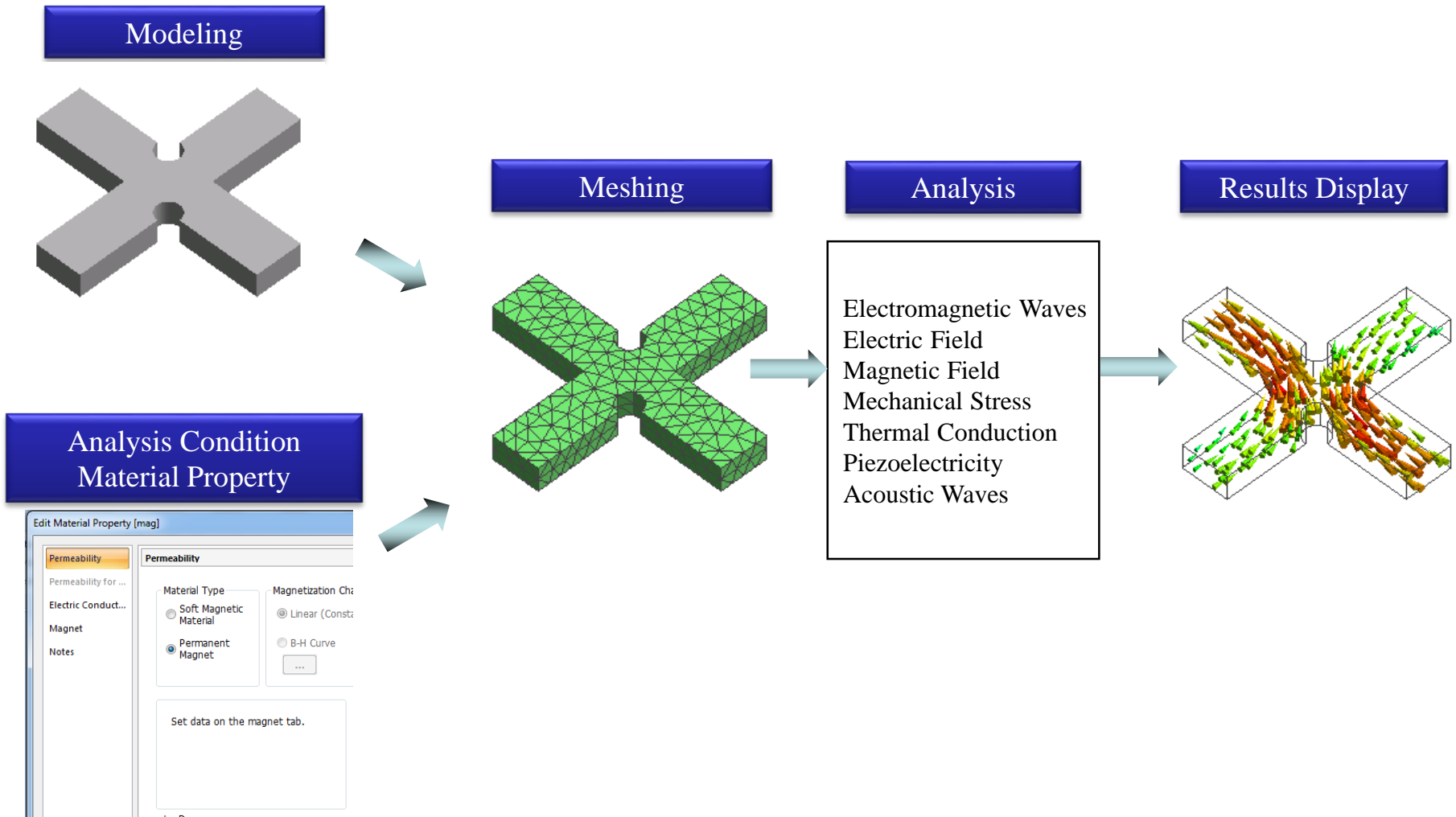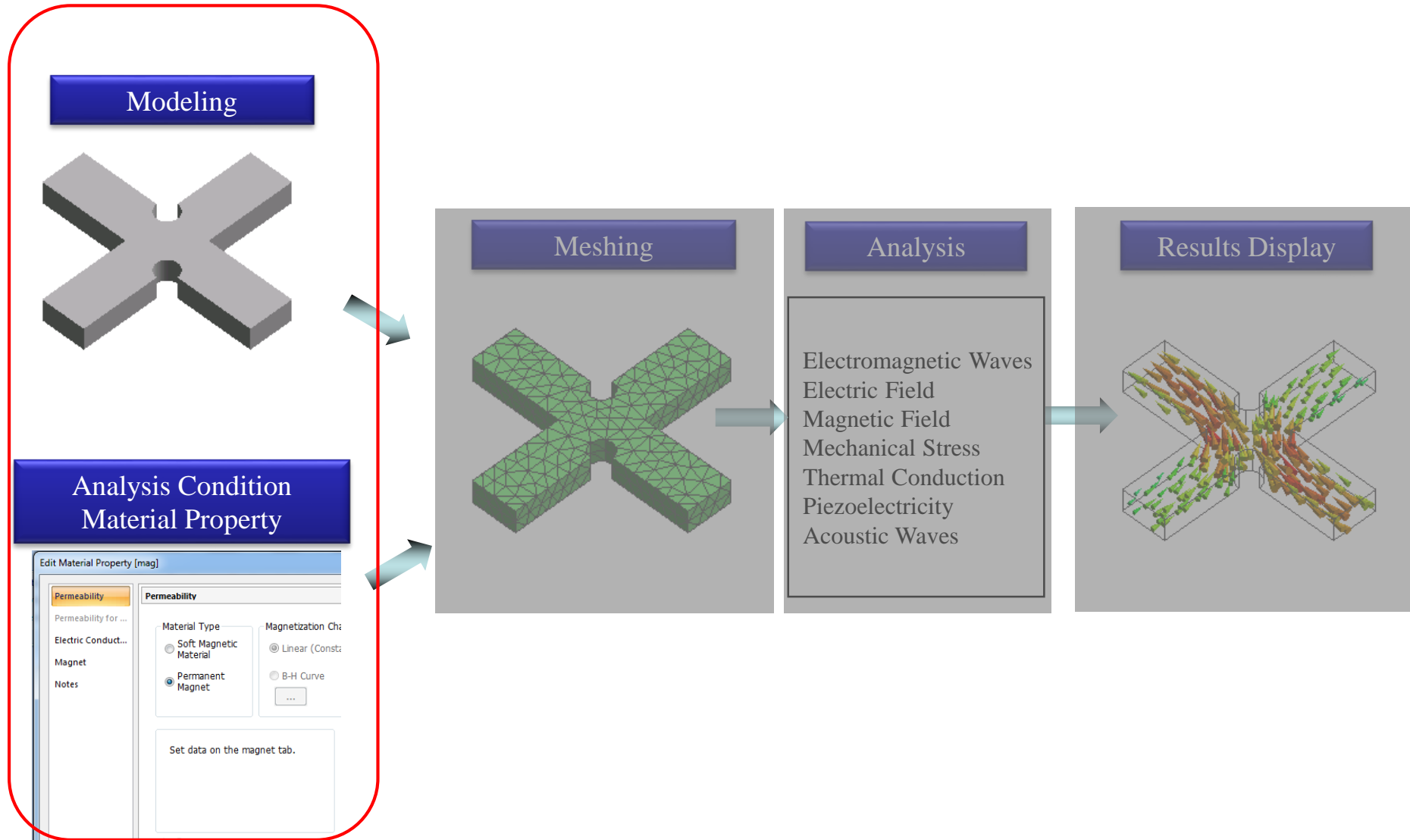# Understanding Femtet
- Basics -

202009

Objectives

To understand the functionalities of Femtet

- Modeling
- Meshing
- Solvers
- Results Display
- Useful Features

General Workflows

Modeling

Meshing

Analysis

Results Display

**Analysis Condition
Material Property**

Edit Material Property [mag]

Permeability	Permeability
Permeability for ...	Material Type	Magnetization Cha
Electric Conduct...	○ Soft Magnetic Material	◉ Linear (Consta
Magnet	◉ Permanent Magnet	○ B-H Curve
Notes		[...]

Set data on the magnet tab.

Electromagnetic Waves
Electric Field
Magnetic Field
Mechanical Stress
Thermal Conduction
Piezoelectricity
Acoustic Waves

Modeling

Modeling

Analysis Condition
Material Property

Meshing

Analysis

Electromagnetic Waves
Electric Field
Magnetic Field
Mechanical Stress
Thermal Conduction
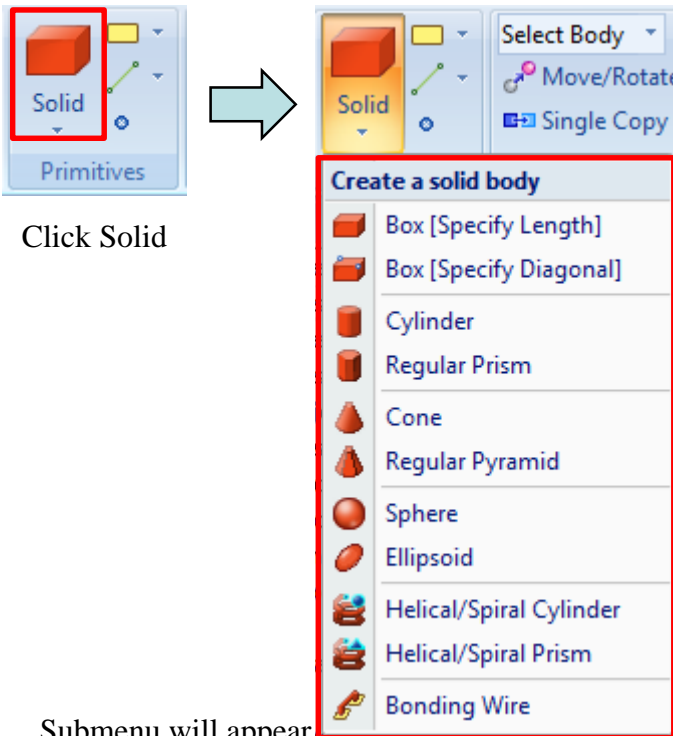Piezoelectricity
Acoustic Waves

Results Display

Modeling

1. Create a basic body.
2. Modify the body (Unite, Subtract, Deform, etc.)
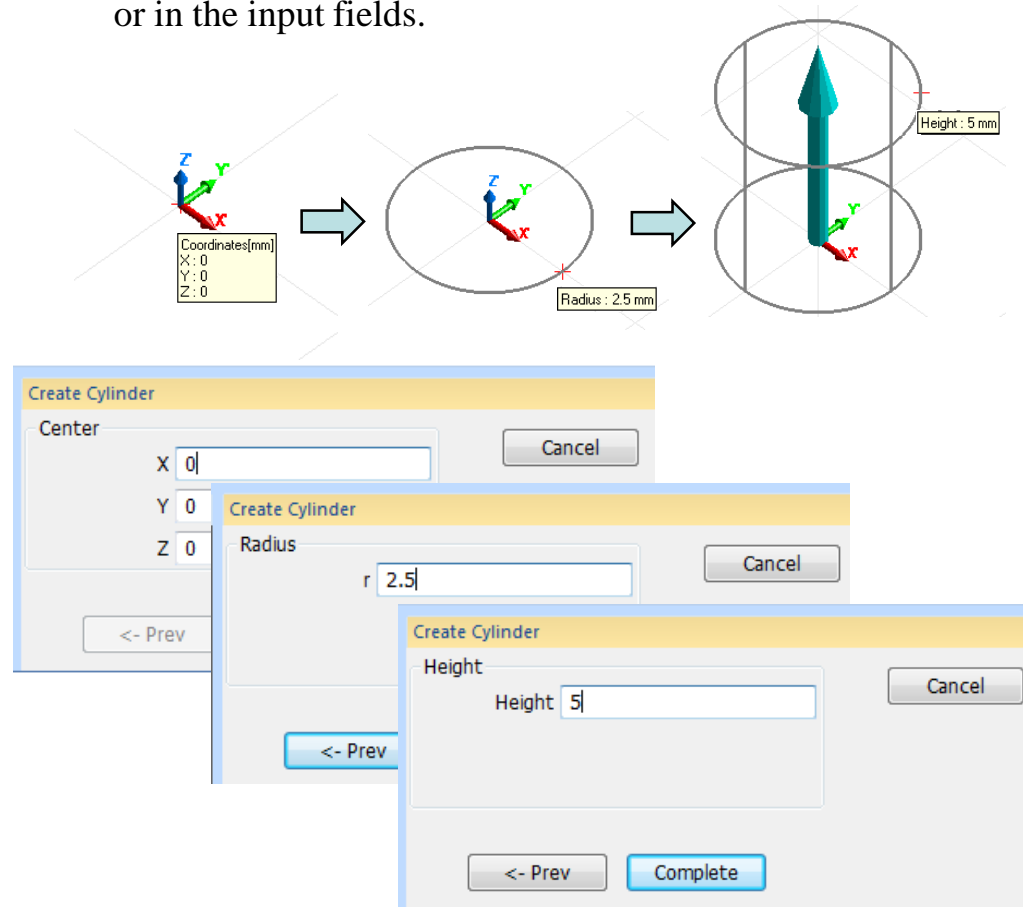3. Setup of body attribute, material property, boundary condition.

Basic Body

Modification

Analysis Model
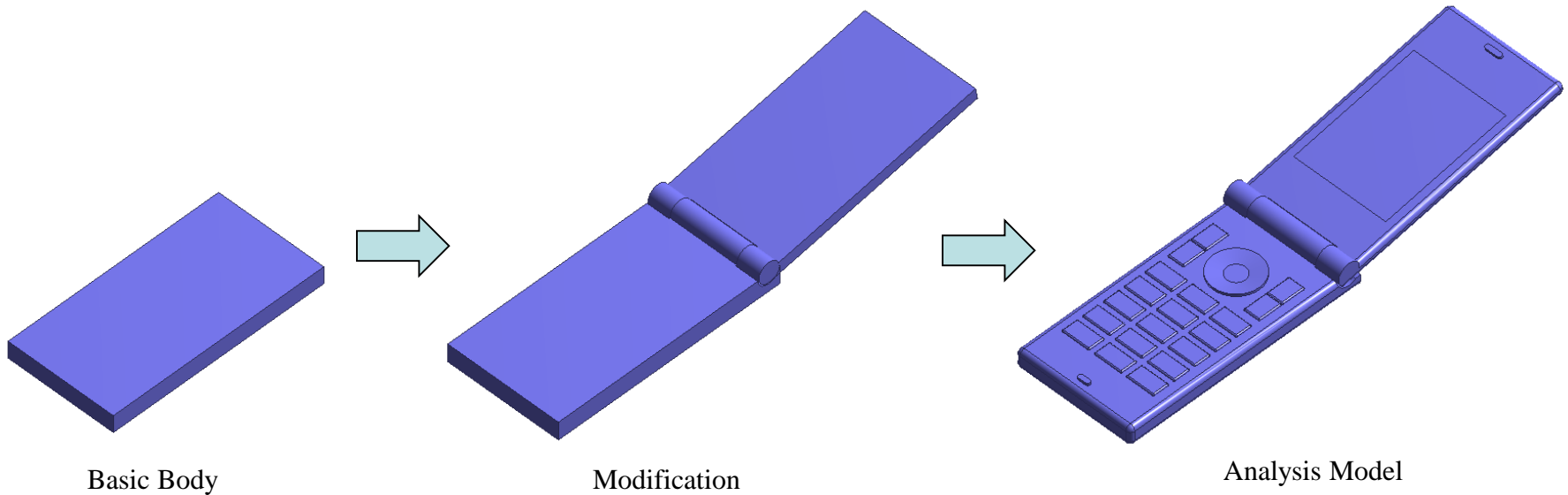
Primitives

Various primitives are available to create a model.

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

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

Click Solid

Submenu will appear

Modeling
Primitives

Various primitives are available to create a model.

Solid Bodies

Box [Specify Length]

Box [Specify Diagonal]

Cylinder

Regular Prism

Cone

Regular Pyramid

Sphere

Ellipsoid

Helical/Spiral Cylinder

Helical/Spiral Prism

Bonding Wire

Wire Bodies

Line

Line [Specify Angle]

Common Tangent Lines of Circles

Arc [Specify 3 Points]

Arc [Specify Center]

Arc [Specify Angle]

Circle

Ellipse

Track

Spline

Helix/Spiral

Polygonal Arc [Specify 3 Points]

Polygonal Arc [Specify Center]

Regular Polygon

Sheet Bodies

Rectangle [Specify Length]

Rectangle [Specify [diagonal]

Circle

Ellipse

Track

Regular Polygon

Polygon

Vertex

Vertex

Enter parameters in each input field or click on a Modeling window.

Create Box
Starting-point
X 0
Y 0
Z 0
Cancel
<- Prev   Next ->

Height : 5 mm
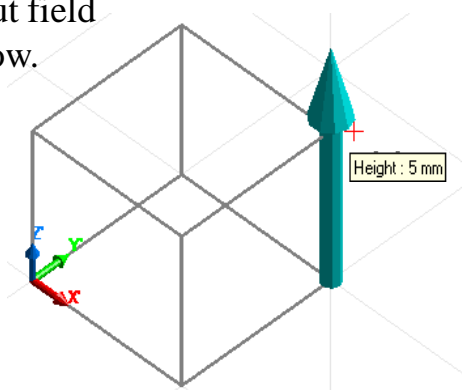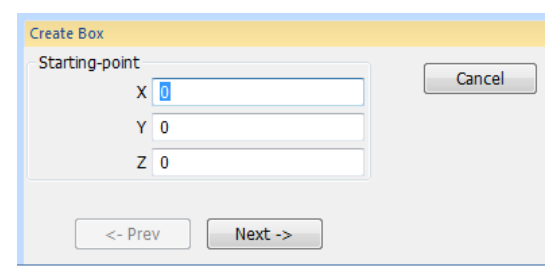
See Help - [Modeling] – [Drawing Process] – [How to Create Primitives]

Drawing Plane

Direction of Parameters Changes

Depending on the drawing plane, the direction of parameters of the commands changes.
In the case of cone, its height direction is normal to the drawing plane.

Drawing planes other than XY, YZ, and XZ can be set as well.
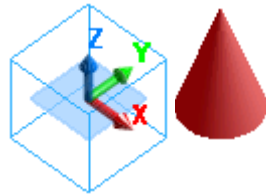
XY Plane
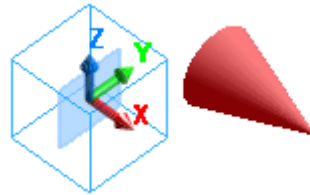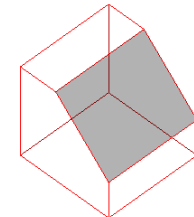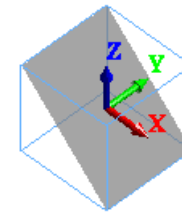
YZ Plane

XZ Plane

User-Specified Plane
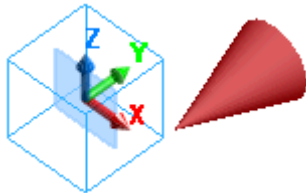
*The direction can be changed also by clicking an axis of normal direction while pressing [Ctrl] key.

**For 2D analysis and axisymmetric analysis, create a model on an XZ plane.

See Help – [Modeling] – [Drawing Process] – [Drawing Plane]

Snap

Snap picks specific coordinates of a model.

When entering parameters for primitives,
enable types of coordinates to pick on the [Snap] tab.
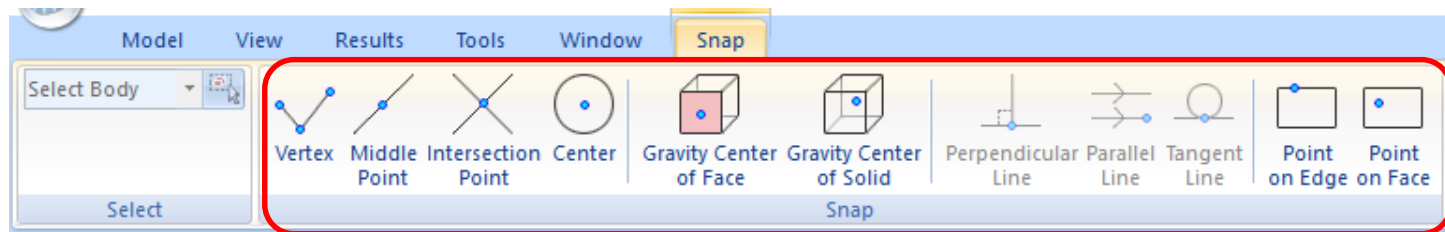
When a mouse is brought near to the specified coordinates,
its values are automatically picked.
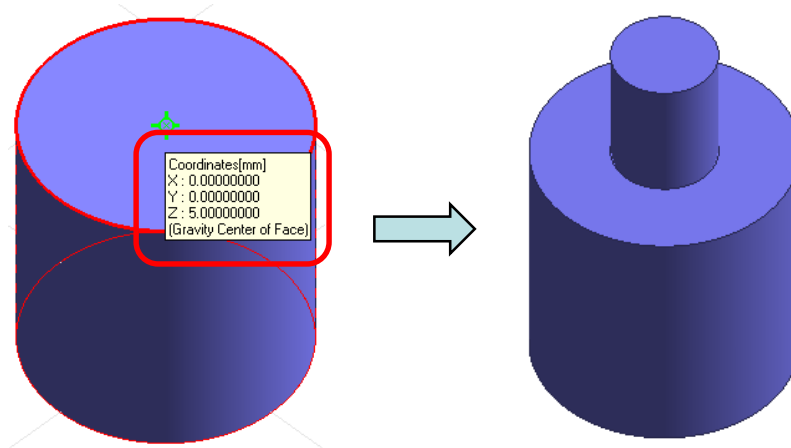
*Multiple snaps are possible.

Snap

Snap picks specific coordinates of a model.

Vertex
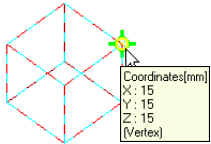
Intersection Point
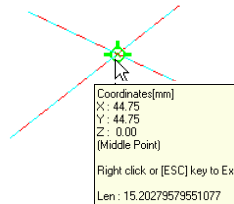
Gravity Center of Face
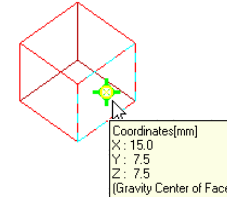
Middle Point
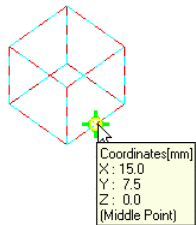
Center
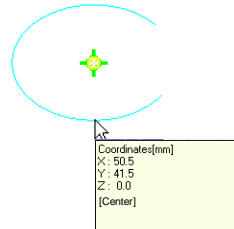
Gravity Center of Solid
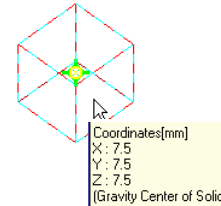
Point on Edge
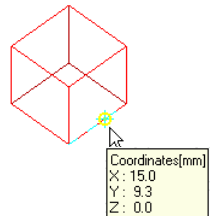
Point on Face
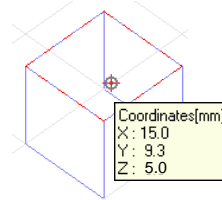
*If multiple snaps are ON, the one nearest to the mouse will be picked.

See Help – [Modeling] – [Drawing Process] – [Snap]

Snap for Lines

When creating a line, the following Snaps are available.
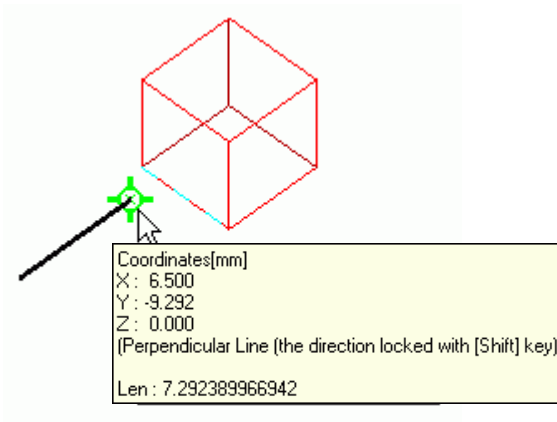
Perpendicular Line	Parallel Line	Tangent Line

Coordinates[mm]
X : 6.500
Y : -9.292
Z : 0.000
(Perpendicular Line (the direction locked with [Shift] key))

Len : 7.292389966942

Coordinates[mm]
X : 19.500
Y : 14.815
Z : 0.000
(Parallel Line (the direction locked with [Shift] key))

Len : 14.31537147234486

Coordinates[mm]
X : 32.863
Y : -8.729
Z : 0.000
(Tangent Line (the direction locked with [Shift] key))

Len : 28.10038926815651
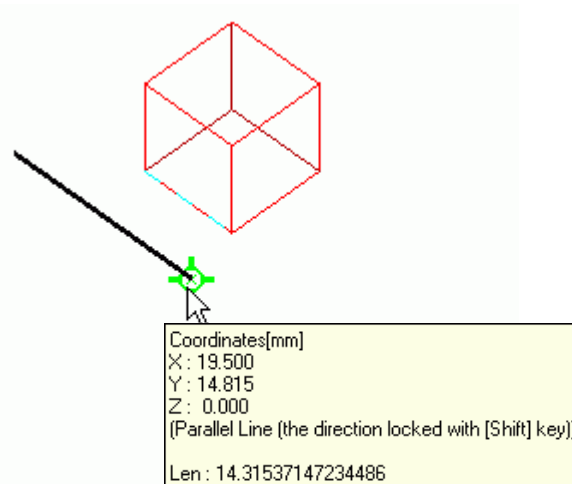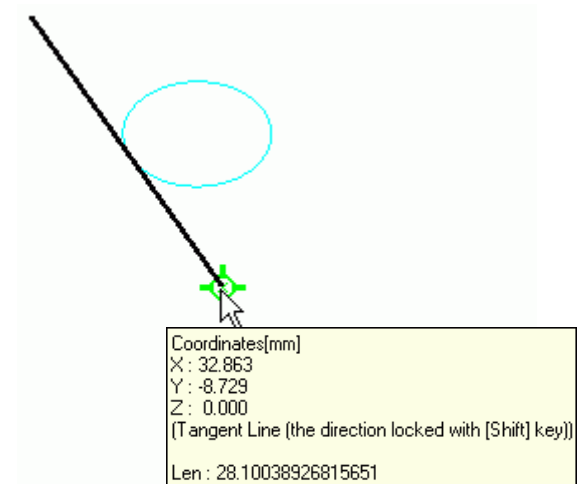
*The line direction is locked if [Shift] key is pressed while snapping.

See Help – [Modeling] – [Drawing Process] – [Snap]

Viewpoint Operation

Viewpoint operation is enabled on the tool panel on a Modeling window or [Model] tab => [Viewpoint Operation].
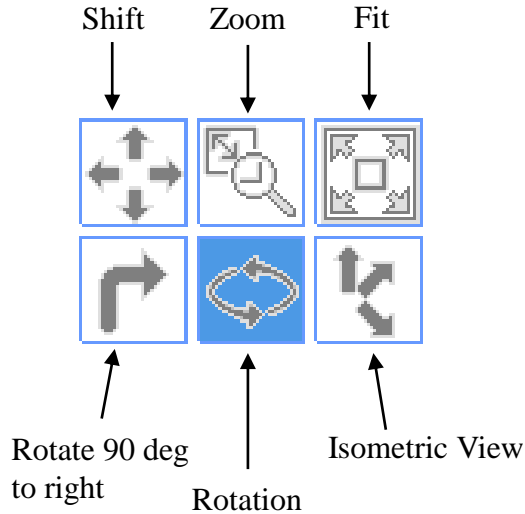
Drag a model while pressing left button of a mouse to get a viewpoint of your wish.

Shift   Zoom   Fit

Rotate 90 deg to right

Rotation

Isometric View

Drag mouse

[Model] tab => [Viewpoint Operation]

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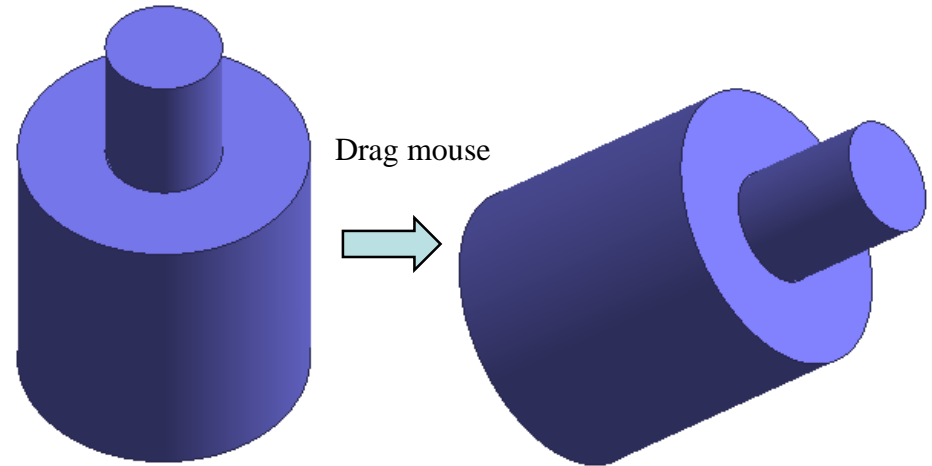
Viewpoint Operation
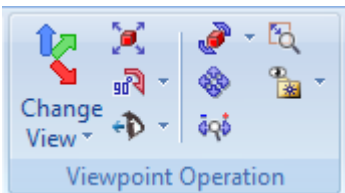
*Some operations, such as [Fit] and [Isometric View], can be executed simply by selecting the menu.

12

Body/Topology Selection

Murata Software

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).

Select Body   Select Face

Select Edge   Select Vertex

[Model] tab ⇒ [Modification Operation]

The target object will be highlighted when a mouse is brought to it.
Left-click it to complete the selection.

Highlighted

Face topology
<topology ID/Body key>: 15/1
<Area>: 3.14159 mm2
<Gravity Center of Face>: 0.0, 0.0, 7.5 mm

Selected

Left-click while highlighted
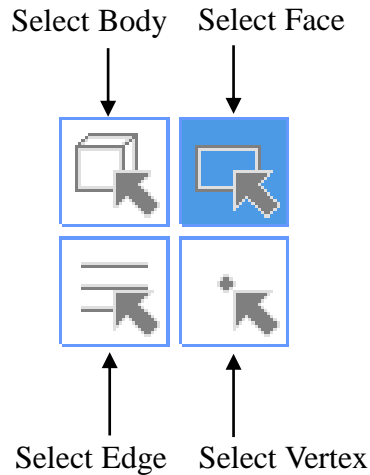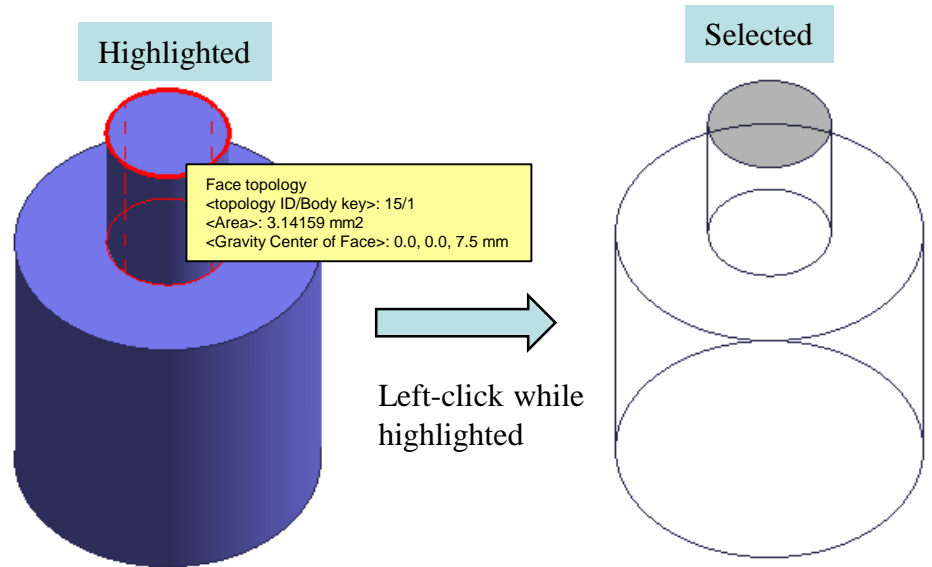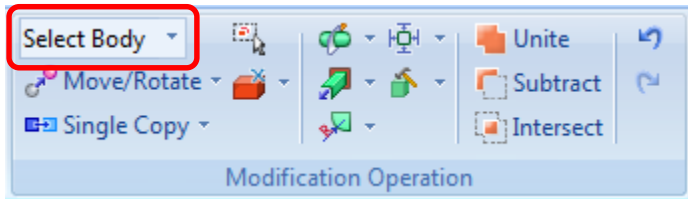
*Be aware that left-click is needed to complete the selection.
**Left-click + [Ctrl] key will allow multiple selection.

Select Body
Move/Rotate   Unite
Single Copy   Subtract
Intersect

Modification Operation

13

Modeling
Body/Topology Selection

Change Target Object

Useful to select a part of body which is hidden from the view.

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

Press a spacebar.
A target object can be switched.

Highlighted

Highlighted

Selected

Face topology
<Topology ID/Body key>: 2 / 1
<Area>: 303.89319 mm2
<Gravity Center of Face>: 8.0, -2.5, 5.7 mm

[Space] key to switch options
[TAB] key to show the option list

Press a spacebar

Face topology
<Topology ID/Body key>: 15 / 1
<Area>: 56.54867 mm2
<Gravity Center of Face>: 8.0, -2.5, 11.4 mm

[Space] key to switch options
[TAB] key to show the option list

Left-click while
highlighted

Selection list   x

Body1 / Face(2)
Body1 / Face(9)
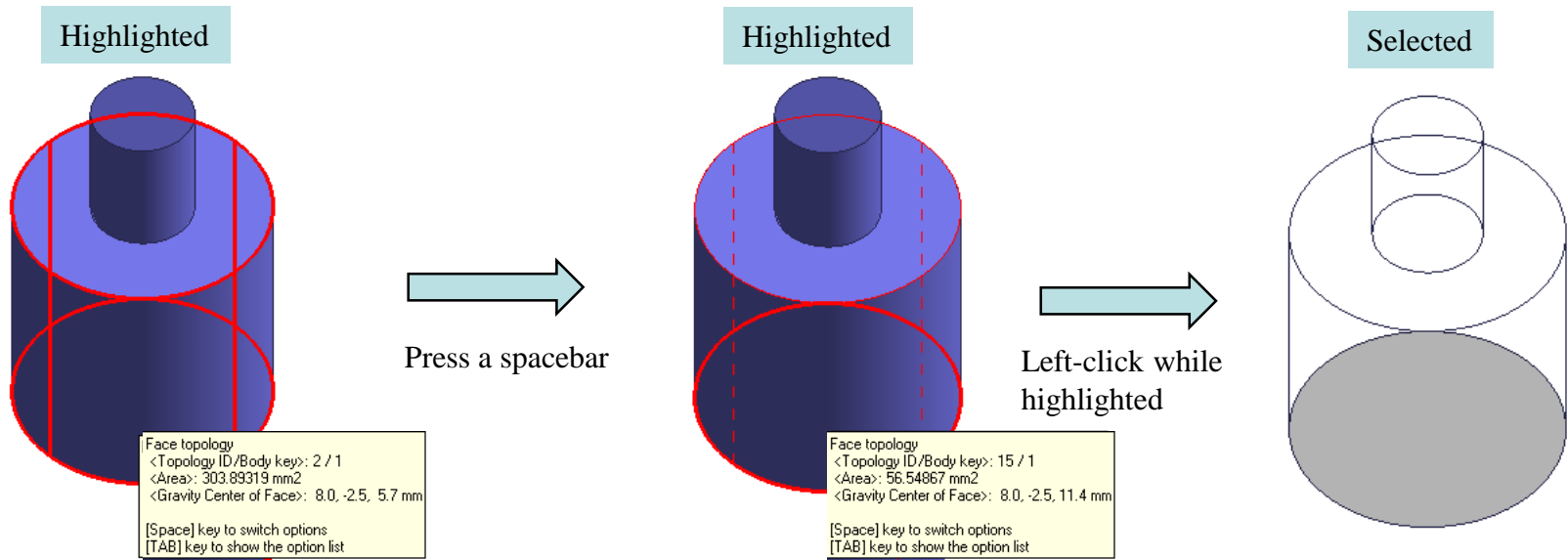
The function is useful when you want to select a place where bodies that are overlapping, or hidden faces.
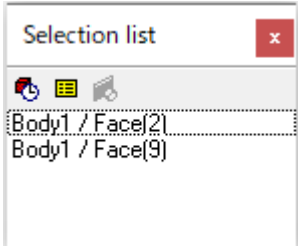
*Press [Tab] key to show Selection list.

Body/Topology Selection

Selection from Body Tree

Bring mouse to an item on a Body Tree.
A corresponding body is highlighted.

Left-click the item on a Body Tree.
The body is now selected.

Highlighted

Selected

Body/Topology Selection
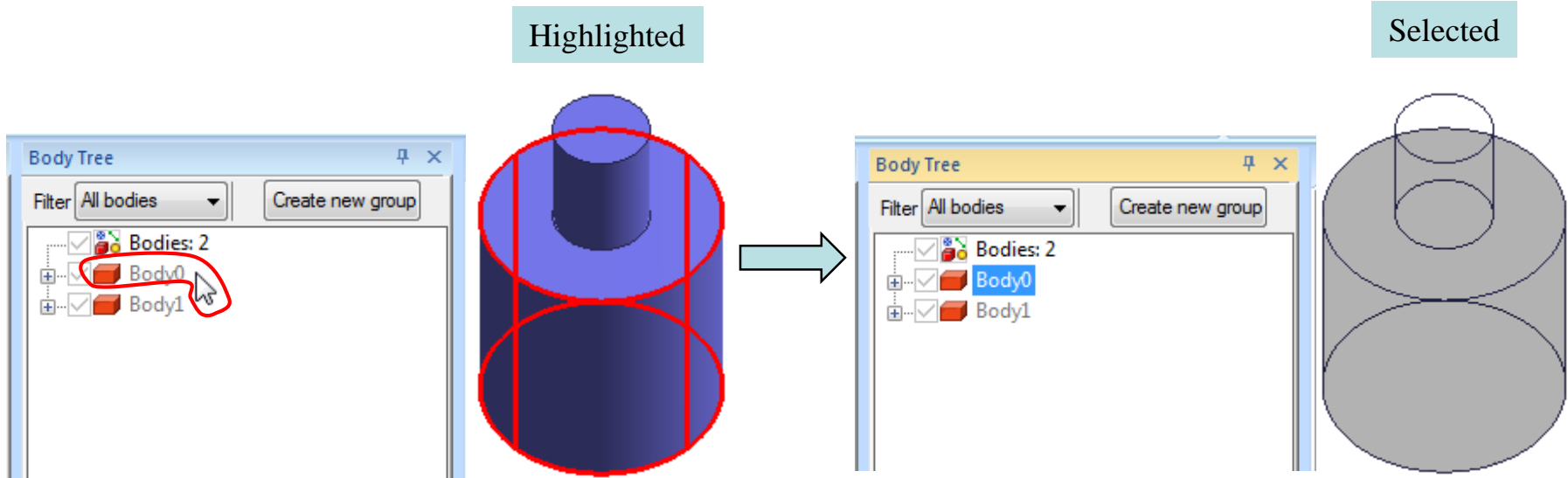
Area Selection

Drag mouse on a Modeling window.
A dotted marquee will appear.
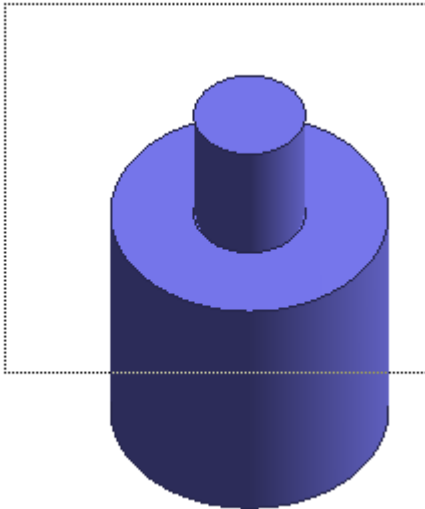
If [Area selection] is OFF, bodies which are contained even partially in the marquee will be selected.
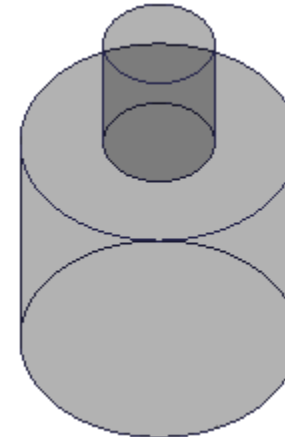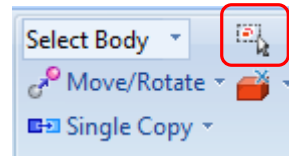
If [Area selection] is ON, only bodies which are wholly contained in the marquee will be selected.

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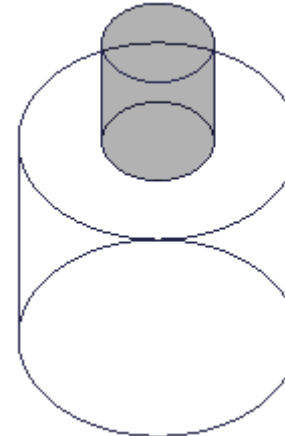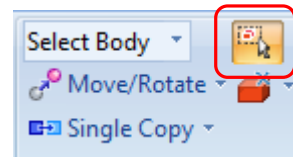
Right-click on a modification target (body or part of body).
Select operation from an appearing menu or
go to[Model] tab => [Modification Operation].

Enter parameters in an input box.

Right-click menu (on a selected face)

Move Face

Boundary Condition
Port
Mesh Size
Copy/Paste/Delete Attribute ▶
Transfer Attributes to User DB

Move/Rotate ▶
Fillet
Chamfer ▶
Shell
Taper
Copy
Imprint Face with Points
Delete Edge/Face

Fit
View ▶

[Model] tab => [Modification Operation]

Select Body ▾   Unite ↩
Move/Rotate ▾   Subtract ↪
Single Copy ▾   Intersect

Modification Operation

Move
Move vector
X 0
Y 0
Z -2
<- Prev   Complete   Cancel

Model Modification

Murata Software
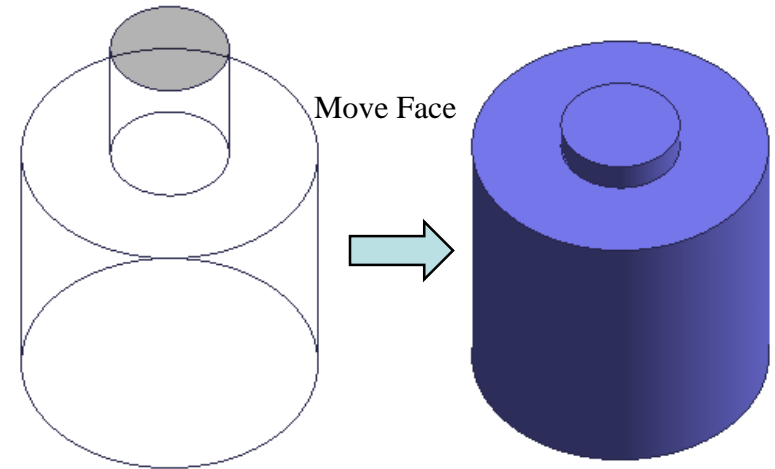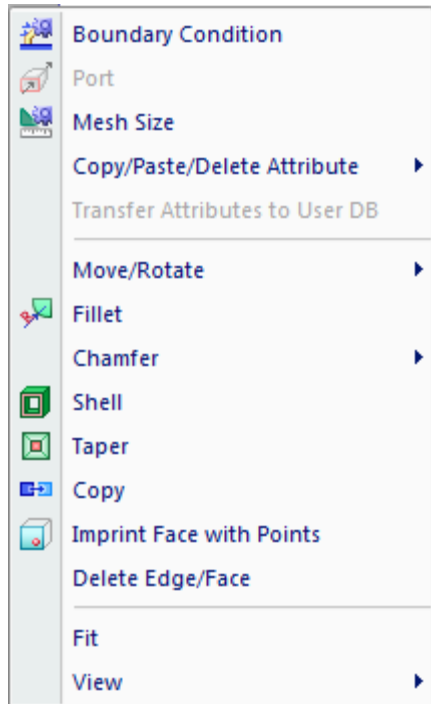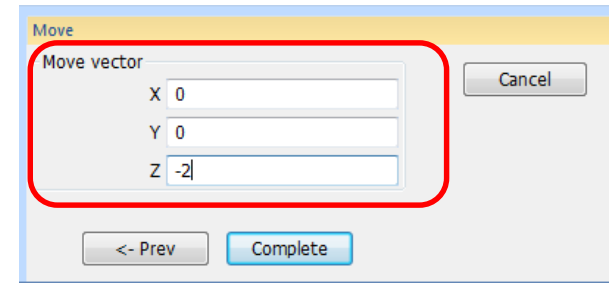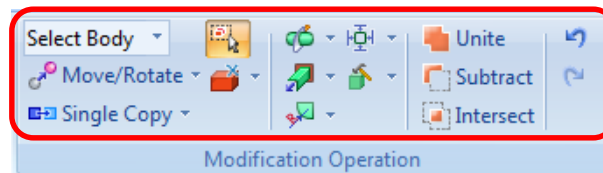
Commands are available to create various forms of model.

Boolean

Unite

Subtract

Copy

Fillet

Extrude
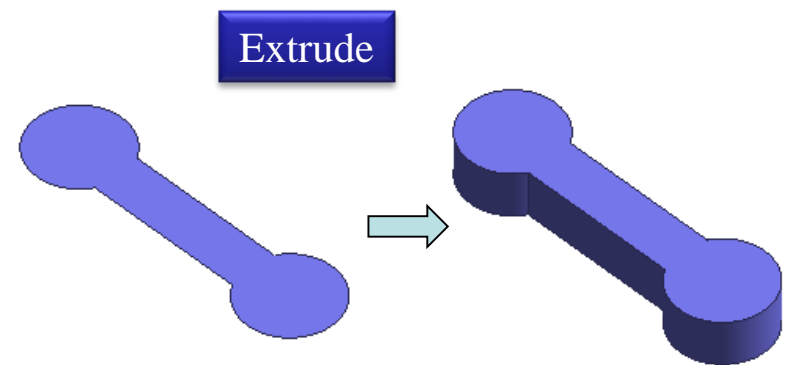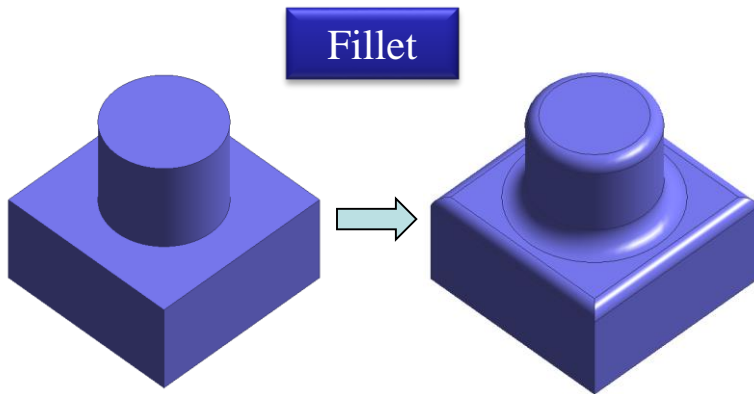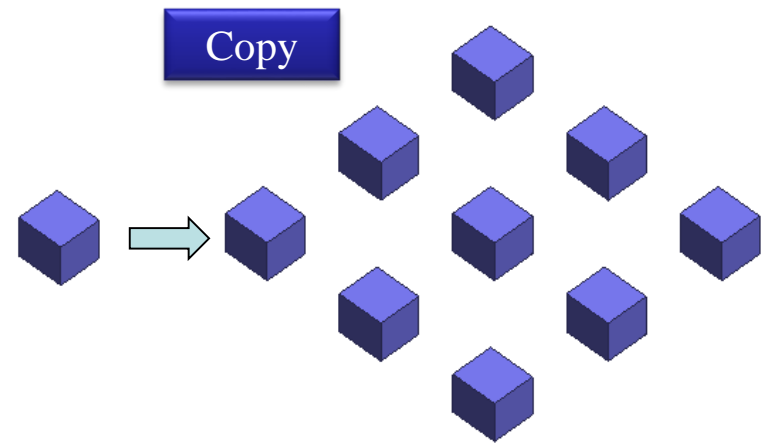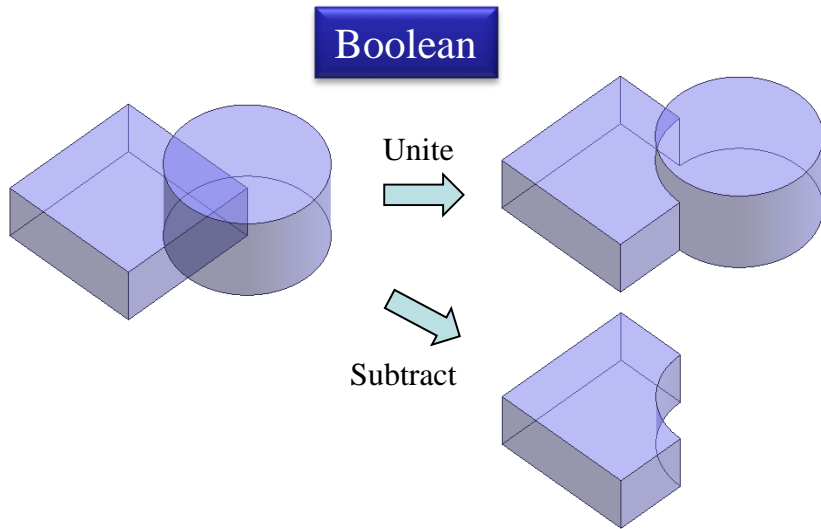
See Help – [Modeling] – [Drawing Process] – [How to Modify Objects] – [Modification Tools]

Modification Commands

Murata Software

Modification Tools		Applicable bodies or topologies	Functions
Boolean	Unite	Sheet bodies Solid bodies	Unites bodies
	Subtract		Subtracts bodies from bodies.
	Intersect		Intersects bodies
Move/Rotate	Move	All types of bodies	Moves bodies
	Rotate		Rotates bodies
	Mirror	All types of bodies	Mirrors bodies
	Cruise	Solid bodies Sheet bodies	Slides the selected bodies on the surface of other body or the drawing plane.
	Move Face Topologies	Face topologies on solid bodies	Moves the selected face topologies to deform the solid bodies.
	Rotate Face Topologies	Face topologies on solid bodies	Rotates the selected face topologies to deform the solid bodies.
Deformation	Scale	All types of bodies	Body dimensions are scaled, either uniformly or nonuniformly.
	Extrude	Vertex bodies Wire bodies Sheet bodies	Extrudes bodies.
	Extrude with Taper	Wire bodies Sheet bodies	Extrudes bodies with taper.
	Revolve	Vertex bodies Wire bodies Sheet bodies	Revolves bodies.

Modification Commands

Modification Tools		Applicable bodies or topologies	Functions
Deformation	Loft	Vertex bodies Wire bodies Sheet bodies	Form a 3D object by lofting the specified cross sections.
	Sweep along Path	Wire bodies Sheet bodies	Sweeps the selected bodies along a wire path.
	Fillet	Wire bodies Sheet bodies Solid bodies	Fillets the body corners.
	Chamfer	Solid bodies	Chamfers the body corners.
	Cut	Sheet bodies Solid bodies	Cuts bodies with a specified plane.
	Cut at the intersection point of wire bodies	Wire bodies	Cut the wire bodies at the intersection point.
	Extract	Sheet bodies Solid bodies	Cuts and removes a portion of the body.
	Shell	Face topologies on solid bodies	Hollows the body at the selected face.
	Taper	Face topologies on solid bodies	Tapers adjacent faces with reference to the selected face.
	Delete Face	Face topologies	Removes the selected face.
	Delete Edge	Edge topologies	Removes the selected edge.

Body Attribute/Material Property

Murata Software

Right-click a body.
Click Body Attribute/Material Property.

Enter Body Attribute Name
and Material Name.
Edit Data.
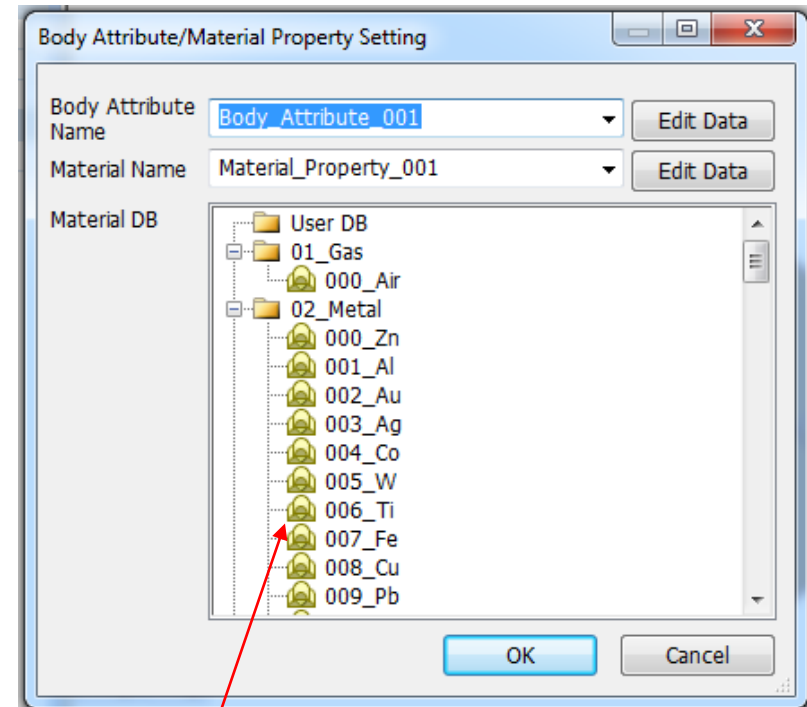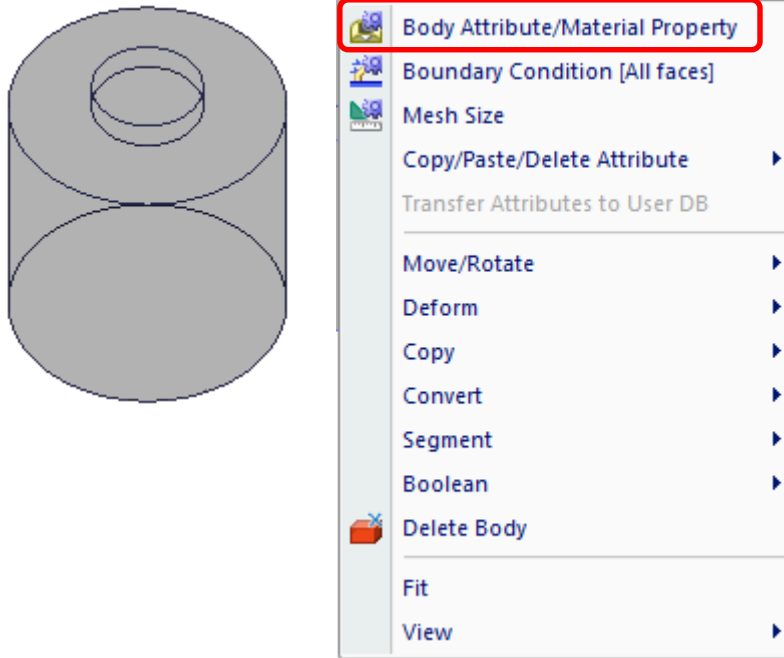
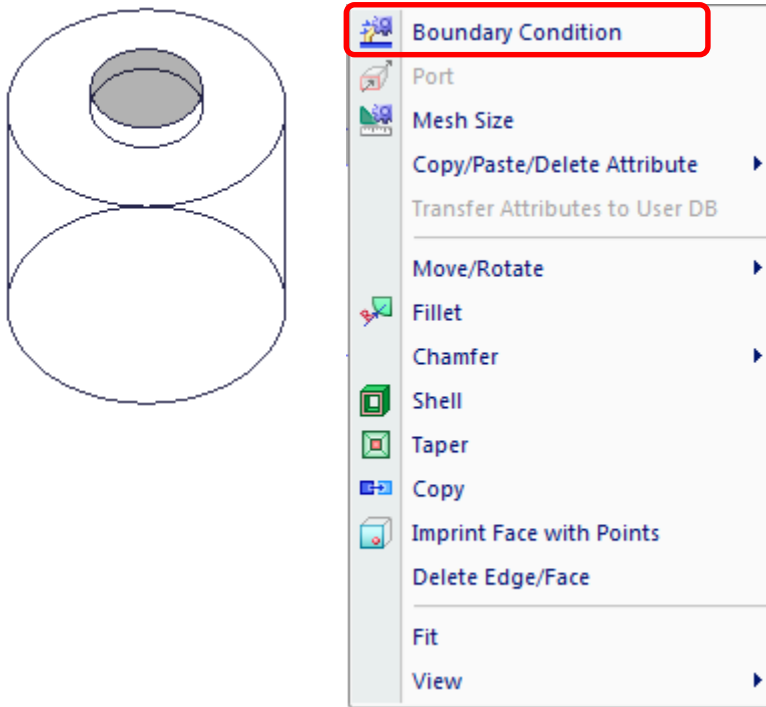*Body Attribute and Material Property can be set to body only

Material database is available

Boundary Condition

Right-click a part of body.
Click Boundary Condition.

*Boundary Condition can be set to a part of body (vertex, edge, face).

🖼	Boundary Condition
🖼	Port
🖼	Mesh Size
	Copy/Paste/Delete Attribute ▶
	Transfer Attributes to User DB
	Move/Rotate ▶
🖼	Fillet
	Chamfer ▶
🖼	Shell
🖼	Taper
🖼	Copy
🖼	Imprint Face with Points
	Delete Edge/Face
	Fit
	View ▶

Enter Boundary Condition Name.
Edit Data.

Boundary Condition

Boundary Condition Name: dary_Condition_001 ▾ [Edit Data]

☐ Add
☐ Delete

[OK] [Cancel]
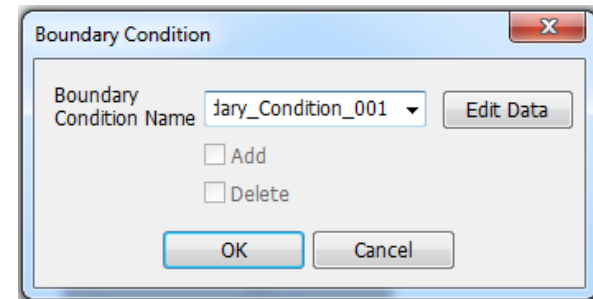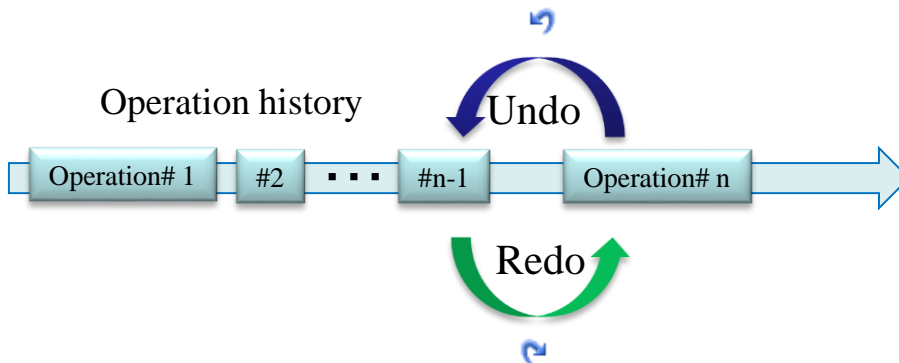
Model creation is completed at this step.

Undo/Redo

Undo ⤺

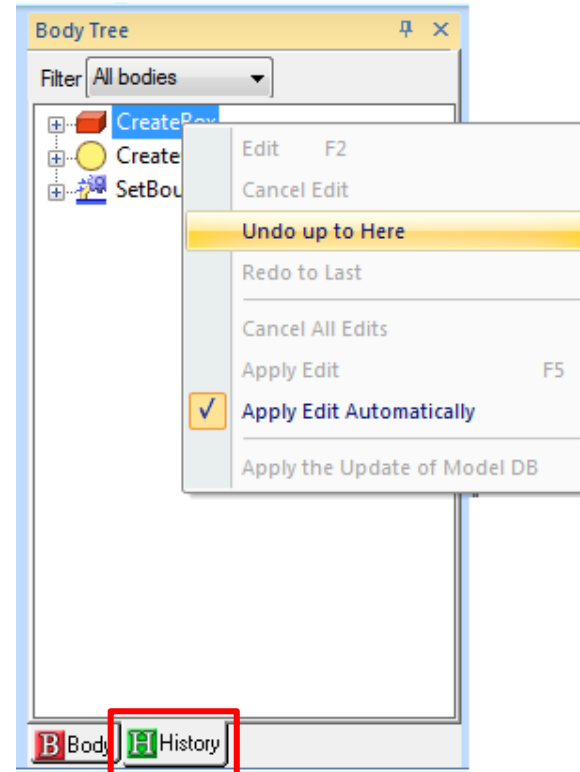Undo the modeling or attribute operation.
Shortcut keys are [Ctrl] + [Z].

Redo ⤻

Redo the modeling or attribute operation.
Shortcut keys are [Ctrl] + [Y].

Operation history

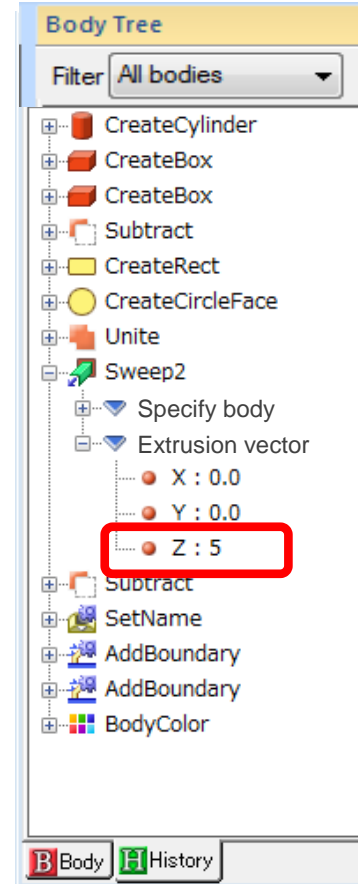| Operation# 1 | #2 | ▪ ▪ ▪ | #n-1 | Operation# n |

Undo
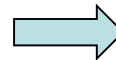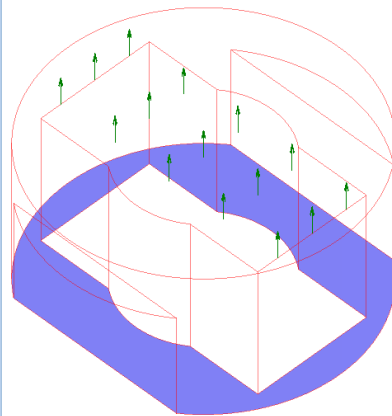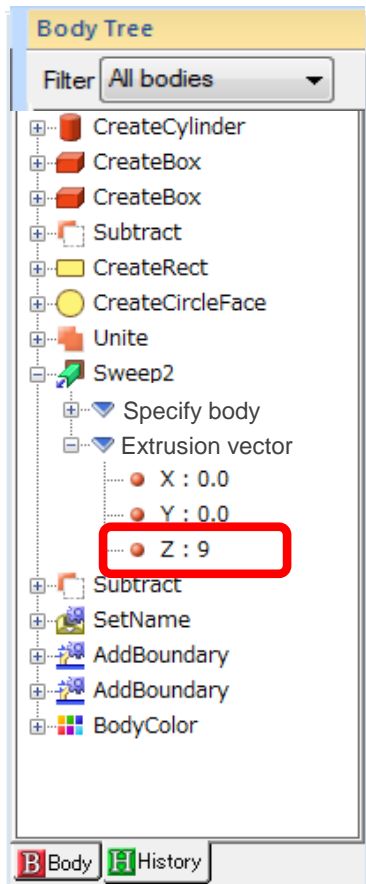
Redo

Undo on History Tree

On History Tree, you can undo up to a specific item.
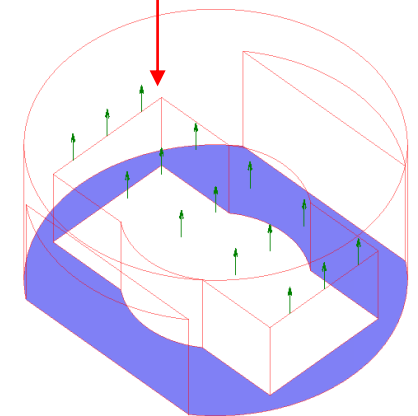Right-click an item and select [Undo up to Here]

See Help – [Modeling] – [Drawing Process] – [How to Modify Objects] – [Modification Tools]

Edit History

A model can be modified by entering parameters for primitives and modifications on a History.

The depth of internal hole is shallower.

See Help – [Modeling] – [Drawing Process] – [How to Modify Objects] – [Modification Tools]
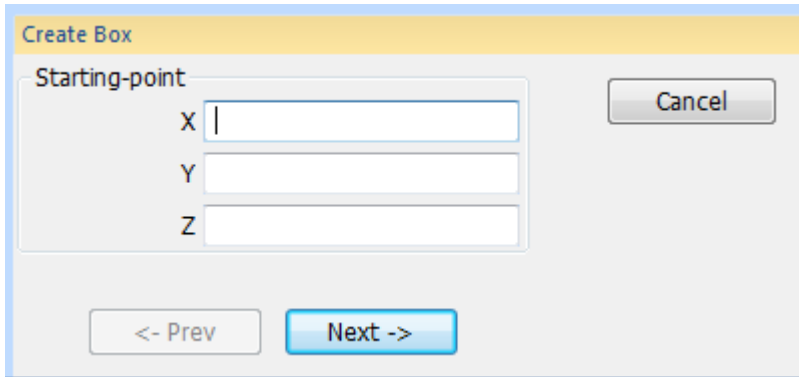
Efficient Parameter Input

Murata Software

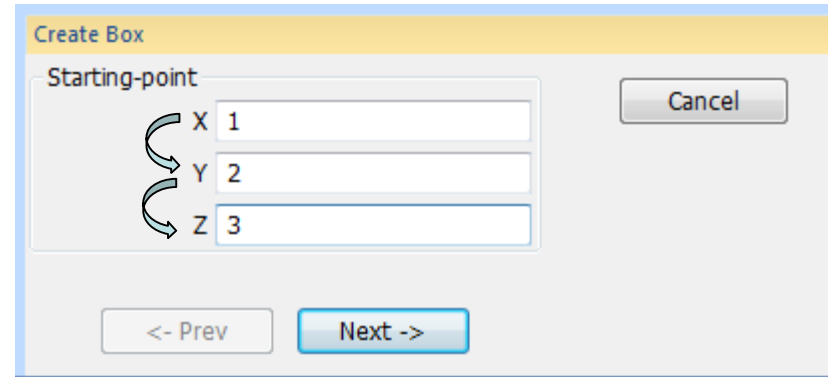## Enter "0" Automatically

Click [Next] without entering anything.

"0" is automatically entered.

Jump to Next Input Field

You can jump from an input field to another by pressing [Tab] key.

Switching Display

Murata Software

Show/Hide

Show/Hide of body is switched by
[Model] tab => [Show]

Show All, Hide All

Show/Hide Selected Bodies

Hide All but Selected Bodies,
Reverse All Show/Hide
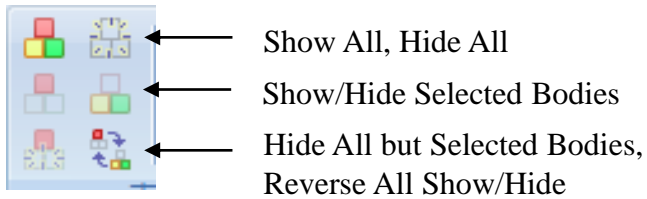
Show/Hide can be switched also by
ticking a box at the left side of each item.

Body Tree

Filter | All bodies ▼ | Create new grou

☑ Bodies: 2
☑ Body0 <imprinting body>
☑ Body1
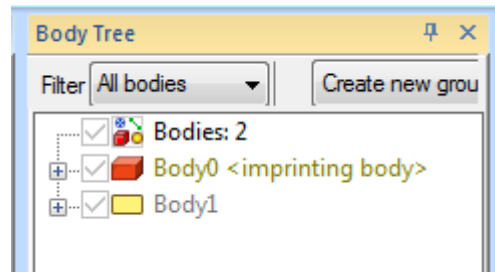
Paint and Shade/Wireframe

Paint and Shade/Wireframe is switched by
[Model] tab => [Show]

Wireframe, Paint and Shade all bodies

Wireframe selected body,
Paint and shade selected body

Paint and Shade

Wireframe

Import CAD Data

<Standard Support>
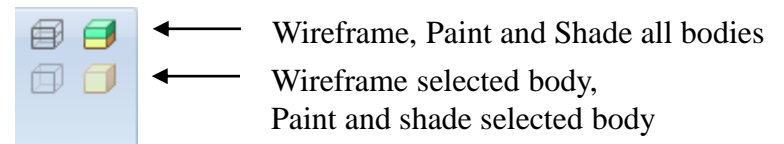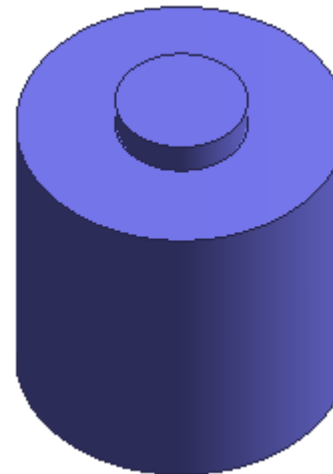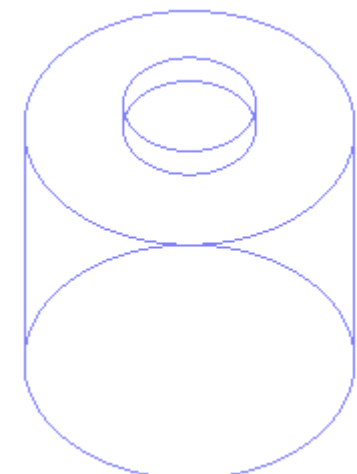
Parasolid (extension: .x_t .x_b)

DXF (extension: .dxf)

*DXF is for 2D only

<Optional Support>

CATIA® V4-V6, Creo™,
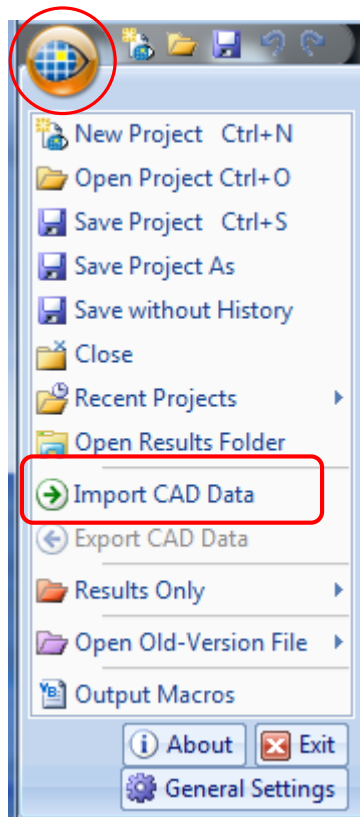Pro/ENGINEER®, I-deas,
JT, Unigraphics®/NX,
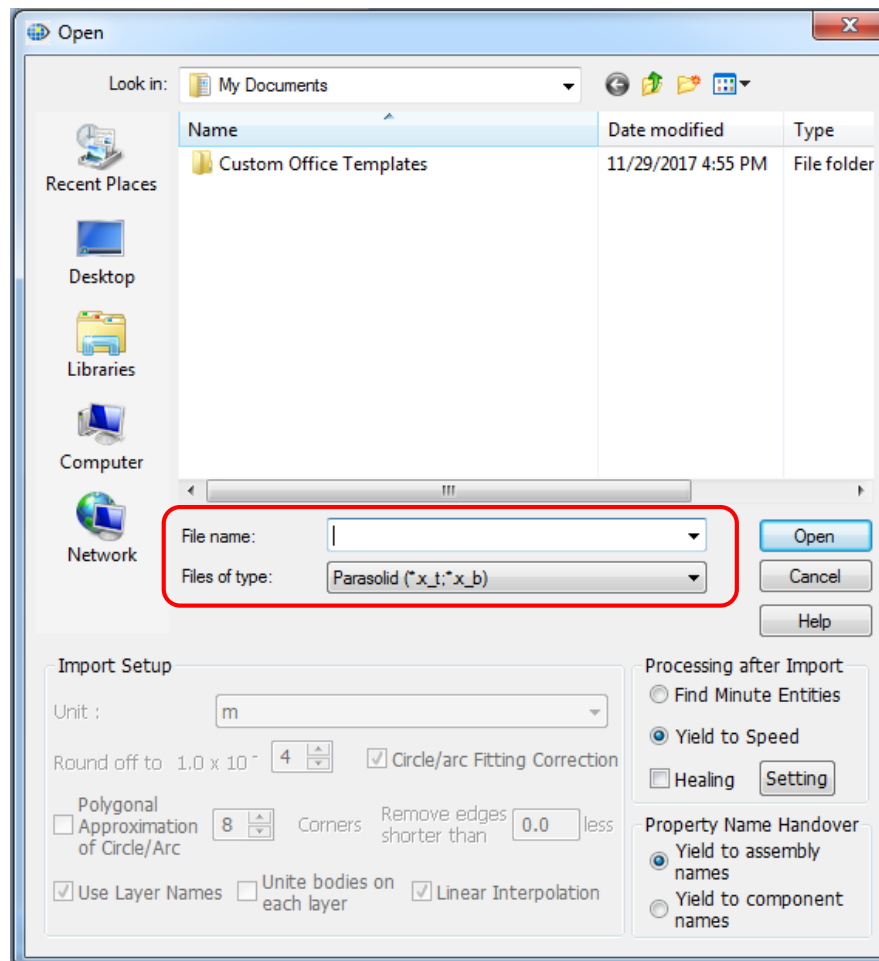SolidWorks®, Solid Edge,
ACIS®, Inventor, IGES,
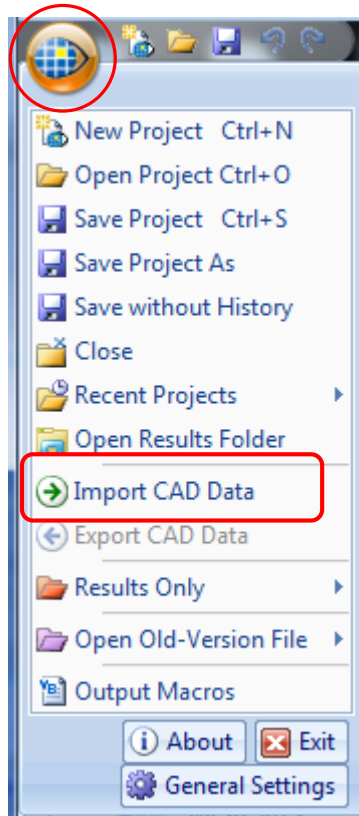STEP, STL, PRC, IFC, VRML

Import CAD Data

Application button
=> [Import CAD Data]

Select File name and Files of type, and click [Open]

Import DXF

DXF data, which is basically 2D, can be imported in 3D by Femtet® setting.

Select DXF file from the menu.

By setting up each layer with [Read], [Height], and [Thickness], a model can be transformed to 3D in Femtet®.

Layer Reading Setup

Layer name	Read	Height [mm]	Thickness [mm]	
No layer name	✓	0	0	Read All
COND1_HATCH	✓	0	0	
COND2_HATCH	✓	1	0	Release All
HOLE	✓	0	1	
OUTLINE	✓	0	1	Reverse

Add-In for CAD Data
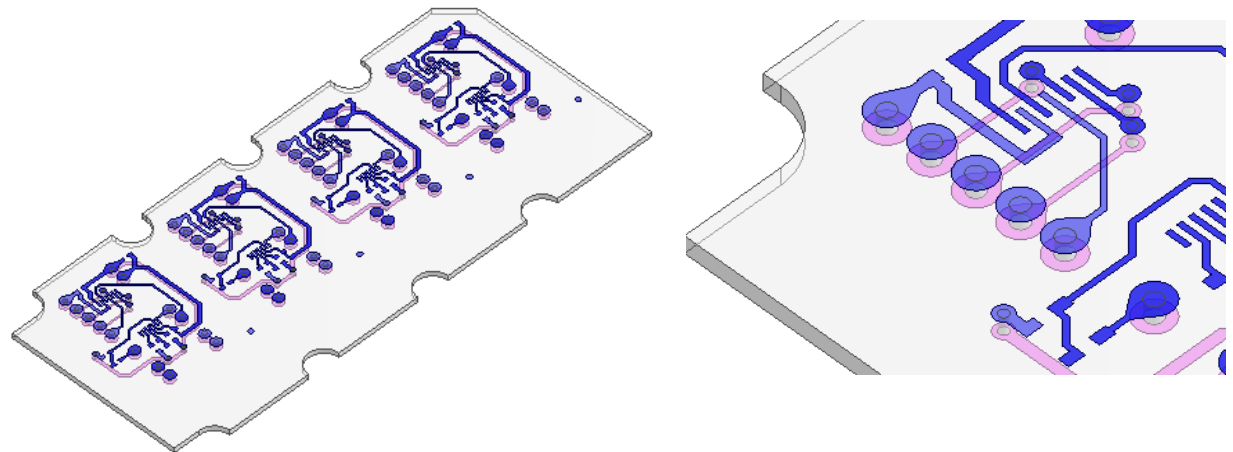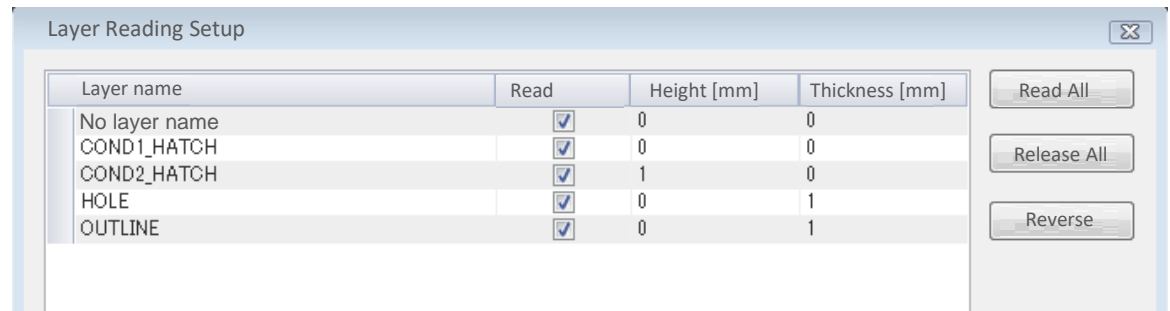
Add-in tools for SolidWorks® and SpaceClaim® are available for free on our website.
Direct model import from SolidWorks® or SpaceClaim® is possible.

*Parasolid conversion is optional for SpaceClaim®.

SolidWorks®

Femtet

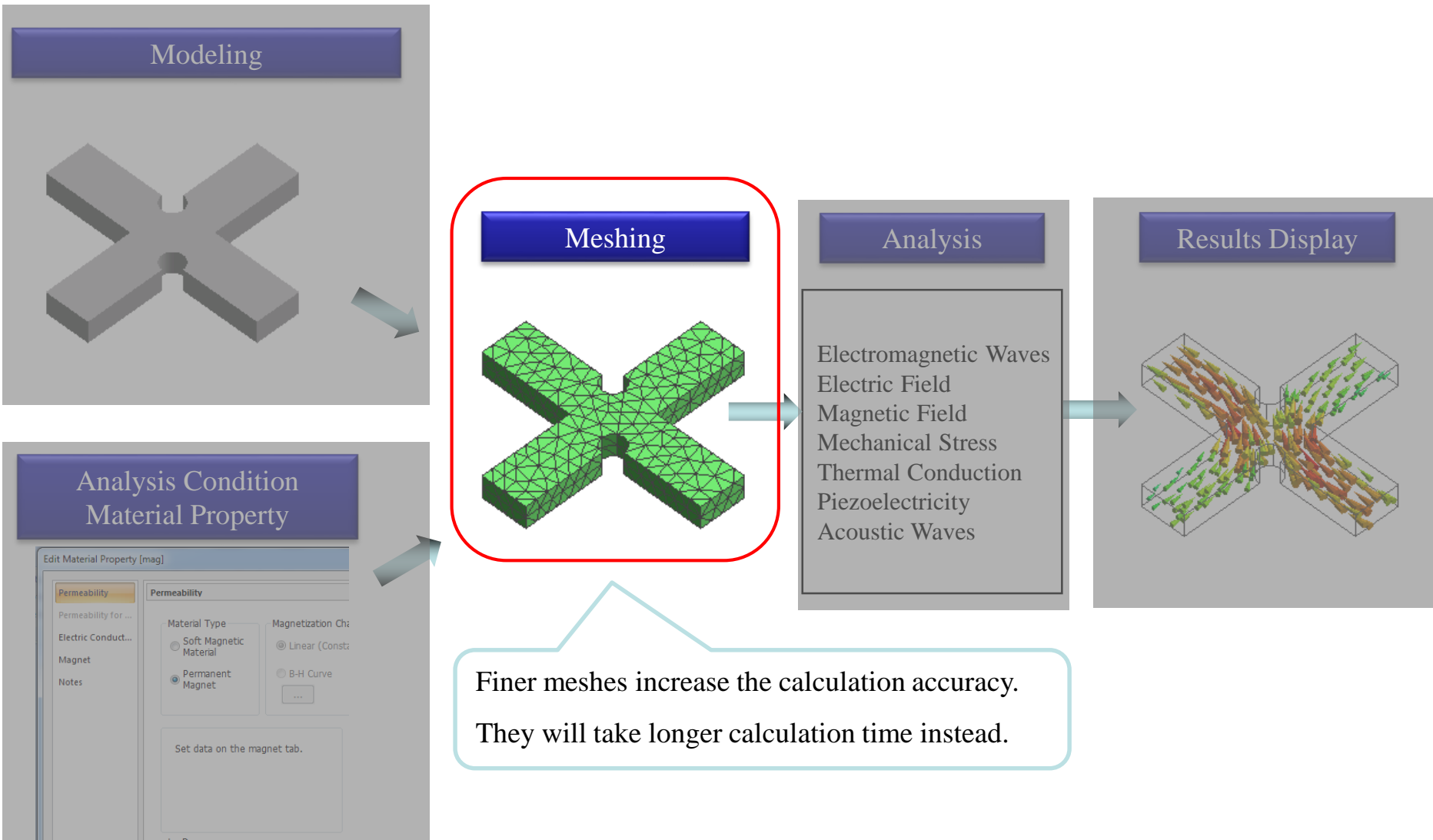# Meshing

Modeling

Analysis Condition
Material Property

Meshing

Analysis

Electromagnetic Waves
Electric Field
Magnetic Field
Mechanical Stress
Thermal Conduction
Piezoelectricity
Acoustic Waves

Results Display

Finer meshes increase the calculation accuracy.

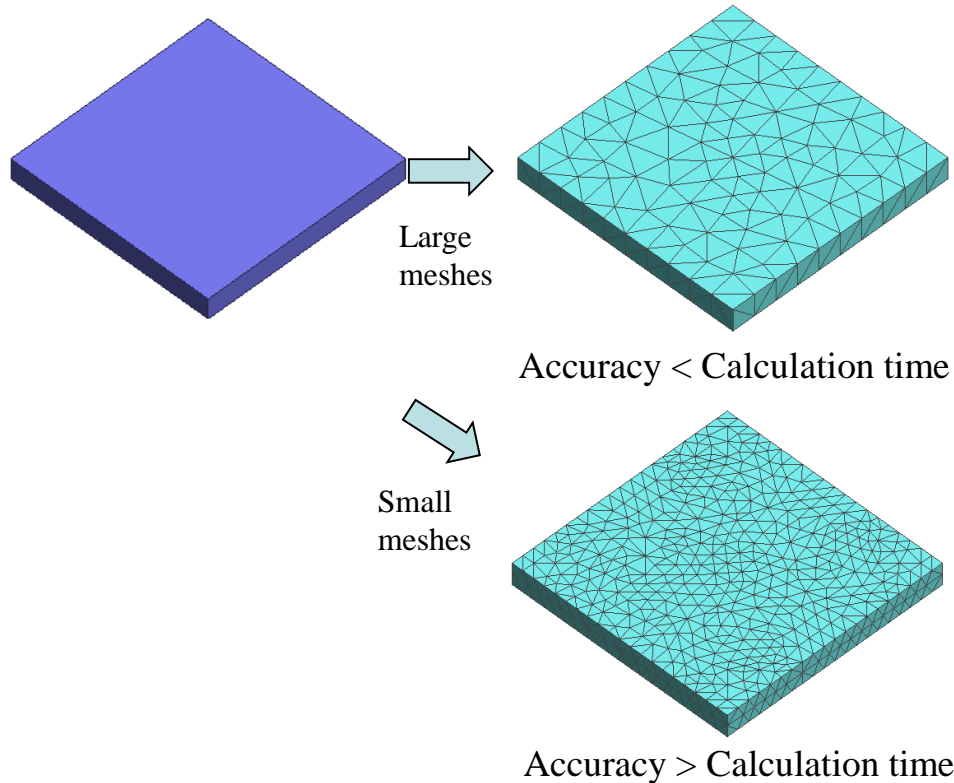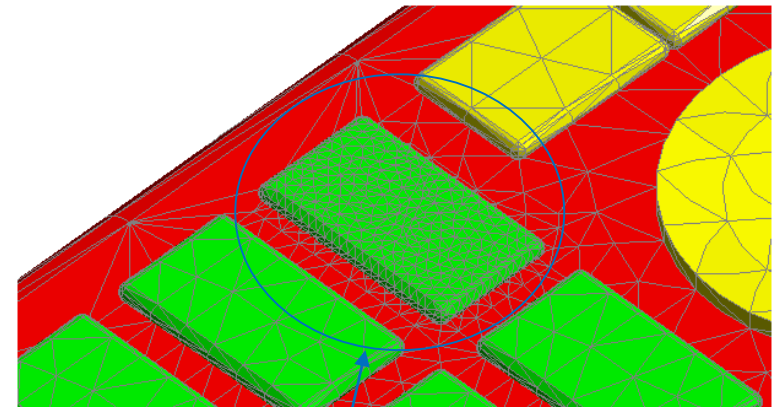They will take longer calculation time instead.
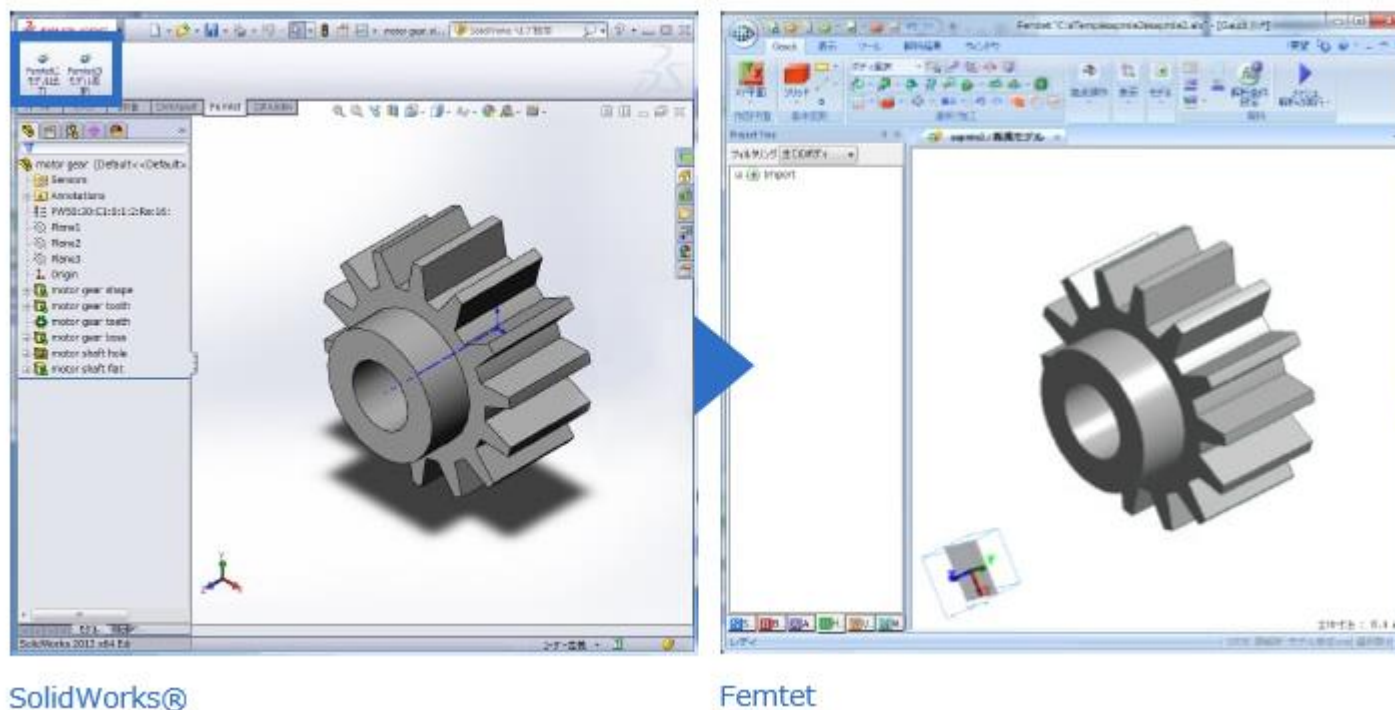
Meshing

Meshing is fully automated.
Triangles or tetrahedrons will be created according to an analysis model.
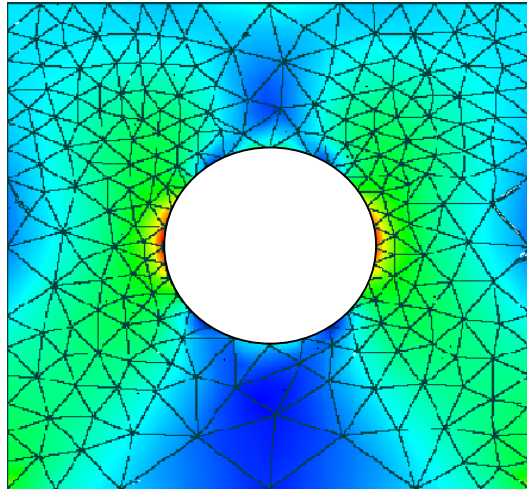Partial mesh size can be set.

Automatic Meshing

Large meshes

Accuracy < Calculation time

Small meshes

Accuracy > Calculation time

Partial Mesh Size
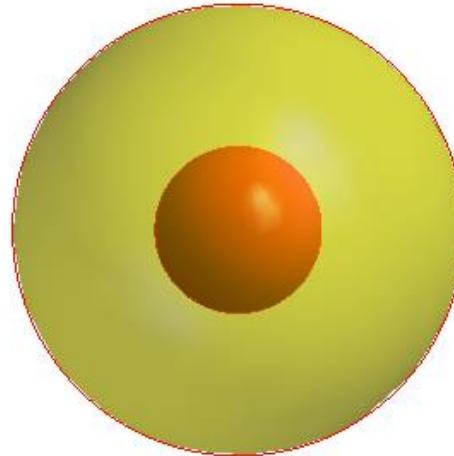
Finer meshes can be set partially
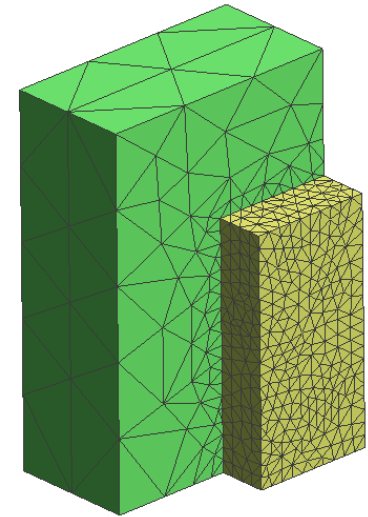
Meshing

Adaptive Mesh

Appropriate mesh size is determined through repeated meshing and analysis.

Automatic Boolean
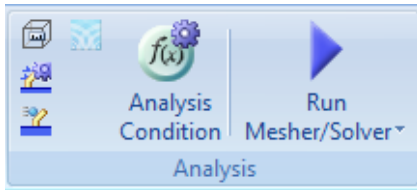

Remove intersection of bodies automatically.

Automatic Imprint

Contacting area of bodies is detected and mesh shape is automatically adjusted.

Mesh Size

General Mesh Size

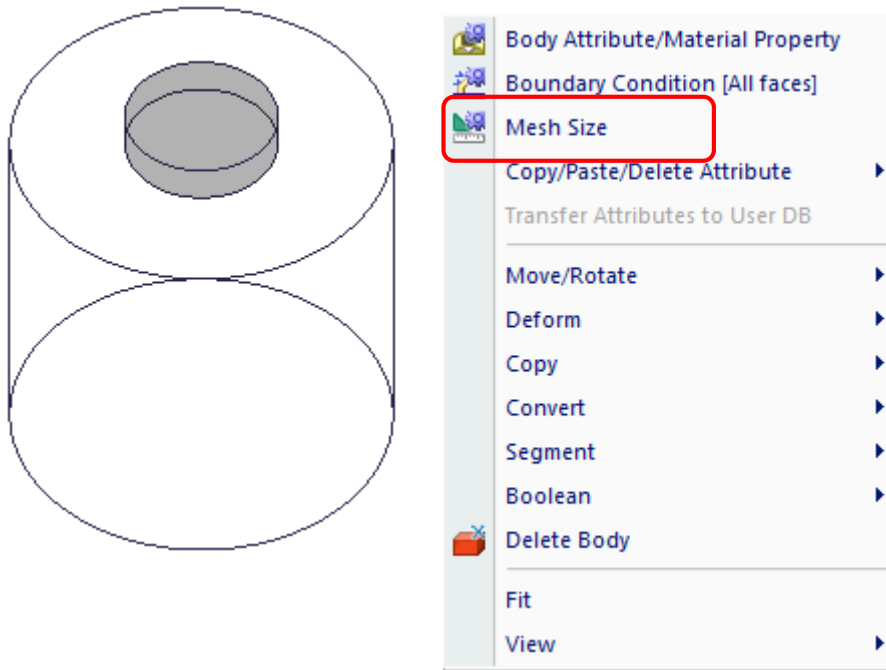[Model] tab => [Analysis] => [Analysis Condition] => [Mesh] tab

If [Automatically set the general mesh size] is selected, the mesh size will be set to a tenth of the model's maximum dimension. You can set the mesh size of your wish by deselecting this option.
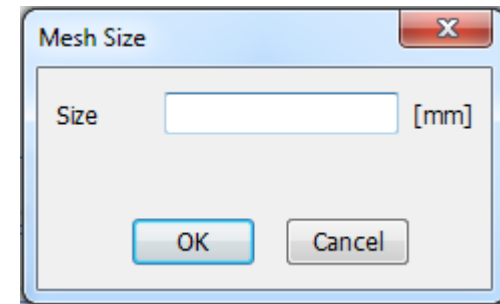

Model's Maximum Dimension: 5.5 mm
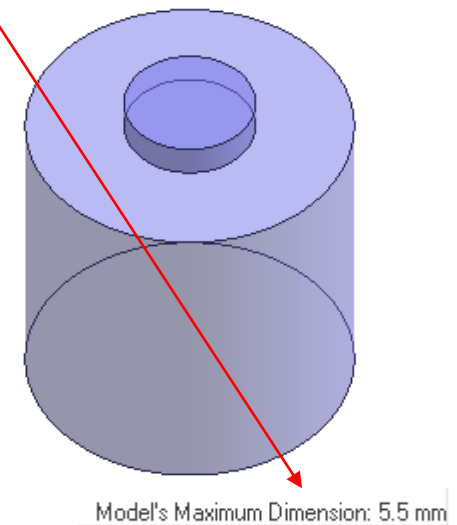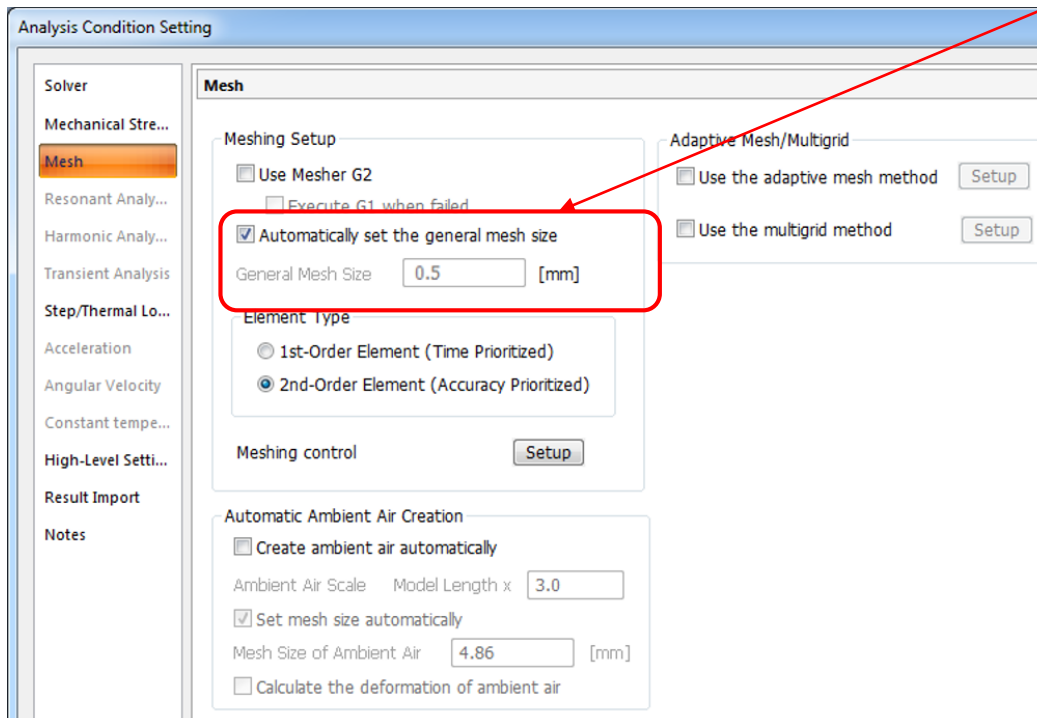
Mesh Size

Partial Mesh Size

Right-click a body of model where you want
to set mesh size partially.
Click [Mesh Size] on an appearing menu.

Enter mesh size of your wish

Menu:
- Body Attribute/Material Property
- Boundary Condition [All faces]
- Mesh Size
- Copy/Paste/Delete Attribute ▶
- Transfer Attributes to User DB
- Move/Rotate ▶
- Deform ▶
- Copy ▶
- Convert ▶
- Segment ▶
- Boolean ▶
- Delete Body
- Fit
- View ▶

Mesh Size dialog:
Size [_____] [mm]
OK   Cancel

*Partial mesh size will override the general mesh size.

Solvers

Modeling

Analysis Condition
Material Property

Meshing

Analysis

Electromagnetic Waves
Electric Field
Magnetic Field
Mechanical Stress
Thermal Conduction
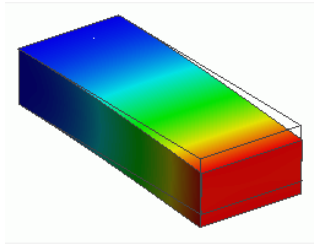Piezoelectricity
Acoustic Waves

Results Display
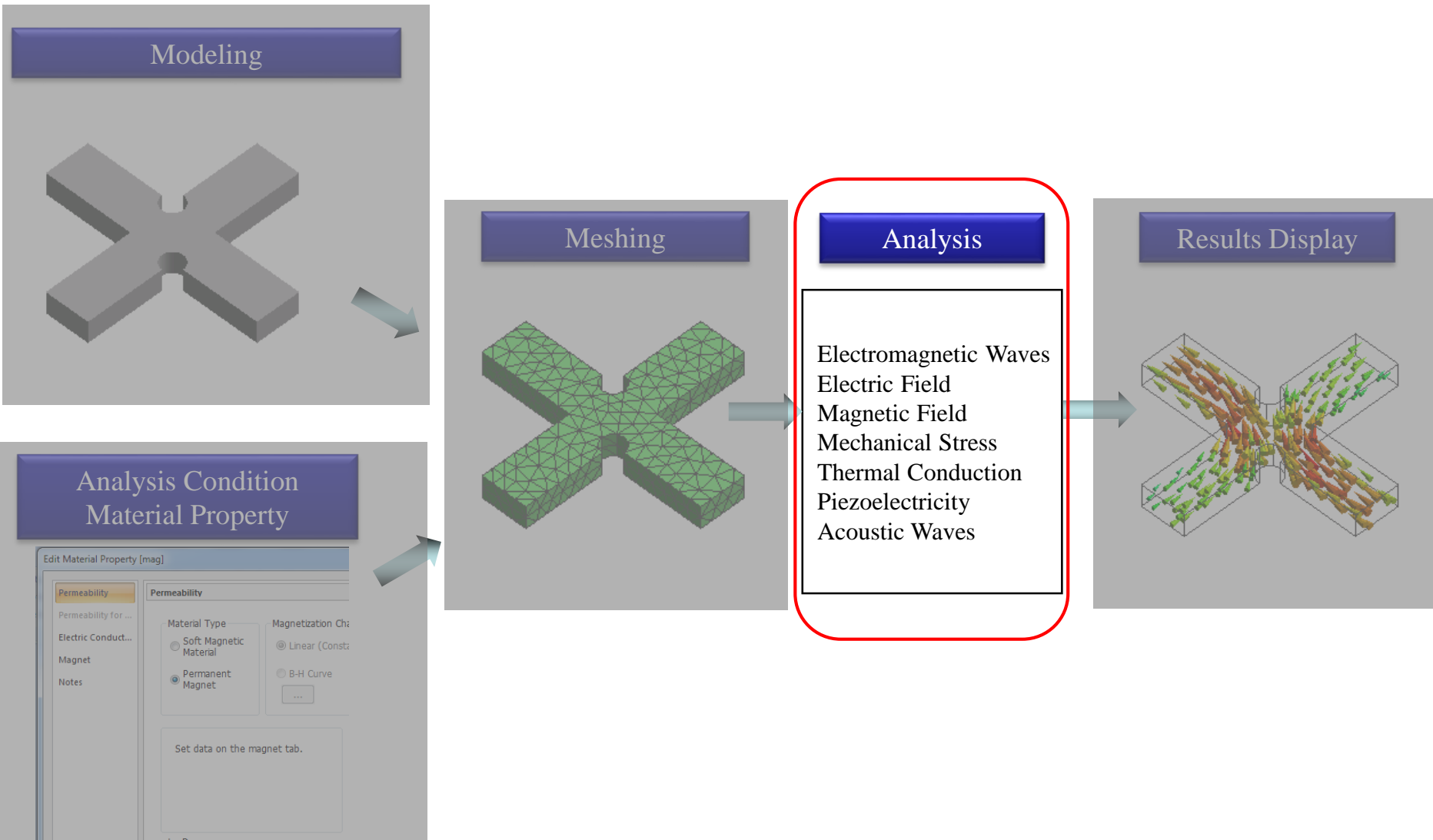
Mechanical Stress Analysis

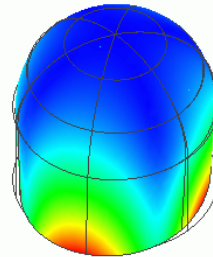Solves deformation and stress due to load or acceleration.
The displacement, stress distribution, and resonant frequency of a structure are calculated.
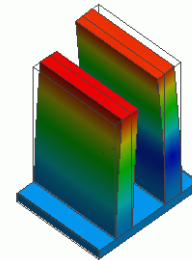
Static Analysis

Cantilever under Distributed Load
(Example 1)

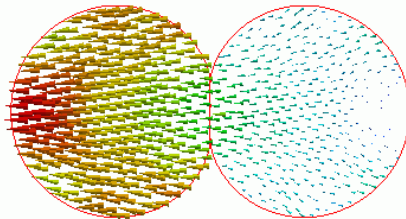**Resonant Analysis**
(Characteristic Vibration)

Resonance of Bell
(Example 11)

Harmonic Analysis
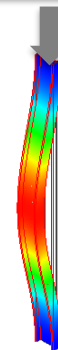(Forced Vibration)

Standing Bars
(Example 16)

Transient Analysis

Elastic Collision of Balls
(Example 32)

Buckling Analysis

H-beam Steel
(Example 48)

Deformation and stress by load and acceleration are solved. Displacement of a structure and stress distribution, and resonant frequency of a structure can be analyzed.
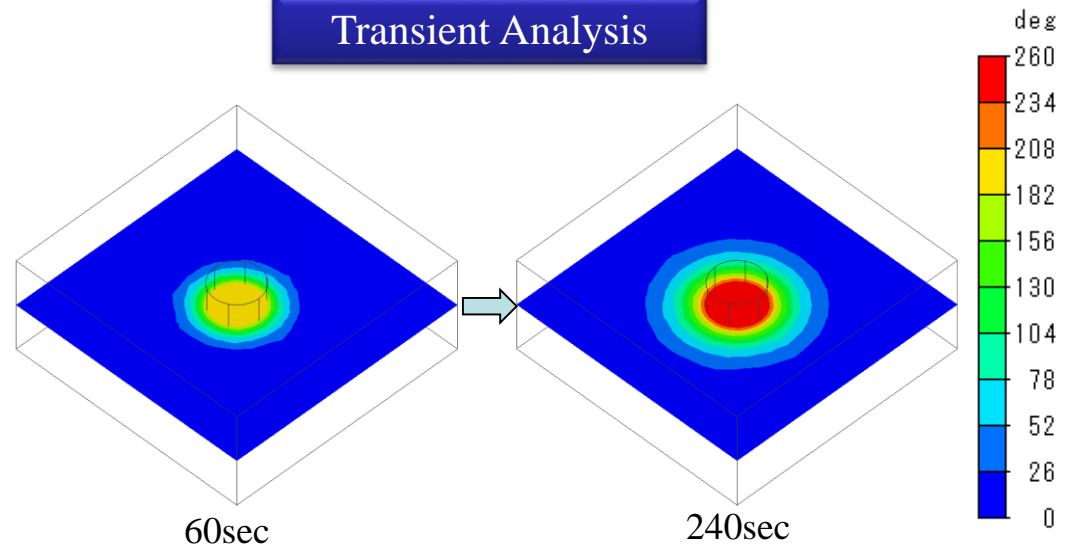
Thermal Analysis

Solves temperature distribution, heat flux, temperature distribution after specific time.

Steady-state Analysis

Transient Analysis

60sec

240sec
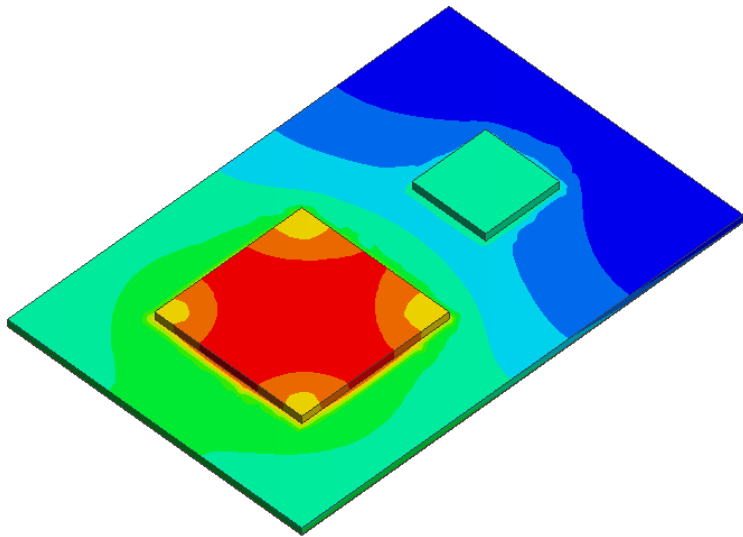
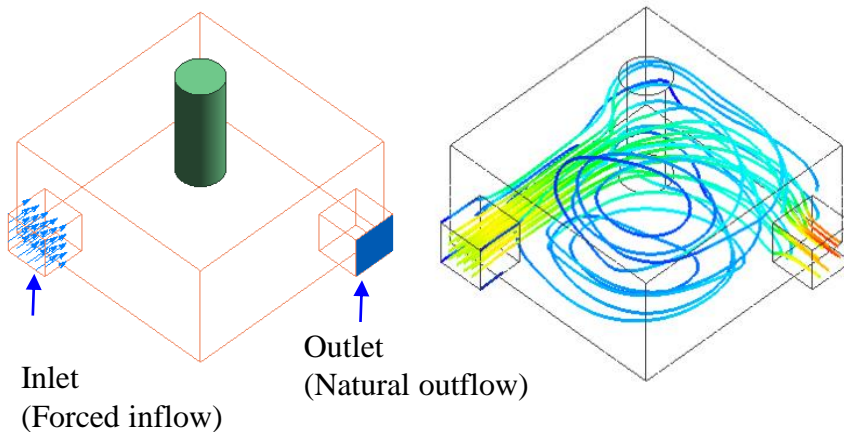Multiple Heat Sources on Substrate
(Example 10)

Block Containing Heating Element
(Example 4)

Fluid Analysis

Flows with obstacle in the path is solved.

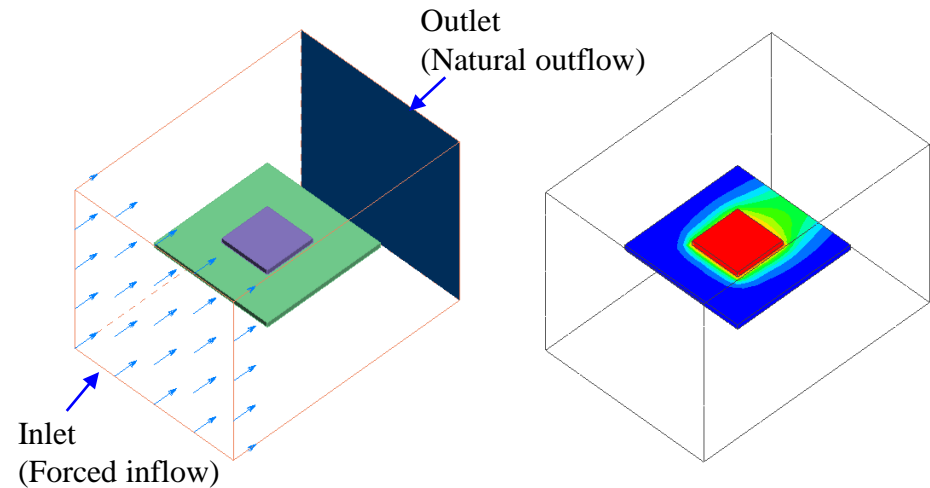By coupling with thermal analysis, heat radiation of a substrate can be analyzed.

Steady-state Analysis

Fluid-Thermal Analysis

Outlet
(Natural outflow)

Inlet
(Forced inflow)

Outlet
(Natural outflow)

Inlet
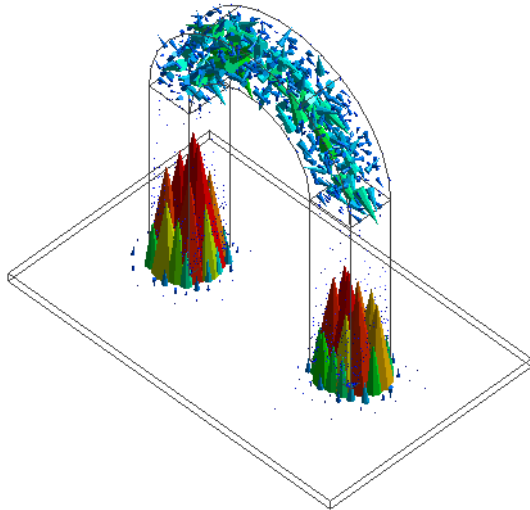(Forced inflow)

Obstacle in the flow path
(see Tutorial)

Heat Radiation of IC by Forced Convection （Example 4）

*Fluid-thermal analysis is optional

Magnetic Analysis

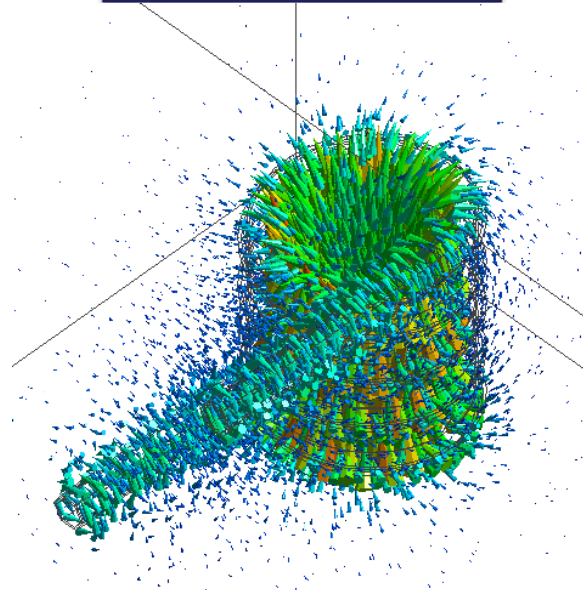Solves magnetic field distribution of magnetic material, inductance of coil, and torque of motor.
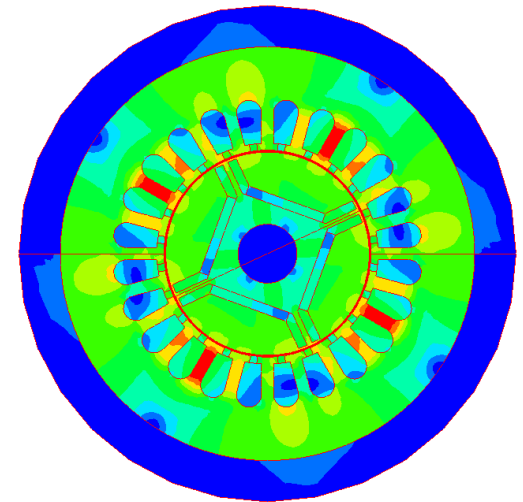
| Static Analysis | Harmonic Analysis | Transient Analysis |

Magnetic Material and
Horseshoe Magnet
(Example 4)

Magnetic Field
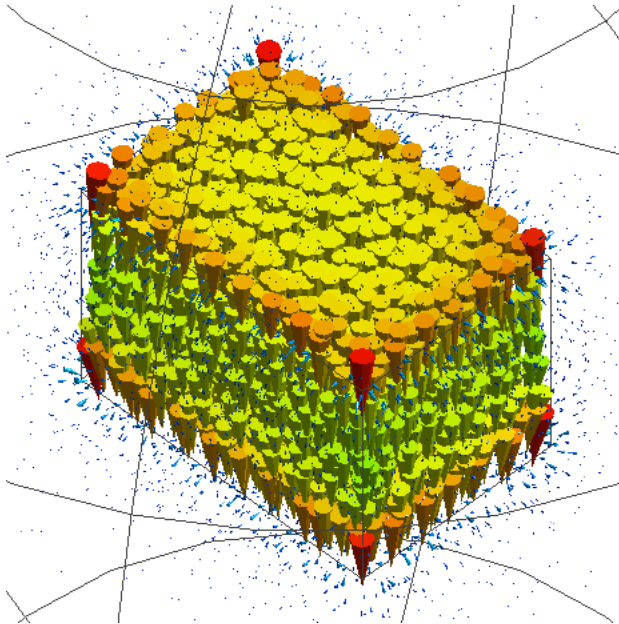Generated by Helical Coil
(Example 11)
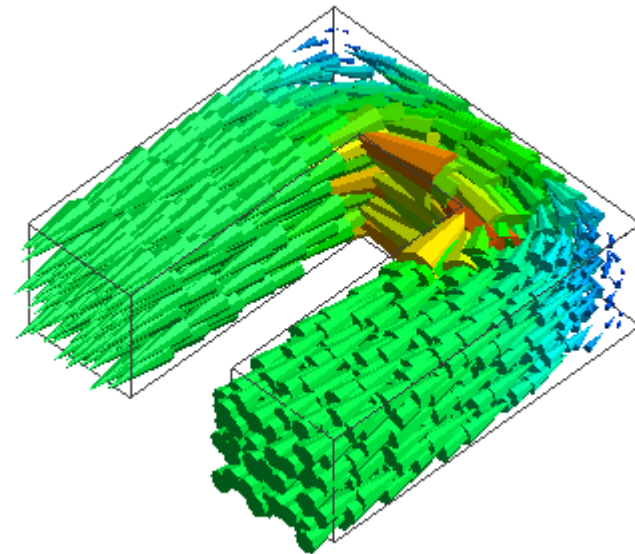
Torque of IPM Motor
(Example 1)

Electric Analysis

Solves electric field distribution of dielectric/conductive material. Capacitance across electrodes, resistance of conductor, and electrostatic force of dielectric material can be analyzed.

Static Analysis

Harmonic Analysis

Electrostatic Capacitance
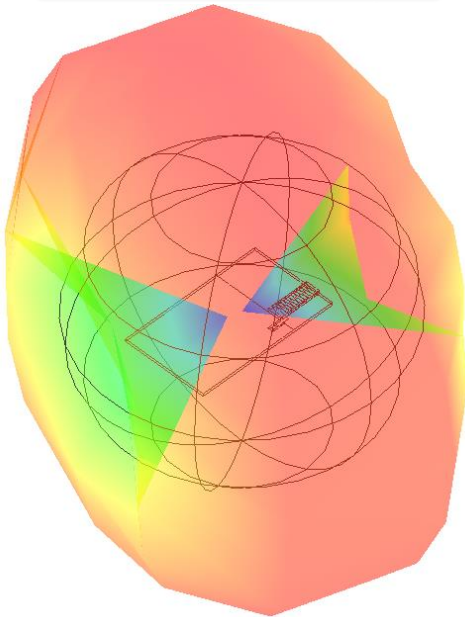of Dielectric Capacitor
(Example 2)

Dielectric and Conductive Materials
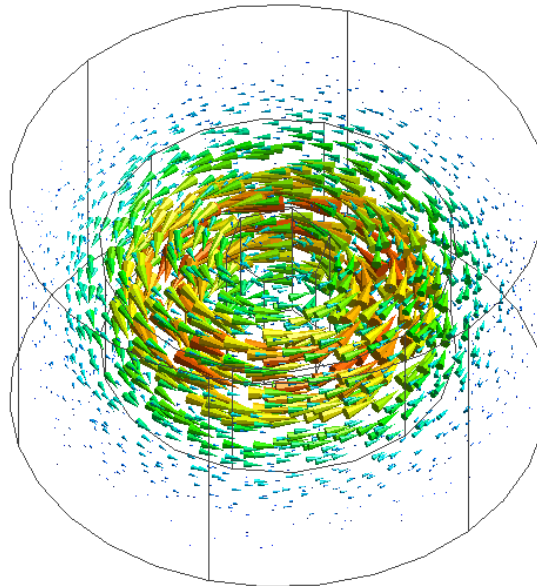(Example 14)

Electromagnetic Analysis

Solves S-parameters of electric circuit, radiation characteristics of antenna, resonant characteristics of microwave resonator.
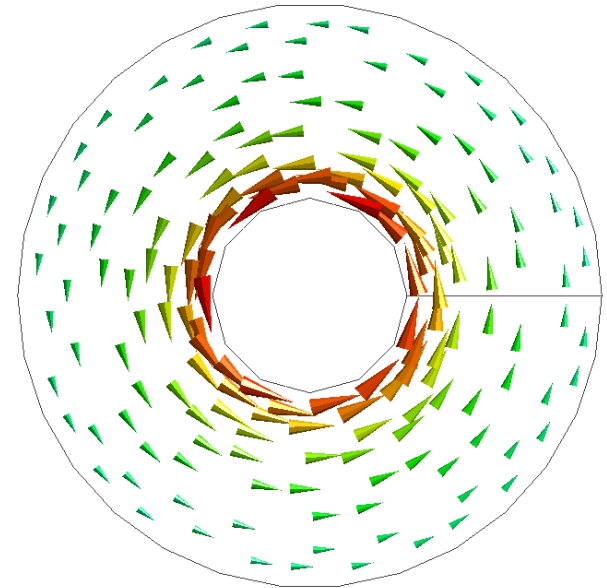
Harmonic Analysis	Resonant Analysis	Waveguide Analysis

Spiral Antenna
(Example 3)
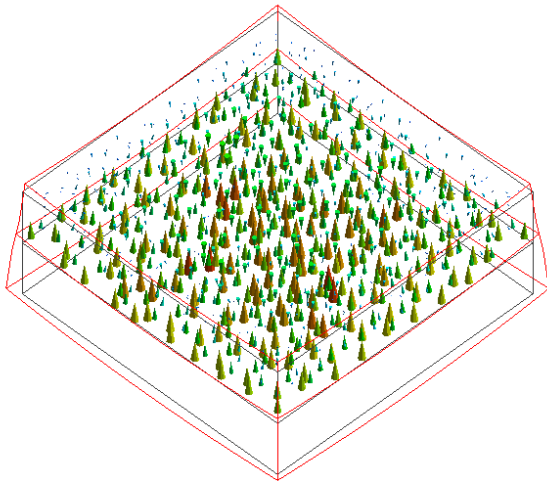
Dielectric Resonator
(Example 11)

Coaxial Cable
(Example 14)

Piezoelectric Analysis

Murata Software

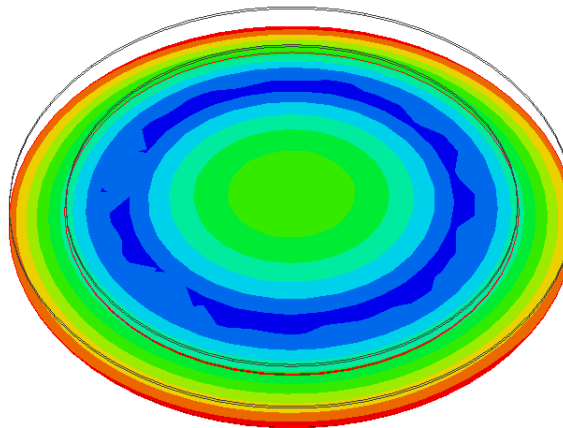Solves piezoelectric single crystal and piezoelectric ceramic.
Potential generated by load or pressure, and deformation due to voltage are analyzed.
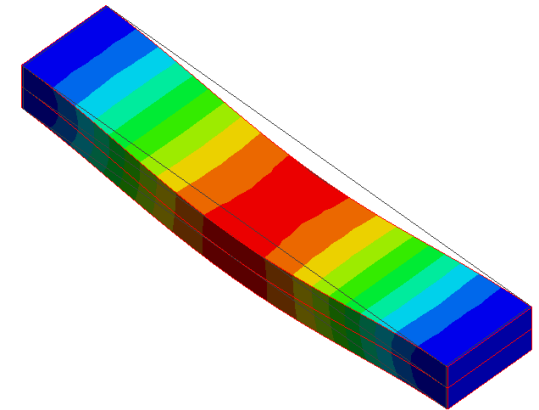
| Static Analysis | Harmonic Analysis | Resonant Analysis |

Thermal Load and
Mechanical Strain
(Example 7)

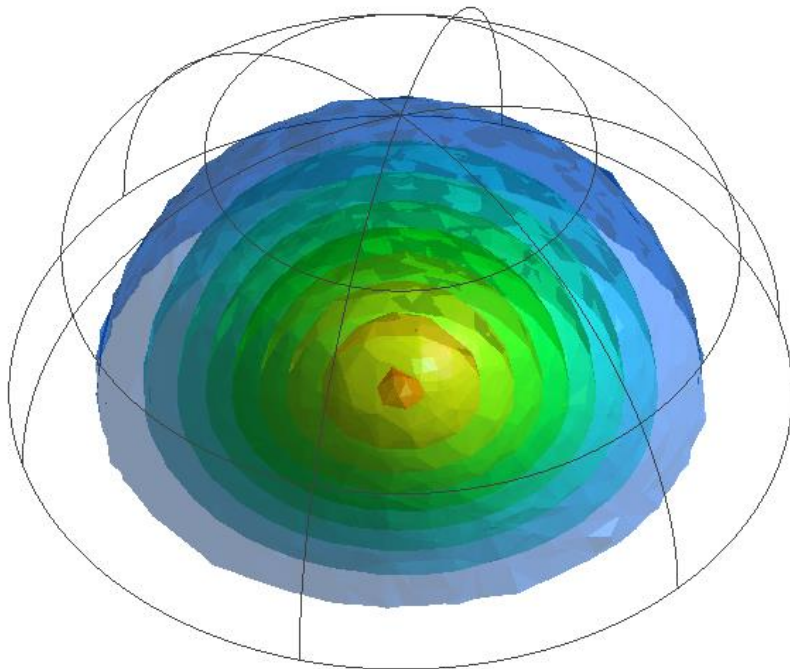Piezoelectric Element
with DC Voltage
(Example 2)
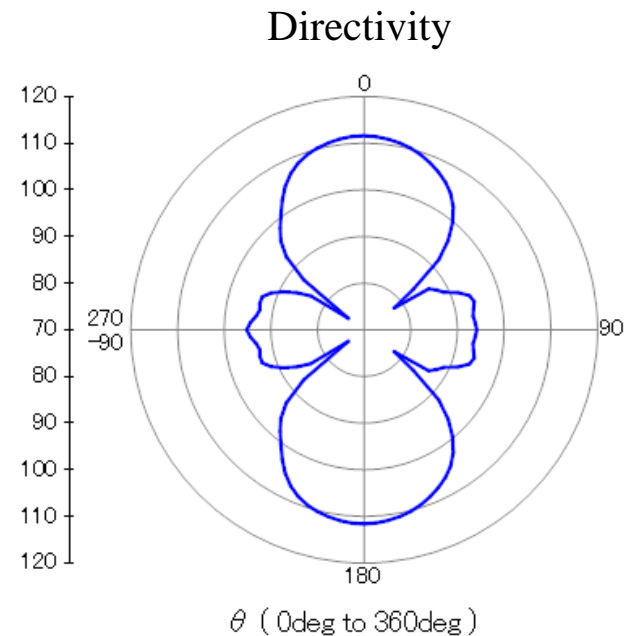

Piezoelectric Element
with Tensile Force
(Example 14)

Acoustic Analysis

Solves sound waves driven by sound pressure or velocity.
Sound pressure distribution and directivity can be analyzed.

Harmonic Analysis

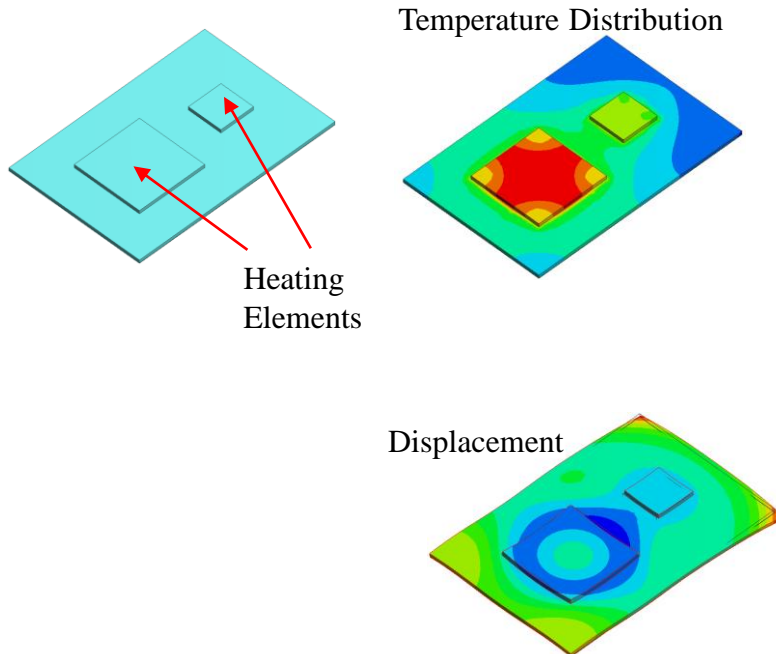Directivity

Vibrating Disc
(Example 2)

Multiphysics Analysis

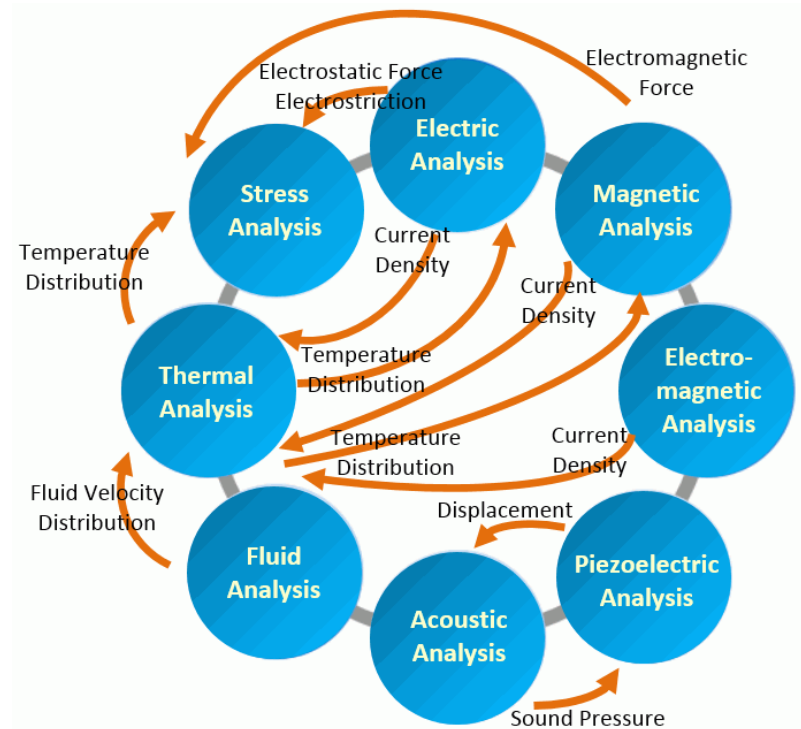In depth analysis is possible by coupling solvers.

For example, deformation and distributions of temperature and stress caused by heat source, or distributions of magnetic field and temperature caused by current flowing in a coil.

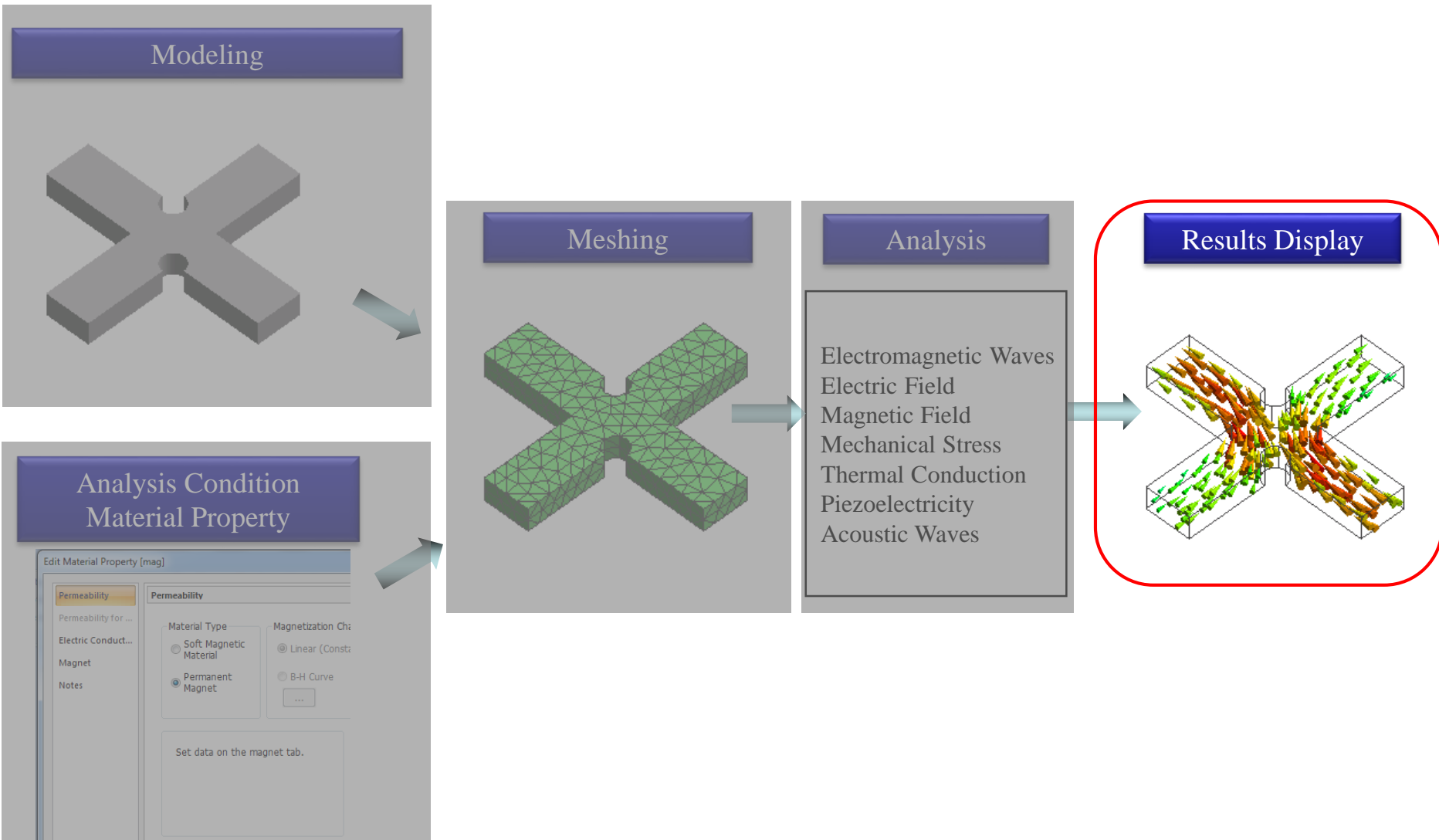### Example of Thermal-Mechanical Stress Analysis
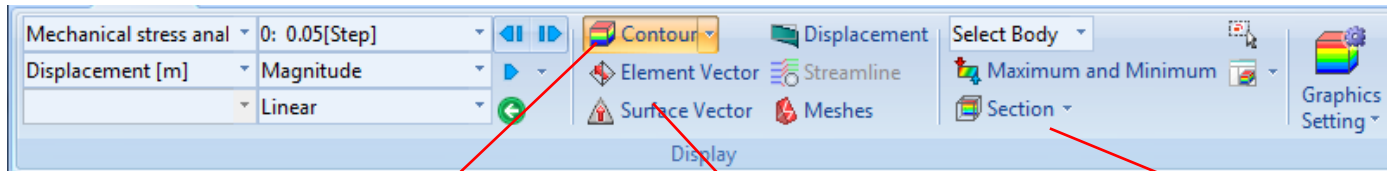### Deformation due to Temperature

Temperature Distribution
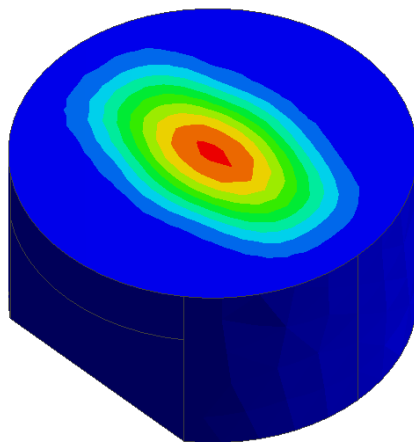

Heating Elements

Displacement

Solver Combinations

Results Display

Modeling

Meshing

Analysis

Electromagnetic Waves
Electric Field
Magnetic Field
Mechanical Stress
Thermal Conduction
Piezoelectricity
Acoustic Waves

Results Display

Analysis Condition
Material Property

Edit Material Property [mag]

Permeability
Permeability for ...
Electric Conduct...
Magnet
Notes

Permeability

Material Type
○ Soft Magnetic
  Material
◉ Permanent
  Magnet

Magnetization Cha
◉ Linear (Consta
○ B-H Curve
  ...

Set data on the magnet tab.
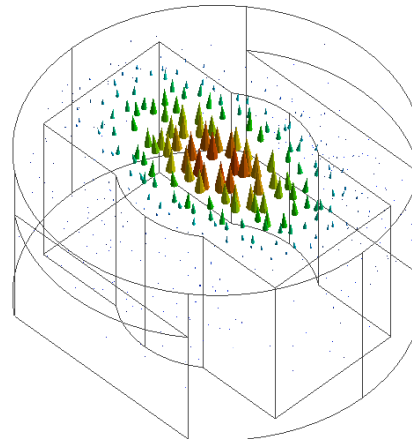
Results Display

 Murata Software

Various ways of results display are available.

Contour   Vectors   Section
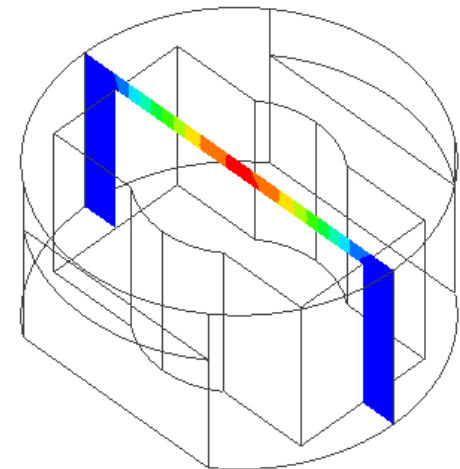
Results Display

Various ways of results display are available.

| Displacement | Max/Min Values | Graph |

Max/Min Values figure annotations:
<Maximum value>:377.423pm
<XYZ coordinates>:0.0, 0.0, 9.5

<Maximum value>:0 m
<XYZ coordinates>:-1.58114, -4.74342, 0.00000 mm

Graph: Displacement-Magnitude
Displacement-Magnitude [pm] vs Distance from starting-point [mm]

Results Display
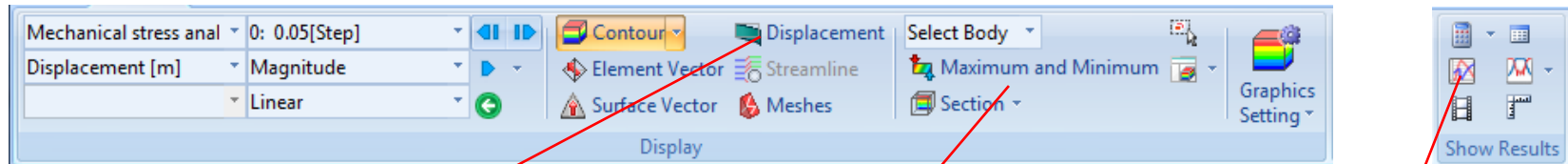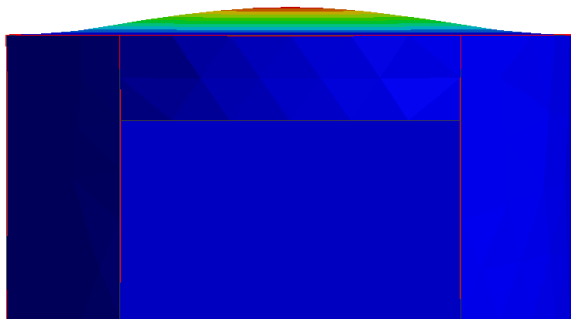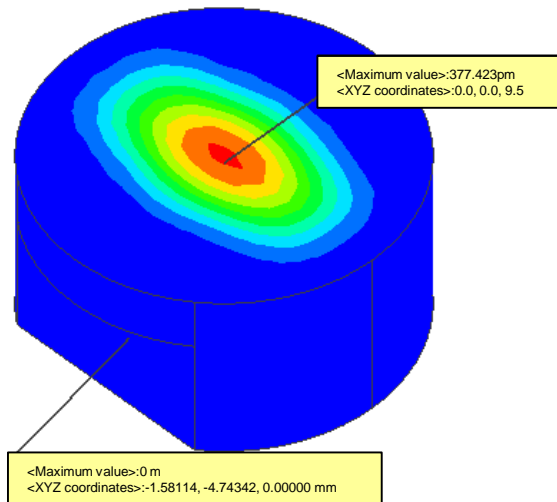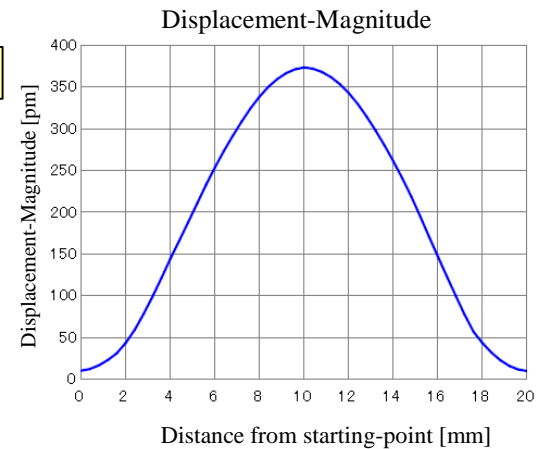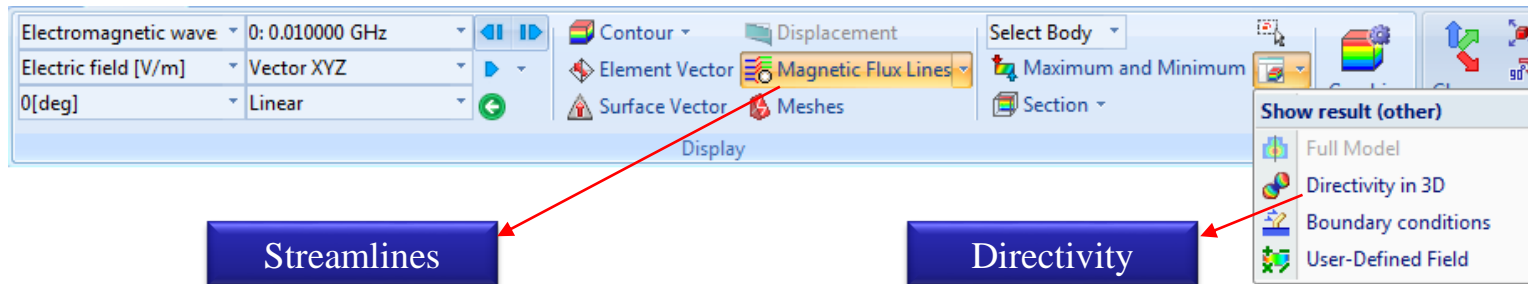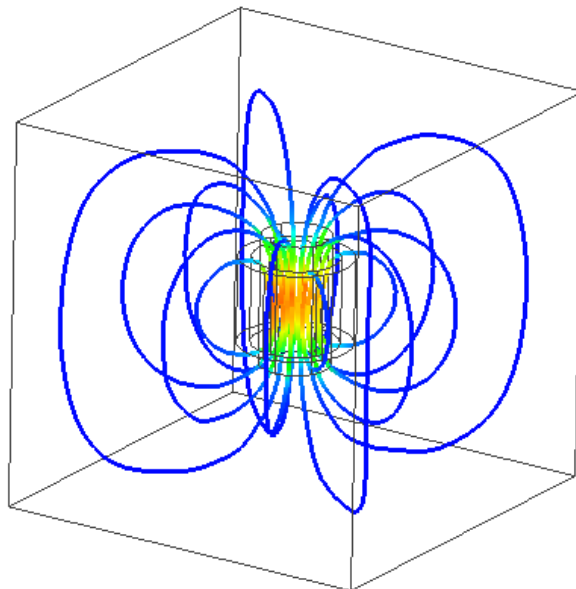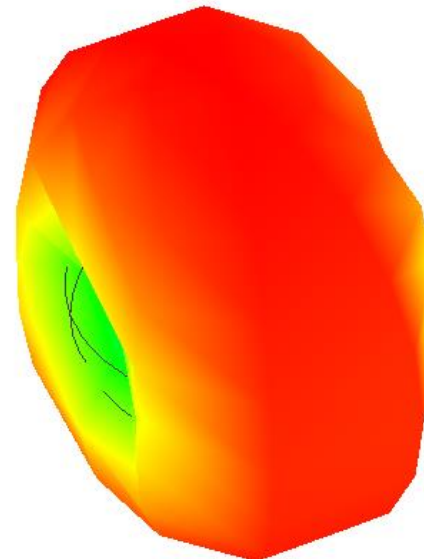
Various ways of results display are available.

Streamlines

Directivity

Streamlines such as fluid velocity
and magnetic flux can be displayed.

Directivity of radiated power of antenna
can be displayed in 3D.

Useful Features

Parametric Analysis

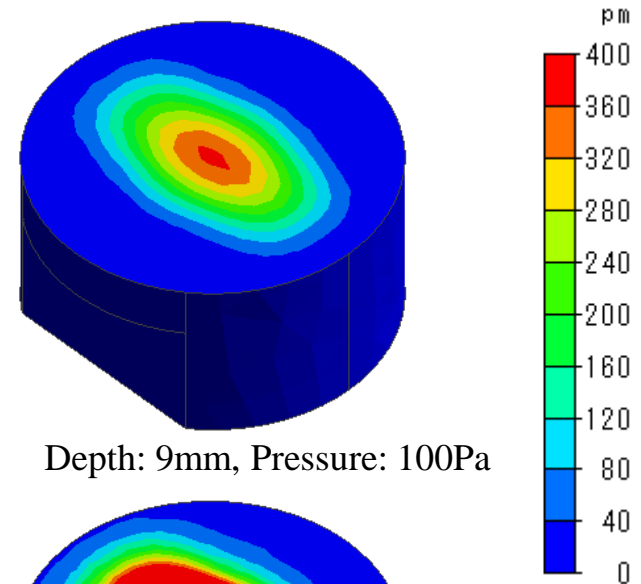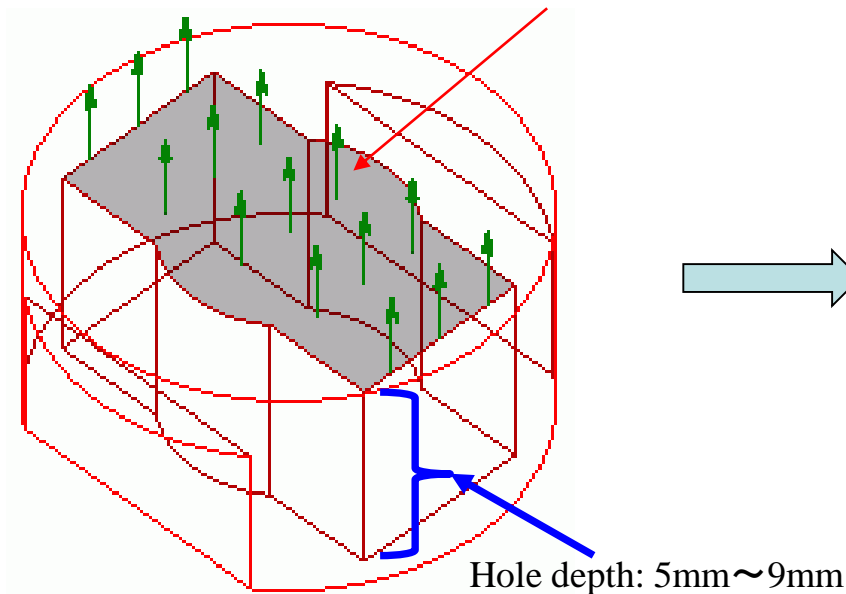Batch Simulation

User Database

Macro

Parametric Analysis

Variables are assigned to model's dimensions and parameters for each condition.
By sweeping the variables in a specified range, the analysis is executed.
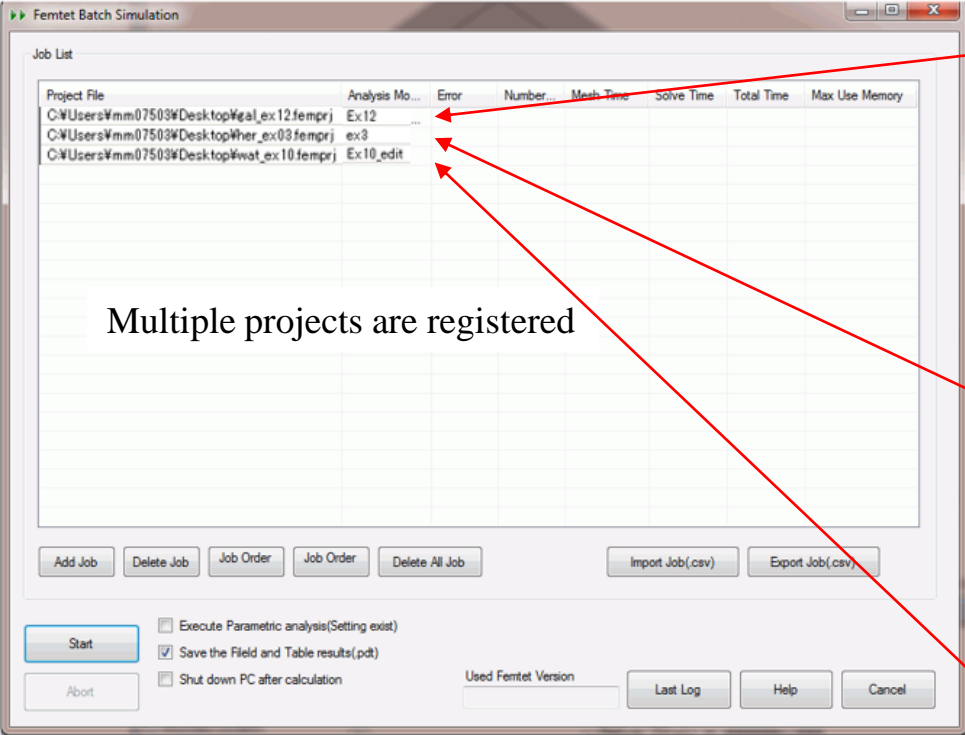
Internal Pressure of Container with a Hole

Pressure: 100Pa～300Pa

Hole depth: 5mm～9mm

Assign variables to internal pressure and a hole depth.

Depth: 9mm, Pressure: 100Pa

Depth: 9mm, Pressure: 300Pa

pm
400
360
320
280
240
200
160
120
80
40
0

Batch Simulation

Murata Software

Multiple projects can be registered as a job for simulation.

Multiple projects are registered
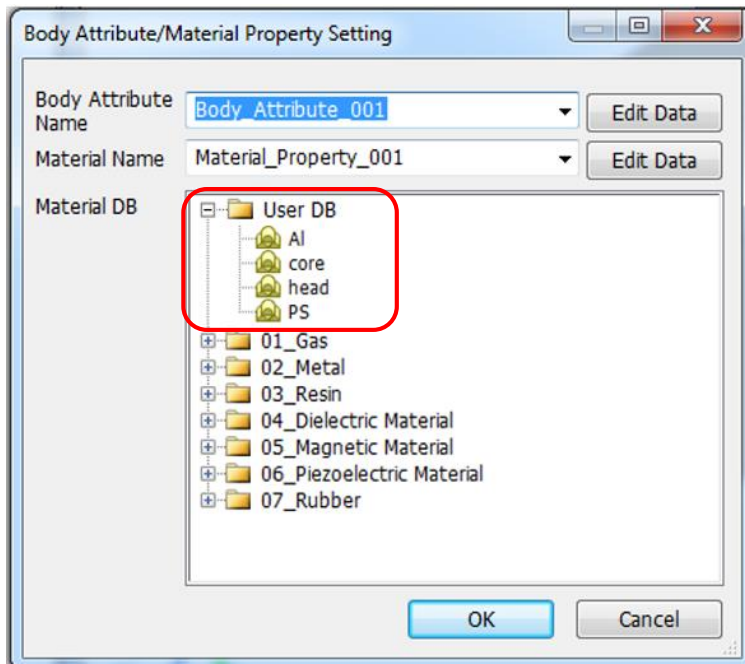
Mechanical Stress

Electromagnetic Waves

Thermal Conductivity

User Database

Body attribute, material property, boundary condition, and model form that are used for an analysis can be registered in the database for another analysis.

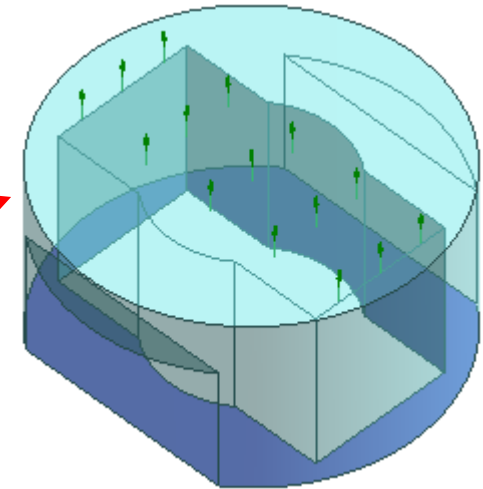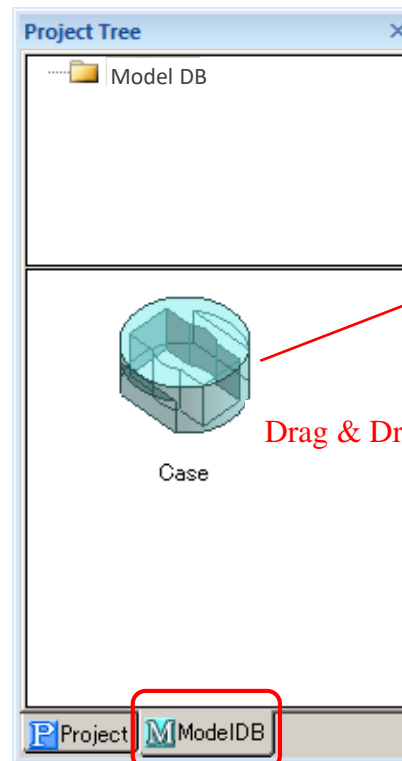| Material Database	Model Database

Register the material property of your own in the User Database.
It can be used for another analysis model.

Register a model in the User Database.
Drag and drop it for another project.

Drag & Drop

Macro

Macro allows you to customize Femtet functions.
For example,
automatic creation of a complicated analysis model,
automatic extraction of results of your choice,
recalculation by modifying model size or analysis conditions.

Visit our site for macro samples (https://www.muratasoftware.com/en/support/macro/)

Macro Samples

No12: Access log check macro.

Techniques Used in the Example

You can see the time you were logged in by this macro.

Simulation Image

Download

No11: Convert Nastran format to STL format with deformed shapes.

Techniques Used in the Example

This macro converts nastran format file that includes mesh data with displacements to STL format file.
It is possible to import the output STL format file into other CAD software.

Simulation Image

Download