Material Property

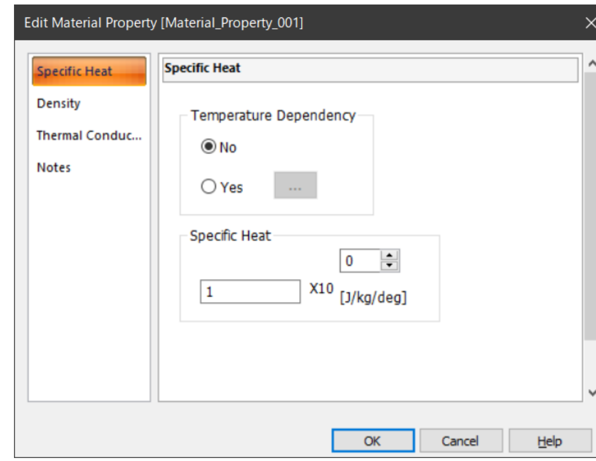
Thermal Conductivity



Density



Specific Heat



Thermal Conductivity [W/m/deg]: Examples

Au	315
Ag	427
Cu	402
SUS301	16

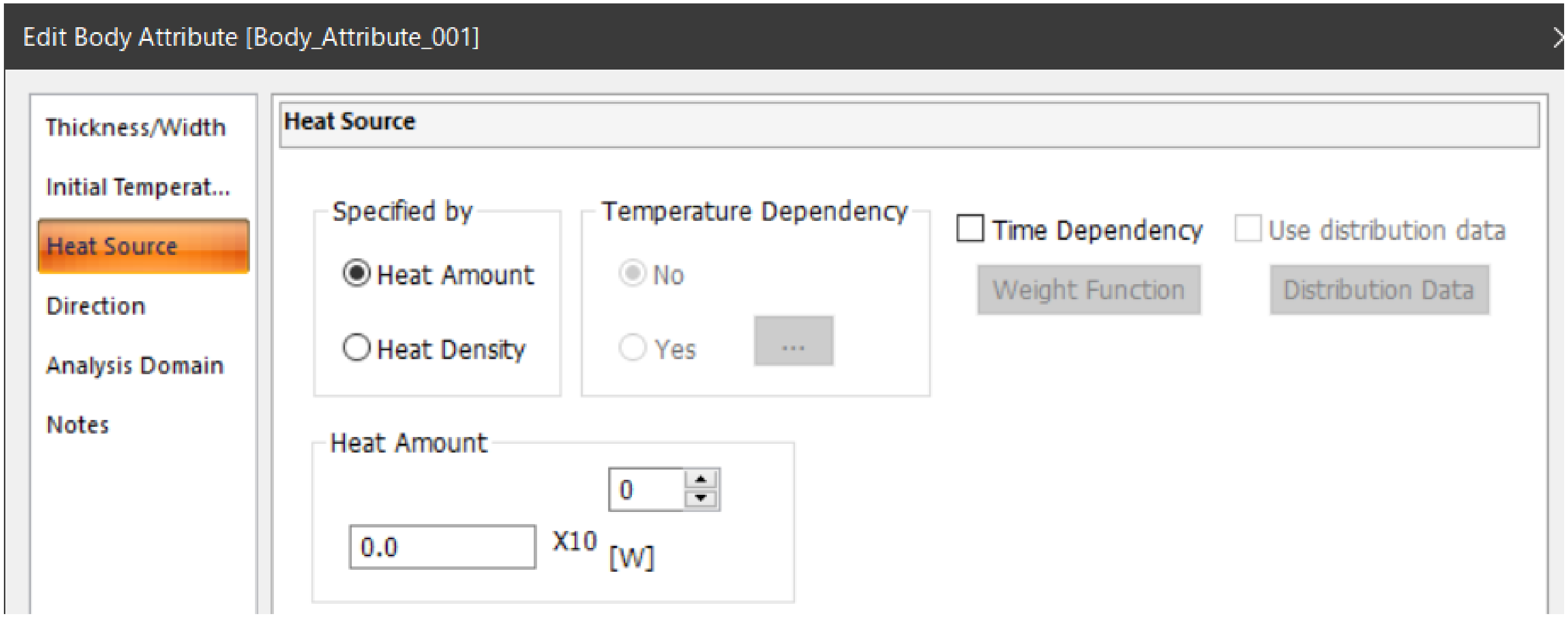
Polycarbonate	0.2
Polyethylene	0.22
Glass epoxy	0.293

In the transient analysis, all of thermal conductivity, density, and specific heat must be set.

In the steady-state analysis, only thermal conductivity must be set.

Body Attribute

Heat source can be set for each body.

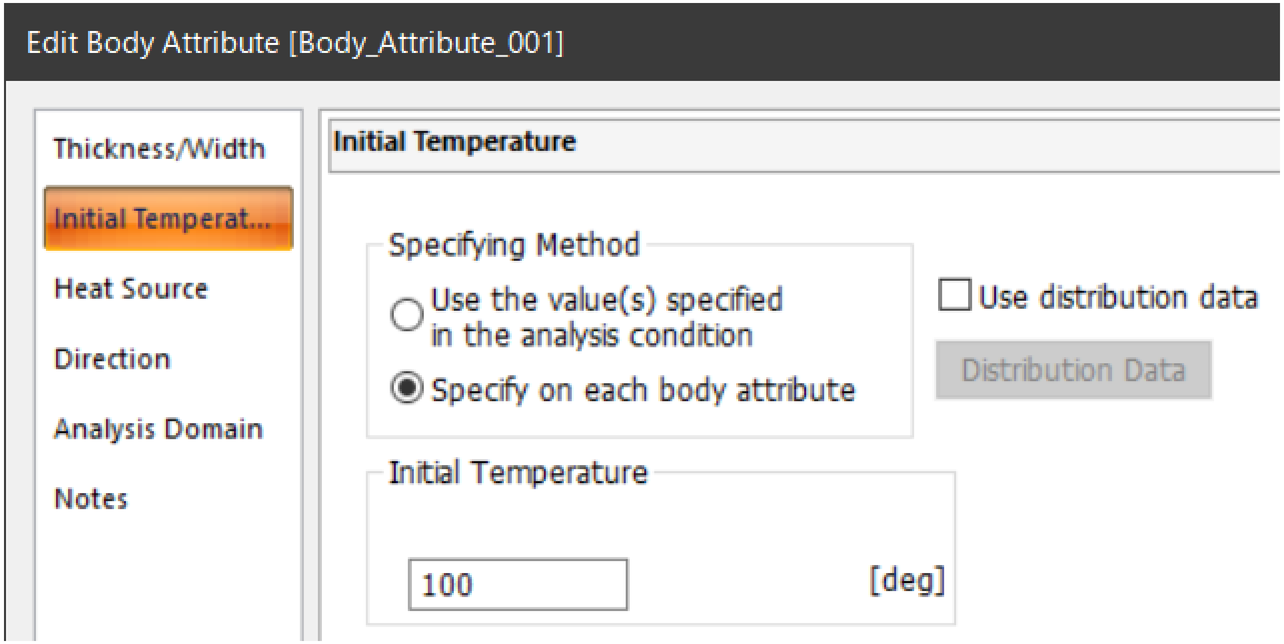


- Can be specified either by [Heat Amount] or [Heat Density].
- Spatial distribution of the heat density can be given.
- Temperature dependency can be given to the heat density.
- Time dependency can be given to the heat source.

Body Attribute

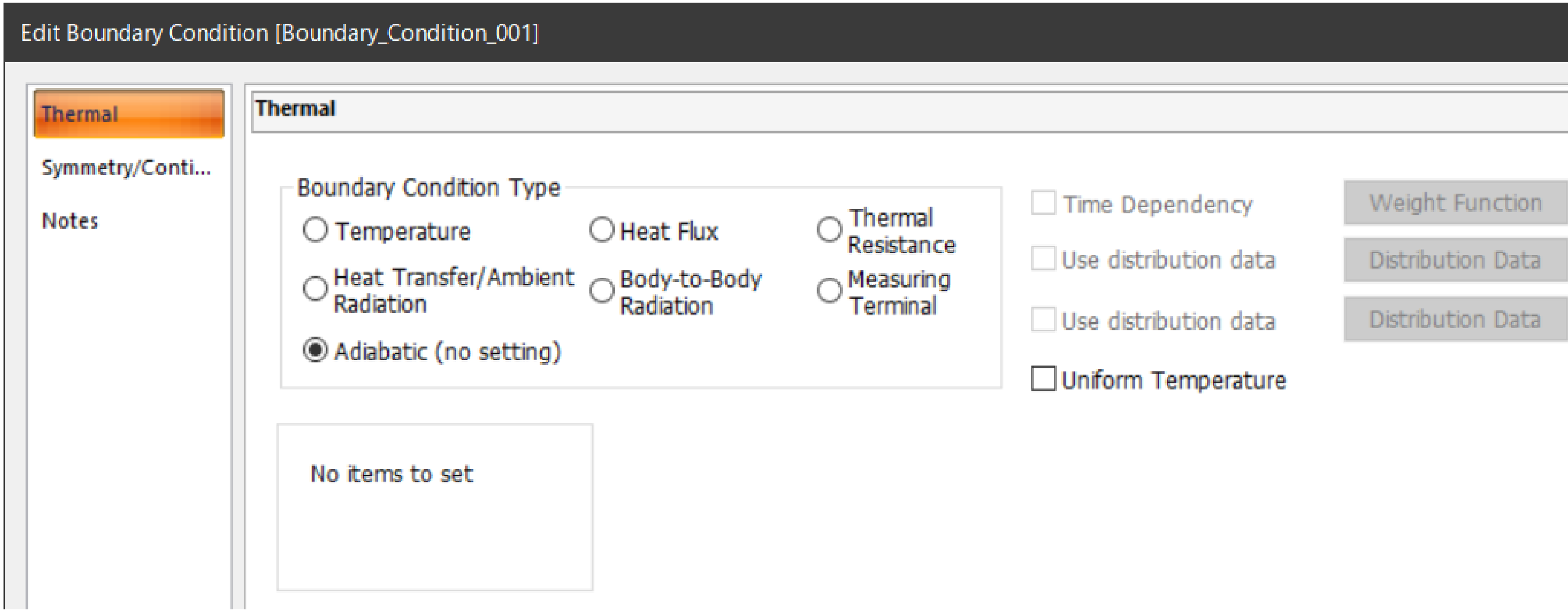
Murata Software

Initial temperature can be set for each body in the transient analysis.



By default, the same initial temperature is set to all bodies.
In the body attribute setting, specific initial temperature can be given to each body.

Boundary Condition

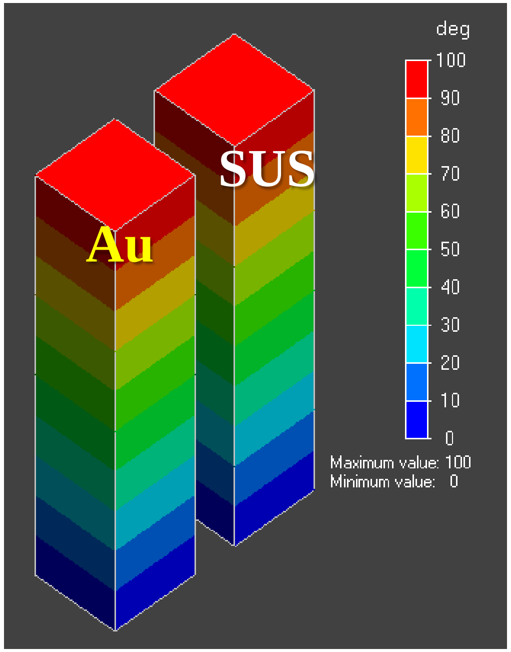


Boundary Condition: Temperature

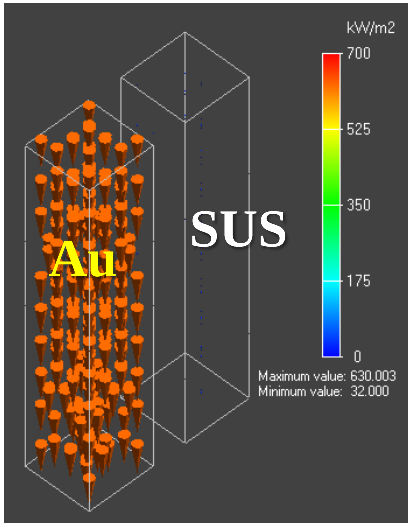
(Example 1)
Top faces of bars: 100°C ←
Bottom faces of bars: 0°C ←
Sides of bars: Adiabatic

Temperature boundary condition

Temperature Distribution



Heat Flux



Heat flux flows in from the top face and flows out at the bottom face.
(Heat flux) = (Thermal conductivity) x (Temperature gradient)

Boundary Condition: Heat Flux

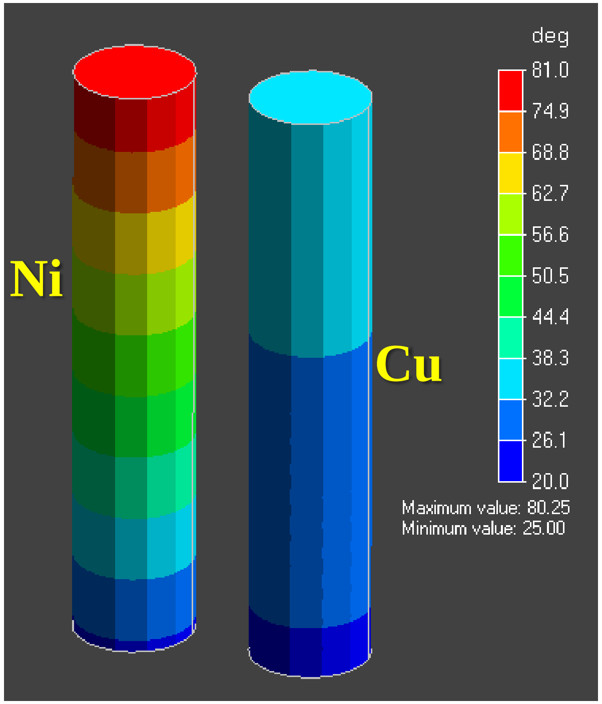
(Example 13)

Top faces of bars: Heat flux 1 x10^5 [w/m^2] inflow ← Boundary condition: Heat flux inflow

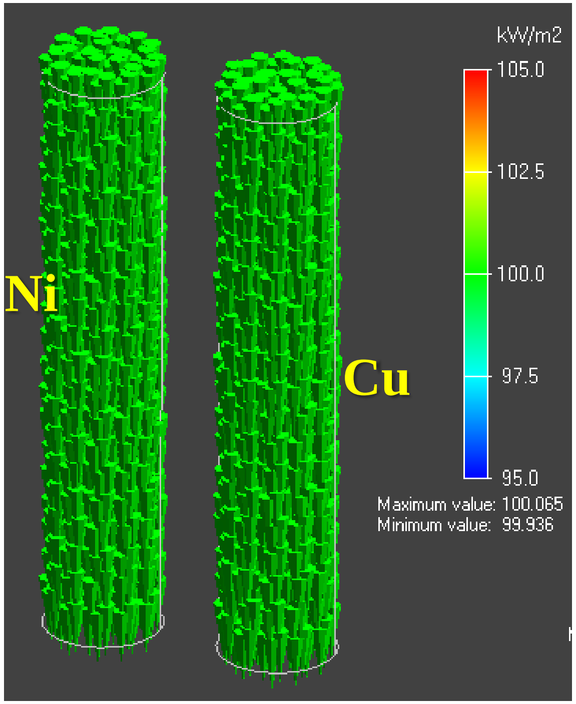
Bottom faces of bars: 25°C

Sides of bars: Adiabatic ← Boundary condition: No heat flux inflow

Temperature Distribution



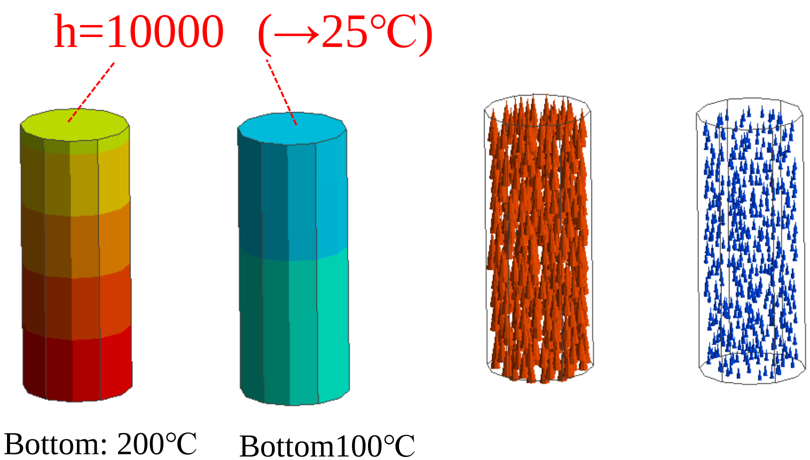
Heat Flux



Boundary Condition: Heat Transfer Coefficient

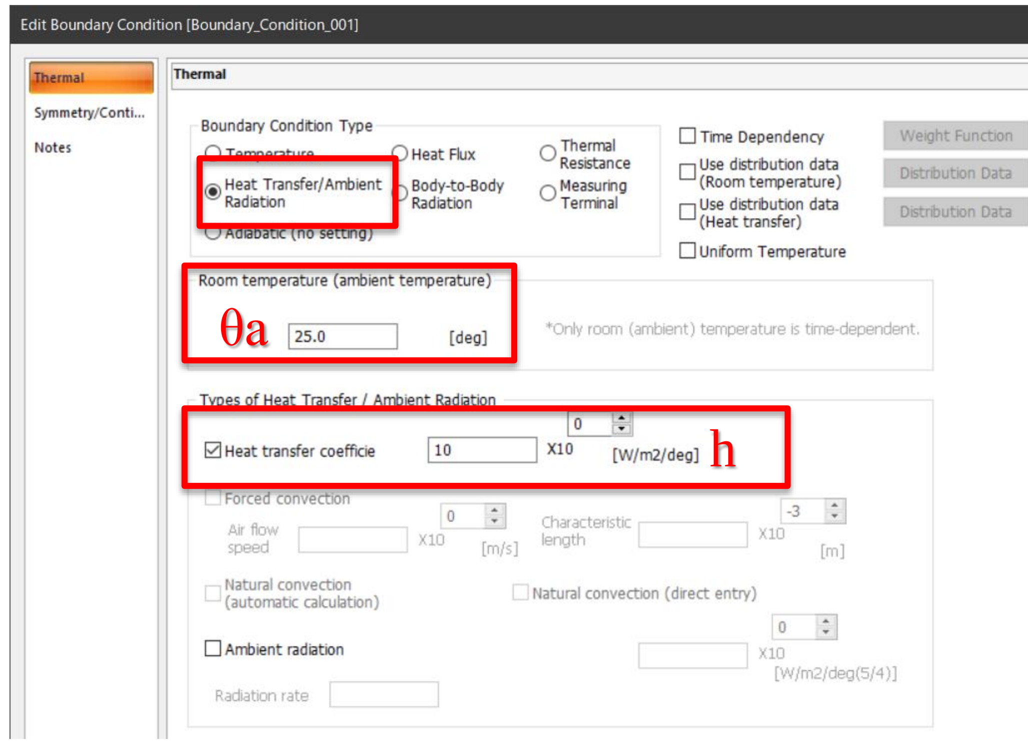
Heat transfer (heat flux outflow) = h x (θ - θa)

where h: heat transfer coefficient, θ: temperature at radiating face, θa: ambient temperature



Temperature Distribution

Heat Flux Distribution

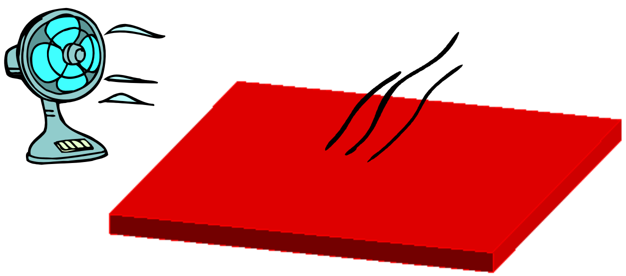


Heat transfer coefficient of the face exposed to the windless air space: 5 ~ 25 [W/m^2K]

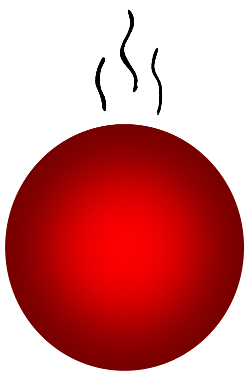
Practical Setting of Heat Transfer

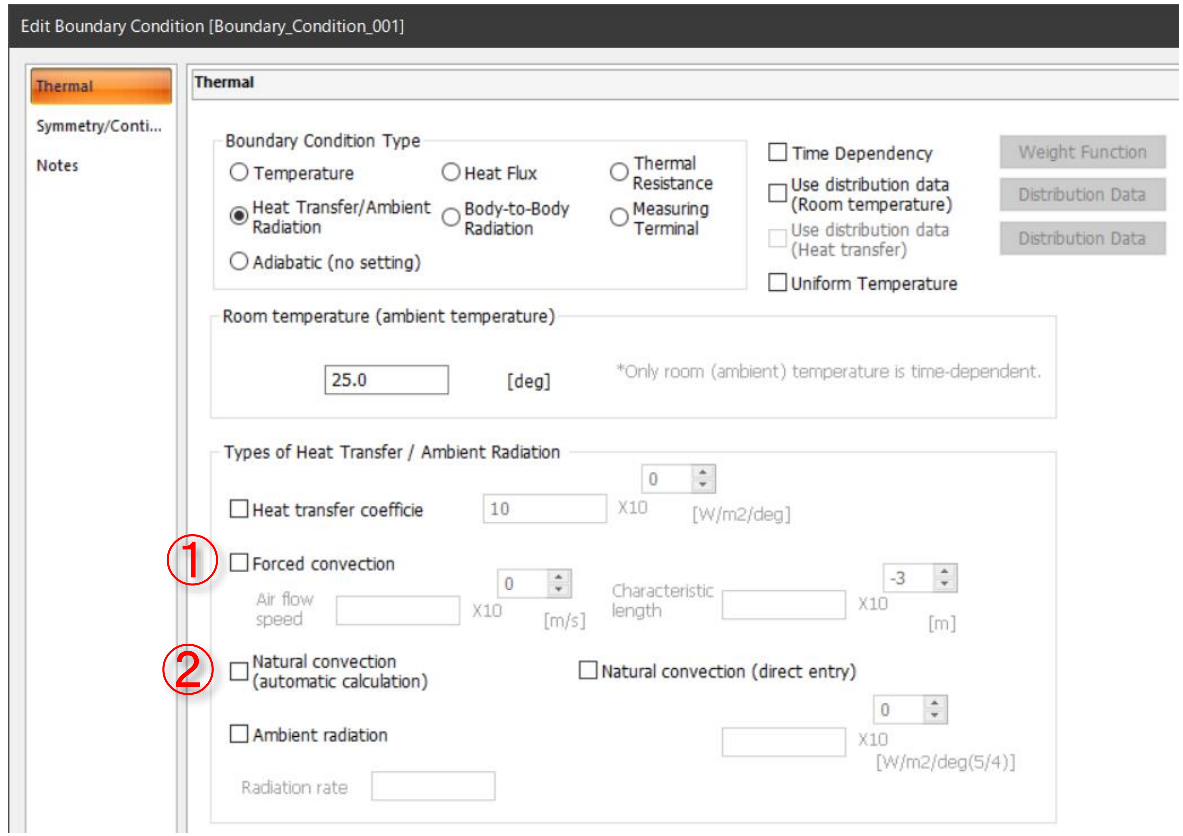
Murata Software

① Forced Convection



② Natural Convection





Boundary Condition: Forced Convection

Equations below are solved.

*Boundary condition is actually an heat transfer coefficient.

<u>Heat transfer by forced convection</u>

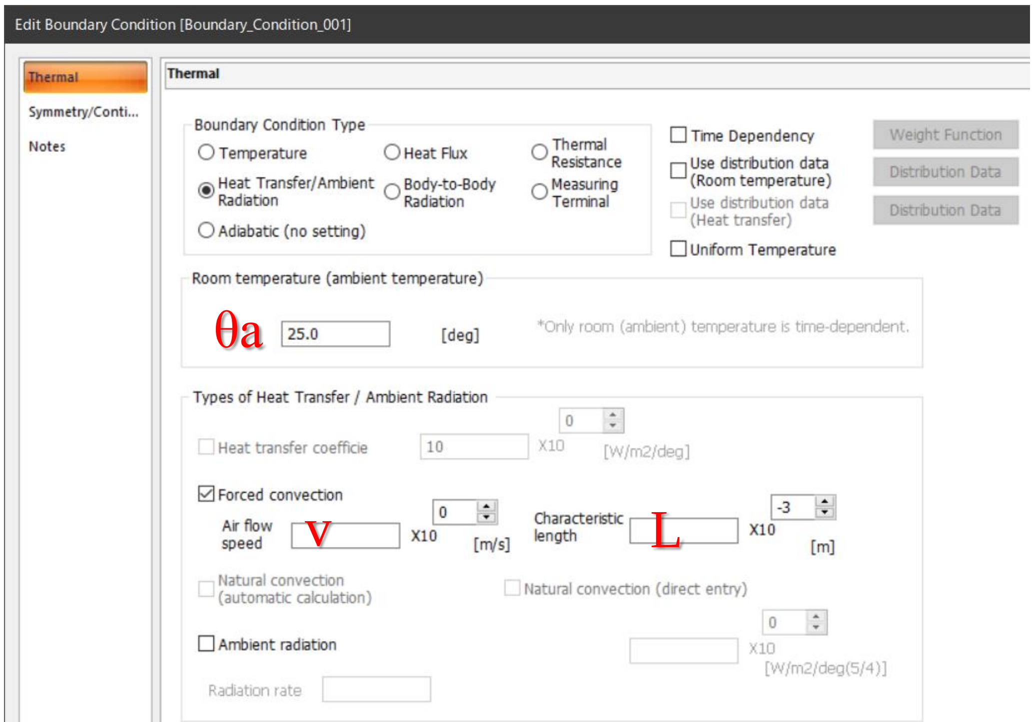
[Heat flux outflow] = h x (θ - θa)

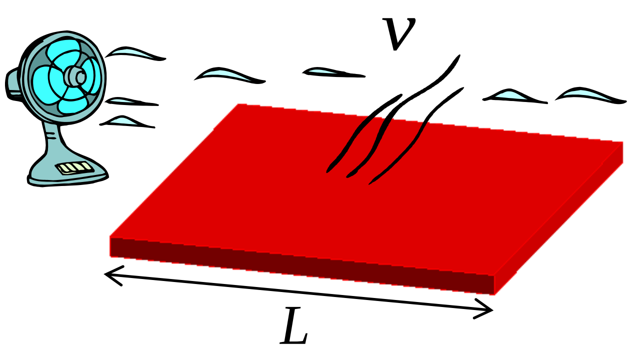
θ: surface temperature
θa: ambient temperature

<u>Average heat transfer coefficient h</u>

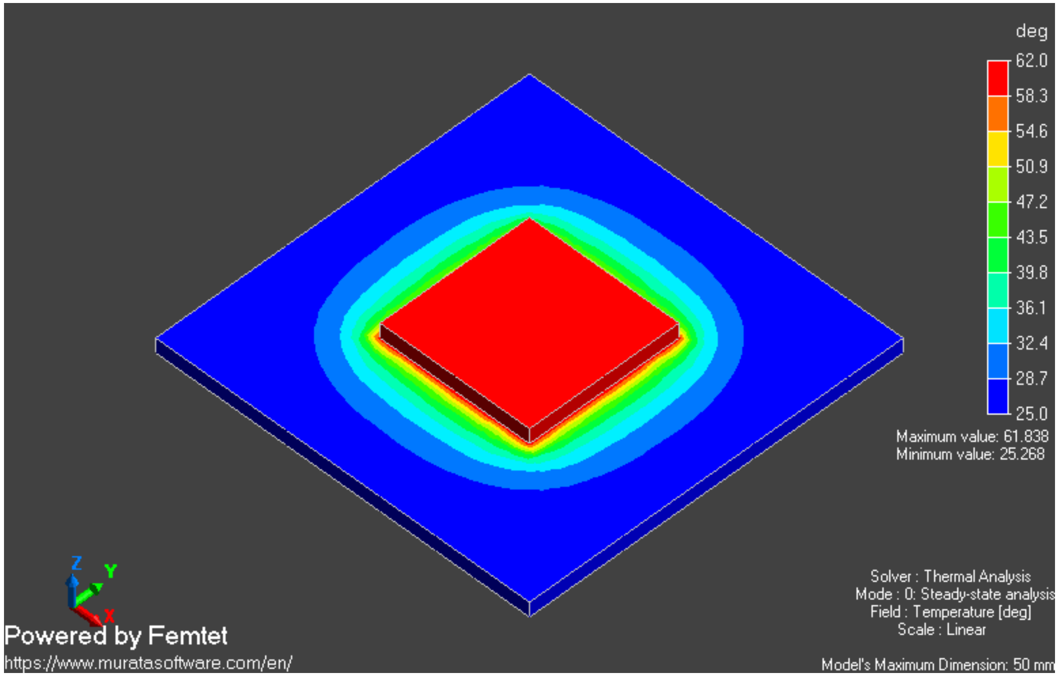
$$h = 3.86\sqrt{v / L}$$

v: velocity
L: typical length





Boundary Condition: Forced Convection



Heat transfer by forced convection

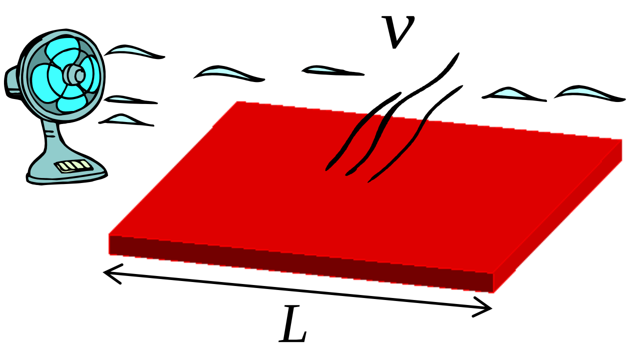
[Heat flux outflow] = h x (θ - θa)

Average heat transfer coefficient h

$$h = 3.86\sqrt{v / L}$$

v: velocity

L: typical length

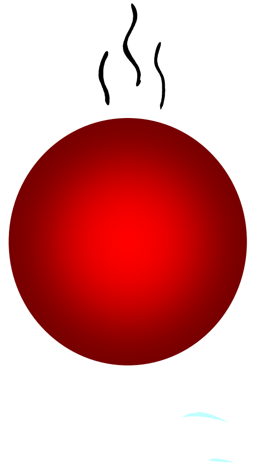


A heat source of 1W is placed on a substrate.
The air is blown in +y direction at 1 [m/s].
The heat transfer by the forced convection is set to the top and bottom face of the substrate. (Example 7)

The overall heat transfer can be calculated since the average heat transfer coefficient is given to the entire surface. But, the distinction of upwind and downwind air flows is not possible.

Boundary Condition: Natural Convection

Natural Convection



Heat transfer by natural convection

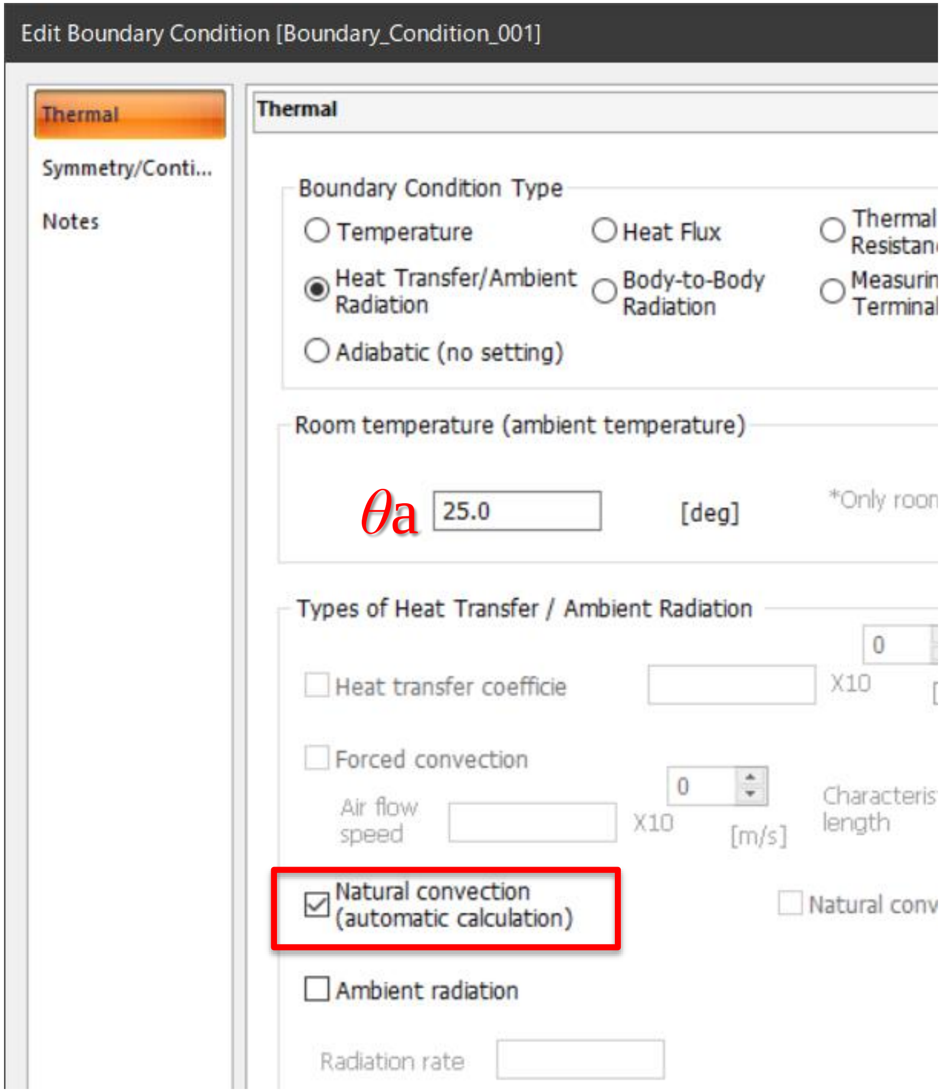
$$[\text{Heat flux outflow}] = h' \times (\theta - \theta a)^{1.25}$$

where
θ: surface temperature
θa: ambient temperature
h': coefficient given by size and direction of face

An object is placed in the wide open space where no forced wind occurs.
It is assumed that the ascending air current is induced by the difference in temperatures.
It is also assumed that the radiating face is exposed to the space large enough.

Boundary Condition: Natural Convection

Murata Software



[Heat flux outflow] = h’ x (θ - θa)$^{1.25}$

where

θ: surface temperature

θa: ambient temperature

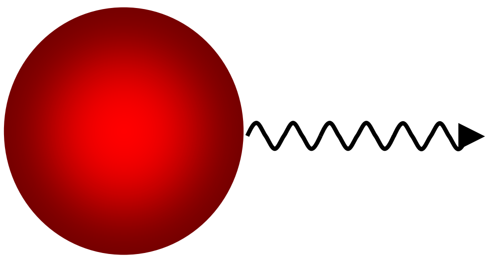
h’: coefficient given by size and direction of face

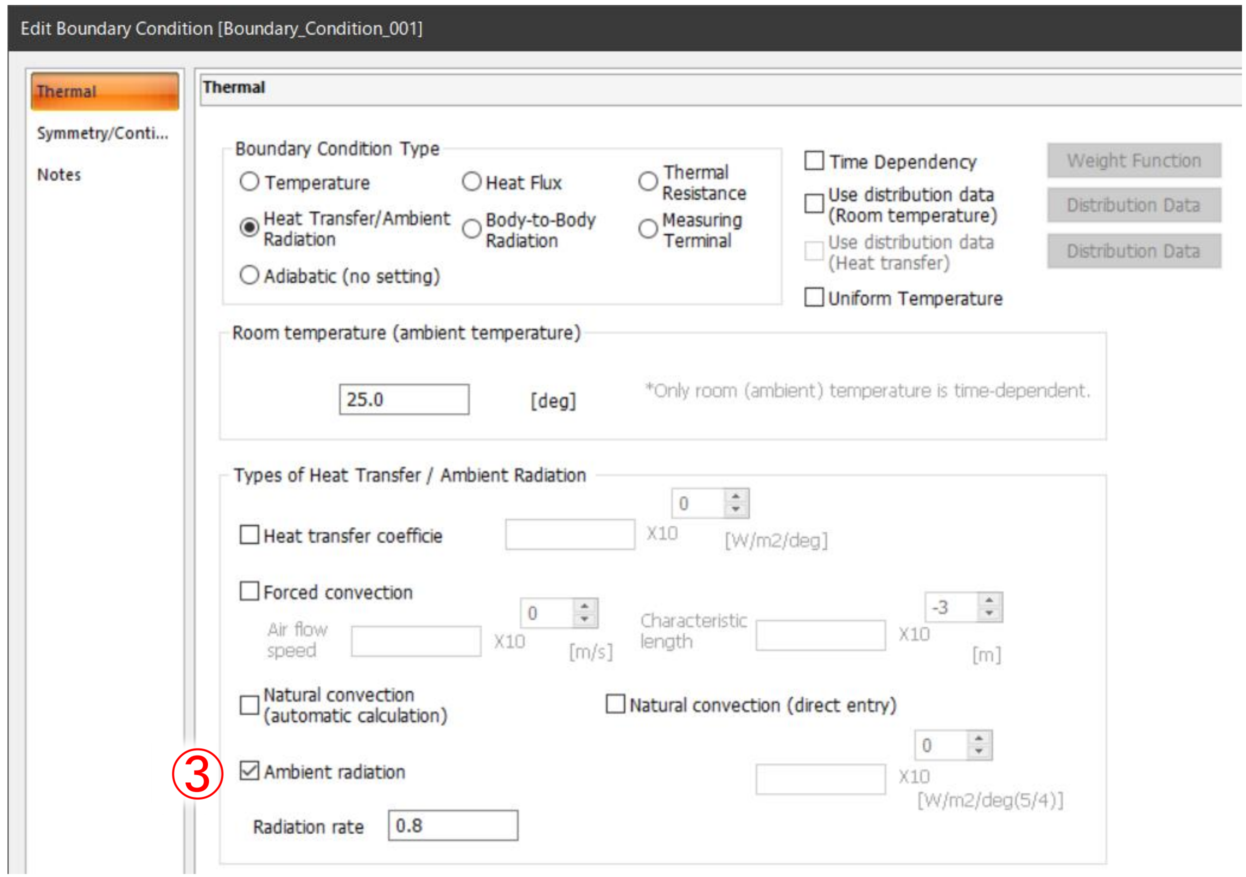
*h’ is automatically calculated.

Boundary Condition: Ambient Radiation

Ambient radiation can be set to the same face as forced or natural convection.

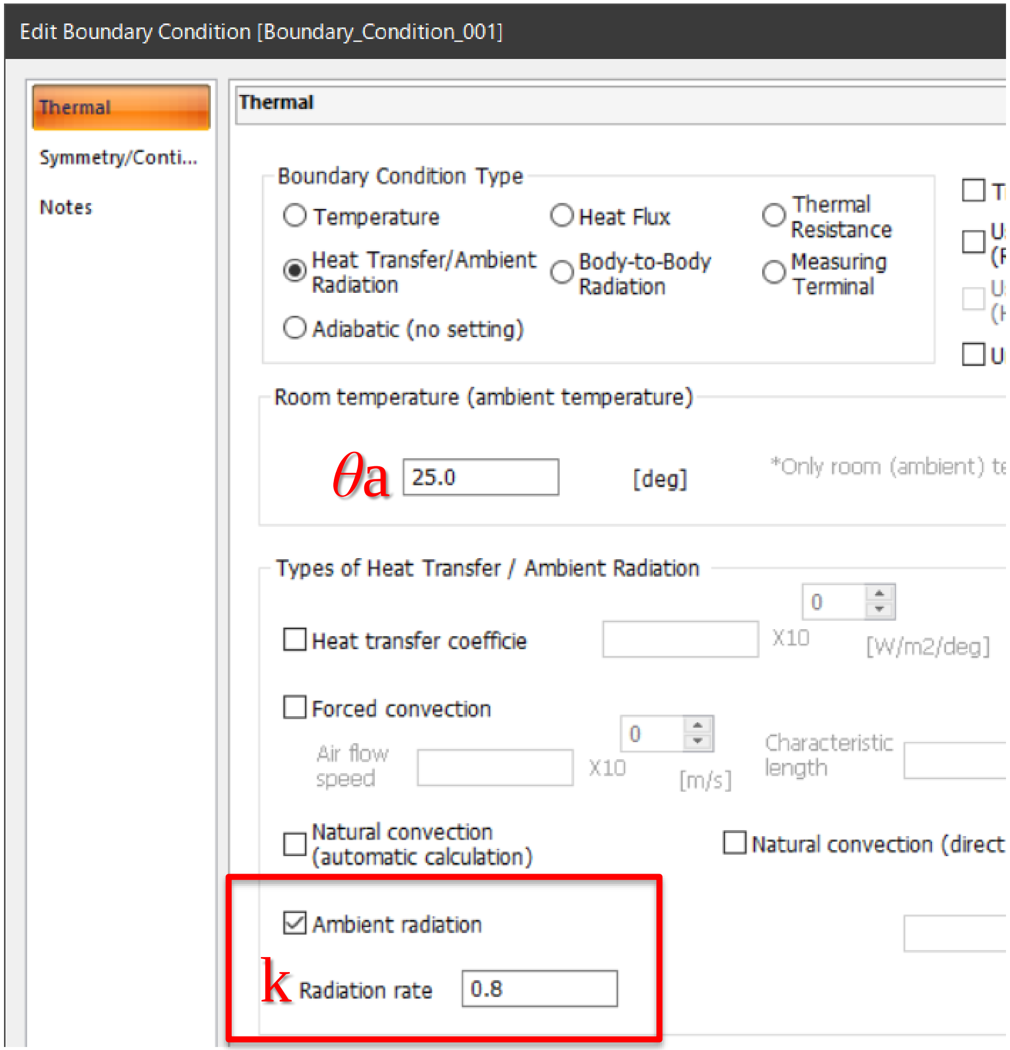
③ Ambient Radiation





Boundary Condition: Ambient Radiation

Murata Software



Heat transfer by ambient radiation

$$f = k \times \sigma (\theta^4 - \theta a^4)$$

where

k: radiation coefficient

σ: Stefan-Boltzmann coefficient

f: heat transfer (heat flux)

θ: surface temperature

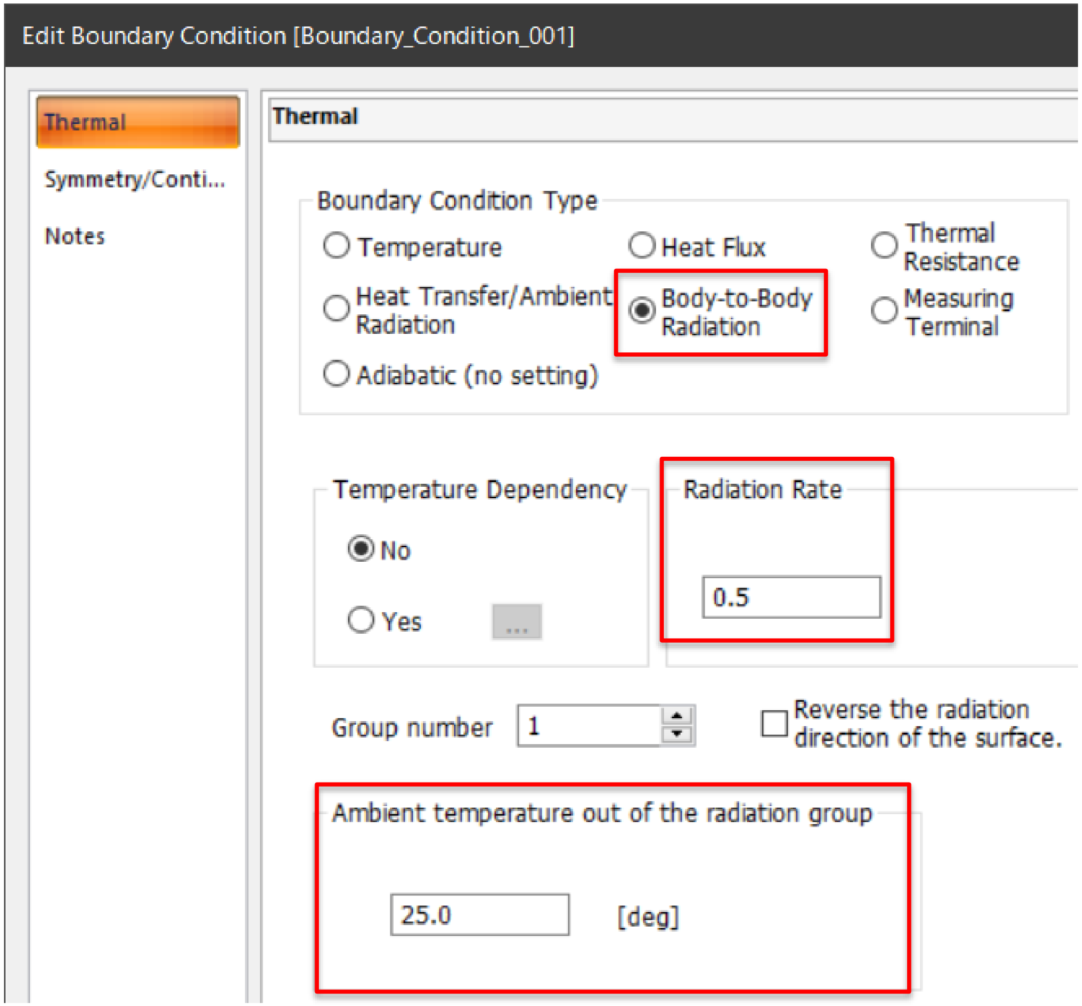
θa: ambient temperature

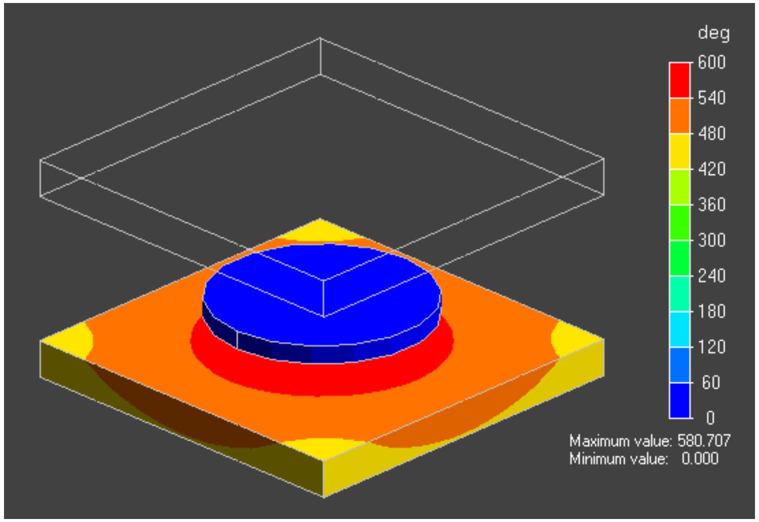
- k is the product of radiation factor and view factor.
- View factor is assumed to be 1.
- k is therefore equal to radiation rate.
- The value of radiation rate is between 0 and 1.
- Radiation rate + Light reflectance=1

Though the view factor is assumed to be 1, it is, strictly speaking, between 0 and 1. It is 1 on the plane or on the convex plane.

Murata Software

Body-to-body radiation is a radiative heat exchange.

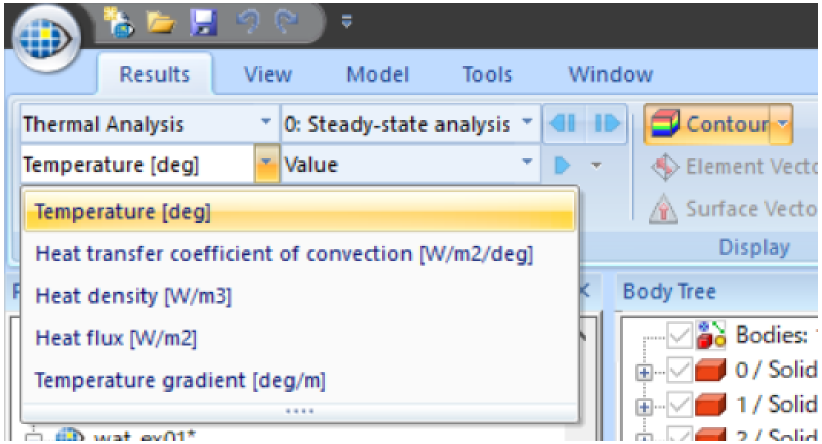


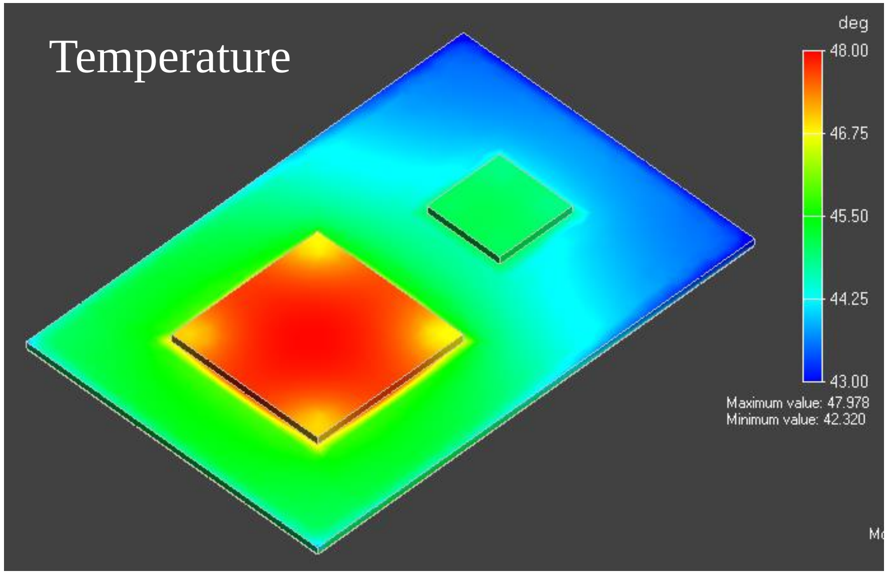


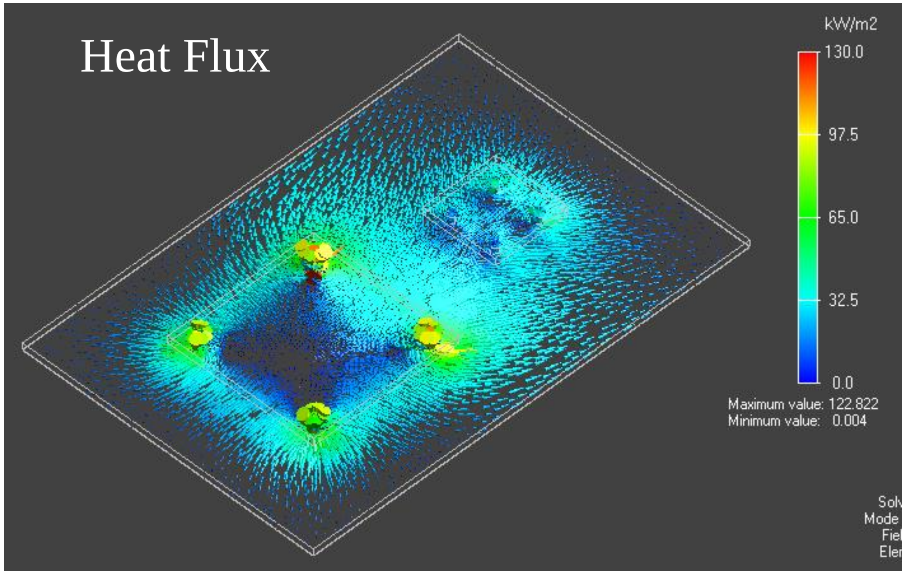
A lower plate is heated by the radiation from an upper plate. A disc in the center is low-temperature shield.
(Example 16)

Results Display

Murata Software







Results Table

Murata Software

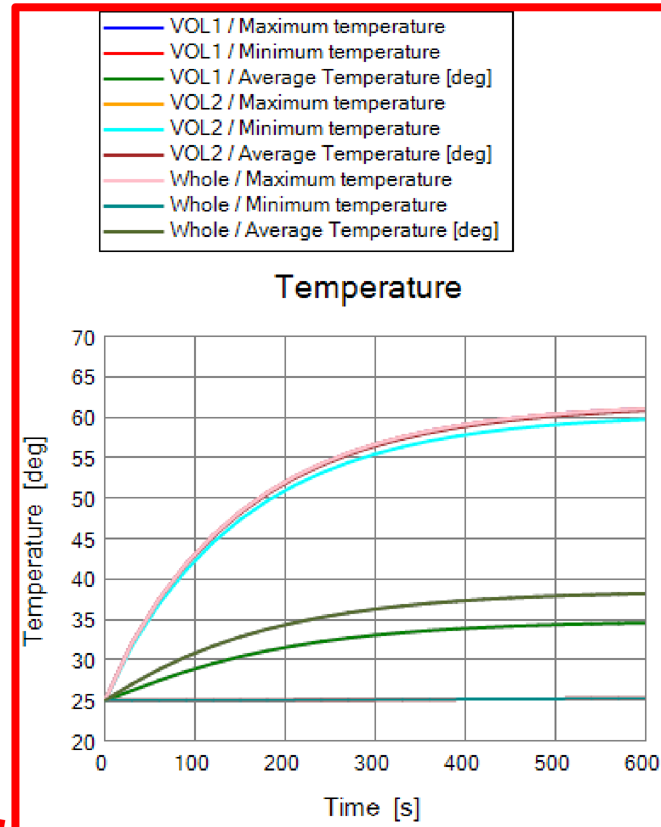
The results are output to the numerical table.
Graphing the results is possible.

Table

Convergence status | Temperature [deg] | Boundary Temperature [deg] | Heat Balance [W] | Heat Flow [W] | Thermal Resistance [deg/W]

	Time [s]	VOL1 / Maxi...	VOL1 / Mini...	VOL1 / Ave...	VOL2 / Max...	VOL2 / Min...	VOL2 / Ave...	Whole /
0: 0.000e+00[s]	0	25.000	25.000	25.000	25.000	25.000	25.000	
1: 3.000e+01[s]	30	32.109	25.000	26.212	32.122	31.738	32.059	
2: 6.000e+01[s]	60	37.527	25.001	27.454	37.540	36.940	37.438	
3: 9.000e+01[s]	90	41.877	25.005	28.578	41.889	41.135	41.759	
4: 1.200e+02[s]	120	45.418	25.010	29.562	45.429	44.560	45.279	
5: 1.500e+02[s]	150	48.317	25.019	30.411	48.328	47.368	48.161	
6: 1.800e+02[s]	180	50.696	25.032	31.135	50.706	49.675	50.527	
7: 2.100e+02[s]	210	52.652	25.047	31.749	52.661	51.572	52.472	
8: 2.400e+02[s]	240	54.261	25.064	32.268	54.270	53.135	54.073	
9: 2.700e+02[s]	270	55.587	25.082	32.704	55.595	54.422	55.392	
10: 3.000e+02[s]	300	56.679	25.101	33.070	56.687	55.483	56.478	
11: 3.300e+02[s]	330	57.579	25.119	33.376	57.587	56.358	57.374	
12: 3.600e+02[s]	360	58.322	25.137	33.631	58.330	57.080	58.113	
13: 3.900e+02[s]	390	58.934	25.153	33.844	58.942	57.675	58.722	
14: 4.200e+02[s]	420	59.440	25.168	34.022	59.447	58.166	59.225	
15: 4.500e+02[s]	450	59.857	25.182	34.169	59.864	58.572	59.640	
16: 4.800e+02[s]	480	60.201	25.194	34.292	60.208	58.906	59.982	
17: 5.100e+02[s]	510	60.485	25.205	34.393	60.492	59.183	60.265	
18: 5.400e+02[s]	540	60.720	25.215	34.478	60.727	59.411	60.498	
19: 5.700e+02[s]	570	60.913	25.223	34.548	60.920	59.599	60.691	
20: 6.000e+02[s]	600	61.073	25.230	34.605	61.080	59.755	60.850	

Thermal Analysis | Show all results summary | Display Options | Graph

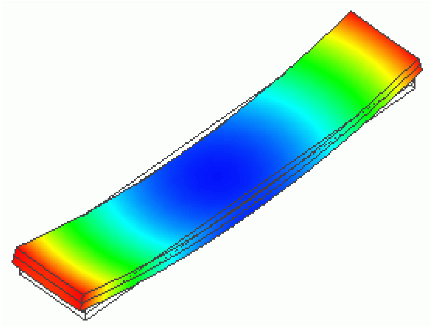


3. Thermal-Mechanical Stress Analysis

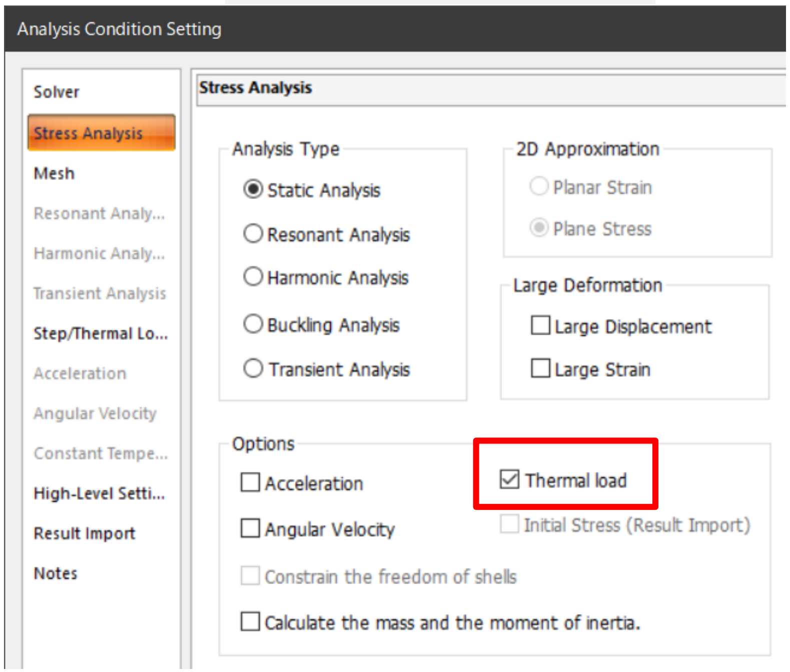
Stress-Static Analysis/Thermal Load

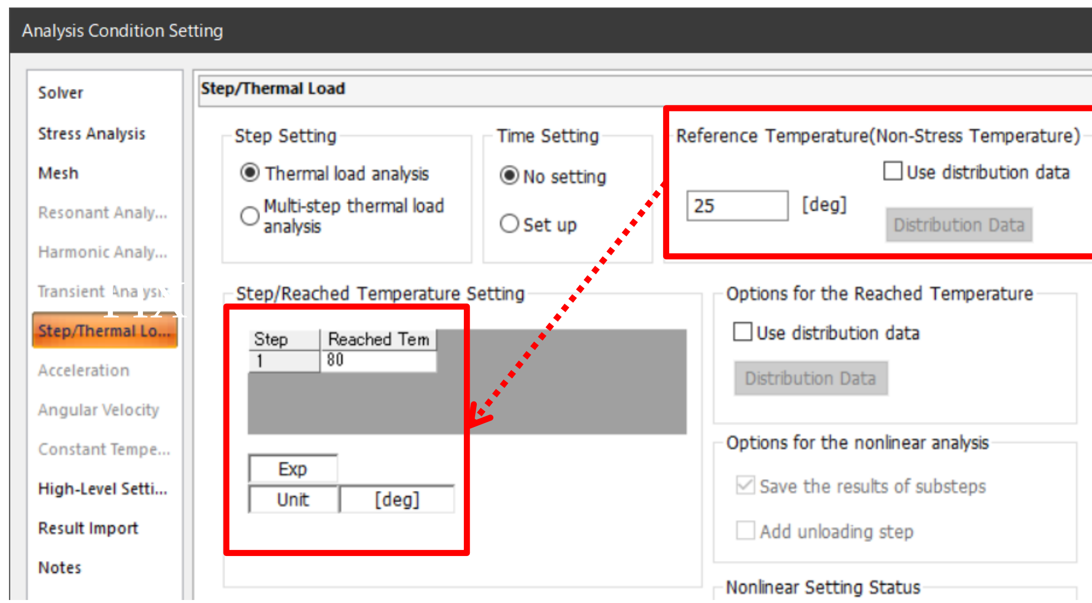
Murata Software

Stress Analysis Example 7: Bimetal under Thermal Load



If the model has no temperature distribution, thermal load can be opted in the stress analysis. Thermal analysis is not needed.



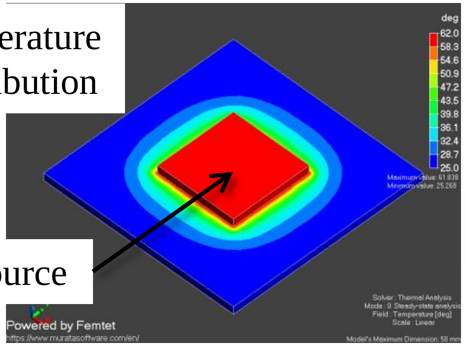


Using Temperature Distribution Obtained in Thermal Analysis

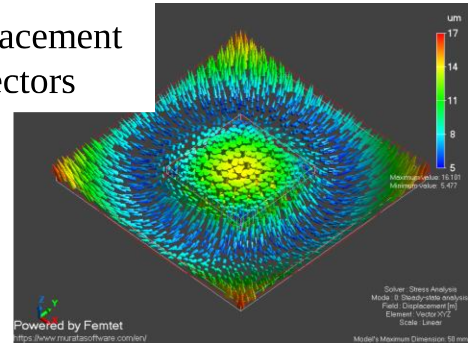
Murata Software

Temperature Distribution

Heat Source

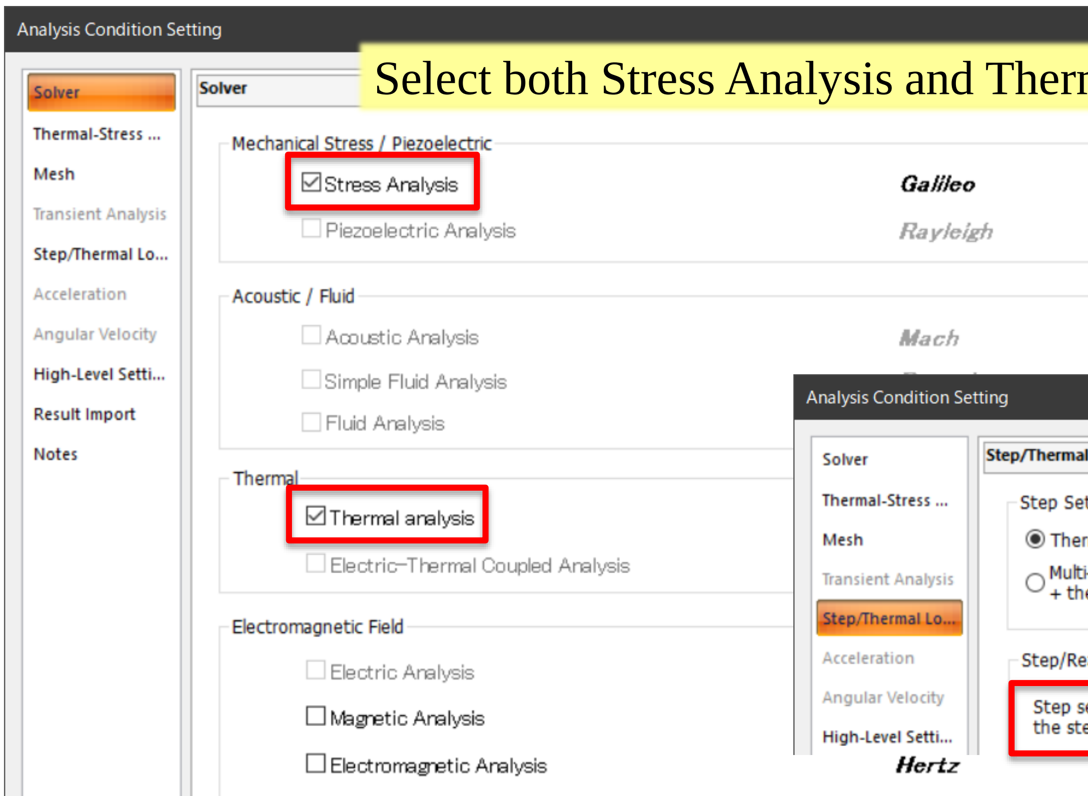


Displacement Vectors

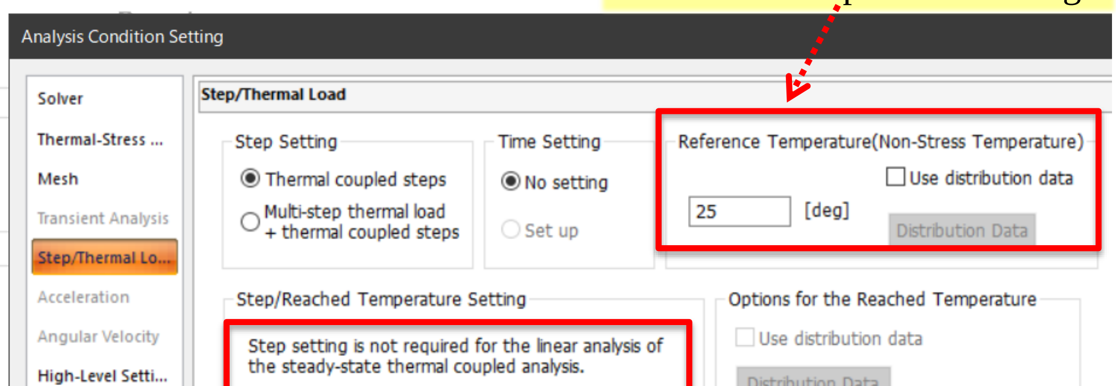


Thermal-Stress Analysis Example 2:
Deformation due to the Temperature Gradient

Select both Stress Analysis and Thermal Analysis



Reference Temperature Setting



4. Symmetric Model

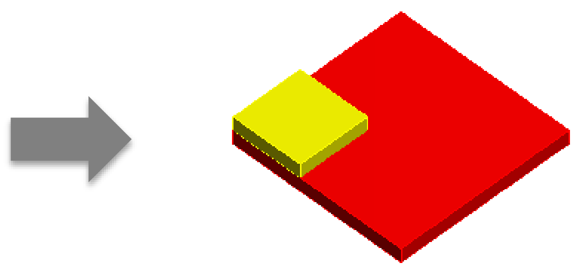
- Advantage of Symmetric Model
- Symmetric Boundary Condition (Stress Analysis)
- Symmetric Boundary Condition (Thermal Analysis)

Advantage of Symmetric Model

If an analysis model has symmetry*, a symmetric model cut out of the full model can be used.

*Symmetry in form, material property, boundary condition, and body attribute

Full Model

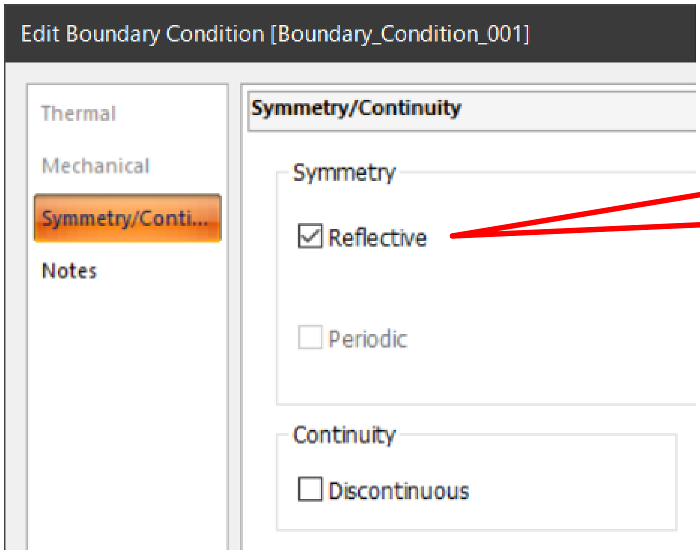


Symmetric Model

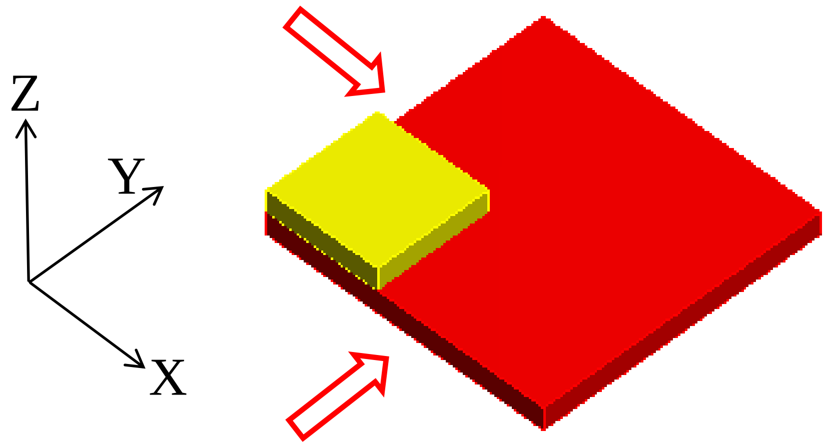
Calculation time and memory consumption are greatly reduced.

Symmetric Boundary Condition

Murata Software



If [Reflective] is selected, appropriate boundary condition is automatically set. Results display in full model is possible.

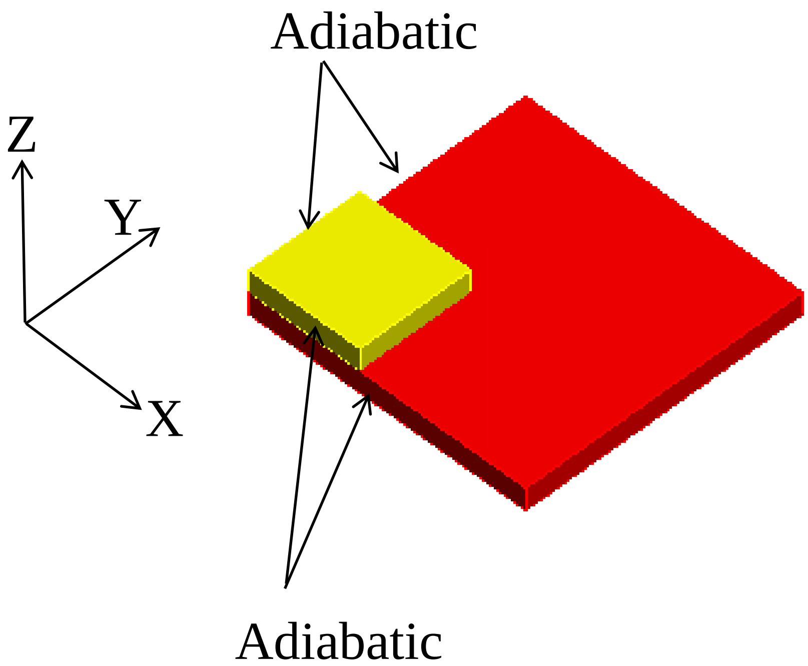


Reflective faces perpendicular to X and Y axis are given boundary conditions of different names.

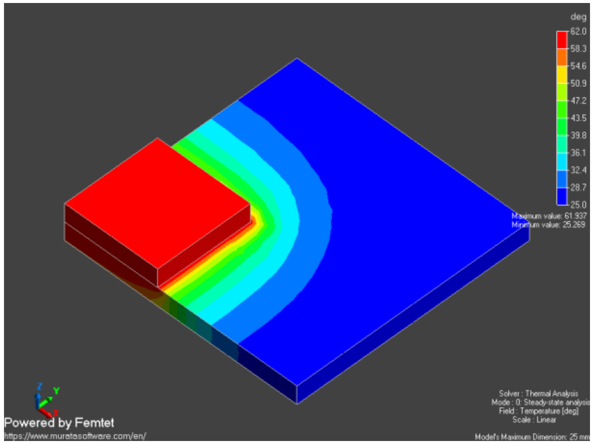
Internal Processing of Symmetric Boundary Condition

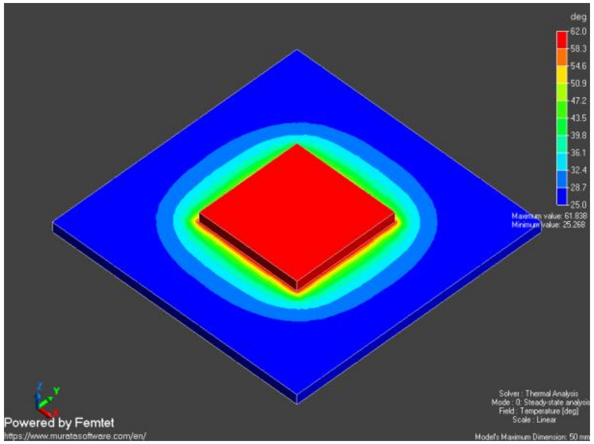
Murata Software

In the thermal analysis, adiabatic boundary is set to the reflective face.



Note
If you segment the heat source,
the heat amount must be adjusted.

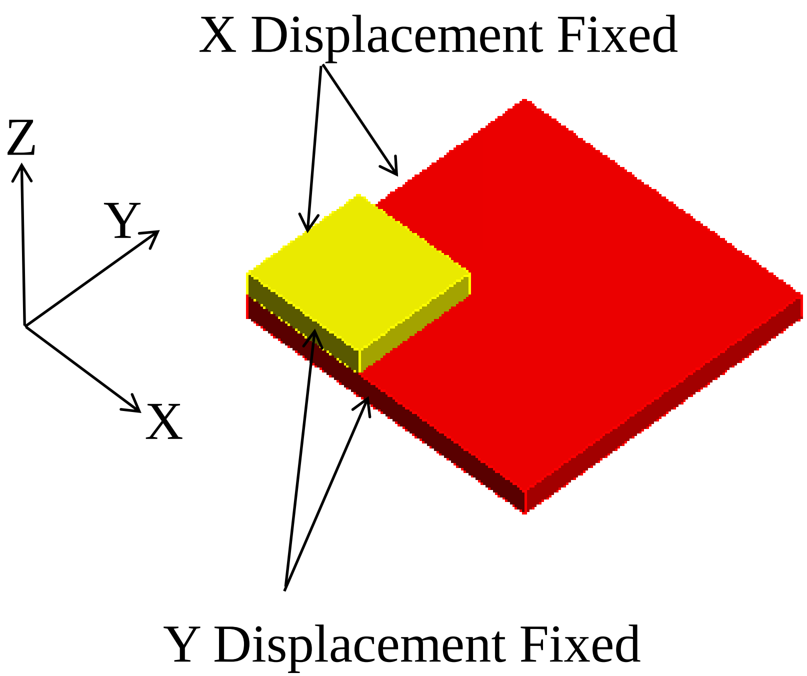


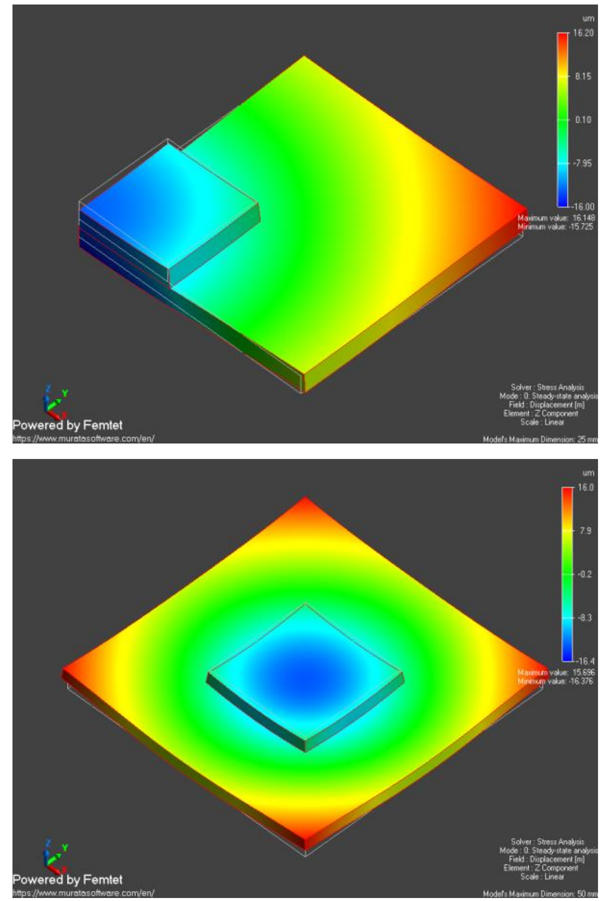


Results in Full Model

Internal Processing of Symmetric Boundary Condition

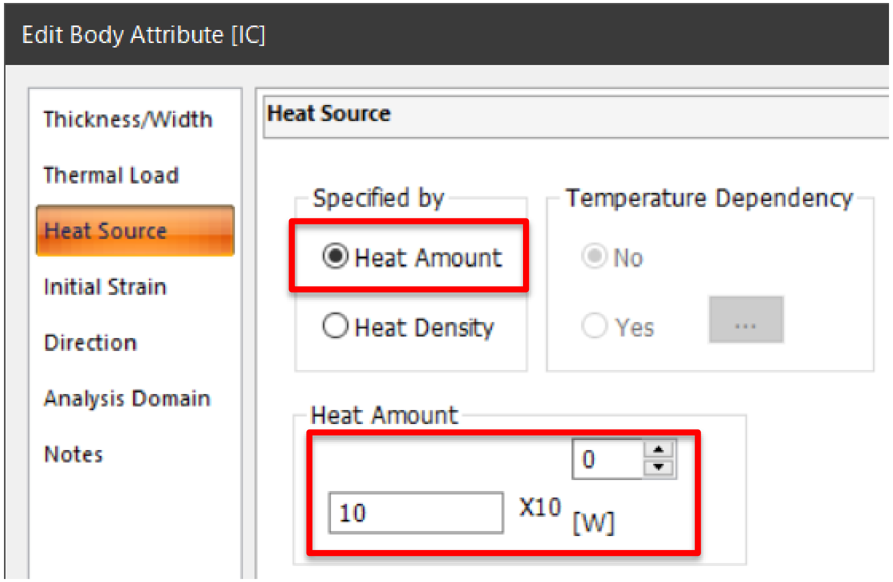
In the stress analysis, the displacement perpendicular to the reflective face is fixed.

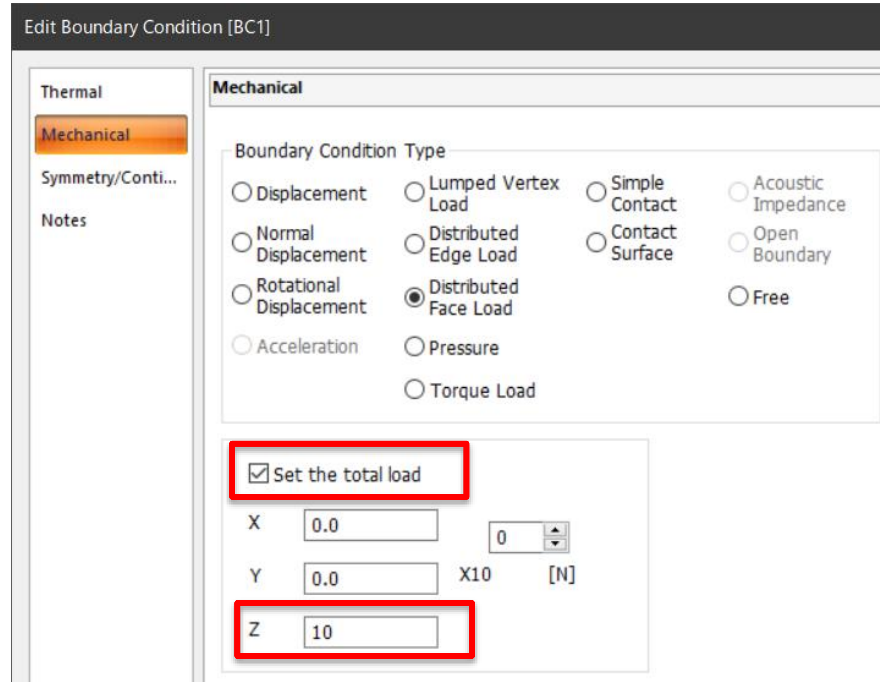




Results in Full Model

Input Values for Symmetric Model





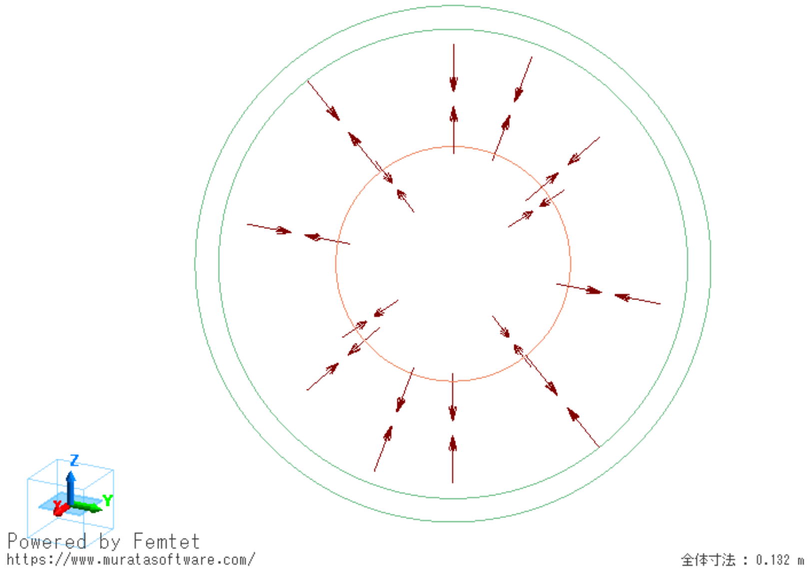
If the distributed load and the pressure are defined by the total load, their values must be adjusted depending on the type of symmetric model. The value of heat amount also needs to be adjusted.

Half model: 0.5 times the values of the full model

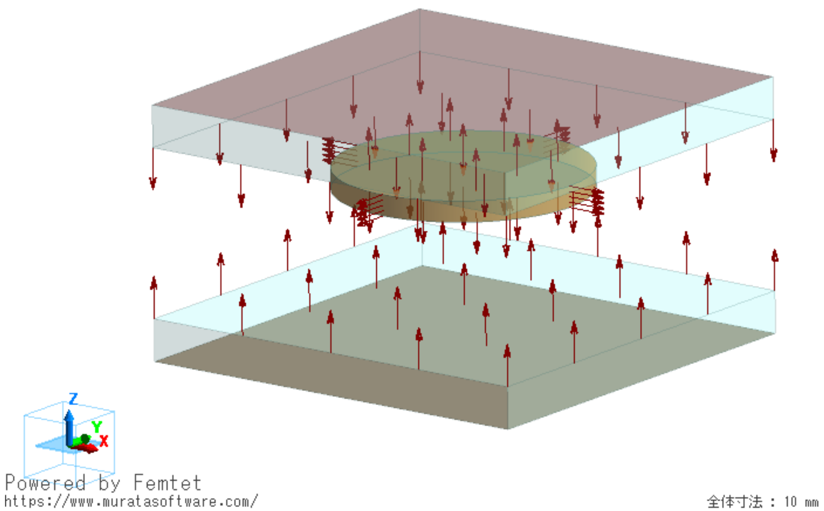
Quarter model: 0.25 times the values of the full model

*No adjustment is needed for the heat density. If the distributed load and the pressure are not defined by the total load, they don't need to be adjusted either.

Accuracy of Body-to-Body Radiation in Symmetric Model



Thermal Analysis Example 9
Body-to-Body Heat Radiation



Thermal Analysis Example 16
Radiation Blocked by Disc

Note:

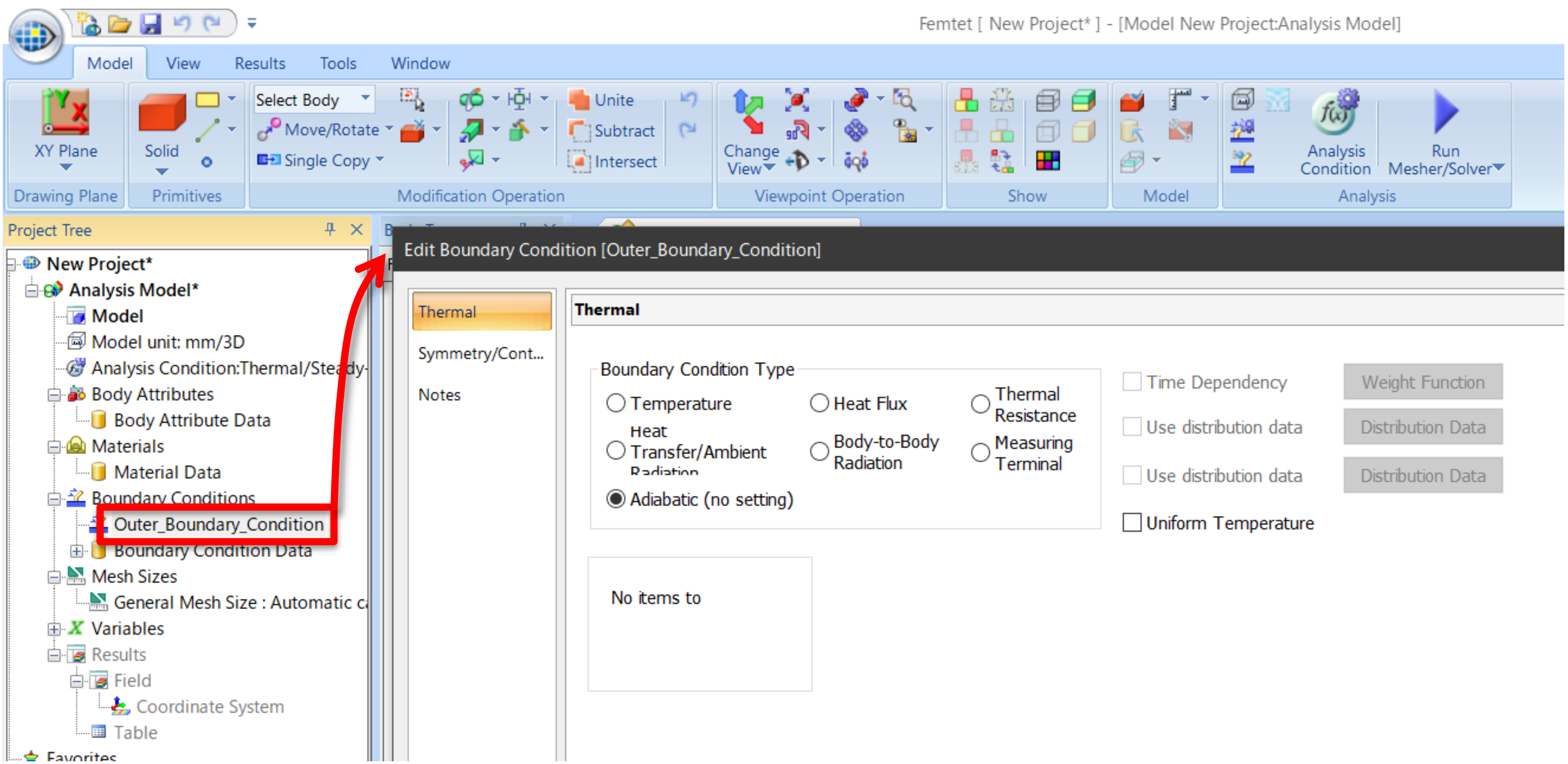
Femtet version 2019 or before

Body-to-body radiation was analyzed accurately only with full model.

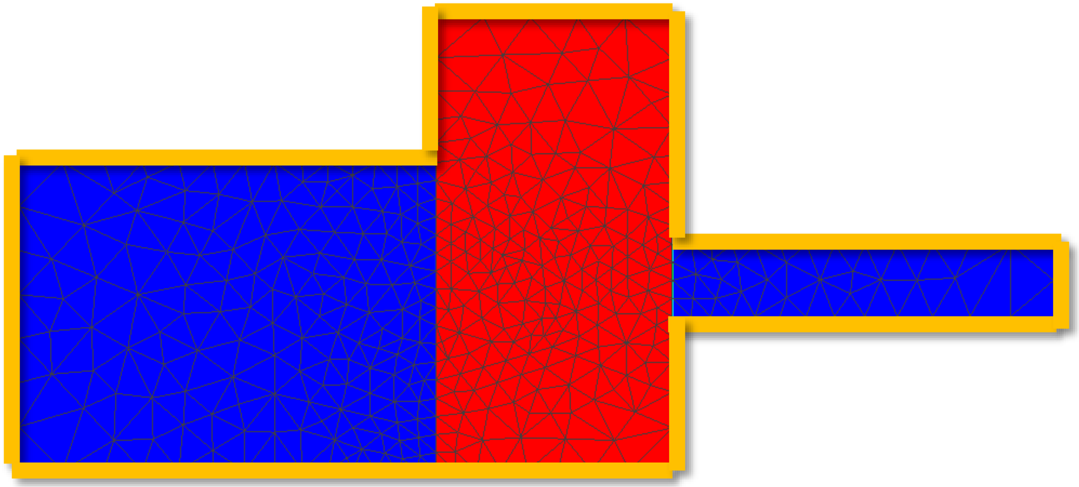
Femtet version 2020

Accurate analysis is possible even with a symmetric model since the radiation passing through the reflective face can be taken into account.

5. Outer Boundary Condition



Outer Boundary Condition by Default

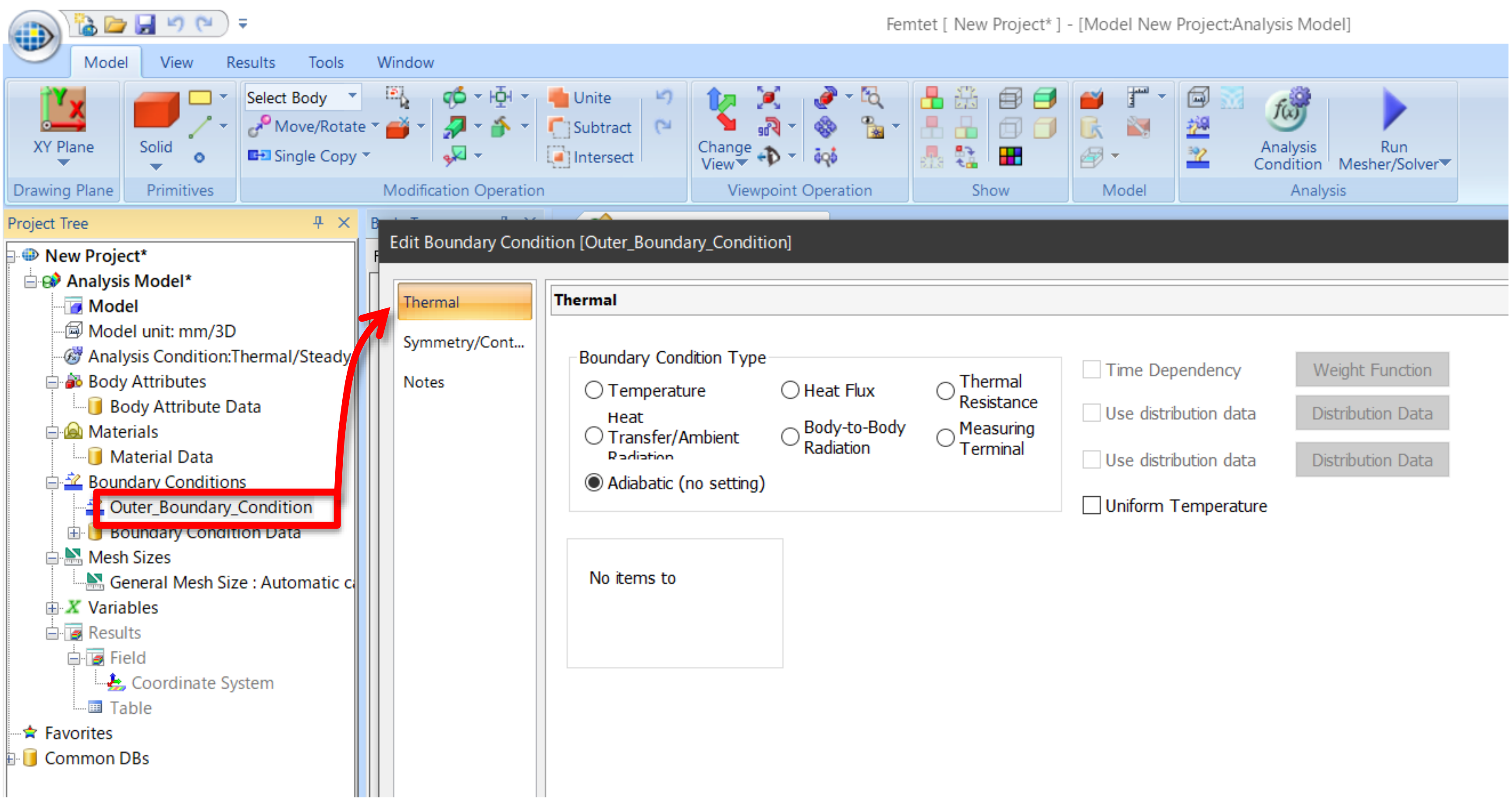


With the finite element method, the outer boundary condition must be set to all faces on the outer perimeter of a model.
The process is time taking for user.
Femtet automatically sets the outer boundary condition to the model's outer perimeter if the condition is not given by user.

Outer Boundary Condition by Default

Murata Software

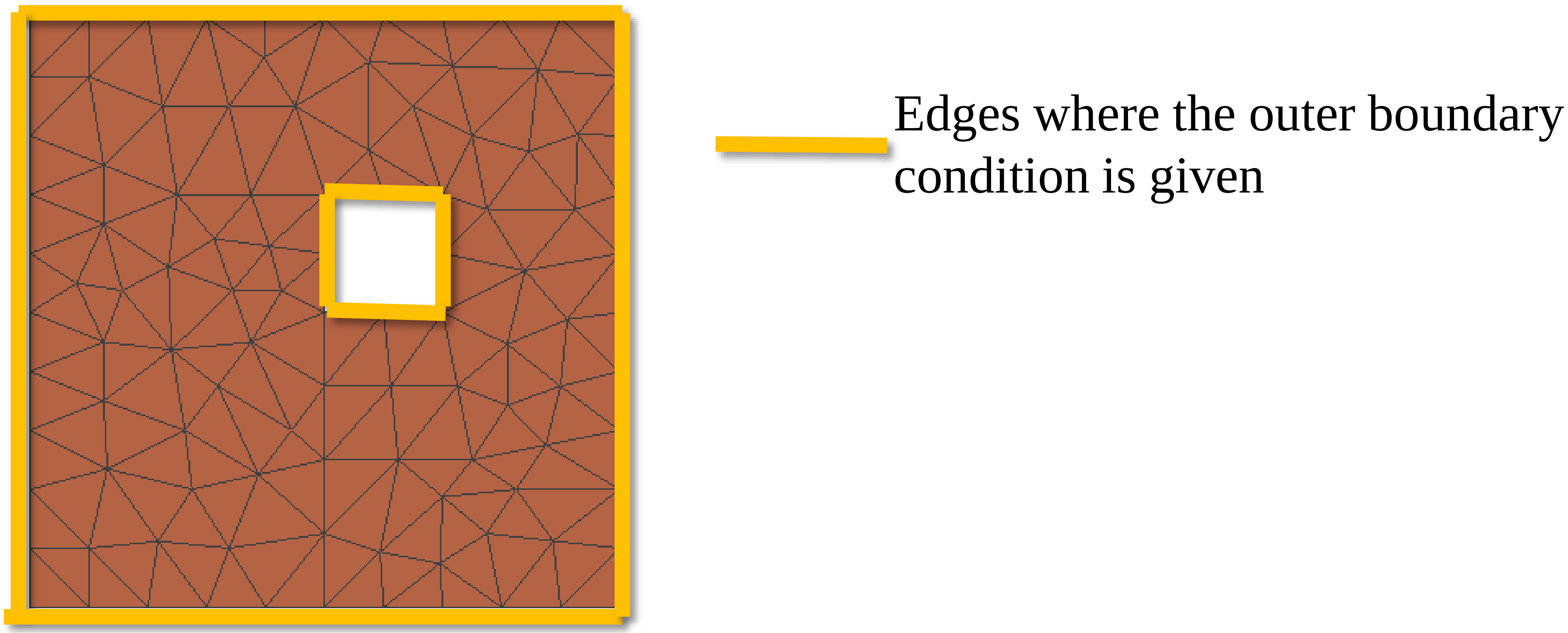
For example, radiating boundary can be set with simple setting.



Notes on Outer Boundary Condition

If boundary condition is not set, outer boundary condition is set to the edges of the hole even if it is inside the body.

For the faces and edges which are not in contact with other bodies, if boundary condition is not set to them, outer boundary condition will be set.



6. Linear/Nonlinear Analysis

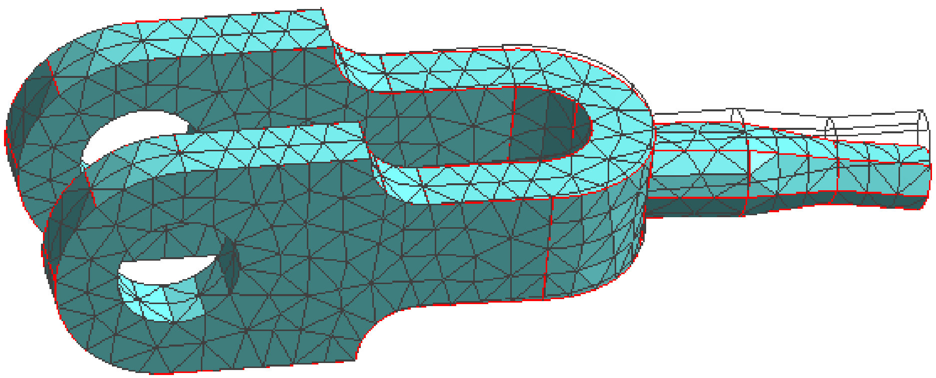
- Linear and Nonlinear Analysis
- Nonlinear Analysis in Stress Analysis
- Nonlinear Analysis in Thermal Analysis

Linear and Nonlinear Analysis

With the finite element method, the simultaneous equation (matrix equation) is solved for the unknown x, which is arranged for each nodal point.

Linear analysis solves the matrix equation by single calculation.
Nonlinear analysis solves the matrix equation by repeating the calculation.

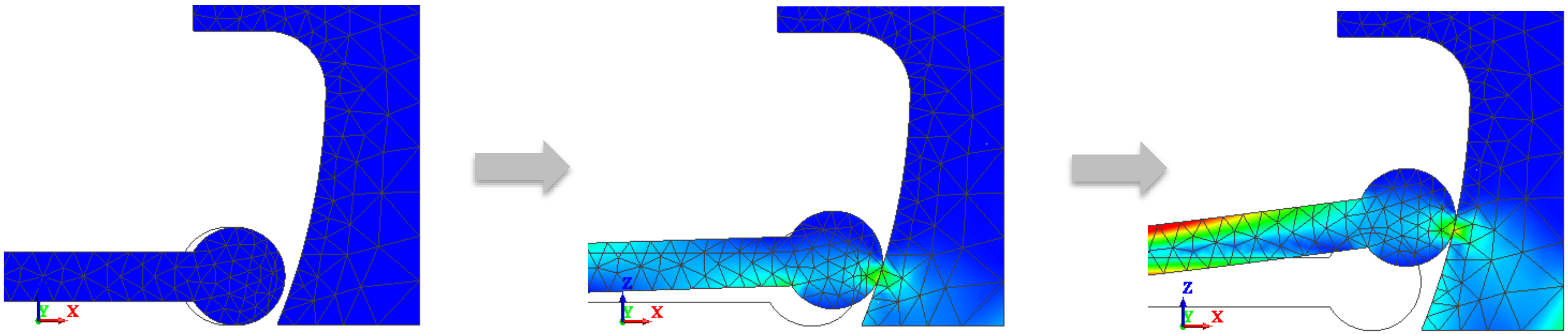
Nonlinear analysis takes long calculation time, and may not converge in some cases.



$$\begin{bmatrix} a_{11} & a_{12} & a_{13} & a_{14} \\ a_{21} & a_{22} & a_{23} & a_{24} \\ a_{31} & a_{32} & a_{33} & a_{34} \\ a_{41} & a_{42} & a_{43} & a_{44} \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \\ x_3 \\ x_4 \end{bmatrix} = \begin{bmatrix} b_1 \\ b_2 \\ b_3 \\ b_4 \end{bmatrix}$$

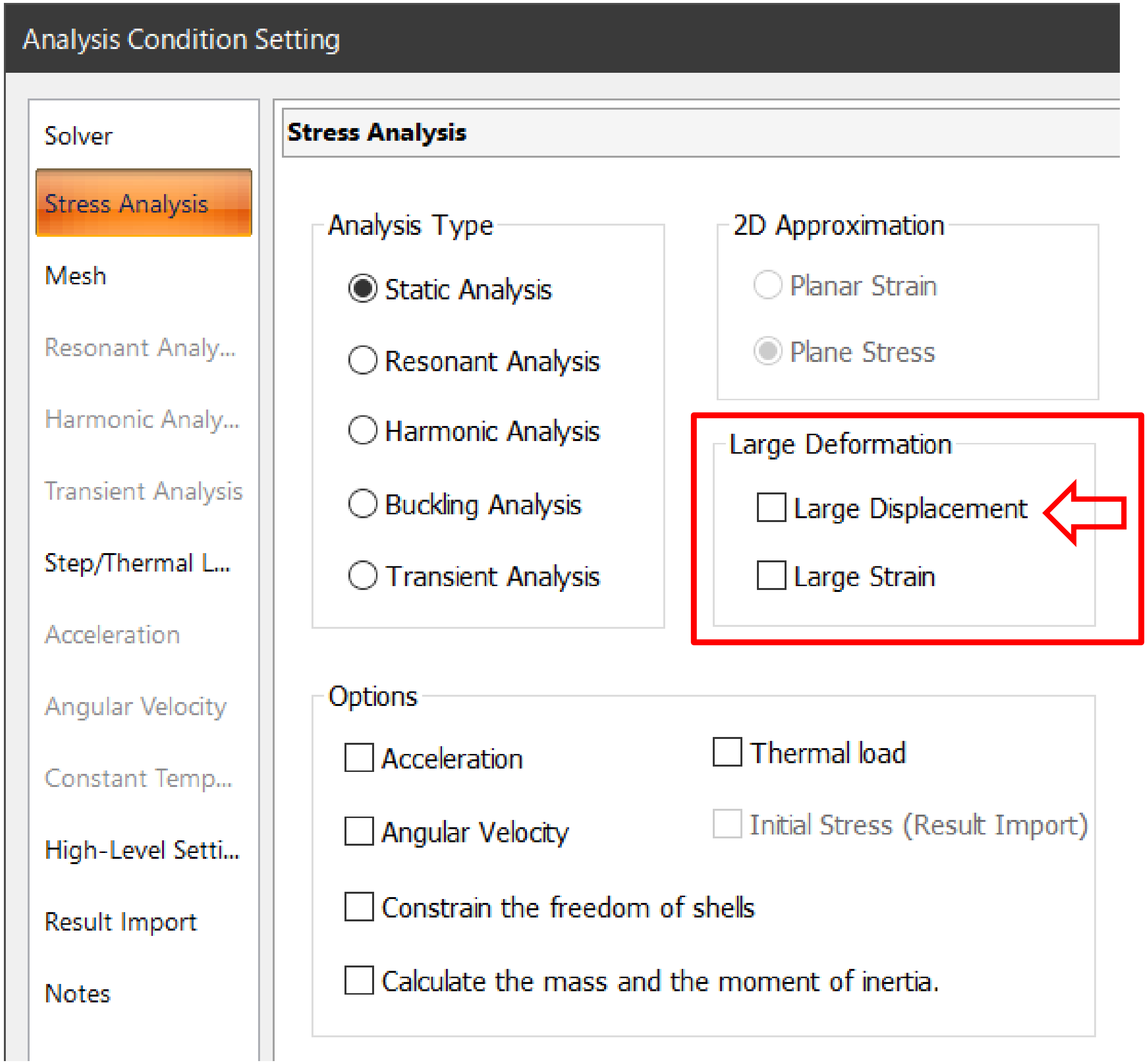
Nonlinear Analysis: Contact

A form changes gradually.
Calculations are repeated by changing the form bit by bit.



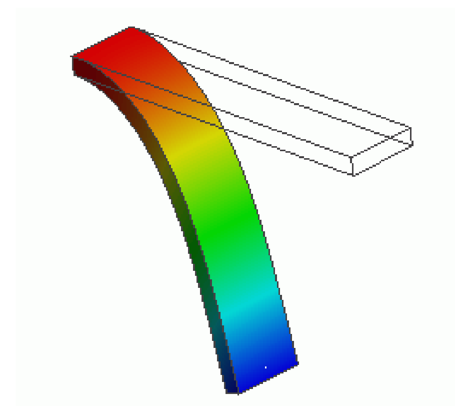
Two Types of Geometric Nonlinearity

Murata Software

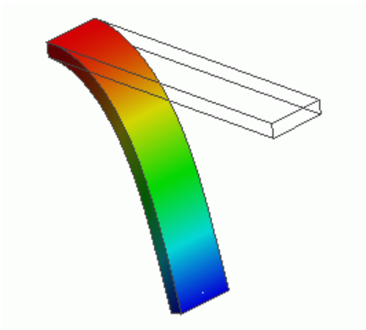


Large Displacement

The impact (displacement) on the tip of a bar generated by the minute strain on the other end of the bar is analyzed.



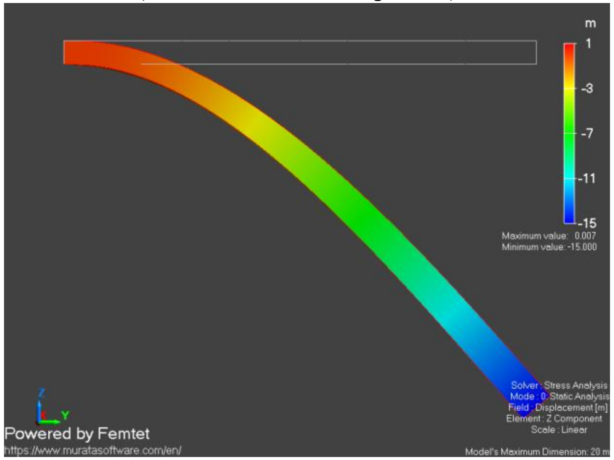
Nonlinear Analysis: Large Displacement



Large Displacement

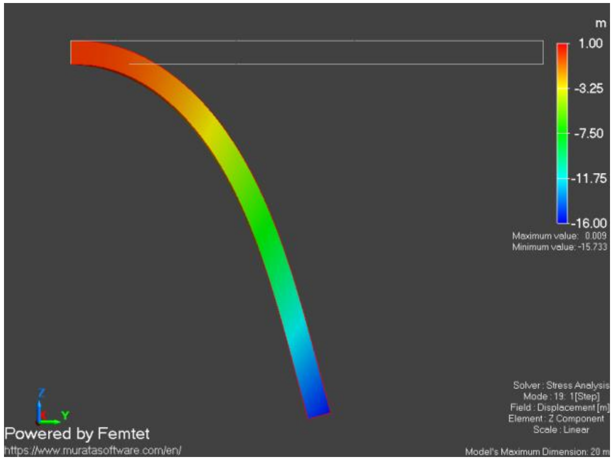
In the analysis, curling is taken into account, which is not calculated in the minute-displacement analysis. (Example 6)

Large Displacement Set OFF
(Linear Analysis)



The results are only magnified version of the minute-displacement analysis results.

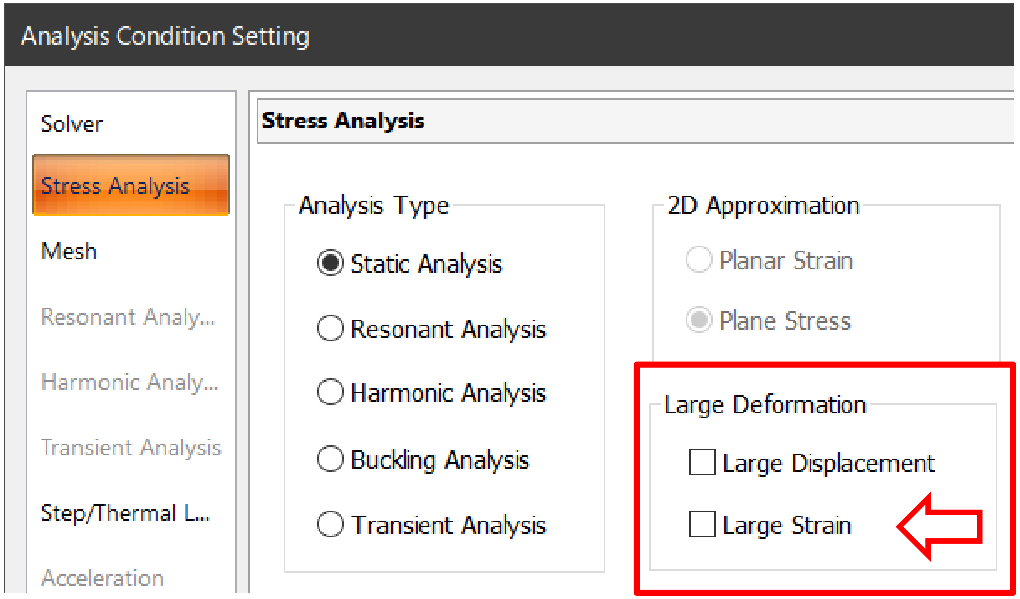
Large Displacement Set ON
(Nonlinear Analysis)



Calculations are repeated by changing the analysis model form bit by bit.

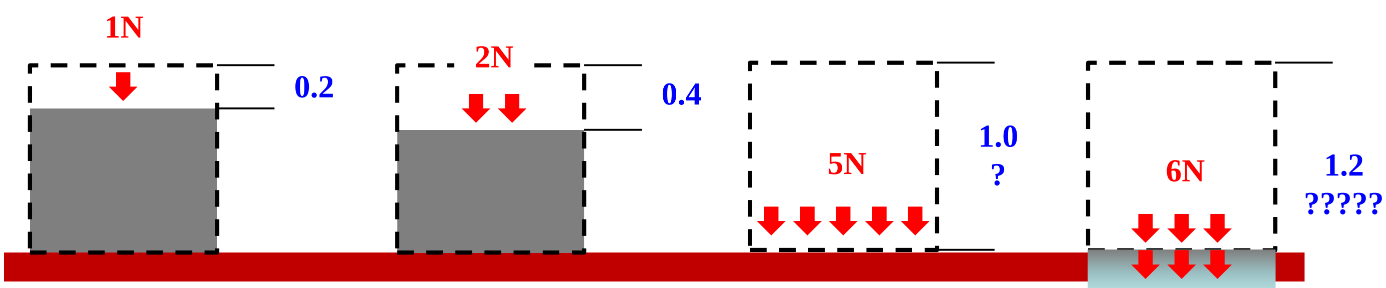
Two Types of Geometric Nonlinearity

Murata Software



Select [Large Strain] if the strain is larger than 0.1.

In the linear analysis, if the load and deformation are large, the unrealistic results will be given.

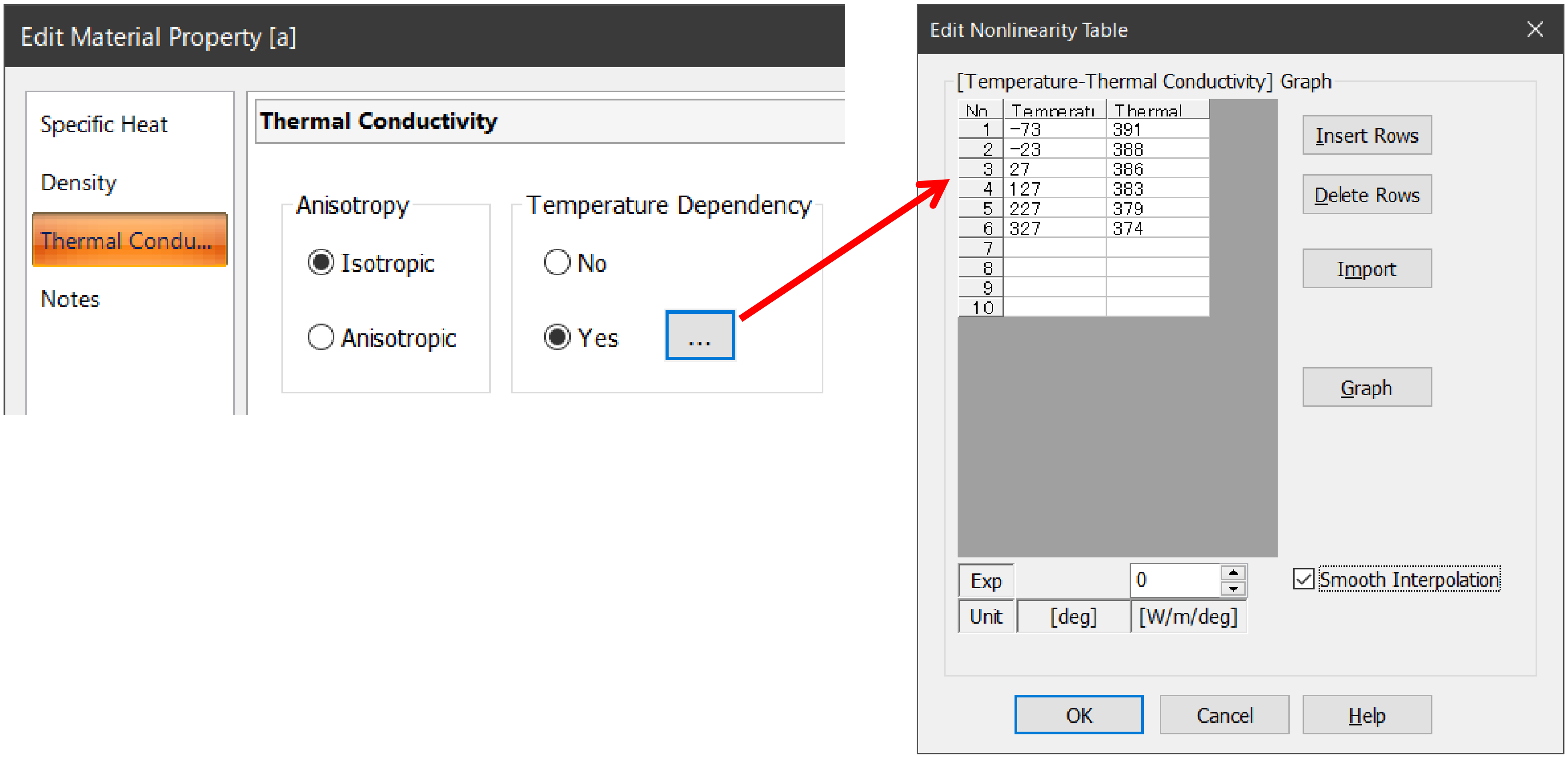


Calculations are repeated by "recreating an analysis model bit by bit with strain"
The equation is modified as well.

Nonlinear Thermal Analysis

Murata Software

If temperature dependency is selected for the thermal conductivity in the material property setting, nonlinear analysis is performed.
The material property is repeatedly changed by making a small change in temperature at every step.



Other Nonlinear Analyses

Stress analysis for the materials below:

- Elastoplastic material
- Creep material
- Viscoelastic material
- Hyperelastic material

Thermal analysis with boundary conditions below:

- Natural convection
- Ambient radiation
- Body-to-body radiation

Note:
As nonlinear analysis takes long time for calculation, it is recommended that you carefully select the nonlinear elements for analysis.

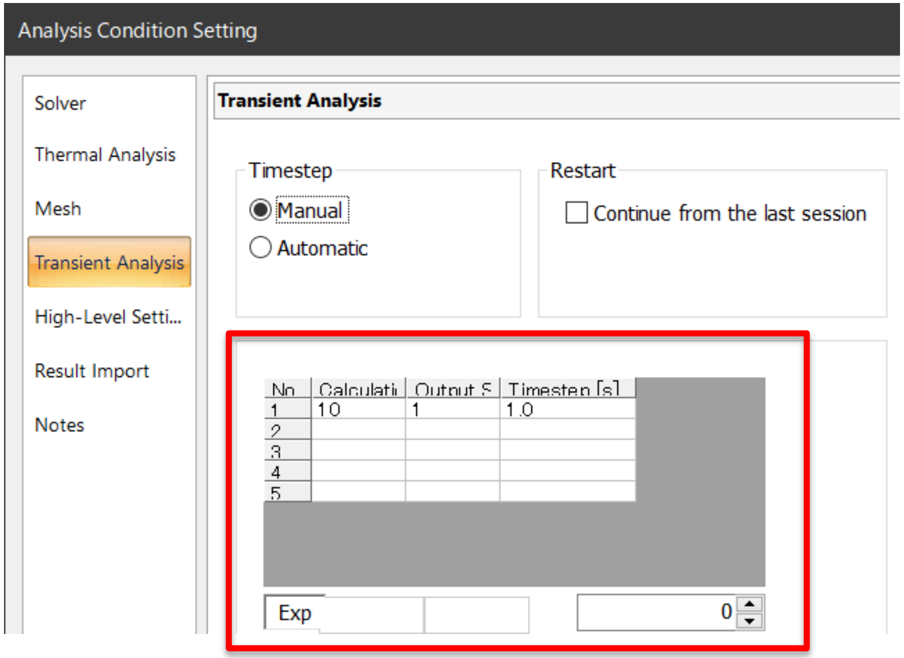
7. Transient Analysis

*Option for Femtet Advanced Mechanical is required to perform transient analysis in the stress analysis.

Transient Analysis Tab

Murata Software

The transient analysis tab is set to perform transient analysis in the thermal or stress analyses.



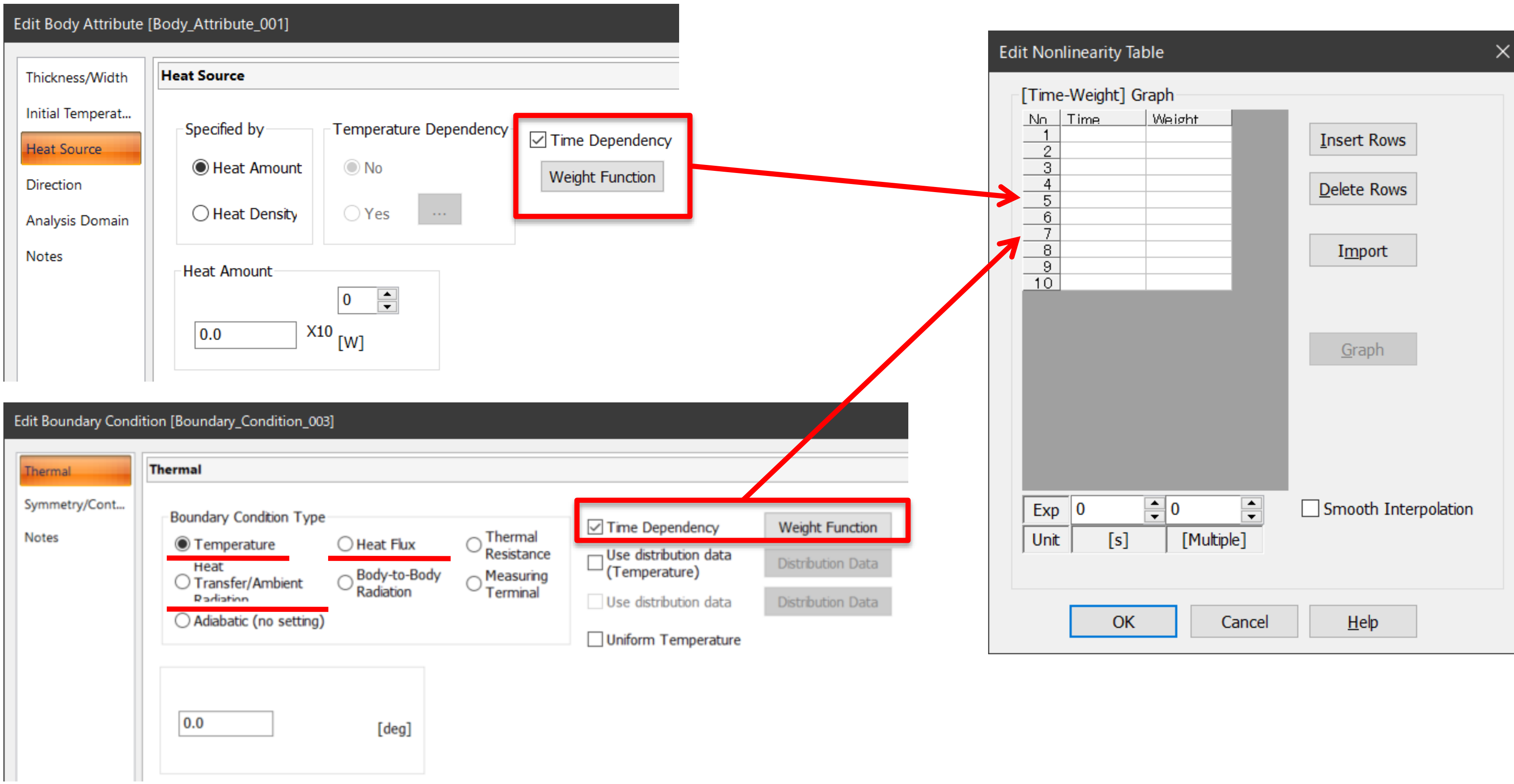
The transient analysis is performed over the small timesteps.
Setting items are:

- Timestep
- Calculation steps
- Output steps
- Initial temperature (for the thermal analysis)

Heat Source Changing over Time

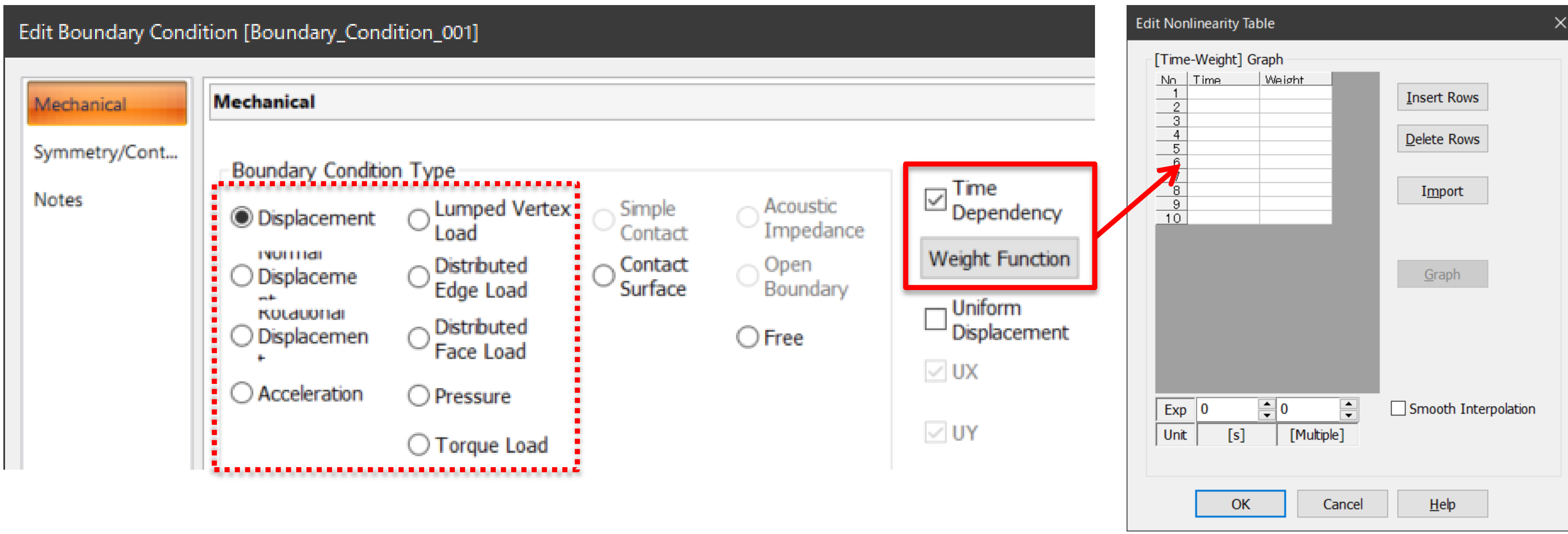
Murata Software

Time dependency can be given to the heat source (body attribute) and the boundary conditions (temperature, ambient temperature for heat transfer/ambient radiation, heat flux).

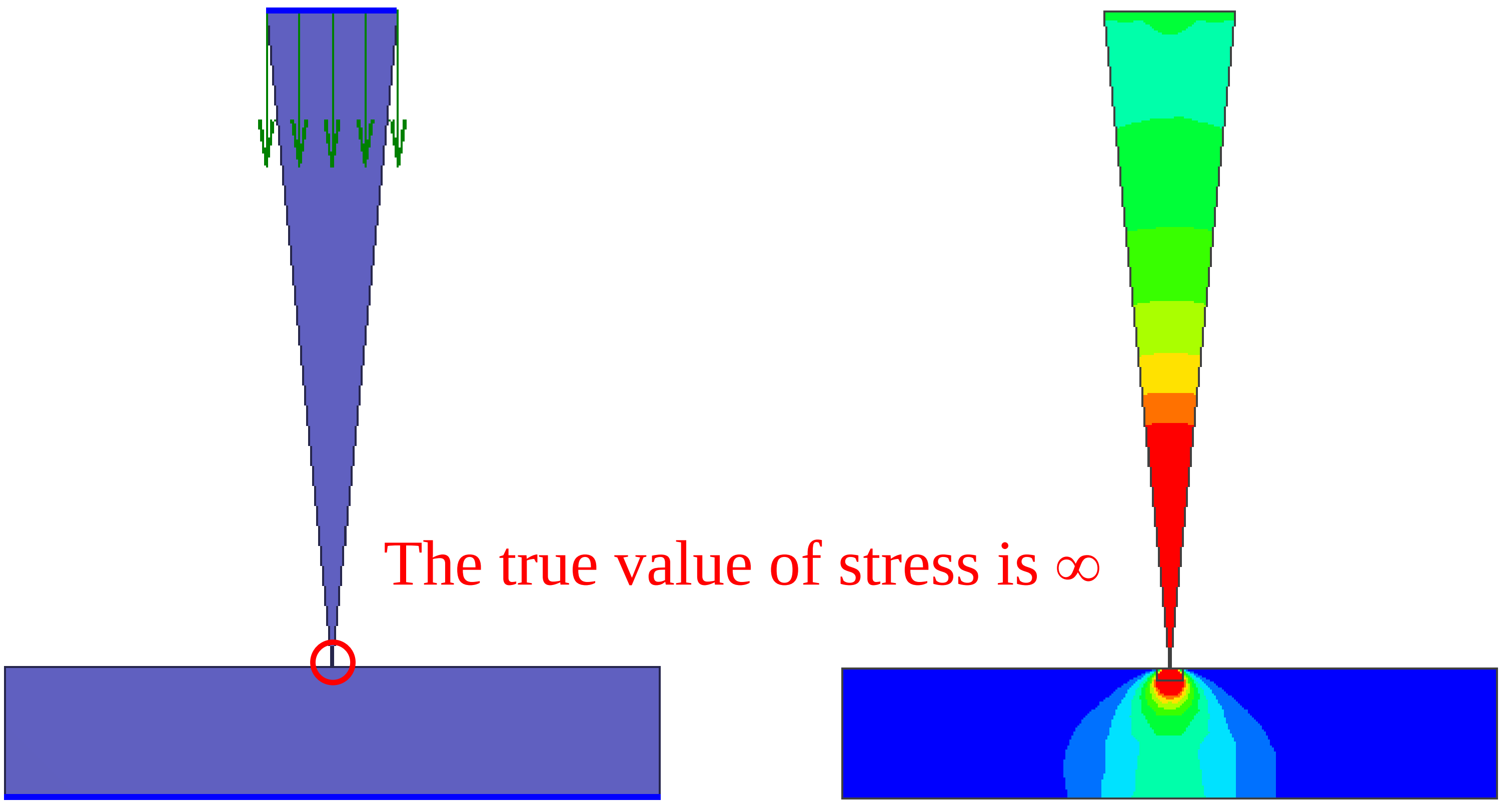


Boundary Condition Changing over Time

In the transient stress analysis, time dependency can be given to the mechanical boundary conditions of displacement, load, and pressure.



8. Singularity

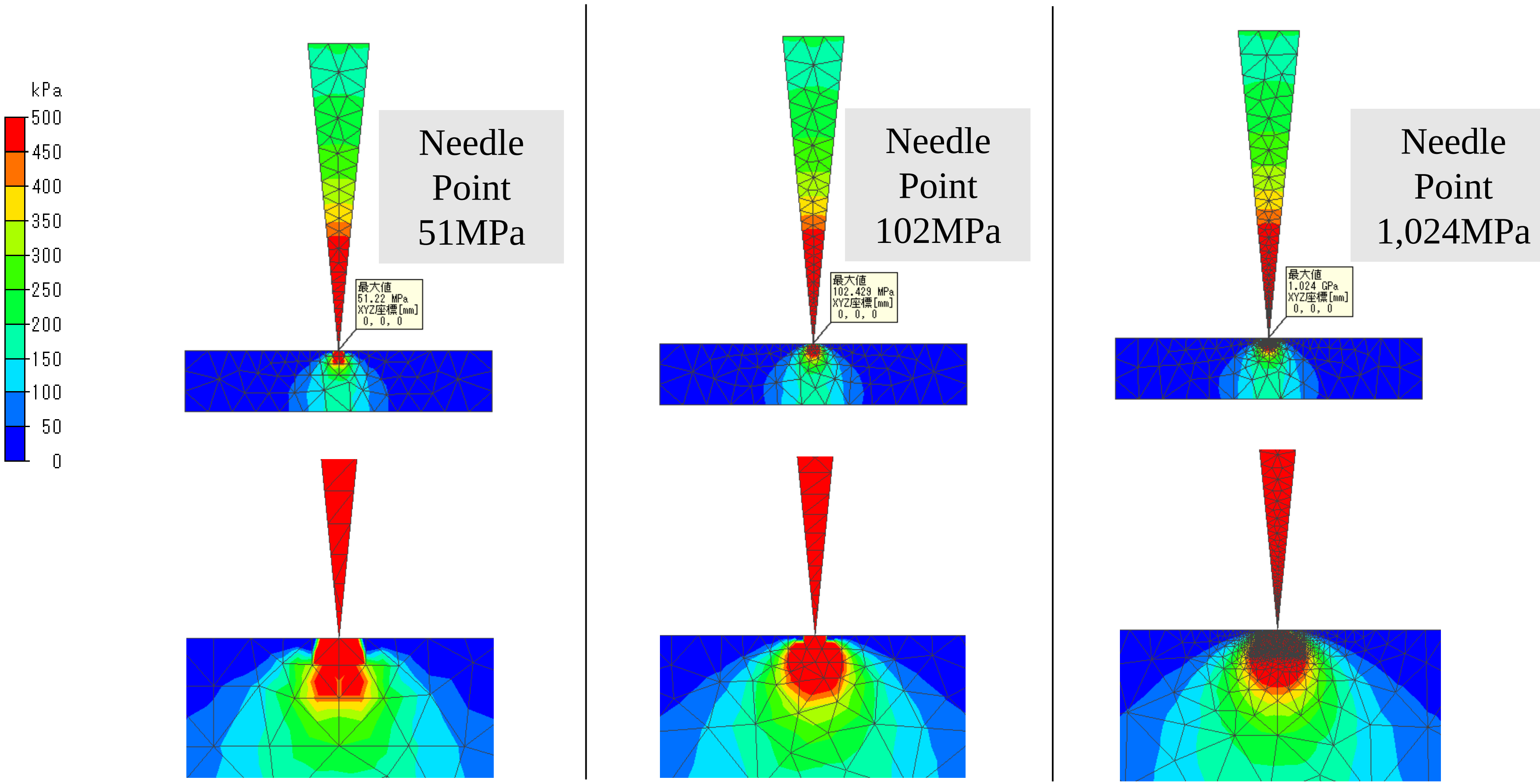


Singularity

Murata Software

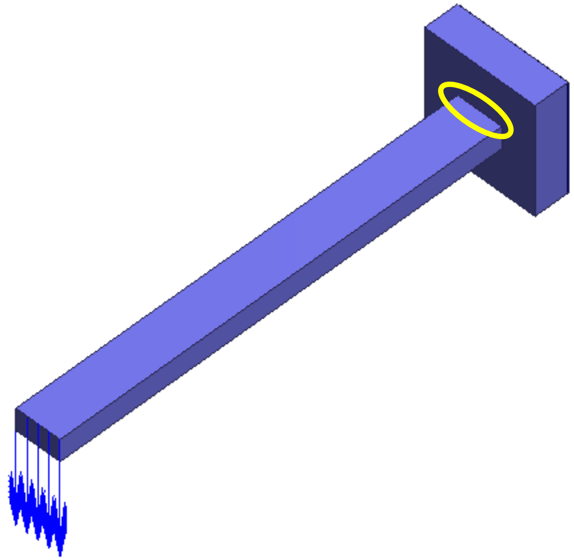
Stress (Hydrostatic Pressure)

Stress singularity does not converge even if the meshes are fine.
It approaches to the true value of ∞.
The results more than two meshes away from the singularity are reliable.



Singularity

In this model, the singularity is at the base of the cantilever.
If the base is rounded, the singularity will disappear.



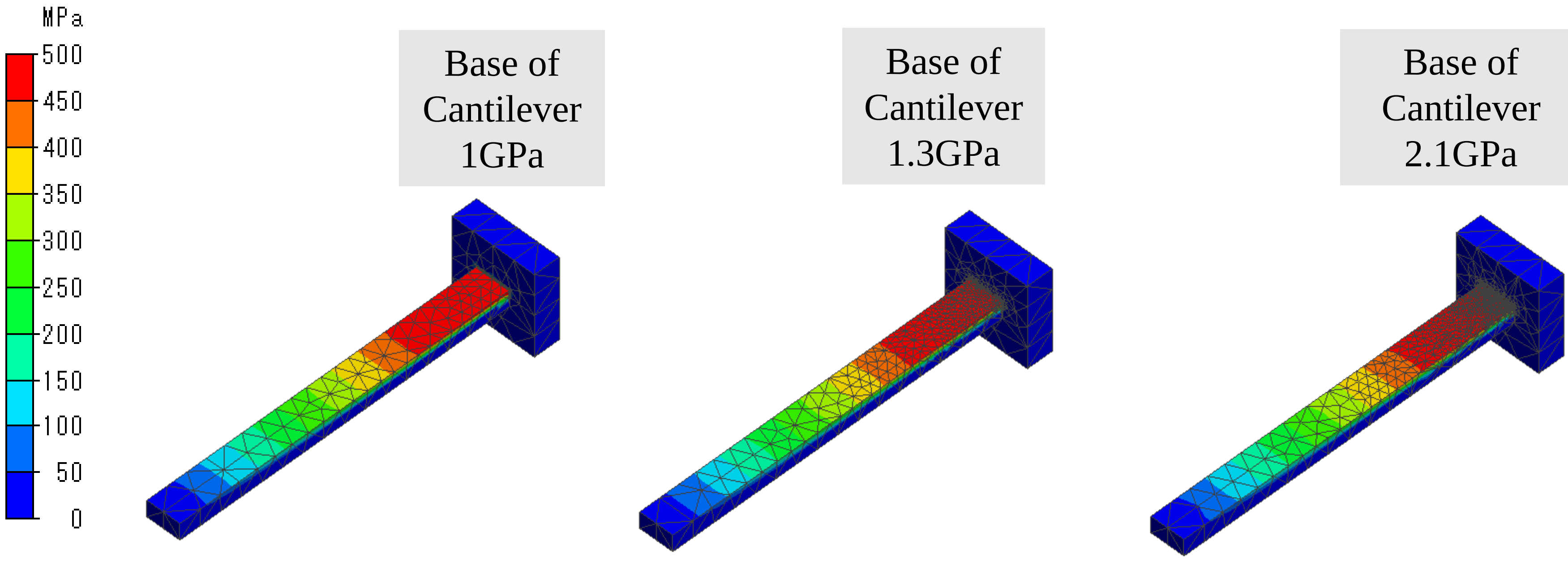
Singularity is not uncommon.
It appears if realistic roundness is not set where the stress is concentrated.
If the stress values do not converge after adopting finer meshes, it may be due to the singularity.

Singularity

Murata Software

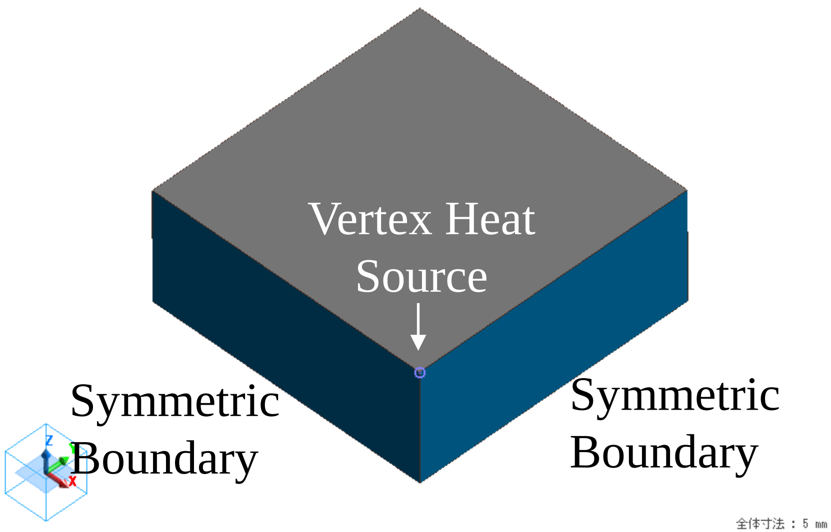
Stress (Maximum Principal Stress)

The singularity stress does not converge even if the meshes are fine.
It approaches to the true value of ∞.
The results more than two meshes away from the singularity are reliable.



Singularity in Thermal Conduction

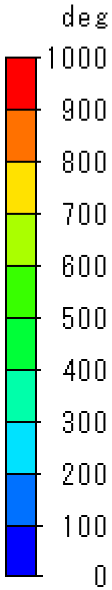
If the heat source is set on the vertex body, it becomes a singularity in the temperature field. That is like a point where the sunlight is concentrated through a magnifying glass.



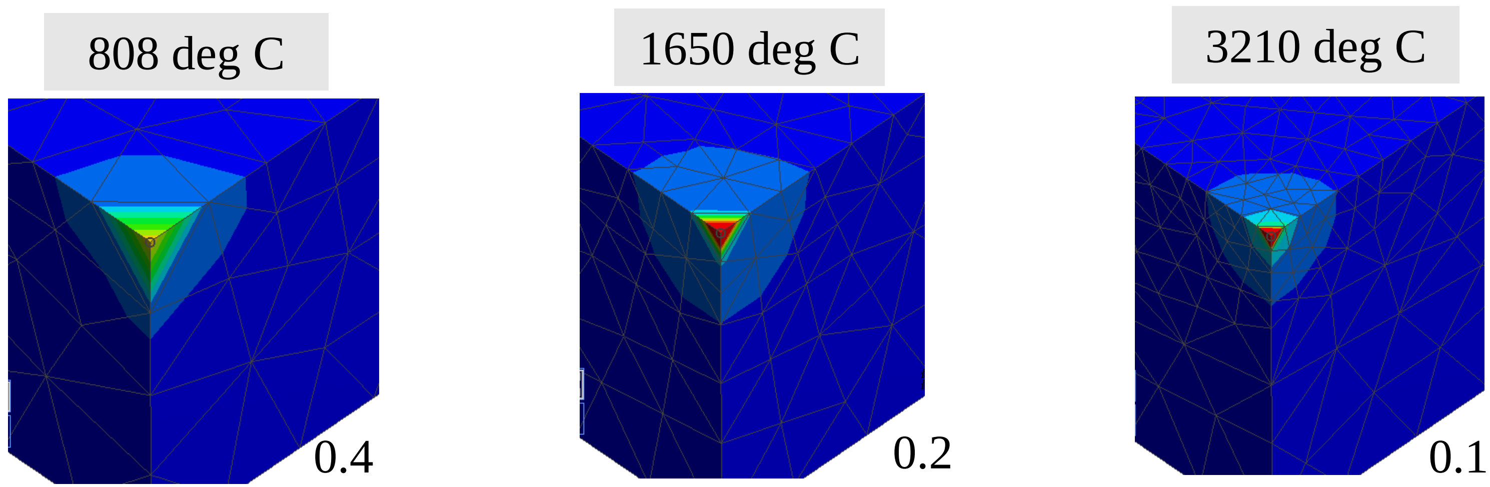
Symmetric Model

Singularity in Thermal Conduction

Murata Software

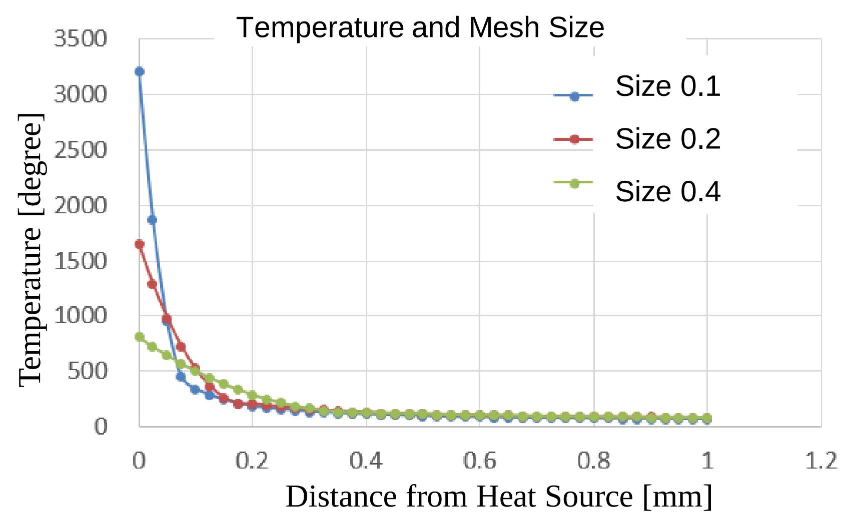


Maximum Temperature



The singularity temperature does not converge even if the meshes are fine. It approaches to the true value of ∞.

The results more than two meshes away from the singularity are reliable.



How to deal with the singularity.

(1) Remove data within two meshes from the singularity for analysis review.

(2) Apply a realistic roundness to the model to delete the singularity.

(3) In a comparison between the levels of the analysis where the structural parameters such as model dimensions are changed, align the mesh size between the levels. Though the analysis values near the singularity are not reliable as the absolute values, it is assumed that the level-to-level relationships are maintained if the mesh size is same.

If you want to evaluate the progressive crack or fracture.

Use the energy release rate.

*Refer to the [J integral] in the Help for the details.