
Femtet Seminar

Understanding

Stress Analysis

&

Thermal Analysis

202009

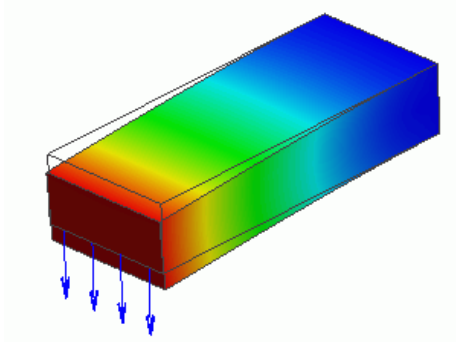
Basic

1. Stress Analysis Overview	3
2. Thermal Analysis Overview	36

Advanced

3. Fluid-Thermal Analysis	56
4. Symmetric Model	59
5. Outer Boundary Condition	66
6. Linear/Nonlinear Analysis	70
7. Transient Analysis	78
8. Singularity	82

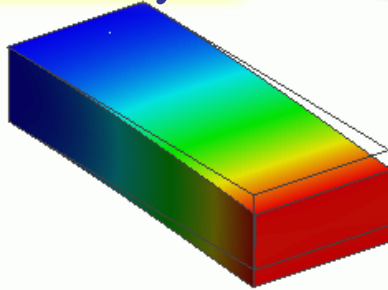
1. Stress Analysis [Galileo] Overview



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

Analysis Type

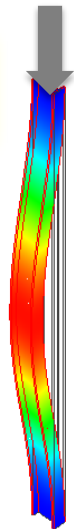
Static Analysis



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

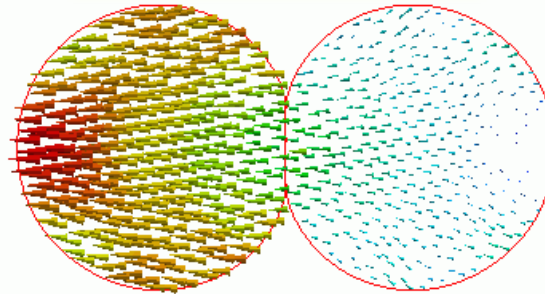
Cantilever under Distributed Load
(Example 1)

Buckling Analysis



H-beam Steel (Example 48)

Transient Analysis

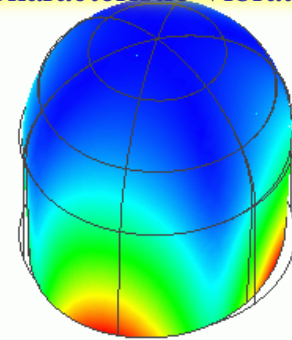


Colliding Balls (Example 32)

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

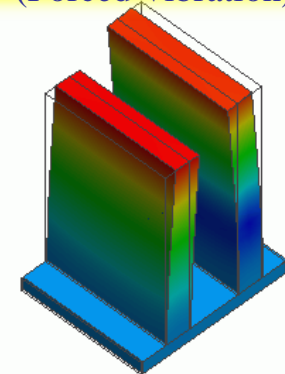
*Option for Advanced Mechanical
is required.

Resonant Analysis (Characteristic Vibration)



Resonance of Bell (Example 11)

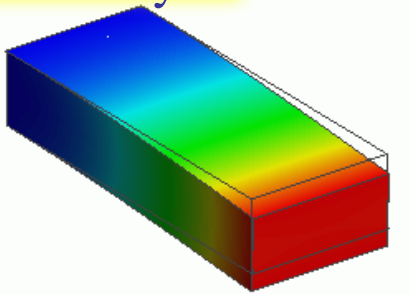
Harmonic Analysis (Forced Vibration)



Standing Bars (Example 16)

Analysis Type

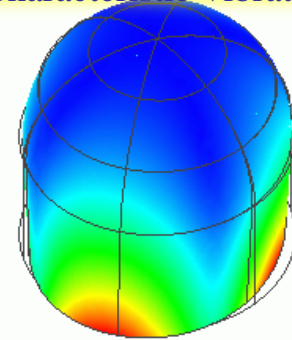
Static Analysis



Cantilever under Distributed Load (Example 1)

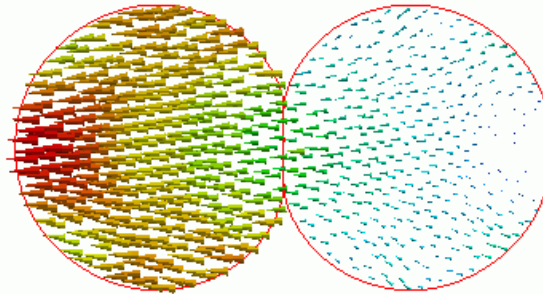
Solves characteristic vibration.
 $\{ [K] - \omega^2 [M] \} \{ x \} = \{ 0 \}$

Resonant Analysis (Characteristic Vibration)



Resonance of Bell (Example 11)

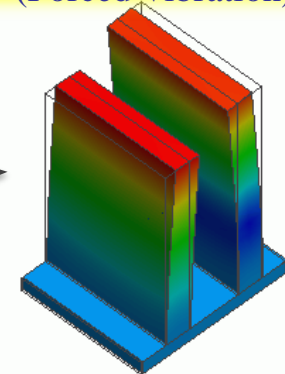
Transient Analysis



Colliding Balls (Example 32)

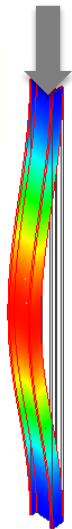
Solves vibration generated by sinusoidal vibration force.
 $\{ [K] - \omega^2 [M] \} \{ x \} = \{ f \}$

Harmonic Analysis (Forced Vibration)



Standing Bars (Example 16)

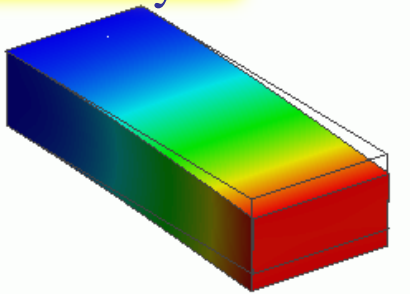
Buckling Analysis



H-beam Steel (Example 48)

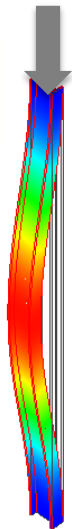
Analysis Type

Static Analysis



Cantilever under Distributed Load
(Example 1)

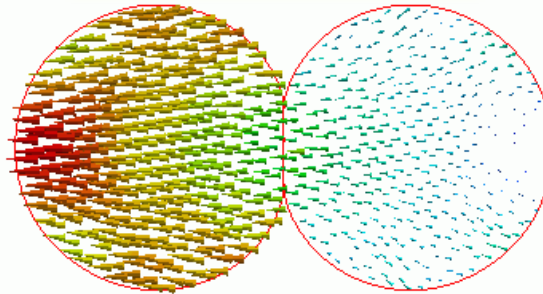
Buckling Analysis



H-beam Steel (Example 48)

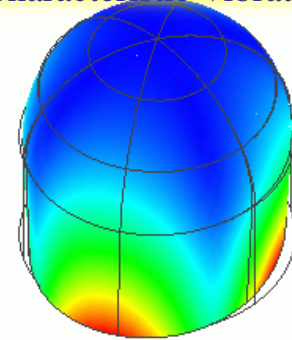
Solves buckling mode generated by a compressive force.

Transient Analysis



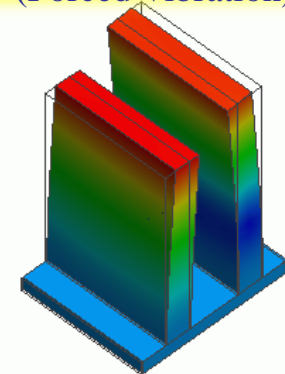
Colliding Balls (Example 32)

Resonant Analysis (Characteristic Vibration)



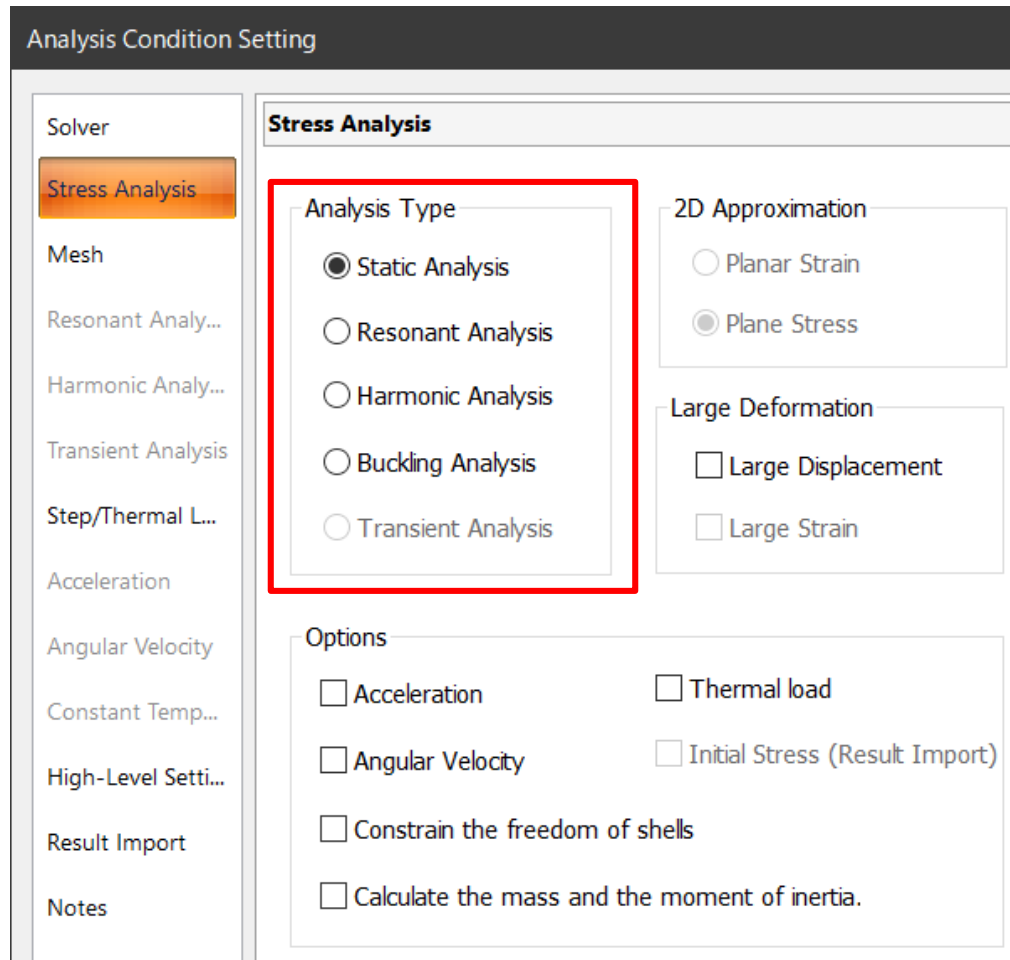
Resonance of Bell (Example 11)

Harmonic Analysis (Forced Vibration)

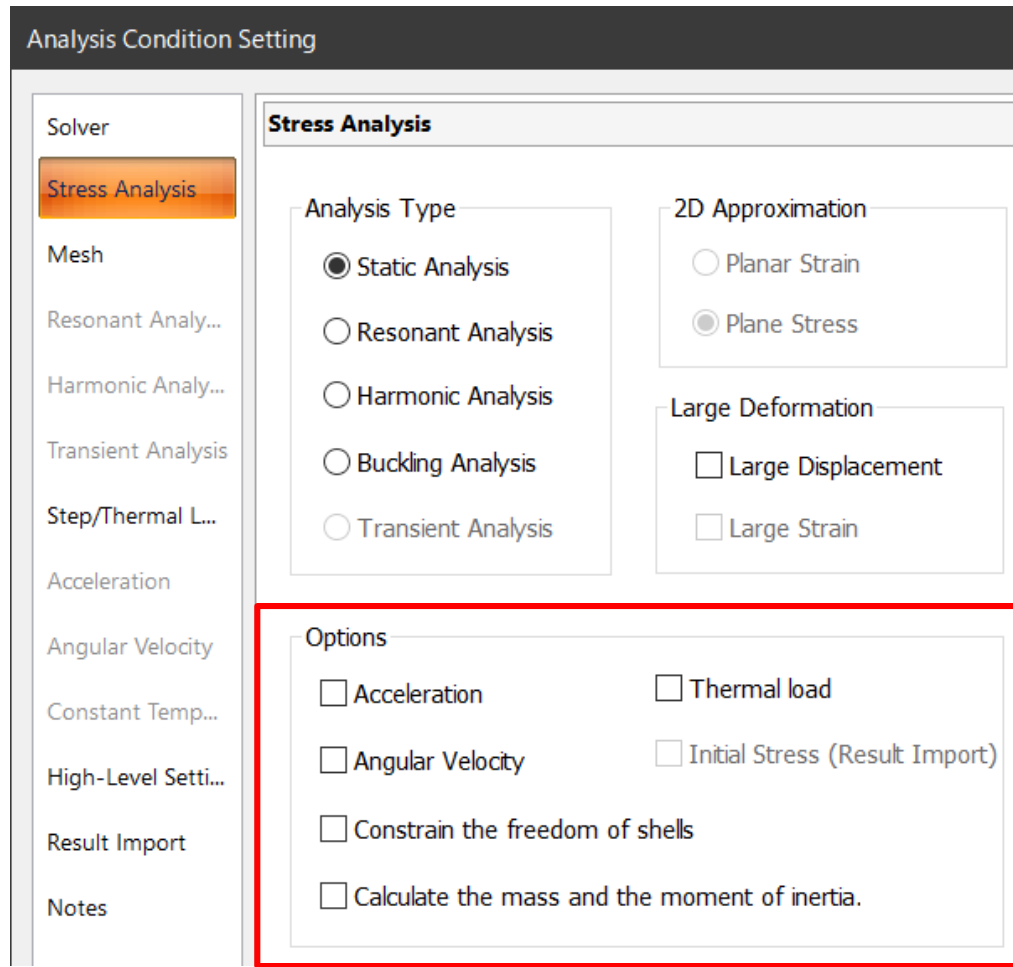


Standing Bars (Example 16)

5 analysis types are available.



Auxiliary options are available.

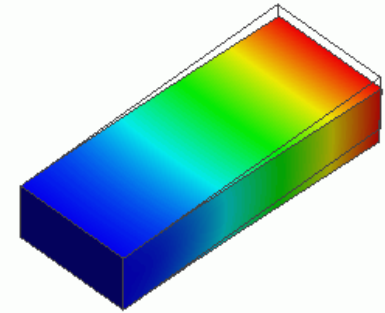


Acceleration
generates inertial force (ma).

Acceleration

X	<input type="text" value="0.0"/>	<input type="text" value="0"/>	[m/s ²]
Y	<input type="text" value="0.0"/>	X10	[m/s ²]
Z	<input type="text" value="-9.8"/>		

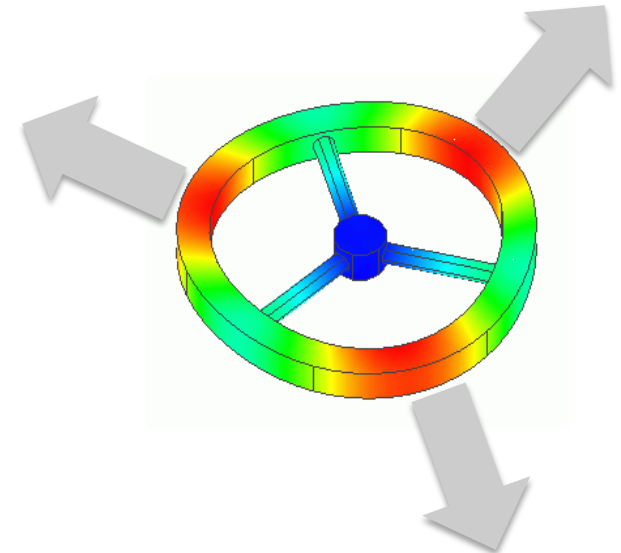
Apply gravity to the model.



Cantilever under Gravity (Example 4)

Angular Velocity
generates centrifugal force ($m\omega^2$).

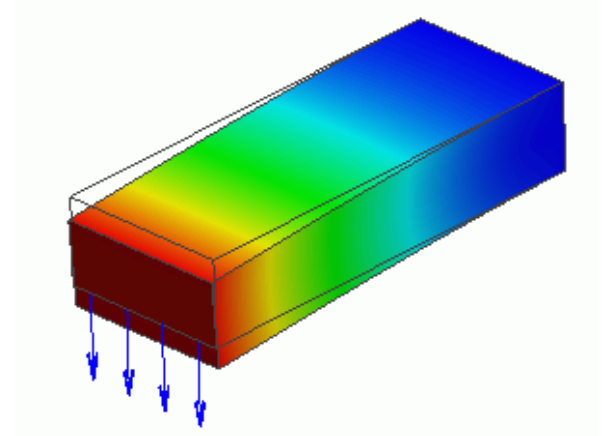
Angular Velocity	Point on Rotation Axis
X <input type="text" value="360"/>	X <input type="text" value="0.0"/>
Y <input type="text" value="0.0"/>	Y <input type="text" value="0.0"/>
Z <input type="text" value="0.0"/>	Z <input type="text" value="0.0"/>
<input type="text" value="0"/>	<input type="text" value="-3"/>
X10 [deg/s]	X10 [m]



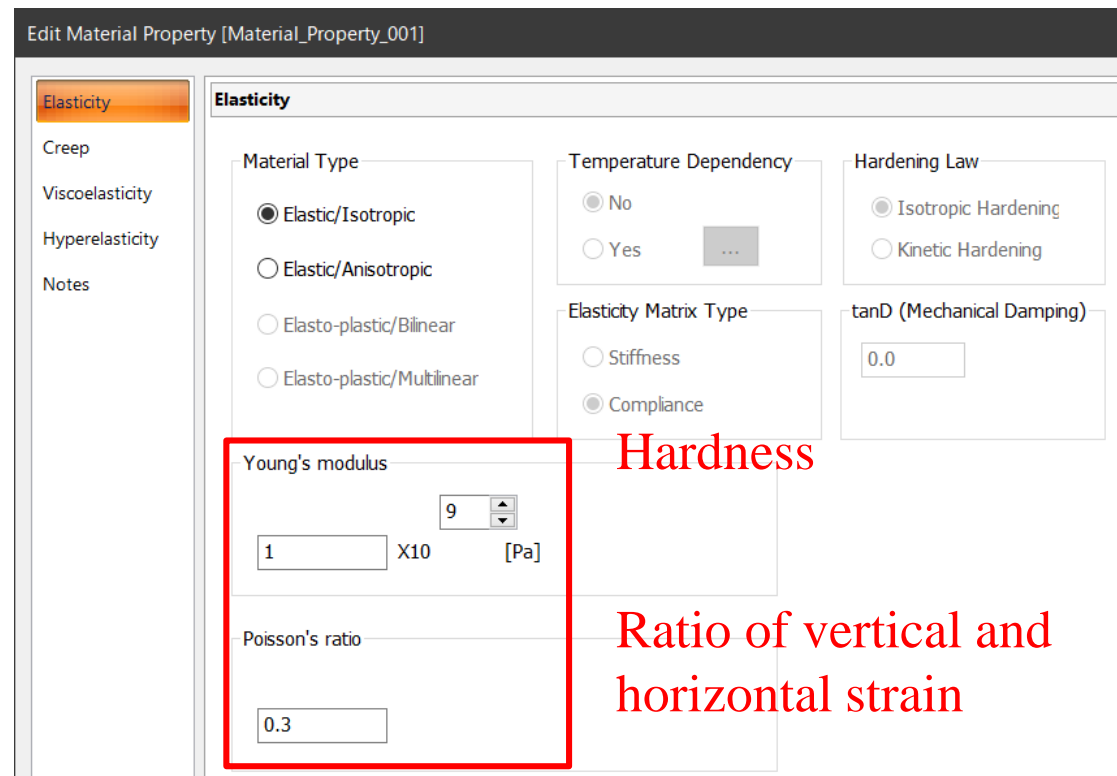
Deformation due to Centrifugal Force (Example 23)

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



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



Edit Material Property [Material_Property_001]

Elasticity

Material Type

- Elastic/Isotropic
- Elastic/Anisotropic
- Elasto-plastic/Bilinear
- Elasto-plastic/Multilinear

Temperature Dependency

- No
- Yes

Hardening Law

- Isotropic Hardening
- Kinetic Hardening

Elasticity Matrix Type

- Stiffness
- Compliance

tanD (Mechanical Damping)

0.0

Young's modulus

1 X10⁹ [Pa]

Poisson's ratio

0.3

Hardness

Ratio of vertical and horizontal strain

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

Edit Material Property [Material_Property_001]

Density

Density

Elasticity

Creep

Viscoelasticity

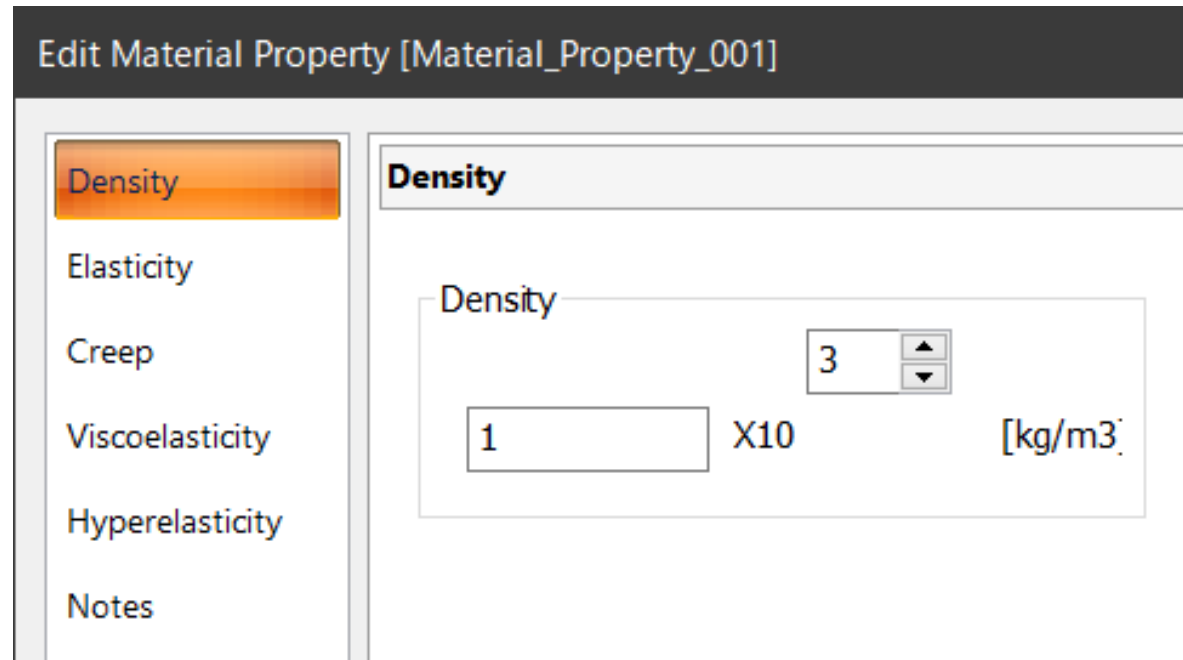
Hyperelasticity

Notes

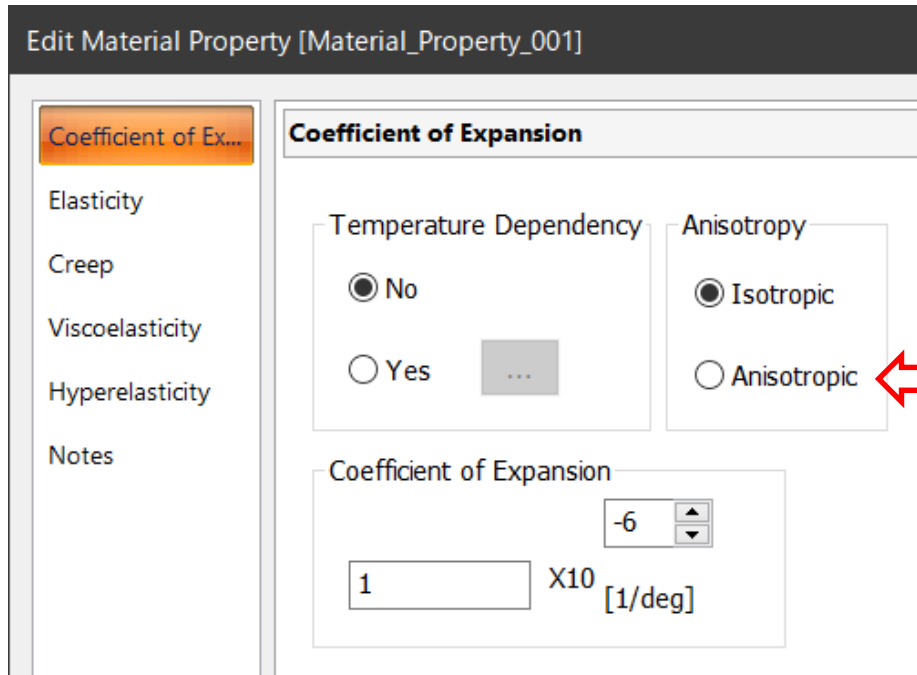
Density

3

1 X10 [kg/m³]



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



Edit Material Property [Material_Property_001]

Coefficient of Expansion

Temperature Dependency

No

Yes ...

Anisotropy

Isotropic

Anisotropic

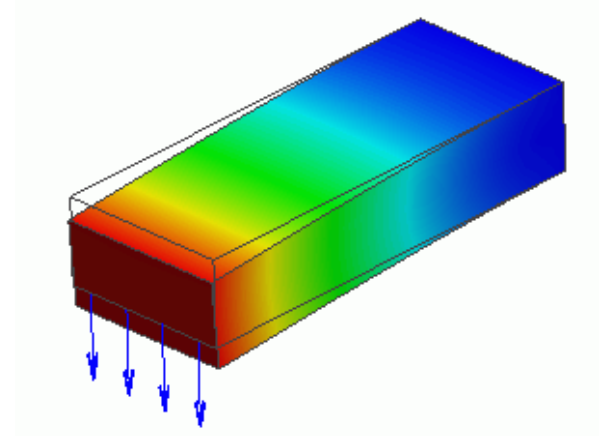
Coefficient of Expansion

1 X10⁻⁶ [1/deg]

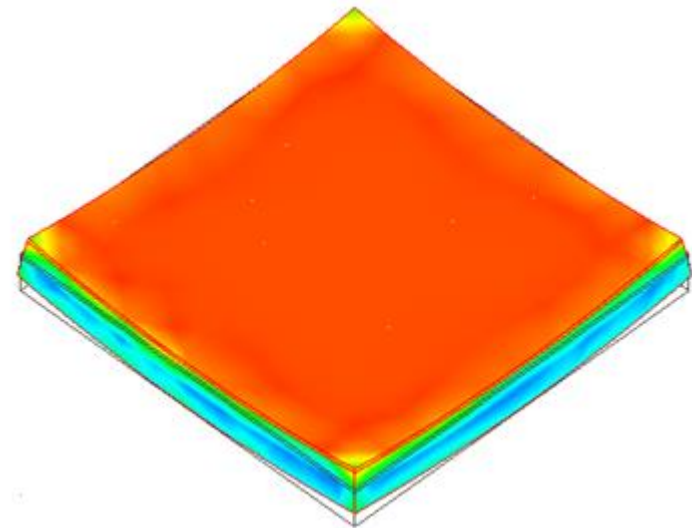
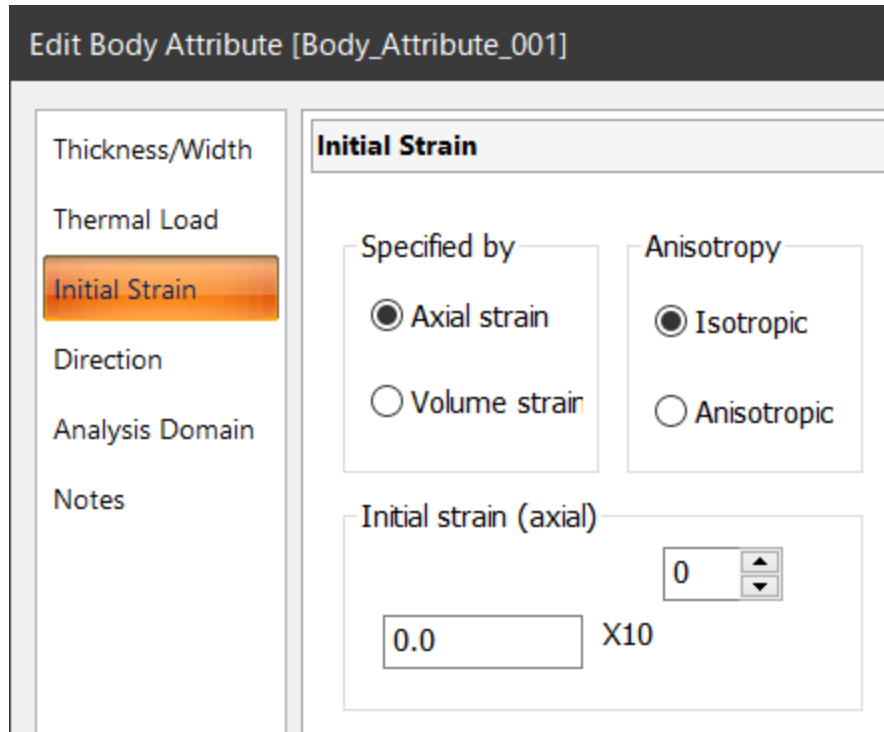
[Anisotropic] is available for Coefficient of thermal expansion and Elasticity.

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

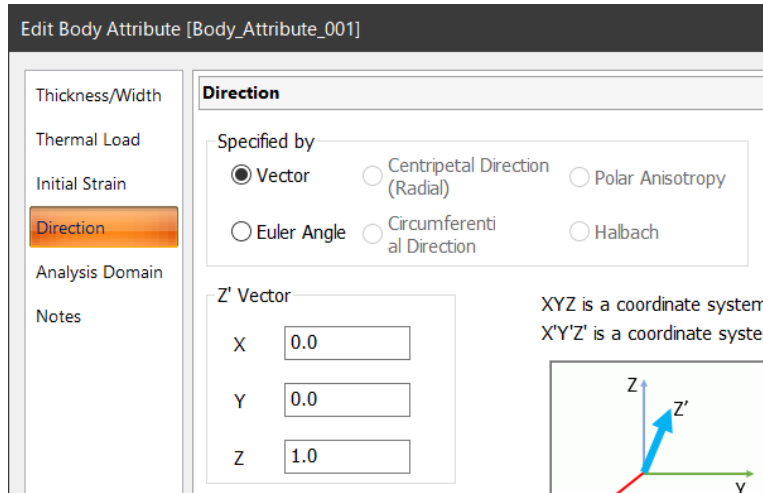


Initial stress, such as thin film stress, can be set.

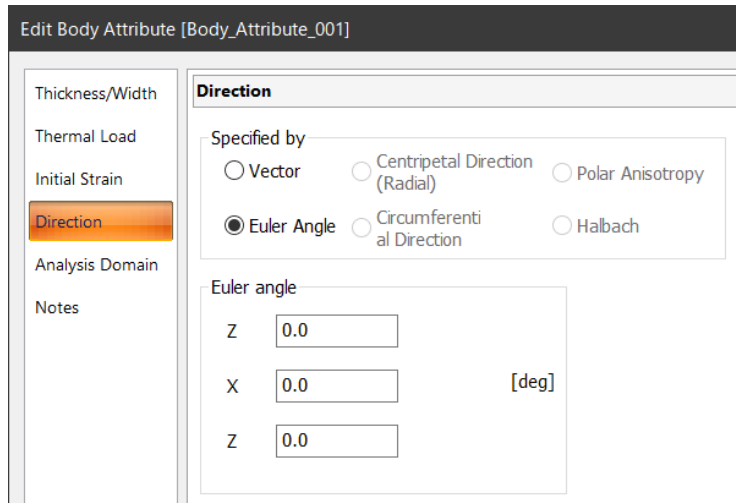


Mechanical Stress of Substrate Coated with Thin Film
(Example 41)

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.

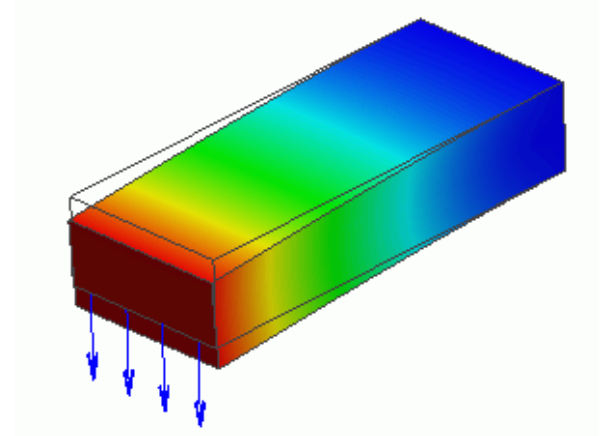


Two vectors can be specified.

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

Edit Boundary Condition [Boundary_Condition_001]

Mechanical

Symmetry/Conti...

Notes

Mechanical

Boundary Condition Type

Displacement

Normal Displacement

Rotational Displacement

Acceleration

Lumped Vertex Load

Distributed Edge Load

Distributed Face Load

Pressure

Torque Load

Simple Contact

Contact Surface

Acoustic Impedance

Open Boundary

Free

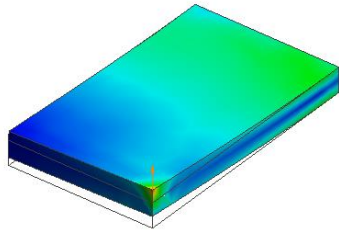
Displacement

Force

Contact

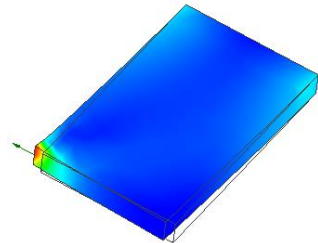
Boundary Condition [Load]

The load type is specified.



Lumped Vertex Load

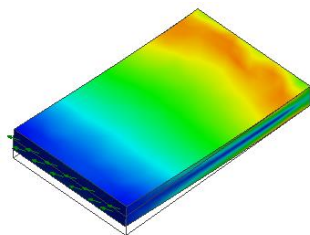
X	<input type="text" value="0.0"/>	<input type="text" value="0"/>	[N]
Y	<input type="text" value="0.0"/>	X10	
Z	<input type="text" value="1.0"/>		



Distributed Edge Load

Set the total load

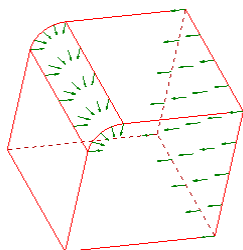
X	<input type="text" value="0.0"/>	<input type="text" value="0"/>	[N/m]
Y	<input type="text" value="0.0"/>	X10	
Z	<input type="text" value="0.0"/>		



Distributed Face Load

Set the total load

X	<input type="text" value="0.0"/>	<input type="text" value="0"/>	[N/m]
Y	<input type="text" value="0.0"/>	X10	
Z	<input type="text" value="0.0"/>		



Pressure on Face

<input type="text" value="6.0"/>	X10	<input type="text" value="0"/>	[Pa]
----------------------------------	-----	--------------------------------	------

※ Unit for the total load is [N]

Boundary Condition [Torque]

Torque is specified.

Boundary Condition Type

Displacement Lumped Vertex Load Simple Contact Acoustic Impedance

Normal Displacement Distributed Edge Load Contact Surface Open Boundary

Rotational Displacement Distributed Face Load Free

Acceleration Pressure Torque Load

Coordinates on Axis

X [m]

Y X10 [m]

Z

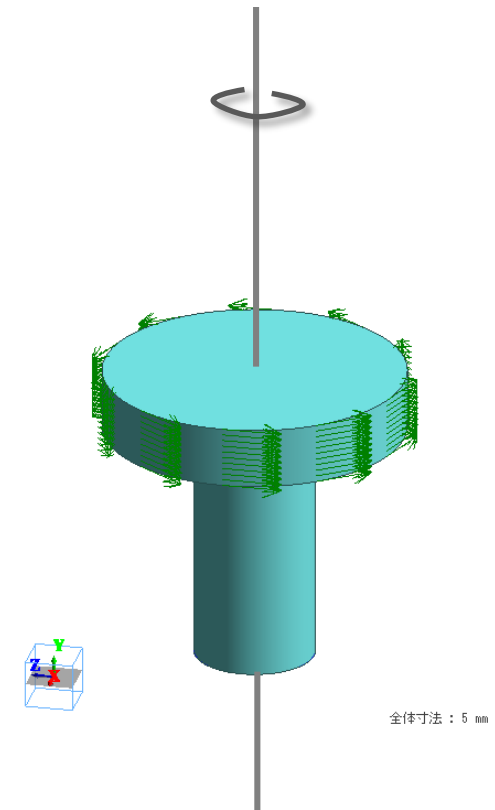
Vectors of Axis

X

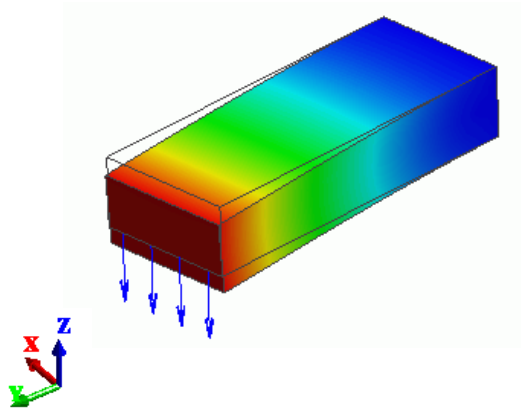
Y

Z

Torque X10 [N*m]



The displacement is specified.



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

<input type="checkbox"/> UX	<input type="text" value="0.0"/>	-3	[m]
<input type="checkbox"/> UY	<input type="text" value="0.0"/>	X10	[m]
<input checked="" type="checkbox"/> UZ	<input type="text" value="-0.5"/>		

X- and Y-direction are not constrained.

<input type="checkbox"/> UX	<input type="text" value="0.0"/>	-3	[m]
<input checked="" type="checkbox"/> UY	<input type="text" value="0.0"/>	X10	[m]
<input checked="" type="checkbox"/> UZ	<input type="text" value="-0.5"/>		

Y-direction is constrained to zero.

Boundary Condition [Rotational Displacement]

Rotational angle is specified.

Boundary Condition Type

<input type="radio"/> Displacement	<input type="radio"/> Lumped Vertex Load	<input type="radio"/> Simple Contact	<input type="radio"/> Acoustic Impedance
<input type="radio"/> Normal Displacement	<input type="radio"/> Distributed Edge Load	<input type="radio"/> Contact Surface	<input type="radio"/> Open Boundary
<input checked="" type="radio"/> Rotational Displacement	<input type="radio"/> Distributed Face Load		<input type="radio"/> Free
<input type="radio"/> Acceleration	<input type="radio"/> Pressure		
	<input type="radio"/> Torque Load		

Coordinates on Axis

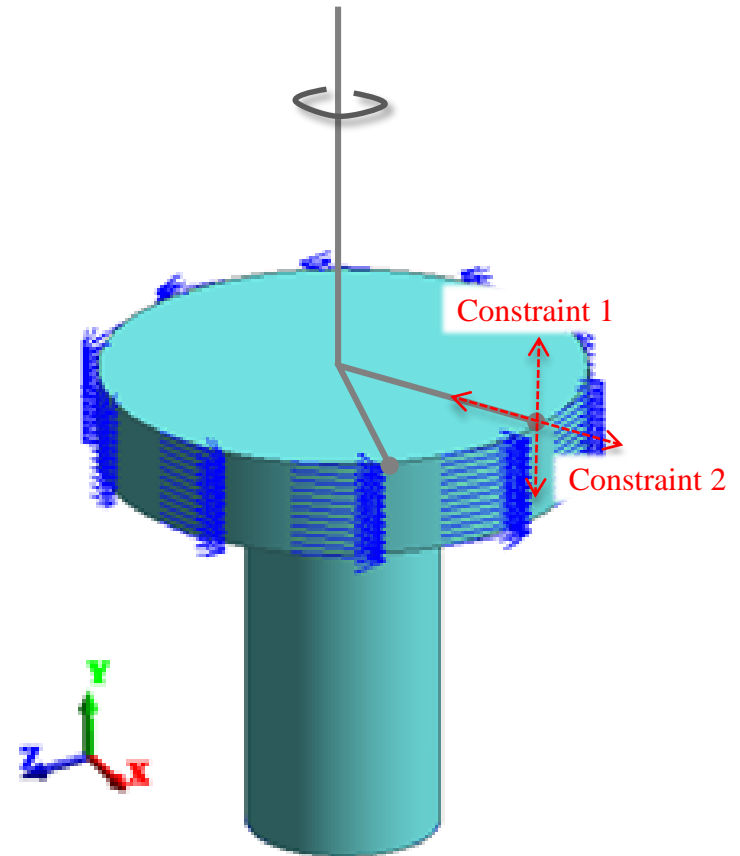
X	<input type="text" value="0.0"/>	<input type="text" value="-3"/>	
Y	<input type="text" value="0.0"/>	X10	[m]
Z	<input type="text" value="0.0"/>		

Vectors of Axis

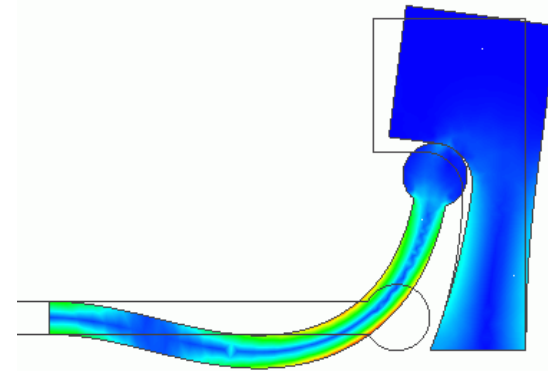
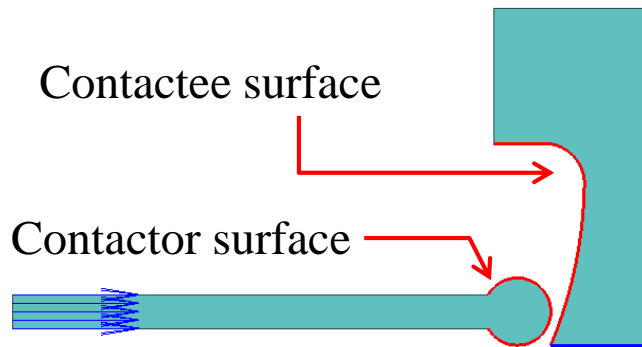
X	<input type="text" value="0.0"/>		
Y	<input type="text" value="1.0"/>		
Z	<input type="text" value="0.0"/>		

Rotation Angles

	<input type="text" value="30"/>	[deg]
<input type="checkbox"/> Rotate at constant radius		← Constraint 2
<input type="checkbox"/> Constrain displacement in axis direction.		← Constraint 1



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



Contact Analysis with Large Deformation (Example 28)

Contact Surface Classification

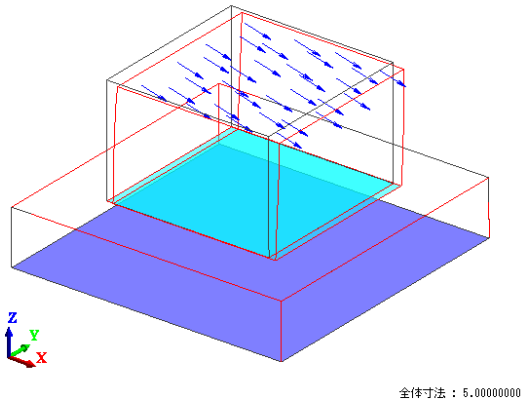
Contactor surface Contactee surface

Coefficient of Friction

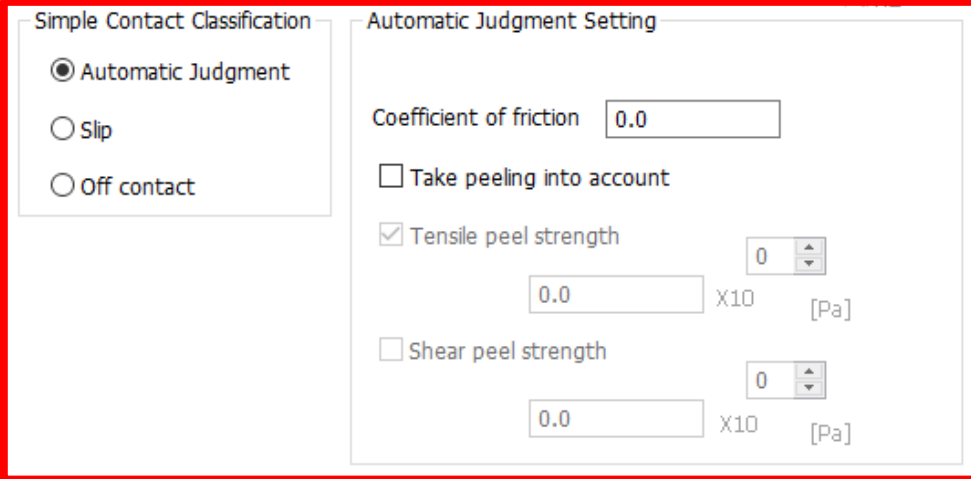
0.0

Boundary Condition

[Simple
Contact]



Frictional Contact
(Example 43)



Simple Contact Classification

- Automatic Judgment
- Slip
- Off contact

Automatic Judgment Setting

Coefficient of friction

Take peeling into account

Tensile peel strength X10 [Pa]

Shear peel strength X10 [Pa]

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

Advantages:

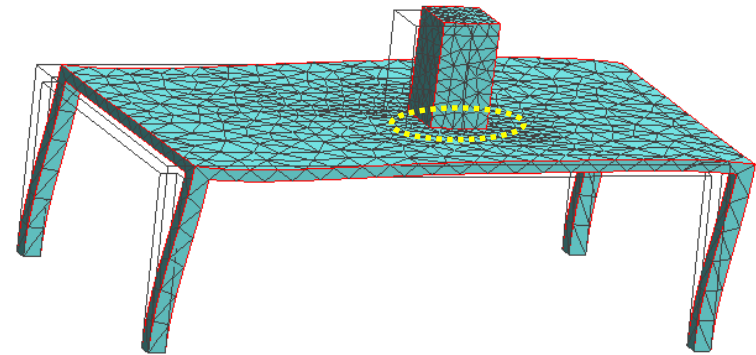
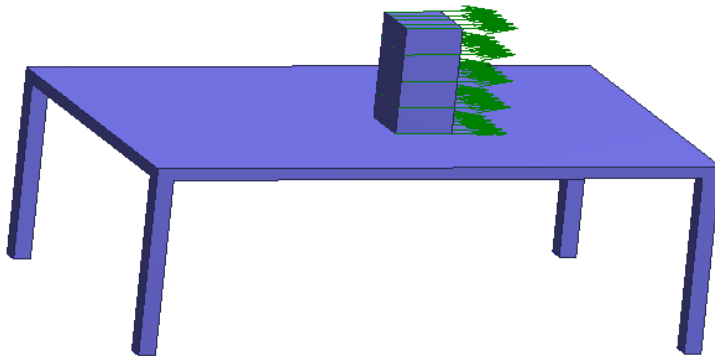
- Shorter calculation time than the usual contact setting.
- Peeling can be taken into account.

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.

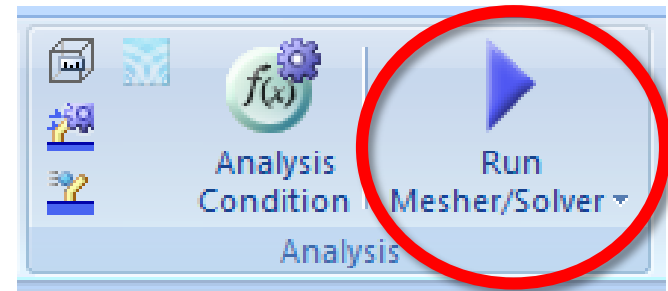
When drawing a model, if two bodies contact each other graphically, and no boundary conditions are set on the boundary, they are considered to be *bonded*.

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.



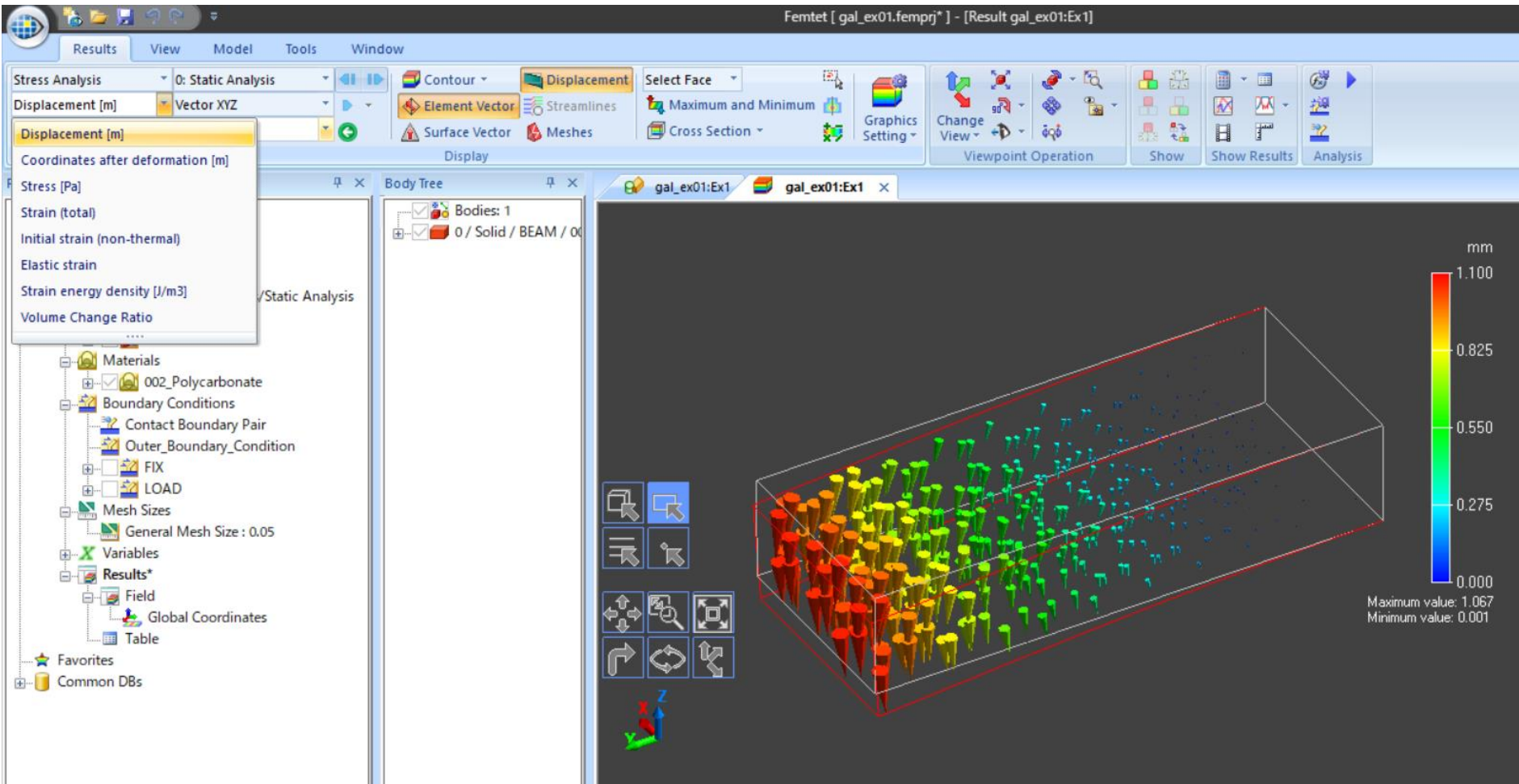
Flows of Stress Analysis

- 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



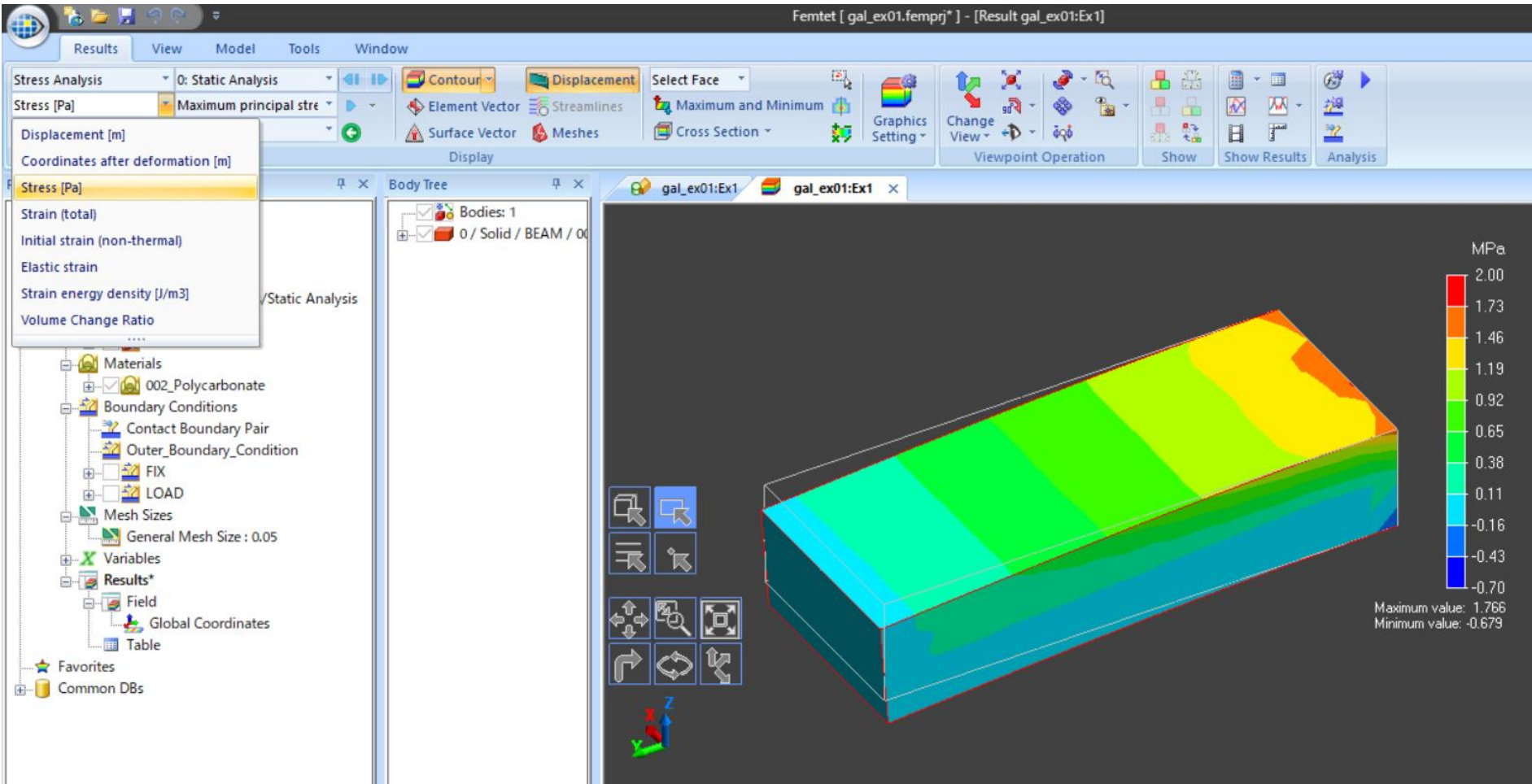
Meshes are created automatically

Results Display



Vector Display

Results Display



Color Contour

Results Table

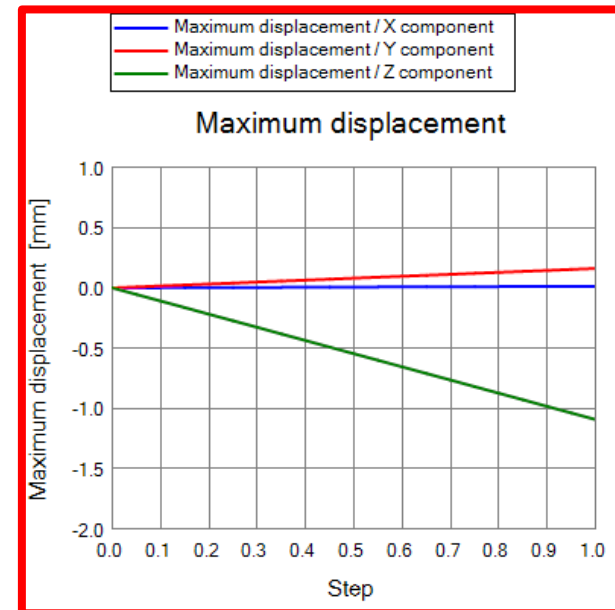
The results are output to the numerical summary table.
Graphing is possible from the table.

Table

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

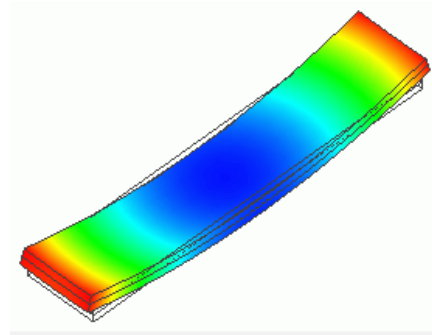
	Step	Maximum displacement / X component	Maximum displacement / Y component	Maximum displacement / Z component
0:	0.05[Step]	5.000e-2	6.865e-7	8.063e-6
1:	0.1[Step]	1.000e-1	1.373e-6	1.613e-5
2:	0.15[Step]	1.500e-1	2.060e-6	2.419e-5
3:	0.2[Step]	2.000e-1	2.746e-6	3.225e-5
4:	0.25[Step]	2.500e-1	3.433e-6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

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



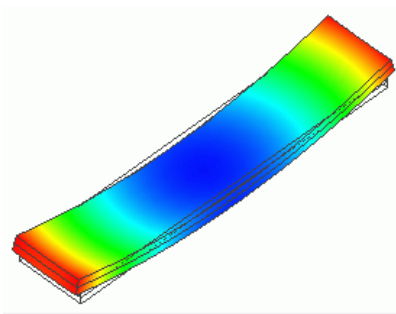
Mechanical Stress Analysis: Appendix

Step/Thermal Load Setting



Bimetal under Thermal Load
(Example 7)

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



[Step/Thermal Load] tab for the setup.

Analysis Condition Setting

Step/Thermal Load

Step Setting

Thermal load analysis
 Multi-step thermal load analysis

Time Setting

No setting
 Set up

Reference Temperature(Non-Stress Temperature)

Use distribution data
25 [deg]

Step/Reached Temperature Setting

Step	Reached Tem
1	80

Use distribution data

Options for the nonlinear analysis

Save the results of substeps
 Add unloading step

Nonlinear Setting Status

Exp
Unit [deg]

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

Multi-Step Thermal Load

Temperature change over the multiple steps is solved.

Analysis Condition Setting

Solver

Stress Analysis

Mesh

Resonant Analy...

Harmonic Analy...

Transient Analysis

Step/Thermal Lo...

Acceleration

Angular Velocity

Constant Tempe...

High-Level Setti...

Result Import

Notes

Step/Thermal Load

Step Setting

Thermal load analysis

Multi-step thermal load analysis

Time Setting

No setting

Set up

Reference Temperature(Non-Stress Temperature)

220 [deg]

Use distribution data

Distribution Data

Step/Reached Temperature Setting

Step	Substeps	Reached Tem
1	1	25
2	1	120
3	1	25
4	1	85
5		

Exp [deg]

Unit [deg]

Insert Rows Delete Rows

Temperature Graph

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

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

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

Analysis Condition Setting

Solver

- Stress Analysis
- Mesh
- Resonant Analy...
- Harmonic Analy...
- Transient Analysis
- Step/Thermal Lo...**
- Acceleration
- Angular Velocity
- Constant Tempe...
- High-Level Setti...
- Result Import
- Notes

Step/Thermal Load

Step Setting

Thermal load analysis

Multi-step thermal load analysis

Time Setting

No setting

Set up

Step/Reached Temperature Setting

Step	Substeps	Reached Tem
1	1	25
2	1	120
3	1	25
4	1	85
5		

Exp [deg]

Unit [deg]

Insert Rows Delete Rows Import

Temperature Graph **Table**

Time/Step Table

Birth/Death		Material Change		Boundary Condition ON/OFF			Weight function
Calculation step	Initial temperat	Reached tempe	BGA	PCB	SB	UF	
1	220	25	Yes	Yes	Yes	No	
2	25	120	Yes	Yes	Yes	No	
3	120	25	Yes	Yes	Yes	Yes	
4	25	85	Yes	Yes	Yes	Yes	

Step 1 (25°C)

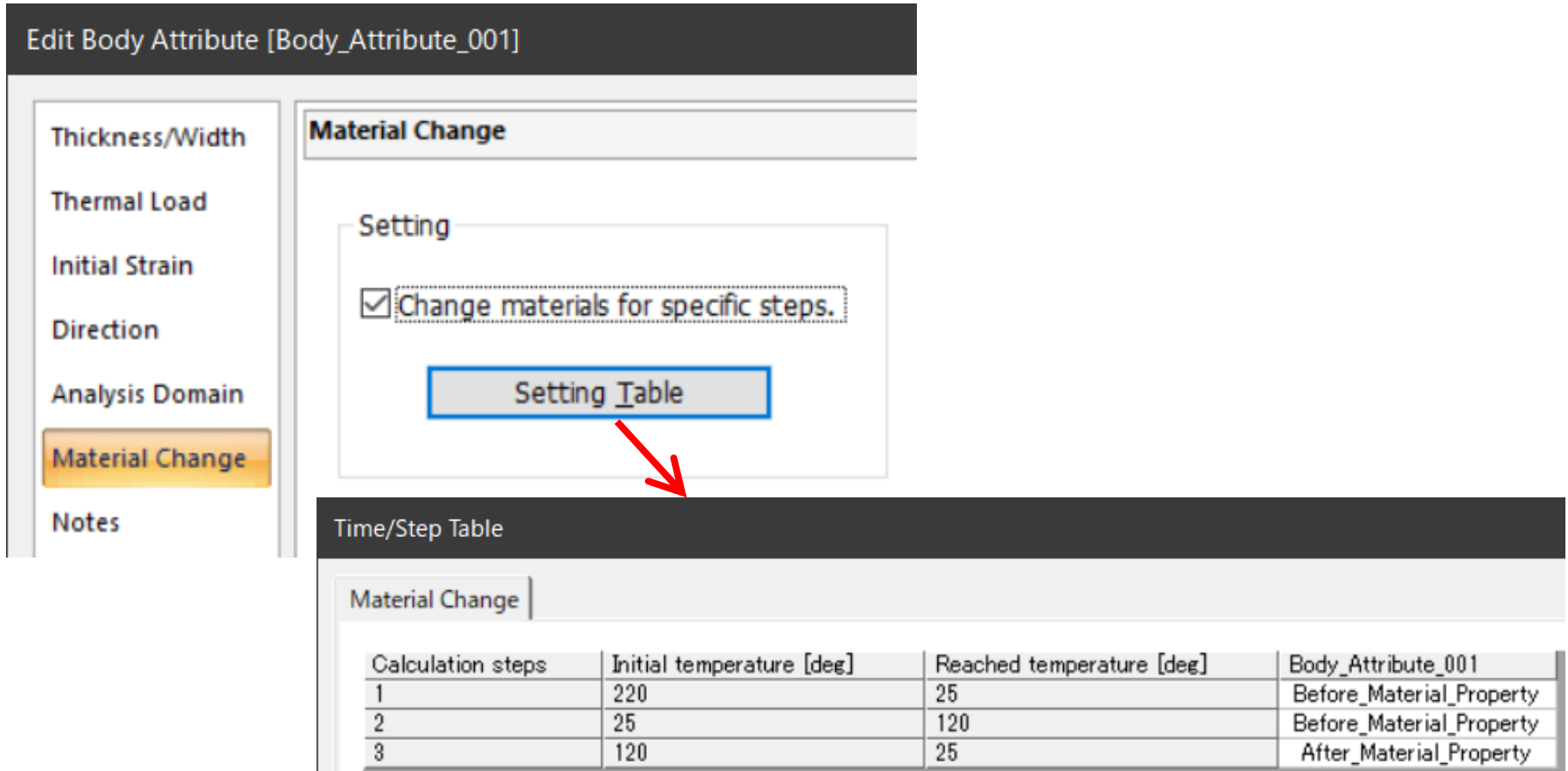
Step 2 (120°C)

Step 3 (25°C)

Step 4 (85°C)

Material Property Change

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



Setting

Change materials for specific steps.

Setting Table

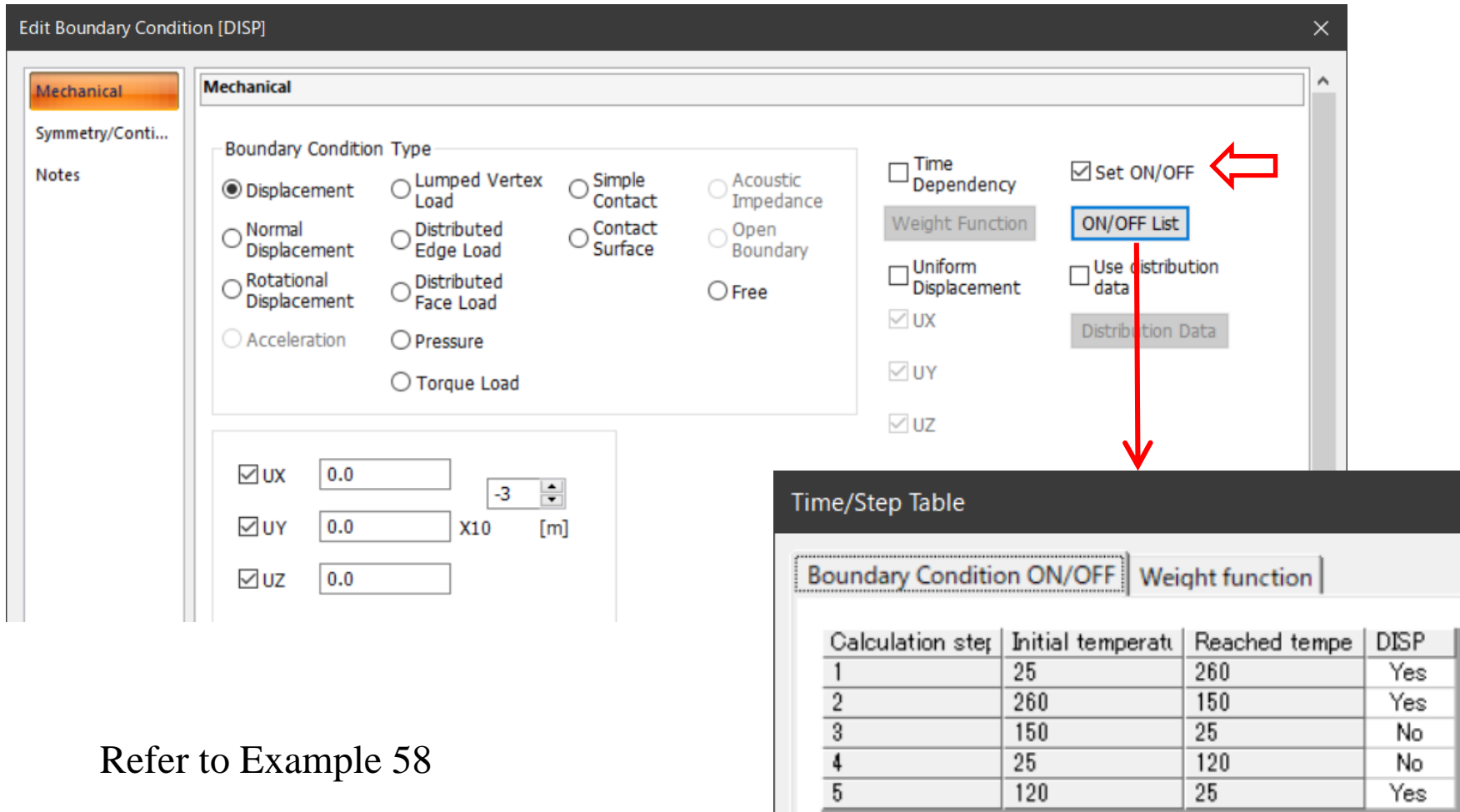
Time/Step Table

Calculation steps	Initial temperature [deg]	Reached temperature [deg]	Body_Attribute_001
1	220	25	Before_Material_Property
2	25	120	Before_Material_Property
3	120	25	After_Material_Property

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.



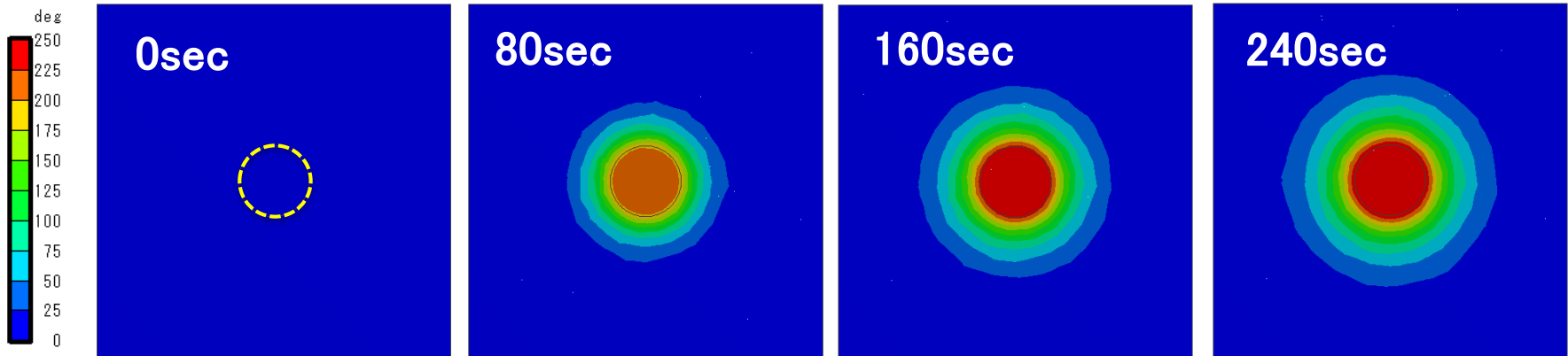
Time/Step Table

Calculation step	Initial temperature	Reached temperature	DISP
1	25	260	Yes
2	260	150	Yes
3	150	25	No
4	25	120	No
5	120	25	Yes

Refer to Example 58

2. Thermal Analysis [Watt] Overview

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



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

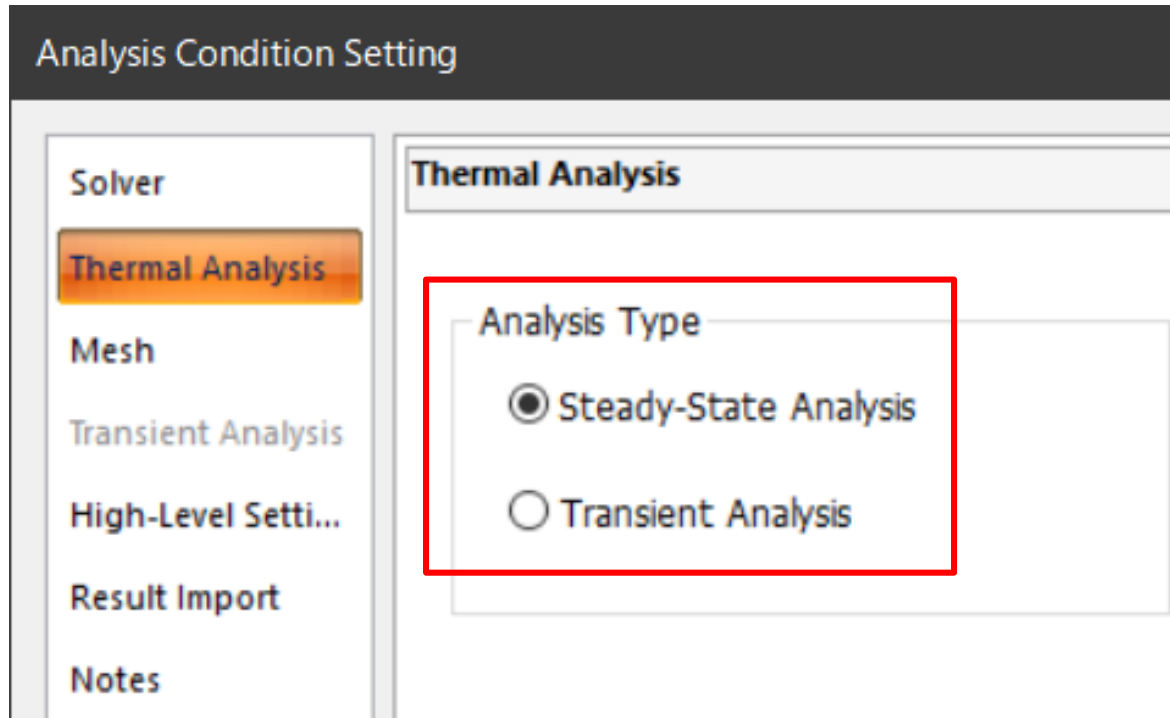
Transient Analysis

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.

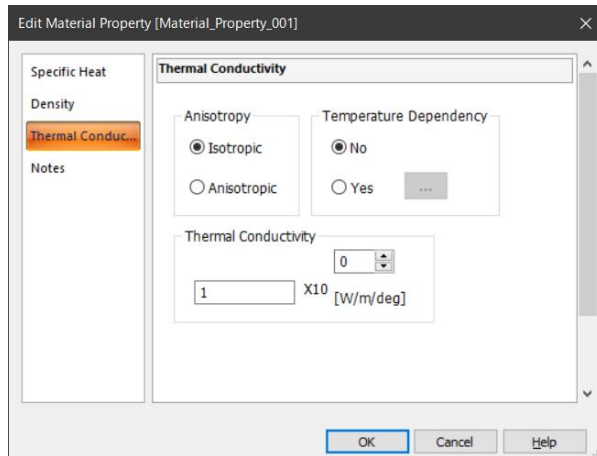
*Steady-state analysis cannot solve the problem where the steady state does not exist.
(example: where the temperature continues to go up permanently)



Thermal Conductivity

Density

Specific Heat



Edit Material Property [Material_Property_001]

Specific Heat
Density
Thermal Conduc...
Notes

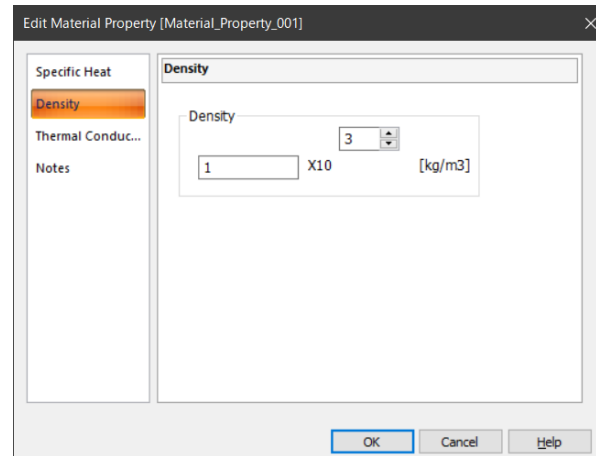
Thermal Conductivity

Anisotropy
 Isotropic
 Anisotropic

Temperature Dependency
 No
 Yes ...

Thermal Conductivity
0
1 X10 [W/m/deg]

OK Cancel Help



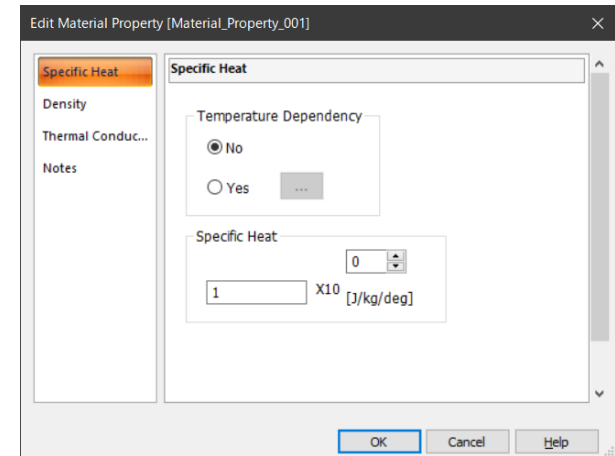
Edit Material Property [Material_Property_001]

Specific Heat
Density
Thermal Conduc...
Notes

Density

Density
3
1 X10 [kg/m3]

OK Cancel Help



Edit Material Property [Material_Property_001]

Specific Heat
Density
Thermal Conduc...
Notes

Specific Heat

Temperature Dependency
 No
 Yes ...

Specific Heat
0
1 X10 [J/kg/deg]

OK Cancel Help

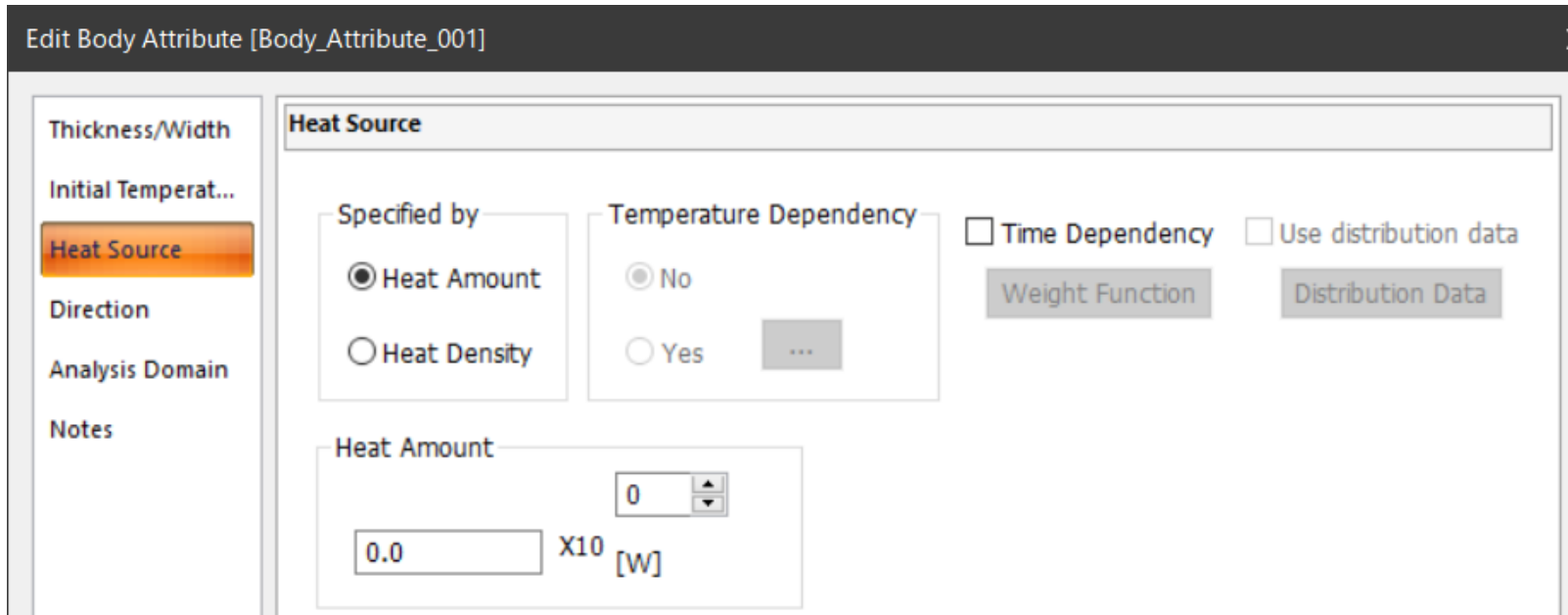
Thermal Conductivity [W/m/deg]: Examples

Au	315	Polycarbonate	0.2
Ag	427	Polyethylene	0.22
Cu	402	Glass epoxy	0.293
SUS301	16		

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.

Heat source can be set for each body.



Edit Body Attribute [Body_Attribute_001]

Thickness/Width

Initial Temperat...

Heat Source

Direction

Analysis Domain

Notes

Heat Source

Specified by

Heat Amount

Heat Density

Temperature Dependency

No

Yes

...

Time Dependency

Use distribution data

Weight Function

Distribution Data

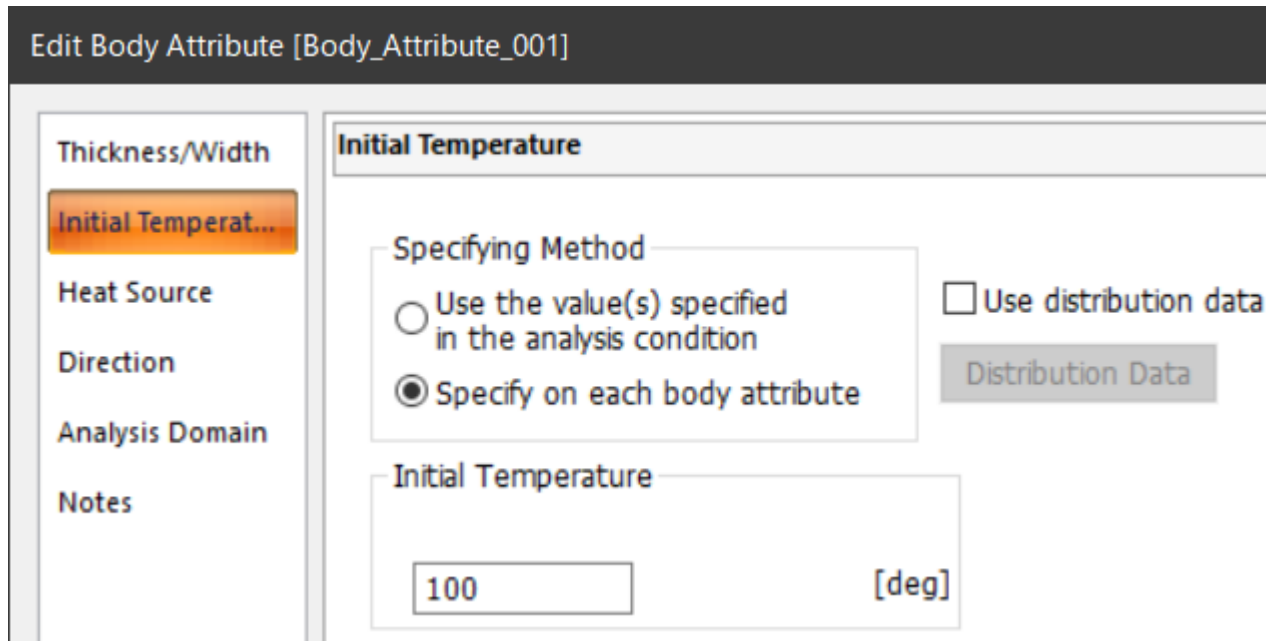
Heat Amount

0

0.0 X10 [W]

- 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.

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



Edit Body Attribute [Body_Attribute_001]

Thickness/Width

Initial Temperat...

Heat Source

Direction

Analysis Domain

Notes

Initial Temperature

Specifying Method

Use the value(s) specified in the analysis condition

Specify on each body attribute

Use distribution data

Distribution Data

Initial Temperature

100 [deg]

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

Edit Boundary Condition [Boundary_Condition_001]

Thermal

Symmetry/Conti...

Notes

Thermal

Boundary Condition Type

Temperature Heat Flux Thermal Resistance

Heat Transfer/Ambient Radiation Body-to-Body Radiation Measuring Terminal

Adiabatic (no setting)

Time Dependency Use distribution data Use distribution data Uniform Temperature

No items to set

(Example 1)

Top faces of bars: 100°C

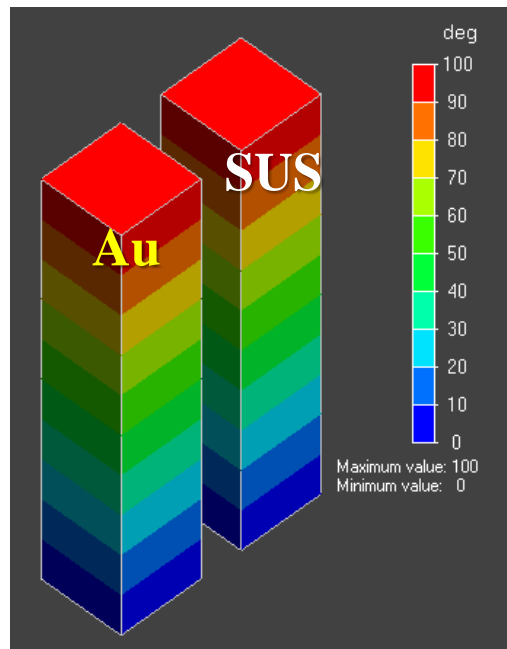
← Temperature boundary condition

Bottom faces of bars: 0°C

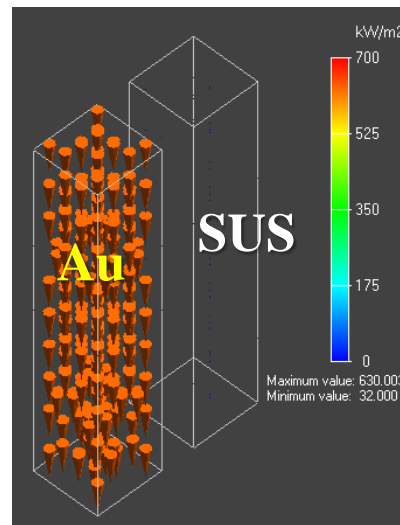
←

Sides of bars: Adiabatic

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)

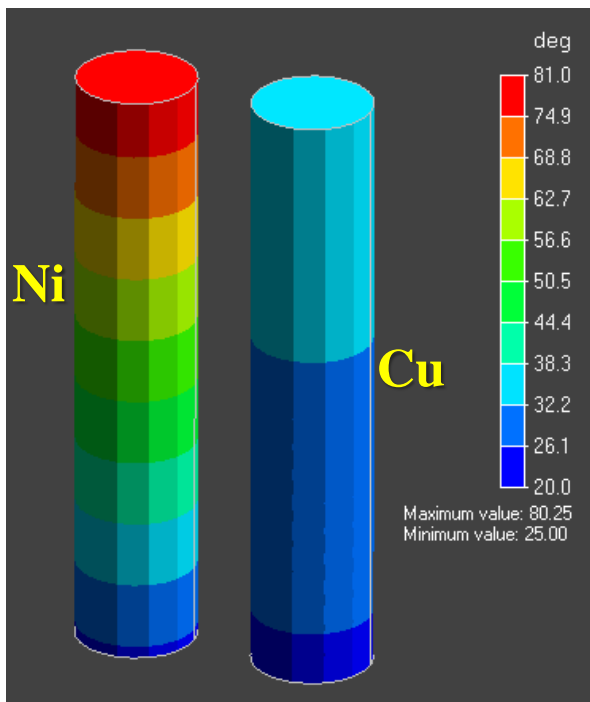
(Example 13)

Top faces of bars: Heat flux 1×10^5 [w/m²] inflow ← Boundary condition: Heat flux inflow

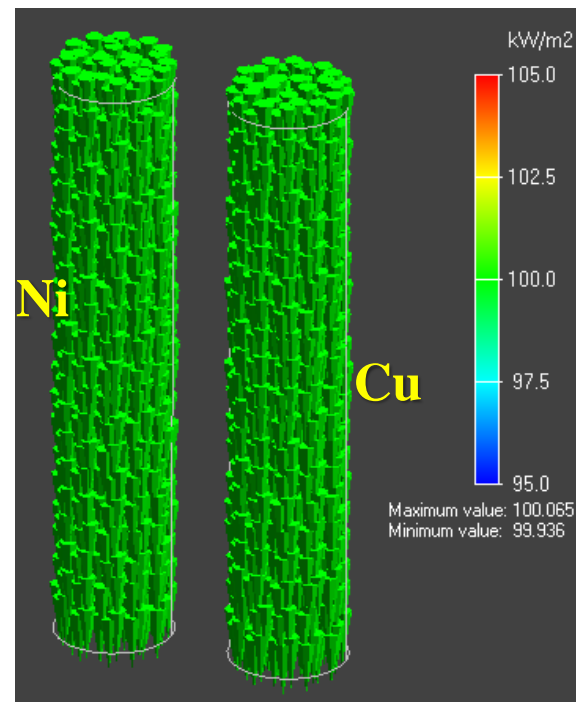
Bottom faces of bars: 25°C

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

Temperature Distribution

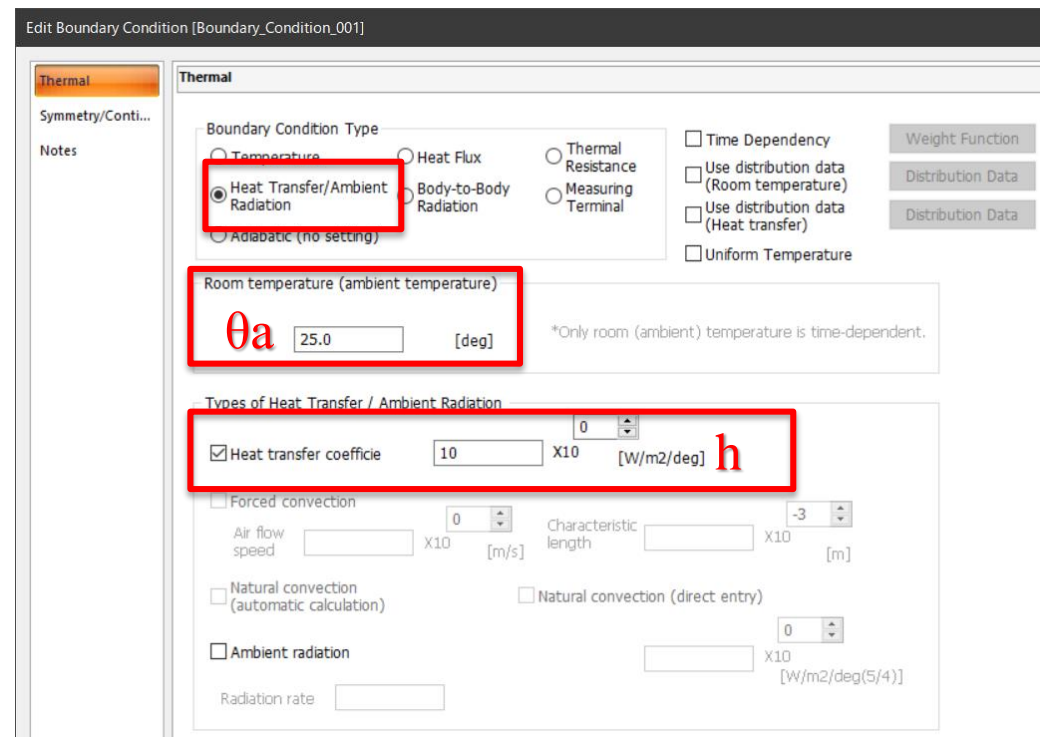
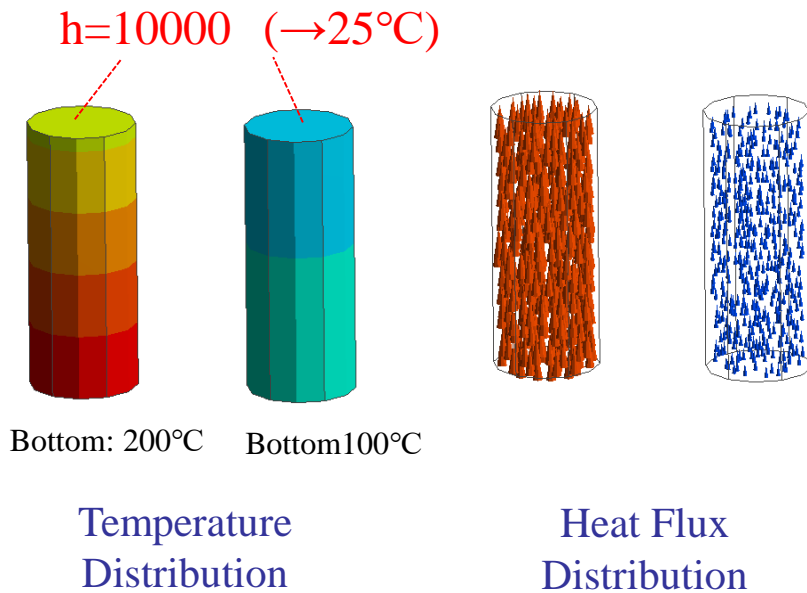


Heat Flux



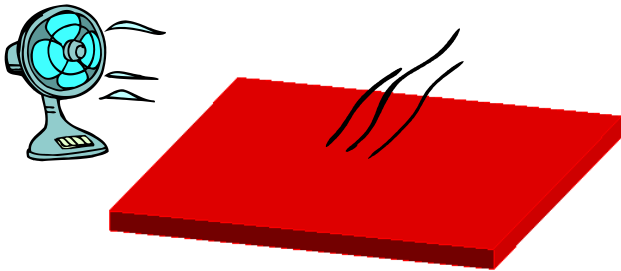
$$\text{Heat transfer (heat flux outflow)} = h \times (\theta - \theta_a)$$

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

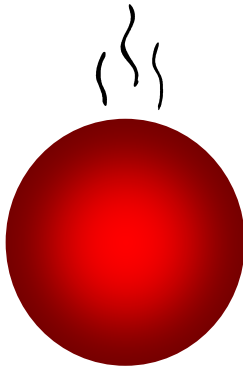


Heat transfer coefficient of the face exposed to the windless air space: 5 ~ 25 [W/m²K]

① Forced Convection



② Natural Convection



Edit Boundary Condition [Boundary_Condition_001]

Thermal

Symmetry/Conti...
Notes

Thermal

Boundary Condition Type

Temperature Heat Flux Thermal Resistance
 Heat Transfer/Ambient Radiation Body-to-Body Radiation Measuring Terminal
 Adiabatic (no setting)

Time Dependency Use distribution data (Room temperature)
 Use distribution data (Heat transfer) Uniform Temperature

Room temperature (ambient temperature)

[deg] *Only room (ambient) temperature is time-dependent.

Types of Heat Transfer / Ambient Radiation

Heat transfer coefficient X10 [W/m2/deg]

Forced convection

Air flow speed X10 [m/s] Characteristic length X10 [m]

Natural convection (automatic calculation) Natural convection (direct entry)

Ambient radiation X10 [W/m2/deg(5/4)]

Radiation rate

Equations below are solved.

*Boundary condition is actually an heat transfer coefficient.

Heat transfer by forced convection

$$[\text{Heat flux outflow}] = h \times (\theta - \theta_a)$$

θ : surface temperature

θ_a : ambient temperature

Average heat transfer coefficient h

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

v : velocity

L : typical length

Edit Boundary Condition [Boundary_Condition_001]

Thermal

Symmetry/Conti...
Notes

Thermal

Boundary Condition Type

Temperature
 Heat Flux
 Thermal Resistance
 Time Dependency
 Weight Function

Heat Transfer/Ambient Radiation
 Body-to-Body Radiation
 Measuring Terminal
 Use distribution data (Room temperature)
 Distribution Data

Adiabatic (no setting)
 Use distribution data (Heat transfer)
 Distribution Data

Uniform Temperature

Room temperature (ambient temperature)

θ_a [deg] *Only room (ambient) temperature is time-dependent.

Types of Heat Transfer / Ambient Radiation

Heat transfer coefficient X10 [W/m2/deg]

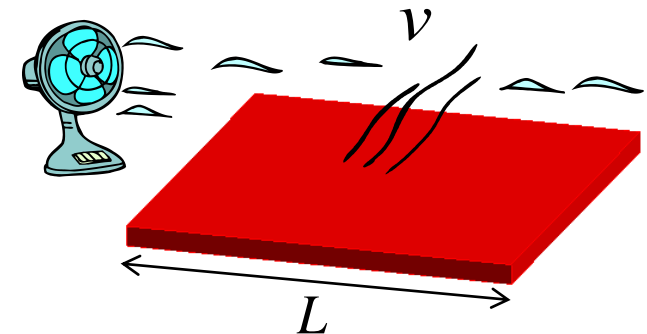
Forced convection

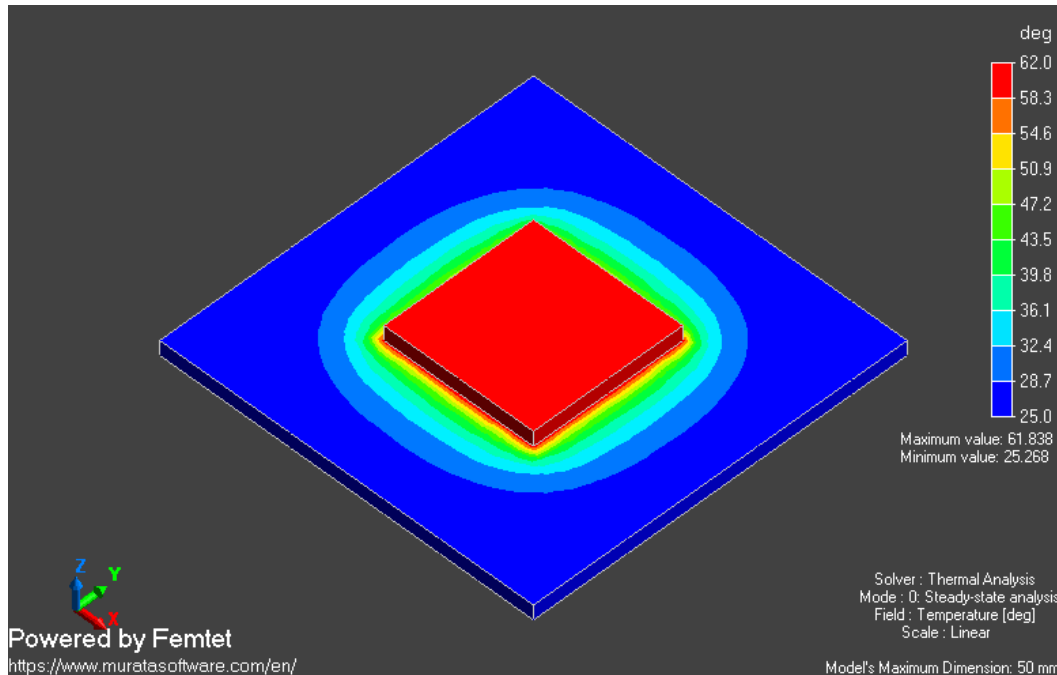
Air flow speed X10 [m/s]
 Characteristic length X10 [m]

Natural convection (automatic calculation)
 Natural convection (direct entry)

Ambient radiation X10 [W/m2/deg(5/4)]

Radiation rate





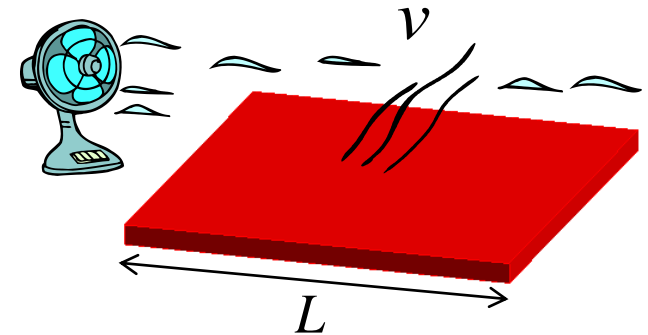
Heat transfer by forced convection
[Heat flux outflow] = $h \times (\theta - \theta_a)$

Average heat transfer coefficient h

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

v : velocity

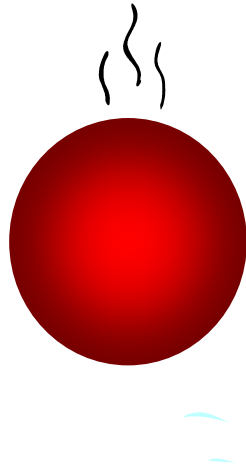
L : typical length



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.

Natural Convection



Heat transfer by natural convection

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

where

θ : surface temperature

θ_a : 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.

Edit Boundary Condition [Boundary_Condition_001]

Thermal

Symmetry/Conti...
Notes

Thermal

Boundary Condition Type

Temperature Heat Flux Thermal Resistan
 Heat Transfer/Ambient Radiation Body-to-Body Radiation Measurin Terminal
 Adiabatic (no setting)

Room temperature (ambient temperature)

θ_a [deg] *Only room

Types of Heat Transfer / Ambient Radiation

Heat transfer coefficie X10 [

Forced convection

Air flow speed X10 [m/s] Characteris length

Natural convection (automatic calculation) Natural conv

Ambient radiation

Radiation rate

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

where

θ : surface temperature

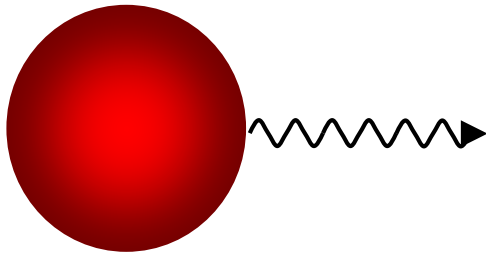
θ_a : ambient temperature

h' : coefficient given by size and direction of face

* h' is automatically calculated.

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

③ Ambient Radiation



Edit Boundary Condition [Boundary_Condition_001]

Thermal

Symmetry/Conti...
Notes

Thermal

Boundary Condition Type

Temperature Heat Flux Thermal Resistance
 Heat Transfer/Ambient Radiation Body-to-Body Radiation Measuring Terminal
 Adiabatic (no setting)

Time Dependency Use distribution data (Room temperature)
 Use distribution data (Heat transfer)
 Uniform Temperature

Weight Function
Distribution Data
Distribution Data

Room temperature (ambient temperature)

[deg] *Only room (ambient) temperature is time-dependent.

Types of Heat Transfer / Ambient Radiation

Heat transfer coefficient X10 [W/m2/deg]

Forced convection

Air flow speed X10 [m/s] Characteristic length X10 [m]

Natural convection (automatic calculation) Natural convection (direct entry)

Ambient radiation X10 [W/m2/deg(5/4)]

Radiation rate

Edit Boundary Condition [Boundary_Condition_001]

Thermal

Symmetry/Conti...
Notes

Thermal

Boundary Condition Type

Temperature Heat Flux Thermal Resistance T

Heat Transfer/Ambient Radiation Body-to-Body Radiation Measuring Terminal U (F)

Adiabatic (no setting) U (f)

Room temperature (ambient temperature)

θ_a 25.0 [deg] *Only room (ambient) te

Types of Heat Transfer / Ambient Radiation

Heat transfer coefficient [] X10 [W/m2/deg]

Forced convection

Air flow speed [] X10 [m/s] Characteristic length []

Natural convection (automatic calculation) Natural convection (direct)

Ambient radiation

k Radiation rate 0.8

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)

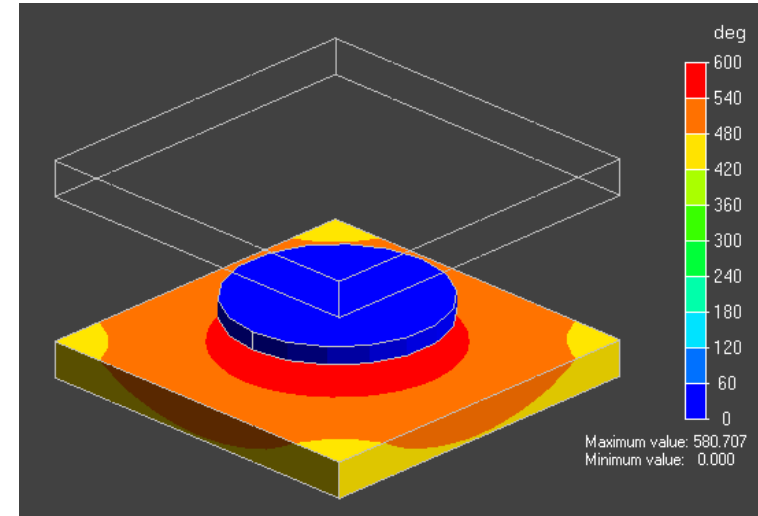
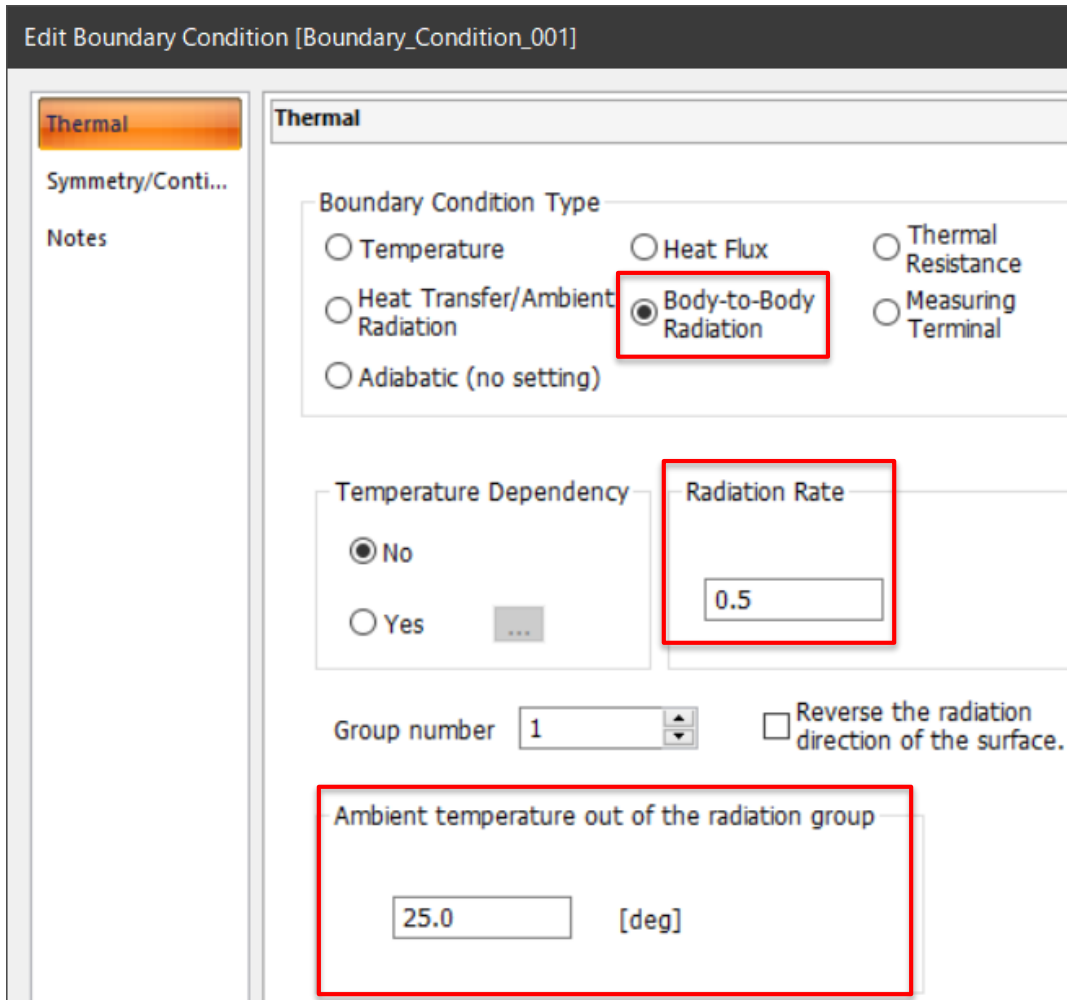
θ : surface temperature

θ_a : 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

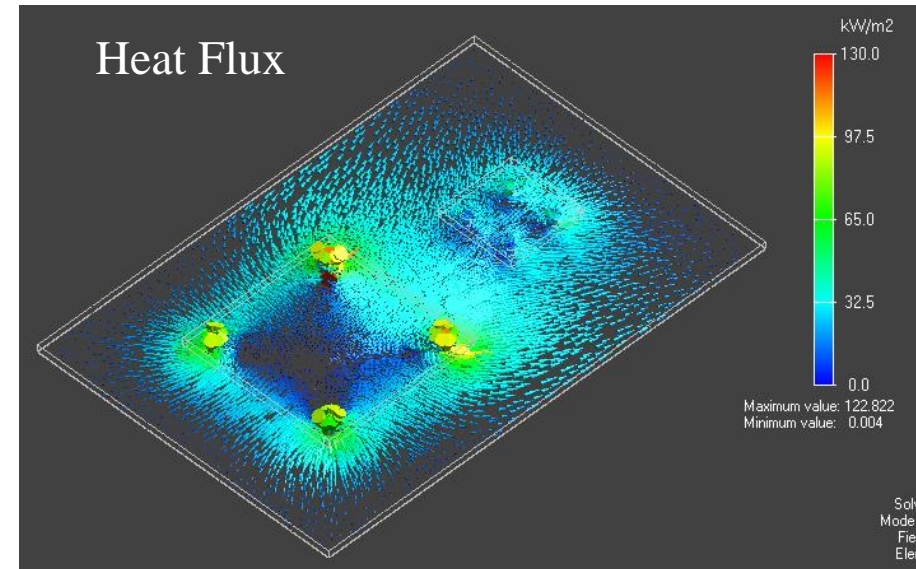
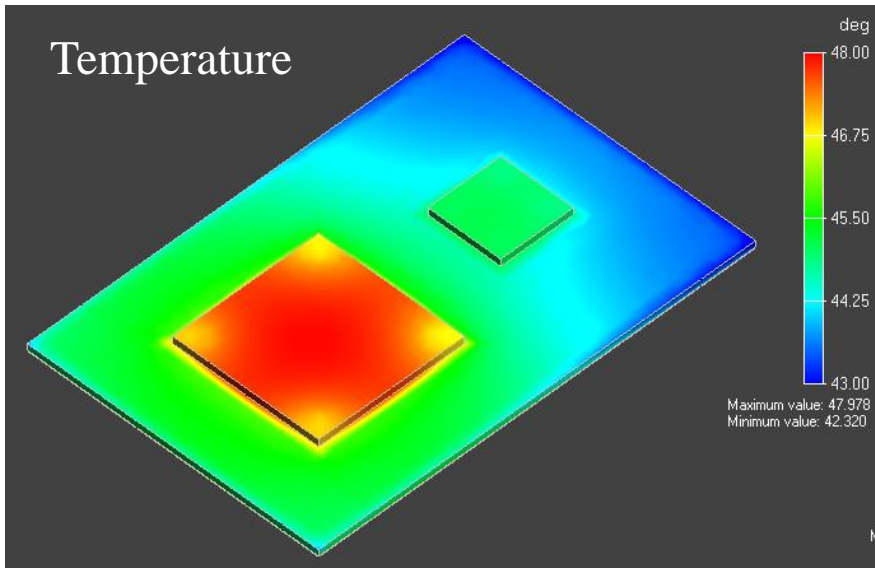
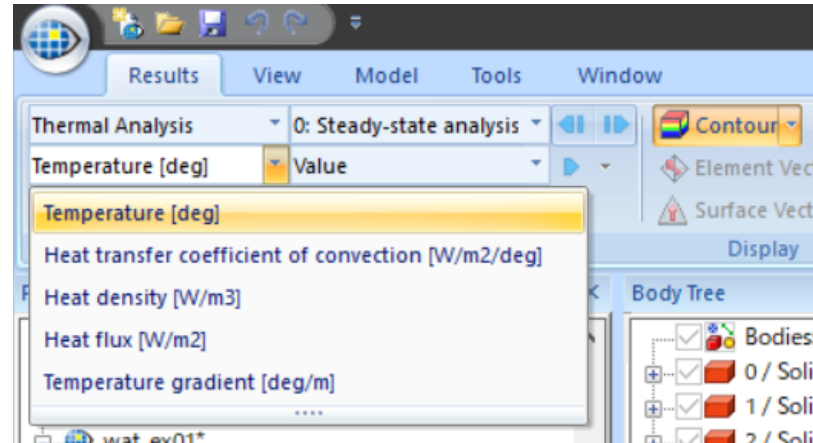
Though the view factor is assumed to be 1, it is, strictly speaking, between 0 and 1. It is 1 on the plane or on the convex plane.

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



Results Table

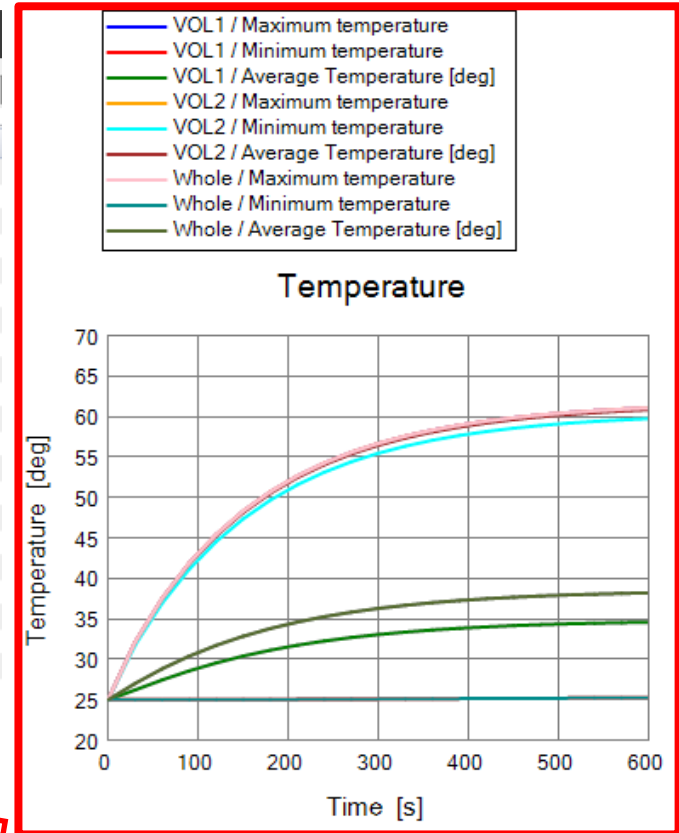
The results are output to the numerical table.
Graphing the results is possible.

Table

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

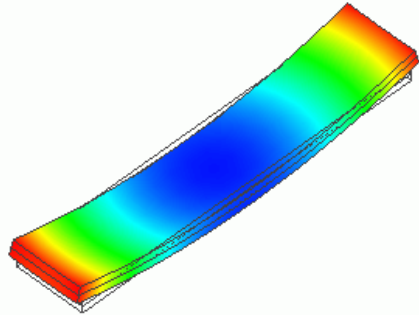
	Time [s]	VOL1 / Maxi...	VOL1 / Mini...	VOL1 / Ave...	VOL2 / Max...	VOL2 / Min...	VOL2 / Ave...	Whole /
0:	0.000e+00[s]	0	25.000	25.000	25.000	25.000	25.000	25.000
1:	3.000e+01[s]	30	32.109	25.000	26.212	32.122	31.738	32.059
2:	6.000e+01[s]	60	37.527	25.001	27.454	37.540	36.940	37.438
3:	9.000e+01[s]	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

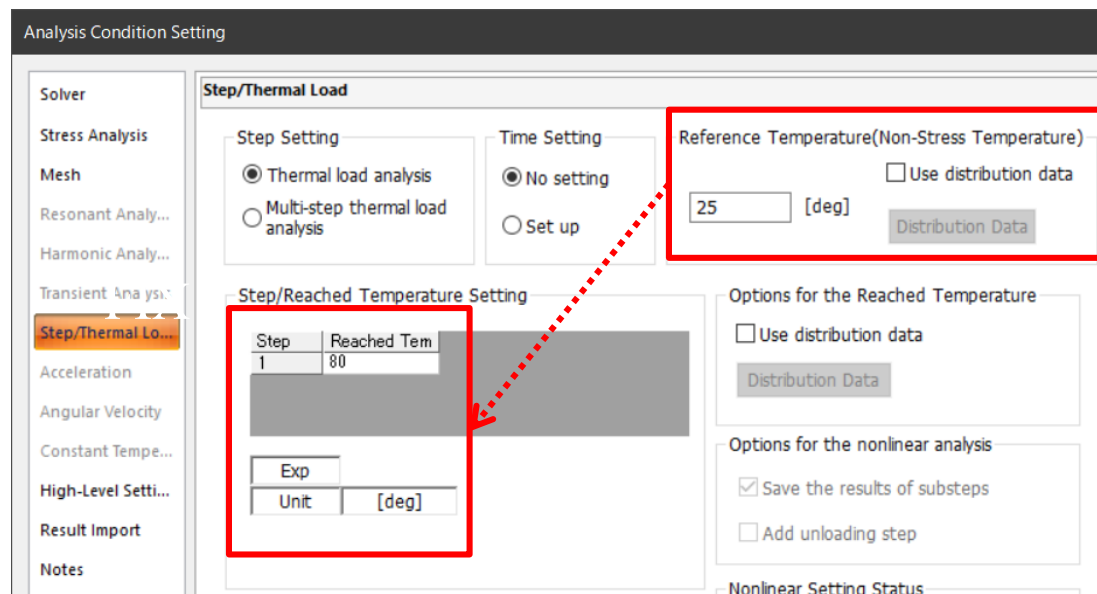
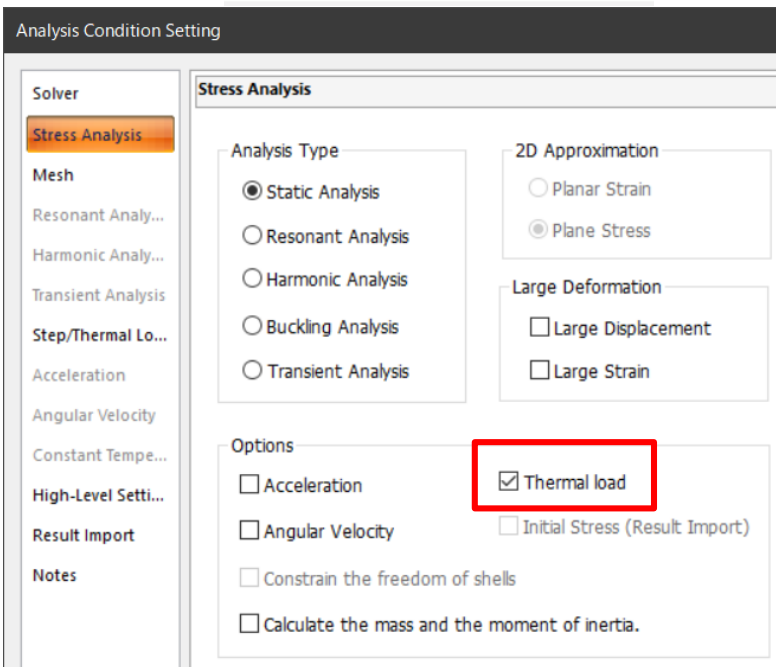


3. Thermal-Mechanical Stress Analysis

Stress Analysis Example 7: Bimetal under Thermal Load

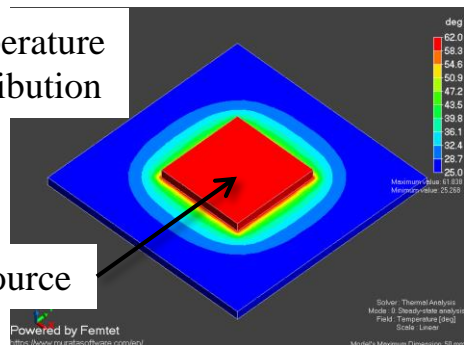


If the model has no temperature distribution, thermal load can be opted in the stress analysis. Thermal analysis is not needed.

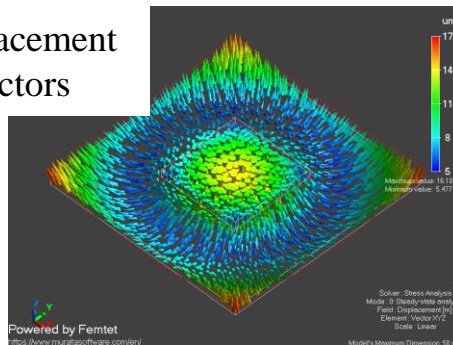


Using Temperature Distribution Obtained in Thermal Analysis

Temperature Distribution



Displacement Vectors



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

Heat Source

Powered by Femtet

Powered by Femtet

Analysis Condition Setting

Select both Stress Analysis and Thermal Analysis

Solver

Thermal-Stress ...

Mesh

Transient Analysis

Step/Thermal Lo...

Acceleration

Angular Velocity

High-Level Setti...

Result Import

Notes

Solver

Mechanical Stress / Piezoelectric

Stress Analysis

Piezoelectric Analysis

Galileo

Rayleigh

Acoustic / Fluid

Acoustic Analysis

Simple Fluid Analysis

Fluid Analysis

Mach

Thermal

Thermal analysis

Electric-Thermal Coupled Analysis

Electromagnetic Field

Electric Analysis

Magnetic Analysis

Electromagnetic Analysis

Hertz

Reference Temperature Setting

Analysis Condition Setting

Step/Thermal Load

Step Setting

Thermal coupled steps

Multi-step thermal load + thermal coupled steps

Time Setting

No setting

Set up

Reference Temperature(Non-Stress Temperature)

25 [deg]

Use distribution data

Distribution Data

Step/Reached Temperature Setting

Step setting is not required for the linear analysis of the steady-state thermal coupled analysis.

Options for the Reached Temperature

Use distribution data

Distribution Data

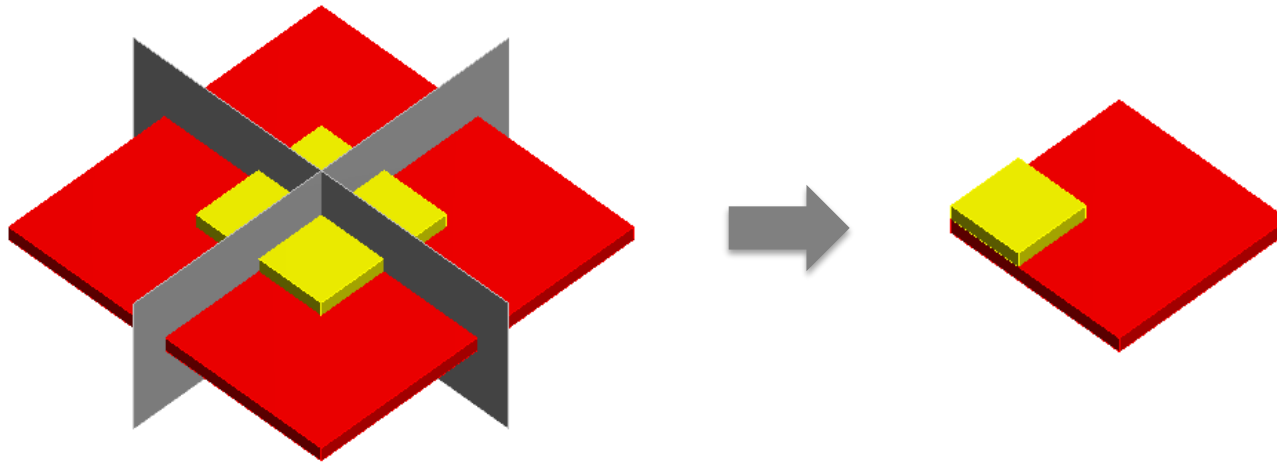
4. Symmetric Model

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

Advantage of Symmetric Model Murata Software

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

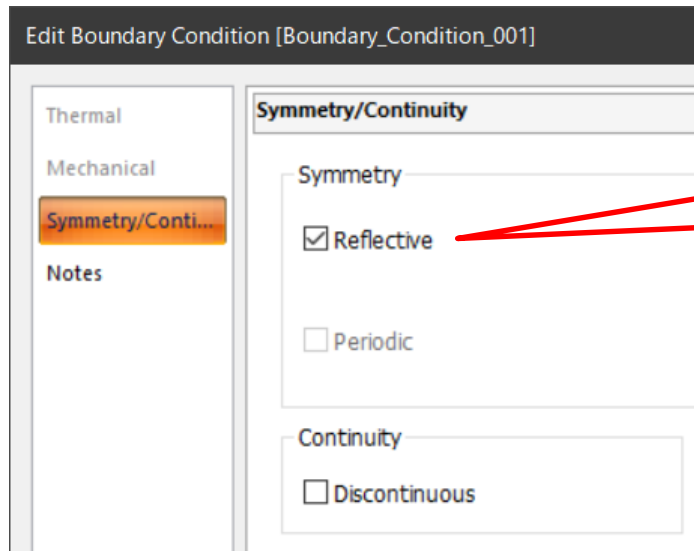


Full Model

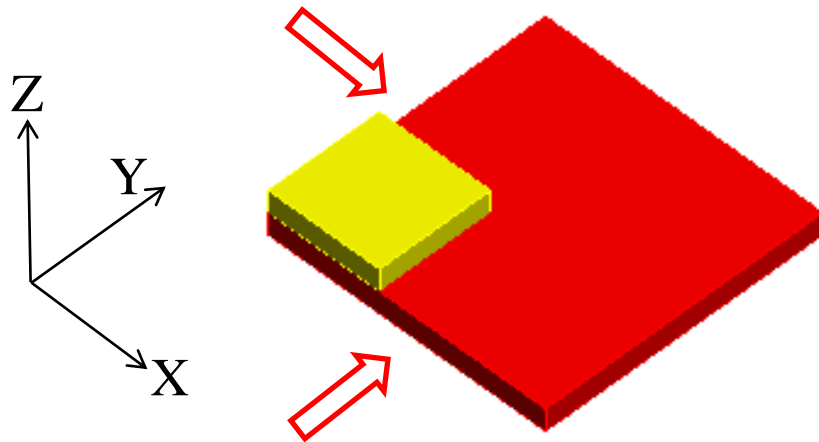
Symmetric Model

Calculation time and memory consumption are greatly reduced.

Symmetric Boundary Condition Murata Software



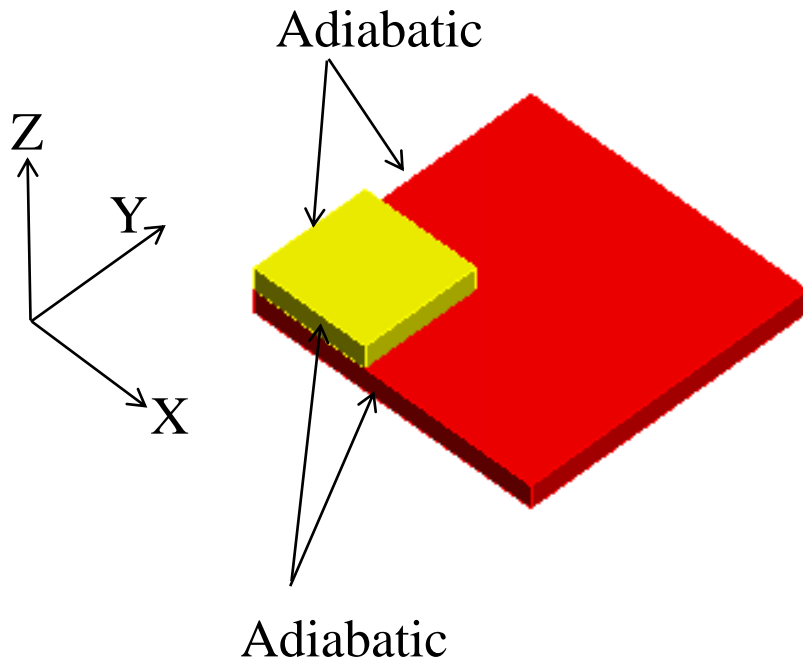
If [Reflective] is selected, appropriate boundary condition is automatically set. Results display in full model is possible.



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

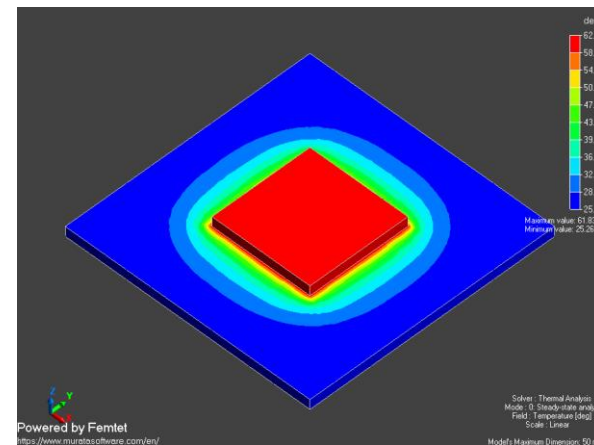
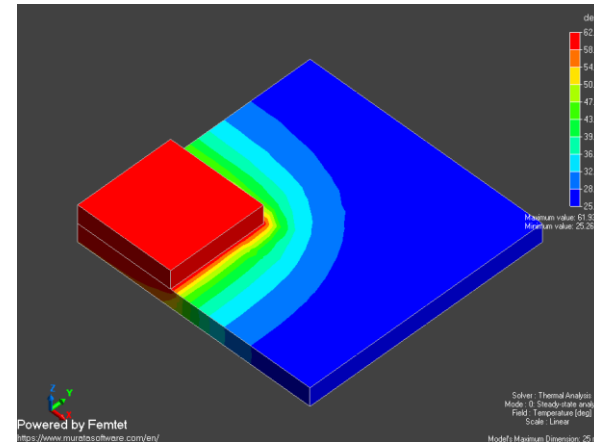
Internal Processing of Symmetric Boundary Condition

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



Note

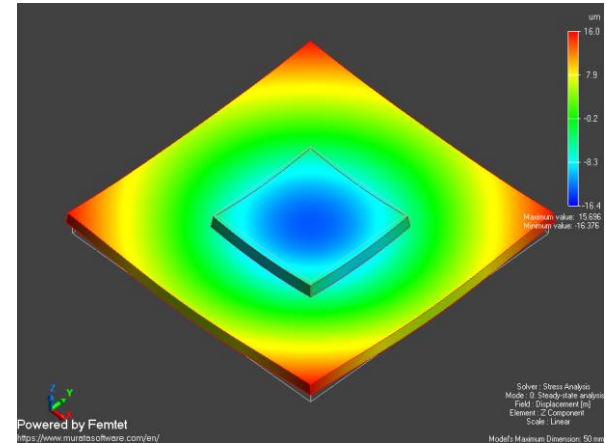
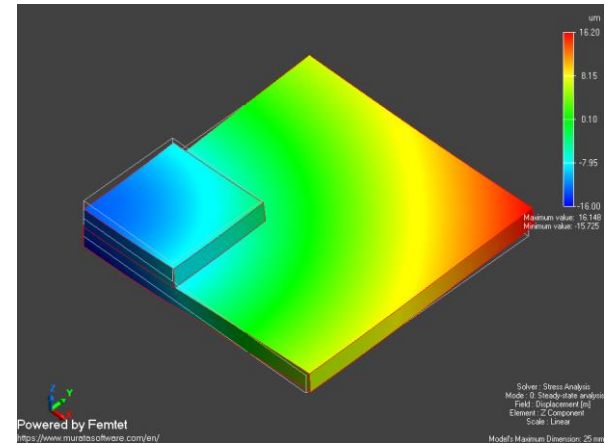
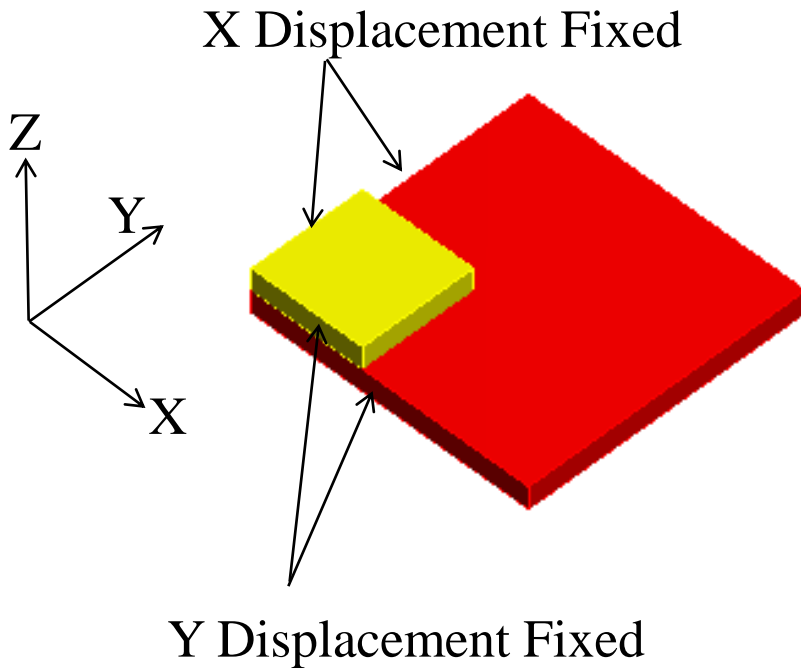
If you segment the heat source, the heat amount must be adjusted.



Results in Full Model

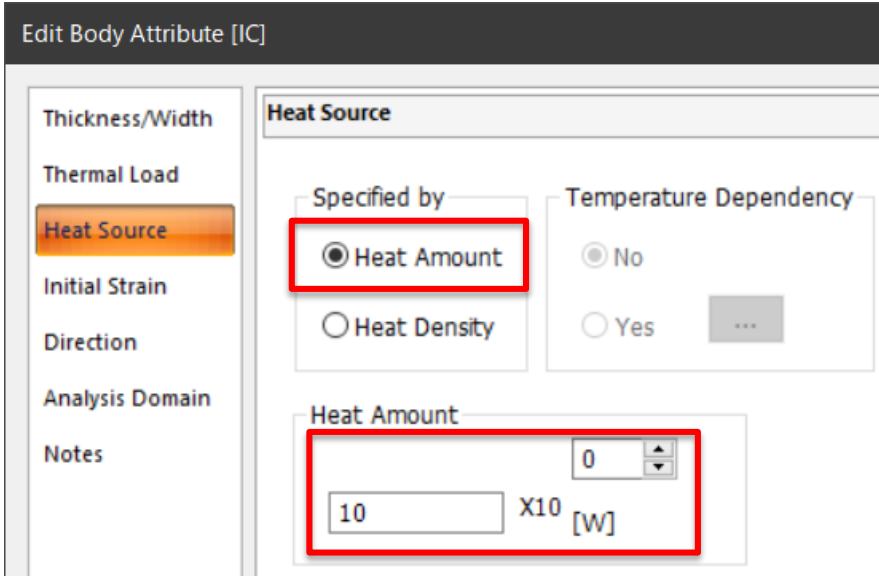
Internal Processing of Symmetric Boundary Condition

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



Results in Full Model

Input Values for Symmetric Model Murata Software



Edit Body Attribute [IC]

Thickness/Width

Thermal Load

Heat Source

Initial Strain

Direction

Analysis Domain

Notes

Heat Source

Specified by

Heat Amount

Heat Density

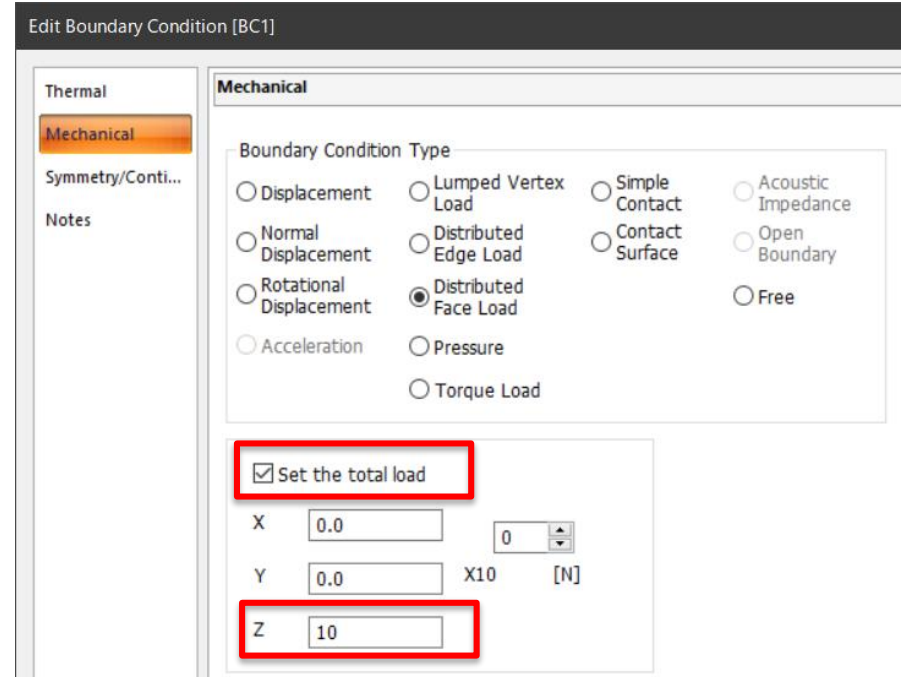
Temperature Dependency

No

Yes

Heat Amount

10 X10 [W]



Edit Boundary Condition [BC1]

Thermal

Mechanical

Symmetry/Conti...

Notes

Boundary Condition Type

Displacement

Normal Displacement

Rotational Displacement

Acceleration

Lumped Vertex Load

Distributed Edge Load

Distributed Face Load

Pressure

Torque Load

Simple Contact

Contact Surface

Acoustic Impedance

Open Boundary

Free

Set the total load

X 0.0

Y 0.0 X10 [N]

Z 10

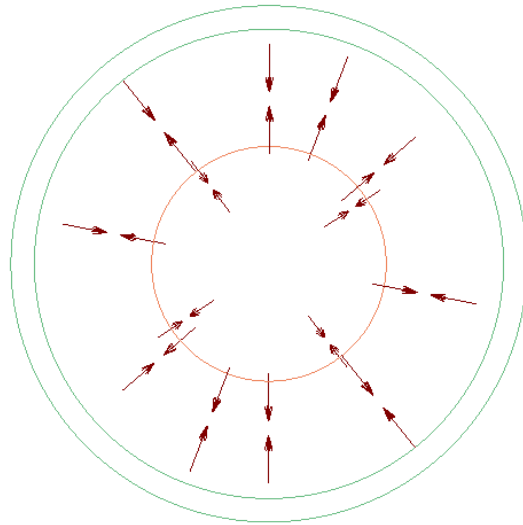
If the distributed load and the pressure are defined by the total load, their values must be adjusted depending on the type of symmetric model. The value of heat amount also needs to be adjusted.

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.

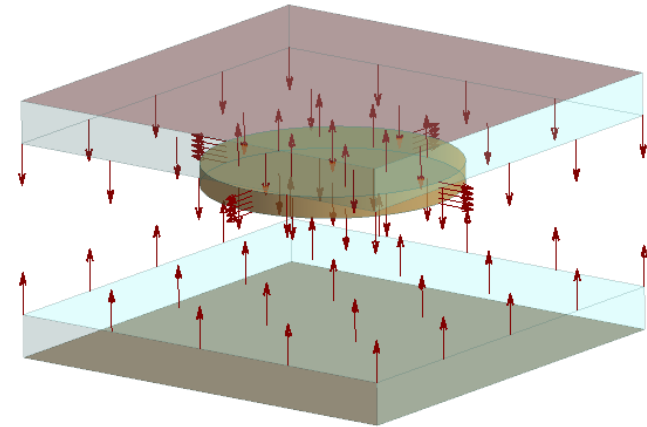
Accuracy of Body-to-Body Radiation in Symmetric Model



Powered by Fentet
<https://www.muratasoftware.com/>

全体寸法 : 0.132 m

Thermal Analysis Example 9
Body-to-Body Heat Radiation



Powered by Fentet
<https://www.muratasoftware.com/>

全体寸法 : 10 mm

Thermal Analysis Example 16
Radiation Blocked by Disc

Note:

Fentet version 2019 or before

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

Fentet 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

The screenshot displays the Murata Software FEMtet interface. The title bar reads "Femtet [New Project*] - [Model New Project:Analysis Model]". The menu bar includes "Model", "View", "Results", "Tools", and "Window". The ribbon contains several tool groups: "Drawing Plane" (XY Plane), "Primitives" (Solid), "Modification Operation" (Select Body, Move/Rotate, Single Copy, Unite, Subtract, Intersect), "Viewpoint Operation" (Change View), "Show", "Model", and "Analysis" (Analysis Condition, Run Mesher/Solver).

The Project Tree on the left shows a hierarchy for "New Project*":

- Analysis Model*
 - Model
 - Model unit: mm/3D
 - Analysis Condition:Thermal/Steady...
 - Body Attributes
 - Body Attribute Data
 - Materials
 - Material Data
 - Boundary Conditions
 - Outer_Boundary_Condition** (highlighted with a red box)
 - Boundary Condition Data
 - Mesh Sizes
 - General Mesh Size : Automatic c...
 - Variables
 - Results
 - Field
 - Coordinate System
 - Table

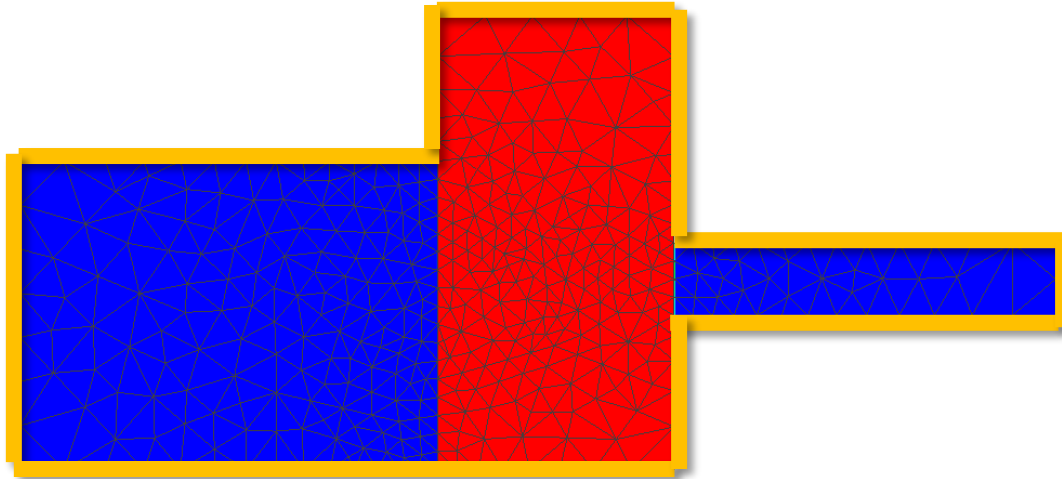
The "Edit Boundary Condition [Outer_Boundary_Condition]" dialog is open, showing the "Thermal" tab. The "Boundary Condition Type" section includes:

- Temperature
- Heat Flux
- Thermal Resistance
- Transfer/Ambient Radiation
- Body-to-Body Radiation
- Measuring Terminal
- Adiabatic (no setting)

 Additional options include:

- Time Dependency
- Use distribution data
- Use distribution data
- Uniform Temperature

 Buttons for "Weight Function", "Distribution Data", and "Distribution Data" are visible on the right. A red arrow points from the "Outer_Boundary_Condition" in the Project Tree to the dialog title bar.

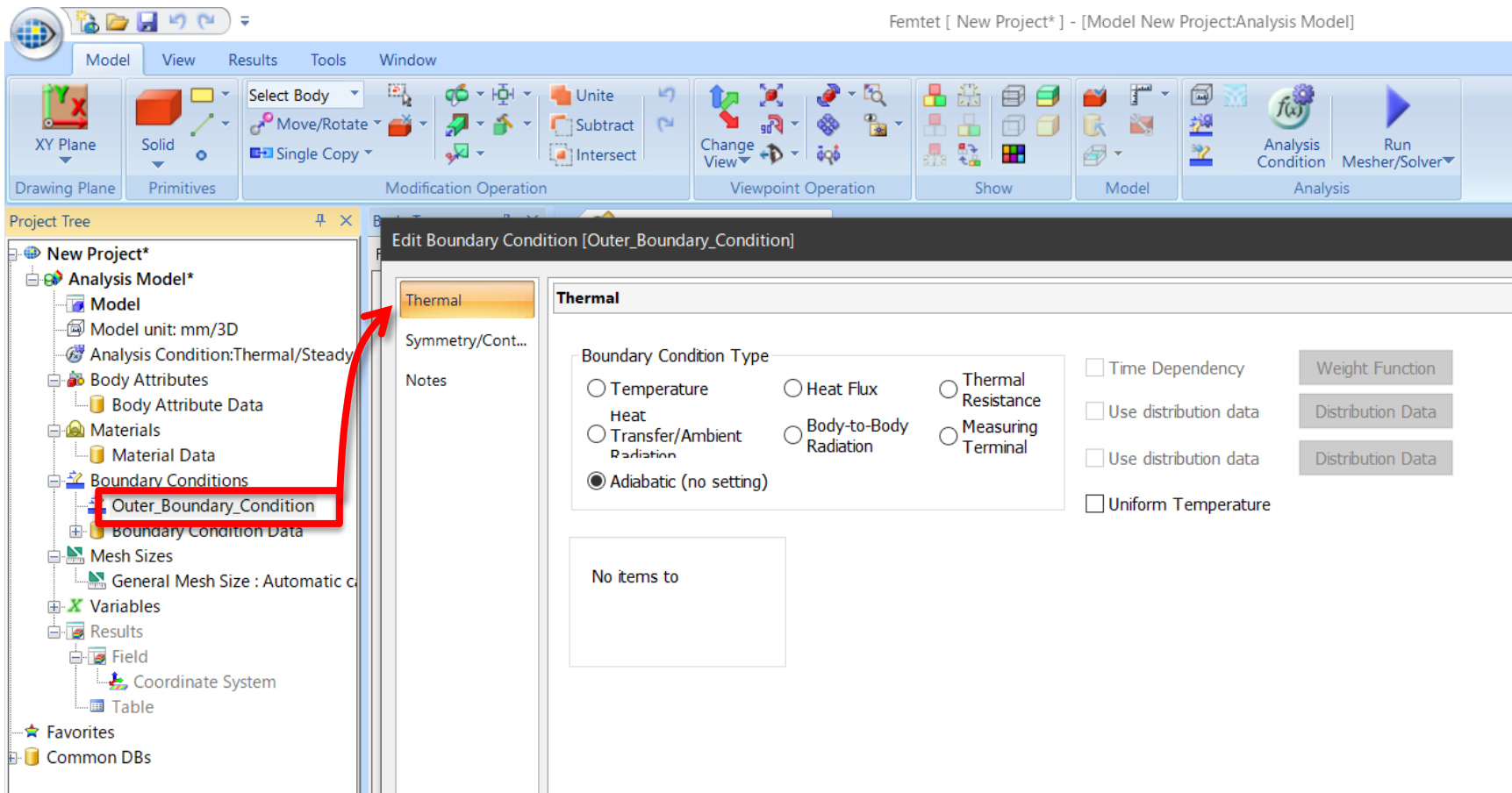


With the finite element method, the outer boundary condition must be set to all faces on the outer perimeter of a model.

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.

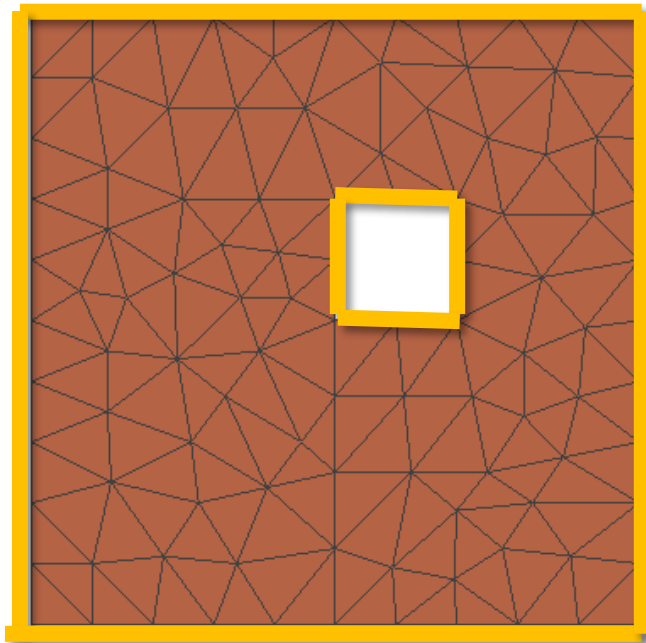
For example, radiating boundary can be set with simple setting.




Notes on Outer Boundary Condition Murata Software

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.



 Edges where the outer boundary condition is given

6. Linear/Nonlinear Analysis

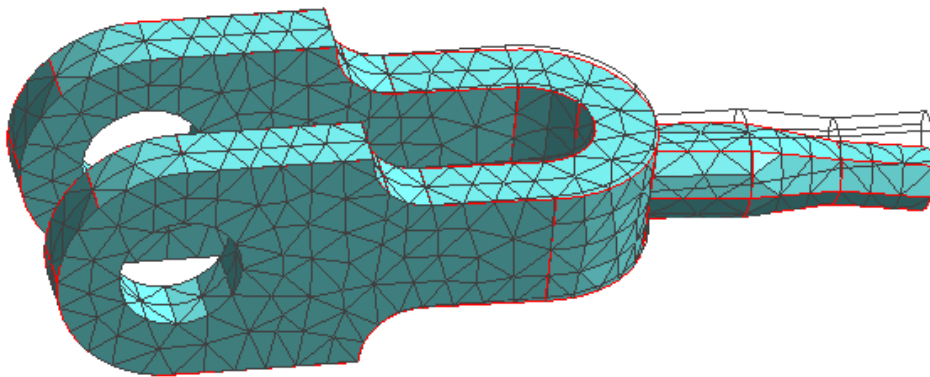
- Linear and Nonlinear Analysis
- Nonlinear Analysis in Stress Analysis
- Nonlinear Analysis in Thermal Analysis

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.

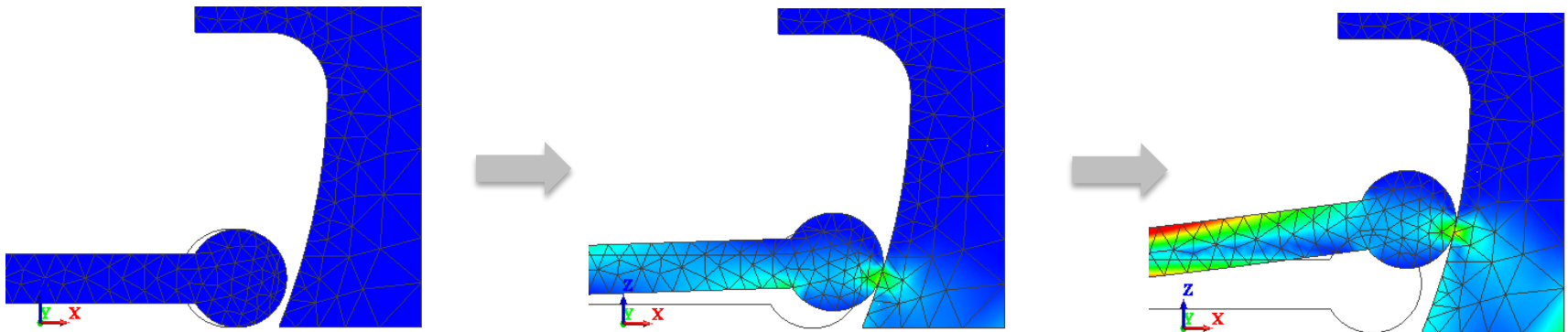
Nonlinear analysis takes long calculation time, and may not converge in some cases.



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

A form changes gradually.

Calculations are repeated by changing the form bit by bit.



Analysis Condition Setting

Solver

Stress Analysis

Mesh

Resonant Analy...

Harmonic Analy...

Transient Analysis

Step/Thermal L...

Acceleration

Angular Velocity

Constant Temp...

High-Level Setti...

Result Import

Notes

Stress Analysis

Analysis Type

- Static Analysis
- Resonant Analysis
- Harmonic Analysis
- Buckling Analysis
- Transient Analysis

2D Approximation

- Planar Strain
- Plane Stress

Large Deformation

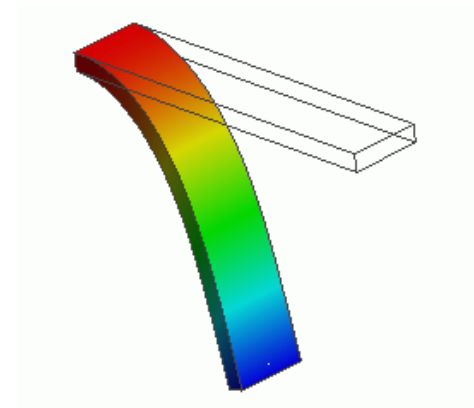
- Large Displacement
- Large Strain

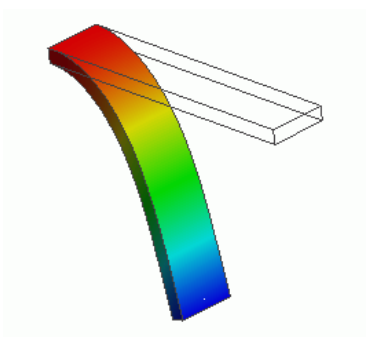
Options

- Acceleration
- Thermal load
- Angular Velocity
- Initial Stress (Result Import)
- Constrain the freedom of shells
- Calculate the mass and the moment of inertia.

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.

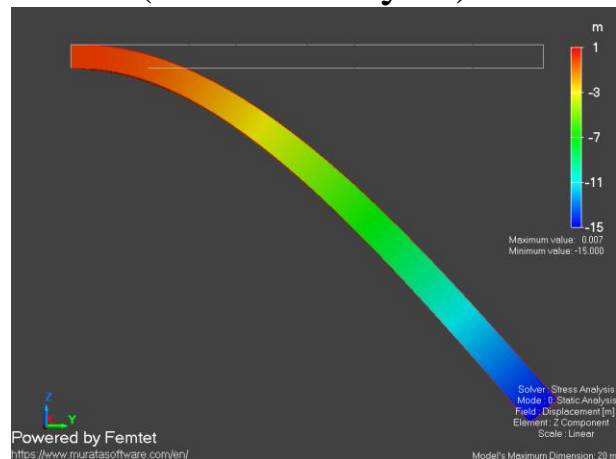




Large Displacement

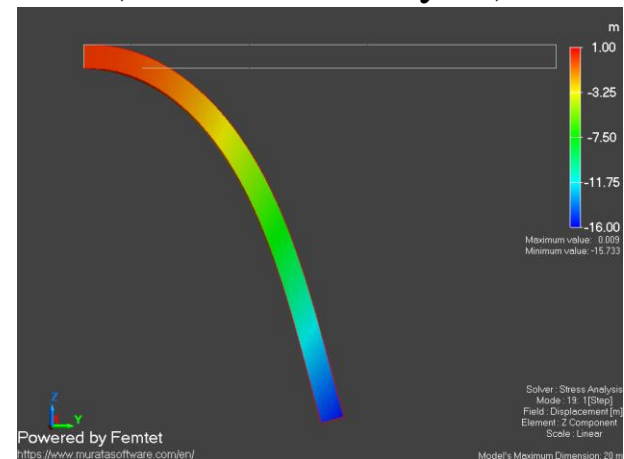
In the analysis, curling is taken into account, which is not calculated in the minute-displacement analysis.
(Example 6)

Large Displacement Set OFF (Linear Analysis)

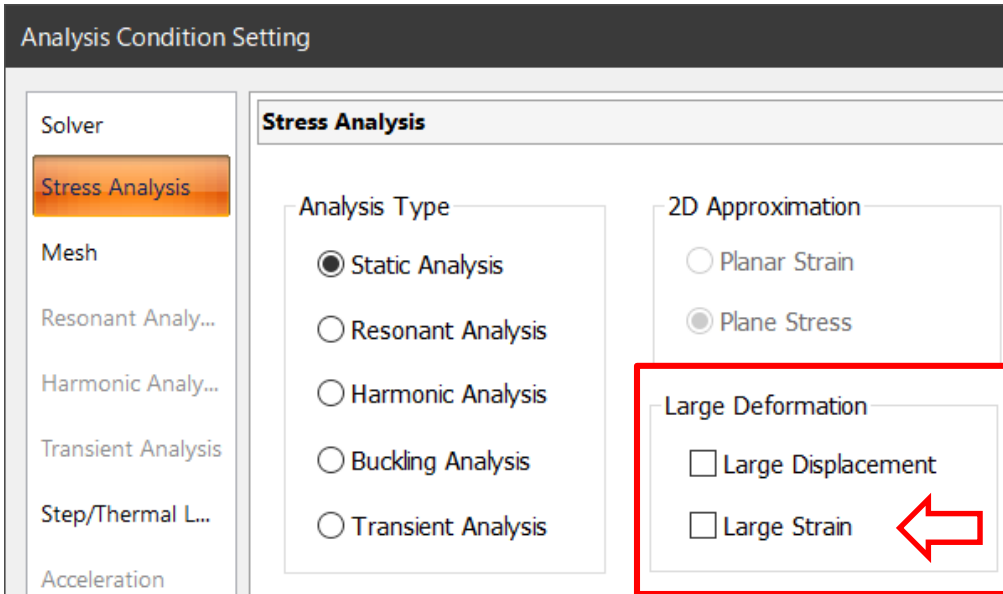


The results are only magnified version of the minute-displacement analysis results.

Large Displacement Set ON (Nonlinear Analysis)

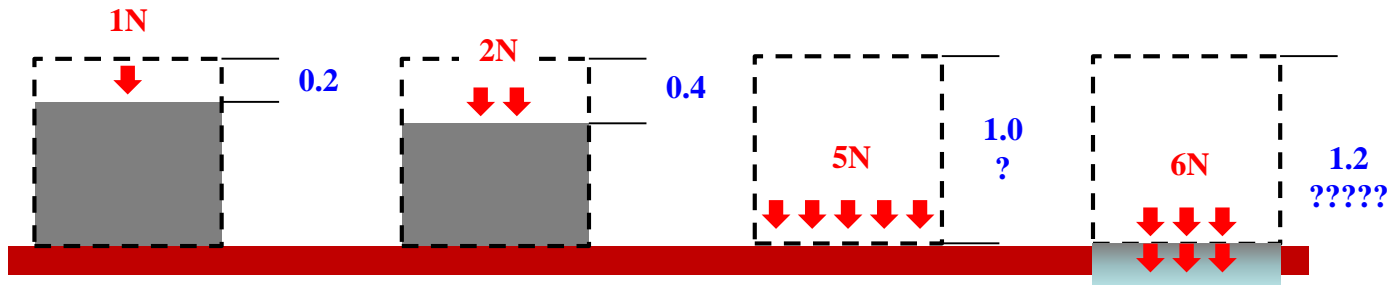


Calculations are repeated by changing the analysis model form bit by bit.



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.

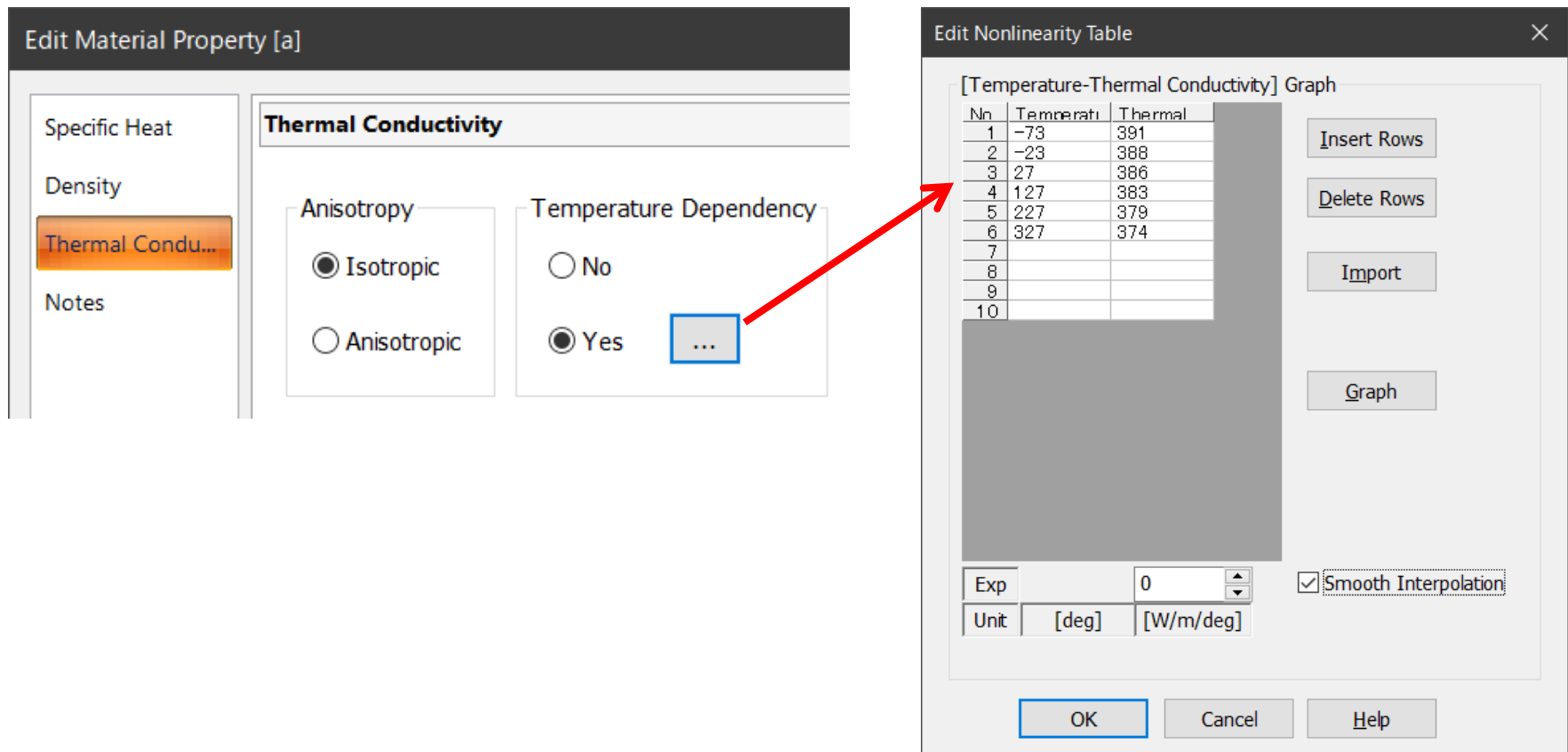


Calculations are repeated by “recreating an analysis model bit by bit with strain”
The equation is modified as well.

Nonlinear Thermal Analysis

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.



The image shows two overlapping dialog boxes from a software application. The left dialog, titled "Edit Material Property [a]", has a sidebar with "Thermal Condu..." selected. The "Thermal Conductivity" section has "Temperature Dependency" set to "Yes". A red arrow points from the "..." button next to "Yes" to the "Edit Nonlinearity Table" dialog.

The "Edit Nonlinearity Table" dialog shows a table with the following data:

Nr	Temperatu	Thermal
1	-73	391
2	-23	388
3	27	386
4	127	383
5	227	379
6	327	374
7		
8		
9		
10		

Below the table, there are input fields for "Exp" (0) and "Unit" ([deg] [W/m/deg]). A "Smooth Interpolation" checkbox is checked. Buttons for "Insert Rows", "Delete Rows", "Import", "Graph", "OK", "Cancel", and "Help" are also visible.

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:

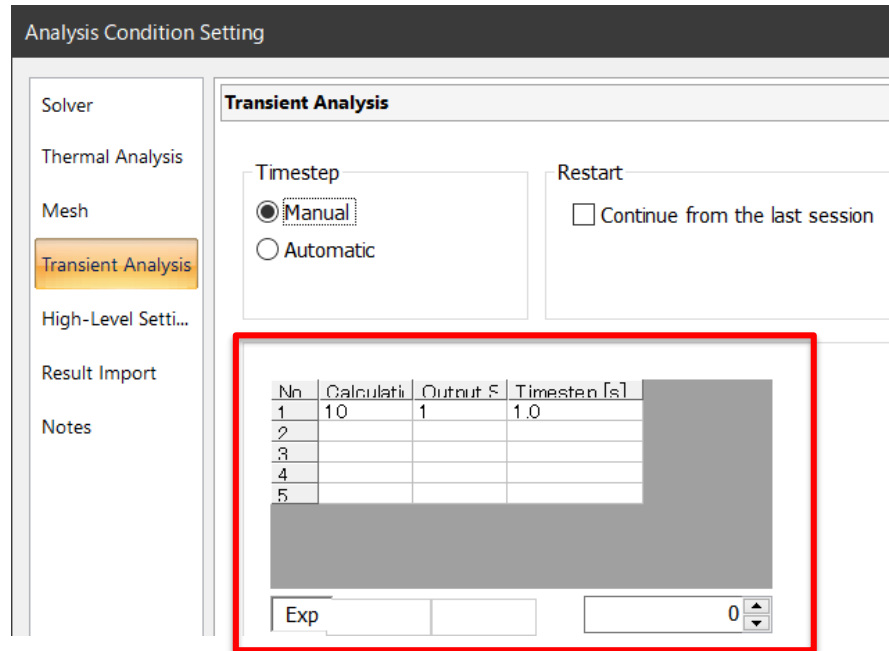
As nonlinear analysis takes long time for calculation, it is recommended that you carefully select the nonlinear elements for analysis.

7. Transient Analysis

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

Transient Analysis Tab

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

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).

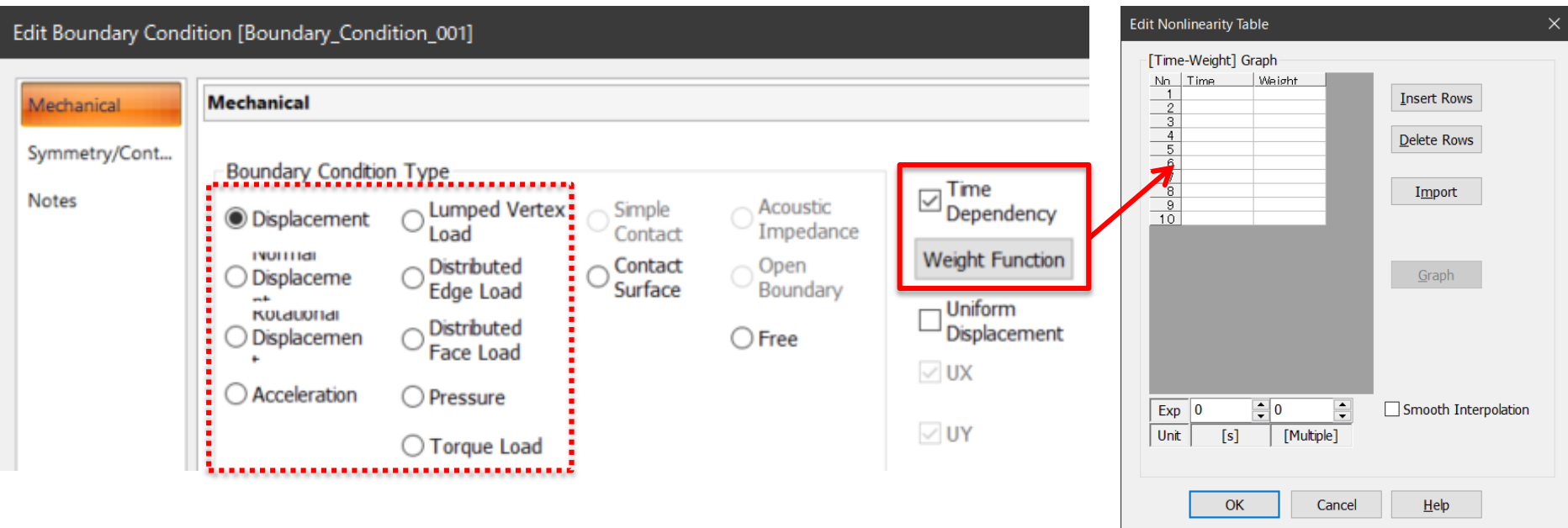
The image displays three software windows illustrating time dependency settings:

- Edit Body Attribute [Body_Attribute_001]:** Shows the 'Heat Source' section. The 'Time Dependency' checkbox is checked and highlighted with a red box. Below it is a 'Weight Function' button.
- Edit Boundary Condition [Boundary_Condition_003]:** Shows the 'Thermal' section. The 'Time Dependency' checkbox is checked and highlighted with a red box. Below it is a 'Weight Function' button.
- Edit Nonlinearity Table:** A dialog for defining a 'Time-Weight' graph. It contains a table with columns 'No.', 'Time', and 'Weight'. The table has 10 rows. To the right of the table are buttons for 'Insert Rows', 'Delete Rows', 'Import', and 'Graph'. Below the table are input fields for 'Exp' (set to 0) and 'Unit' (set to [s]).

No.	Time	Weight
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		

Boundary Condition Changing over Time Murata Software

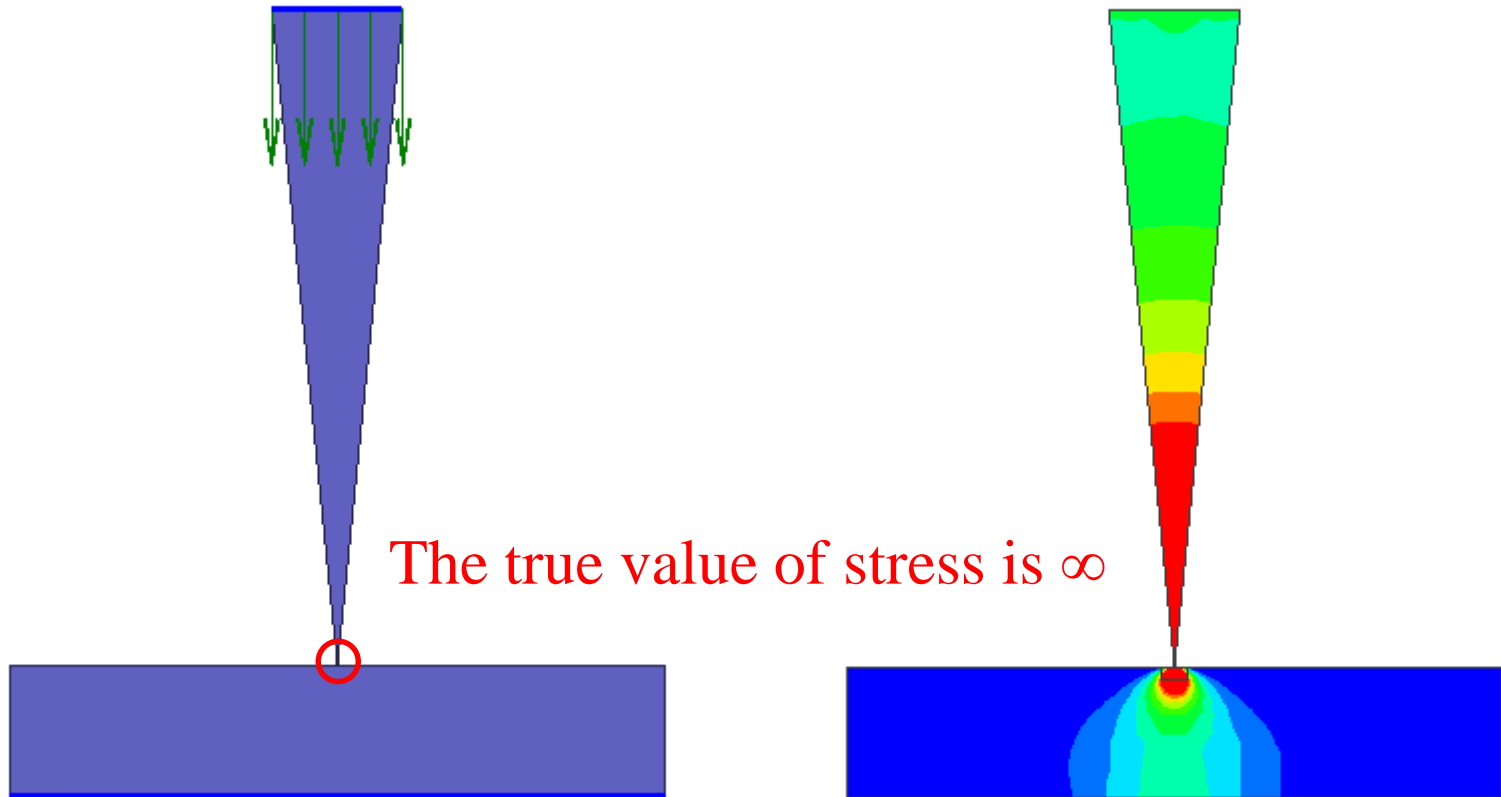
In the transient stress analysis, time dependency can be given to the mechanical boundary conditions of displacement, load, and pressure.



The image shows two overlapping software windows. The background window is titled "Edit Boundary Condition [Boundary_Condition_001]" and has a "Mechanical" tab selected. Under "Boundary Condition Type", the "Displacement" option is selected and highlighted with a red dashed box. To its right, the "Time Dependency" checkbox is checked and also highlighted with a red box. Below it, the "Weight Function" button is visible. The foreground window is titled "Edit Nonlinearity Table" and contains a table with columns "No.", "Time", and "Weight". The table has 10 rows, with the first row containing the values 1, 0, and 0. To the right of the table are buttons for "Insert Rows", "Delete Rows", "Import", and "Graph". At the bottom of this window are "OK", "Cancel", and "Help" buttons. A red arrow points from the "Weight Function" button in the background window to the table in the foreground window.

No.	Time	Weight
1	0	0
2		
3		
4		
5		
6		
7		
8		
9		
10		

8. Singularity



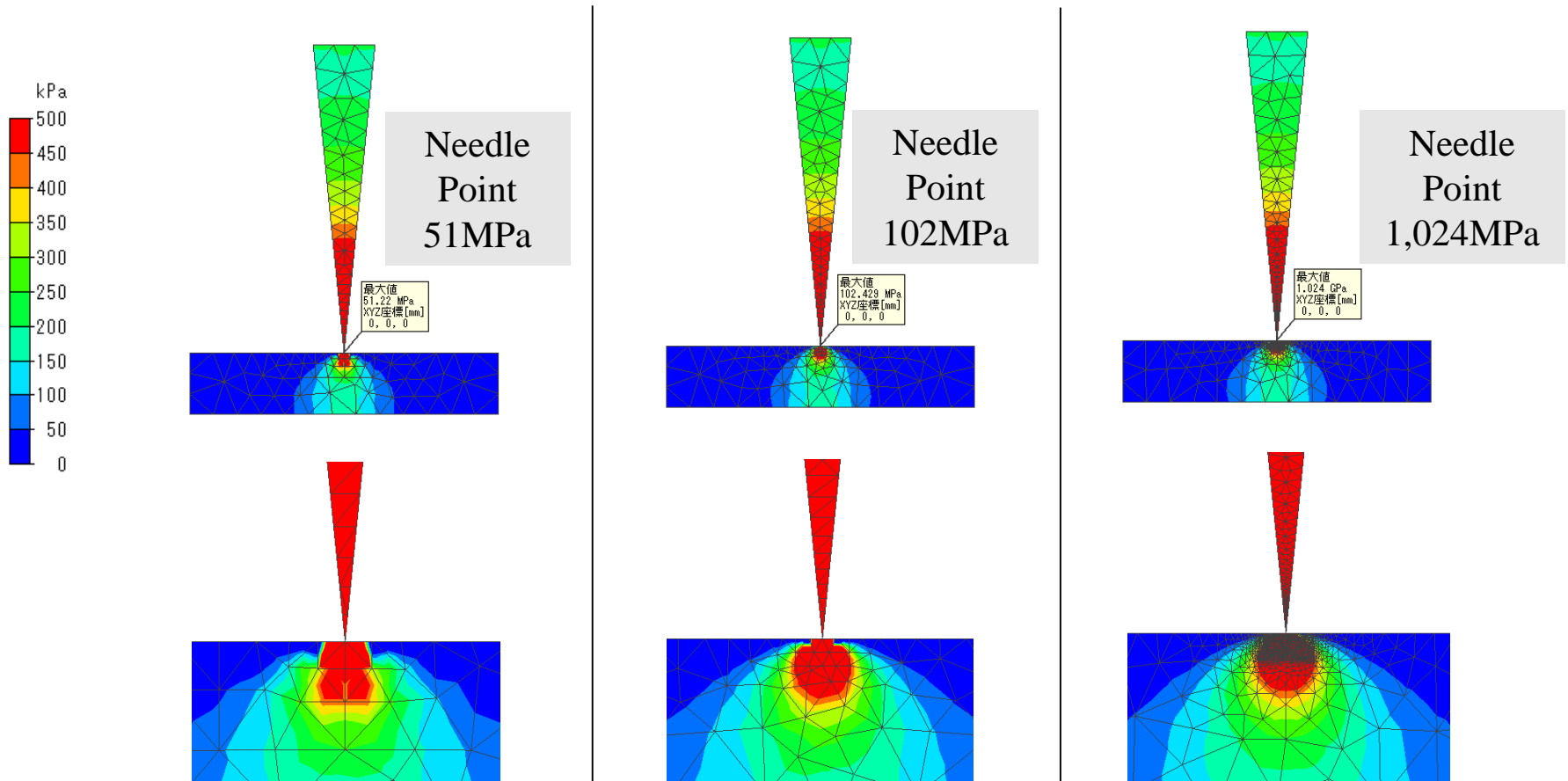
Singularity

Stress (Hydrostatic Pressure)

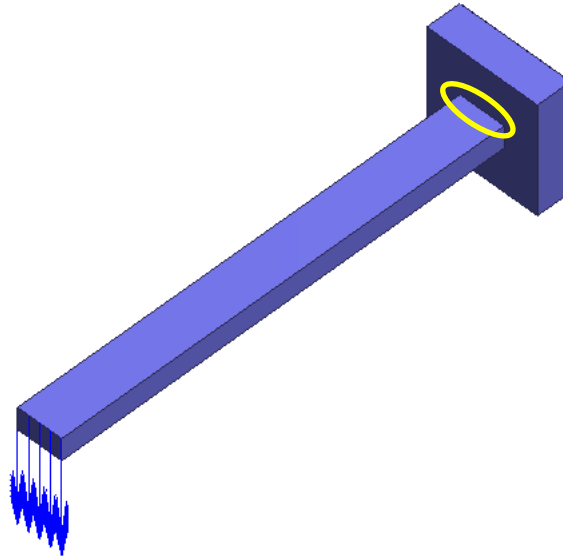
Stress singularity does not converge even if the meshes are fine.

It approaches to the true value of ∞ .

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



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



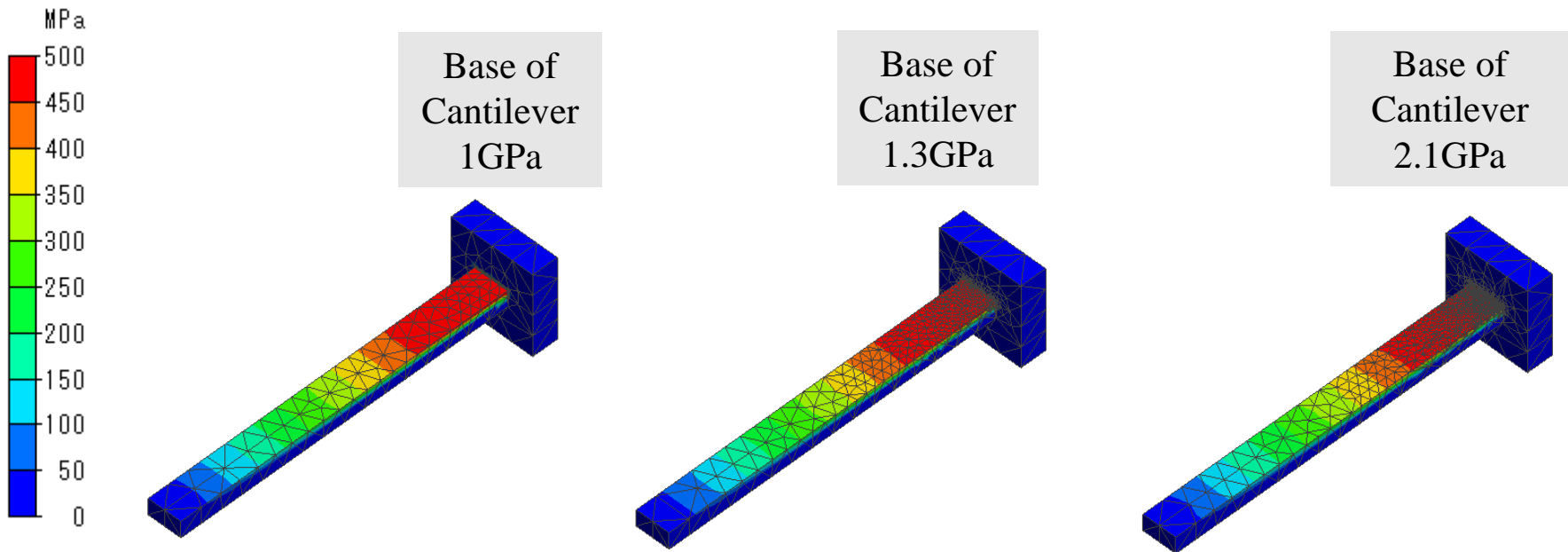
Singularity is not uncommon.

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.

Stress (Maximum Principal Stress)

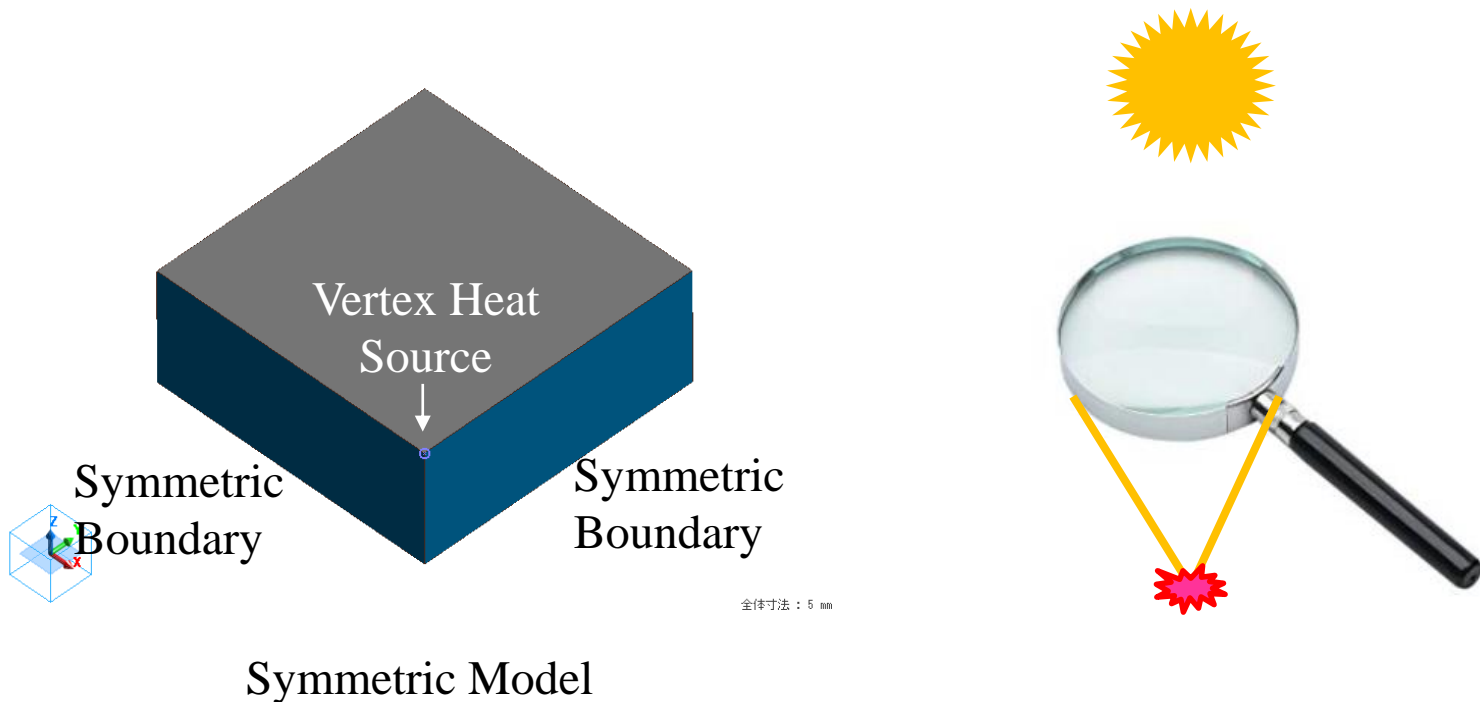
The singularity stress does not converge even if the meshes are fine.
It approaches to the true value of ∞ .

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

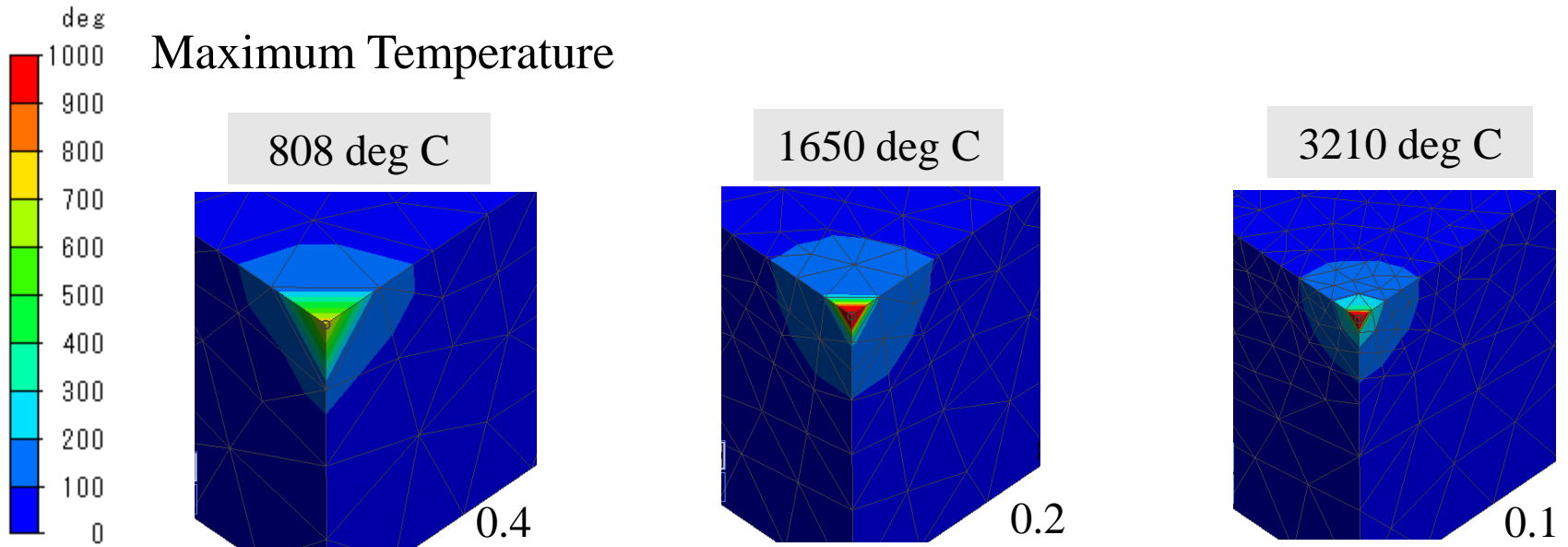


Singularity in Thermal Conduction Murata Software

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.

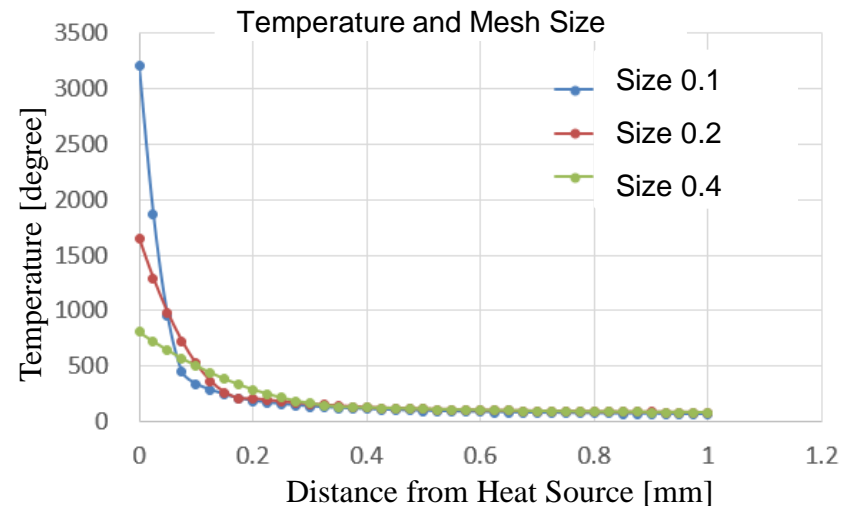


Singularity in Thermal Conduction Murata Software



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.