

Femtet

- Operations Exercise -

202009

- Basic Operations of Modeler
 - Primitives
 - Snap
 - Viewpoint
 - Object Selection
 - Model Modification


- Exercise with Stress Analysis
 - Modeling
 - Setup of Body Attribute, Material Property, and Boundary Condition
 - Run Solver
 - Results Confirmation

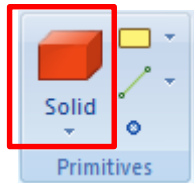
Basic Operations of Modeler Murata Software

Primitives

From ribbon menu, select primitives.

Pressing ▼ on the right side or below the menu, submenu will appear.

Example: [Model] tab — [Primitives] — [Solid] 



Press Solid

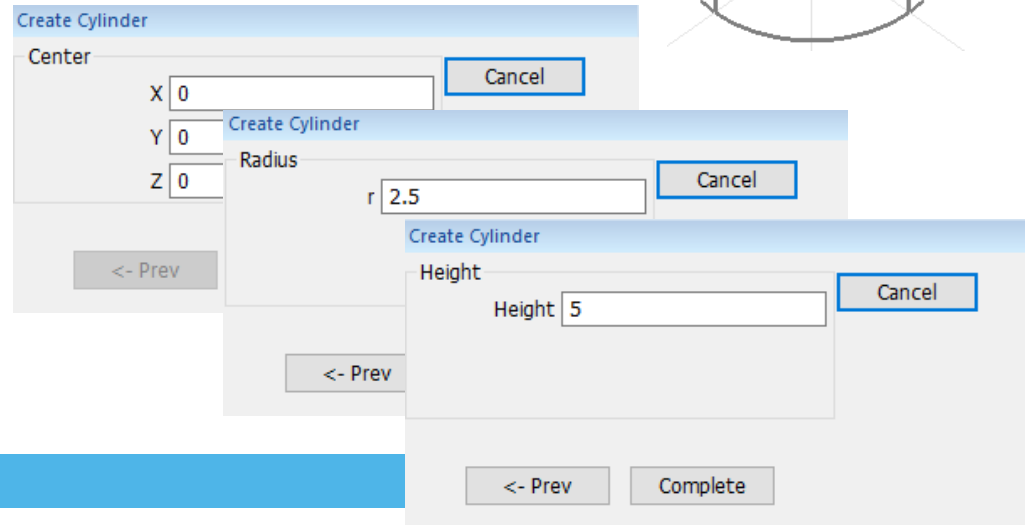
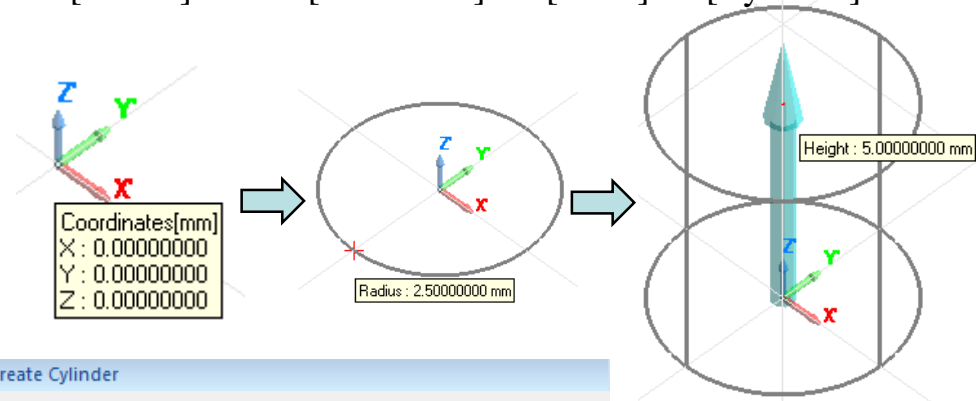


Submenu

Enter parameters of primitives on a Modeling window or in the input fields.

Example

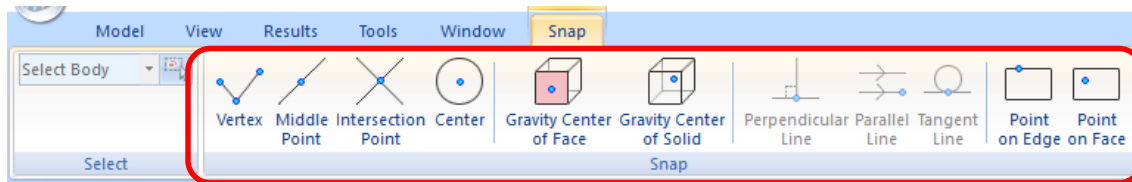
[Model] tab => [Primitives] => [Solid] => [Cylinder]



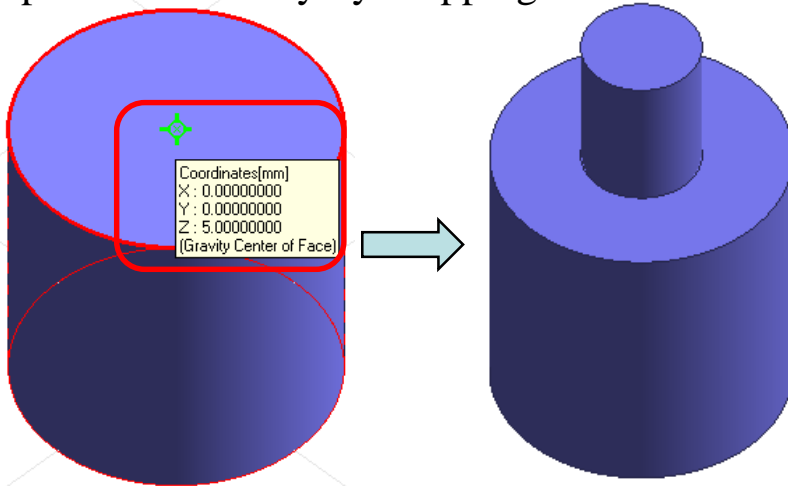
Basic Operations of Modeler Murata Software

Snap

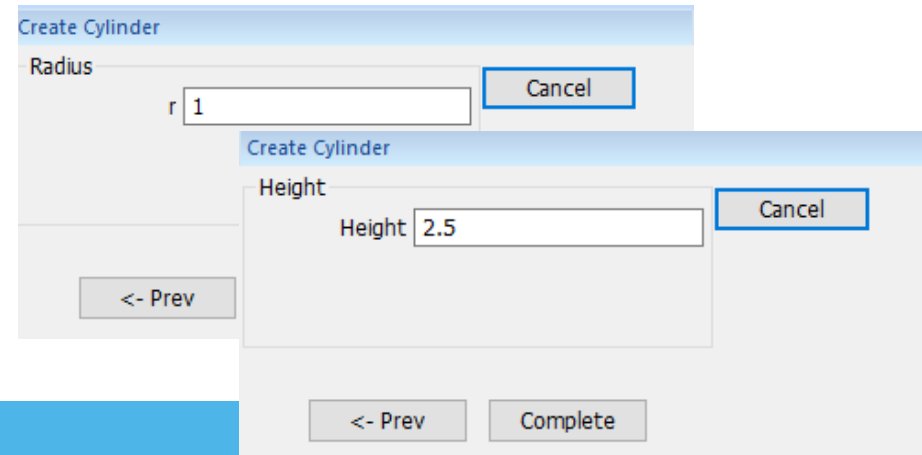
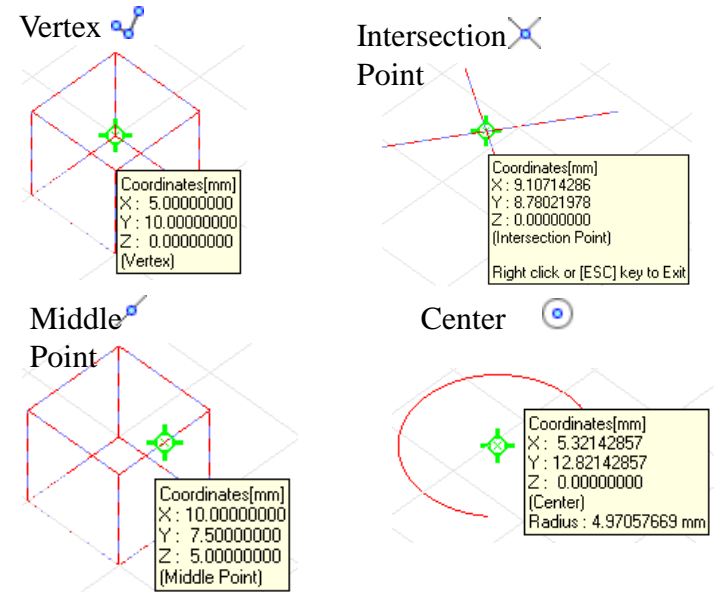
When entering parameters, enable the type of coordinates to snap.



Bring mouse near to the enabled coordinates, the parameters are input automatically by snapping the coordinates.

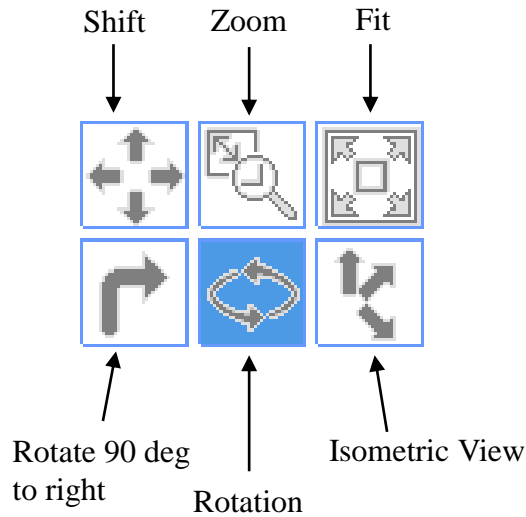


スナップ機能で拾える座標例

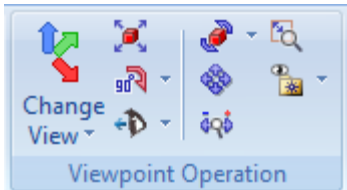


Viewpoint Operation

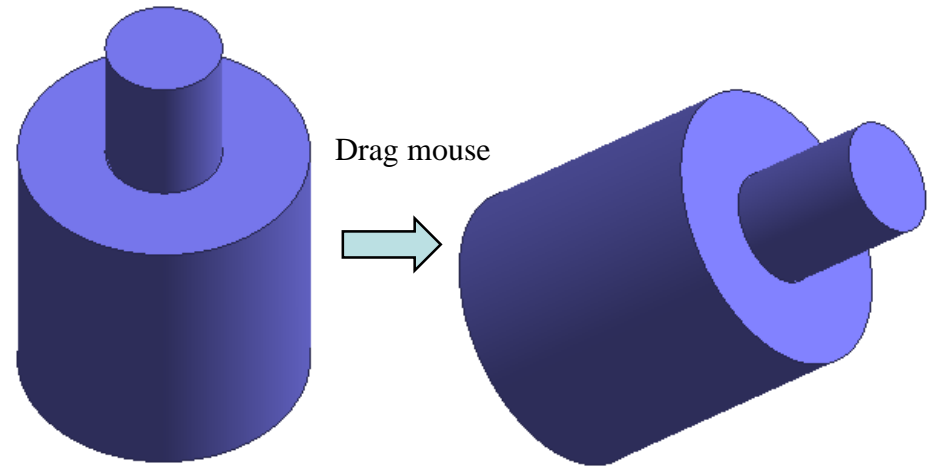
Viewpoint operation is enabled on the tool panel on a Modeling window or [Model] tab => [Viewpoint Operation].



[Model] tab => [Viewpoint Operation]



Drag a model while pressing left button of a mouse to get a viewpoint of your wish.



*Some operations, such as [Fit] and [Isometric View], can be executed simply by selecting the menu.

Basic Operations of Modeler Murata Software

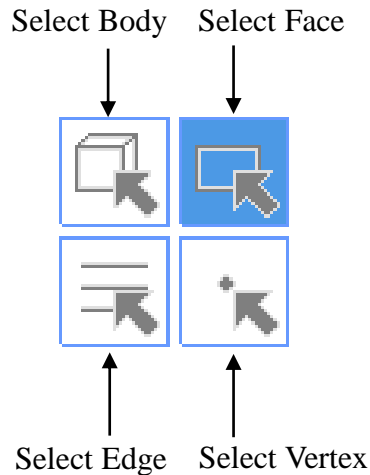
Object Selection

Select a body or topology (vertex, edge, face) from the tool panel on a Modeling window or [Model] tab => [Modification Operation]

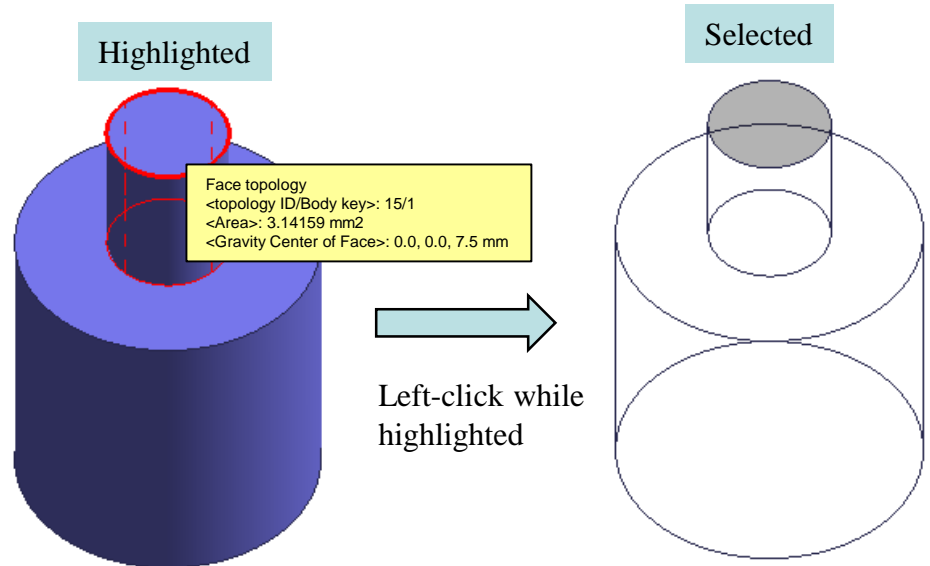
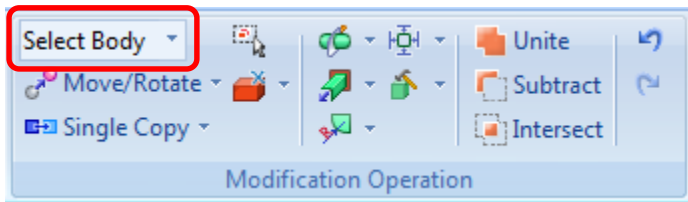
*A body consists of topologies (vertex, edge, face).

The target object will be highlighted when a mouse is brought to it.

Left-click it to complete the selection.



[Model] tab => [Modification Operation]



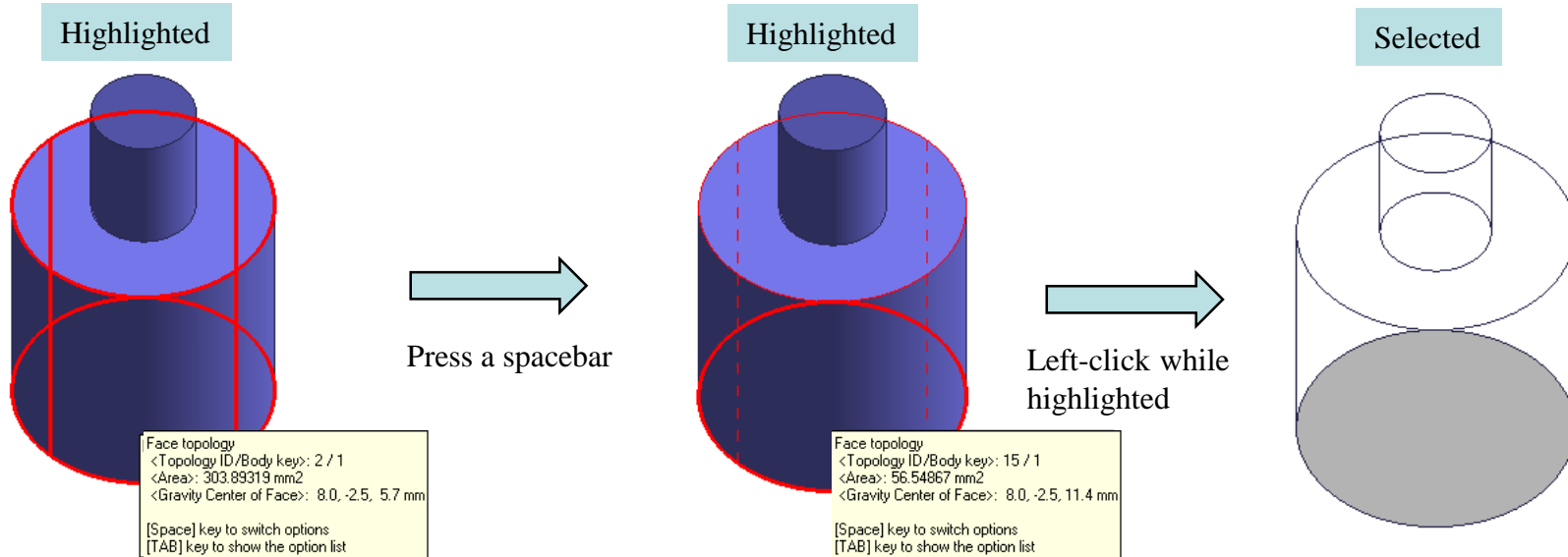
*Be aware that left-click is needed to complete the selection.

**Left-click + [Ctrl] key will allow multiple selection.

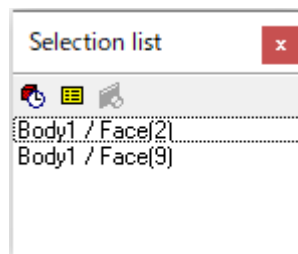
Change Target Object

Bring mouse to the place to select.
Stay there for a second.

Press a spacebar.
A target object can be switched.



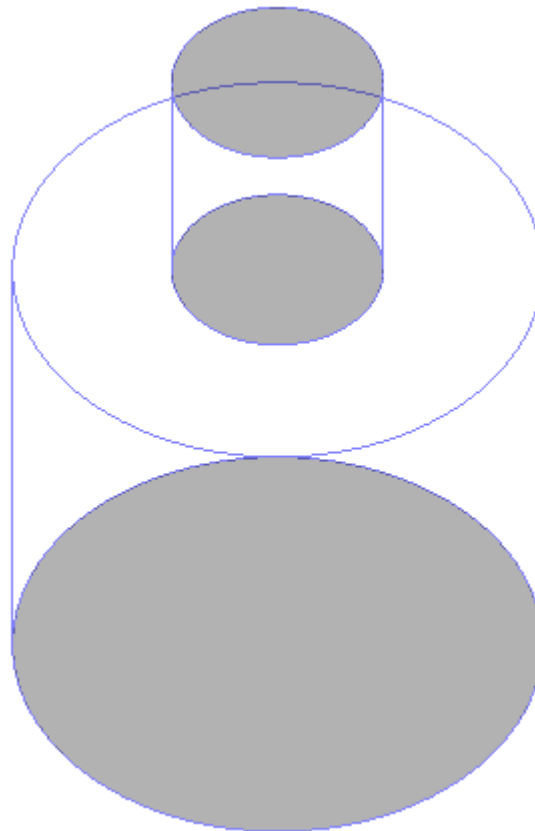
The function is useful when you want to select a place where bodies that are overlapping, or hidden faces.



*Press [Tab] key to show Selection list.

Check Your Understanding

You've learnt Object Selection, Change Target Object, and Multiple Selection. Based on what you've learnt, select three faces below.

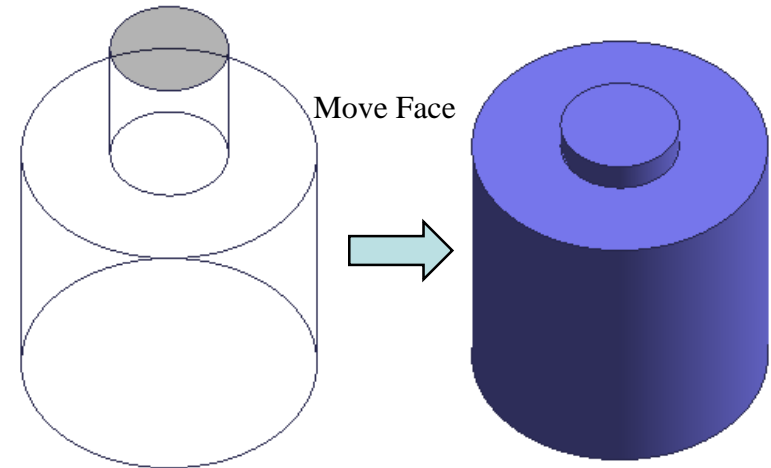
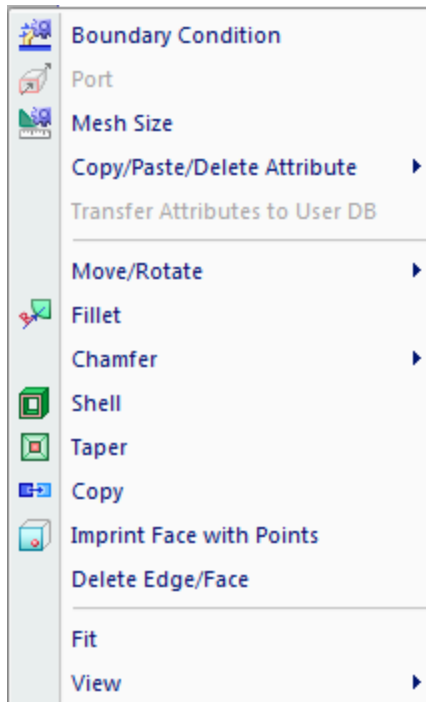


Model Modification

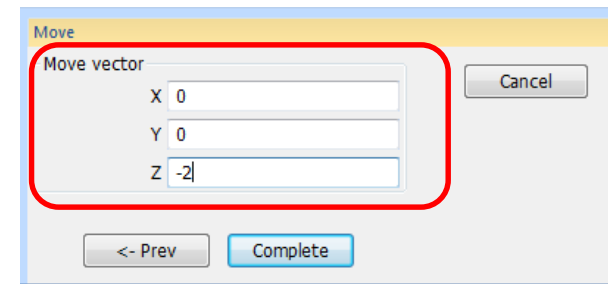
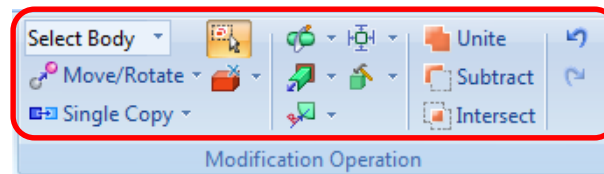
Right-click on a modification target (body or part of body).
Select operation from an appearing menu or
go to [Model] tab => [Modification Operation].

Enter parameters in an input box.

Right-click menu (on a selected face)

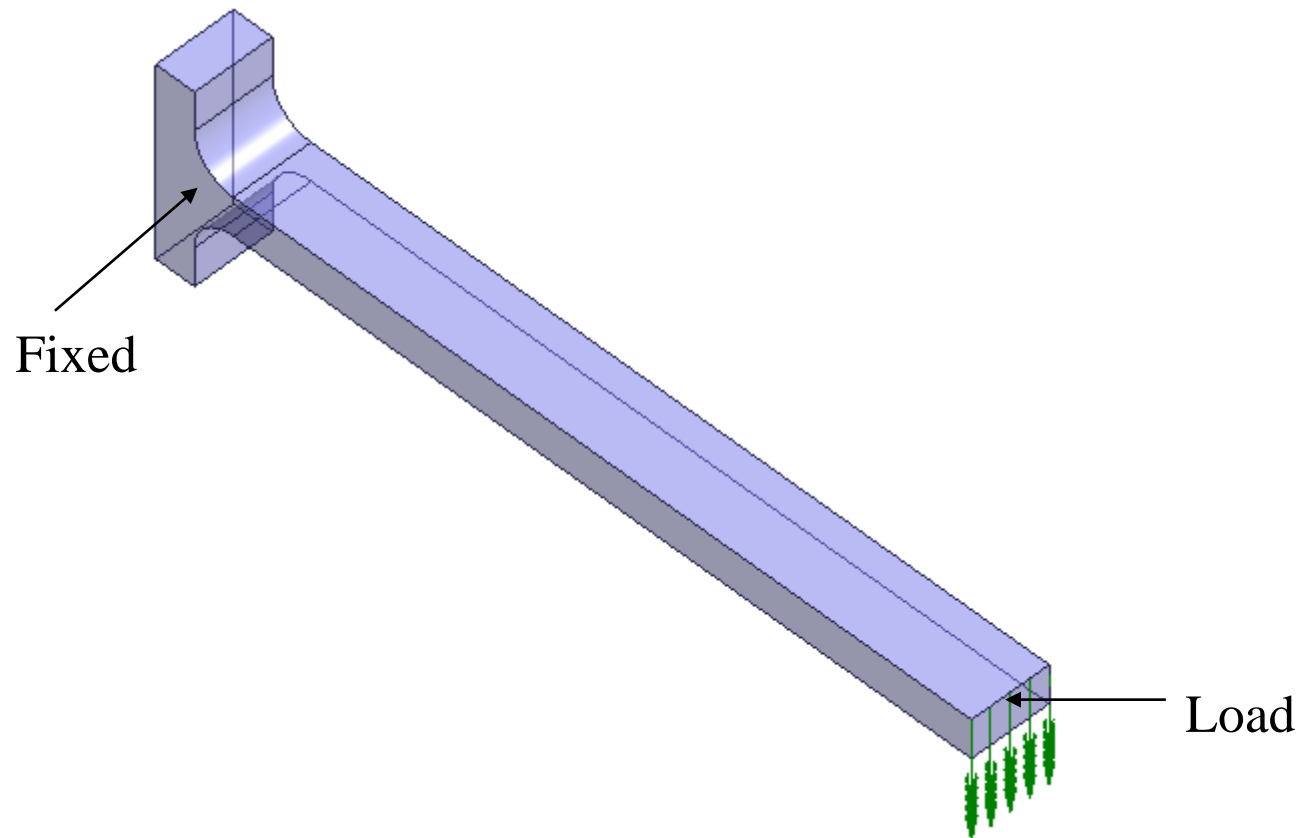


[Model] tab => [Modification Operation]



Analysis Model

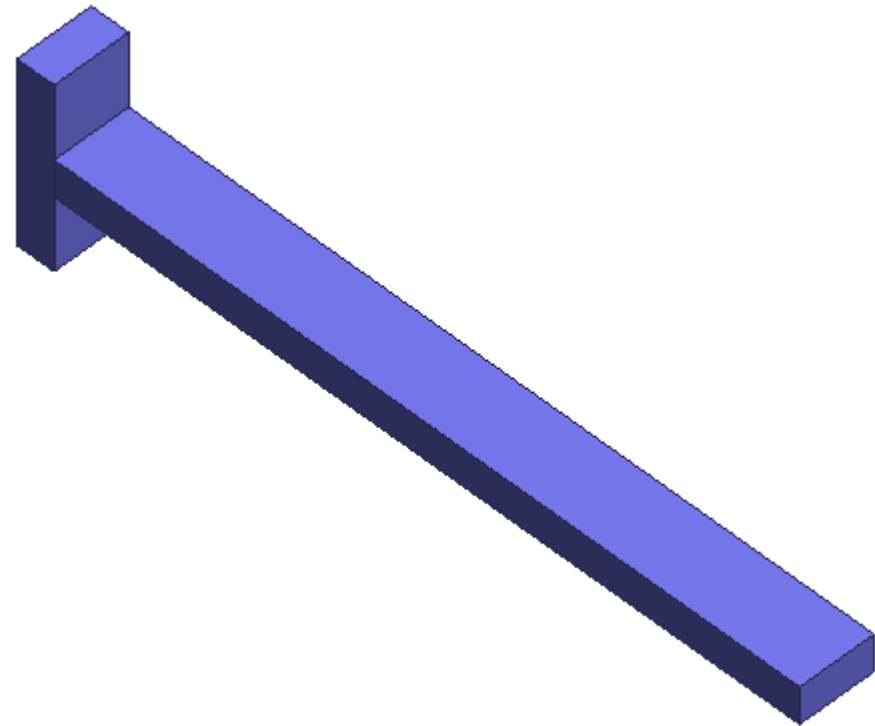
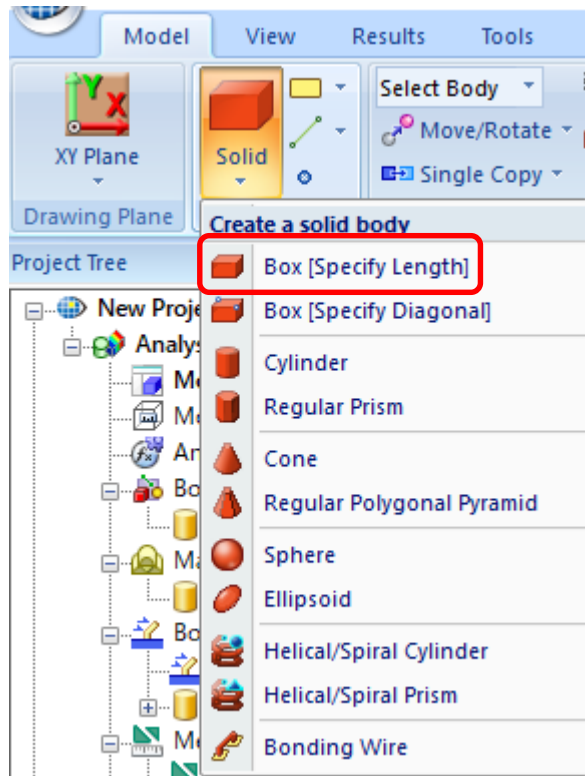
Displacement and stress are analyzed when a beam is applied load on its end.



Modeling

[Model] > [Primitives] > [Box [Specify Length]]

Enter Startpoint [0, 0, 0], Width[20], Depth [2], and Height [1].

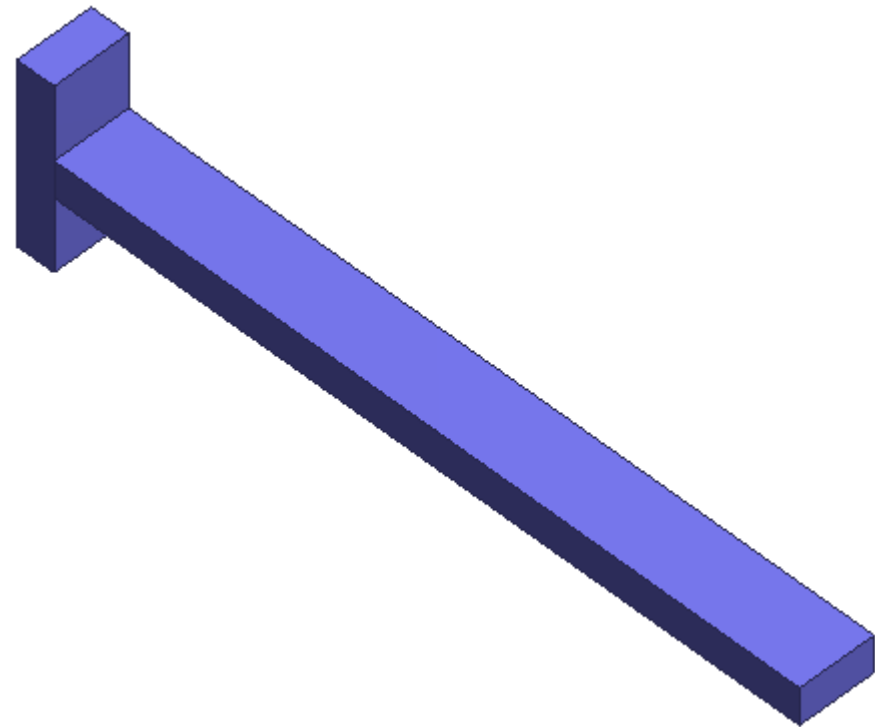
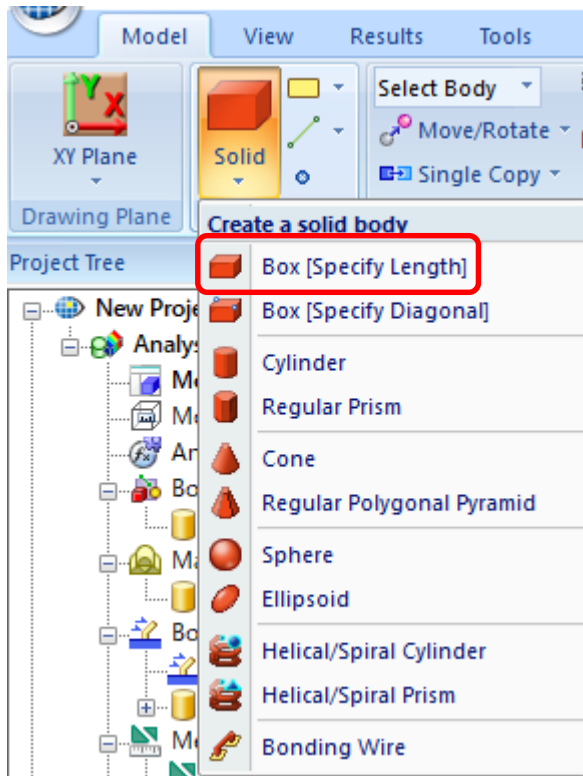


Exercise - Stress Analysis

Modeling

[Model] > [Primitives] > [Box [Specify Length]]

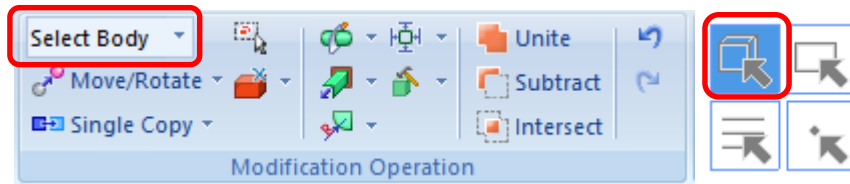
Enter Startpoint [0, 0, -2], Width[-1], Depth [2], and Height [5].



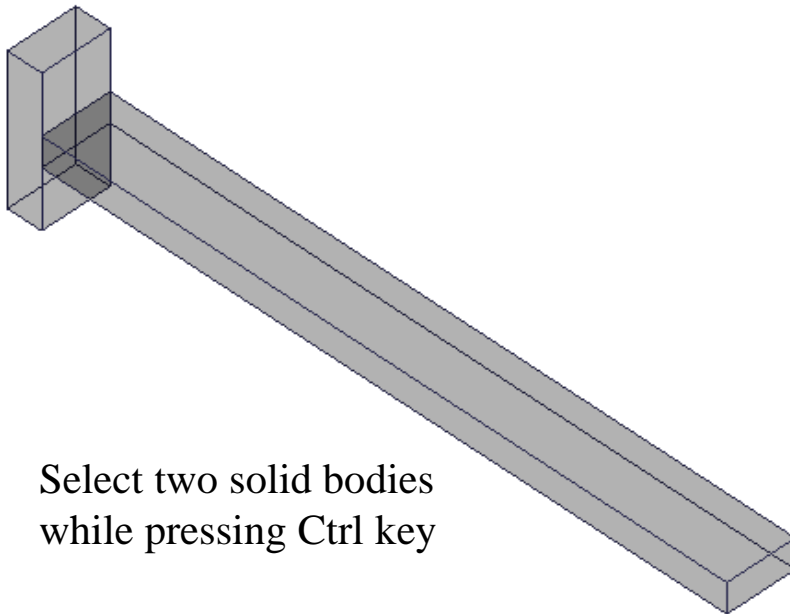
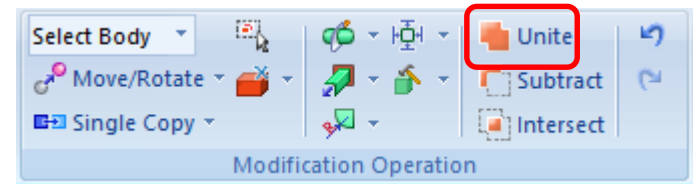
Exercise - Stress Analysis

Modeling

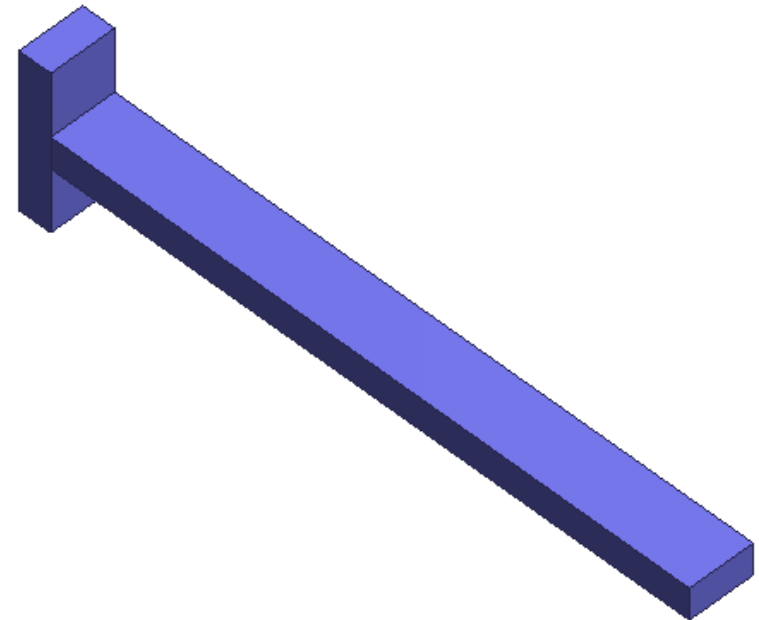
Switch target to Body and select two bodies



[Model] > [Modification Operation] > [Unite]



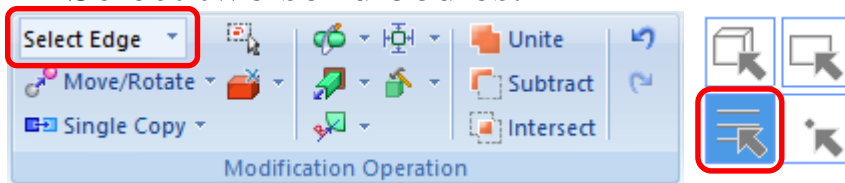
Select two solid bodies
while pressing Ctrl key



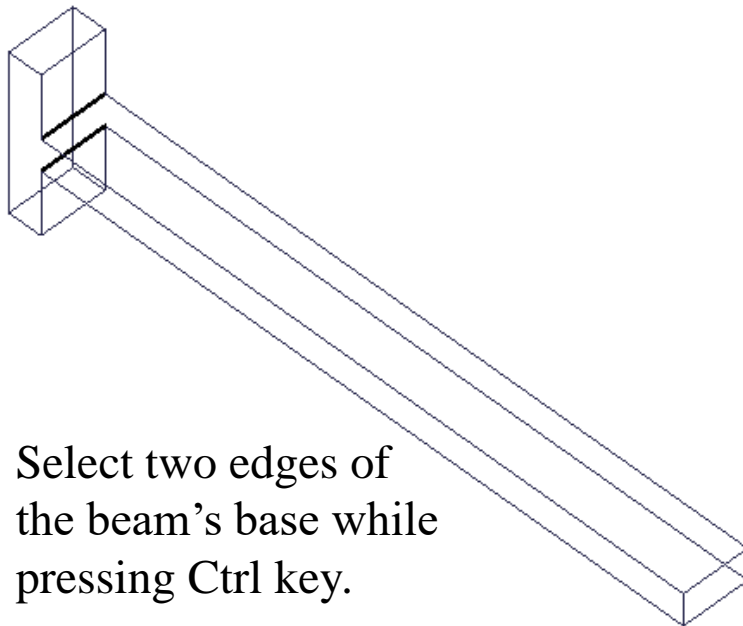
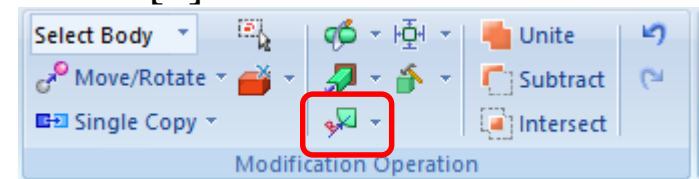
Exercise - Stress Analysis

Modeling

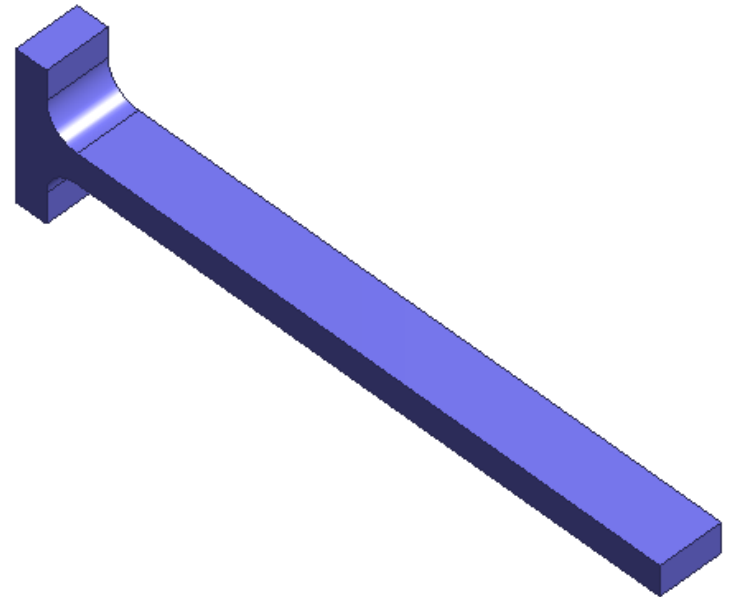
Switch target to Edge.
Select two solid bodies.



[Model] > [Modification Operation] > [Fillet]
Enter radius [1]



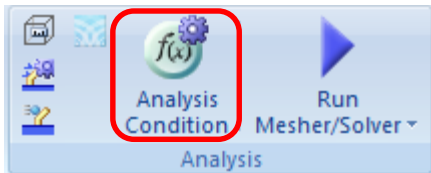
Select two edges of
the beam's base while
pressing Ctrl key.



Exercise - Stress Analysis

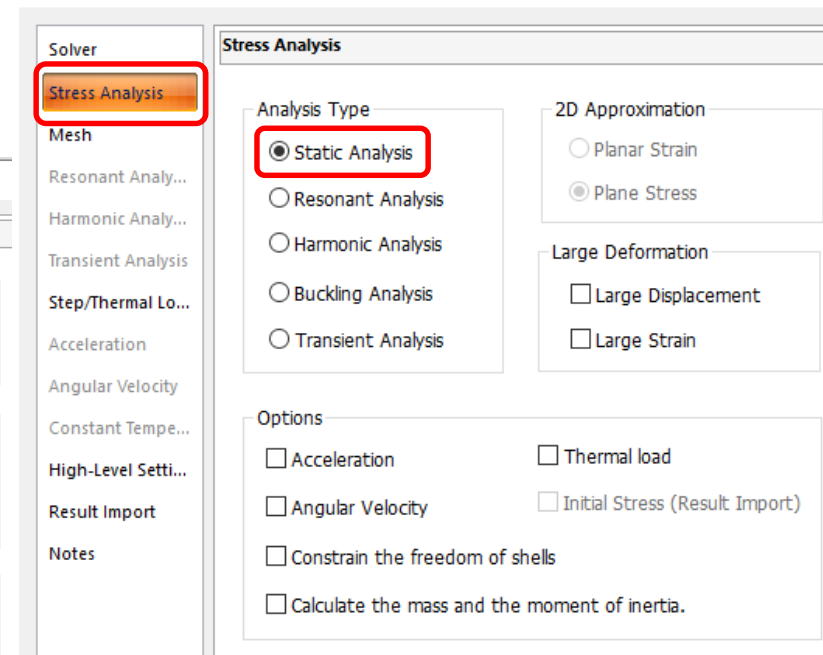
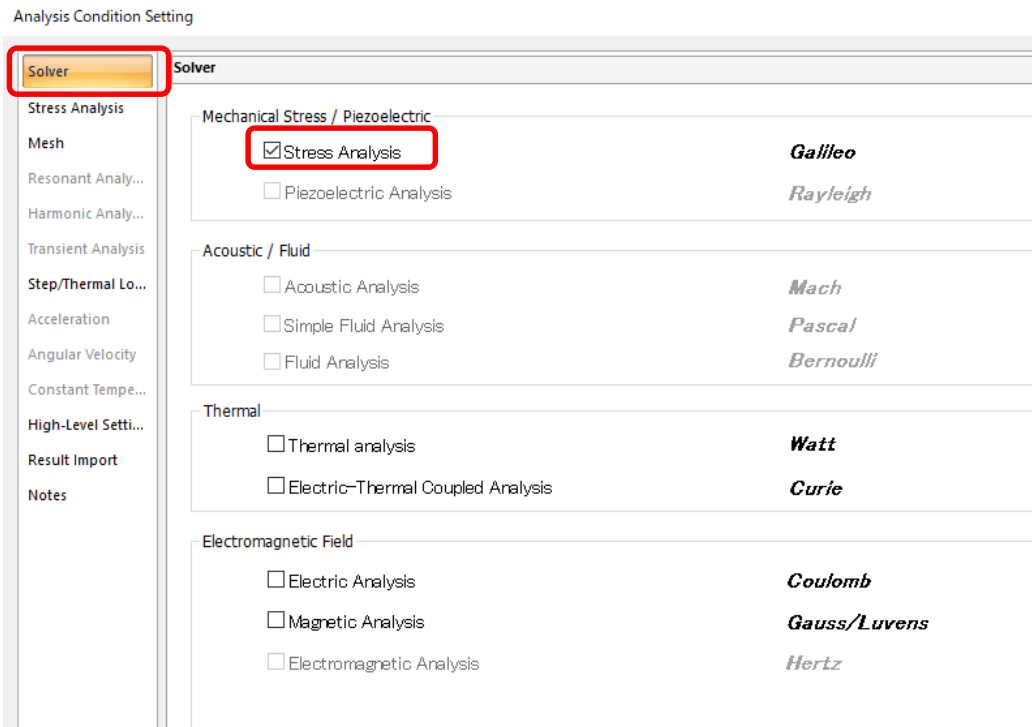
Analysis Condition Setting

[Model] > [Analysis] > [Analysis Condition]
Select [Stress Analysis] on the Solver tab.



Select [Static Analysis] on the [Stress Analysis] tab.

Analysis Condition Setting



Exercise - Stress Analysis

Body Attribute/Material Property Setting

Switch target to Body.

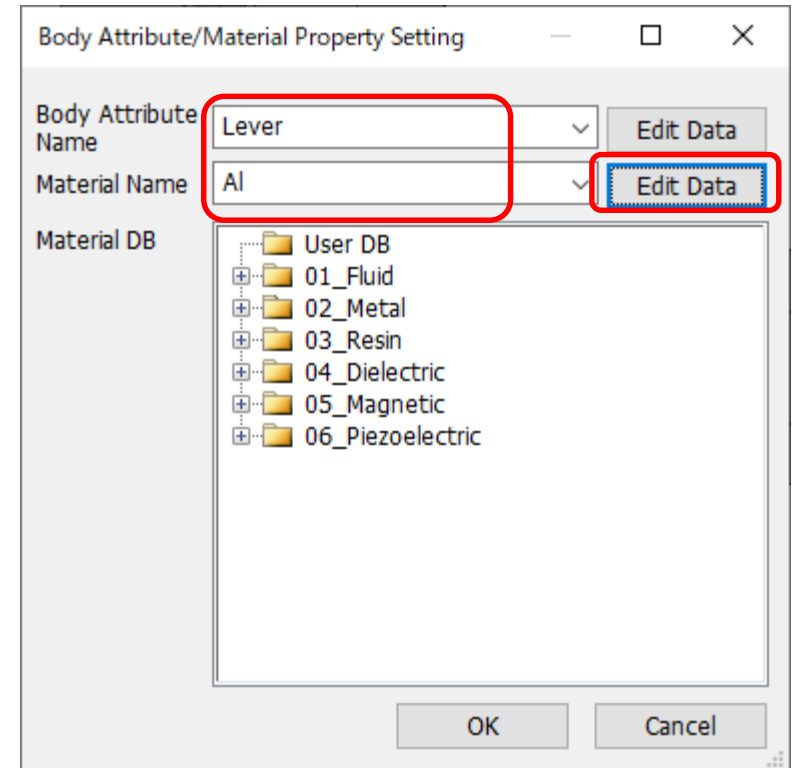
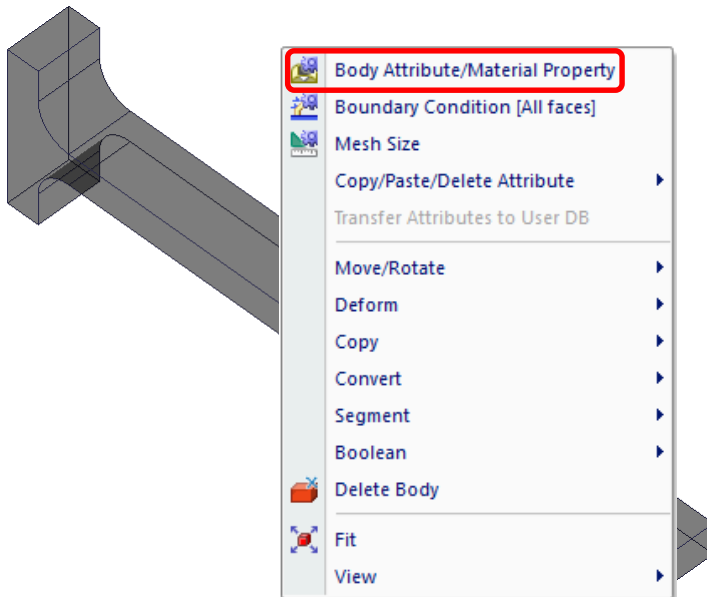
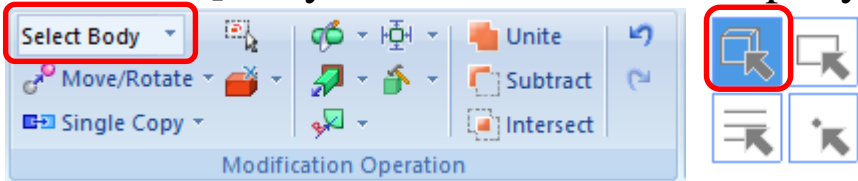
Right-click on the beam body.

Select [Body Attribute/Material Property]

Enter [Lever] as Body Attribute Name.

Enter [Al] as Material name.

Press Edit Data button.



Exercise - Stress Analysis

Body Attribute/Material Property Setting

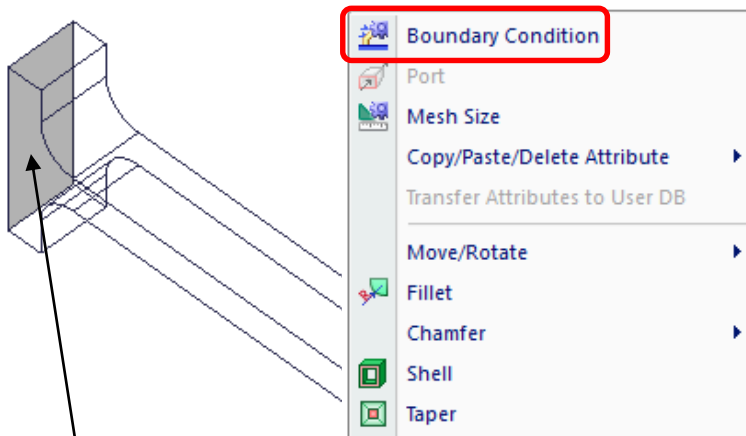
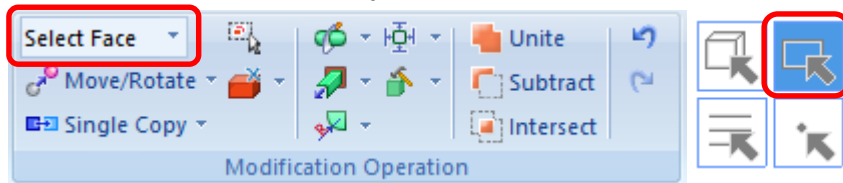
Select [Elastic/Isotropic] on the Elasticity tab.
Type 6.85 in the Young's modulus and 10 for exponent.
Enter 0.34 in the Poisson's ratio.

The screenshot shows the 'Elasticity' tab in a software interface. The 'Material Type' section has 'Elastic/Isotropic' selected. The 'Temperature Dependency' section has 'No' selected. The 'Hardening Law' section has 'Isotropic Hardening' selected. The 'Elasticity Matrix Type' section has 'Compliance' selected. The 'tanD (Mechanical Damping)' section has a value of 0.0. The 'Young's modulus' section has a value of 6.85 X10 [Pa] and an exponent of 10. The 'Poisson's ratio' section has a value of 0.34. The 'OK' button is highlighted.

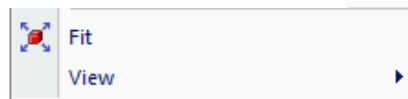
Exercise - Stress Analysis

Boundary Condition Setting

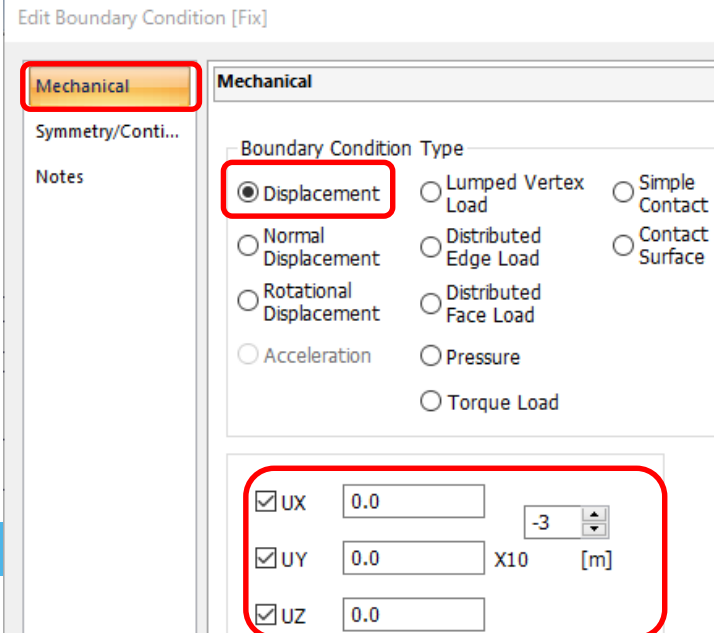
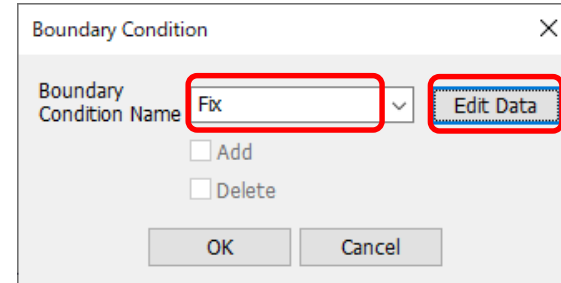
Select the base of the beam and right-click.
Select [Boundary Condition].



Bring mouse to the base of the beam.
Press a space key to highlight the area.



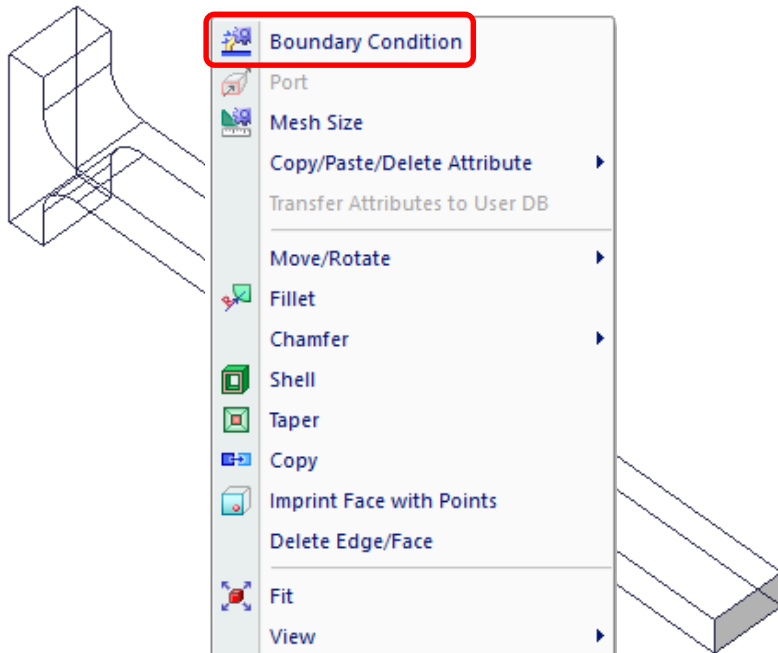
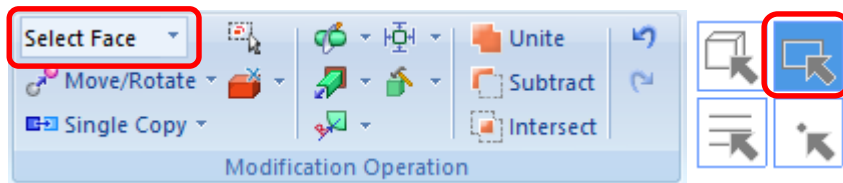
Type [Fix] as Boundary Condition Name.
Press Edit Data button.
Select [Displacement] on the Mechanical tab.
Type $UX=0$, $UY=0$, and $UZ=0$.



Exercise - Stress Analysis

Boundary Condition Setting

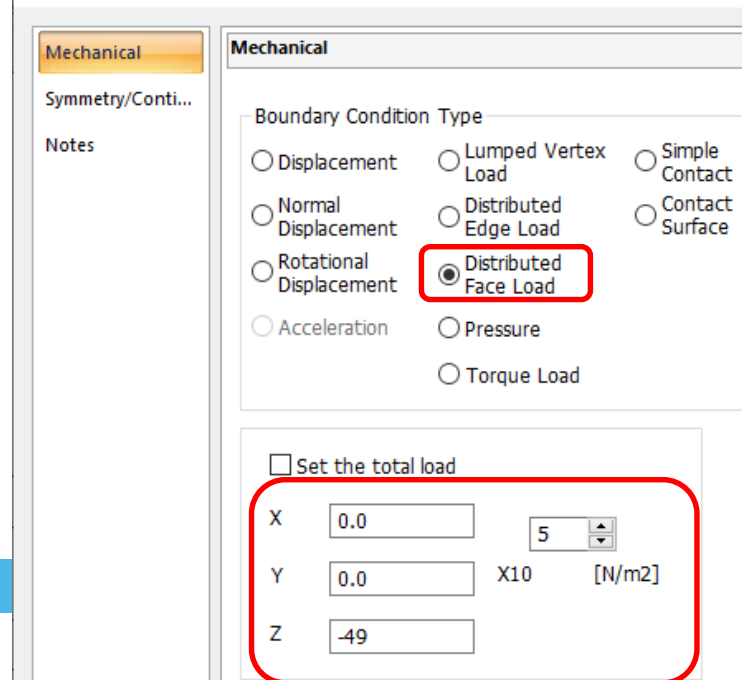
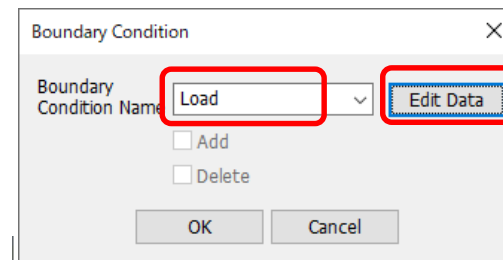
Right-click on the front end of the beam.
Select [Boundary Condition].



Type [load] as Boundary Condition Name.
Press Edit Data button.

Select [Distributed Face Load] on the Mechanical tab.

Type X=0, Y=0, Z= -49, and exponent=5.



Mesh Size Setting

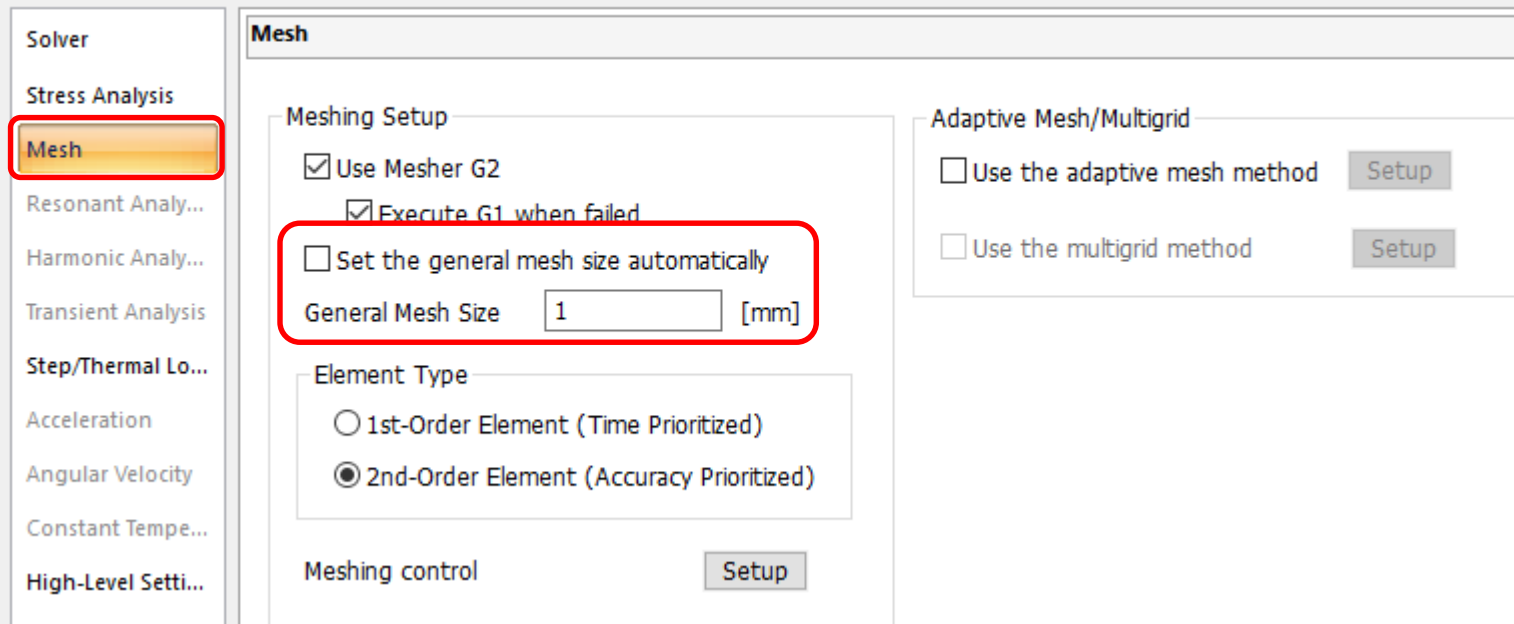
[Model] > [Analysis] > [Analysis Condition]

Deselect [Set the general mesh size automatically] on the Mesh tab.

Enter General Mesh Size = 1.



Analysis Condition Setting

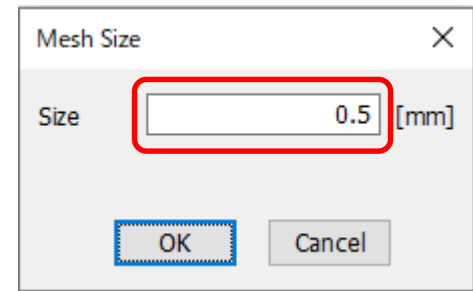
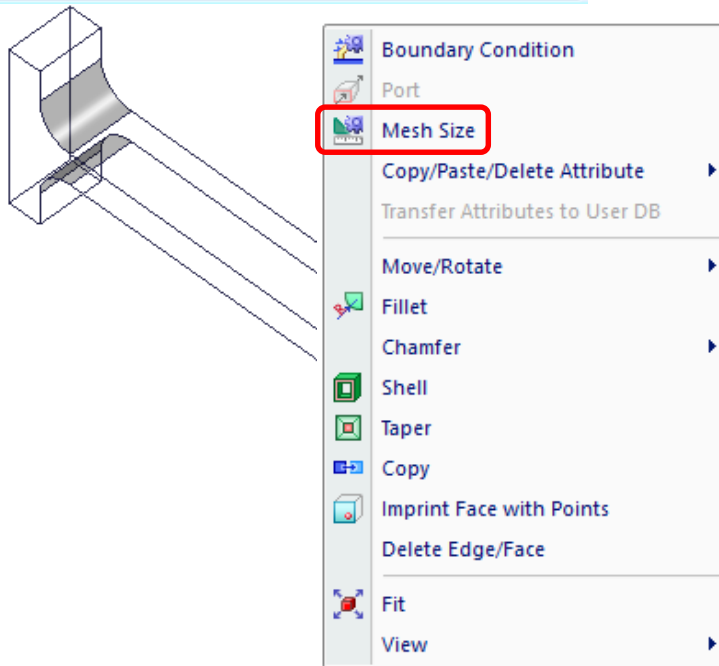
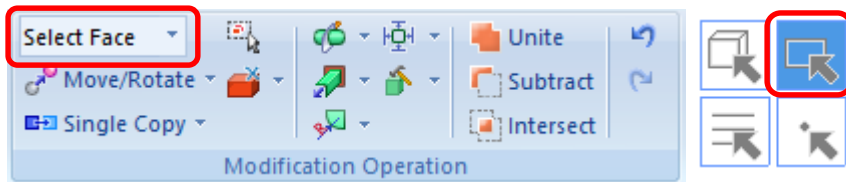
A screenshot of the 'Analysis Condition Setting' dialog box. On the left is a sidebar with a list of analysis types: 'Solver', 'Stress Analysis', 'Mesh' (highlighted with a red box), 'Resonant Analy...', 'Harmonic Analy...', 'Transient Analysis', 'Step/Thermal Lo...', 'Acceleration', 'Angular Velocity', 'Constant Tempe...', and 'High-Level Setti...'. The main area is titled 'Mesh' and contains several sections: 'Meshing Setup' with checkboxes for 'Use Mesher G2' (checked), 'Execute G1 when failed' (checked), and 'Set the general mesh size automatically' (unchecked, highlighted with a red box); a 'General Mesh Size' input field containing the value '1' with '[mm]' units; 'Element Type' with radio buttons for '1st-Order Element (Time Prioritized)' and '2nd-Order Element (Accuracy Prioritized)' (selected); and 'Meshing control' with a 'Setup' button. To the right is the 'Adaptive Mesh/Multigrid' section with checkboxes for 'Use the adaptive mesh method' and 'Use the multigrid method', each with a 'Setup' button.

Exercise - Stress Analysis

Mesh Size Setting

Select two faces of the beam's base and right-click.
Select [Mesh Size].

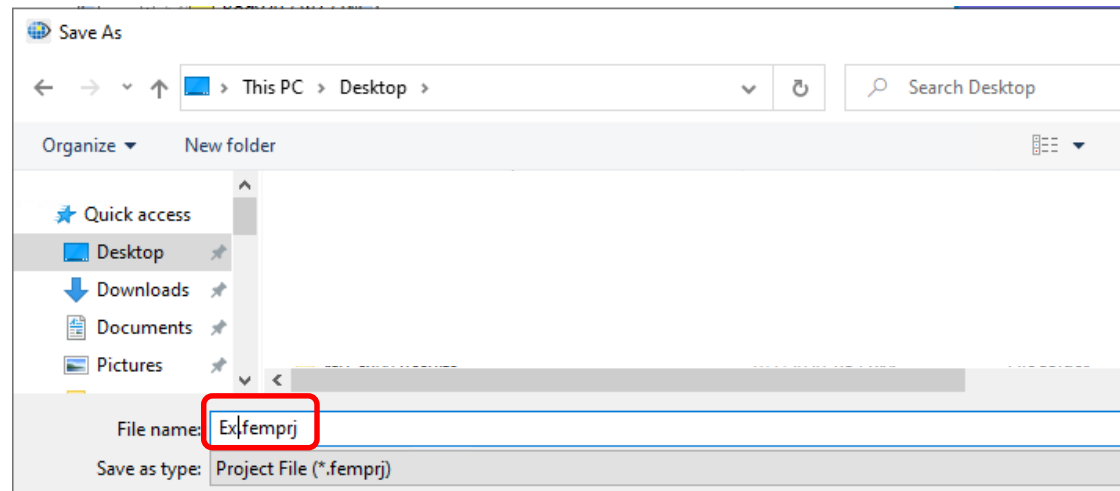
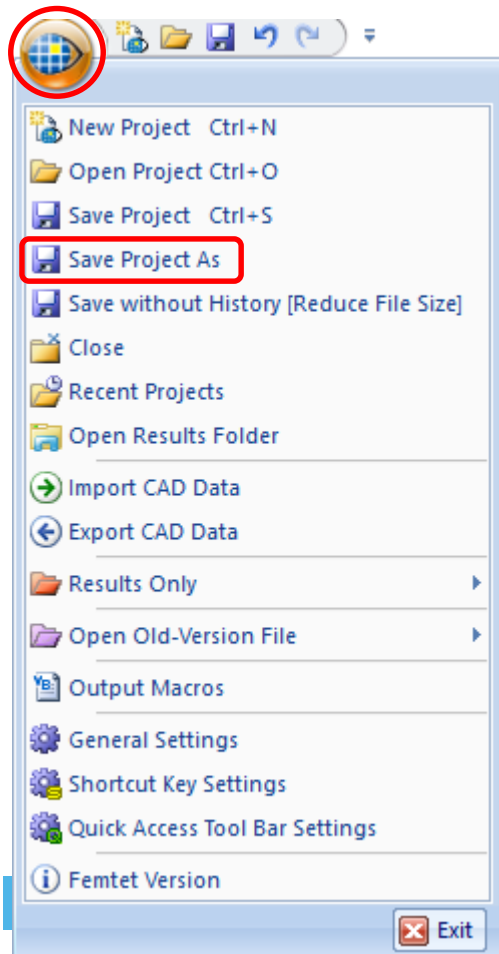
Type 0.5 in the Mesh Size dialog box.



Exercise - Stress Analysis

Save Model

Application button > [Save Project As].
Type file name as you wish.



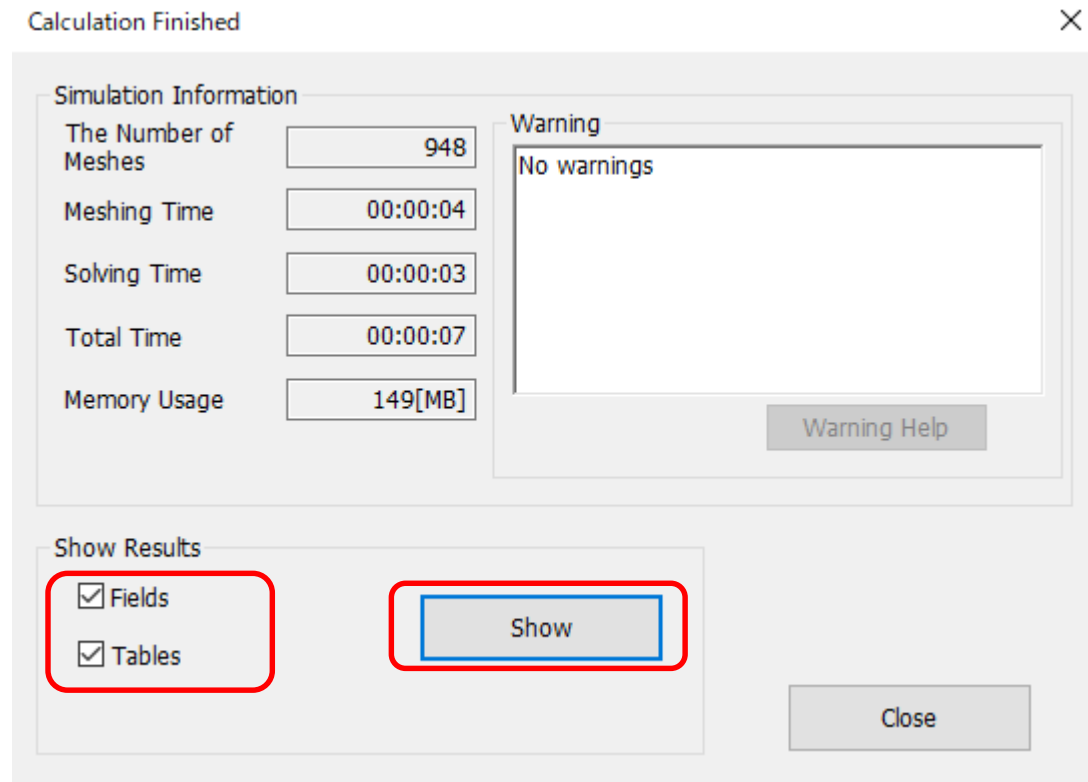
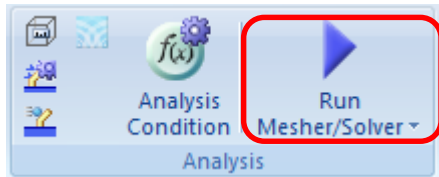
* You can save file anytime while you work on the project.

Exercise - Stress Analysis

Run Solver

[Model] > [Analysis] > [Run Mesher/Solver].

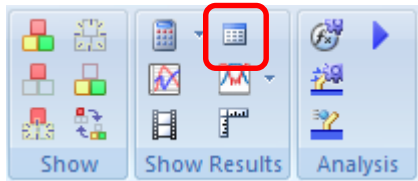
In the [Calculation Finished] dialog box, select [Fields] and [Tables], and press Show button.



Table

If [Table] is selected in the Calculation Finished dialog box, the table will show up automatically.

Or, [Results] > [Show Results] > [Table]



Table

	External/Reactive Force [N]	Strain energy [J]	Maximum displacement [m]	Maximum stress [Pa]	FEM Info
	X component	Y	Z	Absolute value	
Fix	1.713e-11	5.361e-11	9.8	9.8	
Load	4.053e-11	2.475e-11	-9.8	9.8	

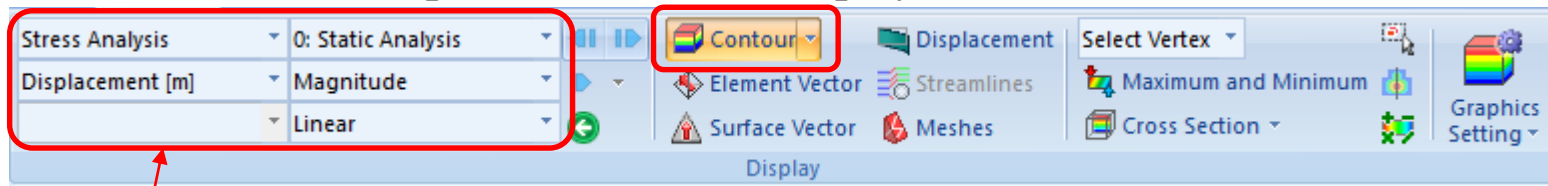
Small values of the X and Y components are output due to a calculation error.

Exercise - Stress Analysis

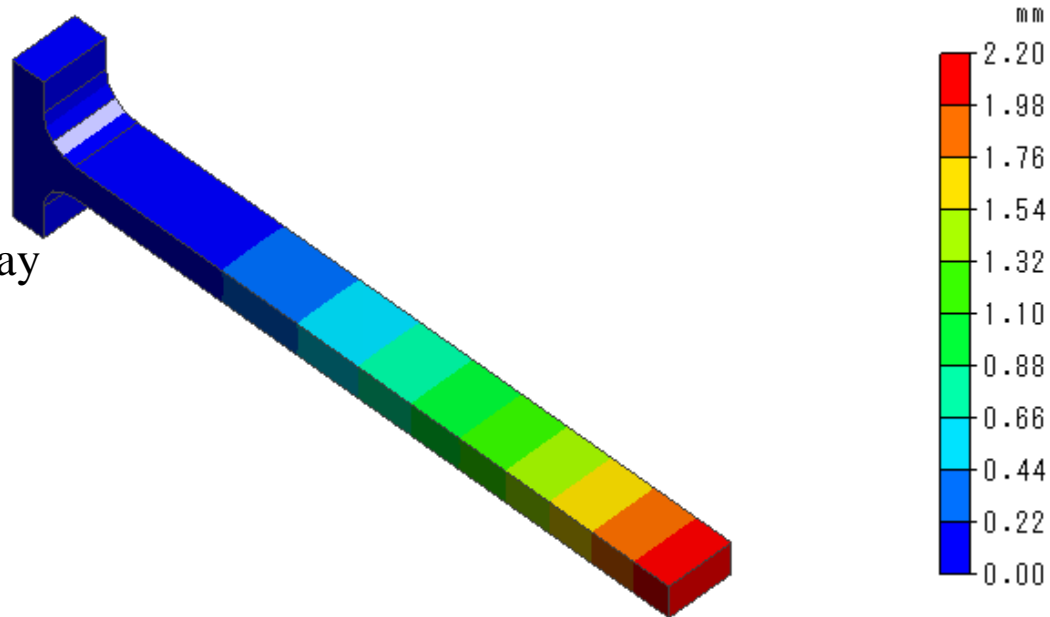
Contour

[Results] > [Display] > [Contour]

The results of specified items are displayed.



Items to display

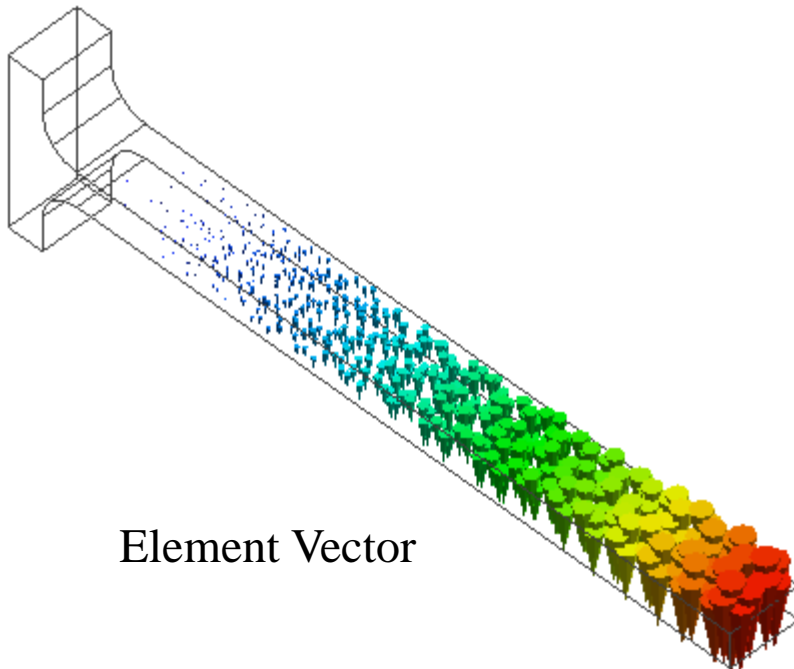
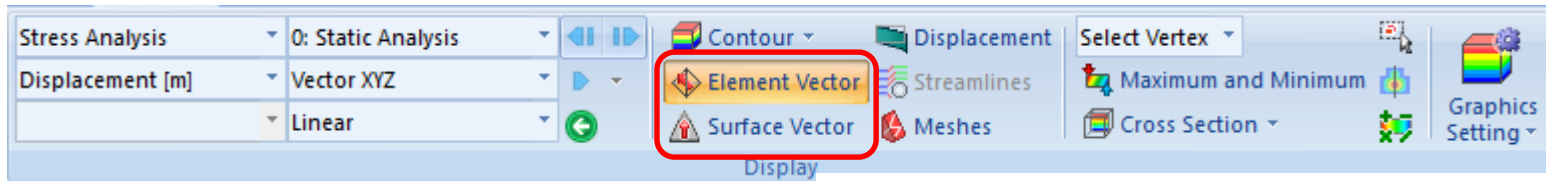


Exercise - Stress Analysis

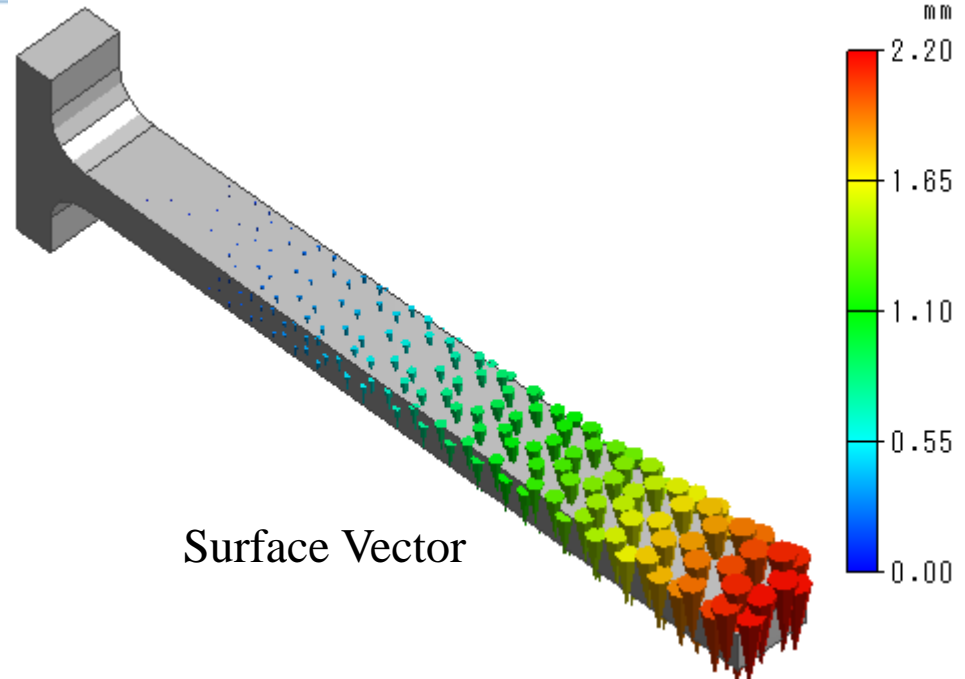
Vector

[Results] > [Display] > [Element Vector] or [Surface Vector]

The surface vectors show the results on the surface of the analysis model only.



Element Vector



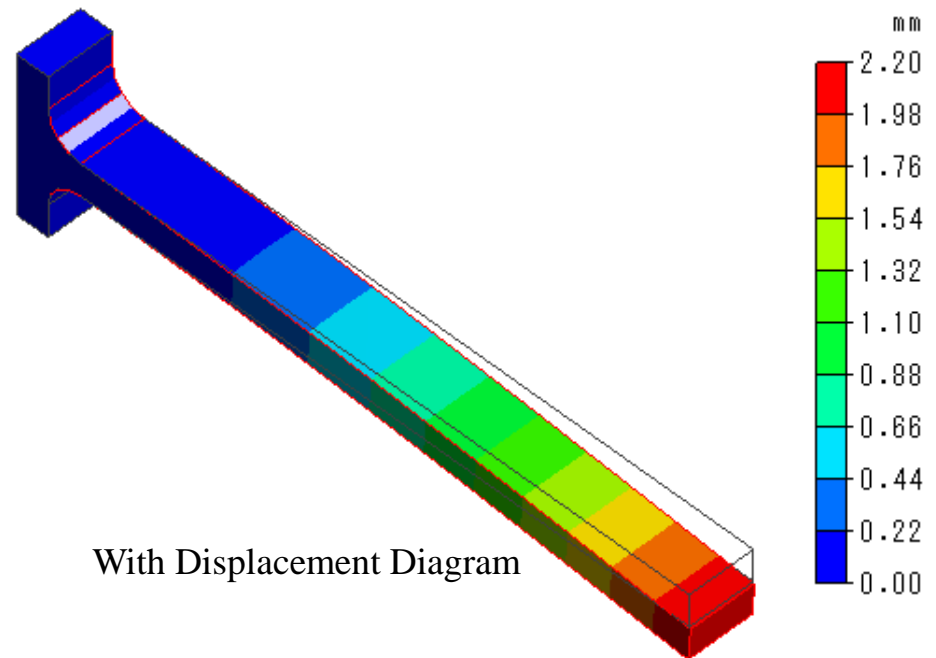
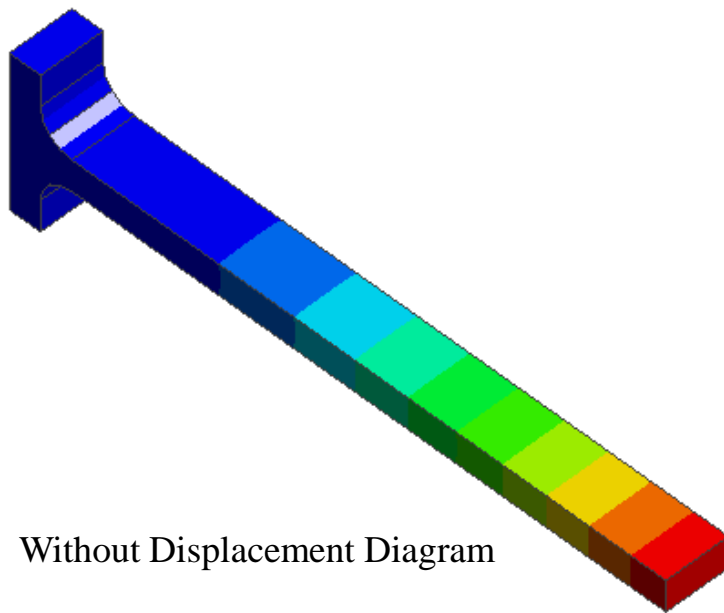
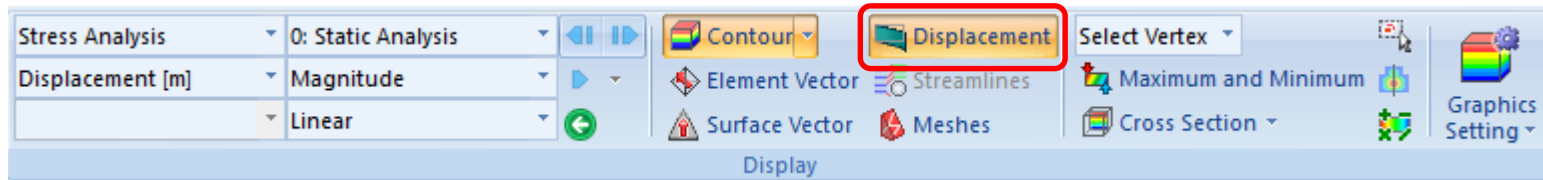
Surface Vector



Exercise - Stress Analysis

Displacement

[Results] > [Display] > [Displacement]

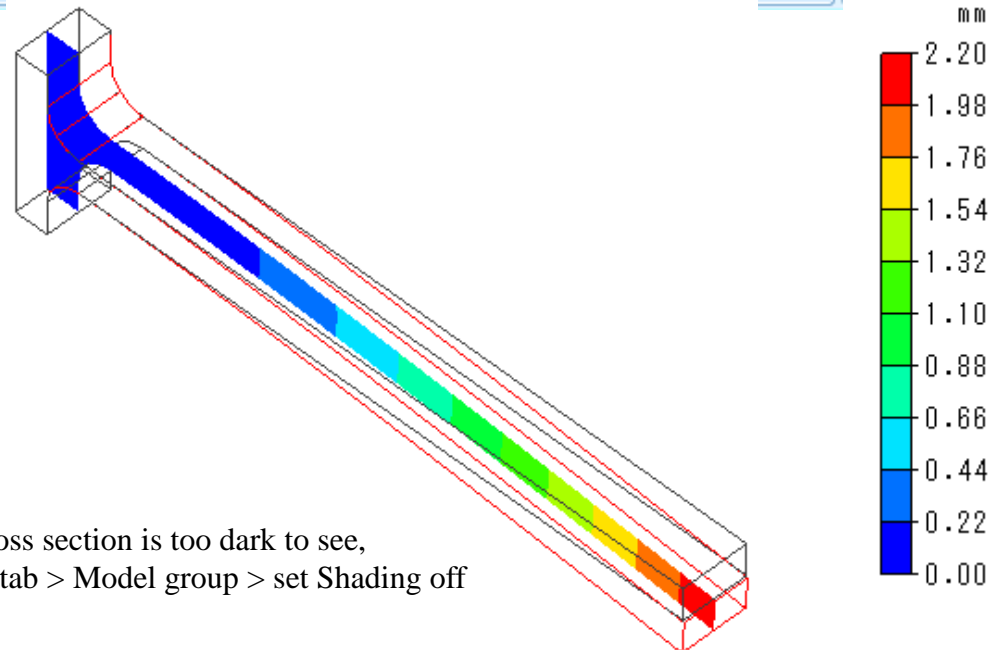
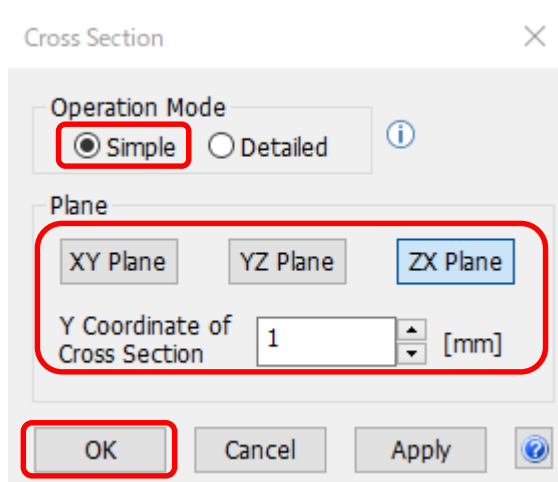
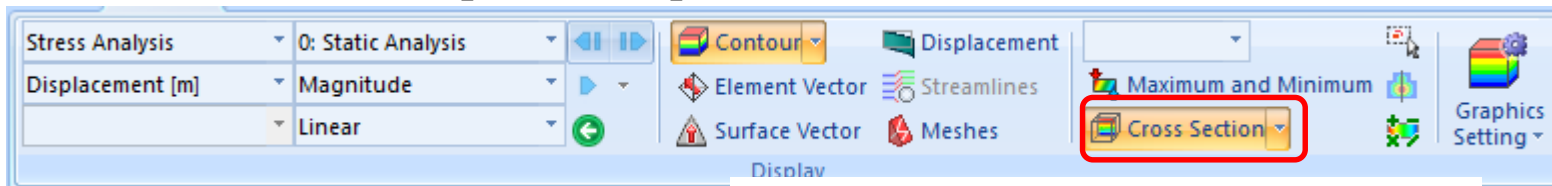


Exercise - Stress Analysis

Cross Section (Simple)

[Results] > [Display] > [Cross Section]

In the dialog box, select Simple Operation Mode, specify the cross section and coordinates of the plane, and press OK.

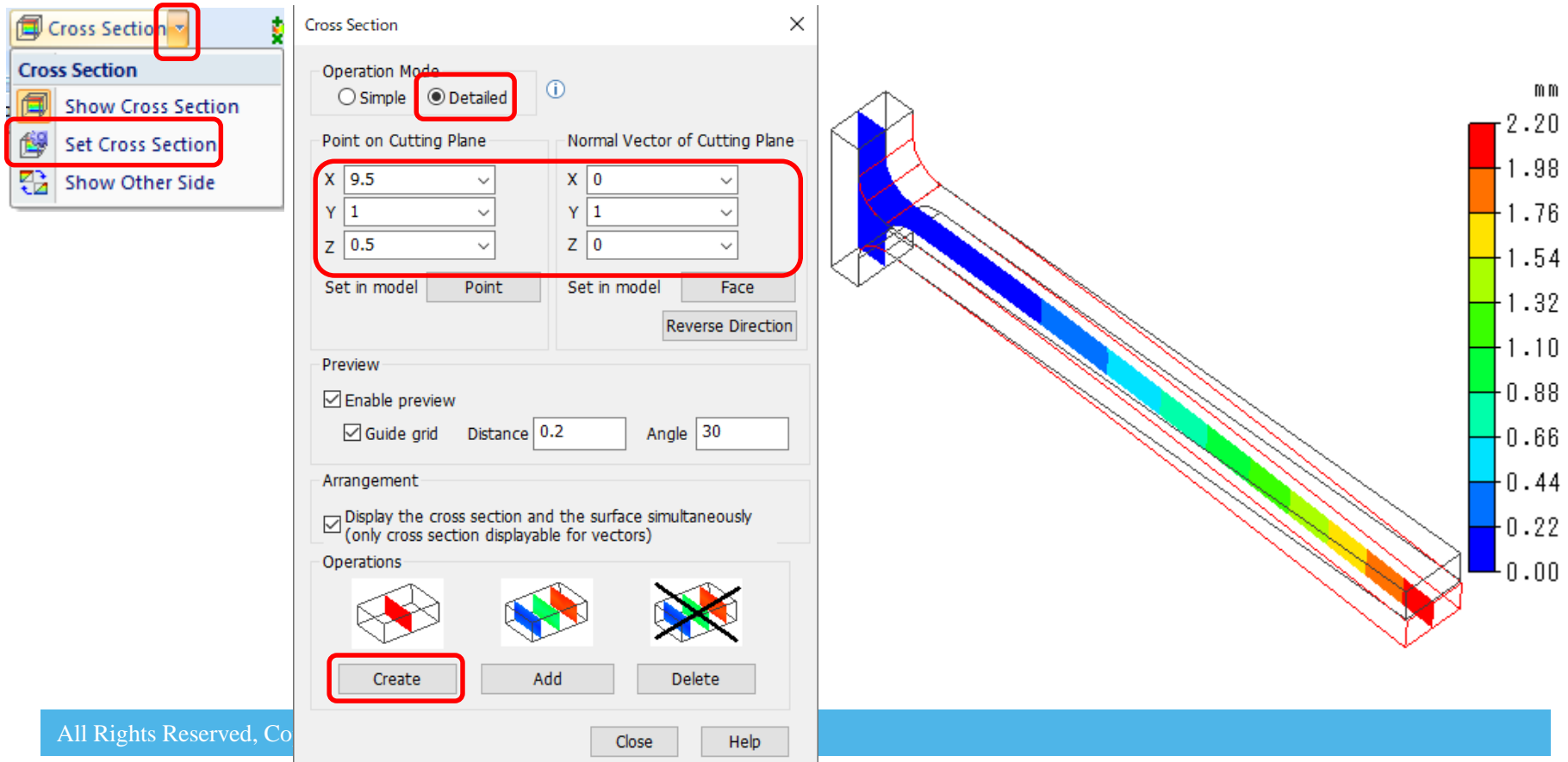


If the cross section is too dark to see,
Display tab > Model group > set Shading off

Cross Section (Detailed)

[Results] > [Display] > Submenu of [Cross Section] > Set Cross Section.

In the dialog box of Cross Section, select Detailed Operation Mode, specify the Point on Cutting Plane and Normal Vector of Cutting Plane, and press Create button.



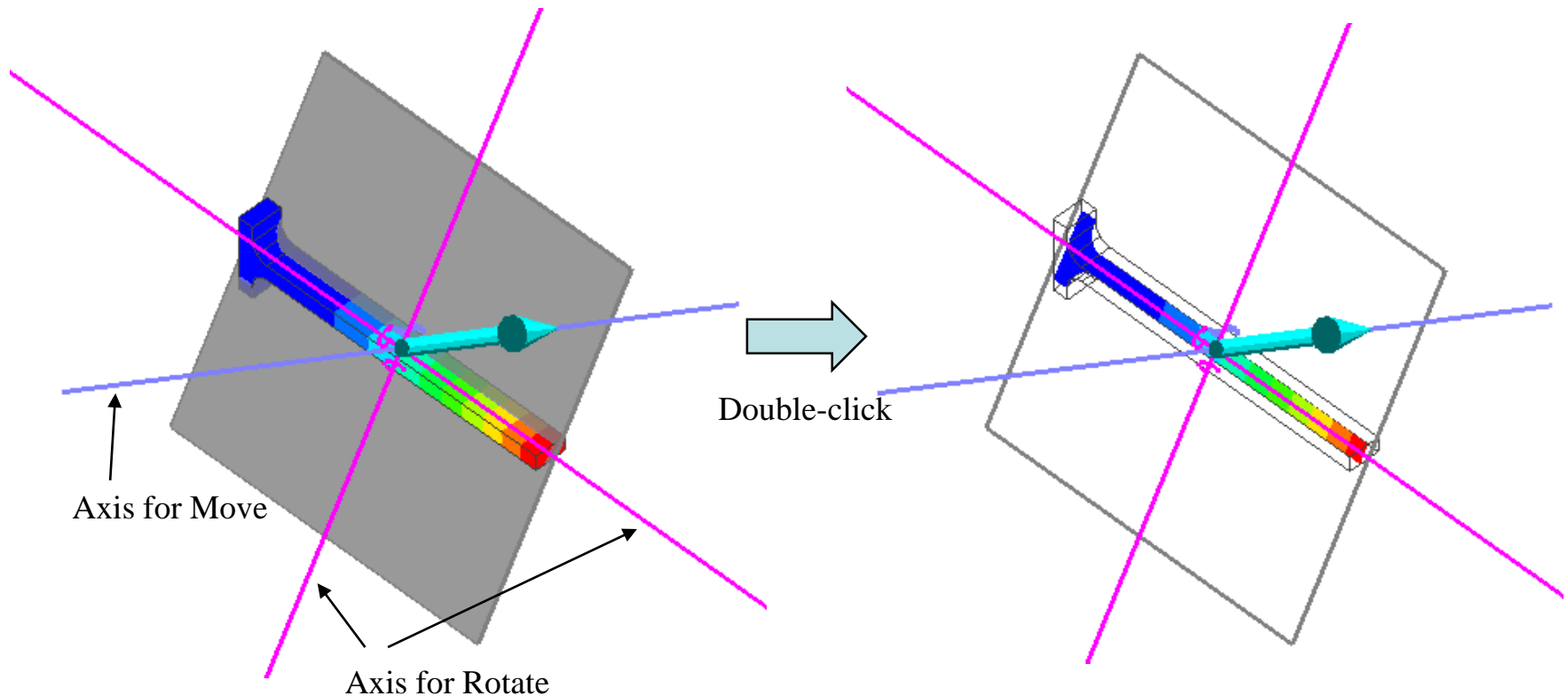
The image shows a software interface for creating a cross-section. On the left, a 'Cross Section' menu is open with 'Set Cross Section' highlighted. The main dialog box, titled 'Cross Section', has the following settings:

- Operation Mode: Detailed
- Point on Cutting Plane: X: 9.5, Y: 1, Z: 0.5
- Normal Vector of Cutting Plane: X: 0, Y: 1, Z: 0
- Preview: Enable preview, Guide grid (Distance: 0.2, Angle: 30)
- Arrangement: Display the cross section and the surface simultaneously (only cross section displayable for vectors)
- Operations: Create, Add, Delete

On the right, a 3D model of a mechanical part is shown with a cross-section. A color scale on the right indicates stress values in mm, ranging from 0.00 (blue) to 2.20 (red).

Cross Section on the Preview Window

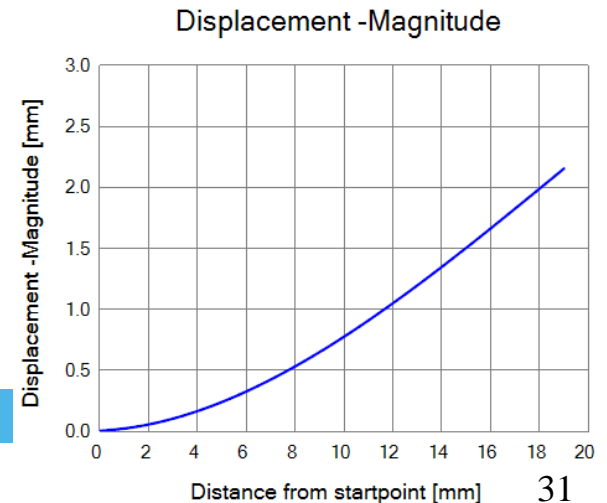
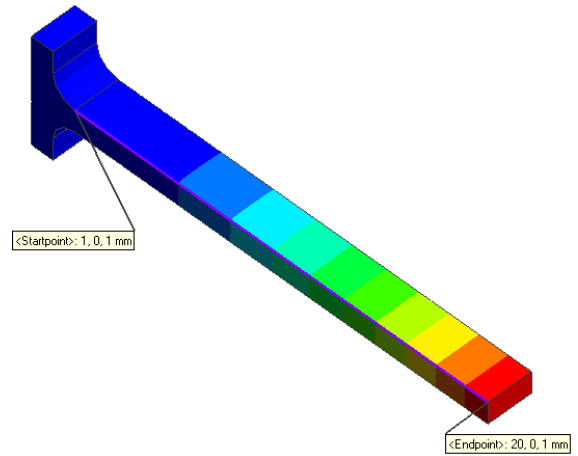
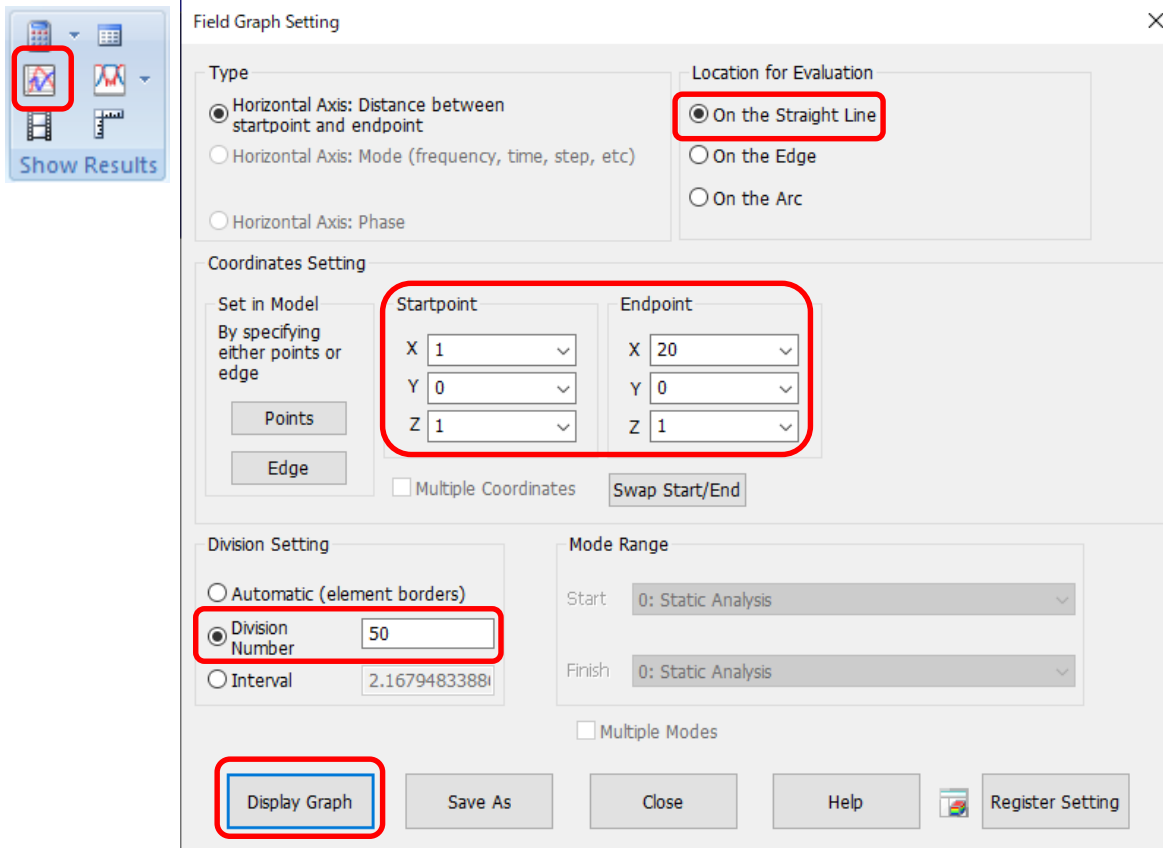
Move the cross section on a blue axis and rotate the cross section on a purple axis.
Show the cross section by double-clicking the position of your wish.



Display Graph on the Straight Line

[Results] > [Display] > [Display Graph]

Select [On the Straight Line] for the location for evaluation, and enter Startpoint Endpoint, and Division Number.
Press Display Graph button.



Exercise - Stress Analysis

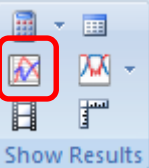
Display Graph on the Edge

[Results] > [Display] > [Display Graph]

Select [On the Edge] for the location for evaluation and specify the edge(s) and enter Division Number.

Press Display Graph button.

Field Graph Setting



Type

Horizontal Axis: Distance between startpoint and endpoint

Horizontal Axis: Mode (frequency, time, step, etc)

Horizontal Axis: Phase

Location for Evaluation

On the Straight Line

On the Edge [Select edge in the window]

On the Arc

Coordinates Setting

Set in Model

Select either point or edge

Points

Edge

Startpoint

X 0

Y 0

Z 0

Endpoint

X 0

Y 0

Z 0

Multiple Coordinates

Swap Start/End

Division Setting

Automatic (element borders)

Division Number 10

Interval 1.4696938456

Mode Range

Start 0: Static Analysis

Finish 0: Static Analysis

Multiple Modes

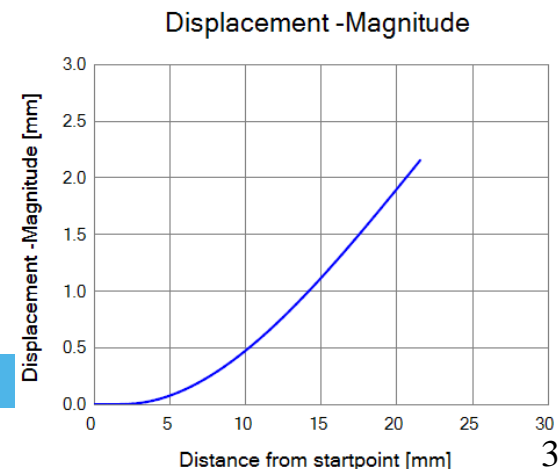
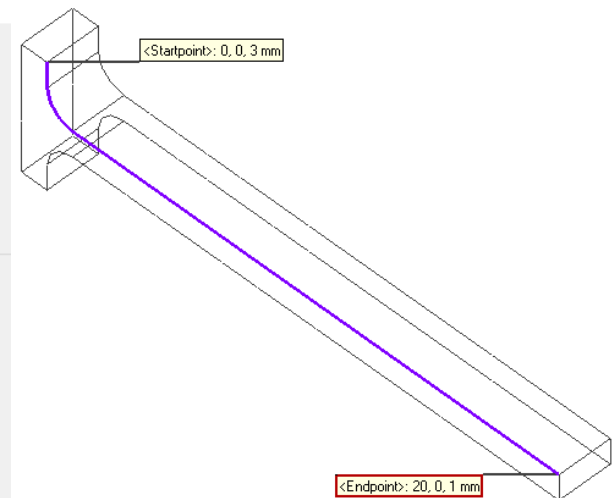
Display Graph

Save As

Close

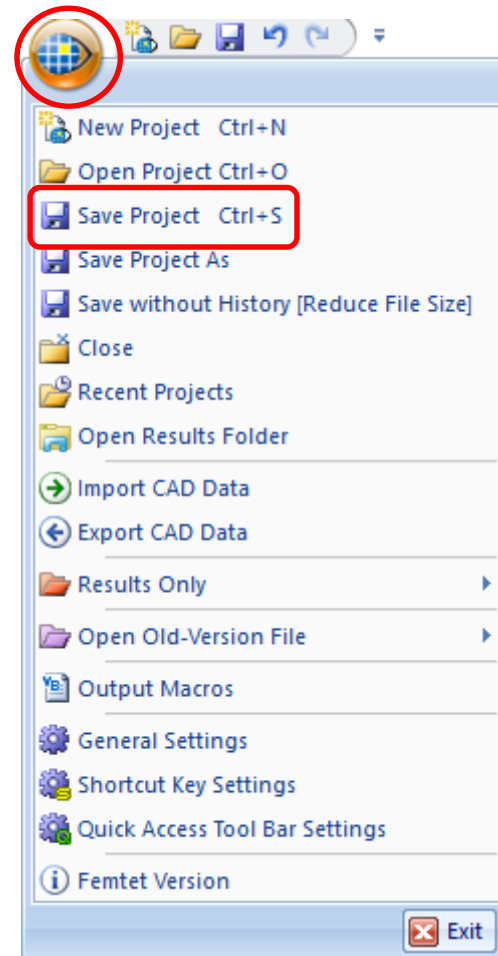
Help

Register Setting



Save the Results

Application button > [Save Project]
Overwrite the project file that was saved
when creating the analysis model.

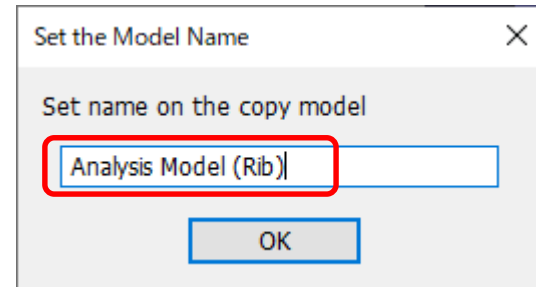
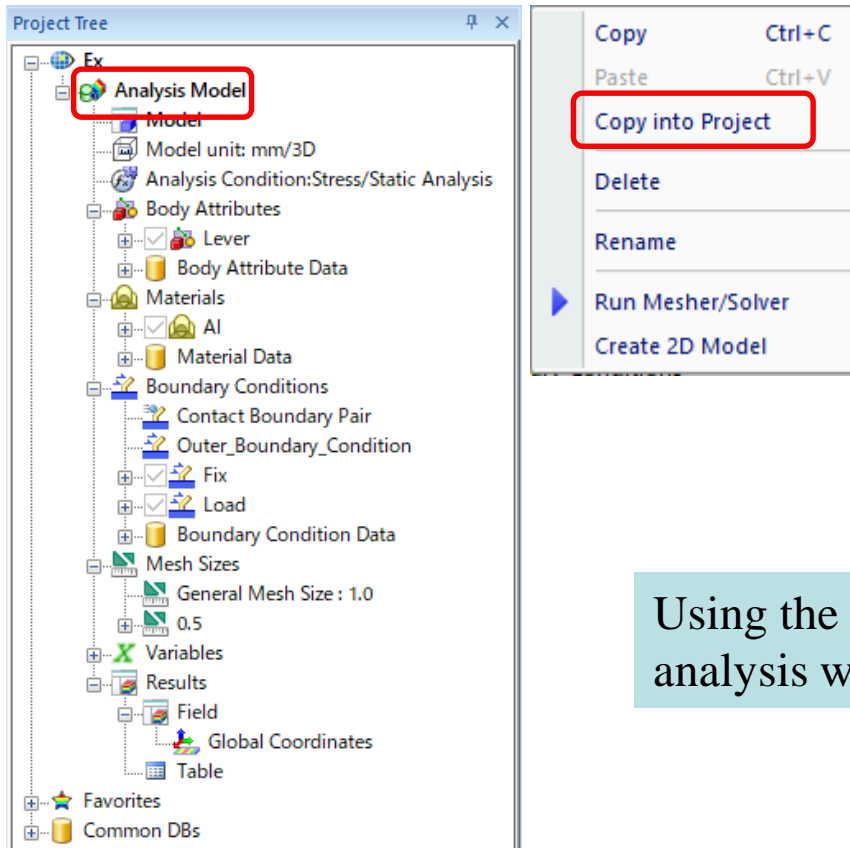


Exercise - Stress Analysis

Copy Analysis Model

Right-click on [Analysis Model] in the project tree.
Execute [Copy into Project].

In the dialog box, enter the analysis model name [Analysis Model (Rib)].



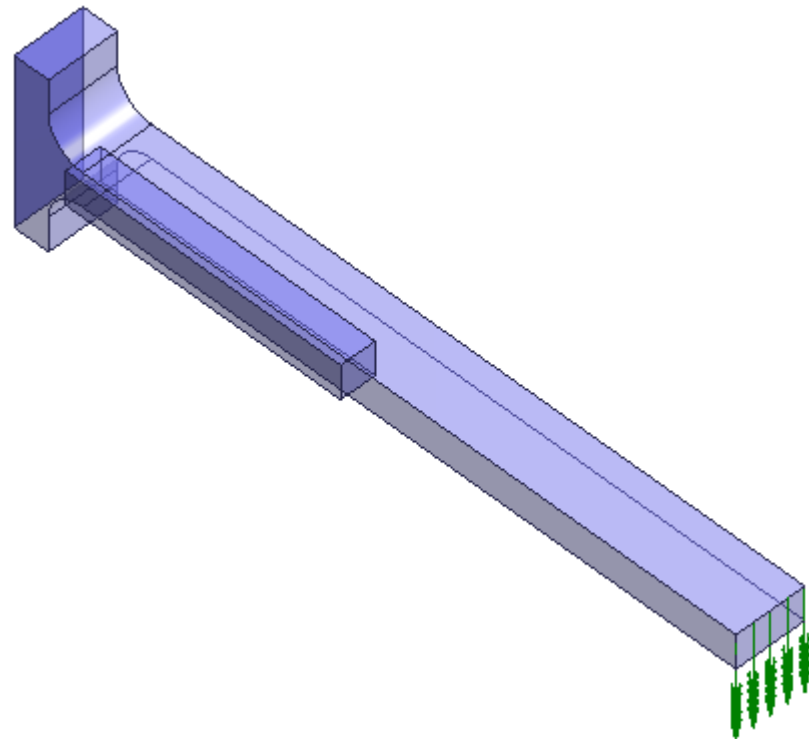
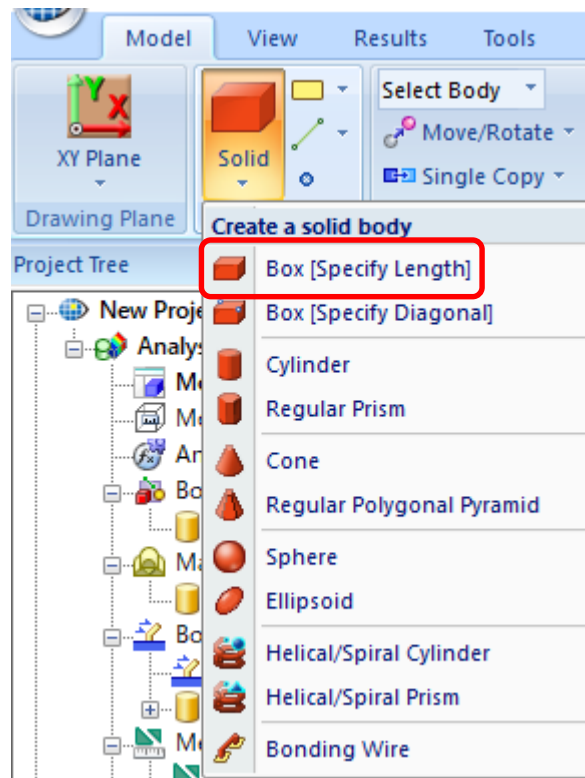
Using the existing analysis model and results,
analysis with different conditions can be executed.

Exercise - Stress Analysis

Modeling

[Model] > [Primitives] > [Box [Specify Length]].

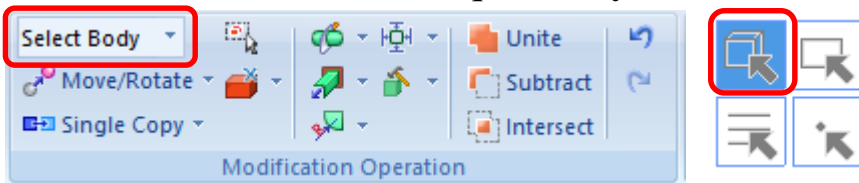
Enter Startpoint [0, 0.5, 0], Width [8], Depth [1], and Height [-1].



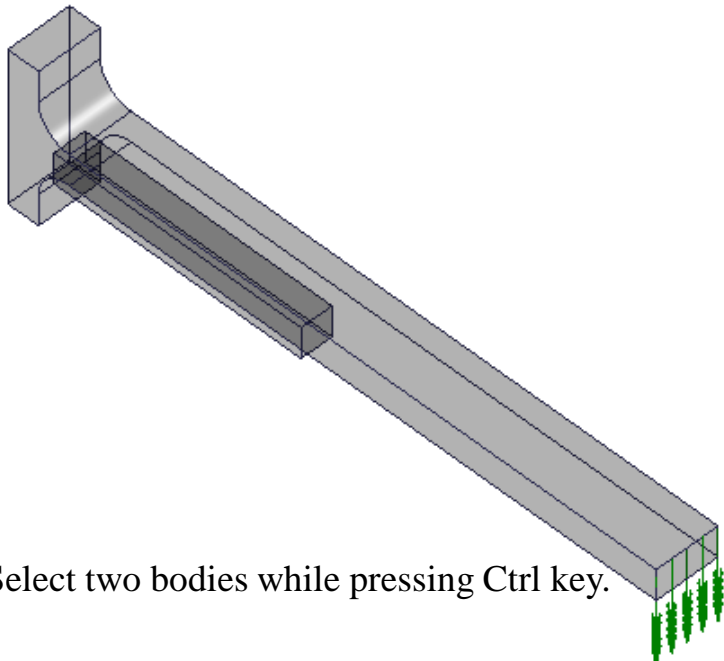
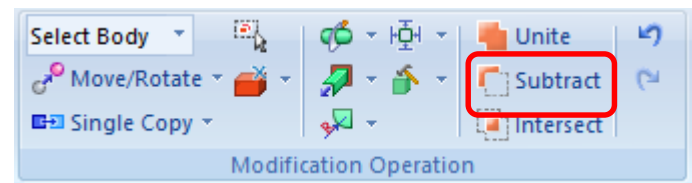
Exercise - Stress Analysis

Modeling

Switch to Select Body.
Select rib and beam sequentially.

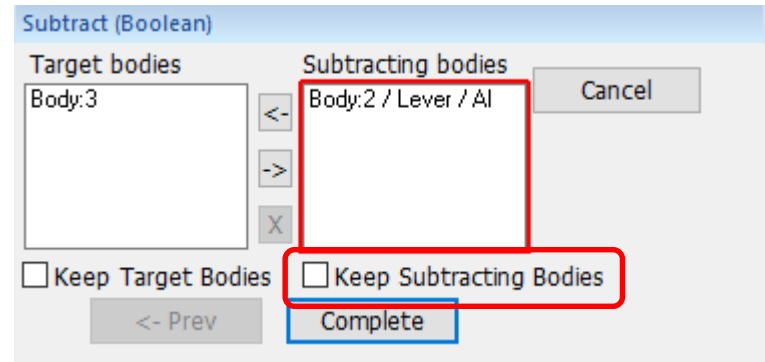


[Model] > [Modification Operation] > [Subtract]



Select two bodies while pressing Ctrl key.

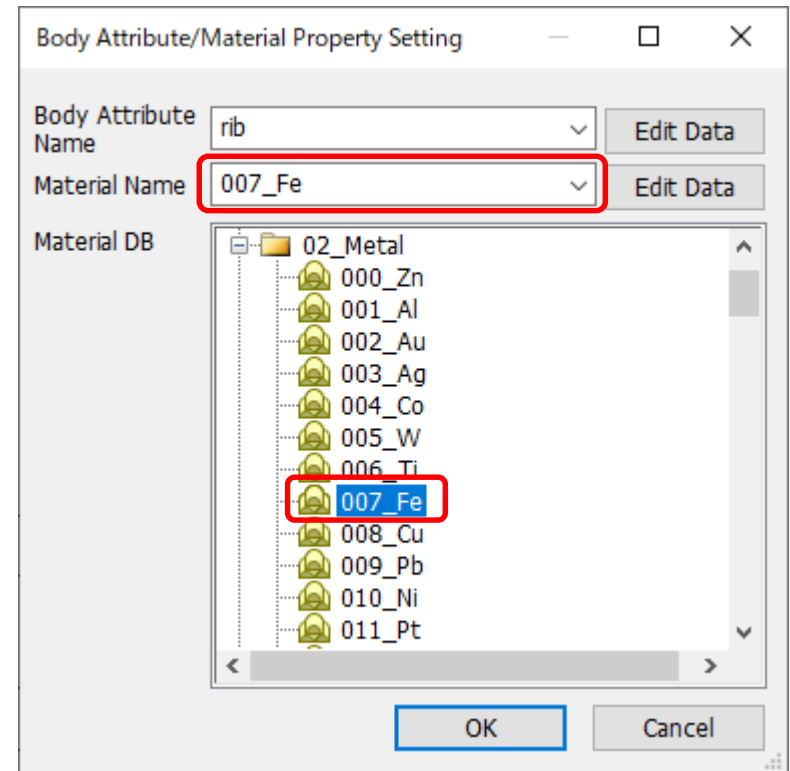
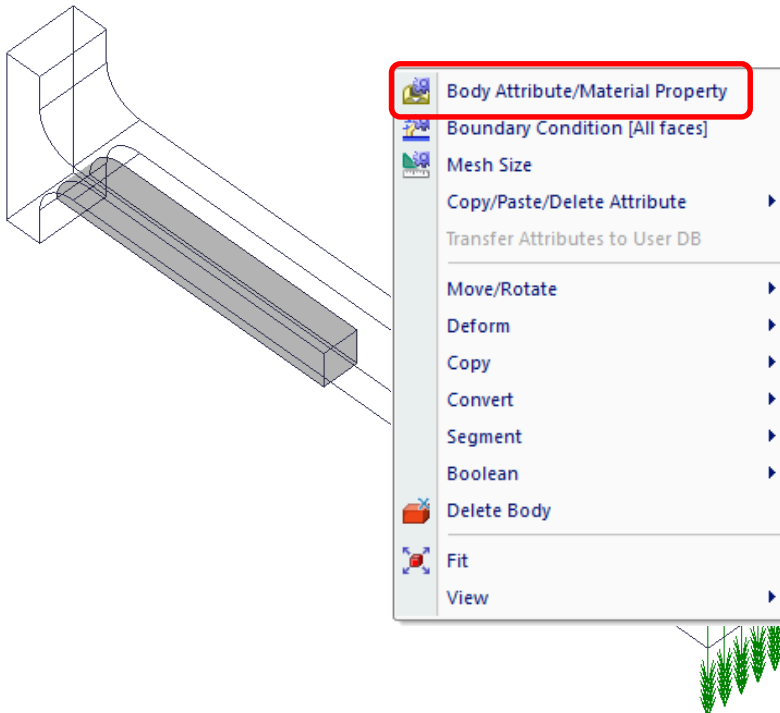
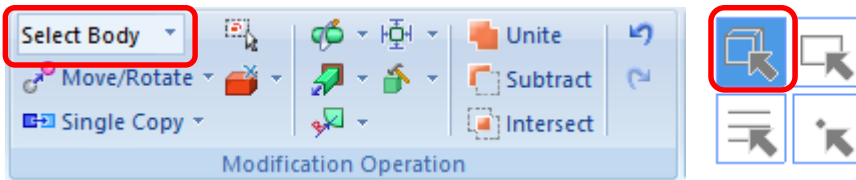
Confirm that the rib is set target body and the beam is set a subtracting body.
Select Keep Subtracting Bodies.



Body Attribute and Material Property Setting

Right-click on the rib body.
Select [Body Attribute/Material Property]

Enter [rib] for the Body Attribute Name.
Material DB > 02_Metal > 007_Fe.

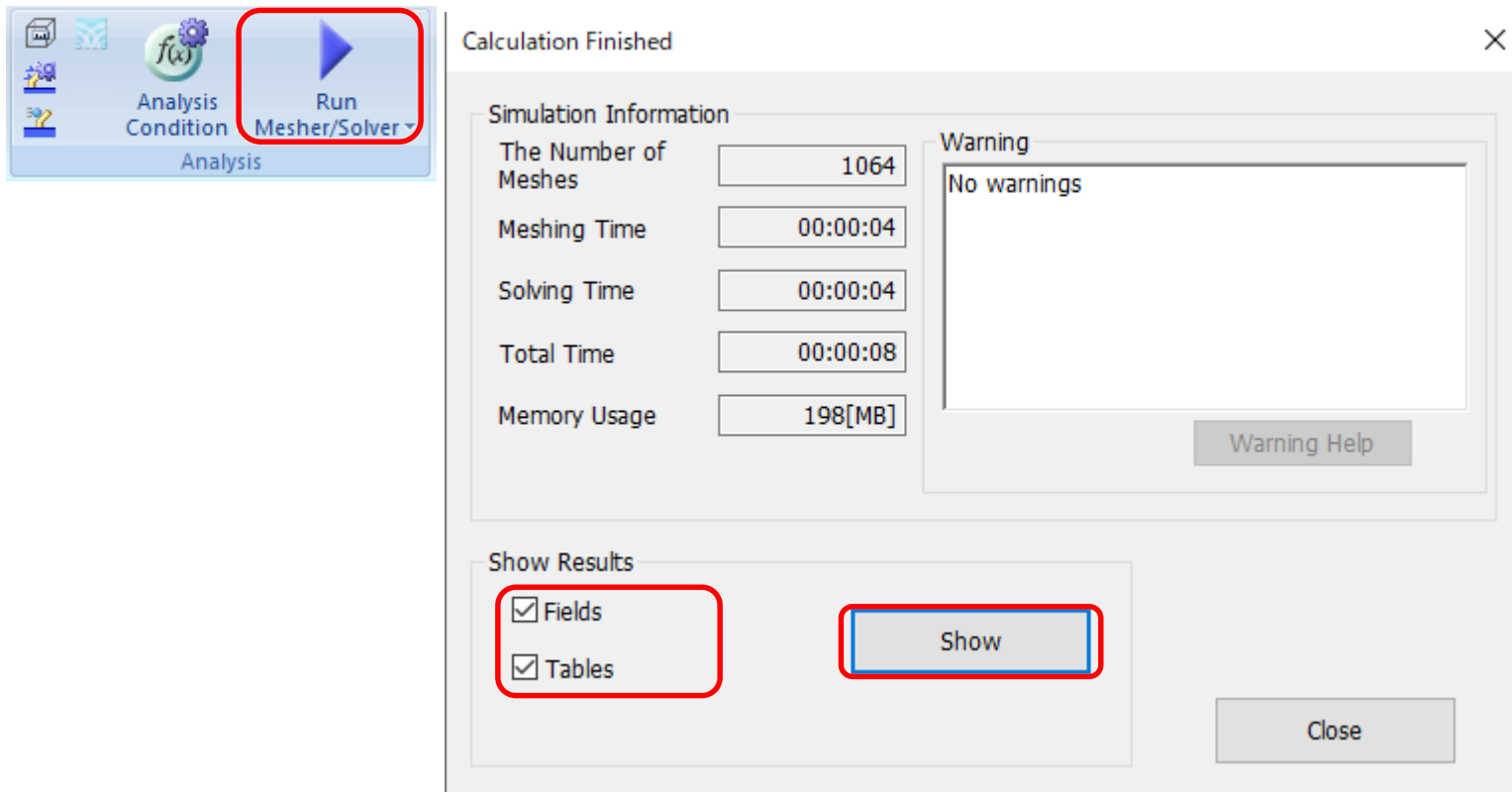


Exercise - Stress Analysis

Execute Analysis

[Model] > [Analysis] > [Run Mesher/Solver]

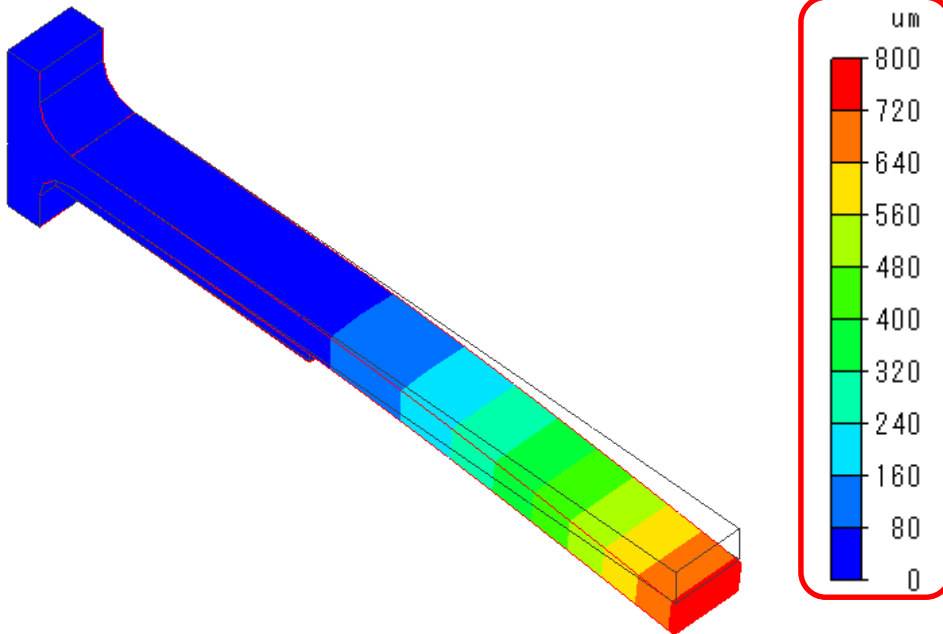
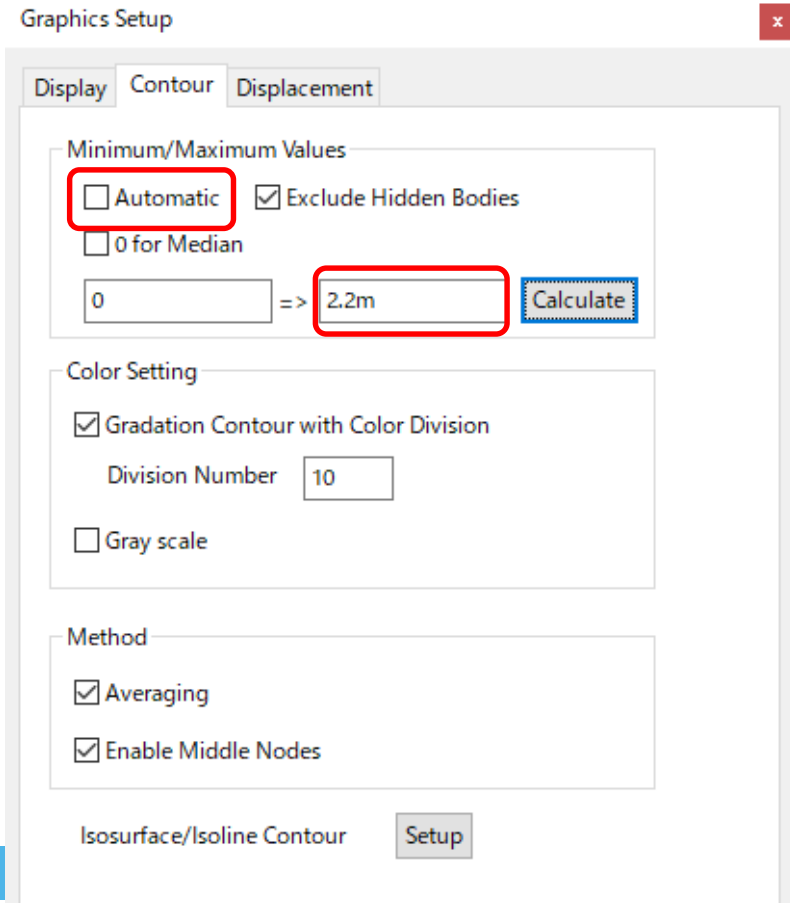
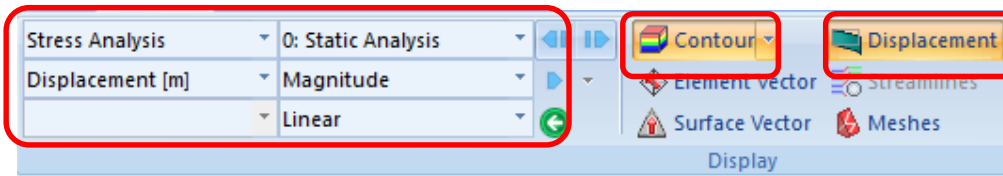
When the calculation is finished, select Fields and Tables in the dialog box, and press Show button.



Exercise - Stress Analysis

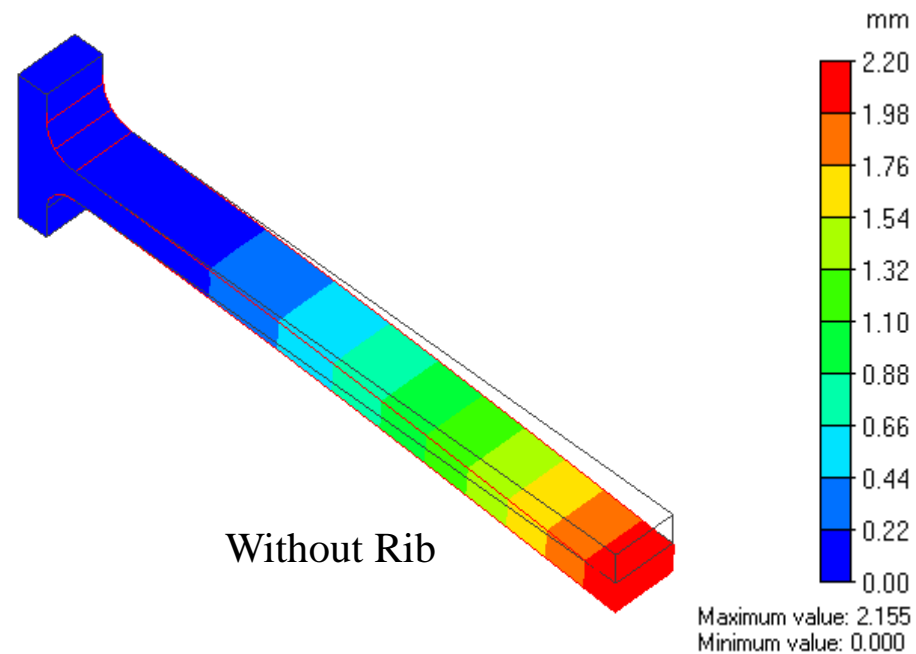
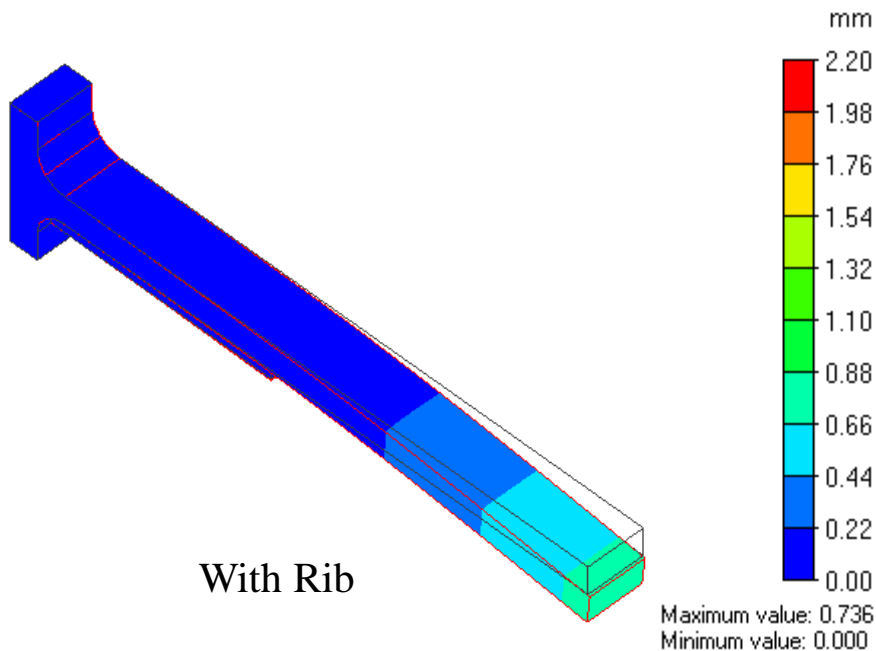
Displacement (Compare the Results with and without Rib)

[Results] > [Display] > Select Contour and Displacement, and double-click the color scale. Deselect [Automatic] of Minimum/Maximum Values, enter [2.2mm] for the maximum value. Press Calculate button.



Displacement (Compare the Results with and without Rib)

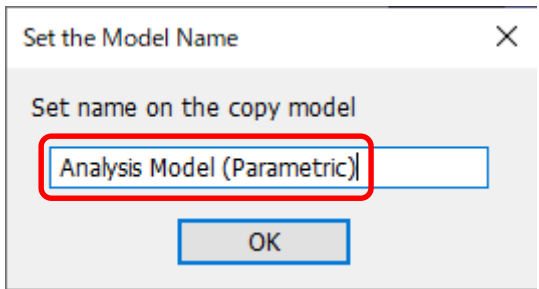
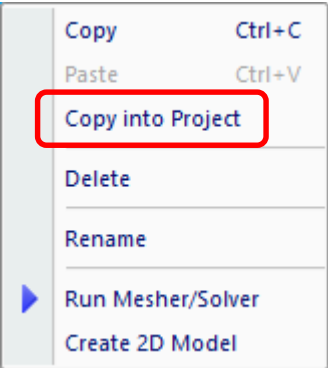
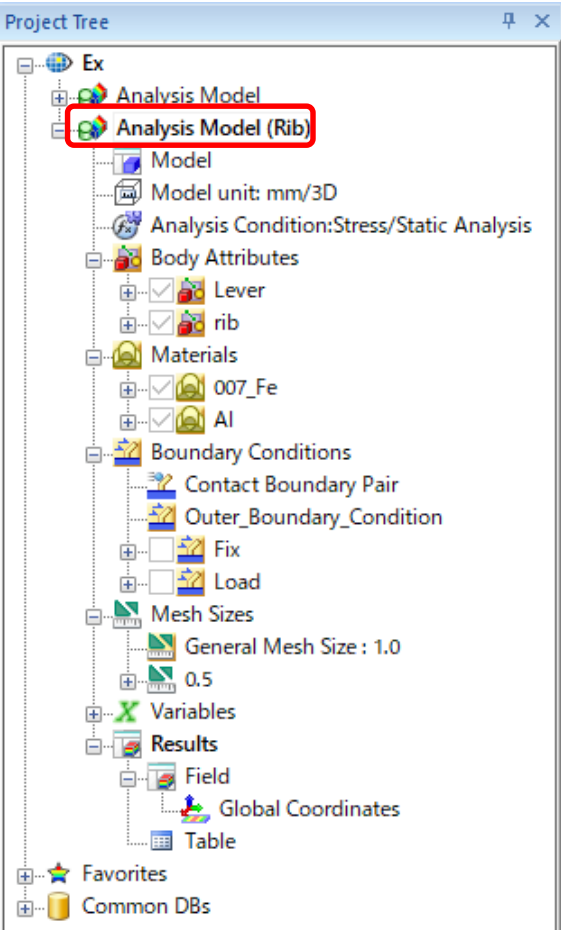
The maximum and minimum values of the color scale is changed as specified on the previous page. Scale adjustment can be done on the displacement tab by double-clicking on the color scale.



Copy Analysis Model

Right-click on [Analysis Model (Rib)] in the project tree. Execute [Copy into Project].

In the dialog box, enter the analysis model name [Analysis Model (Parametric)].

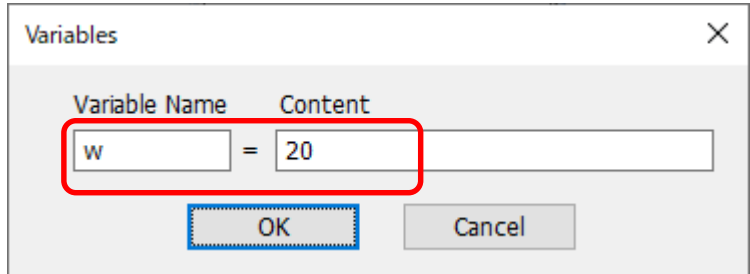
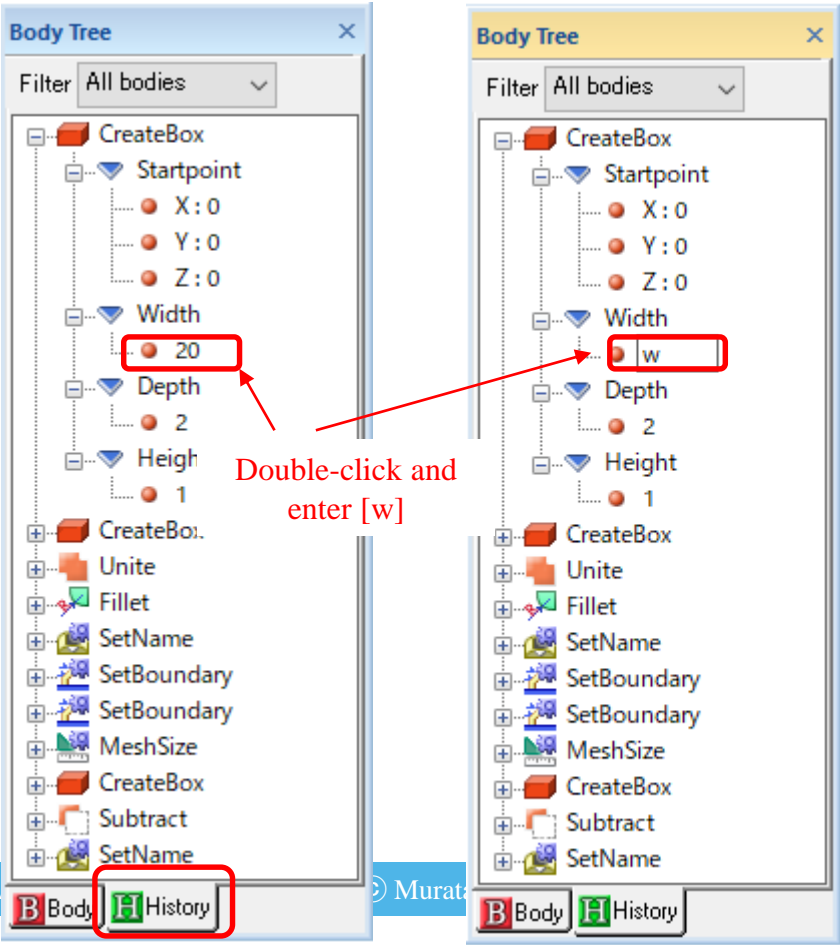


Exercise - Parametric Analysis Murata Software

Parameter Setting

Go to the History tab in the Body Tree.
Modify the value of the width in the 1st Create Box to [w].

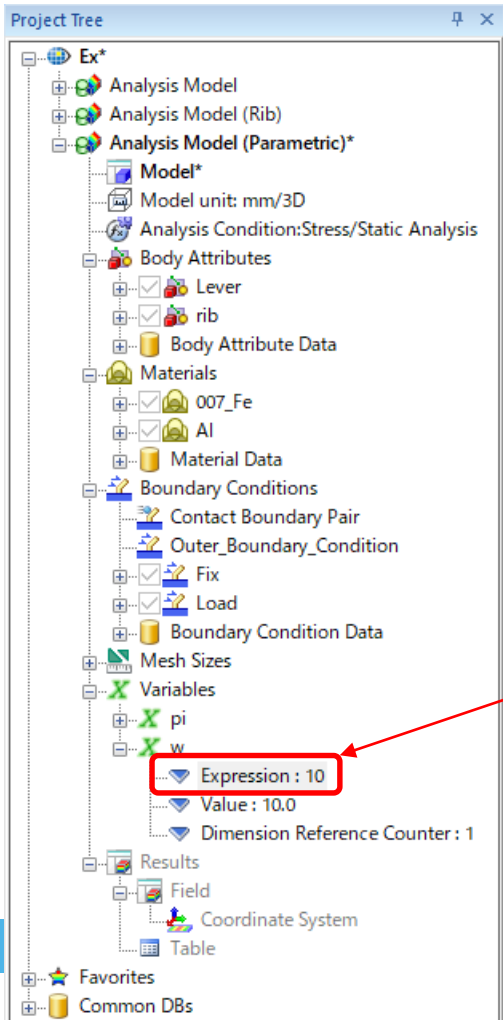
In the dialog box, enter 20



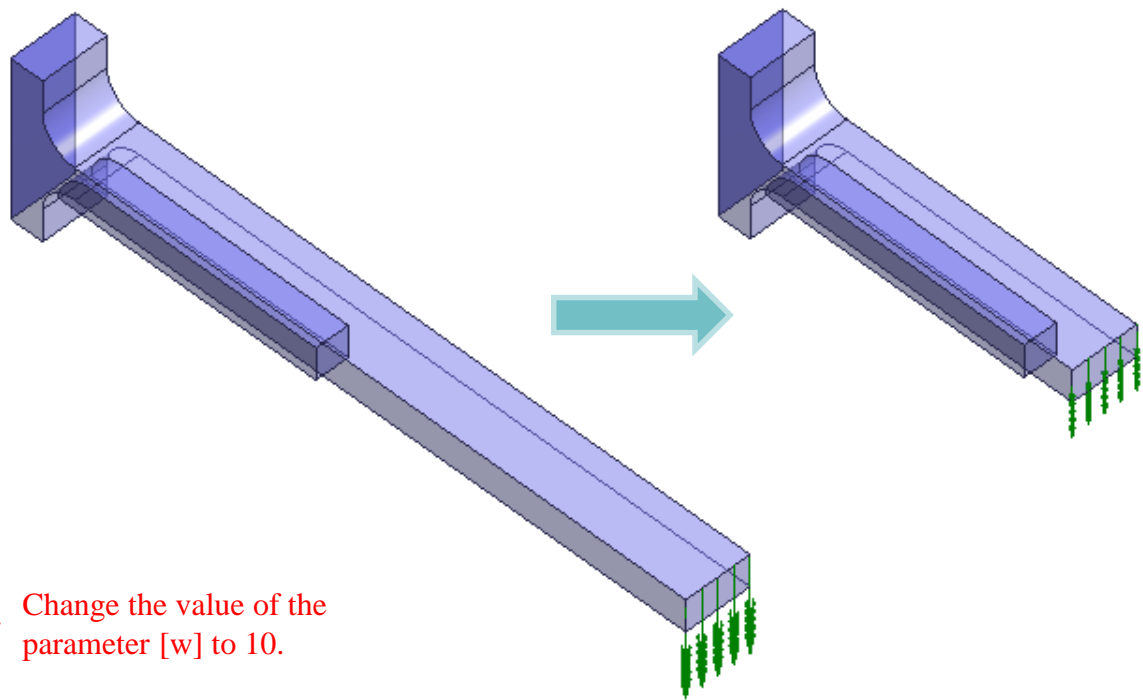
By entering numerical value instead of text, the analysis model is re-created.
In this way, editing the History allows you to re-create the analysis model.

Change Value of Parameter

Confirm the form of the analysis model is changed by modifying the value of the parameter [w].



- Project Tree
 - Ex*
 - Analysis Model
 - Analysis Model (Rib)
 - Analysis Model (Parametric)*
 - Model*
 - Model unit: mm/3D
 - Analysis Condition: Stress/Static Analysis
 - Body Attributes
 - Lever
 - rib
 - Body Attribute Data
 - Materials
 - 007_Fe
 - Al
 - Material Data
 - Boundary Conditions
 - Contact Boundary Pair
 - Outer_Boundary_Condition
 - Fix
 - Load
 - Boundary Condition Data
 - Mesh Sizes
 - Variables
 - pi
 - w
 - Expression : 10
 - Value : 10.0
 - Dimension Reference Counter : 1
 - Results
 - Field
 - Coordinate System
 - Table
- Favorites
- Common DBs

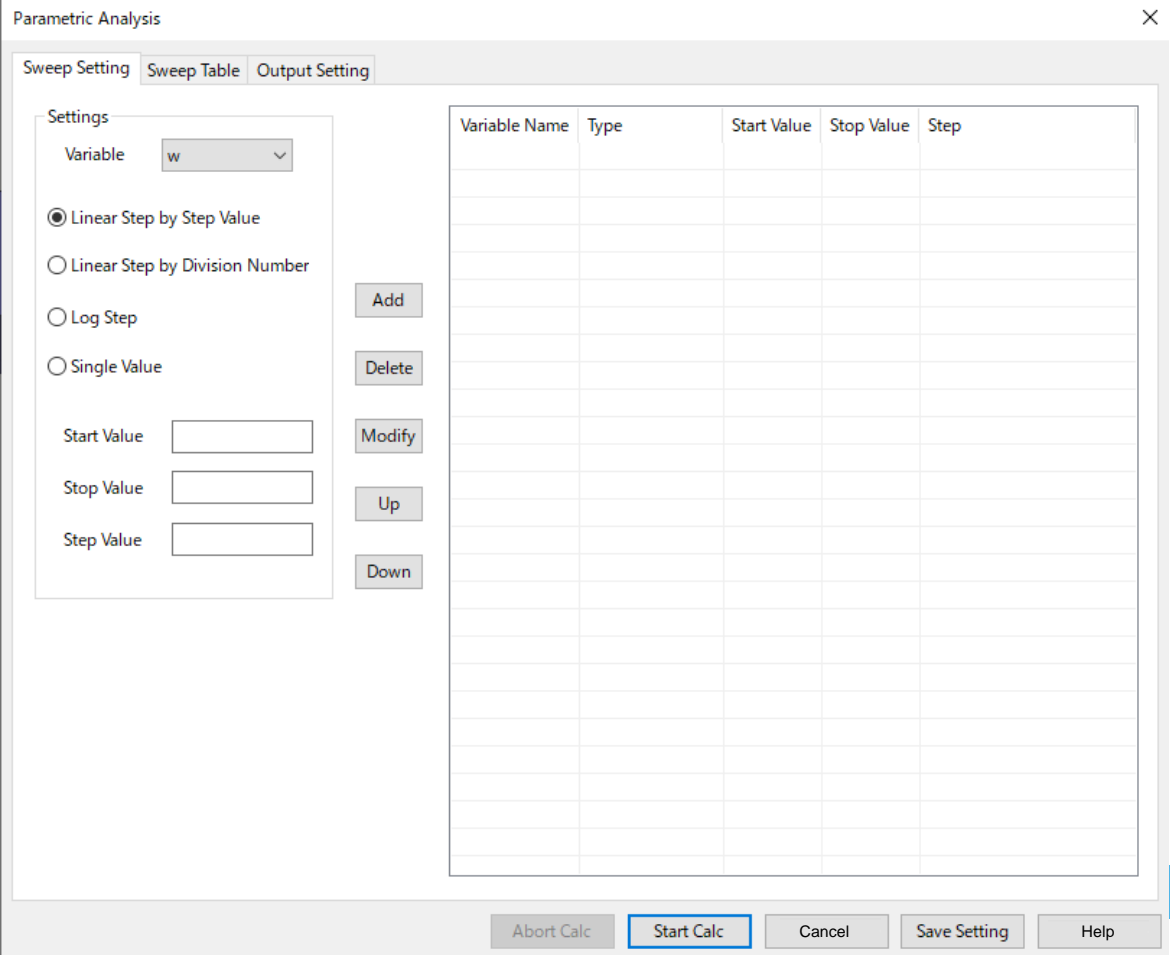
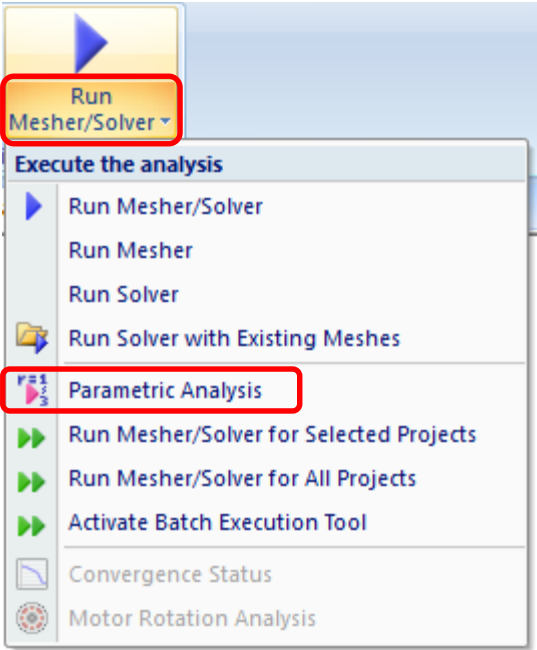


Change the value of the parameter [w] to 10.

Exercise - Parametric Analysis Murata Software

Parametric Analysis Setting

[Model] > [Analysis] > Open submenu of [Run Mesher/Solver].
Select Parametric Analysis.



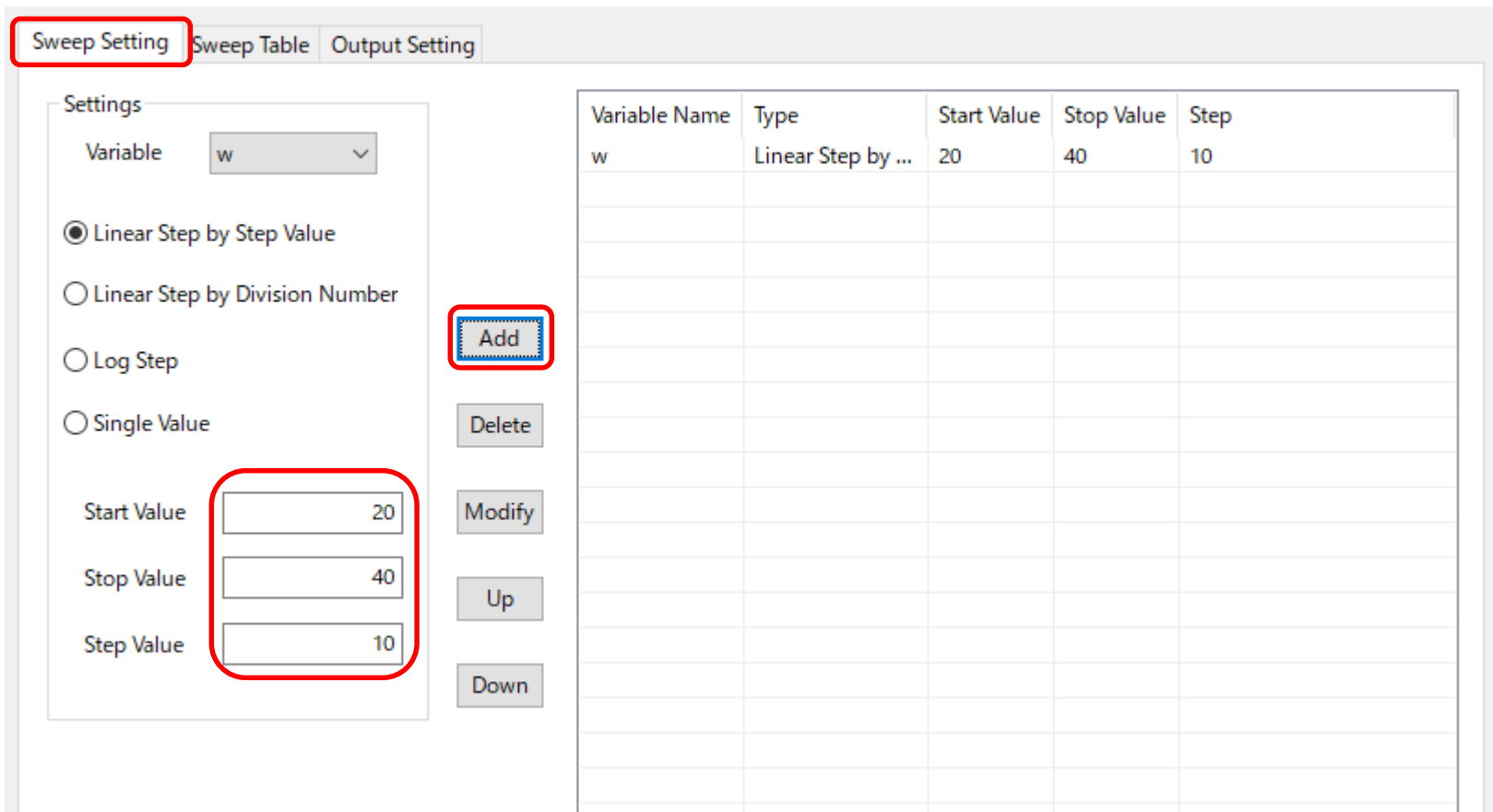
Parametric Analysis Setting

On the Sweep Setting tab, switch to [w] for the Variable setting.

Select [Linear Step by Step Value], enter Start Value [20], Stop Value [40], and Step Value [10].

Press Add button.

Parametric Analysis

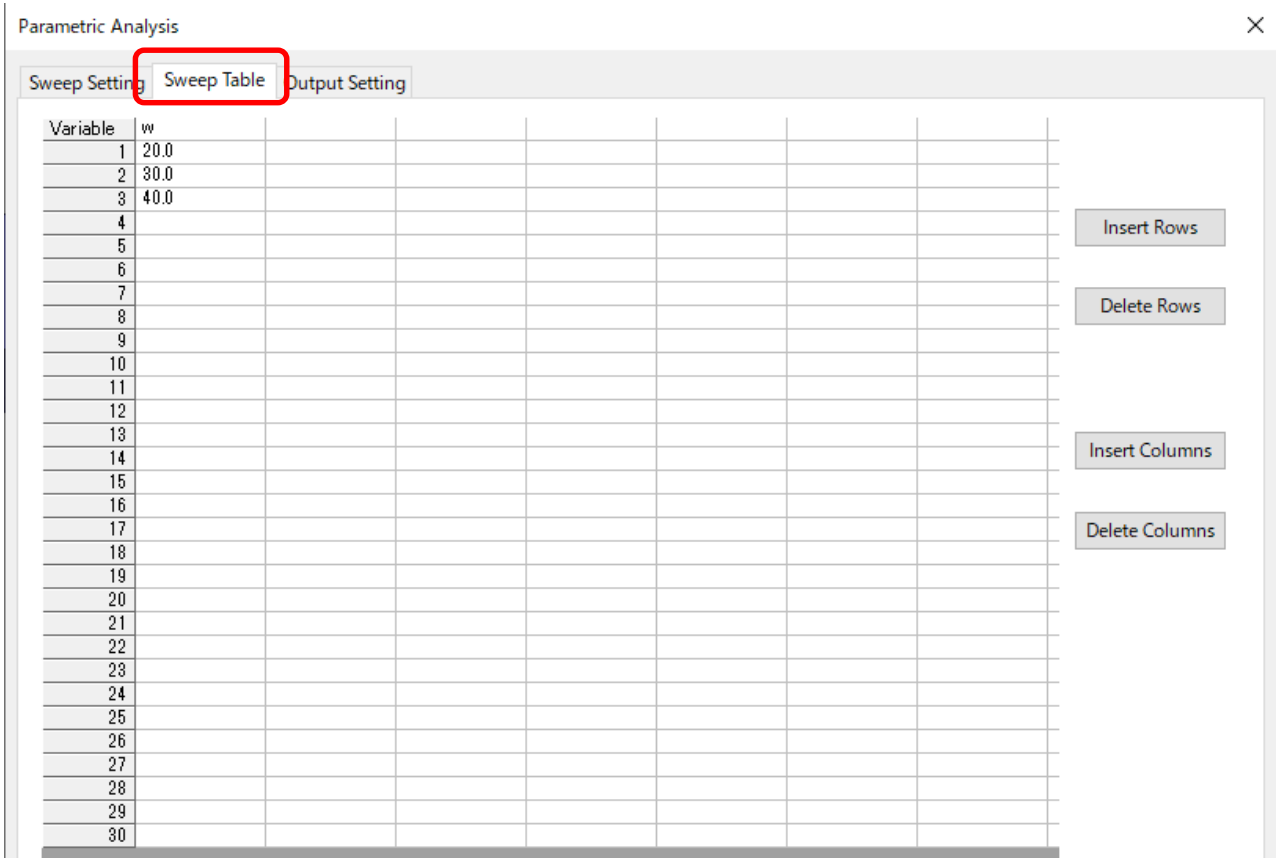


The screenshot shows the 'Sweep Setting' tab of a dialog box. The 'Variable' dropdown is set to 'w'. The 'Linear Step by Step Value' radio button is selected. The 'Start Value' is 20, 'Stop Value' is 40, and 'Step Value' is 10. The 'Add' button is highlighted with a red dashed border. The table below contains the following data:

Variable Name	Type	Start Value	Stop Value	Step
w	Linear Step by ...	20	40	10

Parametric Analysis Setting

Check the combination of variable values on the Sweep Table tab.
Direct edit on the table is available as well.



Parametric Analysis

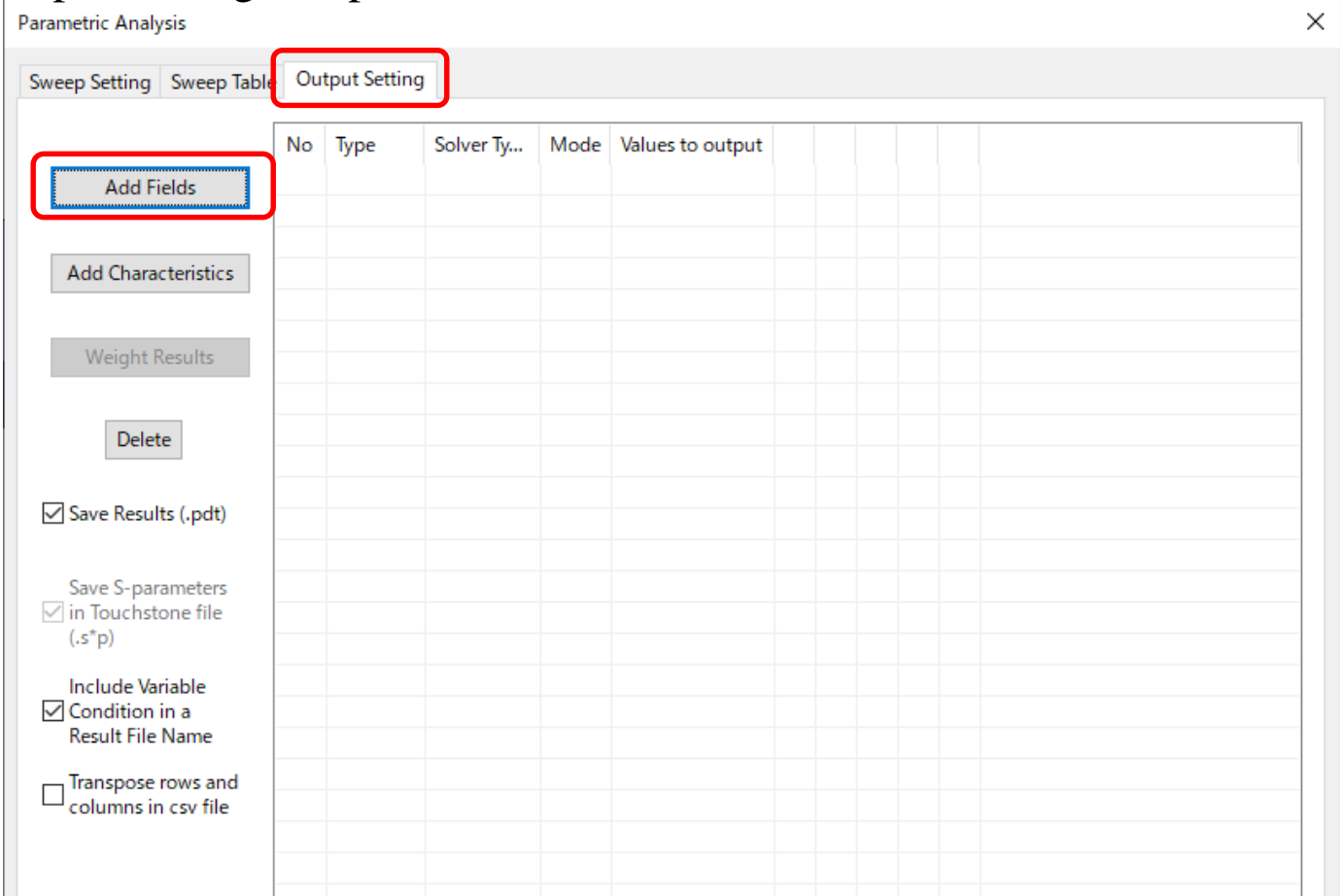
Sweep Setting Sweep Table Output Setting

Variable	w					
1	20.0					
2	30.0					
3	40.0					
4						
5						
6						
7						
8						
9						
10						
11						
12						
13						
14						
15						
16						
17						
18						
19						
20						
21						
22						
23						
24						
25						
26						
27						
28						
29						
30						

Buttons: Insert Rows, Delete Rows, Insert Columns, Delete Columns

Parametric Analysis Setting

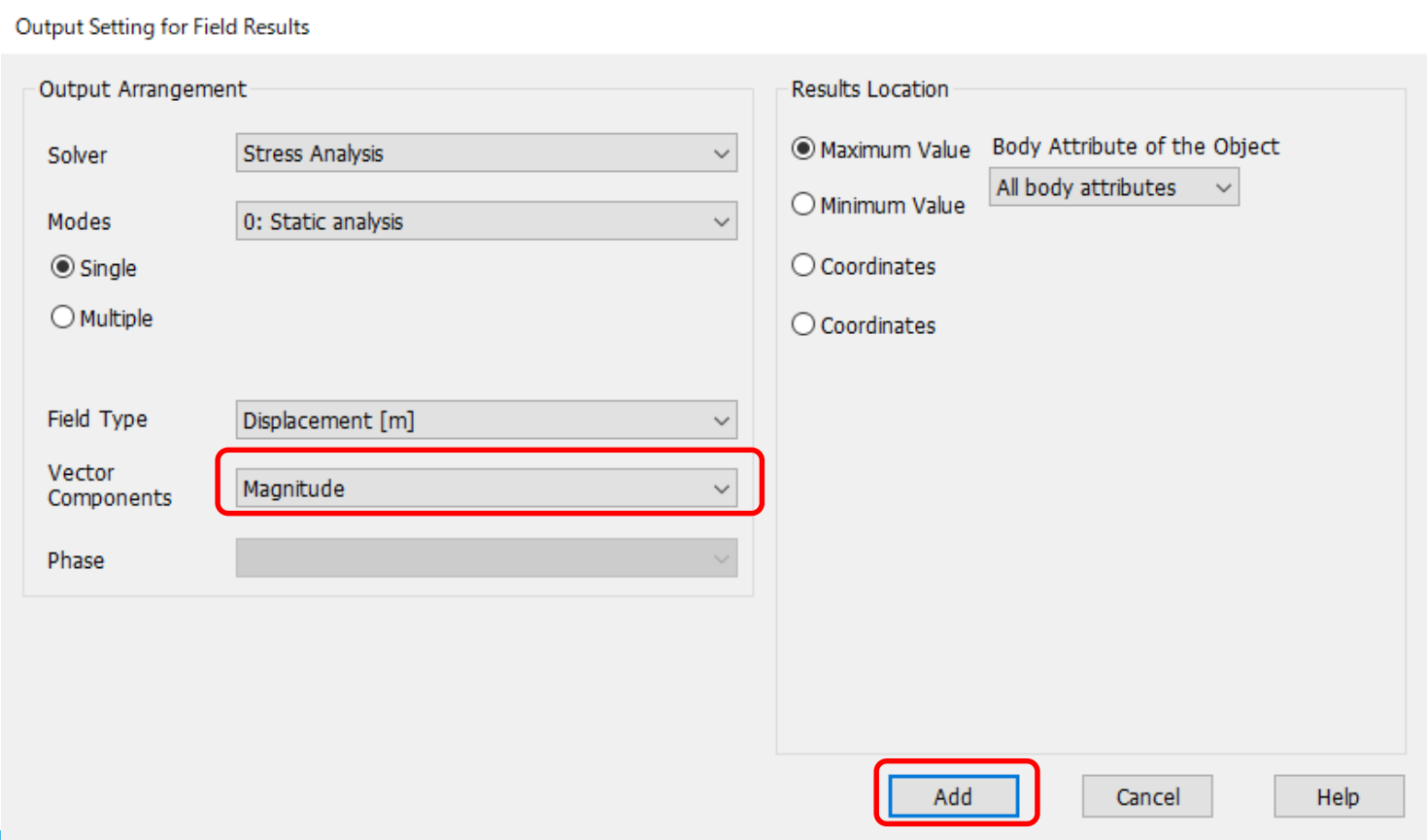
On the Output Setting tab, press Add Fields button.



Exercise - Parametric Analysis Murata Software

Parametric Analysis Setting

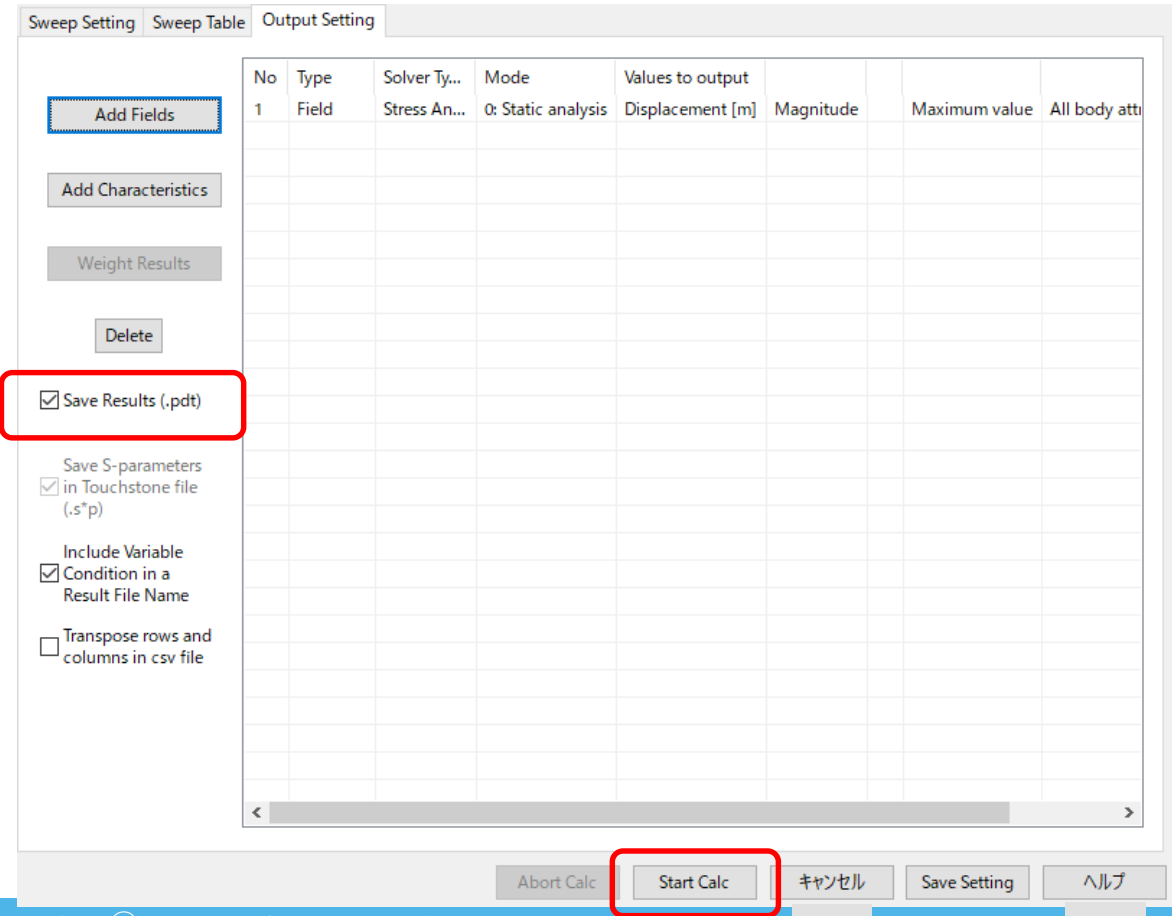
On the Output Setting for Field Results tab, switch Vector Components to [Magnitude]. Press Add button.



Exercise - Parametric Analysis Murata Software

Parametric Analysis Setting

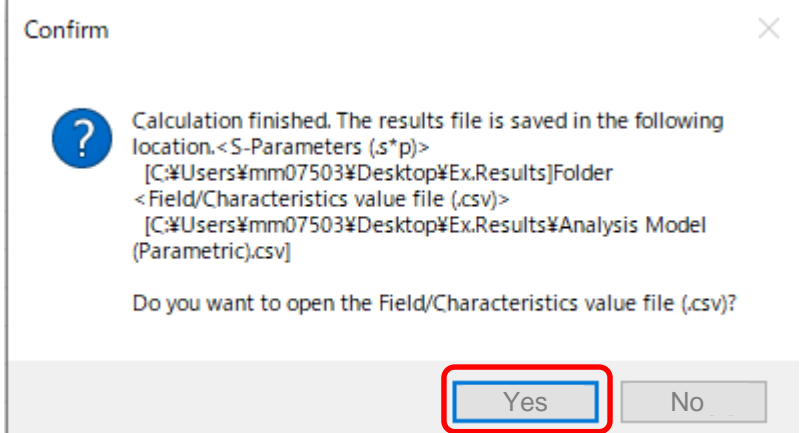
Select [Save Results] and press [Start Calc] button.



Exercise - Parametric Analysis Murata Software

Results Confirmation of Parametric Analysis

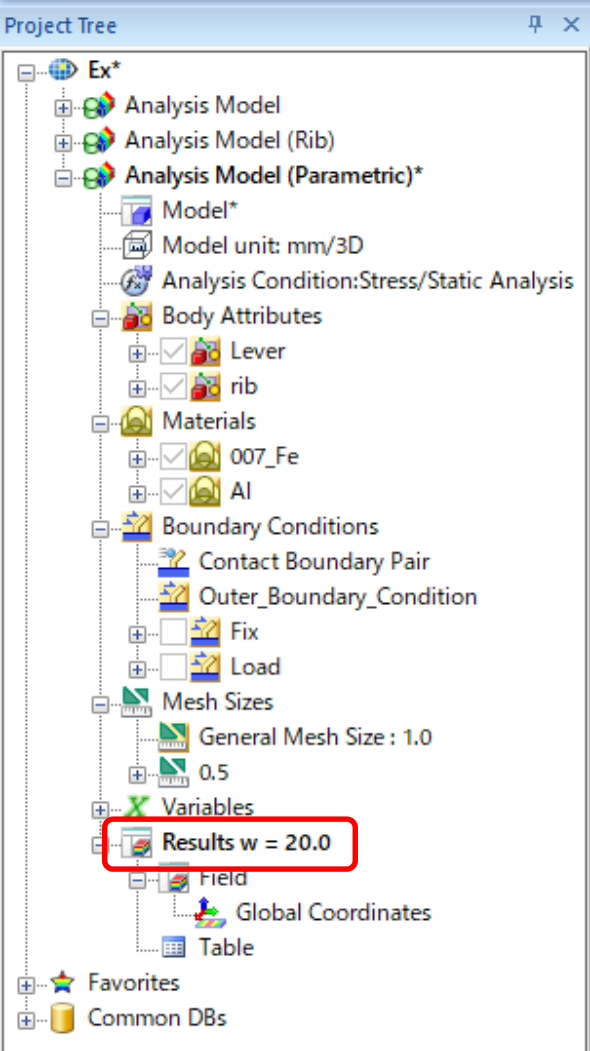
When the message shows up informing the calculation is completed, press [Yes] to see the results on Excel sheet.



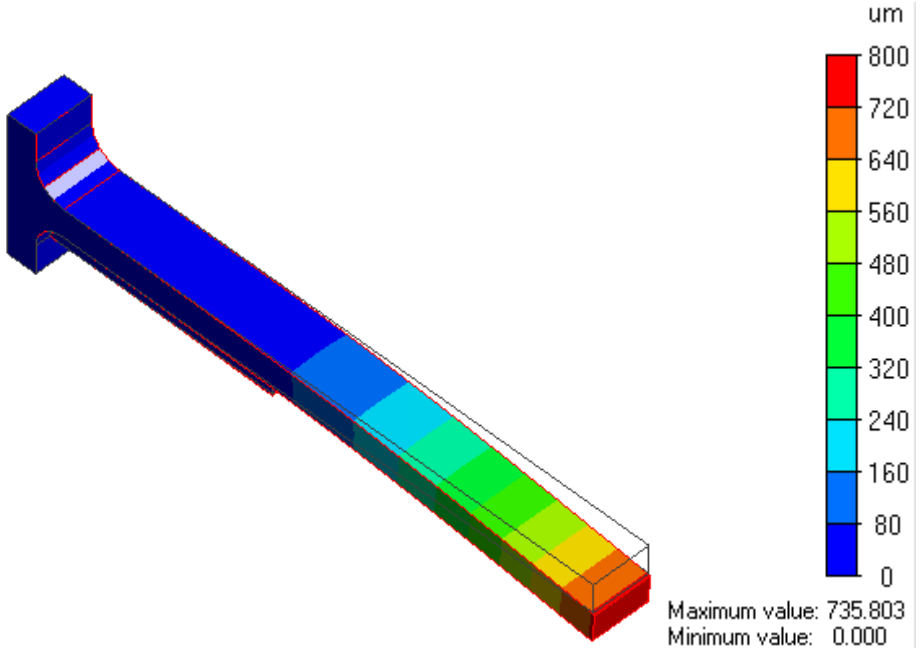
	A	B	C	D	E	F	G	H	I	J	K
1			Solver Type/Mode/Field Type/Component/Phase/Place/Target Body Attribute								
2		Acquired w	Stress Analysis/0: Static analysis/Displacement [m]/Magnitude/ /Maximum value/All body attributes								
3		Acquired X coordinate									
4		Acquired Y coordinate									
5		Acquired Z coordinate									
6											
7	No	w	Value	X coordina	Y coordina	Z coordinate					
8	1	20	7.36E-04	2.00E+01	1.00E+00	1.00E+00					
9	2	30	3.69E-03	3.00E+01	5.00E-01	1.00E+00					
10	3	40	1.06E-02	4.00E+01	1.00E+00	1.00E+00					
11											

Results Confirmation of Parametric Analysis

Double-click on [Results] in the project tree and see the results.

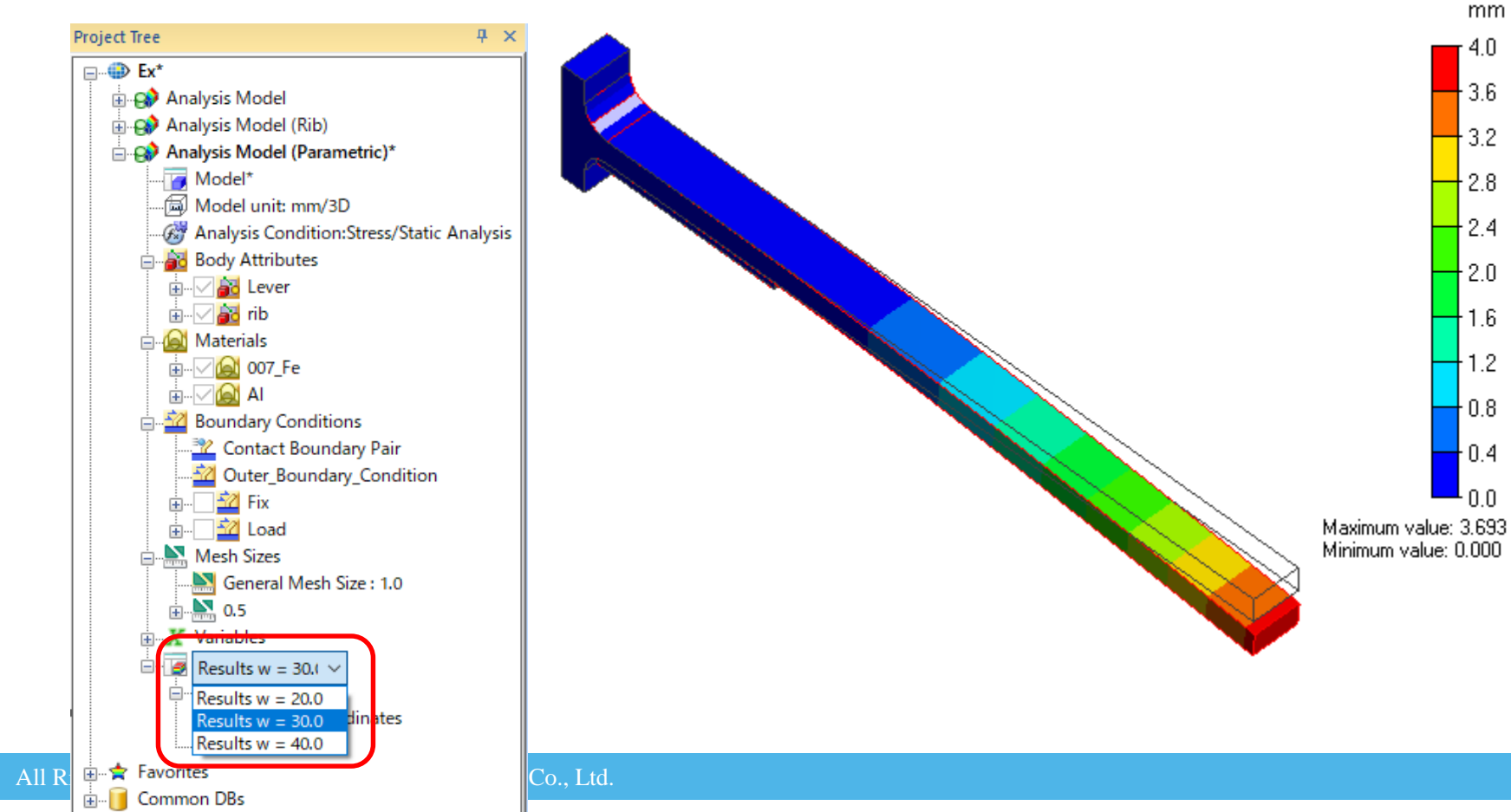


- Ex*
- Analysis Model
- Analysis Model (Rib)
- Analysis Model (Parametric)*
 - Model*
 - Model unit: mm/3D
 - Analysis Condition: Stress/Static Analysis
 - Body Attributes
 - Lever
 - rib
 - Materials
 - 007_Fe
 - Al
 - Boundary Conditions
 - Contact Boundary Pair
 - Outer_Boundary_Condition
 - Fix
 - Load
 - Mesh Sizes
 - General Mesh Size : 1.0
 - 0.5
 - Variables
 - Results w = 20.0**
 - Field
 - Global Coordinates
 - Table
- Favorites
- Common DBs



Results Confirmation of Parametric Analysis

Single-click on [Results] to show the list of combinations of variables.
Select the combination of variables to switch the results.



The screenshot displays a CAD software interface for a parametric analysis. On the left, the 'Project Tree' window shows a hierarchical structure of the model. The 'Results' folder is expanded, showing a list of results for different width values: 'Results w = 30.1', 'Results w = 20.0', 'Results w = 30.0', and 'Results w = 40.0'. The 'Results w = 30.0' item is selected. The main view shows a 3D model of a lever with a color-coded stress distribution. A legend on the right indicates the stress values in mm, ranging from 0.0 (blue) to 4.0 (red). The maximum value is 3.693 mm and the minimum value is 0.000 mm.

Exercise - User Database

Specify the Location for Saving User DB

Application button > [General Settings].

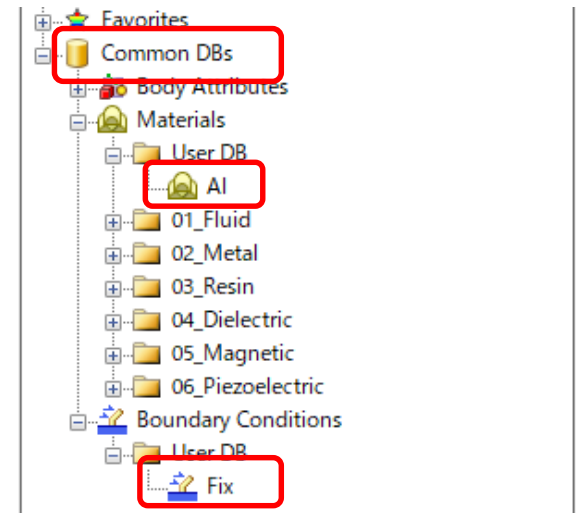
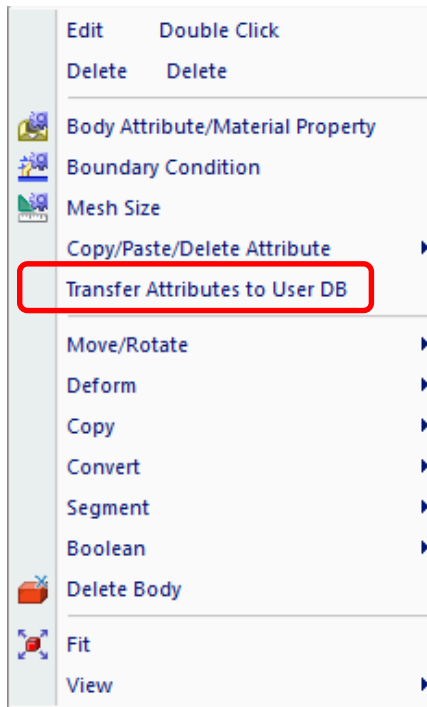
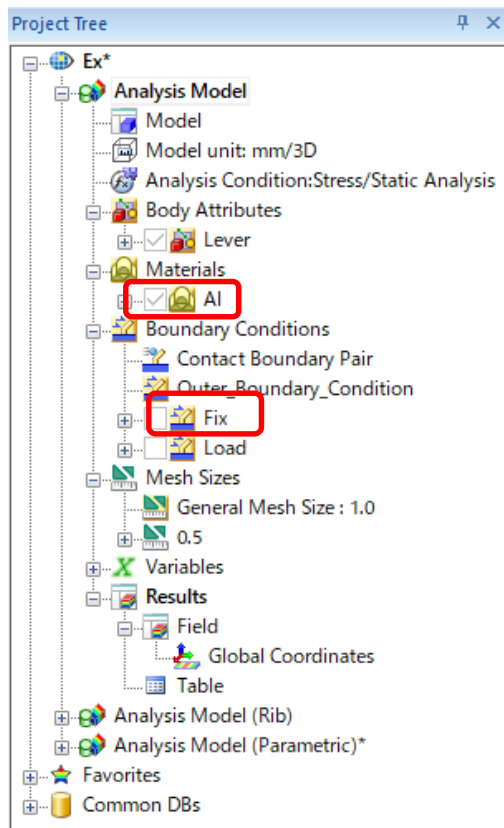
Specify folder to save user DB.

The screenshot displays the 'General Settings' dialog box with the 'Database' tab selected. The 'Folder Name to Save User DB' section is highlighted with a red box, showing fields for 'Body Attribute Data', 'Material Data', 'Boundary Condition Data', and 'Model Data', each with a 'Browse' button. The 'General Settings' menu item is also highlighted with a red box. The 'OK' button at the bottom is also highlighted with a red box.

Exercise - User Database

Register Data in the Database

Open the project and right-click on [AI] in the project tree while showing the model window. Select [Transfer Attributes to User DB]. Transfer [Fix] also in the same way.



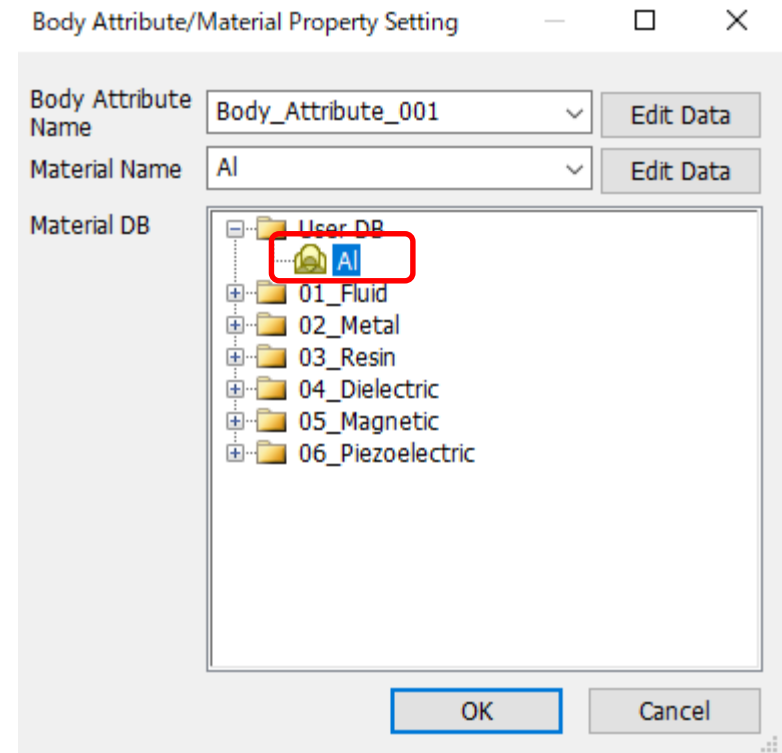
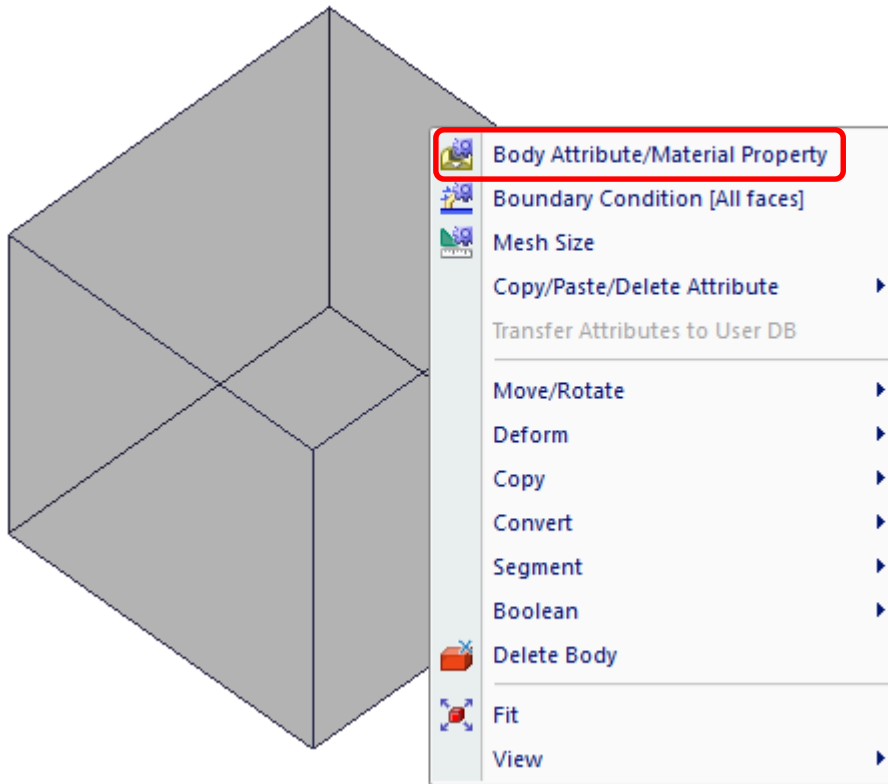
Open [Common DBs] to check the transferred data is registered.

Use Registered Data for Material Property

Create a body in the new project.

Right-click on the body and select [Body Attribute/Material Property]

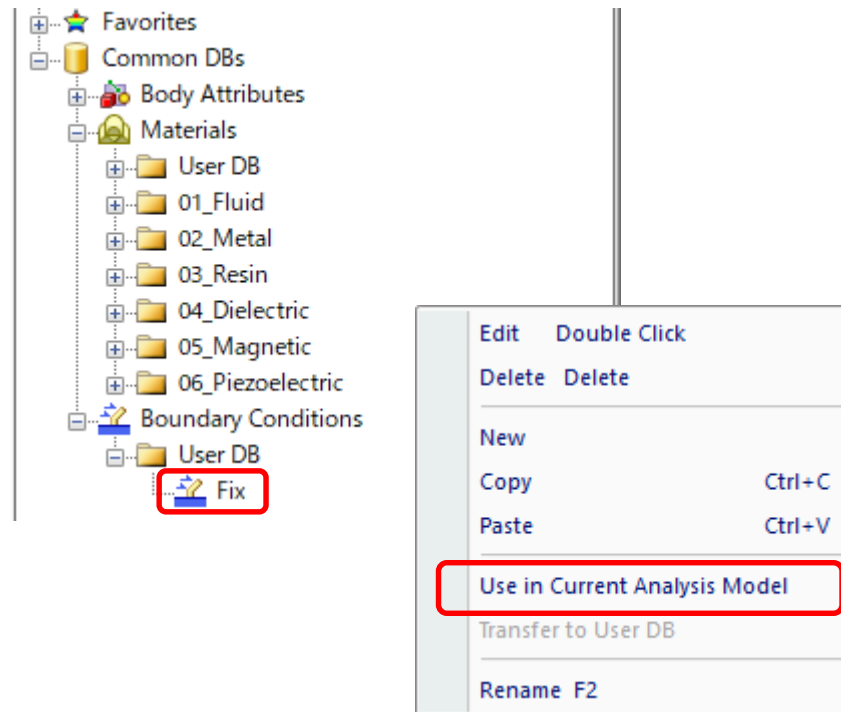
Select the registered material from the material DB.



Exercise - User Database

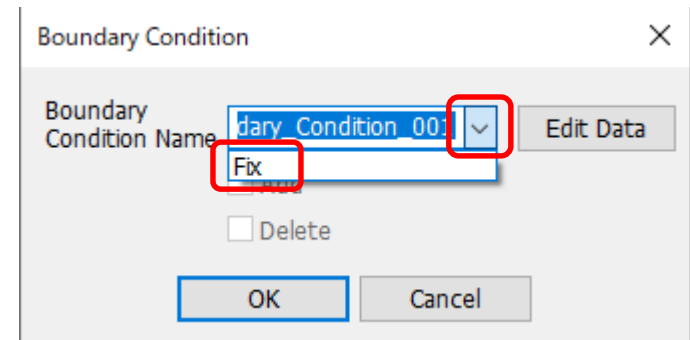
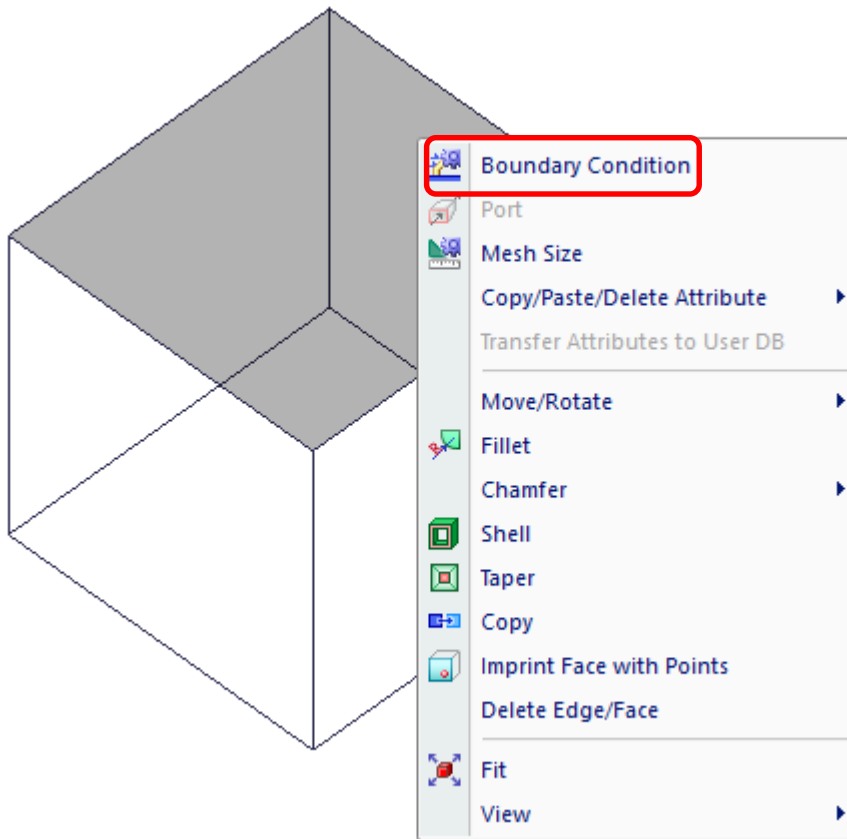
Use Registered Data for Body Attribute and Boundary Condition

In the project tree, select the data to use from the Common DBs.
On the right-click menu, select [Use in Current Analysis Model].



Use Registered Data for Body Attribute and Boundary Condition

Select a face topology of the body. Select [Boundary Condition] on the right-click menu. Click ▼ at the right side of the Boundary Condition Name and select the registered data.

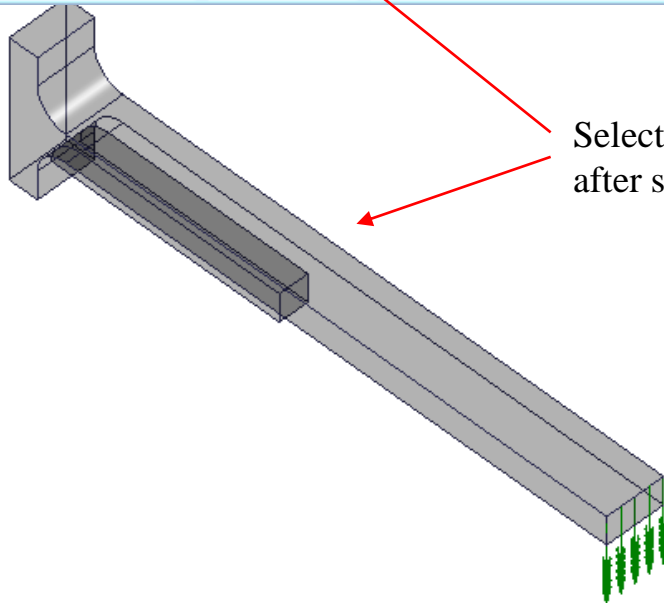
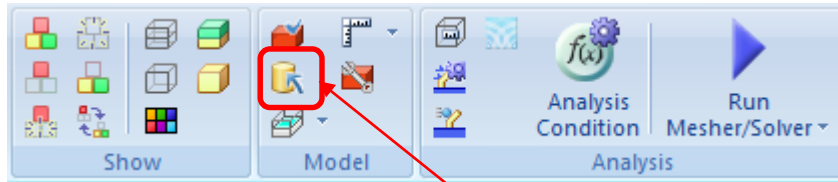


Data Registration of Analysis Model in the Database

Open the project and select all bodies.

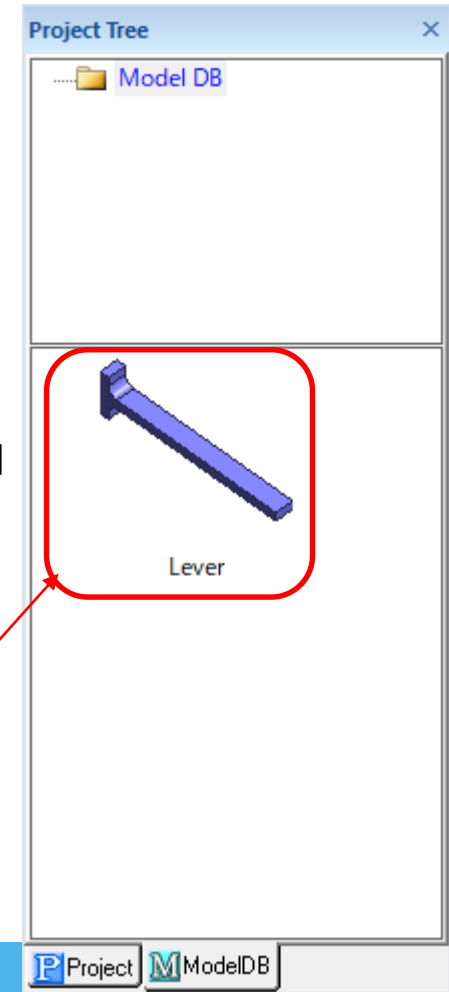
[Model] tab > [Model] group > [Register in Model Database]

The data is registered in the [ModelDB] tab.



Select [Register in Model Database]
after selecting all bodies

Registered



Exercise - User Database

Use Registered Analysis Model Data

Create a new project. Switch the project tree to [ModelDB] tab.
Select an image of data and drag it to the modeling window.

