

POLYGONALmeister  
**Operation Guide**  
**(Basic)**

(ver.7.0.0)

POLYGONALmeister is software for editing and evaluating the shape represented by the triangle group.

Chapters 1 to 5 of this manual explain how to operate POLYGONALmeister, and Chapters 6 and later explain how to set the functions and parameters of each command.

For details on how to use POLYGONALmeister, see "Manual" and "License Guide" on the "Help" tab.

In addition to this manual, the POLYGONALmeister operation manual includes the following documents.

- "Operation Manual (Introduction)"  
Explains the operation method for users who are new to POLYGONALmeister.
- "Operation Manual (Utilization)"  
Explains the recommended usage for each application.
- "Operation Manual (Example)"  
Describes operation examples using POLYGONALmeister. You can learn the functions and operation methods of POLYGONALmeister by following the procedure described.

The operation manuals can be downloaded from the POLYGONALmeister website.

# Table of Contents

Hold down [Ctrl] and click an item to jump to the corresponding page.

## I Overall explanation

### 1. Terms

Shape Terms  
Color Terms  
Operation Terms  
Screen Terms

### 2. Mouse Operation

Mouse operation  
Drawing Line  
Specification of Areas

### 3. Change Display Status

Display State Setting  
Crip  
Coordinate Axis View  
Mesh Comparison  
Screen Setting

### 4. Command Selection

Command Operation  
Order of operation

### 5. Operation Assistance

Operation-related Functions  
Result List Dialog  
Registration of Frequently Used Commands

## II Explanation of each command

### 6. File Tab

Import  
Export  
Import Shell  
Export Shells  
Export Faces  
CAD Surface Export

### 7. Home Tab

Cleaning  
Smoothing  
Simplify  
Subdivision  
Remesh  
Cut  
Trim  
Boolean Operation  
Flip Shells  
Delete Shells  
Scaling  
Move / Copy  
Drag Move  
Arrange Shells  
Primitive Solid

### 8. Edit Tab

Reshape CT Mesh  
Reshape Optical Mesh  
Defeature  
Fill Holes  
Reshape Hole  
Through Hole  
Fill Caves  
Remove Protrusion  
Delete Faces  
Advanced Self-intersection Removal  
Remove Acute Triangles  
Offset  
Thicken  
Morphing  
Projection Solid  
Difference Solid  
Make Flat / Cylindrical  
Transplant  
Loop Cut  
Sharpen  
Delete Boundary Faces  
Extension  
Flip Edge  
Merge Vertices  
Align Vertices  
Divide Faces

### 9. Evaluation Tab

Property  
Distance  
Coordinates  
Circle radius  
Distance Contour  
Display Section Lines  
Find Path  
Crease Detection  
Thickness Inspection  
Unevenness Test  
Display Enlarged Section Lines

### 10. Color Tab

Fill Color  
Texture Alignment  
Transfer Color  
Texture Reconstruction  
Convert to Vertex Color  
Segmentation  
Edit Segment Boundary

### 11. 3D Print Tab

Display Forming Table  
Forming Arrangement  
Pinned Cut  
Z Correction

### 12. Geography Tab

Terrain Meshing  
Create Grid Mesh  
Display Contour Lines  
Soil / Space Volume  
GPS Track Log Display  
Dekigata Heat Map

### 13. Relief Tab

Relief  
Embed Relief  
Stamp  
Letters Relief

### 14. Beta version Tab

Creased Faces Removal  
Fill Region  
Loop Trim  
Wrapping

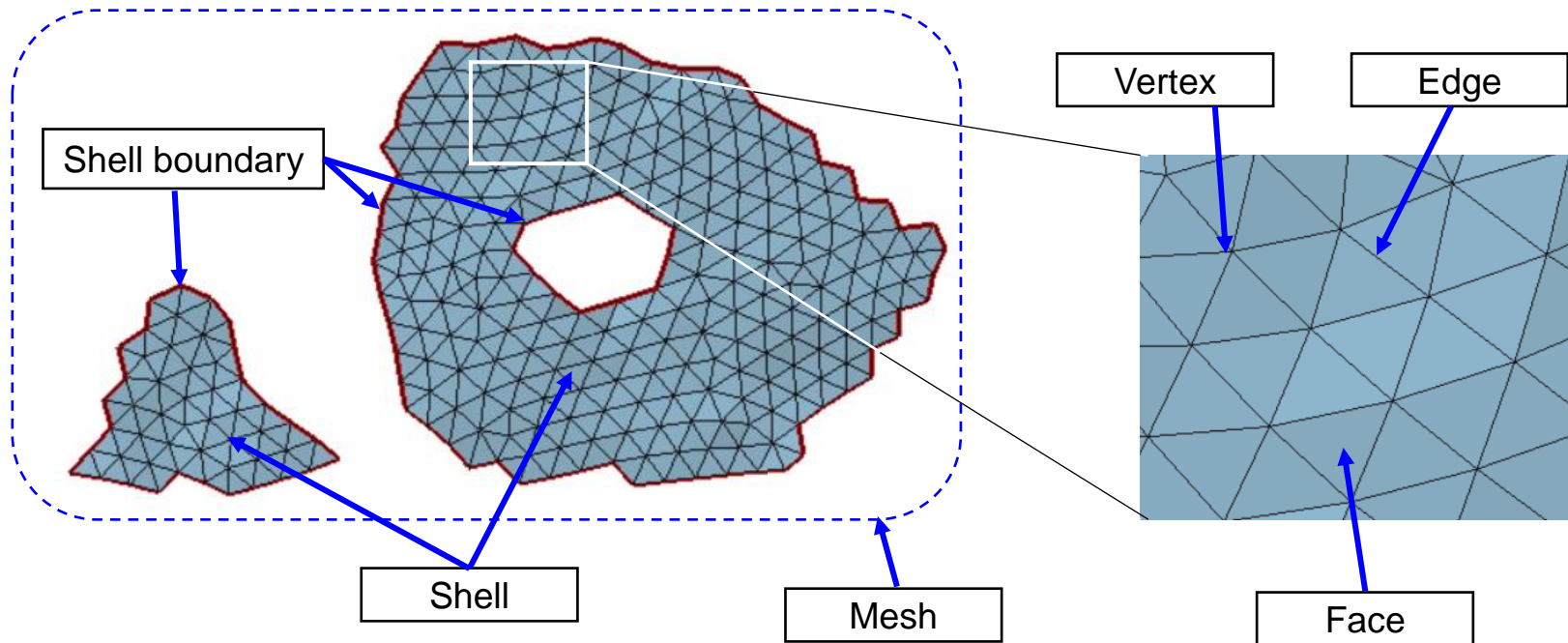
### 15. Help Tab

# Part I

## Overall explanation

# 1. Terms

## Shape Terms (1)



[Mesh] = The whole set of triangles

[Shell] = A collection of connected (sharing edges with adjacent triangles) triangles

[Shell Boundary] = Lines around a shell or lines around a hole in a shell

[Face] = A triangle that composes a mesh

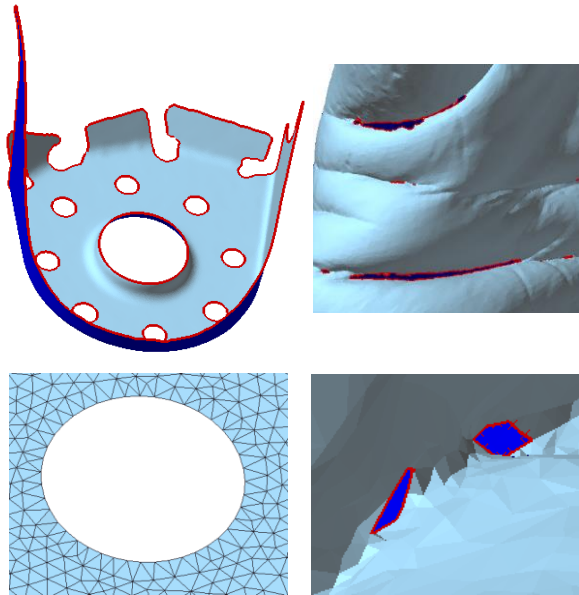
[Edge] = a side of a triangle

[Vertex] = triangle vertex

## Shape Terms (2)

There are three types of mesh holes handled by POLYGONALmeister.

### Hole

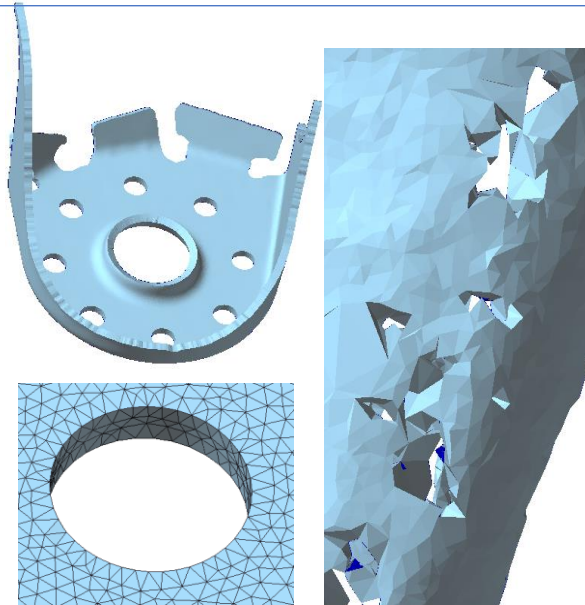


A hole in the surface of the mesh, also called a shell boundary.

Holes are present in optical measurements, such as when the face is missing due to lack of light or when measuring a model with holes.

To fill holes, use the "Fill Holes", "Cleaning (holes)", and "Defeature" commands.

### Through Hole

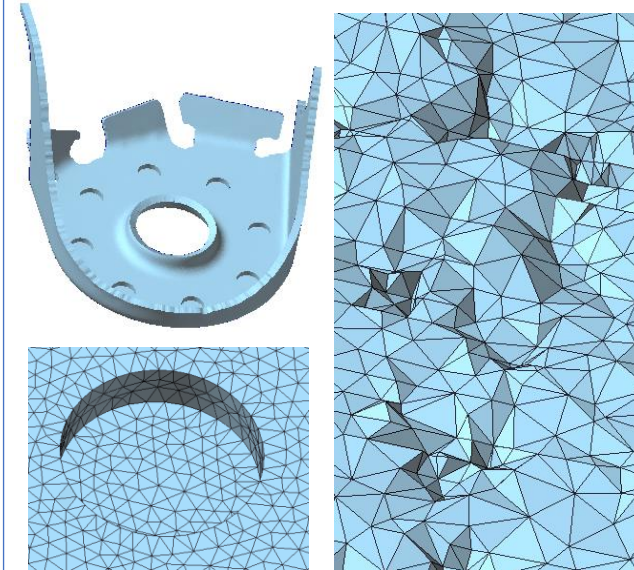


It is a hole that penetrates into a solid.

Through holes are present for bolting or when the model is thin on CT measurements.

To fill through holes, use the "Fill Caves", and "Defeature" commands.

### Blind Hole



It is a dent on the mesh surface.

The blind hole exists in the blind hole of the machine part, the dent of the measurement object, the roughness of the measurement surface, and so on.

To fill blind holes, use the "Fill Caves", and "Defeature" commands.

## Color Terms

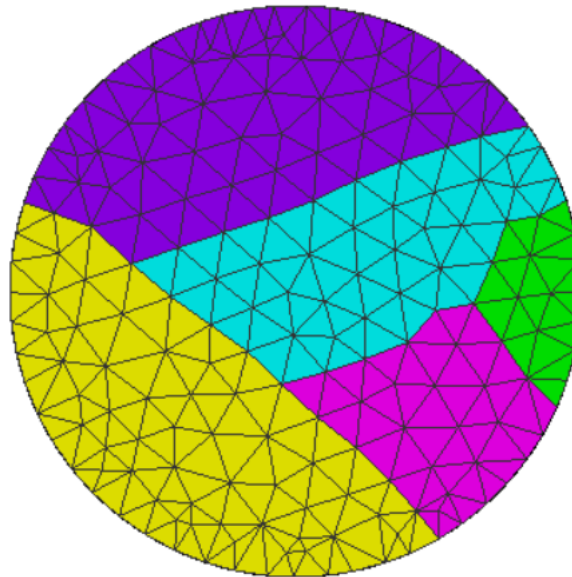
[ Texture Coloring ] = Triangular pieces of 'Texture Image' are assigned to faces.

[ Face Coloring ] = A face is painted in a single color.

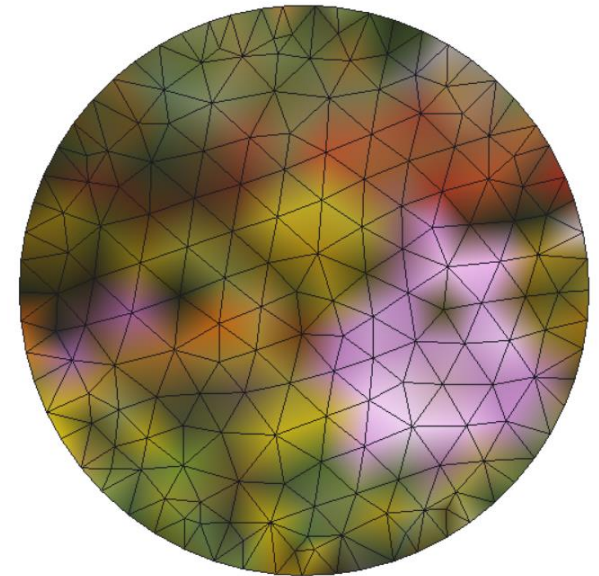
[ Vertex Coloring ] = Each vertex can have its 'Vertex Color'. Blend colors of the three vertices of a face.



Texture Coloring



Face Coloring



Vertex Coloring



# Operation Terms

## [ Commands ]

A command is a unit that a user executes functions.

## [ Temporary Figures ]

Temporary Figures are displayed only temporarily by the System in order to help user's command inputs and display calculation results.

## [ Result List ]

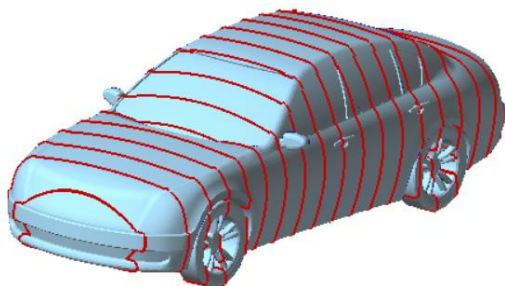
Temporary Figures disappear when a command ends, but it is possible to save them using the Results List function until the System ends. Then, it is possible to switch display / non-display of saved Temporary Figures.

## [ Clip ]

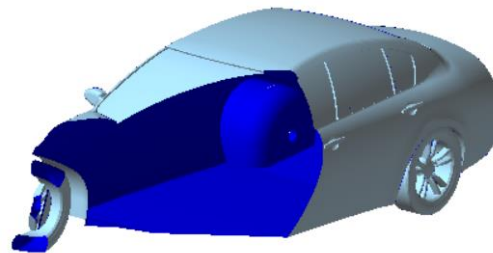
The "Clip" function hides figures that are positioned in the foreground of a plane. The plane specified is referred to as clipping plane, and can be transferred to arbitrary directions and positions. This function helps easy understandings of complicated structures by displaying internal side of solids.

## [ Box ]

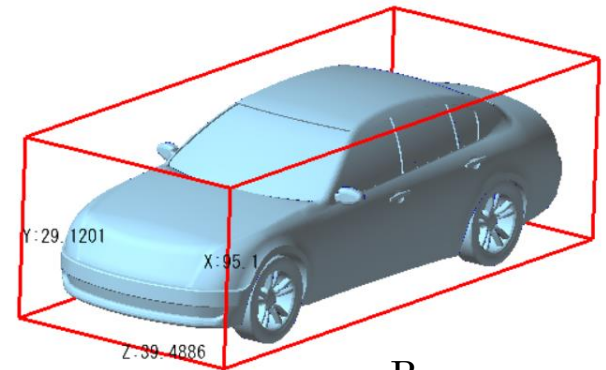
Box is a minimum-size cuboid containing a mesh with edges parallel to the coordinate axes.



Temporary Figures



Clip



Box

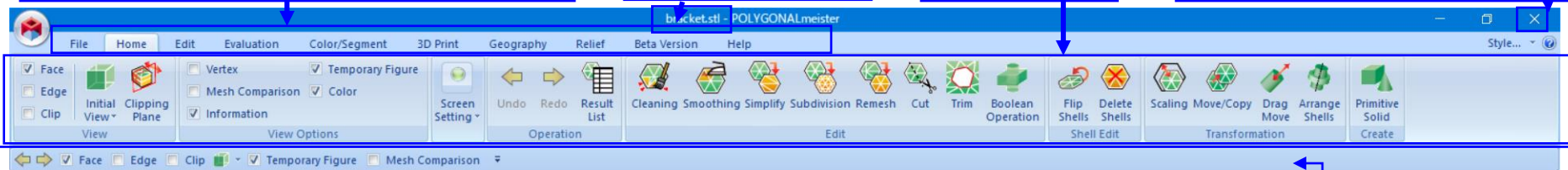
# Screen Terms

Tabs (major classification of commands)

Read file name

Commands

Exiting POLYGONALmeister



A file (size: 196.52 × 194.965 × 147.842) has been imported.

Command Dialog

Cleaning [How to use](#)

Simple [Details](#)

CAD Data  Measurement Data

Inspection Items	Detection Qty	After Repair
Reversed Faces	?	-
Gap between Shells	?	-
Irregular Connection	?	-
Small Holes	?	-

Fill holes with  or less edges (the number of edges around the hole) that make up the shell boundary

[Display Result List](#)

[Inspect](#) [Repair](#)

Faces: 155,680  
Vertices: 79,158

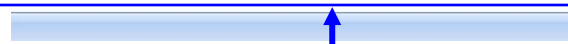
Message from the system

QAT (Register frequently used commands)

Number of faces and vertices

Display of mesh  
Light blue = front side, outside of solid  
Dark blue = back side, inside of solid

Coordinate system



Command Dialogue (Area for displaying command parameters, etc. )

Figure Display Area

Button to stop processing the command

Status bar (Display the progress of processing)

## 2. Mouse Operation

# Mouse Operation

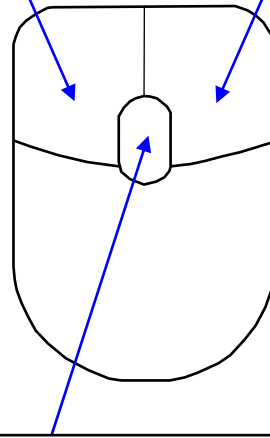
## Left button

Click: Select command or figure  
Drag: Draw a line  
Ctrl + drag: Translate the displayed

## Right button

Click: Change center of enlargement and center of rotation  
Drag: Rotate the displayed figure

(From the next page, there is an explanation of drawing lines and indicating areas by dragging the left button. )



## Mouse wheel

Rotation: Enlarge the displayed figure (PgUp/PgDn can be substituted)  
Drag: Translate the displayed (Ctrl + drag can be substituted)  
Ctrl + rotation: Move the clip plane (Ctrl + PgUp/PgDn can be used instead)

(Some mice do not have wheels, so an alternative operation method is available.)

## Drawing Line

There are commands for entering lines by hand, such as cutting (figure on the right).

Explains how to draw a "Line" and a "Free Line".

Hold down the left button and drag the mouse to draw a line.

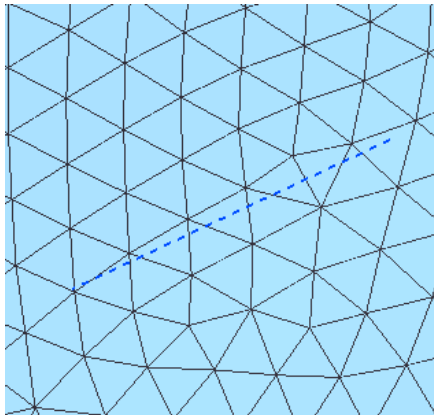
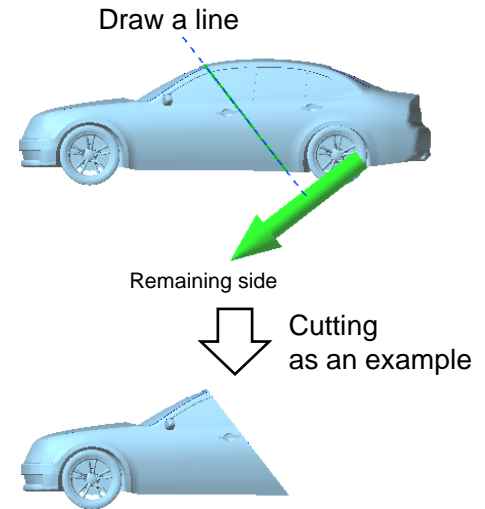
[Line]

A line segment connecting the start point and the end point of the drag is drawn as a "Line".

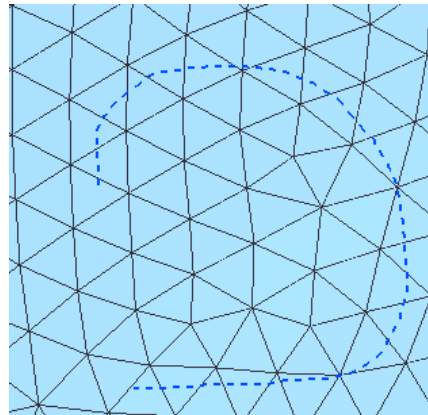
If you drag while holding down the "Shift" key, the line will be parallel to the frame of the screen.

[Free line]

The dragged path is drawn as a "Free Line".



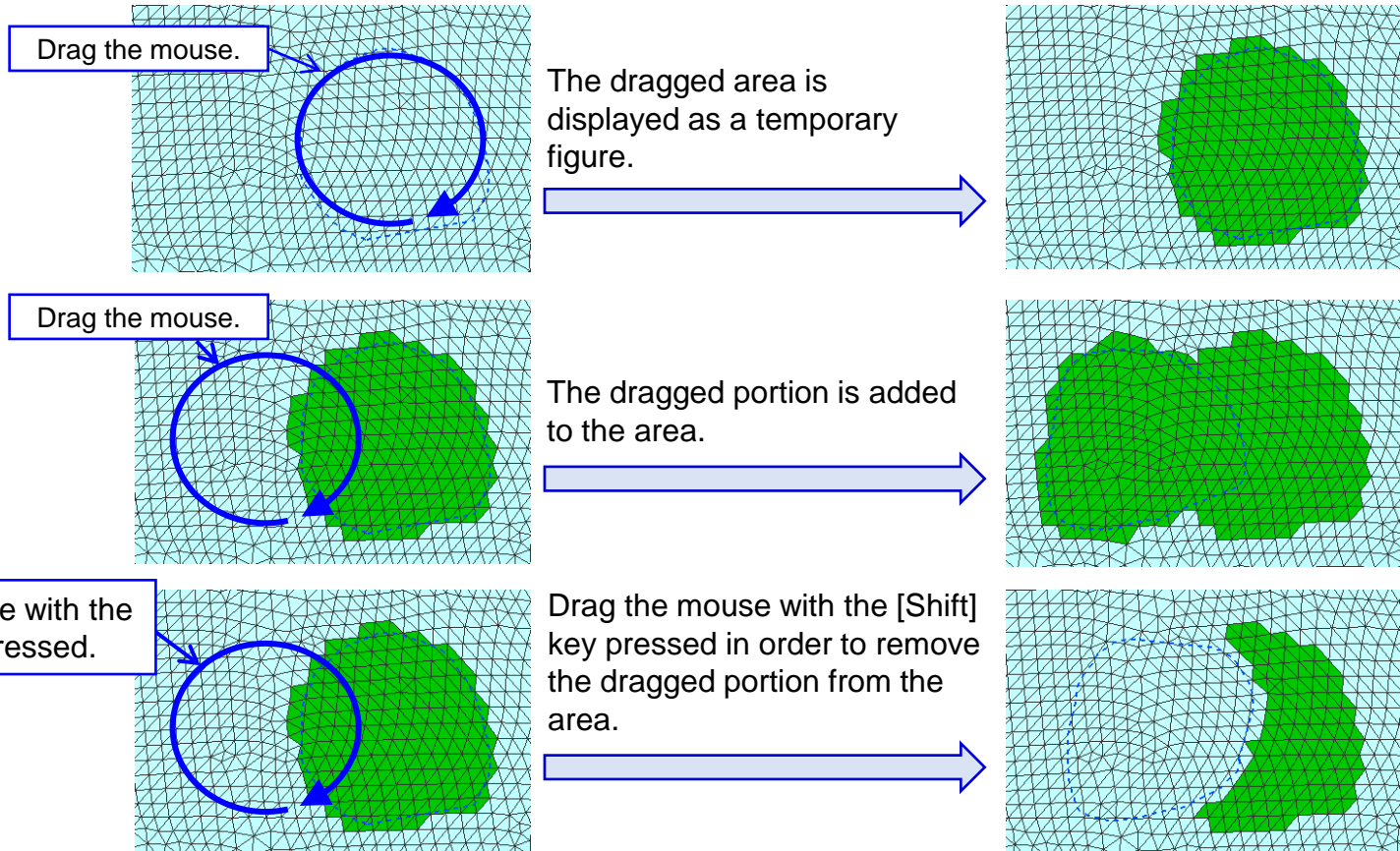
Line



Free Line

## Specification of Areas (1)

This section describes how to specify the area (faces) to be processed. As shown in the figure below, if you drag the mouse in a circle, the faces in the drawn area will be displayed in green, and will be processed by the command. Continue dragging will expand the area, and dragging with the [Shift] key pressed will remove the dragged area from the already specified area.



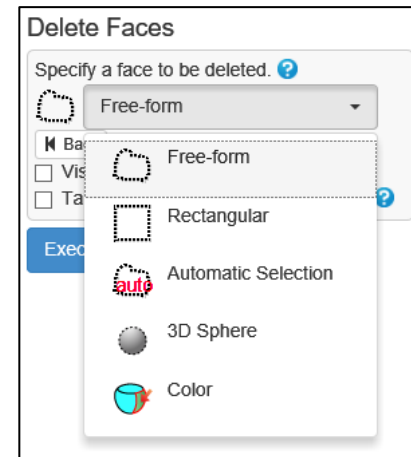
## Specification of Areas (2)

### ■ Area selection method

The figure on the right is the "Delete Face" command dialog.

Click the pull-down menu to see several ways to select an area. (The type of selection method displayed depends on the command.)

The various selection methods are as follows.



[Free-form] = The faces inside the dragged path are selected.

[Rectangular] = A group of faces in a rectangle whose sides are parallel to the screen frame is selected, with the drag start point and end point being two diagonal points.

[Automatic Selection] = A group of faces that smoothly connect to the clicked face is automatically selected.

[3D Sphere] = When the center of the sphere moves along the dragged line, the faces that go inside the moving sphere are selected. You can change the size of the sphere by holding down the "Ctrl" key and rotating the mouse wheel.

[Color] = The faces with the same color type as the clicked face is selected. Use the segmentation command to automatically color the mesh according to its shape features.

[Shell] = The faces that compose the clicked shell or the shell that is (even partly) included in the drag range are selected.

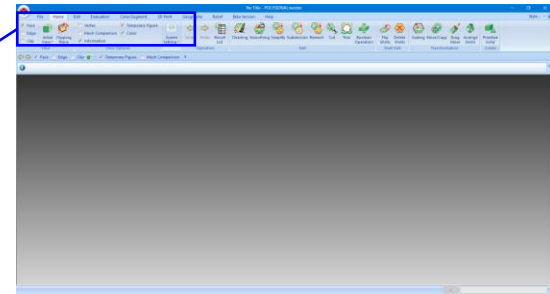
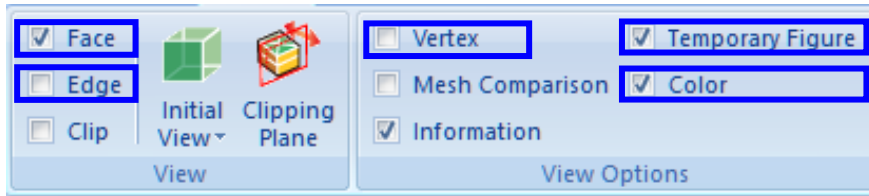
### ■ Click on the face

When you click on a face, if the face is selected (included in the area), it will change to the unselected state (not included in the area), and if it is not selected, it will change to the selected state.

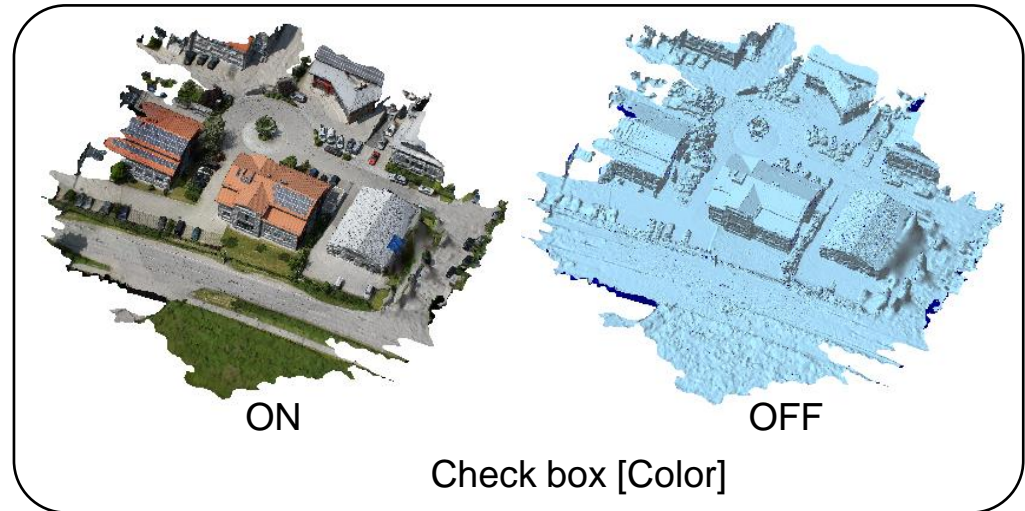
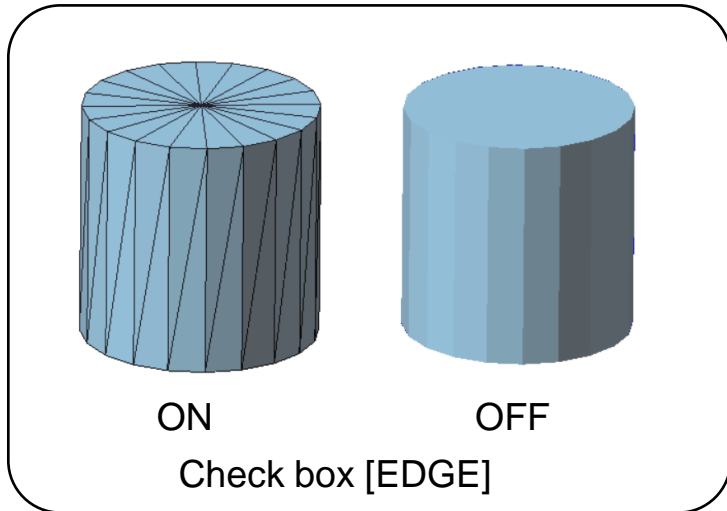
## 3 . Change Display Status



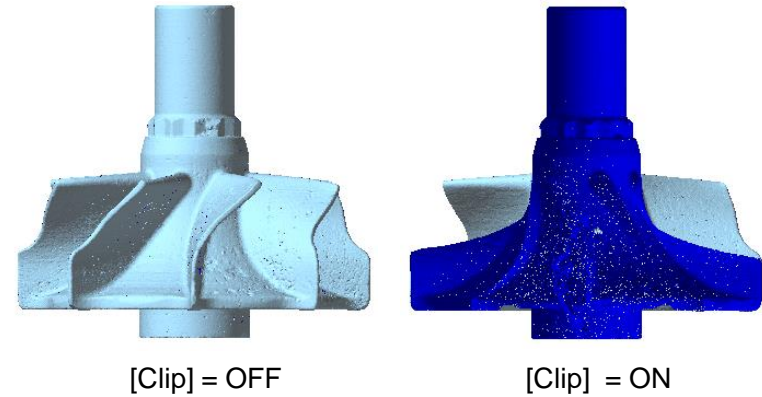
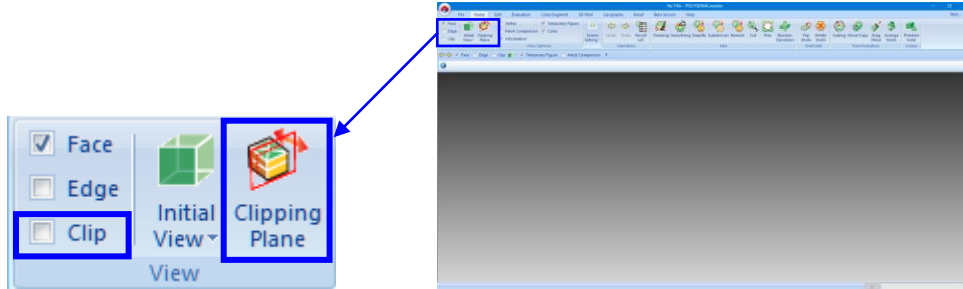
# Display State Setting



- [ Face ]                    Checked: the face (triangular) is displayed.  
There is a distinction between the front and back of the face, the front side is displayed in light blue and the back side is displayed in dark blue.
- [ Edge ]                    If checked, edges (sides of triangles) are displayed. (Left in the figure below)
- [ Vertex ]                  If checked: marks appear at the vertices of the triangles
- [ Temporary Figure ]    It is possible to switch the display (ON) / non-display (OFF) state of the temporary figure that shows the confirmation of the input operation and the processing result.
- [ Color ]                    It is possible to switch the display (ON) / non-display (OFF) state of the color pattern attached to the mesh. (Right in the figure below)



# Clip



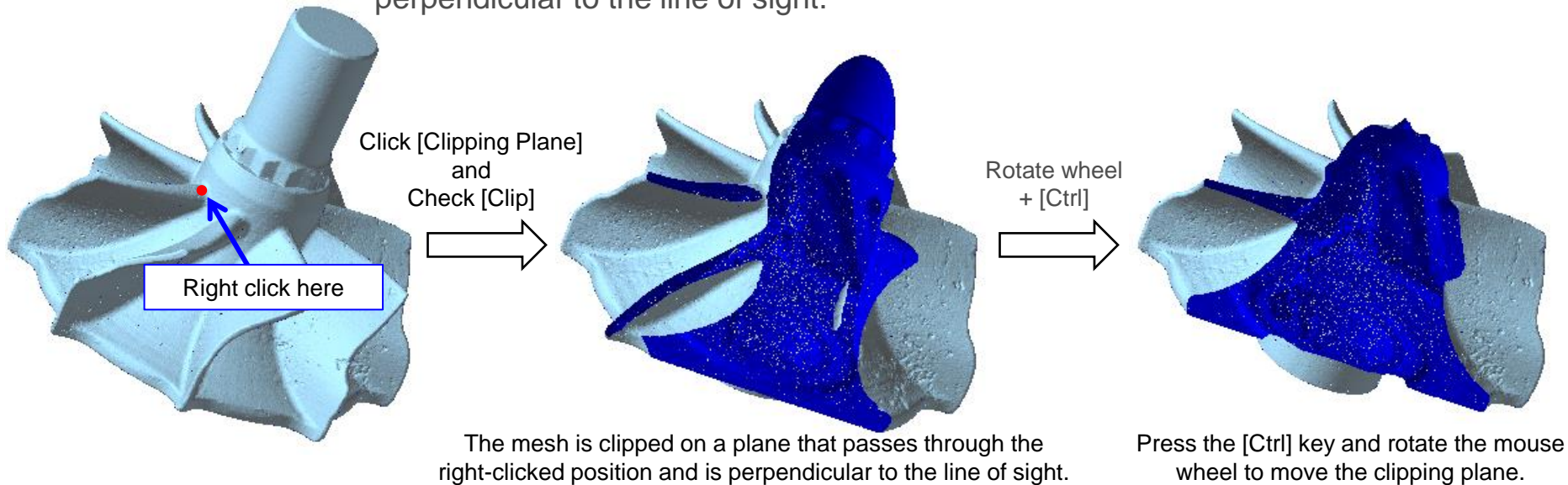
[ Clip ]

If checked, the figure in front of the clipping plane will not be displayed.

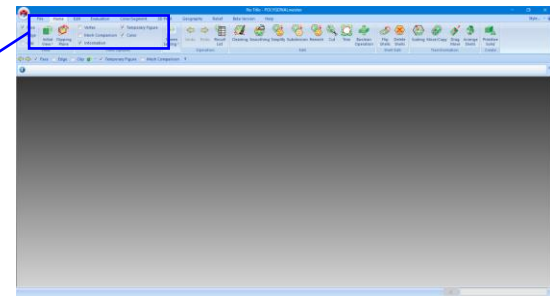
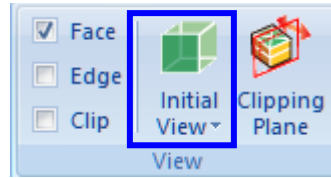
It is possible to move the clip surface by pressing [Ctrl] key and rotating the mouse wheel (or by pressing the [Ctrl] key and pressing the [PgUp] or [PgDn] key).

[ Clipping Plane ]

If you right-click a point on the shape and then click [Clipping Plane], the clipping plane is set to a plane that passes through the right-clicked position and is perpendicular to the line of sight.




# Coordinate Axes View




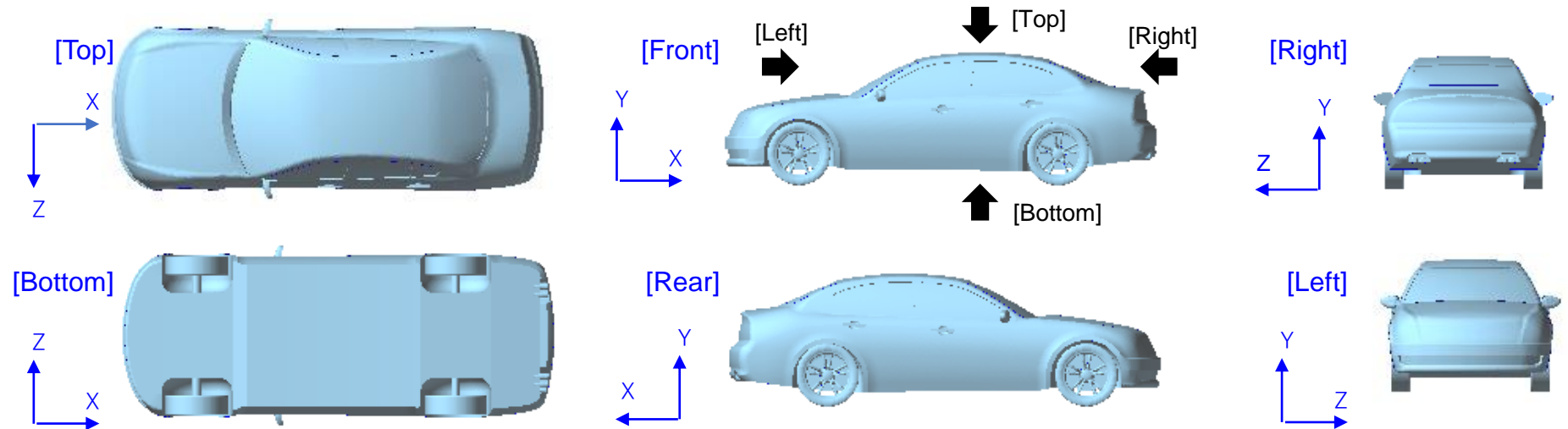
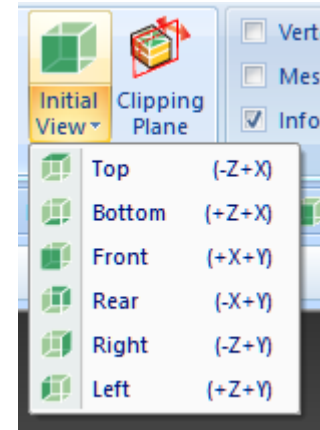
By using [Initial View], it is possible to see the figure from the coordinate axis direction.

The "initial display" is divided into  at the top and  at the bottom.

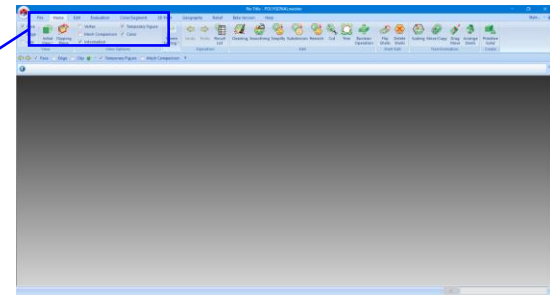
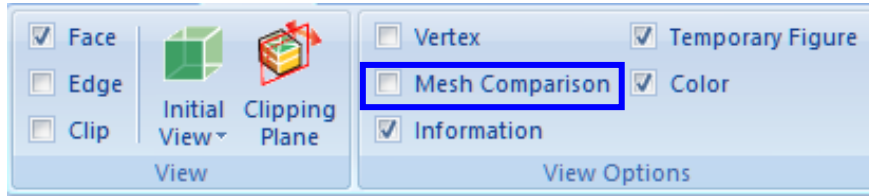
Click the icon [Initial View]  in order to display a view of an imported mesh. In this display state, the figure is viewed from the Z+ direction, and the right direction of the screen is X+ and the upward direction is Y+.

Click  to display the pull-down menu as shown on the right.

If you select [Front], the display will be the same as clicking . In this state, the view from above corresponds to the [Top] of the pull-down menu.

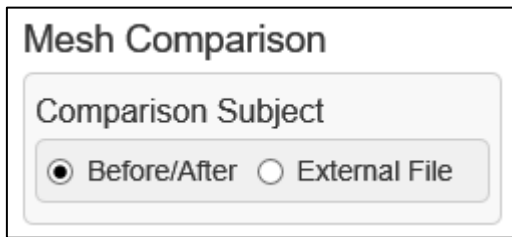


# Mesh Comparison



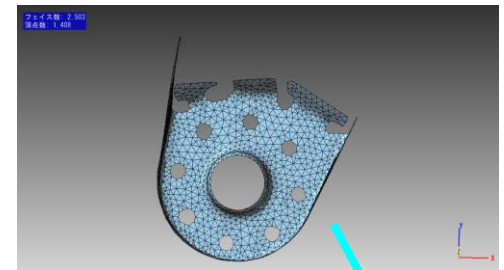
When the "Mesh comparison" check box is turned ON, the graphic display area is divided into left and right, the mesh that was displayed until then is displayed on the right side, and before processing (when "Undo" is performed) on the left side. At the same time, mesh comparison window is displayed. If the viewing state of the mesh on the right side is changed, the viewing state of the mesh on the left side will be changed accordingly.

If you select "External File" from the radio button, the mesh of the external file will be displayed on the left side.



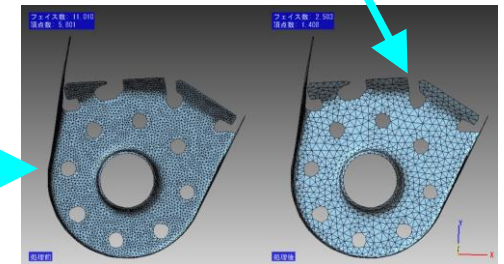
Mesh Comparison window

[Mesh Comparison] OFF



[Mesh Comparison] ON

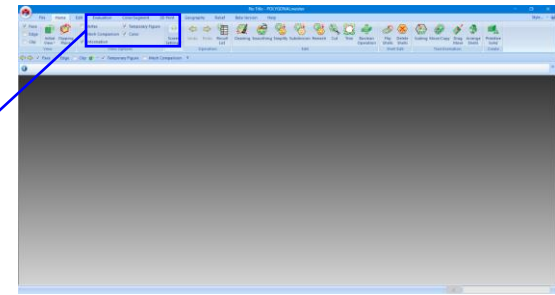
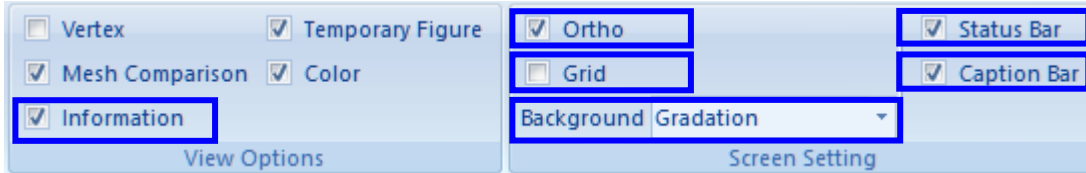
The shape before processing or  
The shape in the external file



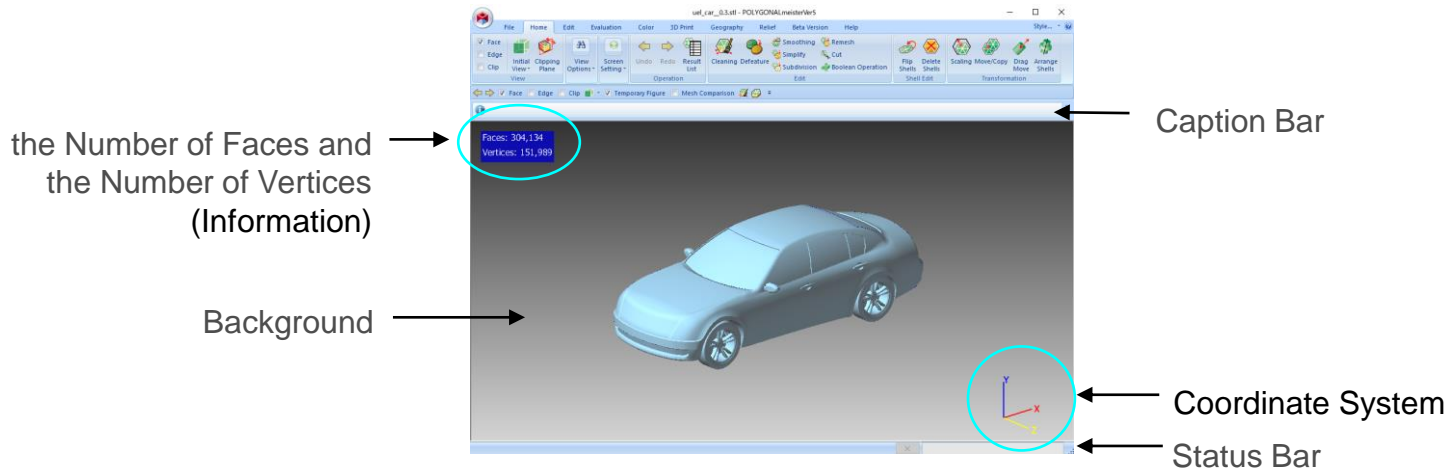
Left side

Right side

# Screen Setting



- [Information] When turned on, the number of faces, the number of vertices, and the coordinate system are displayed.
- [Ortho] It is possible to switch the display method of figures between parallel projection (ON) and one-point perspective projection (OFF).
- [Grid] When turned on, lines parallel to the X and Y axes are displayed, and a red sphere indicating the origin is displayed.
- [Background] It is possible to select the background color of the graphic display area from gradation / black / white / gray.
- [Status Bar] When turned on, the process stop button and progress display area at the bottom of the screen disappear.
- [Caption Bar] When turned on, the area for displaying messages from the system disappears.



# 4 . Command selection

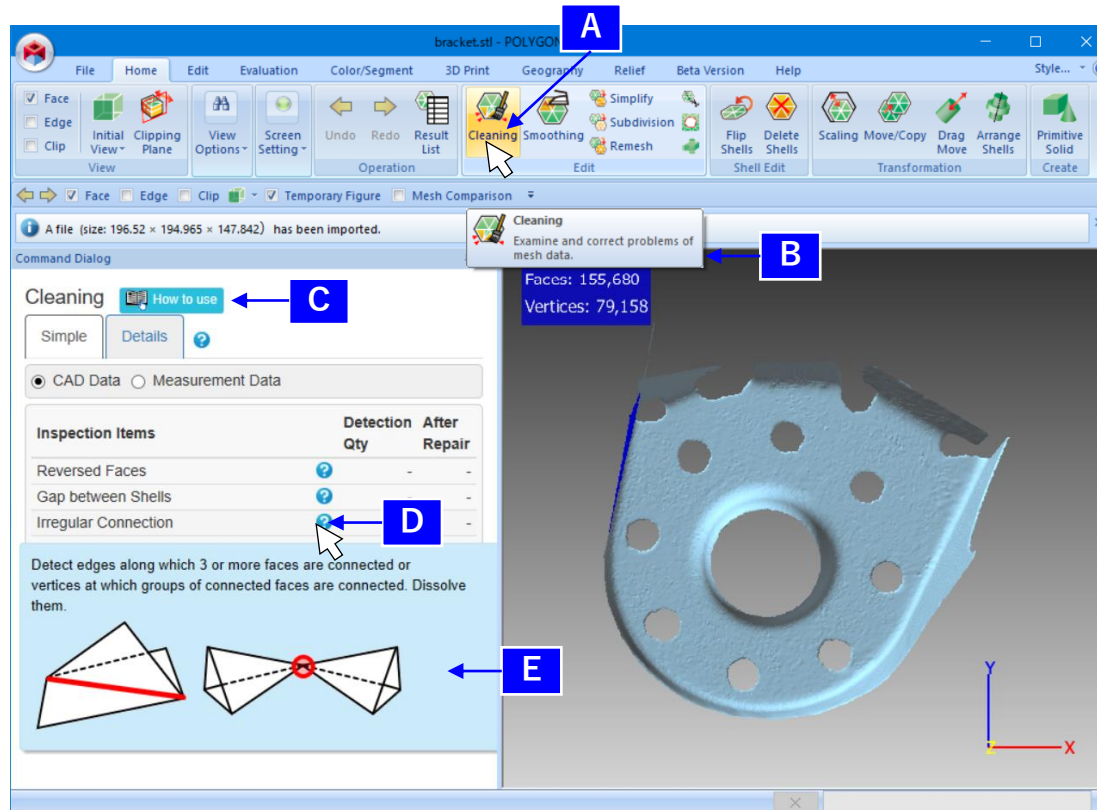
# Command Operation

Switch the "tab" and select the "command" to use.

Hover your mouse pointer over the "command" icon (A in the figure below) to see a functional explanation of the command.

Click the "Command" icon to display the command dialog for entering command input parameters.

Click the "How to use" button (C in the figure below) in the command dialog to open a PDF file of how to use (the same content as the corresponding page in this document). If you hover the mouse pointer over the "?" Mark (D in the figure below), the explanation of the input item (E in the figure below) will be displayed.

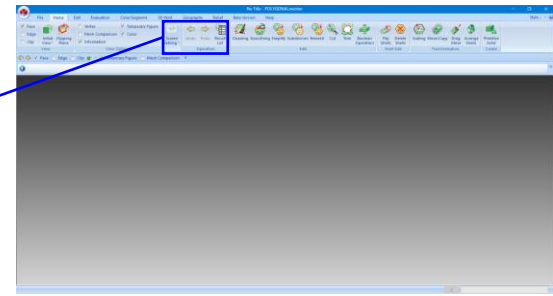






# 5 . Operation Assistance

# Operation-related Functions



- [ Undo ] It is possible to restore the mesh to the state it was in before the last operation (before the command with the shape change).
- [ Redo ] Cancel [Undo] to return to the post-execution state.
- [ Result List ] Click to display the dialog for managing the result list (saved temporary figure). The result list function allows you to save temporary figures that disappear when another command is selected until you exit POLYGONALmeister.

Click to display the "Result List Dialog"

Click to register the temporary figures in the "Result List" and display the "Result List Dialog".

"Result List Dialog" (The operation method will be explained on the next page.)

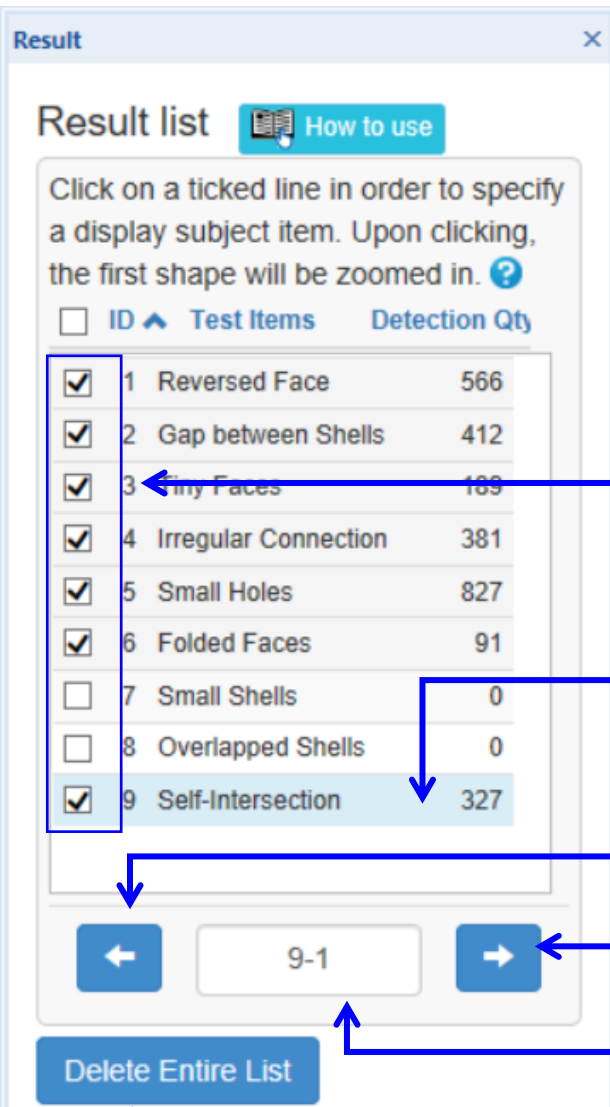
ID	Test Items	Detection Qty
<input type="checkbox"/>	1 Reversed face	0
<input type="checkbox"/>	2 Gap between shell boundaries	0
<input checked="" type="checkbox"/>	3 Tiny face	132
<input checked="" type="checkbox"/>	4 Non-manifold	6081
<input type="checkbox"/>	5 Hole	0
<input checked="" type="checkbox"/>	6 Folded face	169
<input checked="" type="checkbox"/>	7 Small shell	855
<input type="checkbox"/>	8 Overlapped faces	0
<input checked="" type="checkbox"/>	9 Self-intersection	942

Temporary Figure  
(Disappears when another command is selected )

Click the [Display Result list] button in the command dialog

Result List Dialog  
(Remain until POLYGONALmeister is exited)

# Result List Dialog



This section describes the dialog that appears when [Display Result list] button is clicked.

With this dialog, it is possible that:

- Change the display / non-display state of the saved temporary figure for each inspection item.
- Display the figures in the item in order (Click the arrow button to automatically change the view state)

The temporary shape of the checked line is displayed.

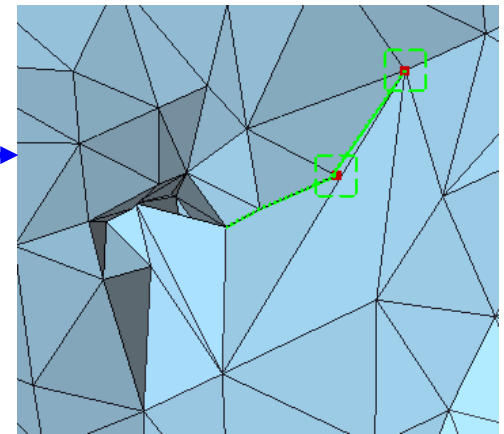
Clicking on a row changes the viewing state to the position of the first saved temporary figure that corresponds to the clicked item.

The viewing state changes to the position of the previous saved temporary figure.

The viewing state changes to the position of the next saved temporary figure.

It means the first saved temporary figure of the ninth item.


Deletes all items saved in the result list.

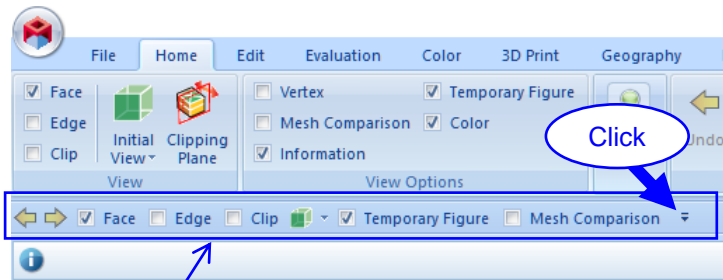


The viewing state changes to the position of the saved temporary figure.

# Registration of Frequently Used Commands (1)

By registering a command in QAT (Quick Access Toolbar), it is possible to start the registered command without switching tabs.

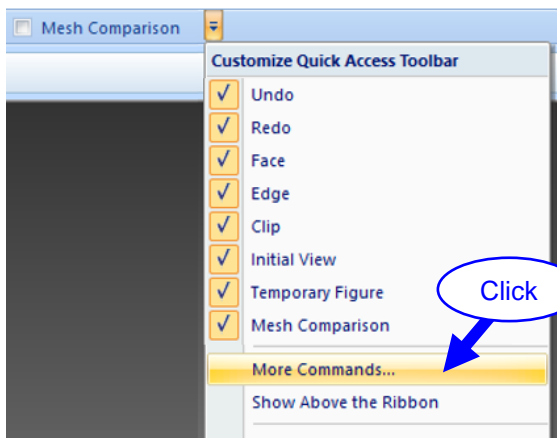
(1) Click the icon  at the right end of the QAT area



QAT area

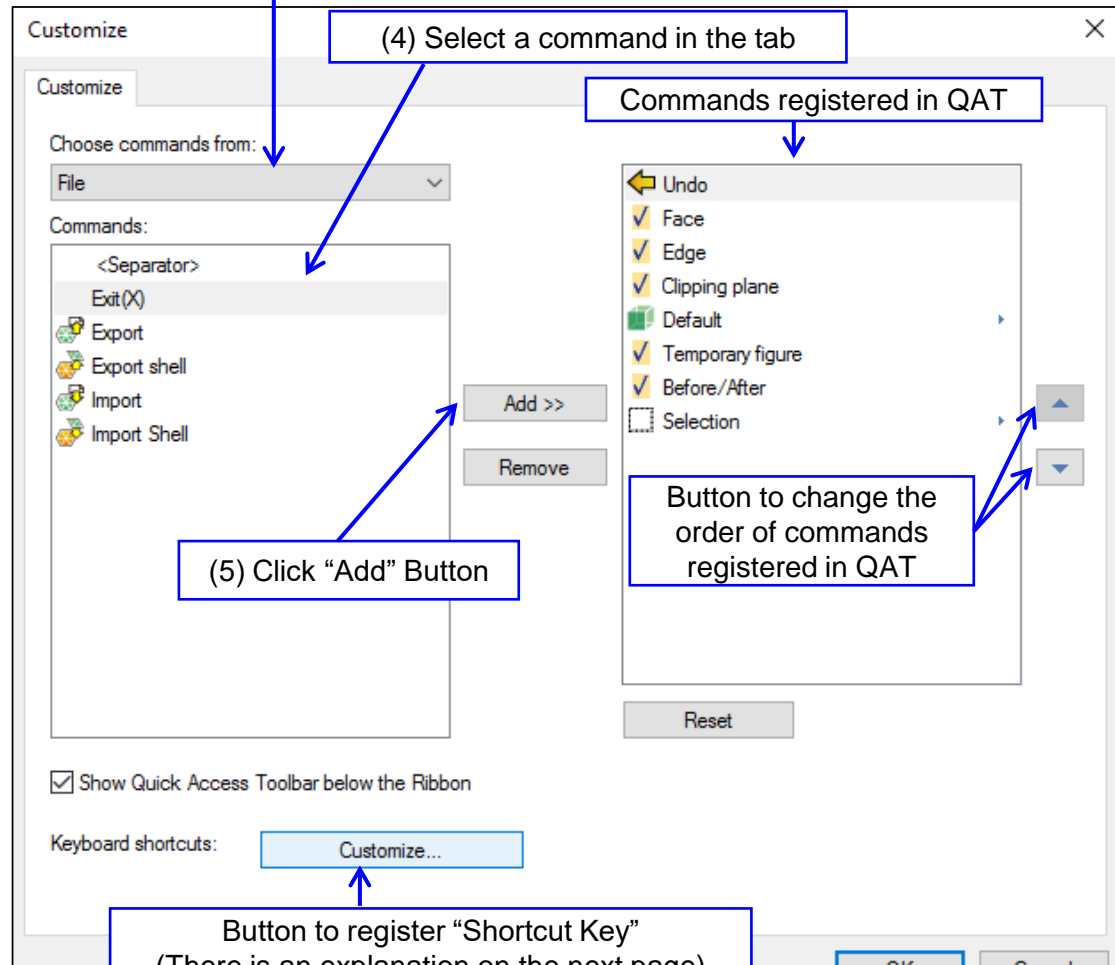
A pull-down menu is displayed

(2) Click "More Command..."



(3) Select tab

(4) Select a command in the tab



(5) Click "Add" Button

Button to change the order of commands registered in QAT

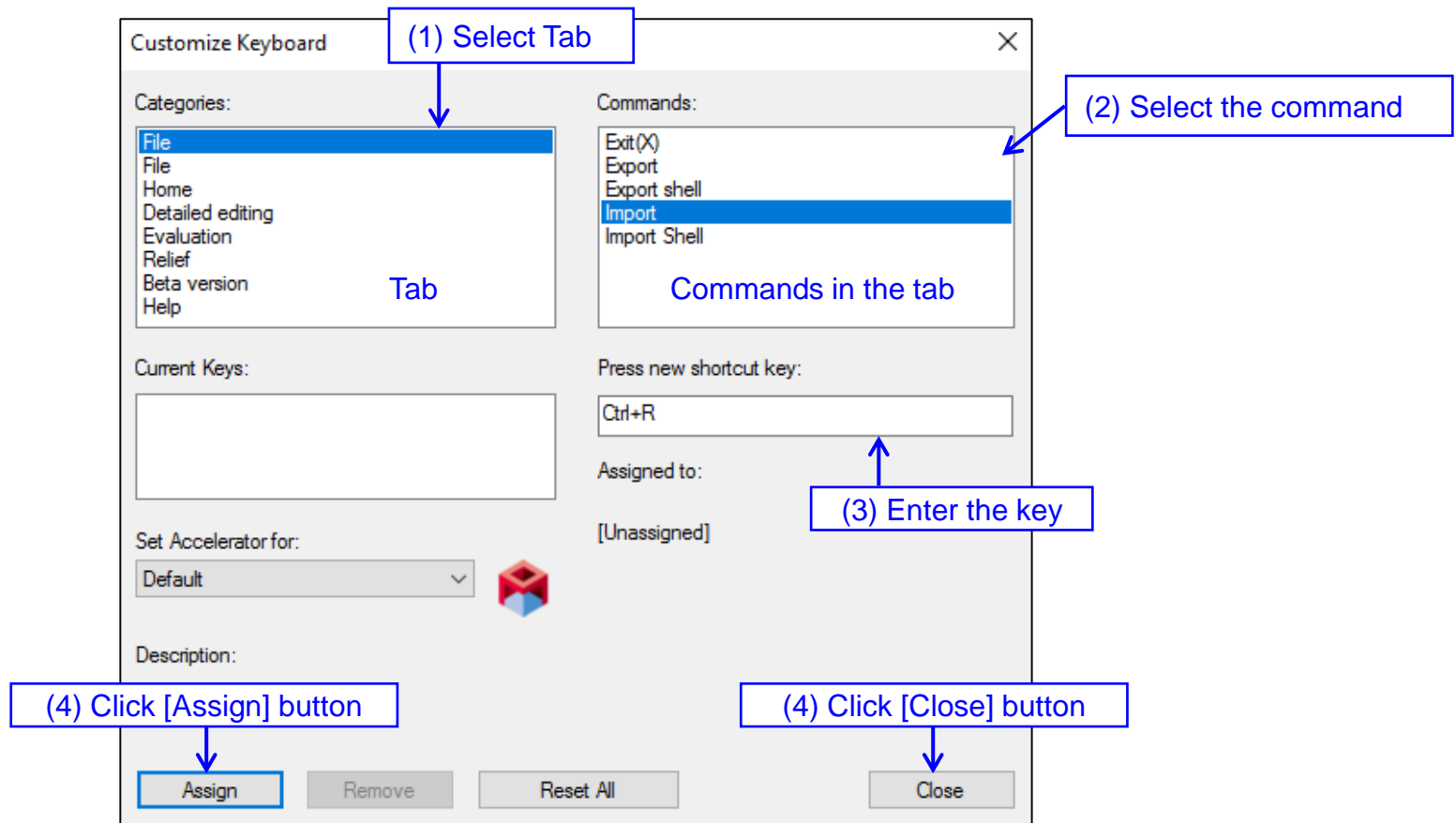
Button to register "Shortcut Key"  
(There is an explanation on the next page)

## Registration of Frequently Used Commands (2)

If a keyboard key is assigned to a command, it is possible to start the command by pressing the key.

This function is useful for selecting frequently used commands.

Click the "Customize..." key on the QAT window on the previous page to display the dialog shown below. The figure below is the example of registering "Ctrl + R" (operation of pressing "Ctrl" key and "R" key at the same time) to start "Import" command.



# Part II

Explanation of each command

# 6 . File Tab

# Import

This command reads polygon data from an external file.

It is possible to import the following types of files:

- OFF file
- OBJ file
- PLY file
- STL file
- VRML file
- NAS file
- BDF file
- BLK file

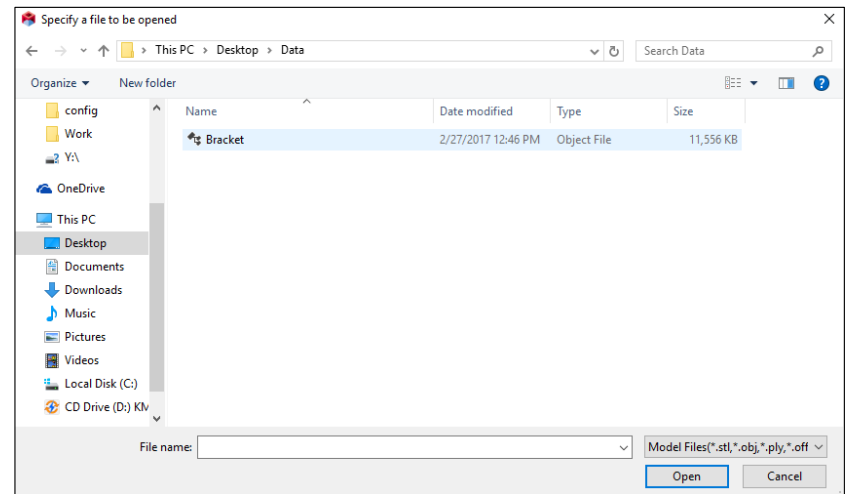
It is possible to import the color information of the following types of files.

	Texture	Face Color	Vertex Color
OBJ	✓	✓	
PLY		✓	✓
VRML	✓	✓	✓

Select "Import" command to display the file selection dialog. (Right figure)

Select the file to import and then click the "Open" button.

If a mesh already exists, the mesh will be deleted and the specified file will be imported.



The file can also be imported by dragging and dropping from Explorer.



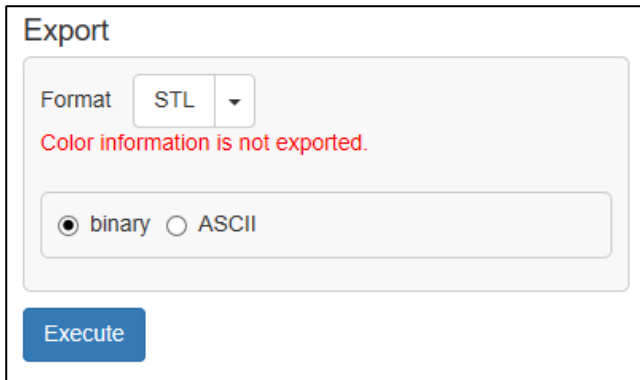
# Export

Export the mesh to an external file.

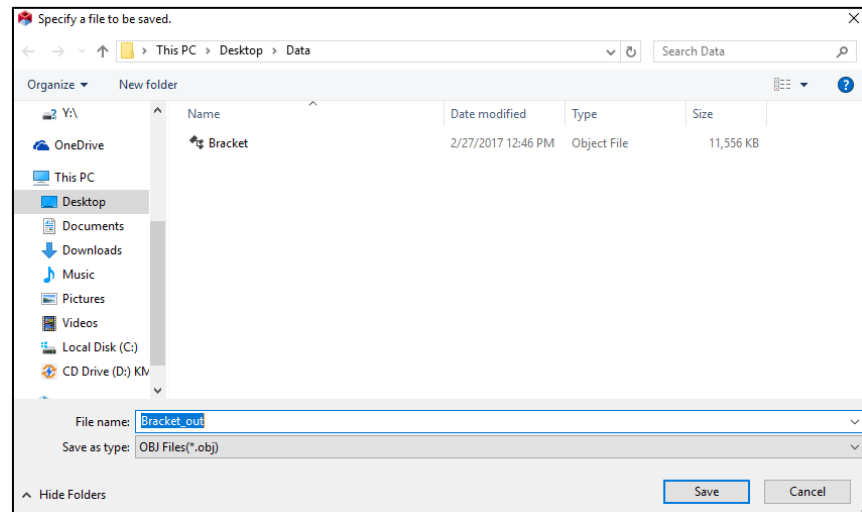
When "Export" command is selected, the command dialog on the left of the figure below is displayed.

Select the "Format" of the exported file and click the [Execute] button.

The file selection dialog on the right of the figure below is displayed. Enter the file name and click the "Save" button.



In "Output format", the file format when reading is set in advance. Change to another format if necessary.



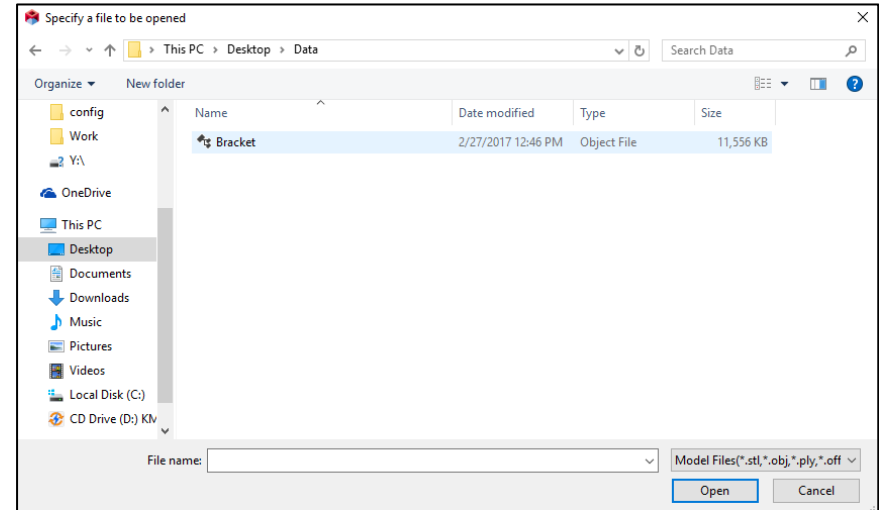
The file name is preset with the name of the imported file followed by "\_out". Rename it if necessary and click the [Save] button.

# Import Shell

Add the polygon data of the external file as shells when the mesh already exists.

When "Import Shell" command is selected, the command dialog is displayed. (Right Figure)

Select the file to import and then click the "Open" button.



The file can also be imported by dragging and dropping from Explorer.

# Export Shells

Export the specified shells to an external file.

Select the "Export Shells" command to display the command dialog shown below.

Select the shell to export and click [Execute] button. A file selection dialog is displayed. Enter the file name and click the "Save" button.

If multiple shells are selected, the shells will be exported to one file.

If checked, the selected shells will be deleted after exporting.

There are three ways to select the shells to export.

- 1) Click or drag the shells in the figure display area.
- 2) A shell consisting of less than the specified number of faces will be exported. Specify the number of faces and click the "Select shells with less than" button.
- 3) Click the row in the list.

The selected shells turn green and the check boxes in the corresponding rows of the list are checked.

Export Shells [How to use](#)

Delete exported shells [?](#)

Specify the target in order to export only some shells. [?](#)

Format

Color information is not exported.

binary  ASCII

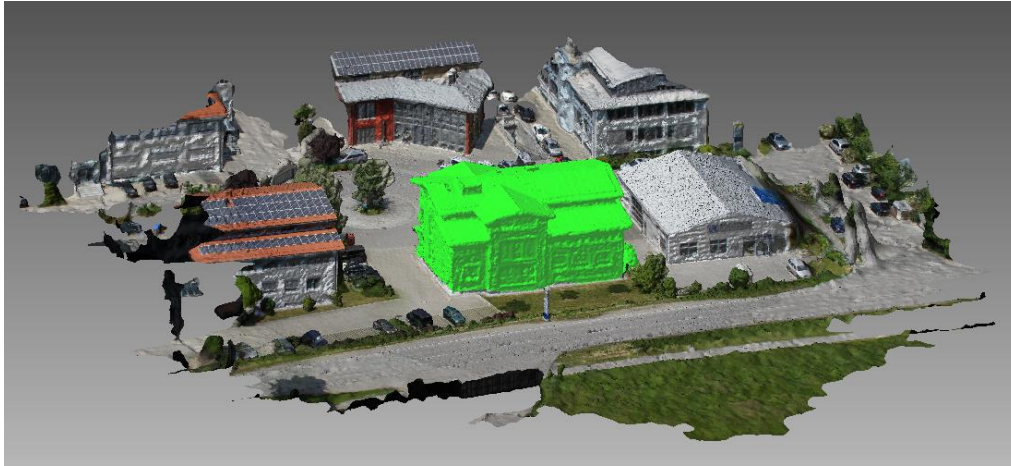
Select shells with less than  faces

<input type="checkbox"/>	Face Qty	Surface Area	Volume	Zoom
<input type="checkbox"/>	54332	93925.9	91134.2	
<input checked="" type="checkbox"/>	51838	92066.1	90002.3	
<input type="checkbox"/>	36380	44127.5	462300	
<input checked="" type="checkbox"/>	10732	6889.7	3476.58	
<input type="checkbox"/>	9262	27976.4	30271.5	
<input type="checkbox"/>	8742	6068.62	3276.25	

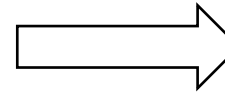
Execute

# Export Faces

Exports the specified faces to an external file.



Only the specified faces will be exported




If checked, the selected faces will be deleted after exporting.

If checked, all faces in the specified area will be deleted.  
If unchecked, the faces connected to the foremost face in the specified area will be deleted.

**Export Faces** [How to use](#)

Delete exported faces [?](#)

Specify the areas to be exported. [?](#)

 Free-form

[Back](#)

Visible Faces

Target all faces in the specified area. [?](#)

Format **STL**

*Color information is not exported.*

binary  ASCII

**Execute**

If checked, only visible faces in the specified area will be deleted.  
If unchecked, invisible faces in the specified area will also be deleted.

# CAD Surface Export (1)

This command creates a CAD surface out of a mesh surface and exports it into an IGES file. The delivered CAD surface quality depends upon the quality of mesh surface. Therefore, preprocess a mesh surface before delivering a CAD surface. Preprocessing is effective in improving CAD surfaces to be delivered. Please read the descriptions about preprocessing in the following pages.  
(\* ) We use the Library by TechnoStar Co., Ltd. for creating CAD surfaces.

If the message "The file could not be exported." is displayed for the first-time attempt to execute the "CAD Surface Export" command, right-click the Icon and choose "Run as Administrator" when booting POLYGONALmeister. This operation is not needed for the second-time execution and thereafter.

### CAD Surface Export [How to use](#)

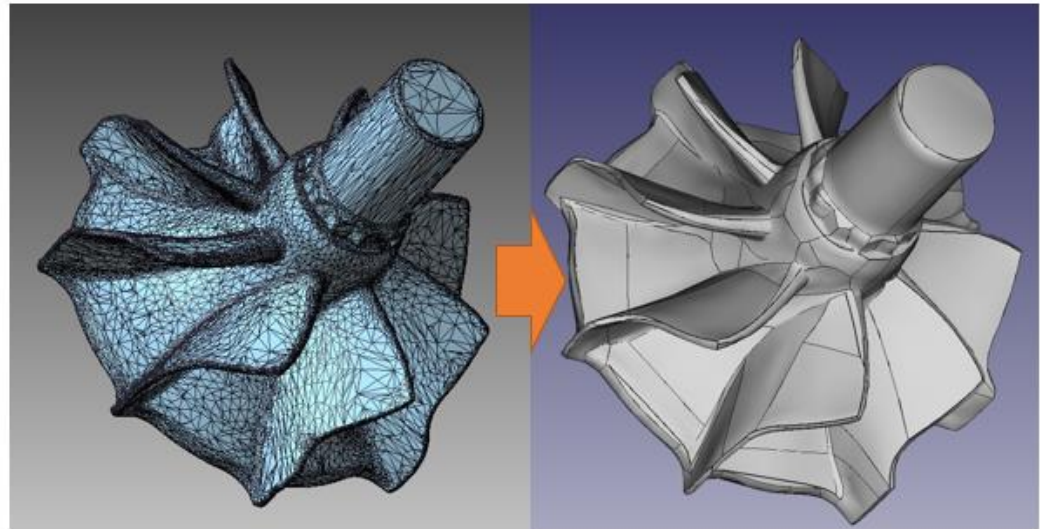
Confirm the exported IGES file with your CAD software.

Estimated calculation time  
100,000 polygons: 2 minutes  
1 million polygons: 15 minutes  
5 million polygons: 2 hours

It is effective to preprocess the meshes.  
Refer to the "How to use".

Export patterns [?](#)

Standard  Different Patterns



Choose "Standard" normally.

If favorable results are not delivered after choosing "Standard" for a mesh surface made of measurement data of human bodies and natural objects, try "Different Patterns".

The message "An IGES file was exported. Confirm with your CAD software" is displayed after the processing is finished.

For your reference, a command prompt may appear on the black backdrop (DOS screen) depending upon your PC environment. A disappearance of this screen indicates the finish of export.

# CAD Surface Export (2)

## ■ Preprocess Mesh Surfaces

The “CAD Surface Export” command automatically creates CAD Surfaces.

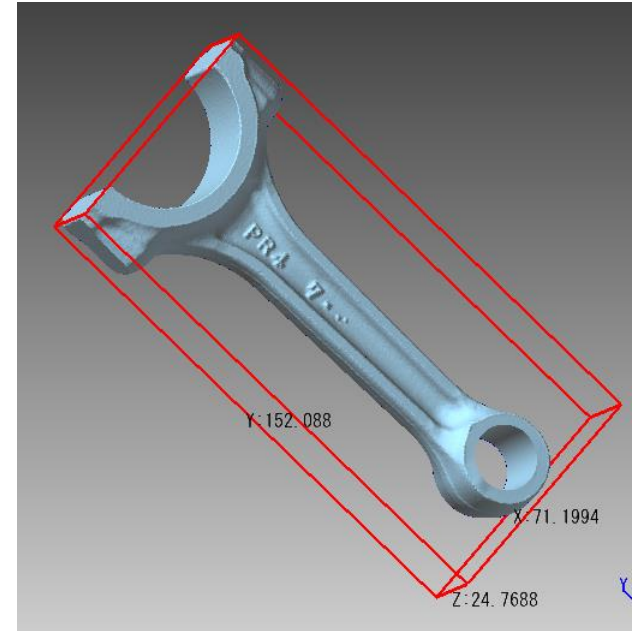
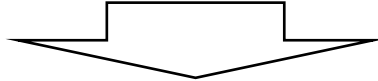
Results to be delivered depend upon mesh surface conditions. Thus, execute the command after improving mesh surfaces as listed below.

Situations of Output Results	Name of Command of Use	Descriptions
<b>Poor export deliveries caused by a wrong mesh size (Export Example 1)</b>	Scale	The command cannot deliver CAD surfaces out of a mesh that is small in light of 3D perspective. An ideal Box that contains the mesh is large enough to contain a few hundreds of diagonals or more. Enlarge the meshes before executing the command and deliver CAD surfaces out of the meshes. Then, use CAD and diminish the size of delivered CAD surfaces into what they should be in accordance with the original meshes.
<b>Poor export deliveries due to the mesh roughness (Export Example 2)</b>	Remesh	If big faces constitute a mesh, the mesh cannot sufficiently reflect changes in the shape. In this case, execute the “Remesh” command in order to reconstitute the mesh into one composed of smaller faces.
<b>Calculation errors, irregular shapes, and surface dropouts (Export Examples 3, 4)</b>	CT Mesh Shape Optical Mesh Shape	The “Mesh Shape” command optimizes automatically mesh surfaces. Accordingly, export results are likely to become better. It is recommended to execute this command.
<b>Process surface edges (Export Example 5)</b>	Cut-off	This command is effective in trimming a surface mesh at the edge derived from measurement data.
<b>Reduce calculation time, and prevent surface fragmentation (Export Example 6)</b>	Simplify	Execute the “Simplify” command in order to reduce calculation time and the quantity of surfaces that become fragmented (phenomenon of plenty of tiny CAD surfaces being generated). In a case of large-size data, it is recommended to execute the command and check generation results.
<b>Prevent surfaces from becoming fragmented</b>	Smoothing CT Mesh Shape Optical Mesh Shape	The commands are effective for rough mesh surfaces. Choose ‘Feature Keeping’ or ‘Polish1’ in the command dialogue of the “Smoothing” command.
<b>Irregular surfaces that are generated where elongated faces exist (Export Example 7)</b>	Remesh	Elongated faces cause irregular surfaces. Execute the “Remesh” command in order to homogenize triangles.
<b>Too few faces (Export Example 8)</b>	Subdivide	CAD data may not be exported due to too few faces. Execute the “Subdivision” command (‘Uniform Division’) in order to divide faces.

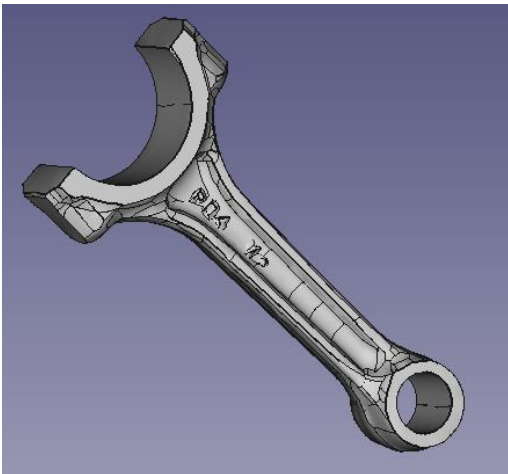
# CAD Surface Export (3)

■ Poor export caused by a wrong mesh size (Export Example 1)  
The command cannot deliver good CAD surfaces out of a small mesh in light of 3D perspective. Execute the command after enlarging small meshes by executing the "Scale" command. What is recommended is that the mesh is large enough to contain X axis, Y axis and Z axis of a few hundreds of sizes or more. An IGES file to be generated needs to be minimized back through the use of CAD.

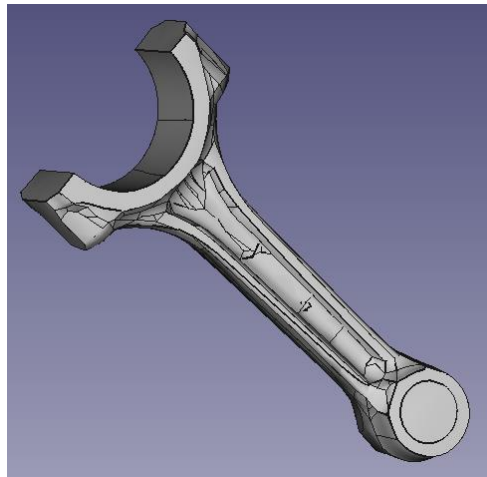
Results from Executing the "CAD Surface Export" command for the Solid Mesh (right) of Different Sizes



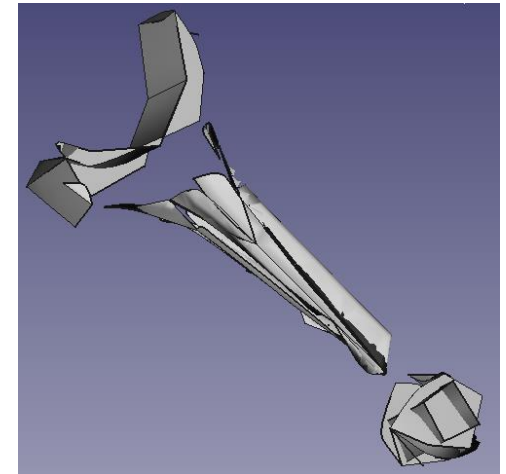
Size : 70×15×2.5  
CAD Surface Export Result



Size : 70×15×2.5  
CAD Surface Export Result



Size : 1.4×3×0.5  
CAD Surface Export Result

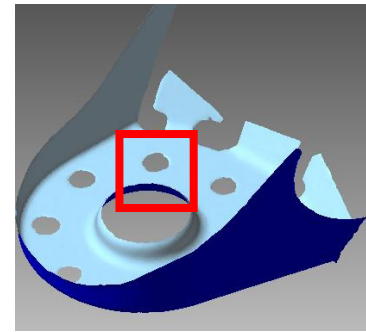


(\* ) Abend if the Solid mesh is of 0.7×1.5×0.25

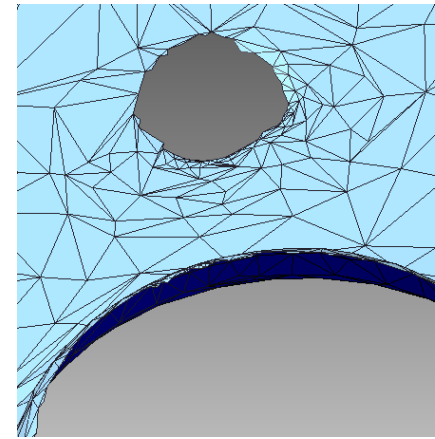
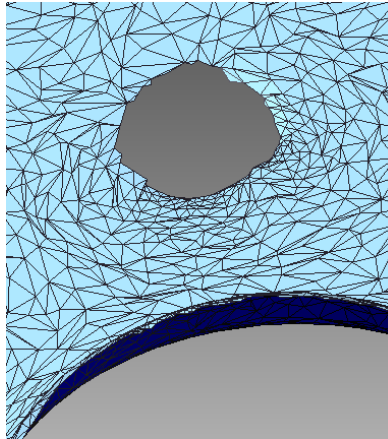
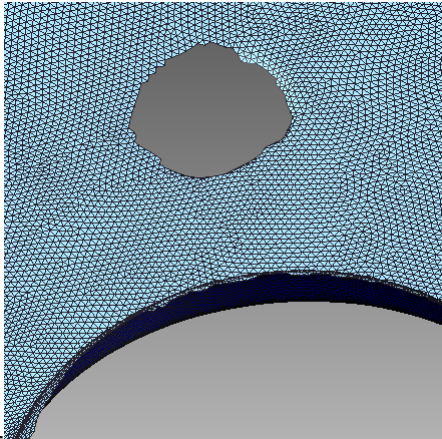
# CAD Surface Export (4)

- Poor export deliveries due to the mesh roughness (Export Example 2)

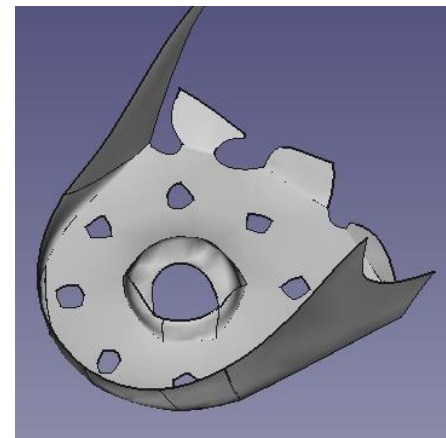
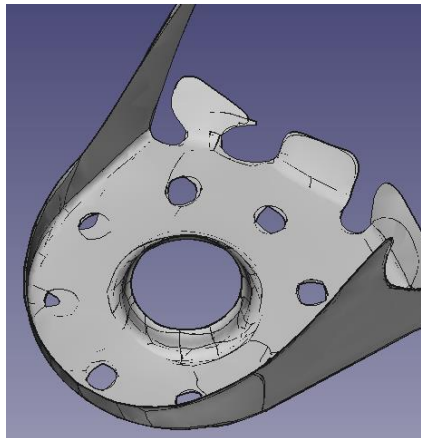
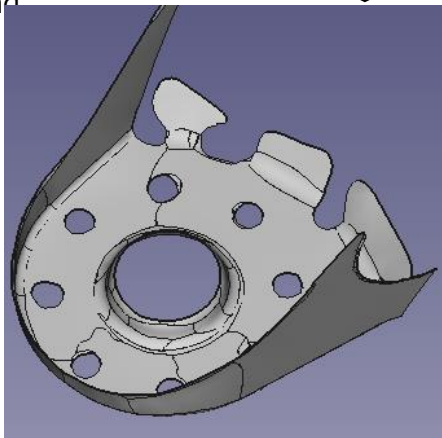
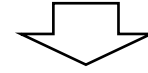
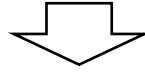
A good CAD surface cannot be generated from a rough mesh. Execute the "Remesh" command ('Triangle Face Size' = 'Adaptive') in order to optimize the granularity before executing the "CAD Surface Export" command.



Mesh Roughness



Results from executing the "CAD Surface Export" command

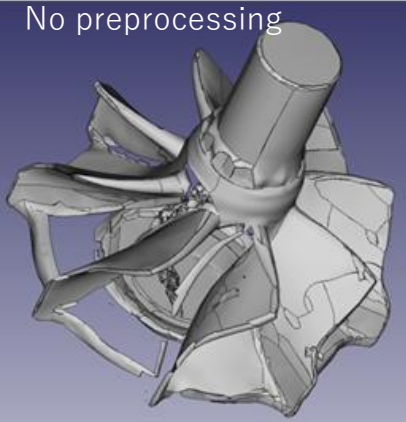




# CAD Surface Export (5)

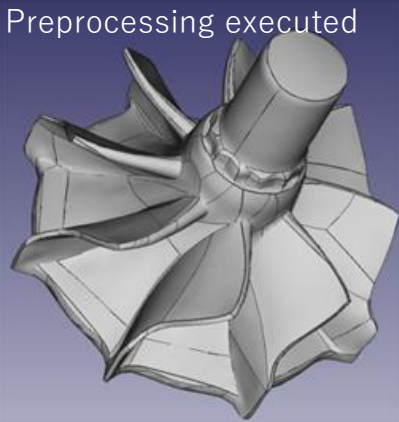
## [Example 3] Surface Dropouts

No preprocessing



Processing time 25min

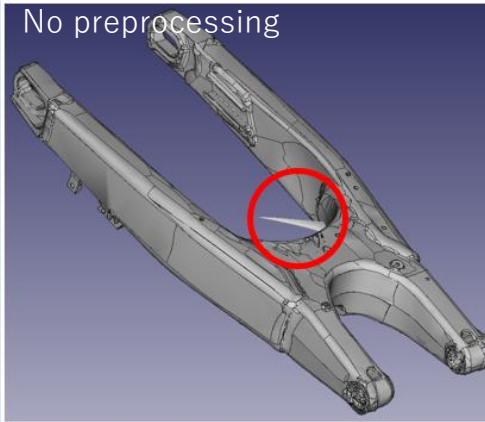
Preprocessing executed



Processing time 1min  
Preprocessing contents : CT Mesh  
Shape → Polish 1 → Simplify

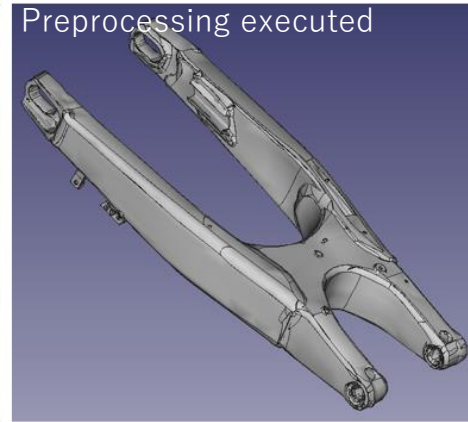
## [Example 4] Irregular Surfaces

No preprocessing



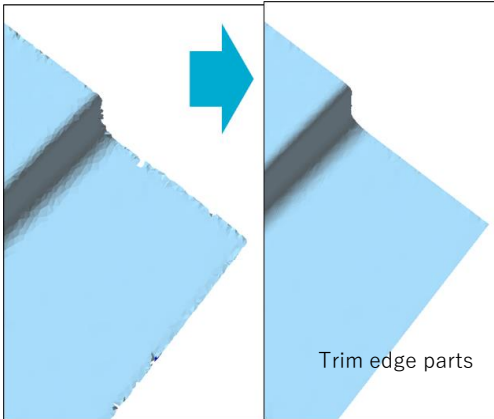
Processing time 2 hr

Preprocessing executed

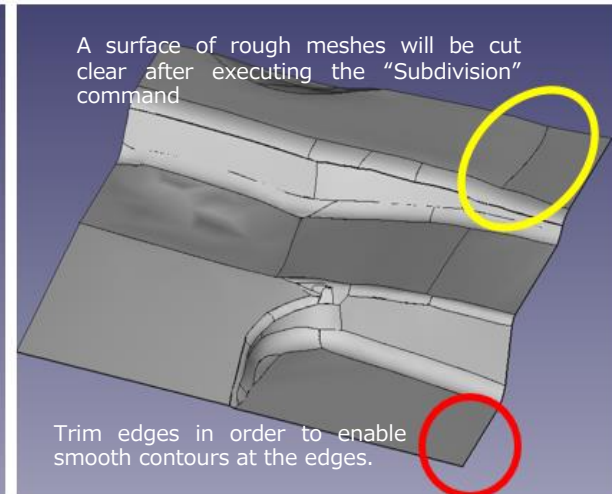
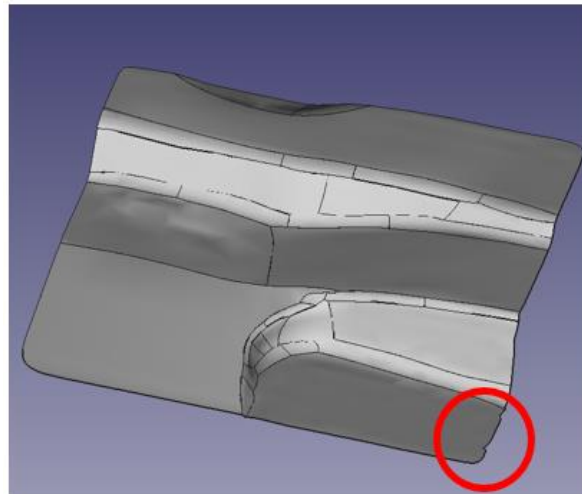


Processing time 10 min  
Preprocessing contents:  
CT Mesh Shape → Simplify

## [Example 5] Process edges Divide surfaces



Trim edge parts



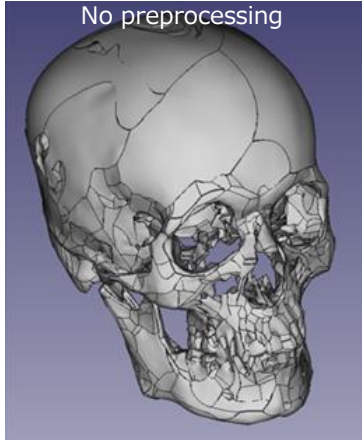
A surface of rough meshes will be cut clear after executing the "Subdivision" command

Trim edges in order to enable smooth contours at the edges.

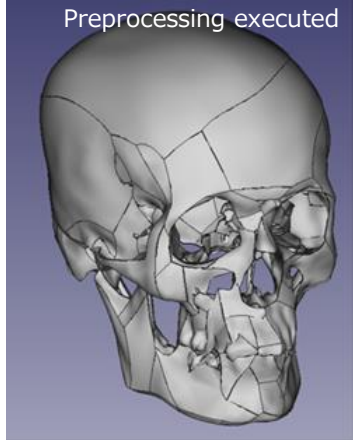
# CAD Surface Export (6)

[Example 6]

Reduce calculation time and prevent surface fragmentation



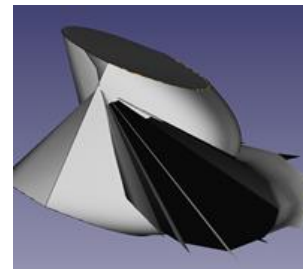
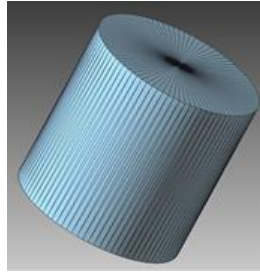
Processing time  
15 min



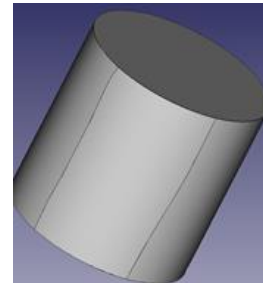
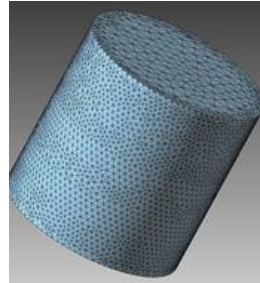
Processing time 3 min  
Preprocessing contents:  
Cleaning → Simplify

[Example 7]

Reduce calculation time and prevent surface fragmentation

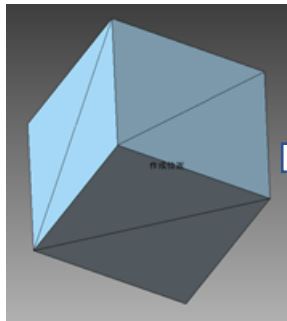


If elongated faces are included, irregular side surfaces are generated from a cylinder made of CAD data.

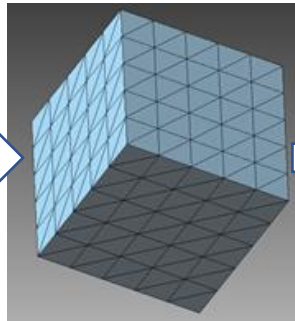


Remeshing may improve the surface.

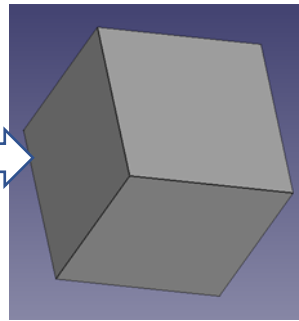
[Example 8] Too few faces



No CAD surfaces can be created



Subdivide (Uniform Subdivision)



CAD surfaces

Standards Calculation Time

Polygon Q'ty	Necessary memory	Calculation time
100,000	16GB	2 min
1,000,000	16GB	15 min
5,000,000	32GB	2 hr.

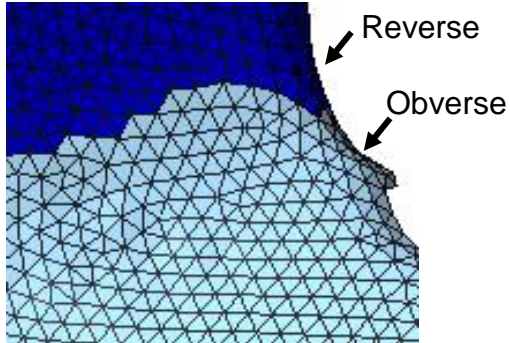
PC of Use: Intel Core i5-8250U 1.8Ghz

The numbers above may depend upon mesh situations and PC specifications. Execute the "Simplify" command first for big data before executing the "CAD Surface Export" command.

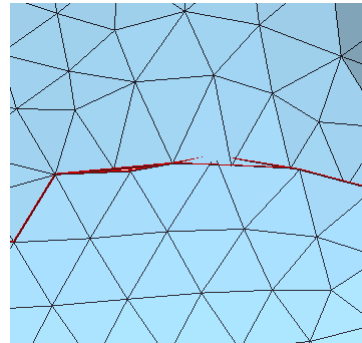
# 7. Home Tab

# Cleaning (1)

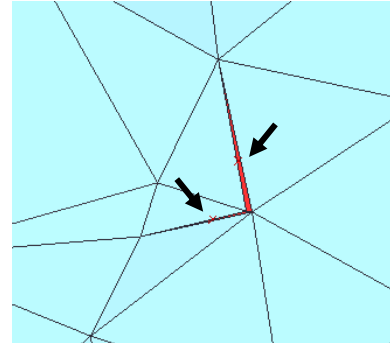
Inspect and repair error parts (illegal parts) in the mesh.



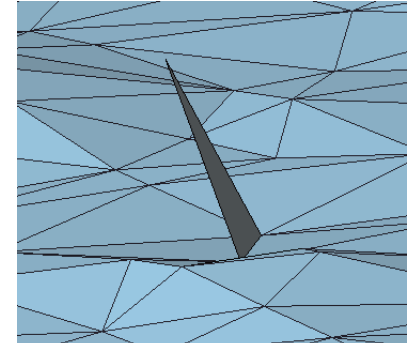
**Reversed Faces**  
(The front and back of adjacent faces do not match)



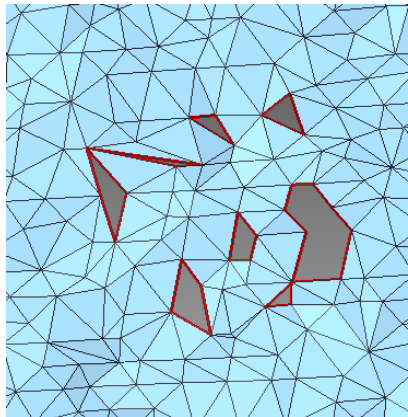
**Gap between Shells**



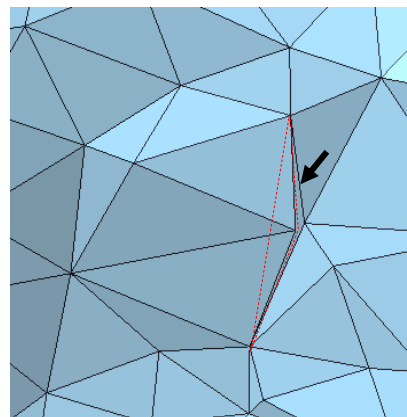
**Tiny Faces**



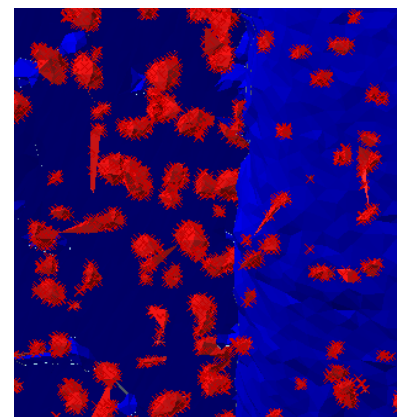
**Irregular Connection**  
(3 faces are connected to one edge, etc.)



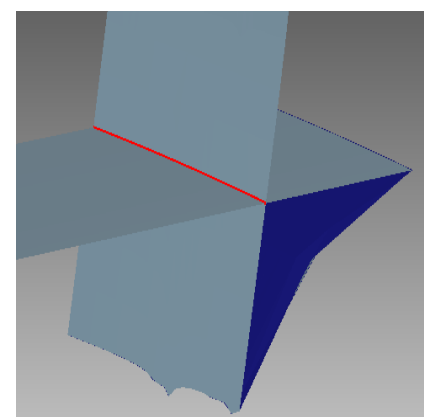
**Small Holes**



**Folded Faces**



**Small Shells**



**Self-intersection**

The target to be processed is the hole in the shell, that is, the missing face.  
For through holes and blind holes, use “Fill Caves” or “Defeature” command.

# Cleaning (2)

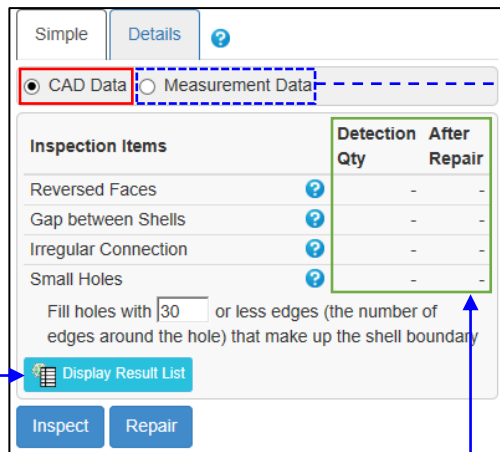
The "Cleaning" command consists of a "Simple" tab (below) and a "Details" tab (explained on the next page).

## "Simple" tab

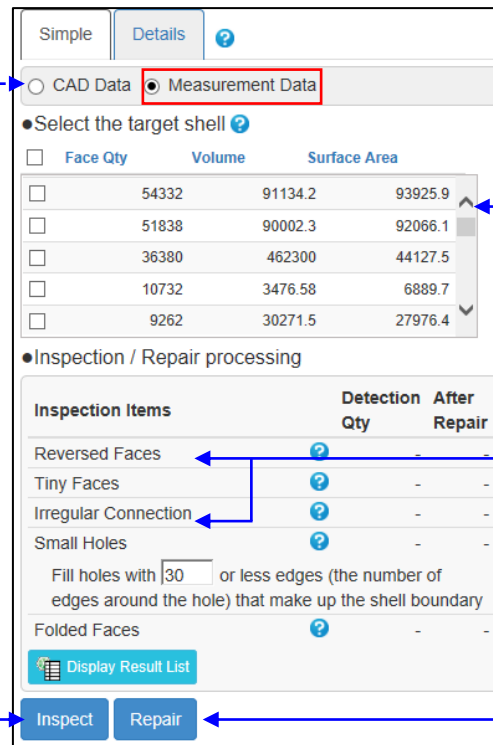
Recommended inspection items are preset according to the origin of the data.

The cleaning function of the "Simple" tab is suitable for the mesh before the editing operation. (Use the function of the "Details" tab for cleaning after editing.)

For mesh derived from CAD data



For mesh derived from measurement data



Click to save the temporary figure of the detected part and display the result list window.

This is useful when you want to use other commands, such as "Defeature", to fix error parts that cannot be repaired with the cleaning command.

In the result list window, you can also zoom in on the position of each error.

The number of errors detected is displayed.

Click to inspect. The detected error parts is displayed in red, and the number of errors is displayed.

Specifies the shell to be processed.

You can also click on the mesh to select a shell.

"Inspect": only the checked shells are inspected.

"Repair": the unchecked shells are deleted.

If there is an error in "Reversed Faces" or "Irregular Connection", POLYGONALmeister may not work properly, so correct these two errors.

Click to repair the error and inspect the mesh again after repair.

# Cleaning (3)

## “Details” tab

The details tab allows you to specify inspection items and detailed parameters individually. Depending on the purpose of using the mesh, it may not be necessary to modify the inspection items. However, if there are errors in “Reversed Faces” or “Irregular Connection”, POLYGONmeister may not work properly, so repair the errors in these two items. .

Check the items to be inspected / corrected.

Inspection Items	Detection Qty	After Repair
<input type="checkbox"/> Reversed Faces	?	-
<input checked="" type="checkbox"/> Gap between Shells	?	-
<input checked="" type="checkbox"/> Tiny Faces	?	-
<input checked="" type="checkbox"/> Irregular Connection	?	-
<input checked="" type="checkbox"/> Small Holes	?	-
<input type="checkbox"/> Folded Faces	?	-
<input type="checkbox"/> Small Shells	?	-
<input type="checkbox"/> Overlapped Shells	?	-
<input type="checkbox"/> Self-Intersection	?	-

Click to open a field for entering detailed parameters, as shown below.

Inspection Items	Detection Qty	After Repair
<input checked="" type="checkbox"/> Reversed Faces	?	-
<input checked="" type="checkbox"/> Gap between Shells Width of Gap between Shell Boundaries: 0.05	?	-
<input checked="" type="checkbox"/> Tiny Faces Tiny Face's Height: 0.01	?	-
<input checked="" type="checkbox"/> Irregular Connection	?	-
<input checked="" type="checkbox"/> Small Holes Delete shells which have 5 or less faces.	?	-
<input type="checkbox"/> Folded Faces	?	-
<input type="checkbox"/> Small Shells Delete shells which have 5 or less faces.	?	-
<input type="checkbox"/> Overlapped Shells Gap between Faces: 0.01	?	-
<input type="checkbox"/> Self-Intersection	?	-

Click to inspect for error parts corresponding to the checked inspection items.

Click to repair the error parts corresponding to the checked inspection item, and then inspect the shape again. If the error parts remain even after the correction, repeating the correction may reduce the error parts.

# Cleaning (4)

By registering errors in the "Result List", you can do the following.  
-You can zoom in for each error.  
-The error display can be left even if you move to another command.

## Inspection

Cleaning [How to use](#)

Simple [Details](#)

<input type="checkbox"/> Inspection Items	Detection Qty	After Repair
<input checked="" type="checkbox"/> Reversed Faces	0	-
<input checked="" type="checkbox"/> Gap between Shells	2	-
<input checked="" type="checkbox"/> Tiny Faces	5	-
<input checked="" type="checkbox"/> Irregular Connection	1	-
<input checked="" type="checkbox"/> Small Holes	5	-
● Flat ○ Smooth		
Fill holes with <input type="text" value="30"/> or less edges (the number of edges around the hole) that make up the shell boundary		
<input checked="" type="checkbox"/> Folded Faces	1	-
<input checked="" type="checkbox"/> Small Shells	0	-
● Delete shells which have <input type="text" value="5"/> or less faces.		
○ Keep <input type="text" value="1"/> shells in descending order of face quantity. Delete the rest.		
<input checked="" type="checkbox"/> Overlapped Shells	0	-
<input checked="" type="checkbox"/> Self-Intersection	2	-

Display Result List

Inspect Repair

Click the [Display Result List] button, if necessary.

Result list [How to use](#)

Click on a ticked line in order to specify a display subject item. Upon clicking, the first detection part will be zoomed in.

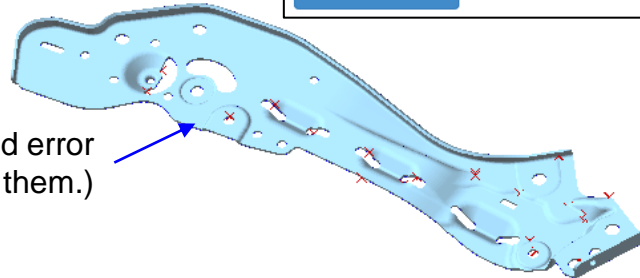
<input type="checkbox"/> ID	Test Items	Detection Qty
<input type="checkbox"/> 1	Reversed Face	0
<input checked="" type="checkbox"/> 2	Gap between Shells	2
<input checked="" type="checkbox"/> 3	Tiny Faces	5
<input checked="" type="checkbox"/> 4	Irregular Connection	1
<input checked="" type="checkbox"/> 5	Small Holes	5
<input checked="" type="checkbox"/> 6	Folded Faces	1
<input type="checkbox"/> 7	Small Shells	0
<input type="checkbox"/> 8	Overlapped Shells	0
<input checked="" type="checkbox"/> 9	Self-Intersection	2

← 2-1 →

Delete Entire List

Click the [Inspect] button.

(The system displays the detected error parts in red and the number of them.)



## Repair

Cleaning [How to use](#)

Simple [Details](#)

<input type="checkbox"/> Inspection Items	Detection Qty	After Repair
<input checked="" type="checkbox"/> Reversed Faces	0	0
<input checked="" type="checkbox"/> Gap between Shells	2	0
<input checked="" type="checkbox"/> Tiny Faces	5	0
<input checked="" type="checkbox"/> Irregular Connection	1	0
<input checked="" type="checkbox"/> Small Holes	5	0
● Flat ○ Smooth		
Fill holes with <input type="text" value="30"/> or less edges (the number of edges around the hole) that make up the shell boundary		
<input checked="" type="checkbox"/> Folded Faces	1	0
<input checked="" type="checkbox"/> Small Shells	0	0
● Delete shells which have <input type="text" value="5"/> or less faces.		
○ Keep <input type="text" value="1"/> shells in descending order of face quantity. Delete the rest.		
<input checked="" type="checkbox"/> Overlapped Shells	0	0
<input checked="" type="checkbox"/> Self-Intersection	2	0

Display Result List

Inspect Repair

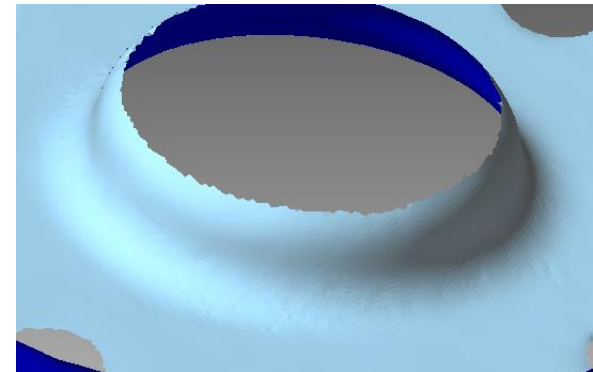
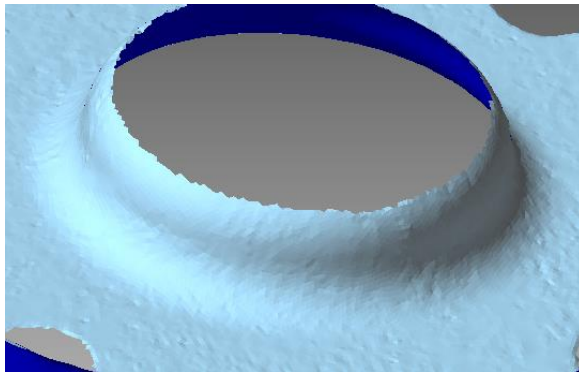
Click the [Repair] button

(The system repairs the errors and performs the inspection again after the repair.)

## Smoothing (1)

Smooth by removing the fine irregularities on the surface of the mesh derived from the measurement. Four types of smoothing methods are available.

- Feature Preserving  
“Feature Preserving” does not smooth the crease parts because they represent the characteristics of the shape, but smooths the other parts so that the change of the normal of the face is small. It is possible to smooth the vertices so that they do not move beyond the specified value.
- Polish 1  
“Polish 1” moves a vertex onto a surface that is approximated (almost passed) using the vertices around it. By doing this for all vertices, the mesh will be smooth.
- Polish 2  
“Polish 2” smooths the mesh by approximating a relatively flat area to a smooth surface and moving each vertex so that it is located on that surface.
- Remove Spikes  
“Remove Spikes” removes vertices that are only one point off the perimeter due to measurement noise.





# Smoothing (2)

It is possible to smooth the entire mesh or a partial area.

If "Weak" is selected, the number of points that are considered to be a broken feature will increase, and if "Strong" is selected, it will decrease.

If checked, you can smooth the mesh so that the vertices do not move beyond the specified value.

If checked, you can see how it becomes smooth.

Increasing the "Smoothing Strength" will reduce the waviness of the shape, and decreasing it will reduce the separation from the original mesh.

The screenshot shows the 'Smoothing' dialog box with the following settings:
 

- Smoothing Modes:**  Feature Preserving,  Polish1,  Polish2,  Remove Spikes
- Smoothing Subject:**  Entirety,  Part
- Smoothing Strength:** Weak    Strong
- Specify Maximum Gap, Maximum Gap: 0.05
- In Progress Display

The screenshot shows the 'Smoothing' dialog box with the following settings:
 

- Smoothing Modes:**  Feature Preserving,  Polish1,  Polish2,  Remove Spikes
- Specify target areas for smoothing:** Free-form
- Smoothing Strength:** Weakest    Strongest
- Remesh after Smoothing
- Maintain shape near region boundaries

If checked, the area near the boundary between the area to be smoothed and the surrounding area will change smoothly.

The screenshot shows the 'Smoothing' dialog box with the following settings:
 

- Smoothing Modes:**  Feature Preserving,  Polish1,  Polish2,  Remove Spikes
- Smoothing Subject:**  Entirety,  Part
- Smoothing Strength:** Weakest    Strongest
- Remesh after Smoothing

It is possible to smooth the entire mesh or a partial area.

Increasing the "Smoothing Strength" will reduce the waviness of the shape, and decreasing it will reduce the separation from the original mesh.

If checked, the shape of the face in the smoothed area will be reconstructed into a triangle that is close to an equilateral triangle.

The screenshot shows the 'Smoothing' dialog box with the following settings:
 

- Smoothing Modes:**  Feature Preserving,  Polish1,  Polish2,  Remove Spikes
- Spike Height:** 1.0
- Spike Vertices Qty:-**

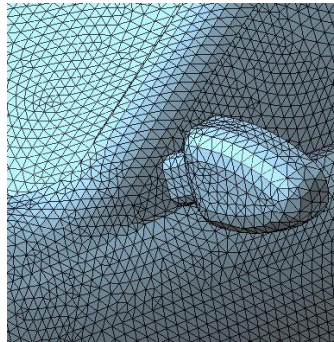
Vertices that are more than this distance in the normal direction from the surrounding vertices are removed.

Detect spikes, display red marks, and display the number of detected vertices.

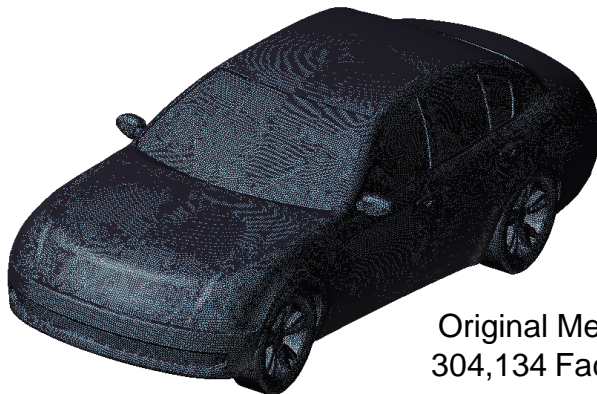
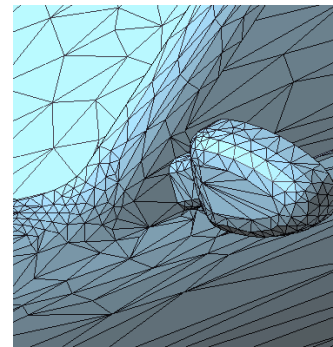
# Simplify (1)

Reduce the number of faces that compose the mesh.

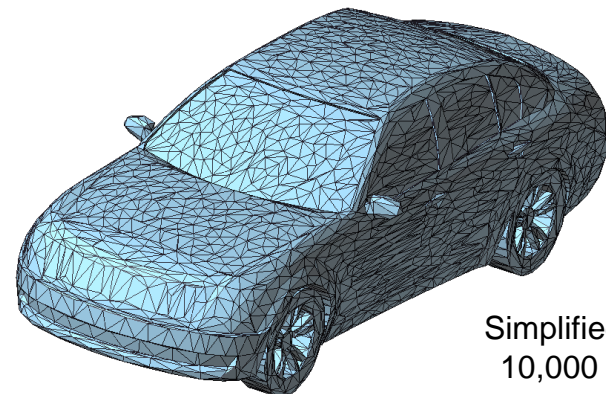
- Simplification to specify precision (“Tolerance”)  
Reduce the number of faces as much as possible, provided that the distance from the original mesh to the simplified mesh does not exceed the specified tolerance.
- Simplification to specify the number of faces after processing (“Face Qty”)  
Specify the number of faces after simplification or the percentage of faces that remain after simplification, and reduce the number of faces without changing the shape as much as possible.



“Tolerance”  
→



“Face Qty”  
→



Original Mesh  
304,134 Faces

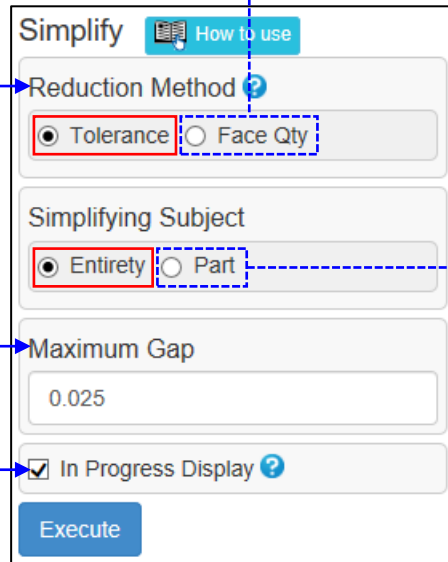
Simplified Mesh  
10,000 Faces

# Simplify (2)

In the simplification to specify the accuracy, not only the entire mesh can be simplified, but also the indication area can be simplified, and the accuracy can be changed in the indication area and other area.

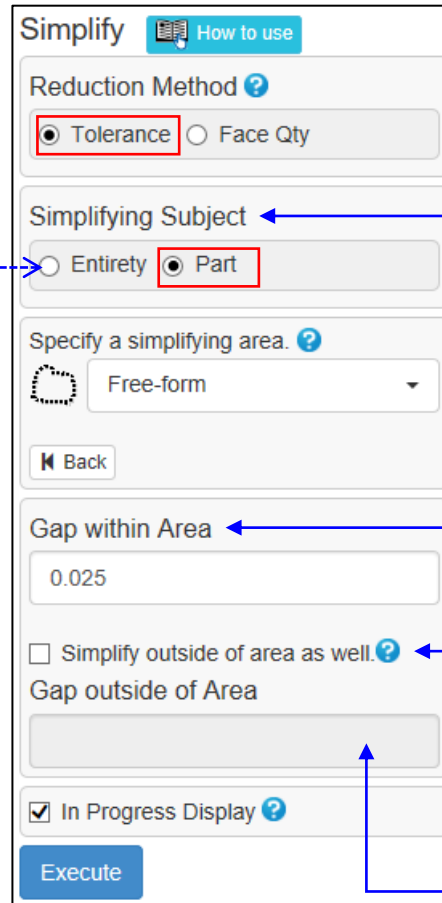
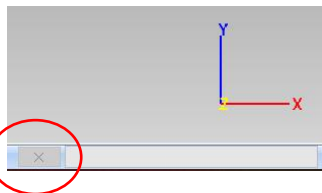
See next page

Select whether to specify the precision for simplification or specify the number of faces for simplification. (See the explanation on the previous page)



Allowable distance for deformation due to simplification.

If you check it, you can see how the faces are decreasing. If you click the stop button "x" at the bottom right of the window, the mesh displayed at the time of clicking will be the processing result of the command.



“Whole”: Simplify the entire mesh with the same accuracy.  
 “Part”: Simplify only a part of the mesh with the specified accuracy, or change the accuracy of one part of the mesh and other parts to simplify.

Allowable distance for deformation due to simplification within the specified area.

If checked, the outside of the specified area can be simplified with a different accuracy than the inside of the area.

Allowable distance for deformation outside the specified area.

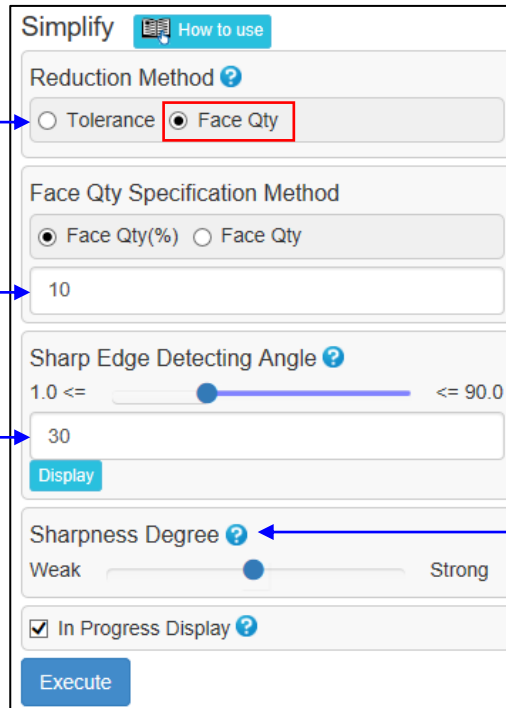
# Simplify (3)

In the simplification to specify the number of faces, simplify so that the number of faces is the specified value. The number of faces can also be specified as a percentage (the ratio of the number of faces after simplification to the number of faces in the mesh before processing).

Select whether to specify the degree of simplification by the ratio to the original number of faces or the number of faces after simplification.

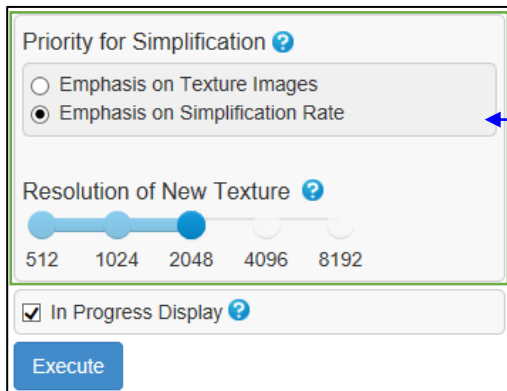
Specify the percentage (%) or the number of faces.

Simplify so as not to break the part that is bent more than the specified angle. Click the "Display" button to display the parts that meet the angle conditions in red.



In order to preserve the shape features, it is not very simplified around the broken edge, but strongly simplified as it goes away.

- Weak: This emphasis processing is not performed.
- Mid: Performs emphasis processing.
- Strong: Performs strong emphasis processing.



For textured meshes, "Priority for Simplification" are added as shown on the left. (See next page)

Selecting "Emphasis on Texture Images" simplifies the texture boundaries so that they do not move (within a continuous area of texture).

Selecting "Emphasis on Simplification Rate" simplifies the mesh without texture and the recreate the texture with reference to the original color.

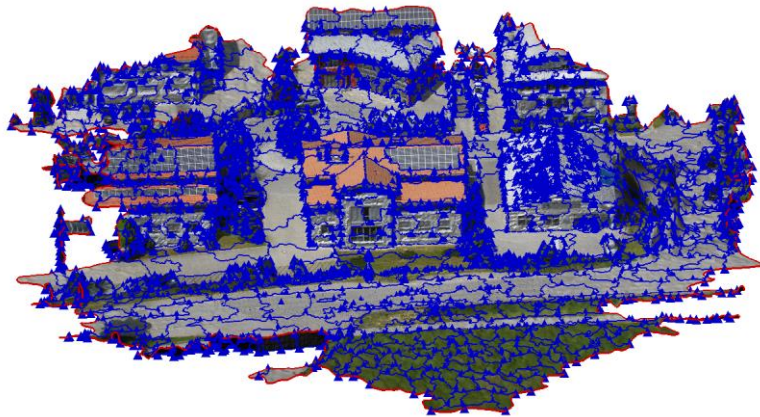
"Resolution of New Texture" is the size of the image file to be created (the number of pixels in the vertical and horizontal directions).

## Simplify (4)

The simplification process removes the color information of the mesh with the texture and processes it, and after processing, the texture can be recreated by referring to the color of the original mesh.

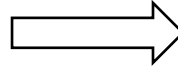


About 1 million faces



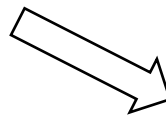
The blue line is the boundary of the continuous texture area

Simplified for each continuous texture area



About 265 thousand faces

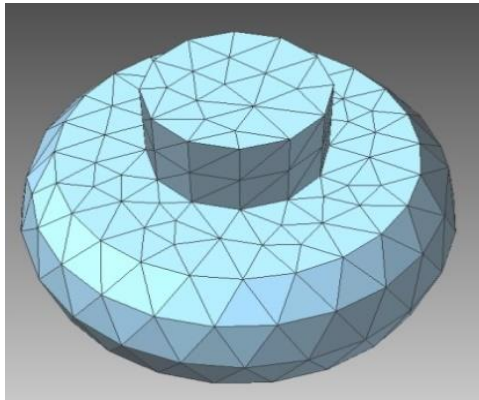
Recreate texture after simplification



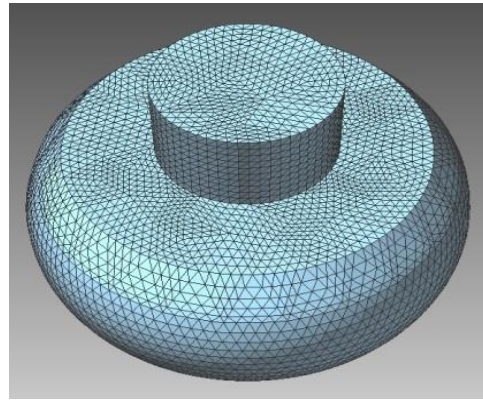
About 70 thousand faces

## Subdivision (1)

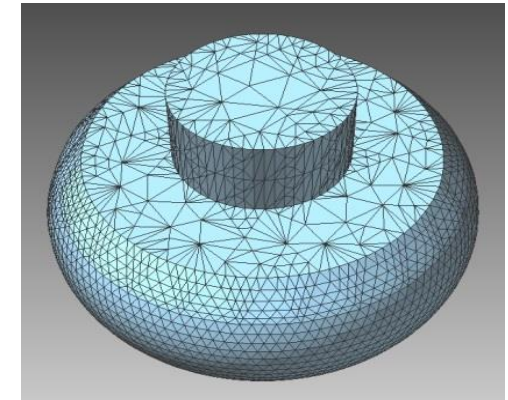
This command divides a face into smaller faces. A curved surface with a bulge is estimated from the bending of the surrounding faces, and the vertices of the divided faces are placed on the curved surface, so the mesh becomes smooth. With this command, you can smooth the mesh when the faces are rough and rugged. This command also allows to divide faces by creating vertices at the intersections of lines and edges drawn on the mesh. In this case as well, the curved surface is estimated and the vertices are created on it.



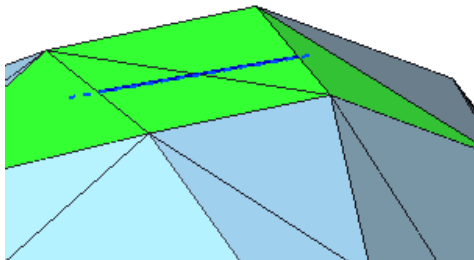
Original Mesh



Uniformly Subdivided  
(Divide each face with the same number of divisions )

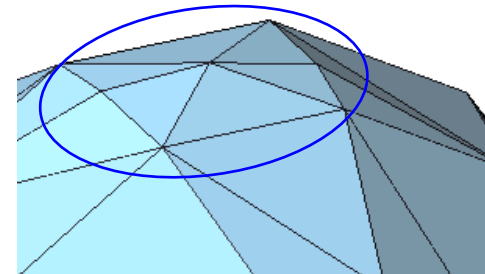
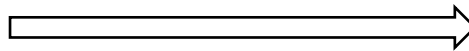


Adaptively Subdivided  
(The bent part is finely divided, and the flat part is roughly divided. )



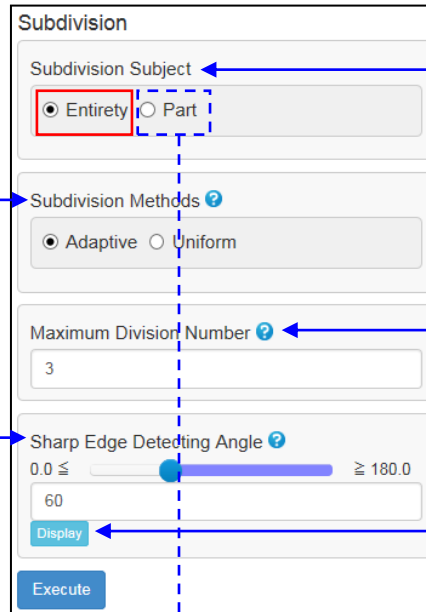
Draw a line

Divide the face at the intersection of the drawn line and the edge



Estimate the curved surface and divide the face so that the vertices to be created are on the curved surface

# Subdivision (2)

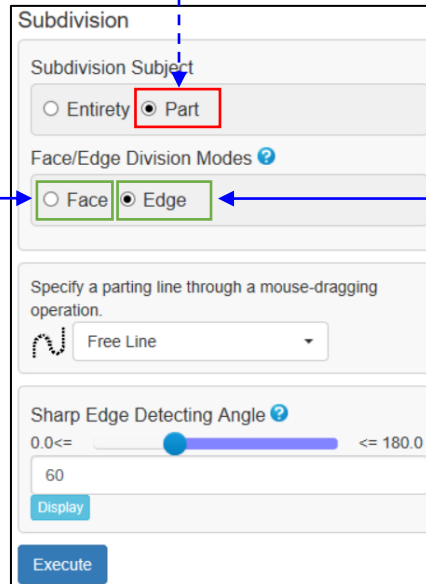
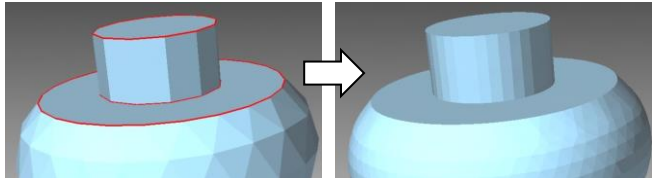


Select whether to subdivide the entire mesh or subdivide the subregion.

“Adaptive” : Fine division at curved areas and coarse division at flat areas.

“Uniform” : Divide each face by the same number of divisions.

Edges that are bent more than the specified angle between adjacent faces (red line in the figure below) are subdivided so that they are not rounded.



[In the case of “Adaptive“ division]  
The number of divisions where the face is divided most finely

[In the case of “Uniform“ division]  
The Number of divisions to divide each face

If clicked, edges that are bent more than the specified angle are displayed in red.

“Face” divides the specified faces by the same number of divisions.

“Edge” divides the edge by the drawn line.

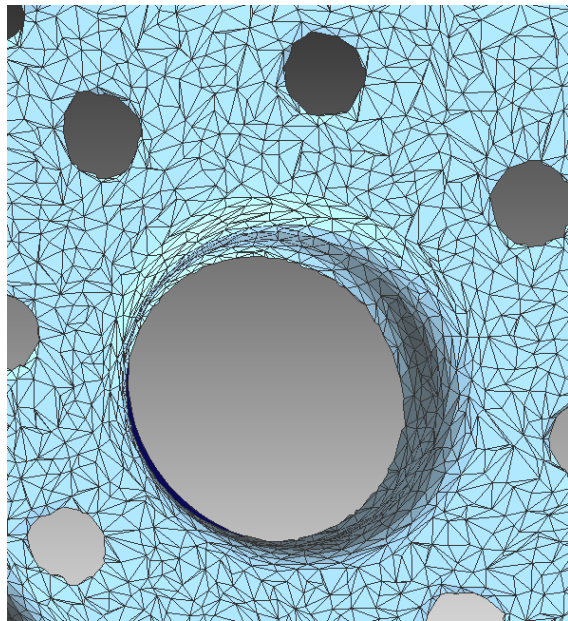
“Edge” of “Subdivision” command divides the edge so that it has a bulge, but if you do not want to have a bulge, use the “Divide Faces” command.

## Remesh (1)

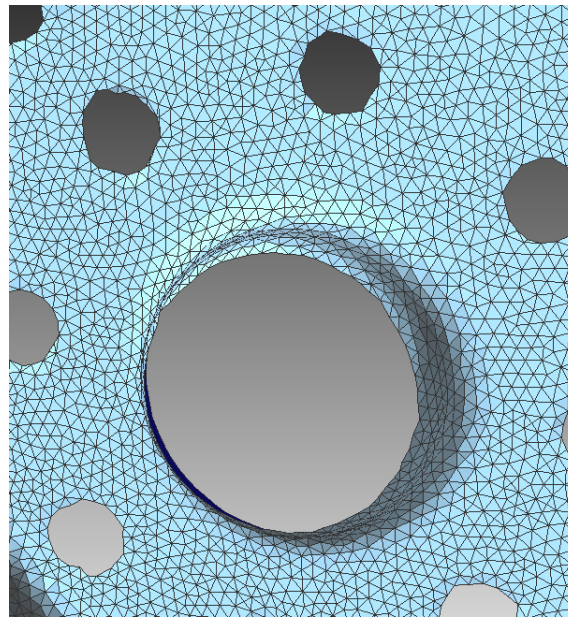
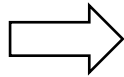
This command reconstructs the mesh so that each face of the mesh is closer to an equilateral triangle.

There are "Uniform Remesh" that creates a triangle of uniform size, and "Adaptive Remesh" that changes the size according to how it bends.

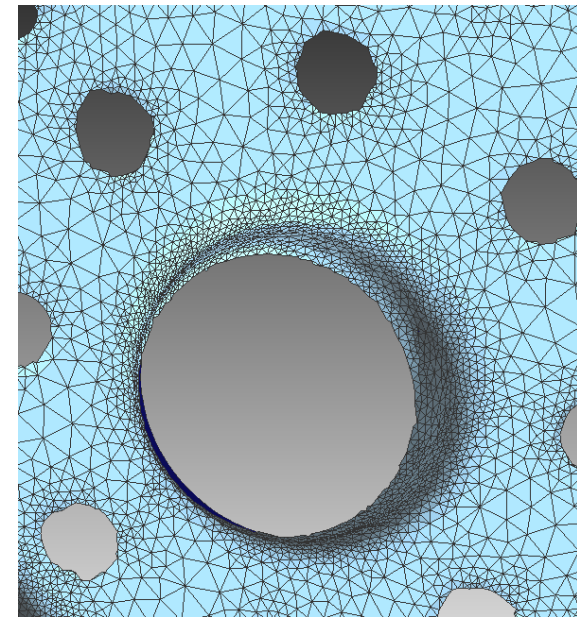
"Optimal Remesh" creates large triangles on flat areas and small triangles on curved areas so that the distance from the original mesh is small.



Original Mesh



Uniform Remesh  
(Triangles of the same size)



Adaptive Remesh  
(Triangles sized according to the bend)



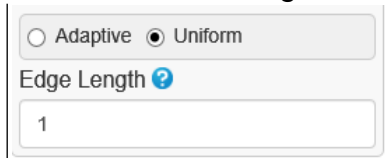
# Remesh (2)

“Adaptive” = Remesh each face to the size according to the bend.

“Uniform” = Remesh each face to the same size.

Maximum edge length = A guideline for the side length of a triangle in flat parts.

Minimum edge length = A guideline for the side length of a triangle with a strong bend. In the case of "uniform", the command dialog for entering one type of edge length changes as shown in the figure below.



It is difficult to make a triangle close to an equilateral triangle because the size of the triangle changes where the degree of bending changes. Use the slider to specify the tolerance of the triangle distortion after remeshing. The more you move it to the left, the easier it is to tolerate.

If checked, the progress of remesh will be displayed. If the stop button "x" at the bottom right of the window is clicked, the mesh displayed at the time of clicking will be the processing result of the command.

**Remesh**

Remeshing Subject  Entirety  Part

---

Triangle (Face) Size  Adaptive  Uniform

Maximum Edge Length

Minimum Edge Length

---

Angle for Detecting Sharp Edges  1.0 <=  <= 180.0

Texture boundaries and face color boundaries are always detected as feature edges.

Do not merge unadjacent sharp edge lines

ON recommended for CAD data

In Progress Display

Recreate the texture

Resolution of New Texture  512 1024 2048 4096 8192

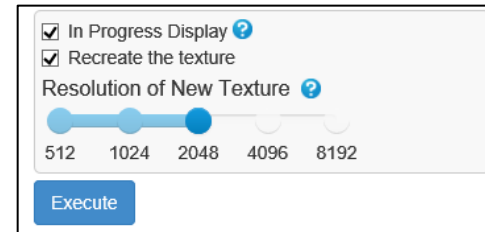
Select whether to remesh the entire mesh or a partial area.

Edges that are bent more than the specified angle between adjacent faces are remeshed with the characteristic of being sharp.

Check if it is necessary to maintain the clear sharp edges found in CAD-derived meshes after remeshing.

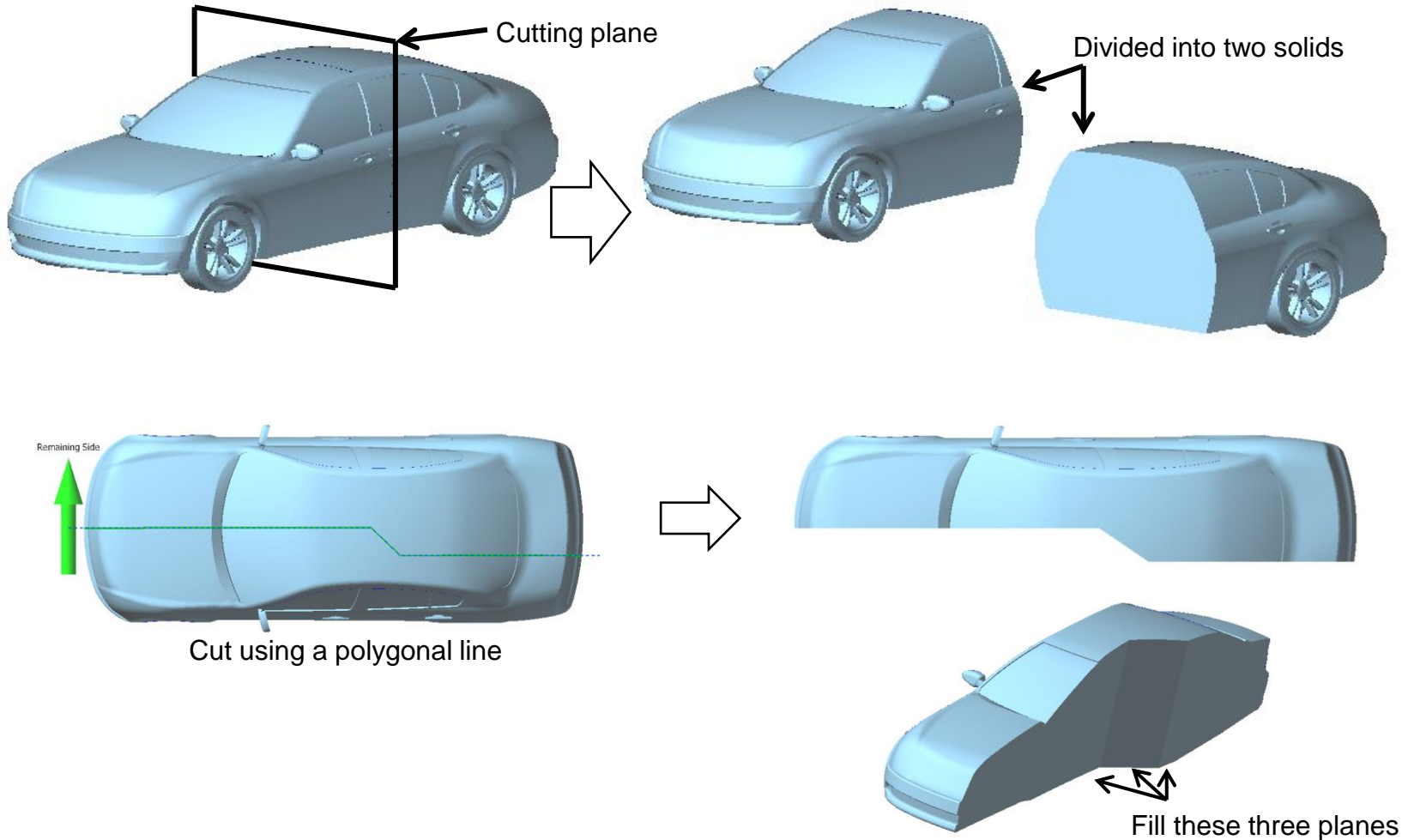
If clicked, edges that are bent more than the specified angle and boundary edges of the color are displayed in blue. Also, the shell boundaries are displayed in red. A triangle is displayed at the corner of the detected line.

For textured meshes, a checkbox "Recreate the texture" will be added as shown below. If checked, the texture will be recreated by referring to the color of the original mesh after remeshing. "Resolution of New Texture" is the size of the image file to be created (the number of pixels in the vertical and horizontal directions).



## Cut (1)

Cut the mesh using a swept surface of a straight line or a polygonal line. It is possible to remove one of the cut pieces or leave both. The cut surface can be filled with a flat surface or left unfilled.



# Cut (2)

After cutting, select whether to leave only one side or both sides.

**Cut** [How to use](#)

**Cutting Patterns ?**  
 Cutoff  Split

**Cutting Surface ?**  
 Specify a cutting surface.  
 Line

**Select Cutting Line**

**Cross Section Processing ?**  
 Filled  Unfilled  
 With Color

Clearance

**Cutting Patterns ?**  
 Cutoff  Split  
 Clearance  
 0.0

If "Split" is selected, an input field for "Clearance" will be added. Specify the width of the gap in the cut part.

When there are multiple lines of intersection, it is possible to cut using only the selected lines. Click the line to change the color. It will be cut using the green lines.

"Select All" : Set all lines green.  
 "Cancel All" : Set all lines red.  
 "Reset" : The state before inputting the section line.

Selected Unselected

Remaining Side

Remaining Side

Specify the line to cut. In the case of "Cutoff", when you specify a line, a green arrow is displayed and the side of the arrowhead remains. Click the arrow to flip the side of the arrowhead.

Select whether to fill the cut end of the cross section with a flat surface or not.

Filled Unfilled

If checked, the color of the faces that fill the cut surface will be colored with reference to the surrounding colors. If unchecked, it will not be colored.

**Cutting Surface ?**  
 Specify a cutting surface.  
 Auxiliary Line

**Coordinate Axis**  
 X  Y  Z

Pitch 100

Specify a reference position.

+ Pick vertex ?

0

[ Auxiliary Lines ]  
 Planes perpendicular to the specified coordinate axis is displayed. Click the cutting plane among them.

Reference Position

Direction perpendicular to the plane

Distance between planes to display

If checked, you can set the reference position by clicking the point on the mesh.

Coordinate value of the plane. It is possible to set the position by clicking a point on the mesh.

Open pull-down menu

Line

Select

Select

Cross

Filled  Unfilled

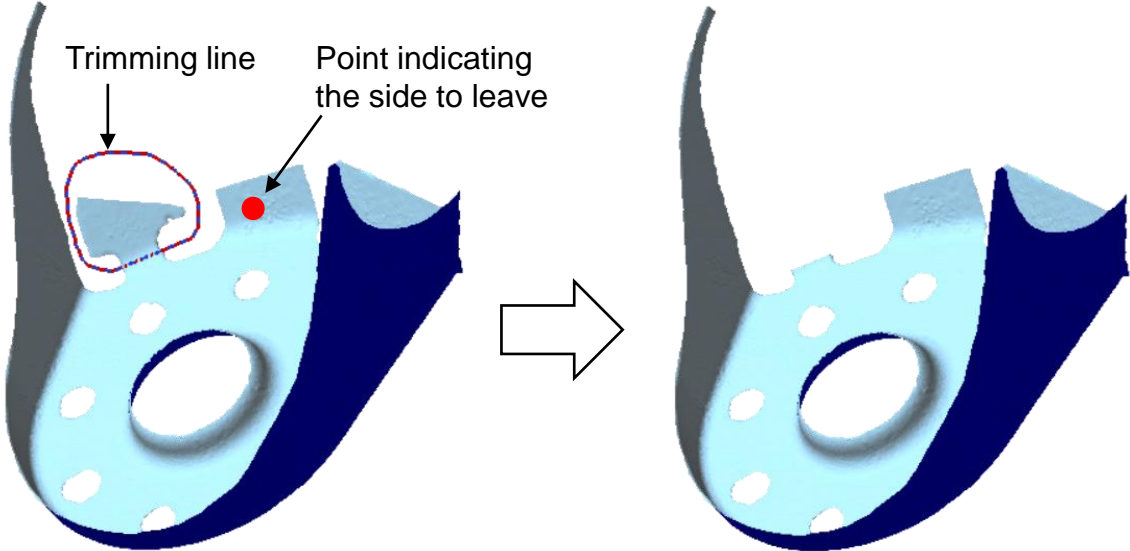
PolyLine

Auxiliary Line

[ Poly Line ]  
 A polyline is created by connecting the clicked points in order. Double-click on the last point. Click with pressing [Shift] key to draw a line horizontally or vertically on the screen.

# Trim

This command divides the mesh with the drawn line as the boundary. Leave the side containing the clicked point and delete the other. It can also leave both sides split.



- Free-form
- Rectangular
- Line
- Free Line
- PolyLine
- Color

Select the type of line to trim (left figure) and draw the line.

**Trim** [How to use](#)

Specify the trim line.

Free-form

Trim  Separation [?](#)

If "Trim" is selected, one of the two split parts will remain and the other will be deleted. Click on the part you want to keep.

If "Separation" is selected, both parts will remain.

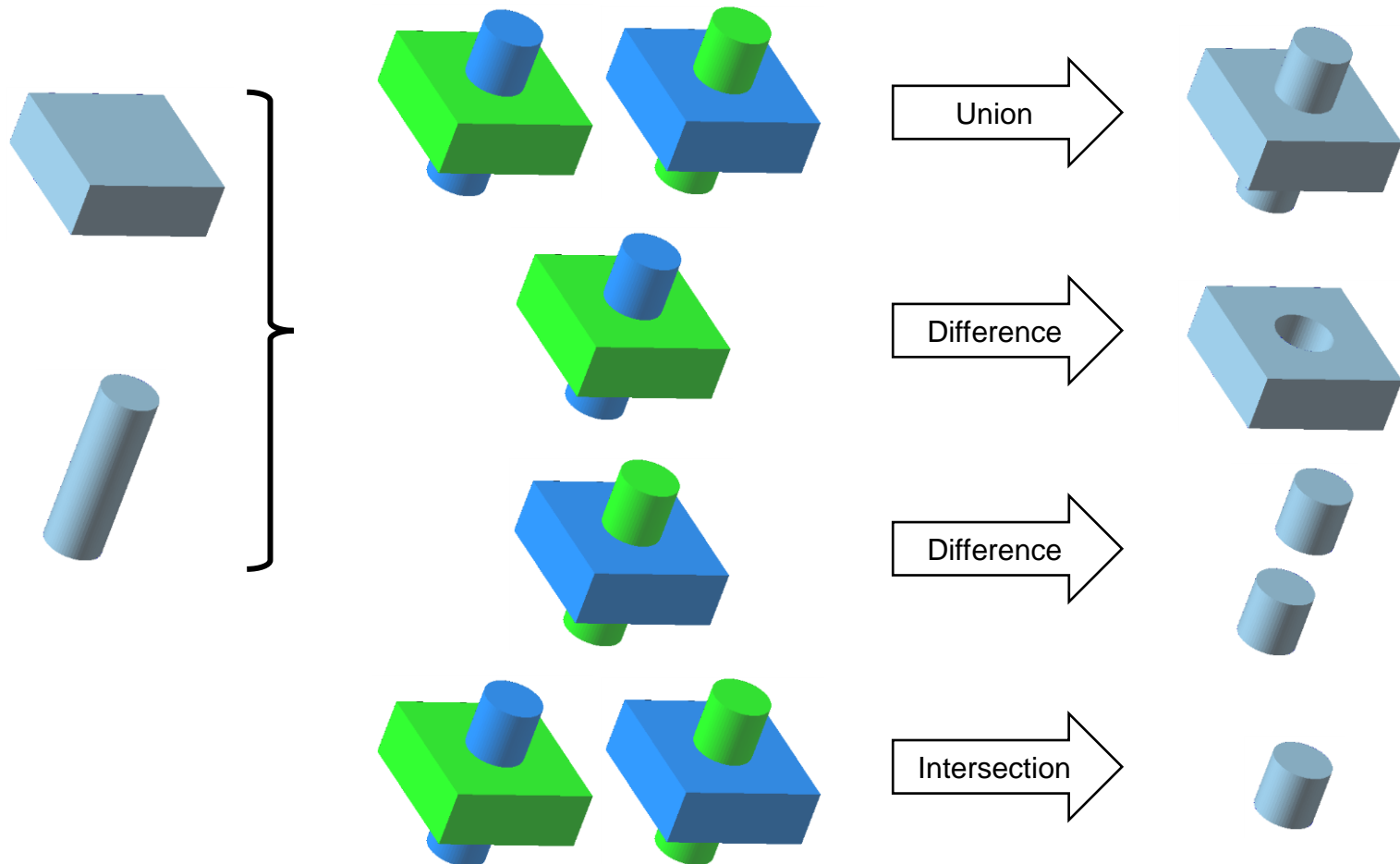
# Boolean Operation (1)

Create a union / difference set / intersection set as solids of two shells.

The result of the difference set depends on the order in which you point the shell.

You can choose the line of intersection used for processing. (Explained on the next page)

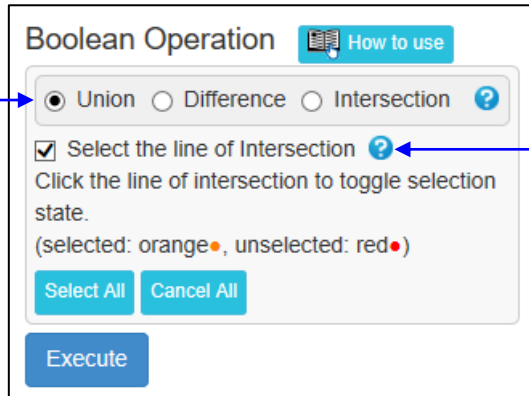
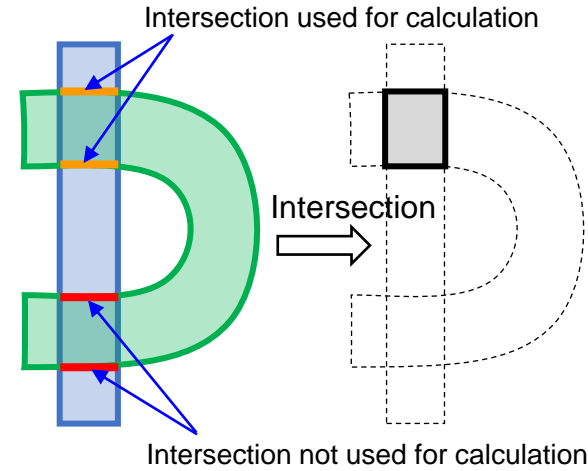
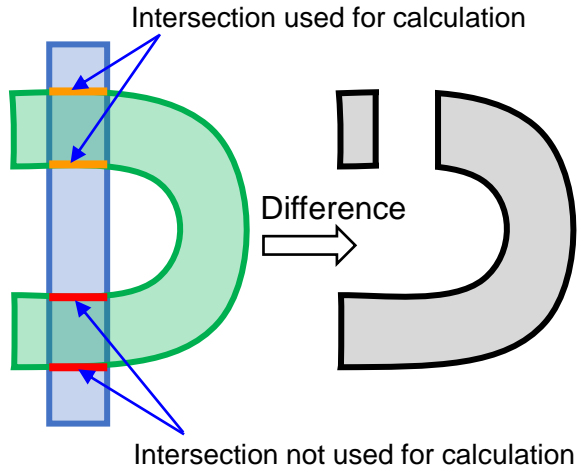
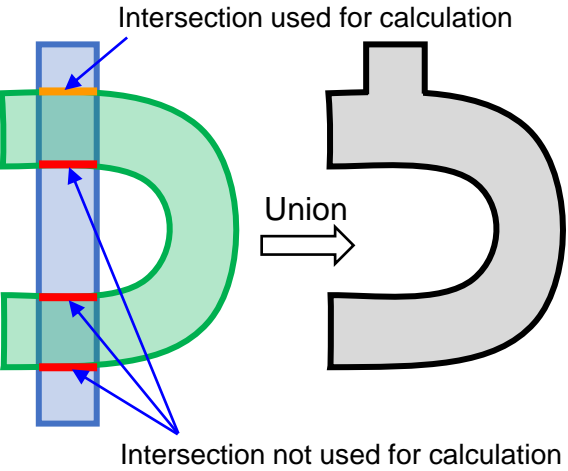
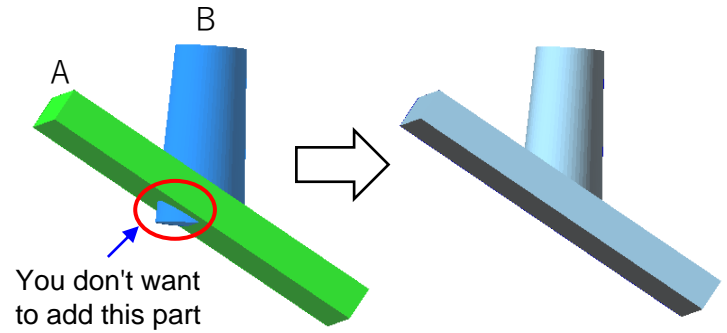
Click two shells. The shell you click the first time is painted green, and the shell you click the second time is painted blue.



# Boolean Operation (2)

## [Intersection Selection]

For example, "You want to combine B (blue shell) with A (green shell) in the right figure as a union, but you don't want to combine B that appears a little at the bottom of A." If you want to perform Boolean Operation only a part like this, use the intersection selection.

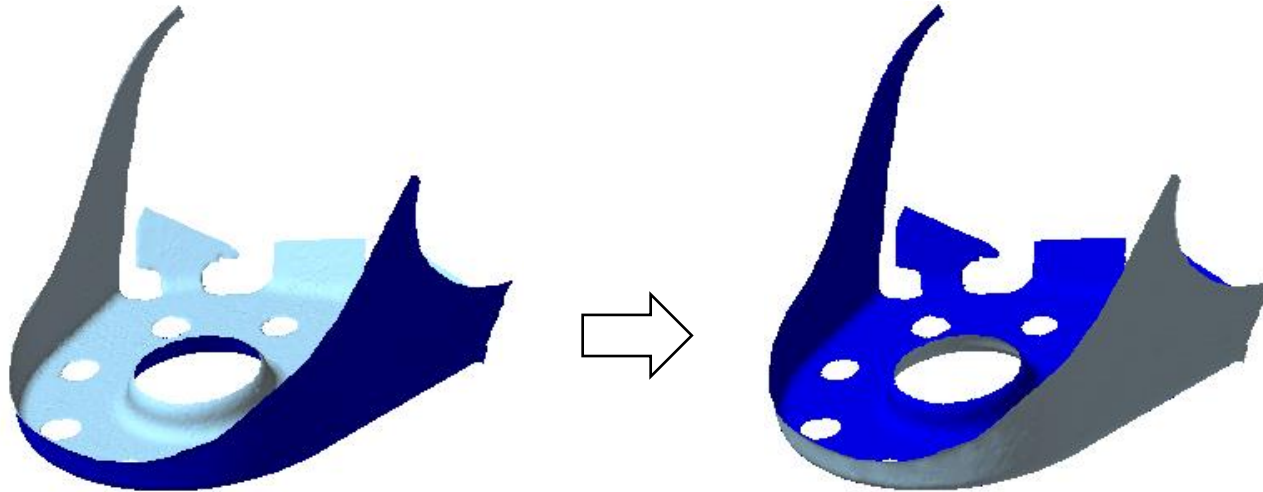


Select the type of Boolean operation and click two shells. The shell you click the first time is painted green, and the shell you click the second time is painted blue.

Click to display the line of intersection in orange. Click the line of intersection and it will turn red. Change the lines of intersection that are not used for Boolean operation red.

# Flip Shells

This command flip the front and back of the specified shells.



There are three ways to select the shell to flip the front and back .

- 1) Click or drag the shell in the figure display area.
- 2) Shells consisting of faces less than the specified number will be processed. Click the “Select shells with less than” button.
- 3) Click the row in the list.

The selected shell turns green and the check boxes in the corresponding rows of the list are checked.

### Flip Shells How to use

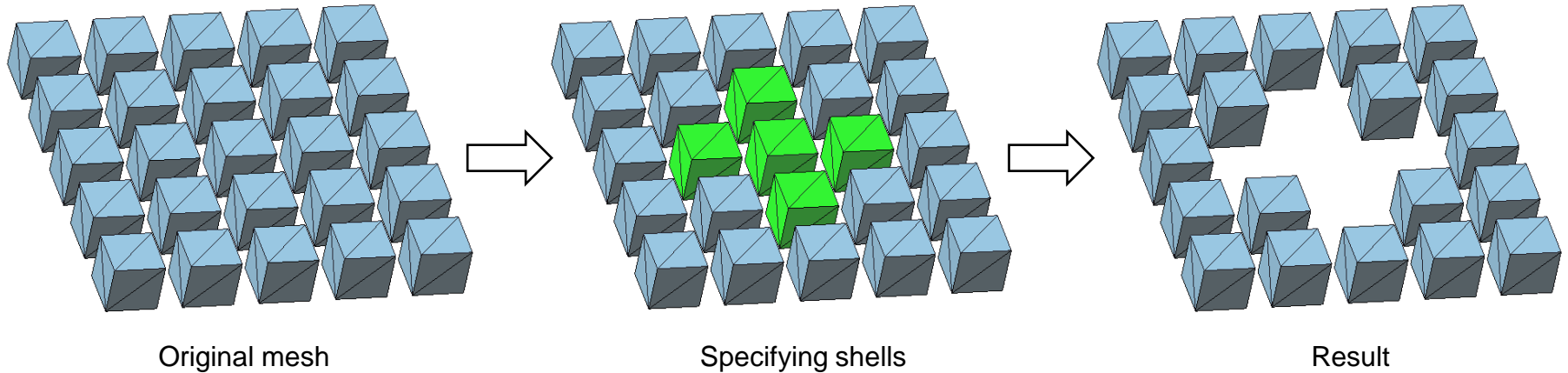
Specify shells that should be displayed reverse. ?

Select shells with less than  faces

<input type="checkbox"/>	Face Qty	Surface Area	Volume	Zoom
<input type="checkbox"/>	54332	93925.9	91134.2	<input type="button" value="🔍"/>
<input type="checkbox"/>	51838	92066.1	90002.3	<input type="button" value="🔍"/>
<input type="checkbox"/>	36380	44127.5	462300	<input type="button" value="🔍"/>
<input checked="" type="checkbox"/>	10732	6889.7	3476.58	<input type="button" value="🔍"/>
<input type="checkbox"/>	9262	27976.4	30271.5	<input type="button" value="🔍"/>
<input type="checkbox"/>	8742	6068.62	3276.25	<input type="button" value="🔍"/>

# Delete Shells

This command remove the specified shells.



There are three ways to select the shell to delete.

- 1) Click or drag the shell in the figure display area.
- 2) Shells consisting of faces less than the specified number will be processed. Click the “Select shells with less than” button.
- 3) Click the row in the list.

The selected shell turns green and the check boxes in the corresponding rows of the list are checked.

**Delete Shells** [How to use](#)

Specify shells to be deleted. ?

Select shells with less than 54332 faces

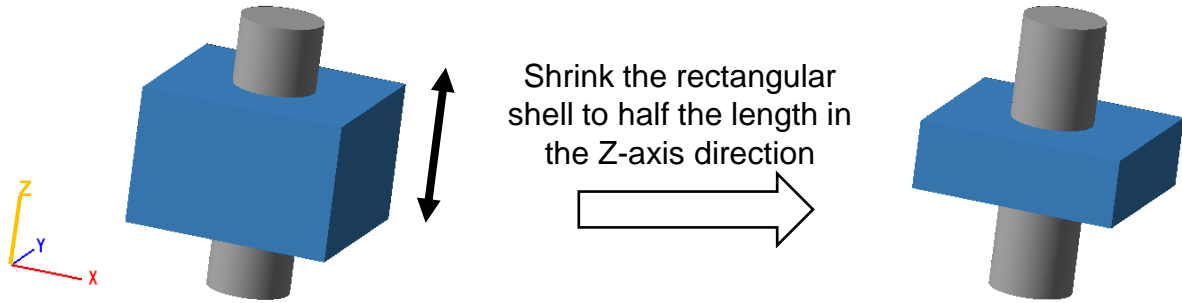
<input type="checkbox"/>	Face Qty	Surface Area	Volume	Zoom
<input type="checkbox"/>	54332	93925.9	91134.2	
<input type="checkbox"/>	51838	92066.1	90002.3	
<input type="checkbox"/>	36380	44127.5	462300	
<input checked="" type="checkbox"/>	10732	6889.7	3476.58	
<input type="checkbox"/>	9262	27976.4	30271.5	
<input type="checkbox"/>	8742	6068.62	3276.25	

Execute



# Scaling

This function scales up and down an entire mesh figure or individual shells. It is capable of changing scaling parameters for each coordinate axis.



### Scaling

**Target**

Mesh  Shell

**Scaling Mode**

Isotropic  Anisotropic

**Reference Point** ?

Reference coordinate value

Coordinate value: 0,0,0

**Scaling Parameters** ?

Scaling Factor  Size  Difference

Specify a scaling factor

1.0

**Execute**

It is a method to specify the scaling ratio. In "Scaling Factor", specify the magnification to enlarge / reduce. In "Size", specify the dimension after scaling. In "Difference", specify the difference in the size of the box containing the target figure. A positive value to increase it and a negative value to decrease it. For example, the parameter for expanding dimension 10 to 12 is:  
 Scaling Factor = 1.2  
 Size = 12  
 Difference = 2.

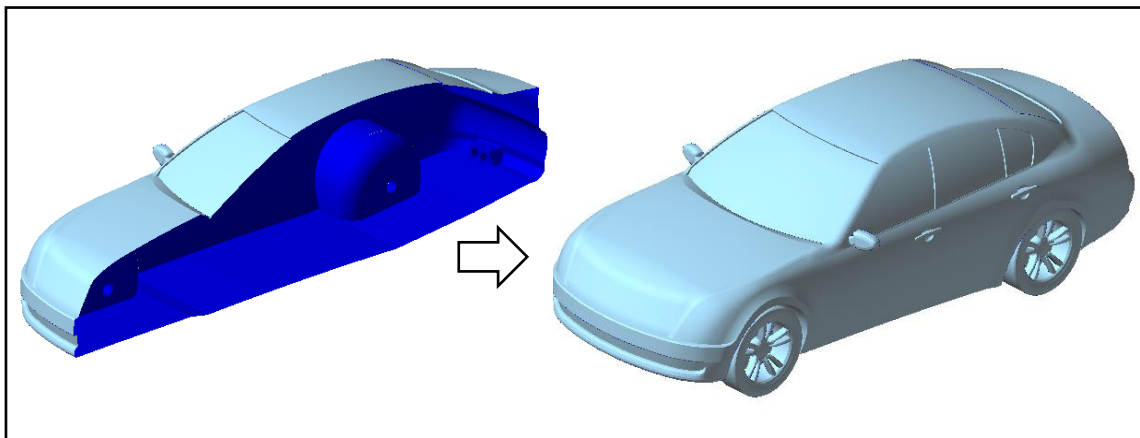
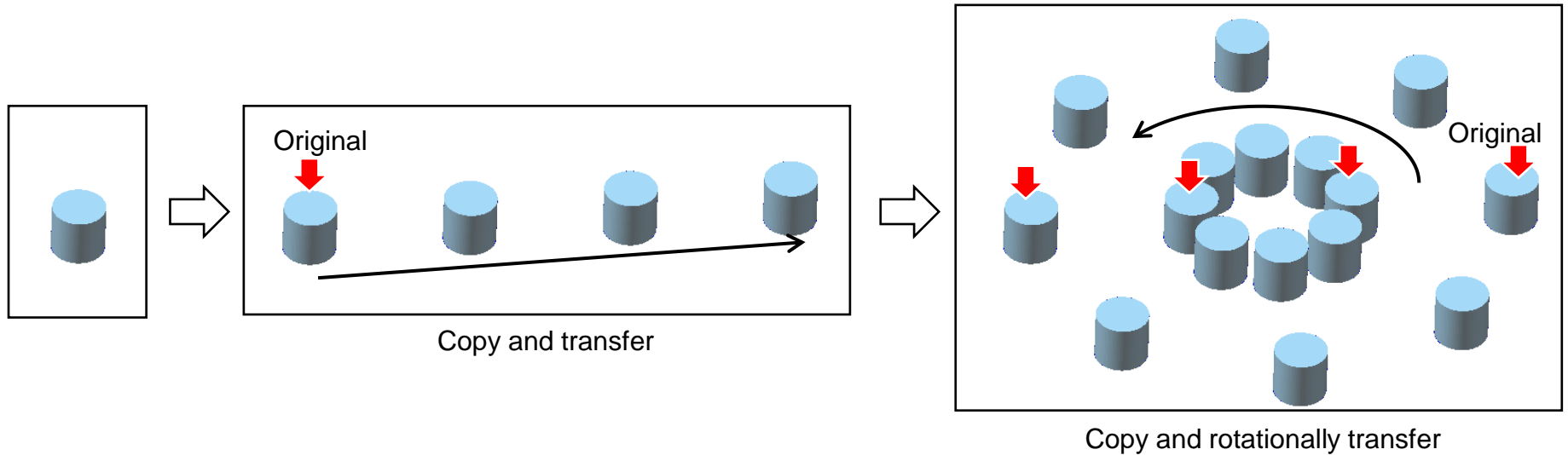
Select whether to scale the mesh or scale the shells.

Select whether to scale in each XYZ axis direction at the same ratio, to scale in different magnifications for each coordinate axis direction.

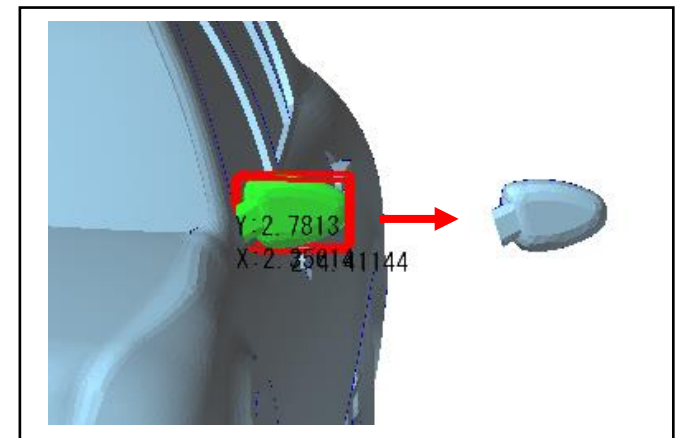
"Reference Point" is the central point of scaling. If checked, the entered coordinate value will be the center of scaling. If unchecked, the center of the box containing the target figure to be scaled will be the center of scaling.

## Move / Copy (1)

This command translates, rotates, mirrors and “Easy Layout” the mesh, shells and faces. Also, it is possible to move the copied figure without changing the original shape.



Copy with mirror movement



Face group movement

# Move / Copy (2)

Select the type of shape to move from mesh, shell, and face.

Select the type of movement from parallel movement, rotational movement, mirror movement, and "Easy Layout".

Amount of movement in the coordinate axis direction


If checked, a copy of the move target will be made, and the copy will move, leaving the move target in its original position.

For translation and rotation, copy and move will be repeated as many times as specified "Repeat Count".

Select the direction of the mirror plane.

If checked, it is possible to specify the passing position of the mirror plane.

If unchecked, the mirror plane is the plane that passes through the origin.


Move/Copy 

Target to Move  
 Mesh  Shells  Faces

Type of Movement  
 Parallel  Rotation  Mirror  Easy Layout

Amount of Movement

X	0
Y	0
Z	0

Leave (copy) the original shape 


Repeat Count 1

Execute

Type of Movement  
 Parallel  Rotation  Mirror  Easy Layout

Axis of Rotation  
 X Axis  Y Axis  Z Axis

Rotation Angle 90


Rotation Center 

Box Center  Coordinates

Coordinates 0,0,0


Type of Movement  
 Parallel  Rotation  Mirror  Easy Layout

Mirror Plane  
 YZ Plane  ZX Plane  XY Plane

Input coordinate value component 

Coordinate Value Component 0

Type of Movement  
 Parallel  Rotation  Mirror  Easy Layout

Leave (copy) the original shape 

Move Box-min to Origin

Move Box-center to Origin

Move Z-min to 0

Specify the rotation axis and rotation angle.

X-axis: direction of rotation from Y-axis to Z-axis

Y-axis: direction of rotation from Z-axis to X-axis

Z-axis: direction of rotation from X-axis to Y-axis

Specify the center of rotation.

When you click the button, the target figure will be translated as follows.  
 "Move Box-min to Origin" = Move the smallest corner of the box so that the X, Y, and Z coordinate values match the origin.

"Move Box-center to Origin" = Move the box so that the center coincides with the origin.

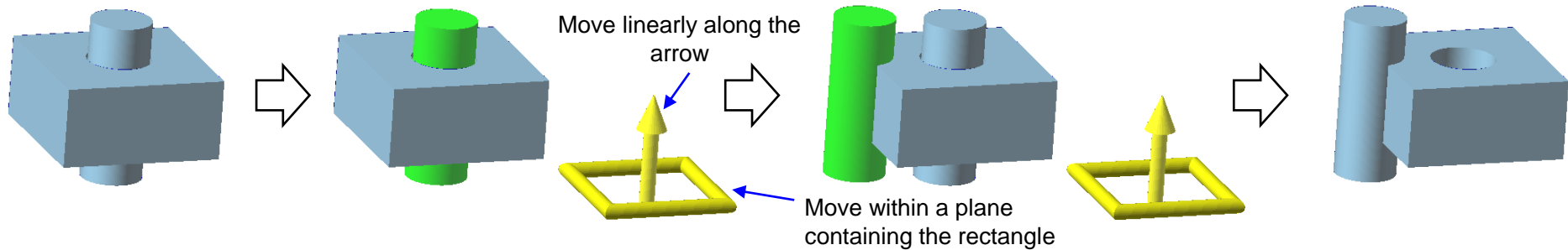
"Move Z-min to 0" = Move so that the minimum Z position of the box becomes zero.

# Drag Move

The specified shells can be translated by dragging.

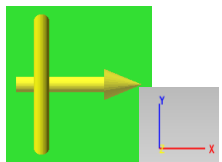
## [Operation]

1. When shells are specified, a green temporary figure for the selected shells are displayed with a yellow arrow and a rectangle. Select the movement method, and in the case of "Face" or "Edge", click the face or edge that determines the direction of the arrow.
2. Grip and drag the arrow to move the green temporary figure in the direction of the arrow, and grip and drag the rectangle to move the green temporary figure in the plane containing the rectangle.
3. Click the "Finalize" button to move the selected shells to the position of the green temporary figure.

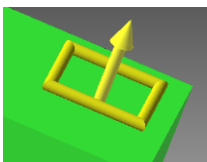


There are three ways to specify the direction of the arrow.

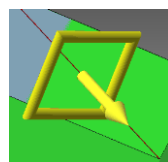
- "Coordinate Axis": Select the coordinate. An arrow parallel to the axis is displayed.
- "Face": Click on a face. An arrow perpendicular to the face is displayed.
- "Edge": Click on an edge. An arrow parallel to the edge is displayed.



Coordinate axis



Face



Edge

**Drag Move** How to use

Specify shells to move. ← Specify the shells to move by clicking or dragging.

⏪ Back

---

Direction ?

Coordinate Axis
  Face
  Edge

X
  Y
  Z

Previous Position ← Click "Previous Position" to return the drag operation to move once.

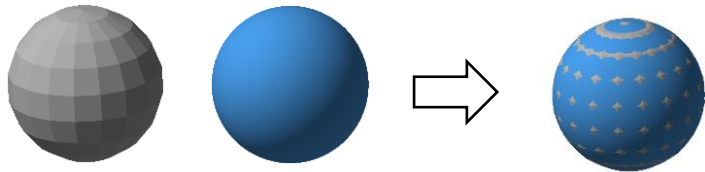
Finalize ← When the dragging operation has been finished, click "Finalize" button.

# Arrange Shells (1)

Arrange shells according to various conditions.

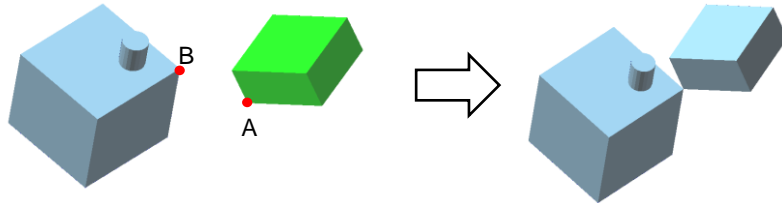
(1) Best fit

Match 2 shells with similar shapes as closely as possible.



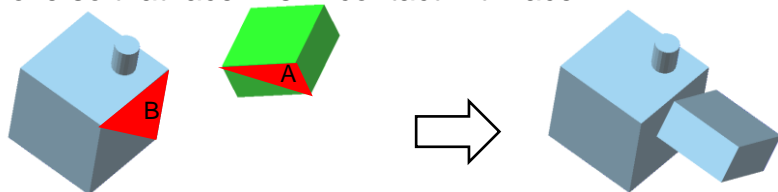
(2) Vertex -> Vertex

Translate so that vertex A matches vertex B.



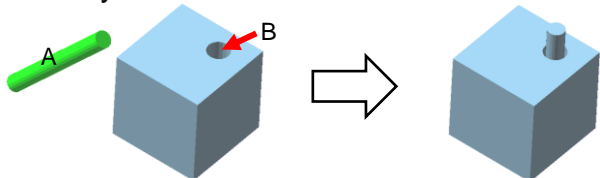
(3) Face -> Face

Move so that face A is in contact with face B.



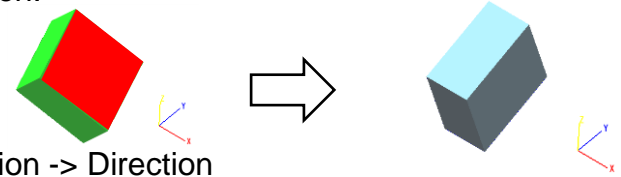
(4) Cylinder -> Cylinder

Move so that the axis of cylinder A coincides with the axis of cylinder B.



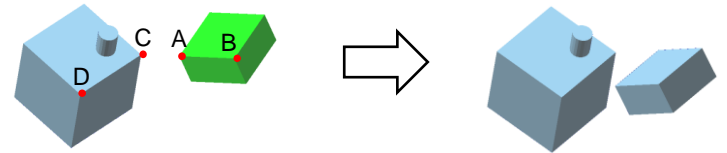
(5) Face -> Coordinate Axis

Moves so that the average normal direction on the front side of the faces matches the specified coordinate axis direction.



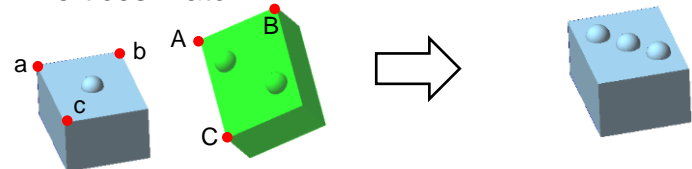
(6) Direction -> Direction

Rotate around A so that the direction from vertex A to B and the direction from vertex C to D match.



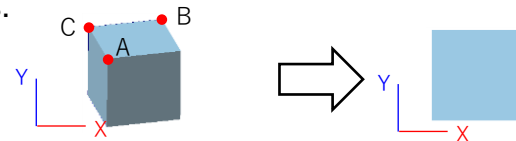
(7) 3 vertices -> 3 vertices

Move so that the directions from the first point to the second point match, and the planes passing through the 3 vertices match.



(8) 3 vertices -> Coordinate Axis

Move so that the direction from the first point to the second point is parallel to the axis, and the plane passing through the 3 vertices is perpendicular to another sitting axis.

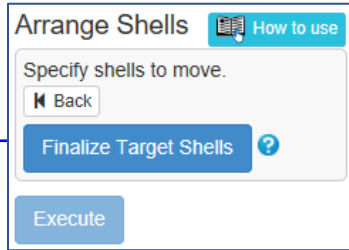


(9) Amount of movement

Translates according to the amount of movement specified for each coordinate axis

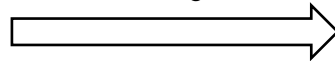
# Arrange Shells (2)

Dialog displayed when a command is selected

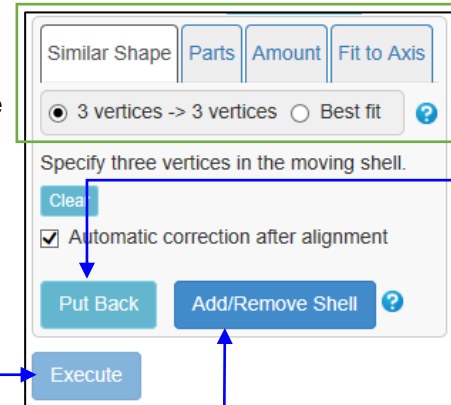


Select the shells to move and click the "Finalize Target Shells" button.

When the "Finalize Target Shells" button is clicked, the displayed contents are changed.



Click to confirm the placement.

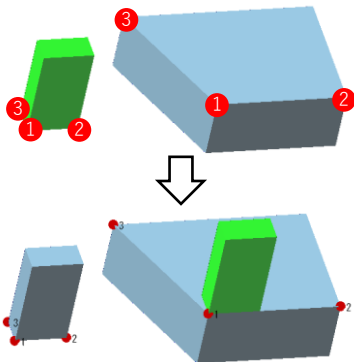


Select the placement method using the tabs and radio buttons.

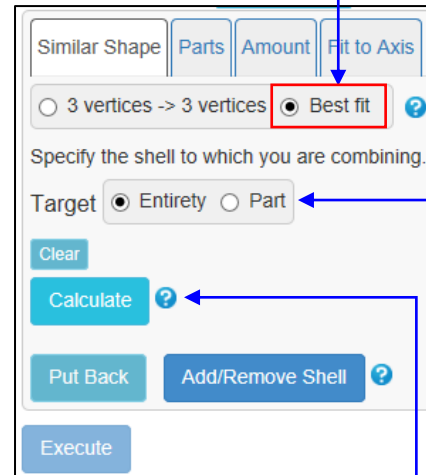
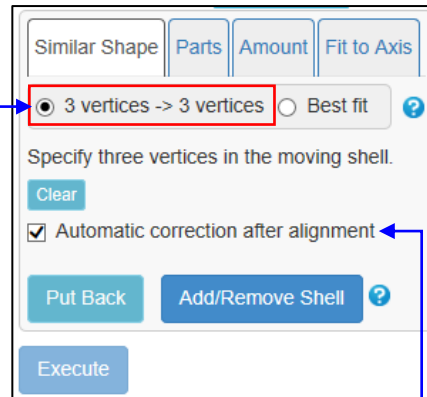
If you set the placement conditions and change the placement method during the preview display, you can specify new placement conditions from the preview state. You can return this placement operation once by clicking the "Put Back" button.

Click to return to the shell selection screen, where you can add or remove shells to place.

Specifies the 3 vertices of the moving shells and the 3 vertices of the non-moving shells. The moving shells translates so that the first vertices match. Next, rotate the shells around the first vertex so that the directions from the first vertex to the second vertex match. Finally, rotate the shells around a line through the first and second vertices so that the planes passing through each of the three vertices match.



The vertices of the two shells are often configured differently, making it difficult to click on the appropriate vertices. If you check "Automatic correction after alignment", the vertices around the clicked vertex will also be used for alignment to improve the alignment accuracy.

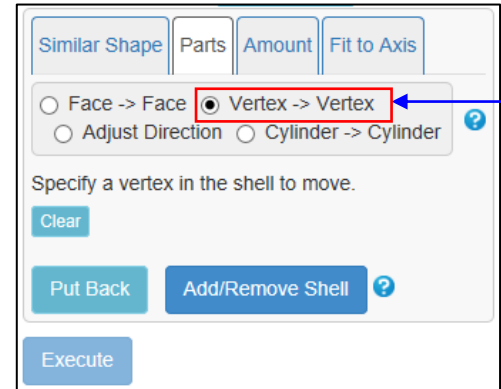


Move the vertices of the moving shell as close as possible to the destination shell. (Before doing the best fit, use other features to bring the two shells closer together.)

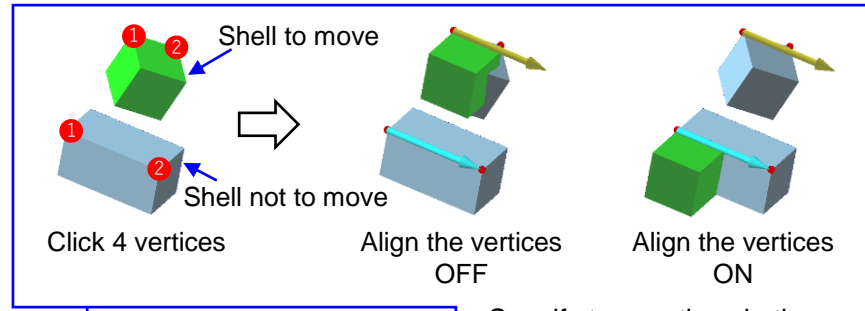
You can select to use the entire moving shell for alignment or the area you want to point to.

Click the destination shell and click the "Calculate" button to start the calculation.

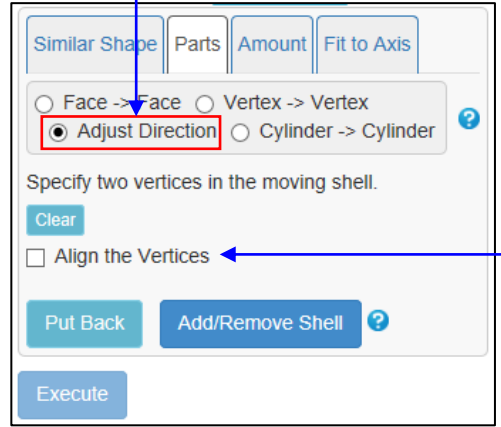
# Arrange Shells (3)



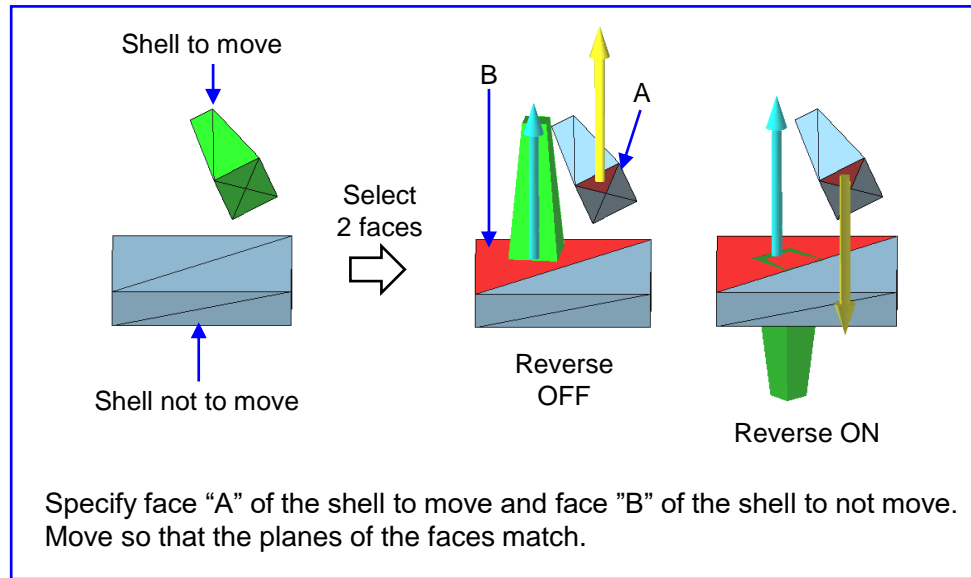
Specify one vertex in the moving shell and one vertex in the non-moving shell. The moving shell translates so that the two vertices match.



Specify two vertices in the shells to move, then two vertices in the shells not to move. Move so that the directions from the first vertex to the second vertex match.

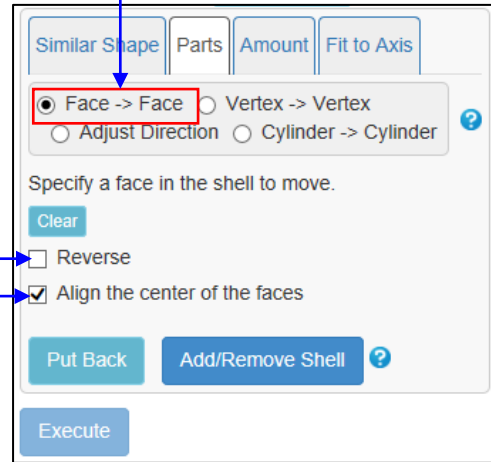


If unchecked, the moving shells will be moved so that the directions from the first vertex to the second vertex match. If checked, the shells will be translated from the above position so that the first vertices match.

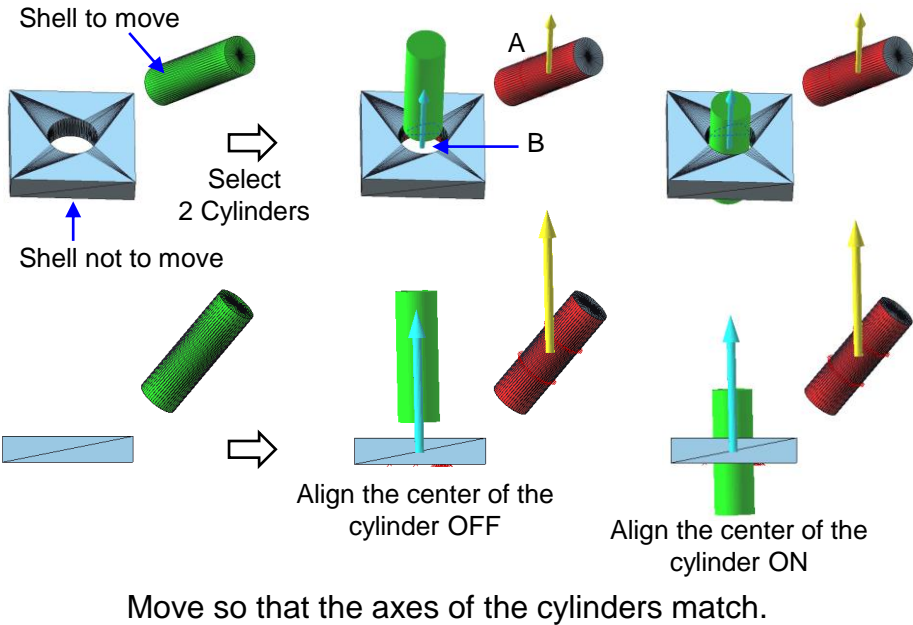


Specify face "A" of the shell to move and face "B" of the shell to not move. Move so that the planes of the faces match.

If unchecked, the front sides of "A" (a face to move) and "B" (a face not to move) will be placed facing each other.  
 If checked, the front side of "A" will be placed facing the back side of "B".  
 If unchecked, A will be placed closest to A on an infinitely wide plane containing B.  
 If checked, A will be moved parallel from the above position so that the centers of gravity of A and B match.



# Arrange Shells (4)



Similar Shape Parts Amount Fit to Axis

Face -> Coordinate Axis

3 vertices -> Coordinate Axis

Specify an axis that aligns the surface direction with faces of the moving shell.

+X  -X  +Y  -Y  +Z  -Z

Free-form

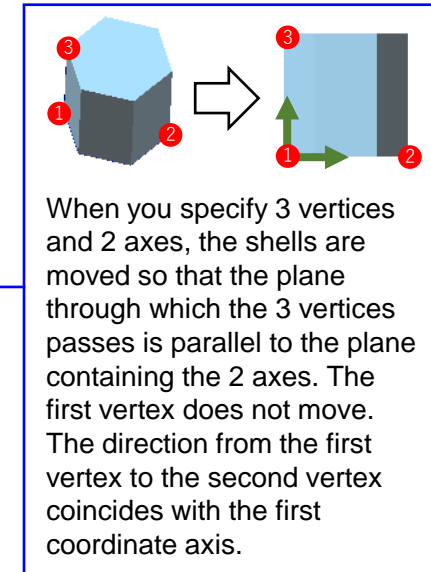
Clear

Put Back Add/Remove Shell

Execute

The shells move so that the average normal direction on the front side of the specified faces coincides with the specified axis direction.

Select the axis direction and specify the faces that align the front normal with the direction.



Similar Shape Parts Amount Fit to Axis

Face -> Coordinate Axis

3 vertices -> Coordinate Axis

Specify an axis that aligns with three vertices in the moving shell.

Direction from First point to Second point

+X  -X  +Y  -Y  +Z  -Z

Rough direction from First point to Third point

+X  -X  +Y  -Y  +Z  -Z

Clear

Put Back Add/Remove Shell

Execute

Specify the coordinate axis directions that match the directions from the first point to the second point.

Specify the rough axis direction from the 1st vertex to the 3rd vertex.

Similar Shape Parts Amount Fit to Axis

Face -> Face  Vertex -> Vertex

Adjust Direction  Cylinder -> Cylinder

Specify cylindrical part in the shell to move.

Clear

Reverse

Align the center of the cylinders

Put Back Add/Remove Shell

Execute

Specify the cylindrical part "A" in the shell that moves and the cylindrical part "B" in the shell that does not move. Click one location to automatically detect the cylindrical area.

If clicked, the direction of the cylinder axis of "A" is reversed.

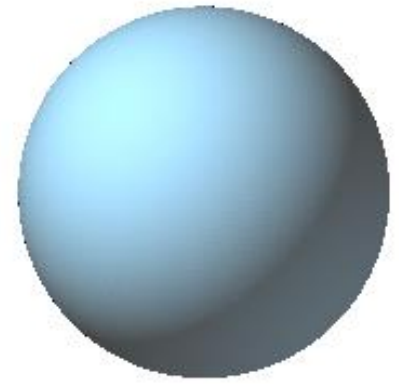
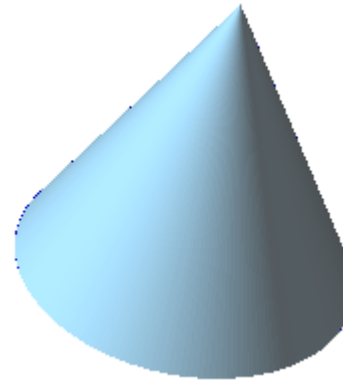
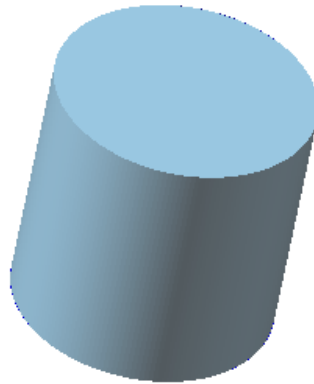
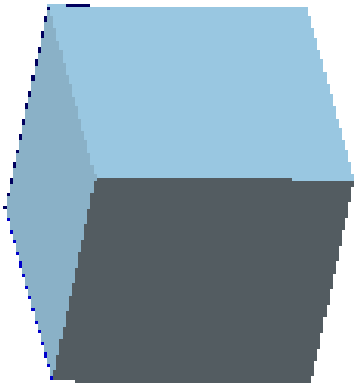
If unchecked, the shells will be placed so that the two axes match and "A" moves to the closest position on that axis.

If checked, the shell will be placed where the two axes match and the axial midpoints of the two cylinders match.



# Primitive Solid

Create a cuboid, cylinder, cone or sphere shell.



Primitive Solid [How to use](#)

Solid Type ?  
 Rectangular  Cylinder  Cone  Sphere

Position ?  
0,0,0

X Length  
10.0

Y Length  
10.0

Z Length  
10.0

Execute

Create a rectangular parallelepiped whose sides are perpendicular to the axes.

The center position of the rectangular parallelepiped.

The length of each side.

Primitive Solid [How to use](#)

Solid Type ?  
 Rectangular  Cylinder  Cone  Sphere

Position ?  
0,0,0

Height  
20.0

Radius  
10.0

Division Number of Circle  
3  360  
20

Execute

Create a cylinder whose axis is parallel to the Z axis.

The position of the center of gravity of the cylinder.

The height and radius of the cylinder

The number of sides of a regular polygon that approximates a circle. (For example, if 6 is specified, a hexagonal column will be created.)

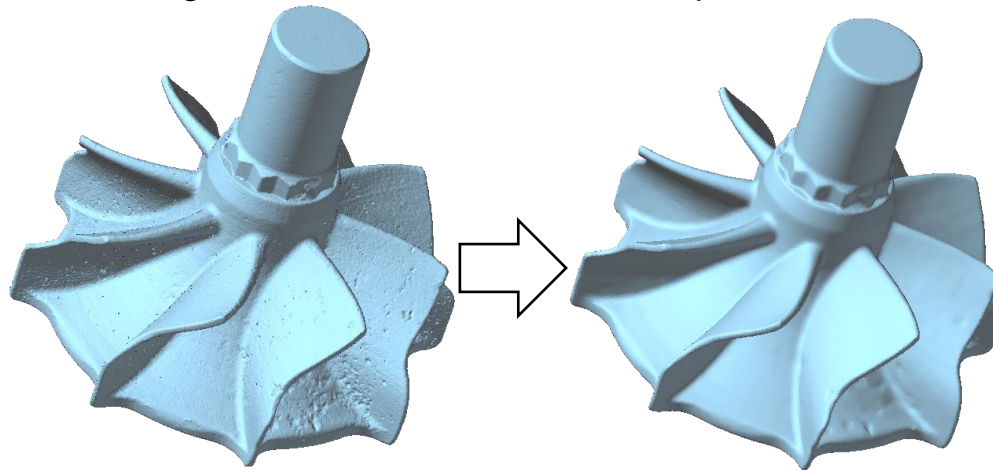
## 8. Edit Tab

# Reshape CT Mesh (1)

This command improves the defective or incorrect parts of the mesh derived from CT measurement as follows.

- Reduce minute irregularities, steps and waviness on the surface
- Remove small protrusions. And fill small dents and small through holes.
- Fix problems detected by cleaning commands such as self-intersection.
- Reshape each face close to an equilateral triangle

It is assumed that this command deals with the mesh immediately after being polygonized from voxel data. Use this command before executing other commands such as simplification.



Cleaning		
<input checked="" type="checkbox"/> Inspection Items	▼	Detection Qty
<input checked="" type="checkbox"/> Reversed Faces	?	0
<input checked="" type="checkbox"/> Gap between Shells	▼ ?	0
<input checked="" type="checkbox"/> Tiny Faces	▼ ?	132
<input checked="" type="checkbox"/> Irregular Connection	?	6081
<input checked="" type="checkbox"/> Holes	▼ ?	0
<input checked="" type="checkbox"/> Folded Faces	?	180
<input checked="" type="checkbox"/> Small Shells	▼ ?	855
<input checked="" type="checkbox"/> Overlapped Shells	▼ ?	0
<input checked="" type="checkbox"/> Self-Intersection	?	942

Cleaning		
<input checked="" type="checkbox"/> Inspection Items	▼	Detection Qty
<input checked="" type="checkbox"/> Reversed Faces	?	0
<input checked="" type="checkbox"/> Gap between Shells	▼ ?	0
<input checked="" type="checkbox"/> Tiny Faces	▼ ?	1
<input checked="" type="checkbox"/> Irregular Connection	?	0
<input checked="" type="checkbox"/> Holes	▼ ?	0
<input checked="" type="checkbox"/> Folded Faces	?	0
<input checked="" type="checkbox"/> Small Shells	▼ ?	0
<input checked="" type="checkbox"/> Overlapped Shells	▼ ?	0
<input checked="" type="checkbox"/> Self-Intersection	?	0

**Reshape CT Mesh** ? [How to use](#)

Strength of Reshaping ? ←

Weakest 
○
○
 Strongest

Leave only one shell

Specify Maximum Gap

Target of the Maximum Gap

0.1

Execute

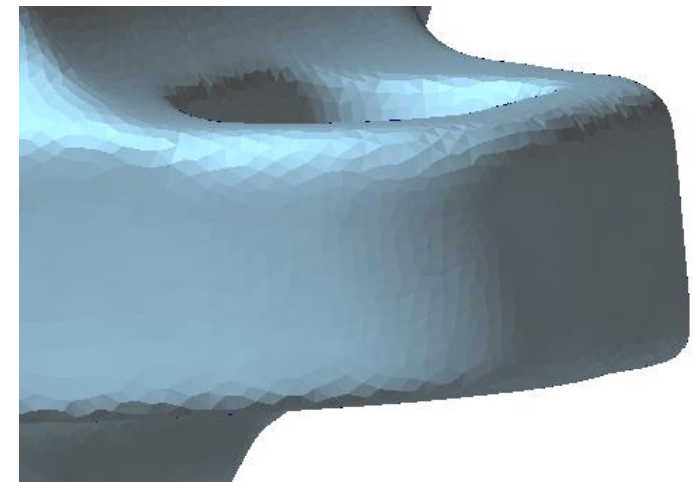
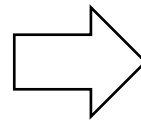
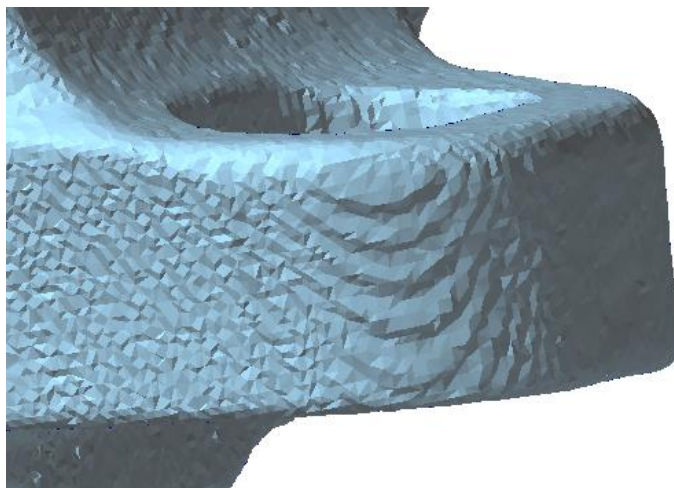
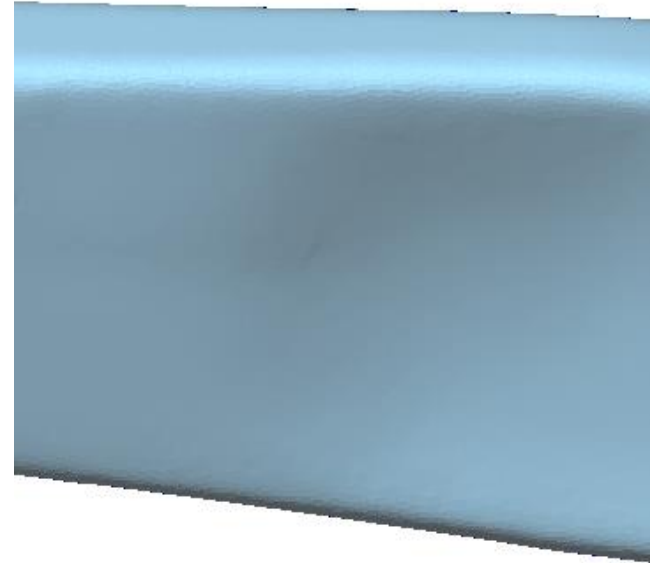
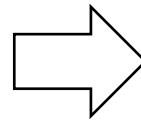
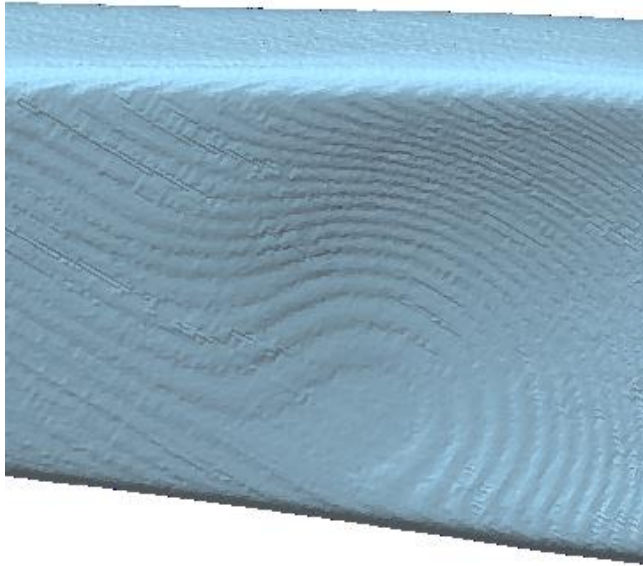
If checked, the shell with the most faces will be left and other shells will be deleted.

If checked, the mesh will be shaped so that the distance from the original shape is within the specified value.

Select "Strength of Reshaping" from 5 levels. If accuracy is important, specify "weak" or "weakest", and if smoothness is important, specify "strong" or "strongest". If it is "weakest", only the problem areas detected by the cleaning command will be fixed, and the operation to make a smooth equilateral triangle will not be performed.

## Reshape CT Mesh (2)

Examples of reducing wavy artifacts

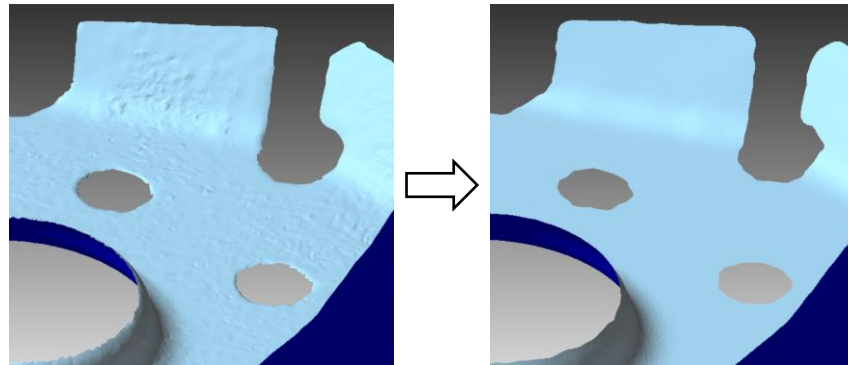


# Reshape Optical Mesh

This command improves the defective and incorrect parts of the mesh derived from optical measurement as follows.

- Reduce minute irregularities on the surface
- Makes up for mesh defects caused by light not reaching (fills holes in the mesh surface)
- It reduces the shape disorder caused by measuring the edge of the object to be measured.
- Fix problems detected by cleaning commands such as self-intersection.
- Reshape each face close to an equilateral triangle

It is assumed that this command deals with the mesh immediately after being polygonized from voxel data. Use this command before executing other commands such as simplification.



If checked, the bend of the mesh near shell boundaries is reshaped and the shape of the shell boundary lines and the surrounding mesh surface are smoothed.

Reshape Optical Mesh ? [How to use](#)

Strength of Reshaping ? ← Weakest Strongest

Shell boundary correction ?

Specify Maximum Gap

Target of the Maximum Gap

0.05

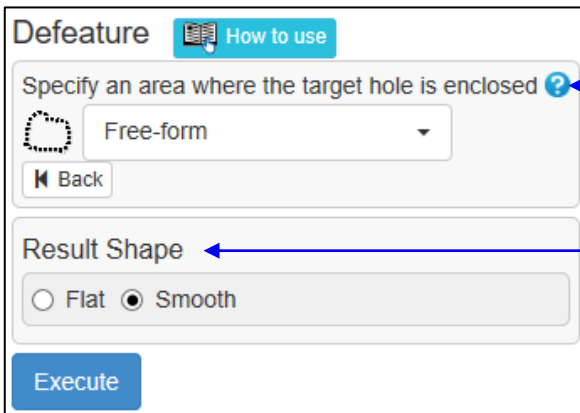
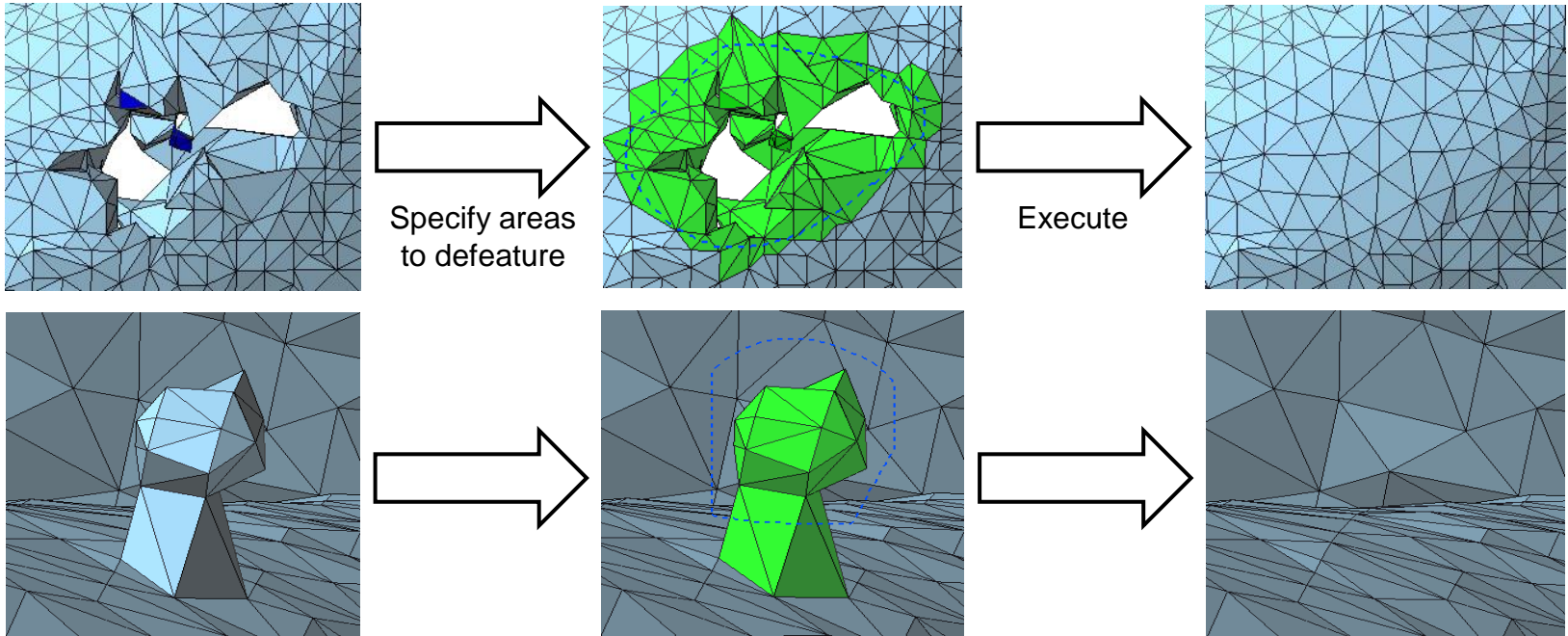
Execute

Select "Strength of Reshaping" from 5 levels. If accuracy is important, specify "weak" or "weakest", and if smoothness is important, specify "strong" or "strongest". If it is "weakest", only the problem areas detected by the cleaning command will be fixed, and the operation to make a smooth equilateral triangle will not be performed.

If checked, the mesh will be shaped so that the distance from the original shape is within the specified value.

# Defeature

This command removes through holes, blind holes, pits, protrusions, self-intersections, etc. in the specified area.



Specify areas to defeature.  
Drag while looking at the area from directly above.

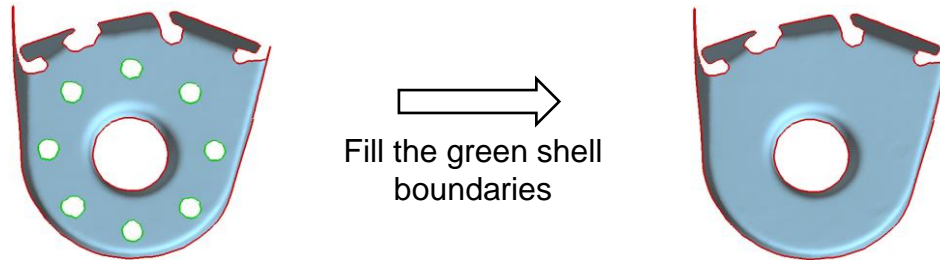
Select whether to flatten the indication area ("Flat") or connect it smoothly with the surroundings ("Smooth").

## Fill Holes (1)

This command fills the holes on the mesh surface. There are three ways to fill the holes. (Use the "Defeature" or "Fill Caves" commands to fill through holes.)

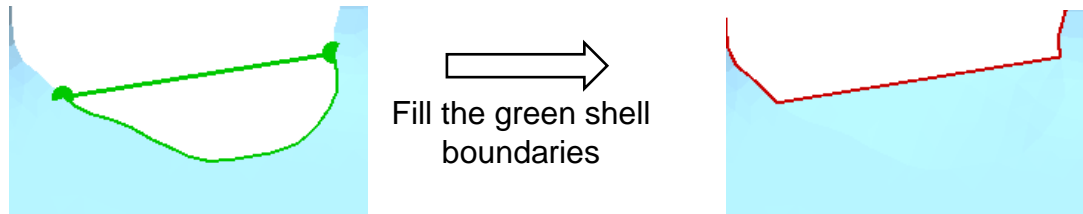
- Filling

Fill the specified holes. Plural holes can be filled at once.



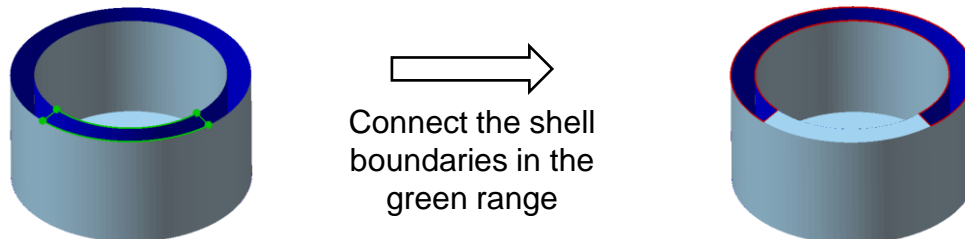
- Partial Filling

Fills only part of the shell boundary, such as the dents on a shell boundary.



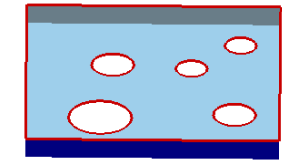
- Bridge

Two shell boundaries can be connected.



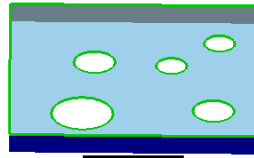
# Fill Holes (2)

## Select holes to fill

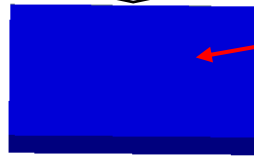
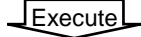


There are 6 holes.  
(The outer boundary is also one of the holes)  
Green: Target to fill holes  
Red: Not target to fill holes

Click "Select All" button

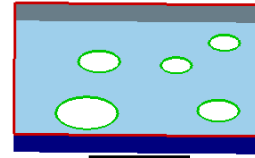


Fill in the holes including the outer boundary



The surface that fills the outer boundary

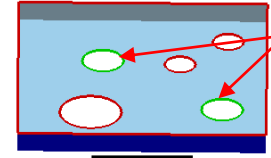
Click "Select holes except largest one" button



Fill holes other than the outer boundary

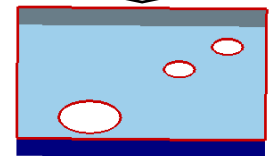


Click the hole to select



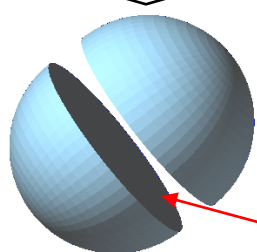
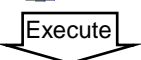
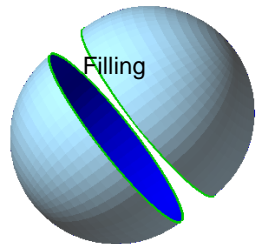
Click holes

Fill the clicked holes

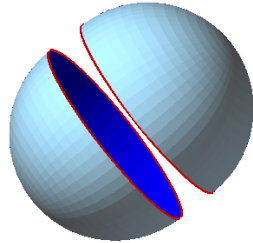


## Join shells

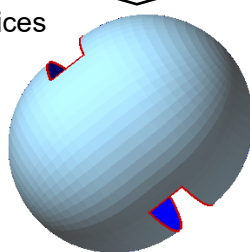
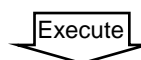
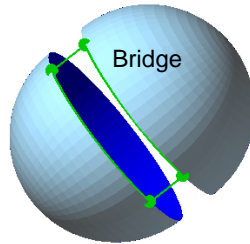
How to connect facing hemispheres



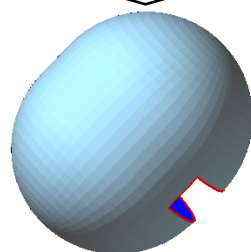
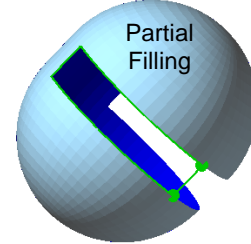
When you fill all the holes in "Filling", each shell becomes a hemispherical solid.



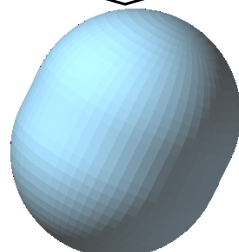
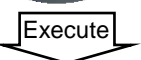
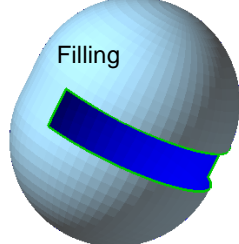
Create faces that connect the shells with "Specify Vertices" of "Bridge".



Expand the connected faces with "Partial Filling".

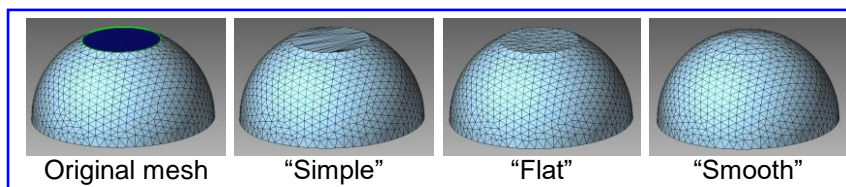


Fill in the remaining hole with "Filling" to complete the connection.





# Fill Holes (3)



There are three ways to make a surface that fills a hole. (See the figure above)  
 "Simple": Connect the vertices around the hole with a line segment.  
 "Flat": Fills flat like "Simple", but creates faces that are close to equilateral triangles.  
 "Smooth": The faces are created so that they connects smoothly with faces around the hole.

"Specify Edges" bridges between two edges.  
 "Specify Vertices" builds a bridge between two sections sandwiched between vertices.  
 (Right in the figure below)

Select the filling method (see previous page).

Click a hole to turn it green (selected).  
 Click it again to return it to red (unselected).

The number of selected holes and the total number of holes are displayed.

Click "Select All" button to select all the holes.

Click "Select holes except largest one" button to select all holes except the largest hole (outer shell boundary).

If you click the "Select holes with" button, the holes whose number of edges is less than or equal to the value in input field will be selected.

**Fill Holes** ? How to use

Methods for Filling Holes ?

Filling  Patial Filling  Bridge

Specify holes to be filled. ?

Selected Hole Qty / All Hole Qty:  
0 / 5

Select All

Select holes except largest one

Select holes with  or fewer  
edges around the hole.

Filling Hole Patterns ?

Simple  Flat  Smooth

Execute

**Fill Holes** ? How to use

Methods for Filling Holes ?

Filling  Patial Filling  Bridge

Specify 2 points that indicate an area to be filled.

Reverse a side that should be filled.

Filling Hole Patterns ?

Simple  Flat  Smooth

Execute

Click on the two vertices on a shell boundary.  
 The space between the two vertices is filled.

Click " Reverse a side that should be filled " to change how to select the section to be filled.

**Fill Holes** ? How to use

Methods for Filling Holes ?

Filling  Patial Filling  Bridge

Specification Mode ?

Specify Edges  Specify Vertices

Specify 2 edges that indicate an area to be bridged.

Reverse a side that should be filled.

Filling Hole Patterns ?

Simple  Flat  Smooth

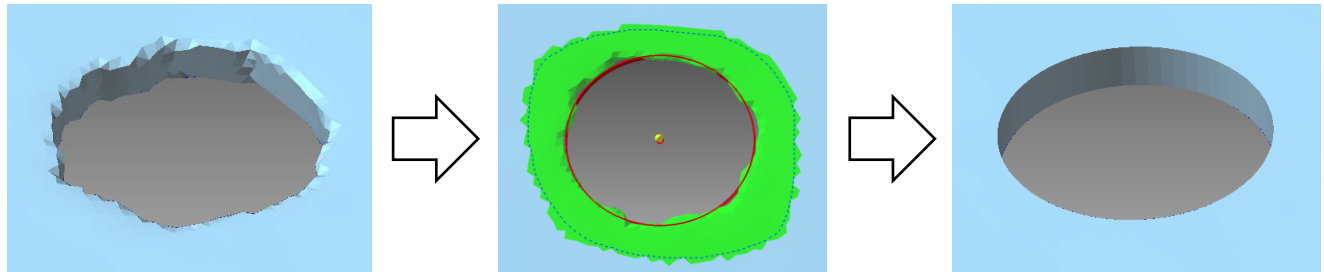
Execute

# Reshape Hole (1)

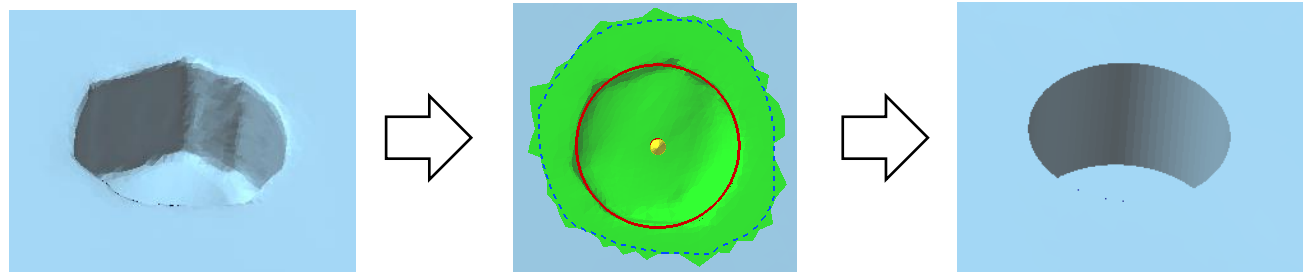
Corrects the turbulence around the holes seen in the mesh derived from the measurement. CAD-derived mesh is also useful when changing the hole diameter.

For round holes, it supports three types of holes: through holes, blind holes, and holes on shell(mesh surface).

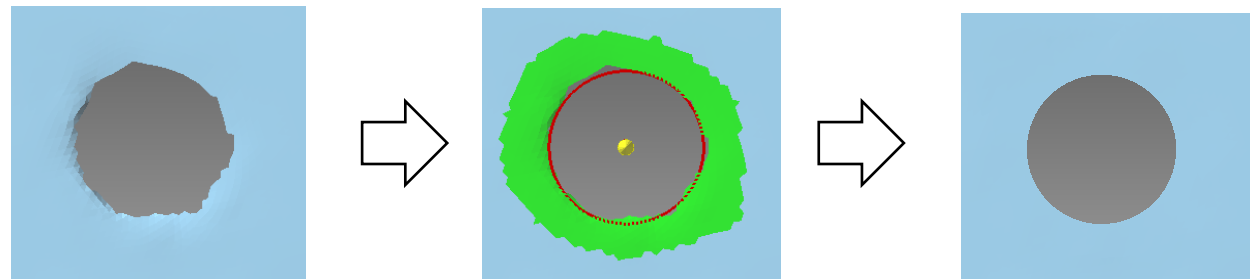
Reshape through holes



Reshape blind holes



Reshape shell holes



Original

Specify the hole area  
to be reshaped

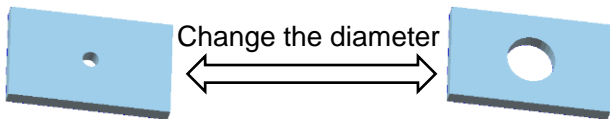
Result

# Reshape Hole (2)

Adjust the viewing state so that the hole is viewed from the axial direction, and drag the area slightly larger than the hole.

Select the hole type (after shaping).

Specify the diameter of the hole.  
(You can change the diameter of the hole)



**Reshape Hole** How to use

Specify the area to be shaped. ?

Free-form

Back

---

Hole type

Through Hole  Blind Hole  Shell Hole

Create a Cylinder ?

Filled  Unfilled

Diameter

Blind Hole Depth

View from the axial direction of the hole

Advanced Setting ▼

Execute

---

Advanced Setting

Division Number of Circle

3  360

Center of Hole (X,Y,Z)

Hole Axis

Execute

Click to change the viewing state so that you can see the hole from the axial direction.

Click this mark to show / hide the following input fields.

The number of sides of a regular polygon that approximates the circle around the hole.

The coordinate value of the point through which the hole axis passes.

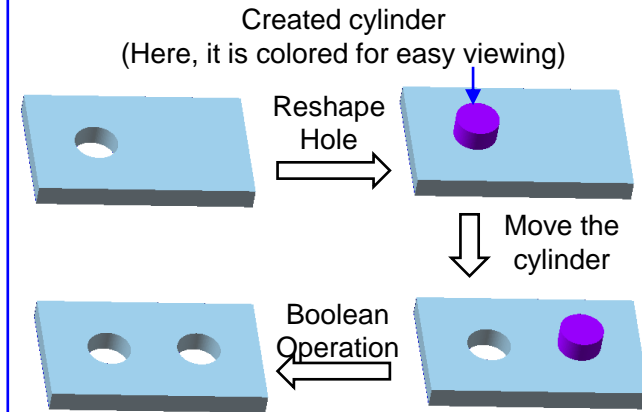
The direction of the hole axis.

If checked, a cylinder with the size of a hole will be created. The holes can be filled or not filled.

The created cylinder can be used for moving or copying the hole.

Select "Filled" to move the hole, and select "Unfilled" to copy the hole.

Example of copying a hole

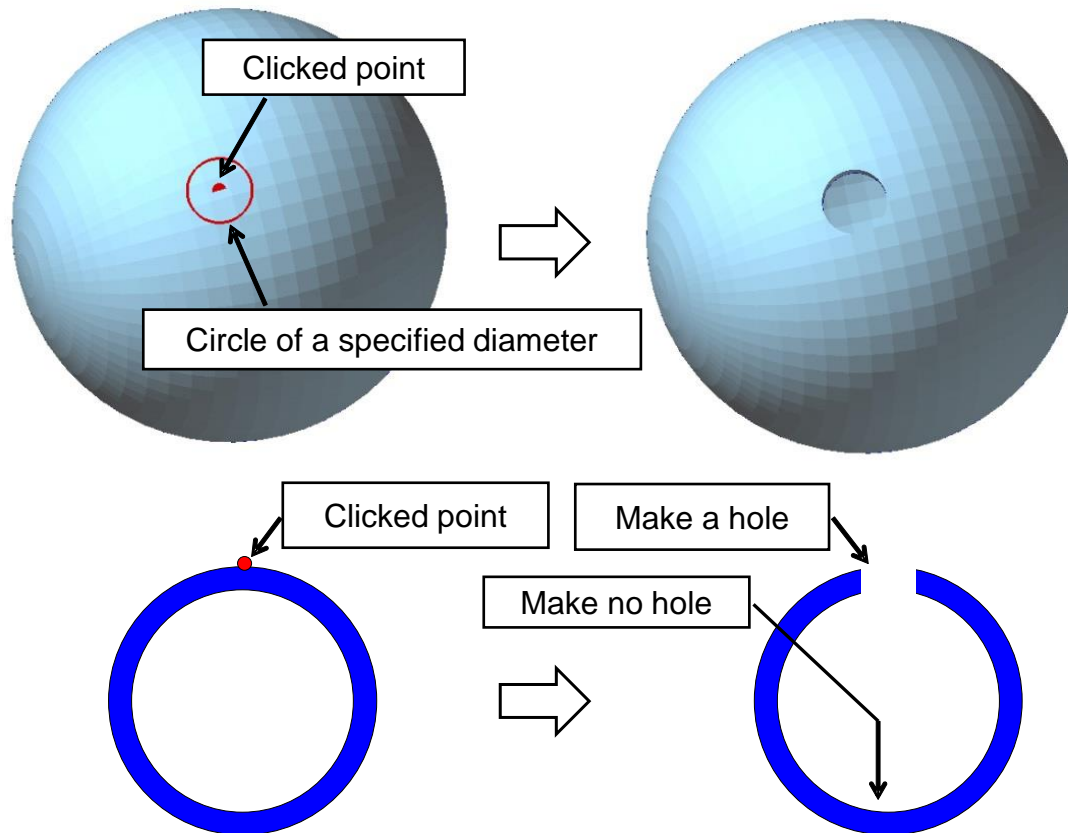


Specify the depth of the blind hole.  
(Displayed when a blind hole is selected)

## Through Hole (1)

Make a round through hole at the specified position.

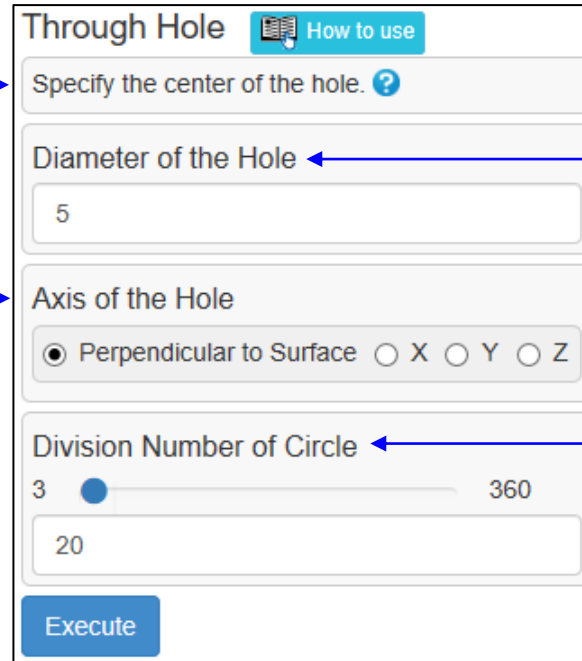
A hole with the specified diameter will be drilled around the clicked position. As shown in the figure below, even if the central axis of the hole intersects the mesh other than the clicked position, the hole will only be made at the clicked position.



## Through Hole (2)

Click where you want to make the hole. The point at the center of the hole and the circle of the size corresponding to the "Diameter of the Hole" are displayed in red.

The axial direction of the hole. Select from "Perpendicular to Surface" (direction perpendicular to the face where the center of the hole is located), "X" (X-axis direction), "Y" (Y-axis direction), and "Z" (Z-axis direction).



The image shows a software dialog box titled "Through Hole" with a "How to use" icon. It contains several input fields and a button. Blue arrows point from text annotations to specific parts of the dialog box.

- The "Specify the center of the hole." field is pointed to by an arrow from the first text block.
- The "Diameter of the Hole" input field, which contains the value "5", is pointed to by an arrow from the second text block.
- The "Axis of the Hole" section, which has radio buttons for "Perpendicular to Surface", "X", "Y", and "Z", is pointed to by an arrow from the third text block.
- The "Division Number of Circle" section, which includes a slider from 3 to 360 and an input field containing "20", is pointed to by an arrow from the fourth text block.
- An "Execute" button is located at the bottom of the dialog box.

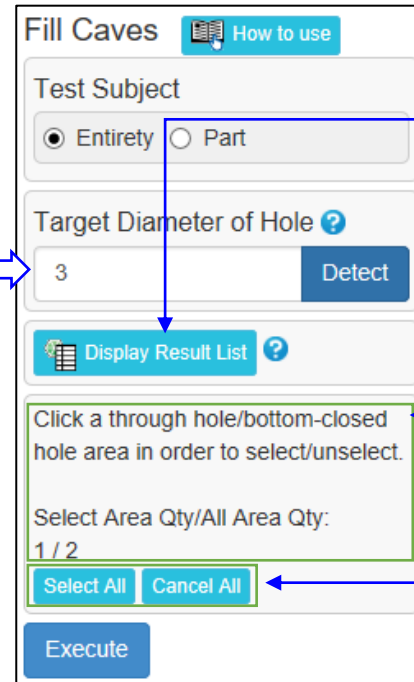
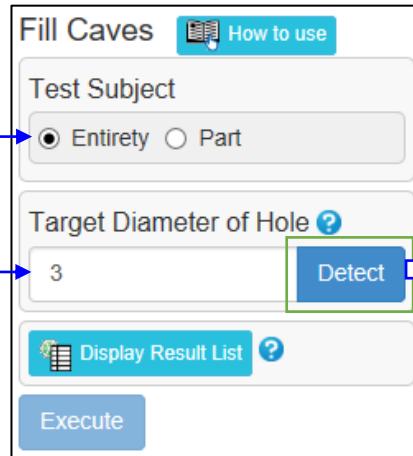
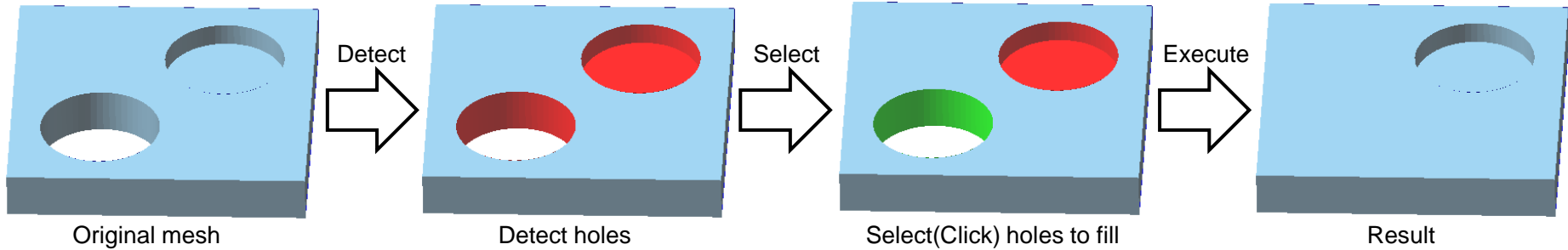
The diameter of the hole to make.

The number of sides of a regular polygon that approximates the circle of a round hole.

# Fill Caves

This function detects through holes and blind holes of a mesh and fills them.

The area where a sphere of the specified diameter cannot be inserted is automatically detected as a hole and displayed in red. You can select holes to fill (holes turn green when selected) and fill it.



Select whether to detect in the entire mesh or in a partial area.

Detects where a sphere with the specified diameter cannot be inserted.

After setting the parameters, click the "Detect" button. When the detection process is complete, the command dialog will change as shown on the right, and the detected area will be displayed in red.

Click this button to save the detection results.

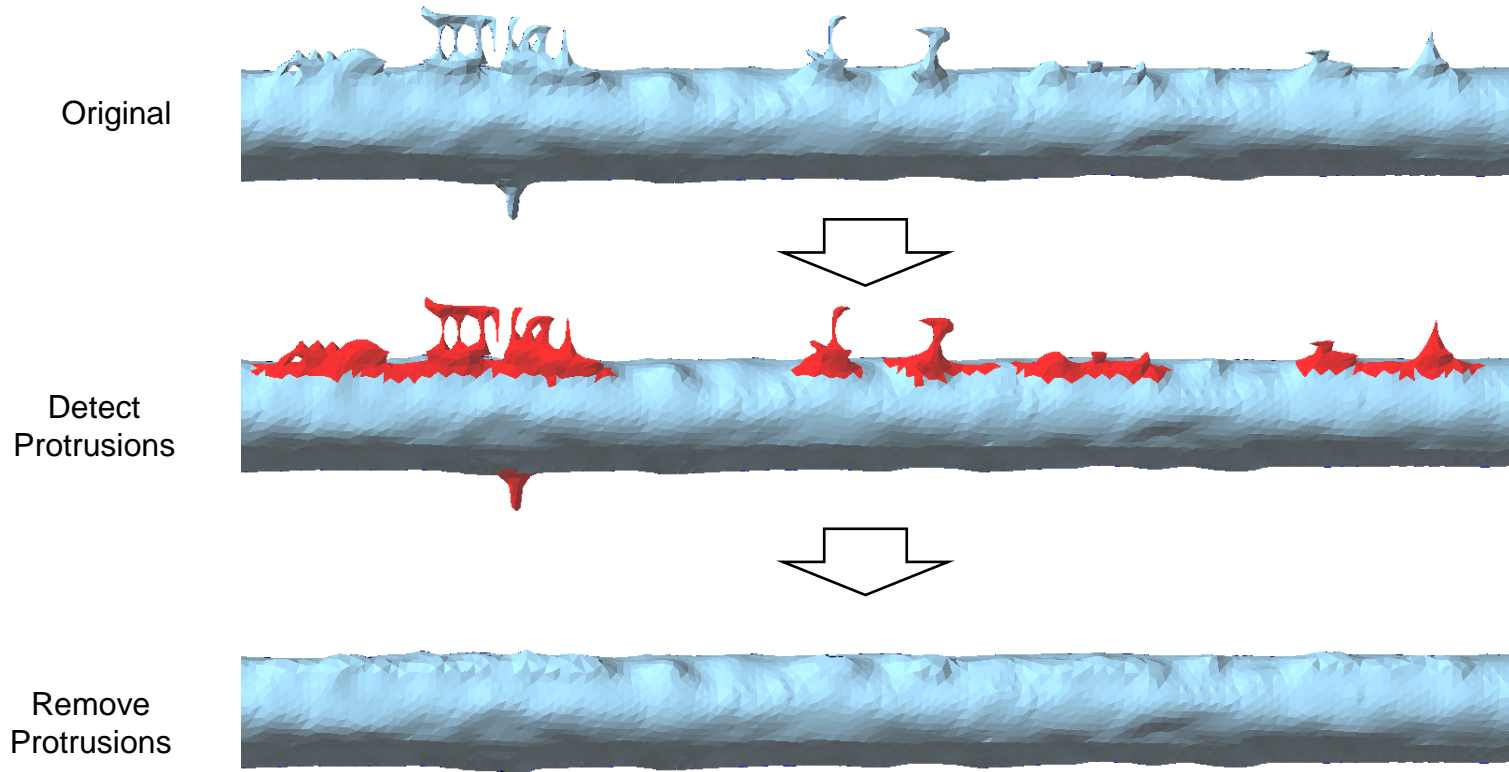
Click the area to be filled from the detected areas in red. It turns green. In this example (above figure), it means that two areas are detected and one of them is selected (green) to fill.

Select All: If clicked, all detected areas are selected (green).  
Cancel All: If clicked, all detected areas are deselected (red).


## Remove Protrusion (1)

Detects and removes protrusions on the mesh.


It automatically detects the part where a sphere of the specified diameter cannot be inserted and displays it in red. The user can select and remove any protrusion from them.





## Remove Protrusion (2)

Remove Protrusion  How to use

Inspection Target  
 Entirety  Part

Estimated Diameter of Protrusion   
4 Detect

 Display Result List 



Execute

Select whether to detect in the entire mesh or in a partial area.

Detects where a sphere with the specified diameter cannot be inserted.

After setting the parameters, click the "Detect" button.

When the detection process is complete, the command dialog will change to the one shown below, and the detected area will be displayed in red.

 Display Result List 

Click protrusion region to switch select / cancel.

Selected Regions / Total Regions:  
21 / 278

Select All Cancel All

Execute

Click this button to save the detection results.

Click on the area to remove. It turns green.

In this example, it means that 278 areas are detected and 21 of them is selected (green) to remove.

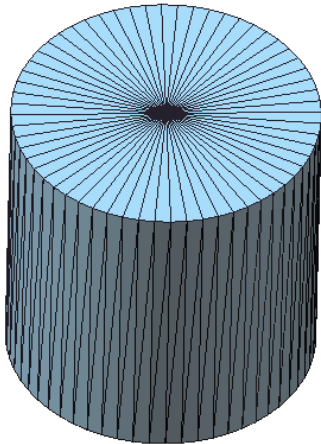
Select All: If clicked, all detected areas are selected (green).

Cancel All: If clicked, all detected areas are deselected (red).

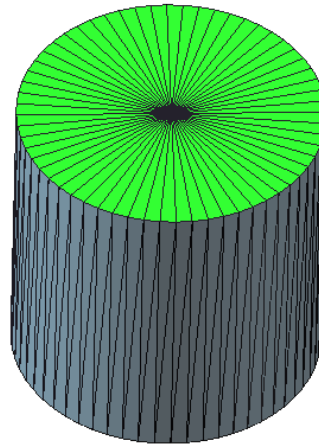
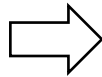


# Delete Faces

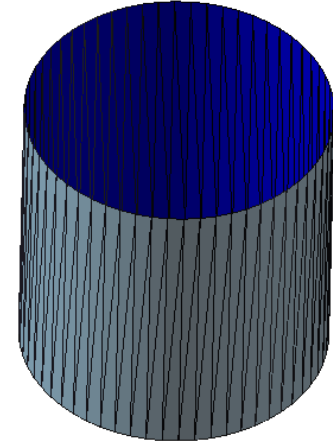
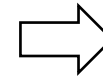
Delete the specified faces.



Original mesh



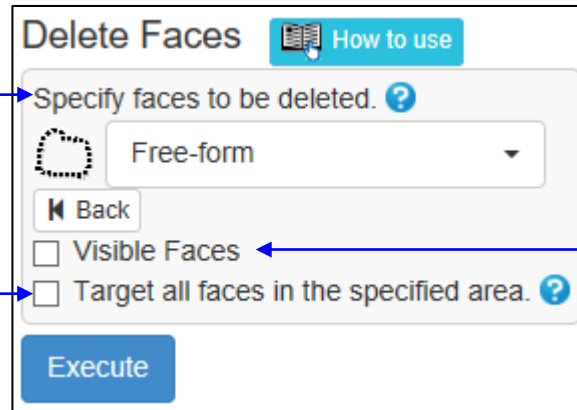
Specify faces (green) to be deleted



Mesh after deletion

Specify the area of the faces to be deleted.

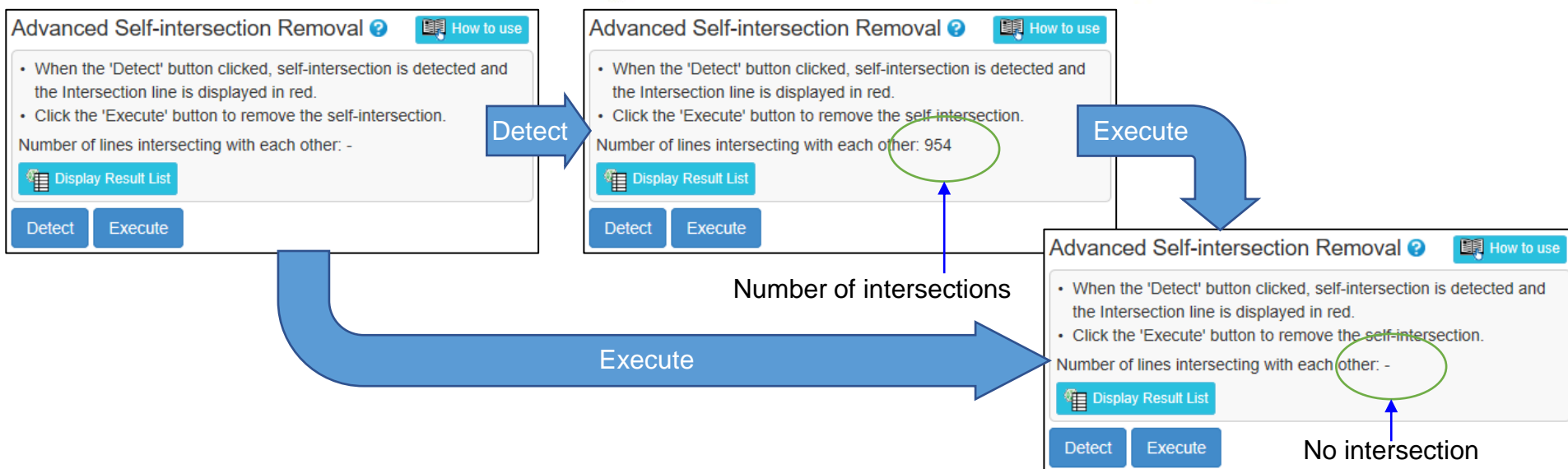
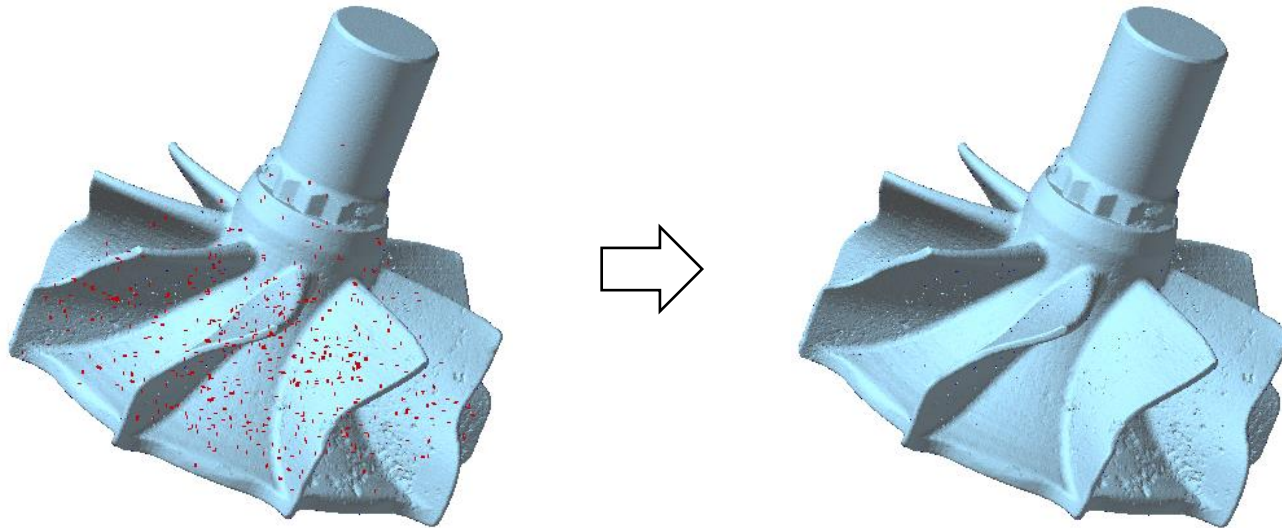
If checked, all faces in the specified area will be deleted.  
If unchecked, faces connected to the foremost face in the specified area will be deleted.



If checked, visible faces in the specified area will be deleted.

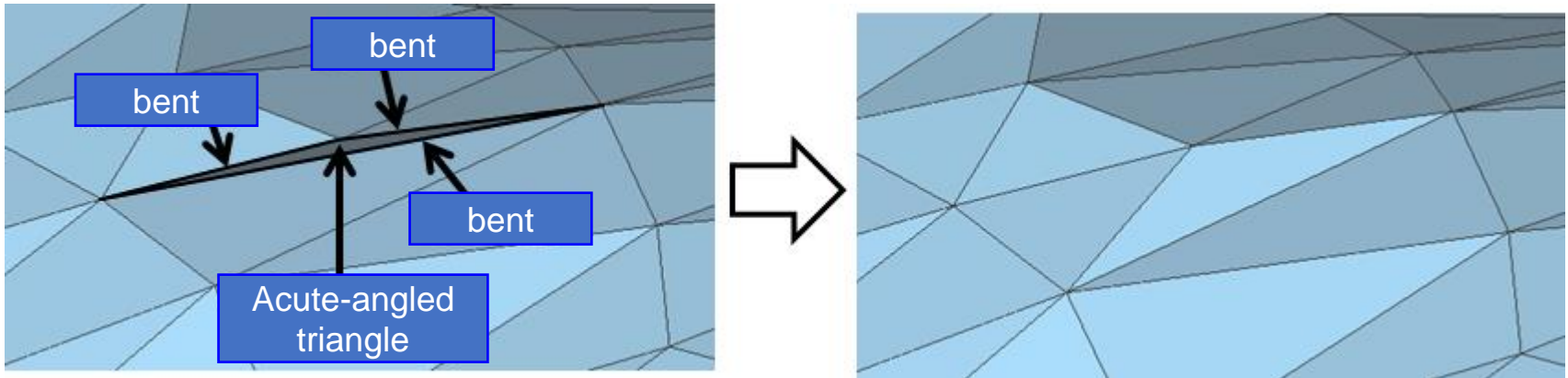
# Advanced Self-intersection Removal

Assuming that the faces are almost the same size, correct the self-intersection.  
Self-intersections that cannot be corrected by cleaning self-intersection removal can be corrected.



# Remove Acute Triangles

This command detects and removes sharp and broken triangles(faces).  
(Sharp and bent faces can cause small steps in the mesh.)



**Remove Acute Triangles** [How to use](#)

Detectable Acute Angles [?](#)  
0.0 <  <= 60.0

Crease Angle [?](#)  
0.0 <=  <= 180.0

Detection Qty: 614

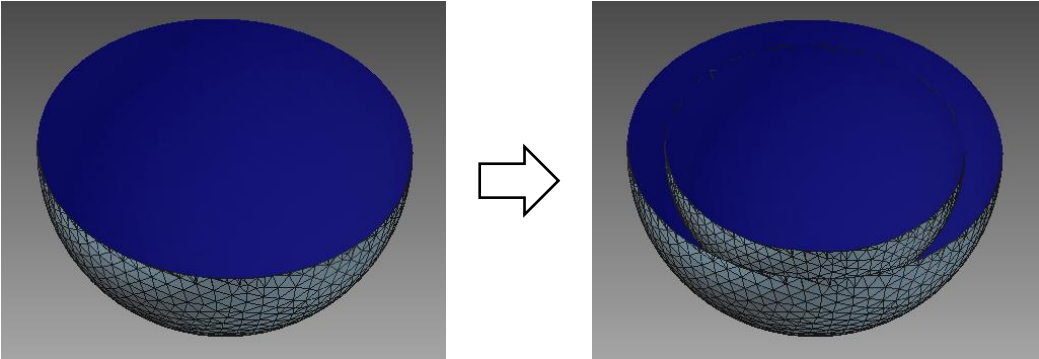
[Display Result List](#) [?](#)

[Detect](#) [Repair](#)

- ← Detects triangles that satisfy both of the following two conditions regarding "Detectable Acute Angles" and "Crease Angle".  
(1) Detects a triangle whose smallest angle (sharp) among the three corners of the triangle is less than or equal to the specified angle of " Detectable Acute Angles".  
(2) If the maximum angle (most bent) among the bending angles (angles formed by normal directions) with adjacent triangles is equal to or greater than the " Crease Angle" specified value, it will be detected.
- ← Click the "Detect" button to display the number of detected faces. This is an example of detecting 614 locations.
- ← Click the "Display Result List" to save the temporary figure of the detected part and open the result list window.

# Offset

This command moves all vertices in their normal direction to create shells that is a certain distance away from the original shells.



Example of offsetting to the "Reverse Side"

**Offset** [How to use](#)

Offset Subject  
 Entirety  Shell

Offset Amount ?  
2.0

Direction  
 Obverse Side  Reverse Side

Keep the original mesh.  
 Remove Self-Intersection ?  
 Create a smooth offset surface ?

Execute

Select whether to offset the entire mesh or the specified shells.

Specify the amount to offset.

Select whether to offset to the "Obverse Side" (front) or the "Reverse Side"(back) of the shells.

An offset shell may self-intersect. If checked, the self-intersection will be corrected automatically.

If checked, the mesh(shells) before offset will remain, and if unchecked, it will be deleted.

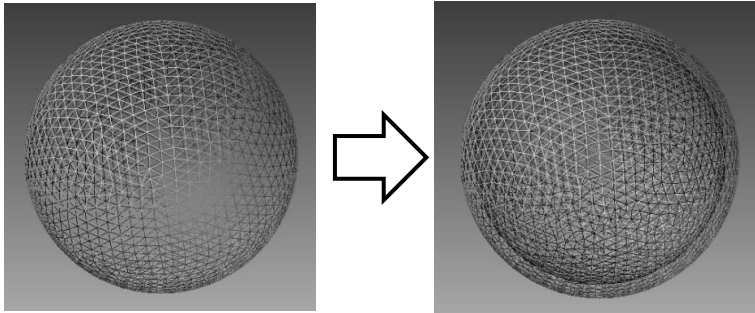
If checked, it will create a smooth offset surface with almost uniform face size.

# Thicken

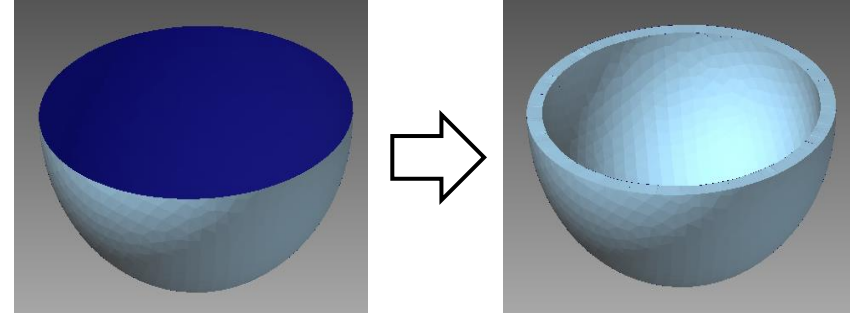
Add thickness to shells.

In the case of a closed shell (solid), it becomes a hollow solid,

In the case of an open shell, it will be a thick solid.



For a closed shell



For an open shell

Select to thicken the entire mesh or the specified shells.

Specify the amount of thickness.

Select whether to add thickness to the Obverse(front) side or the Reverse(back) side of the mesh(shells). Regardless of which side is thicker, the outside of the created mesh will be the front side.

The shell created by thickening may self-intersect. If checked, self-intersection will be corrected automatically.

If checked, it will create a smooth offset surface with almost uniform face size.

### Thicken [How to use](#)

**Thicken Subject**

Entirety  Shell

**Thickness ?**

2.0

**Direction**

Obverse Side  Reverse Side

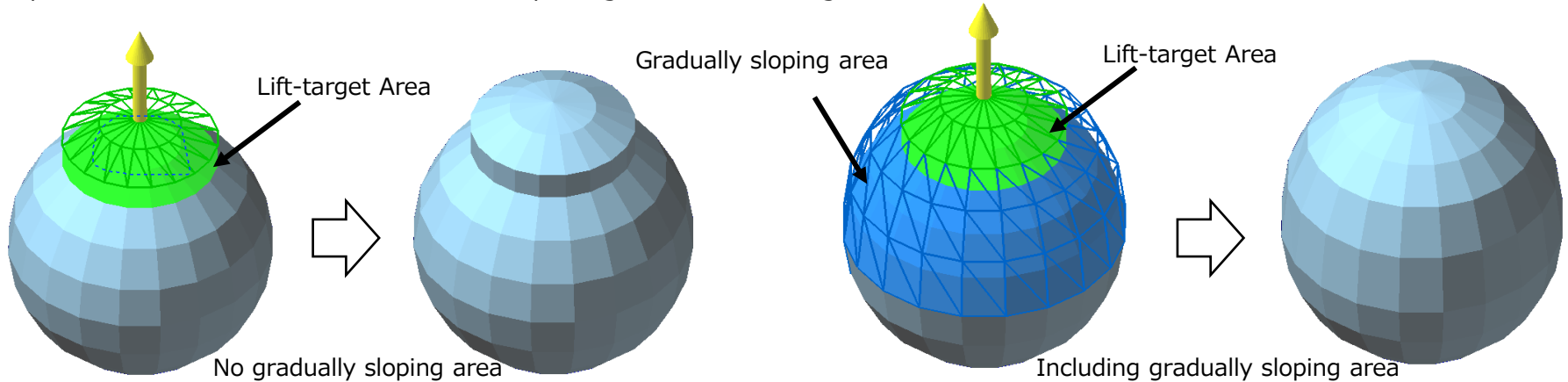
Remove Self-Intersection ?

Create a smooth offset surface ?

**Execute**

# Morphing (1)

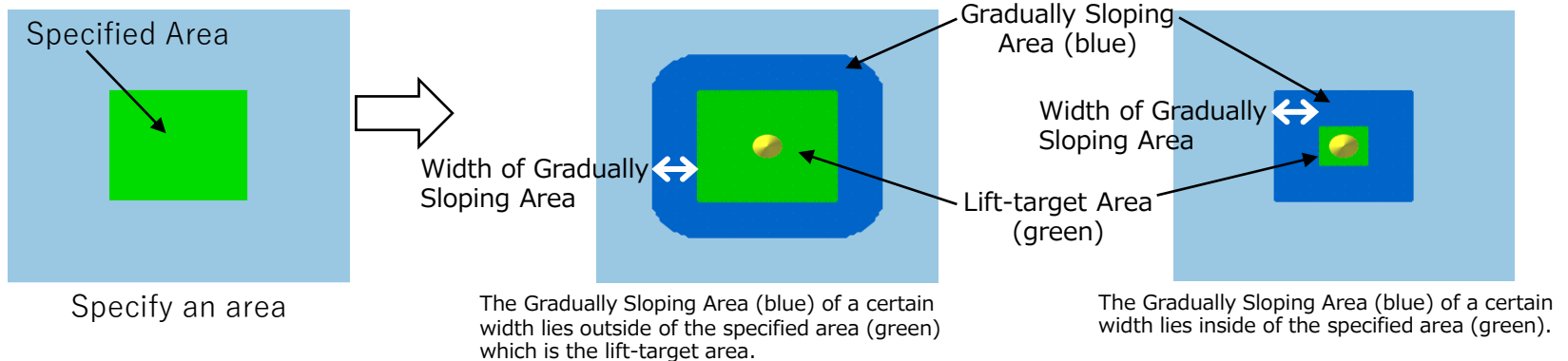
The "Morphing" command can lift an area as specified (hereinafter referred to as the "Lift-target Area"). The command can also lift the peripheral area together with the Lift-target Area by gradually decreasing the lifting amount of the peripheral area in accordance with the area's parting from the Lift-target Area.



Faces are created without lifting between the Lift-target Area and the peripheral area.

Areas (Lift-target Area and Gradually Sloping Area) can be specified by dragging the mesh. The areas can be identified through the use of a specified (green) area as indicated in the figures below.

Specify as a Gradually Sloping Area the belt area (blue) of a certain width inside of or outside of the specific (green) area. (The belt area lies adjacent to the boundary. It can be outside as shown in the middle figure, and it can be inside as shown in the right figure.) A Lift-target Area is created inside of the Gradually Sloping Area. (In other words, the specified green area is the Lift-target Area.)



# Morphing (2)

Choose a method for specifying a Gradually Sloping Area.

Specify an area. (See the previous page for reference.)

A belt-like Gradually Sloping Area with a specified width will be created inside of the specified area and adjacent to the boundary, if you choose 'Create a gradually sloping area in the area'. It will be created outside of the specified area and adjacent to the boundary, if you choose 'Create a gradually sloping area outside of the area'.

A lift-target area will be created inside of the Gradually Sloping Area.

Specify the direction of lifting.

If you choose 'Perpendicular', each vertex moves into a normal line direction as it does if the "Offset" command is executed.

If you choose 'X', 'Y', and 'Z', each vertex moves into the coordinate axis direction.

Specify an amount of lifting for the Lift-target Area.

If you specify a negative value, and if you have specified Perpendicular, each vertex will be repositioned to dent the Lift-target Area.

If you have specified a coordinate axis direction, each vertex will be repositioned to transform the Lift-target Area.

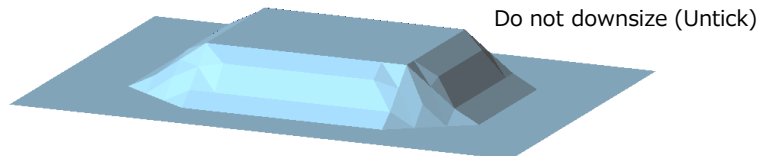
Grip and move the yellow arrow in order to tweak the repositioning amount.

Choose 'Specify a gradually sloping area', in order to show the buttons.

Click the button 'Lift-target area' in order to specify an area as the target for lifting.

Click the button 'Gradually sloping area' in order to specify an area where the lifting amount will be gradually increased or decreased.

A Gradually Sloping Area will be transformed in accordance with the Lift-target Area being transformed.

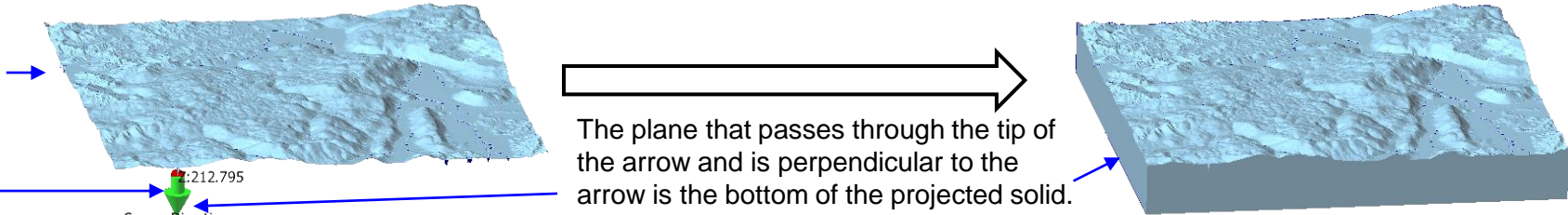


Tick in order to specify granularity on a scale of 1 to 3. (See below the sliding scale with stops at the three positions corresponding to the granularity levels.)

Downsize faces in a Gradually Sloping Area in accordance with the granularity scale. The smaller the faces become, the smoother the area becomes. (See the left figure.)

# Projection Solid

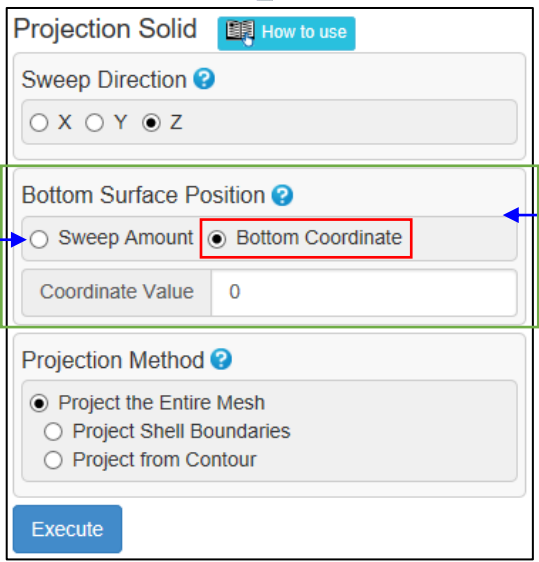
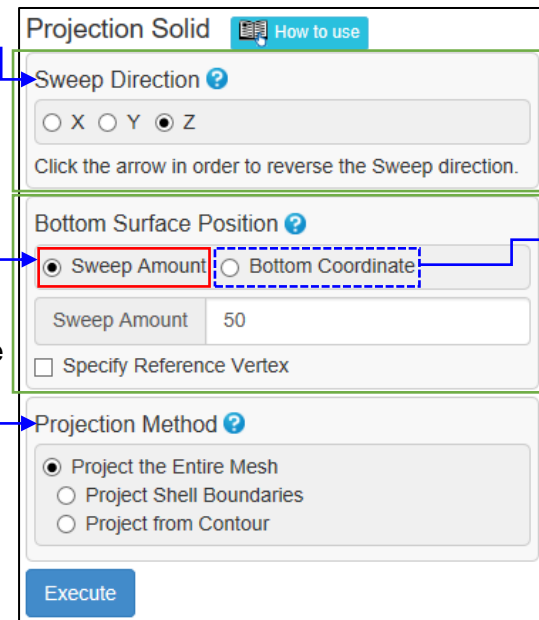
Sweeps the shell boundaries or contours of an open shell to a specified position in the specified coordinate axis direction to make a closed solid shell.



Select the coordinate axis direction to sweep.  
Click the arrow to change the sweep direction (+side / -side of the coordinate axes).

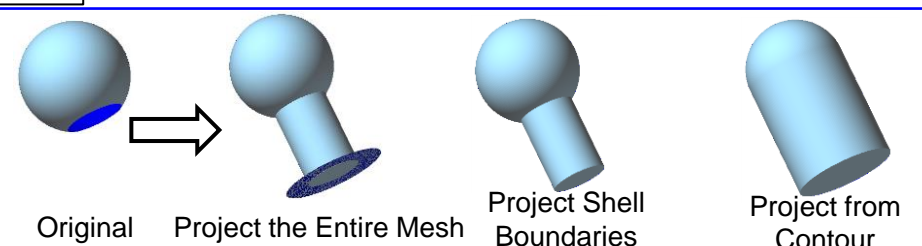
If "Sweep Amount" is selected, a solid will be created with the bottom surface at a position sweep amount away from the reference vertex in the sweep direction.

The reference vertex automatically selects the maximum or minimum vertex in the sweep direction, but if "Specify Reference vertex" is checked, you can also specify by clicking the reference vertex.



If "Bottom Coordinate" is selected, a solid will be created with the position of the "Coordinate Value" as the bottom.

Select how to make the bottom. (See the figure on the right)  
 "Project the Entire Mesh"=Project all faces in the shell onto the projective plane and connect the shell boundaries with the swept faces.  
 "Project Shell Boundaries"=Sweep the shell boundary to the projective plane to create faces on the bottom.  
 "Project from Contour"=Sweep the contour line to the projection plane when viewed from the sweep direction and create faces on the bottom.

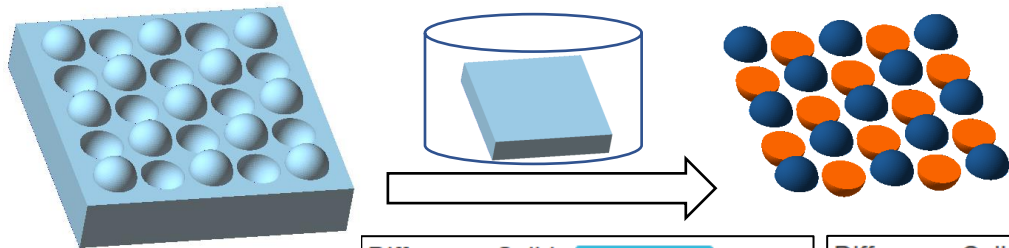




# Difference Solid

Compare the shapes of the mesh (imported into POLYGONALmeister) with the mesh of the external file, or compare the shapes of the two shells to create solid shells that corresponds to the difference. (Measure the distance between the vertices of the “Reference of Measurement” and the other, and detect the area that is more than the threshold as difference.)

The created shells have an orange face color if they are located on the front side of the other (not “Reference of Measurement”) shape, and a blue face color if it is located on the back side.



Create the shape of the difference between the rectangular parallelepiped in the external file and the mesh with spherical depressions and protrusions on the rectangular parallelepiped. (“Reference of Measurement” is an external file)

Click the button and select the external file.

Choose whether to measure the distance from an external file or the mesh.

The difference is the area where the meshes are separated from each other by the “Distance Threshold” or more.

If checked, the mesh is deleted after execution.

**Difference Solid** [How to use](#)

Processing Target  
 External File  Between Shells

Difference detection target file  
[Select File](#)

Display external file mesh

Reference of Measurement ?  
 External File  Mesh

Distance Threshold 1

Output Target ?  
 Front Side\_Orange  Back Side\_Blue  
 Both

Do not color  
 Delete displayed mesh

[Execute](#)

**Difference Solid** [How to use](#)

Processing Target  
 External File  Between Shells

Reference of Measurement ?  
 First Shell\_Green  
 Second Shell\_Blue  
[Deselection](#)

Distance Threshold 1

Output Target ?  
 Front Side\_Orange  Back Side\_Blue  
 Both

Do not color  
 Delete the shell to be processed

[Execute](#)

Specify two shells and select the shell to measure the distance with the radio button.

The difference is the area where the shells are separated from each other by the “Distance Threshold” or more.

You can detect the difference between the front side, back side or both. select which one to detect.

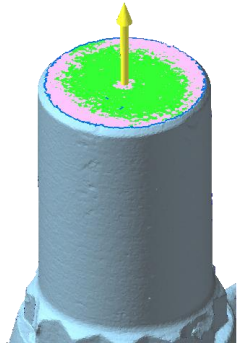
If checked, the face color of blue or orange will not be added.

If checked, the specified shells are deleted after execution.

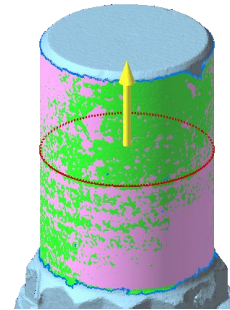
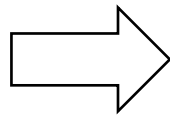
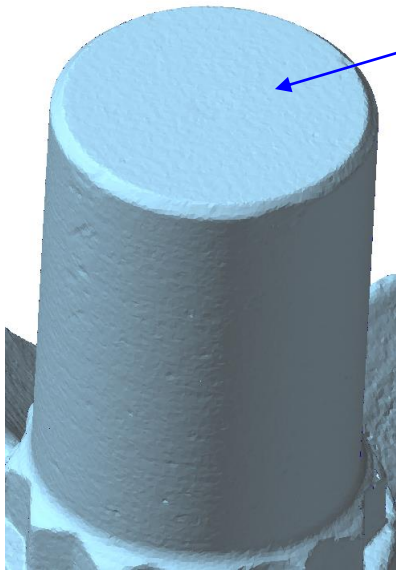
## Make Flat / Cylindrical (1)

The specified area is transformed into a flat or cylindrical shape.

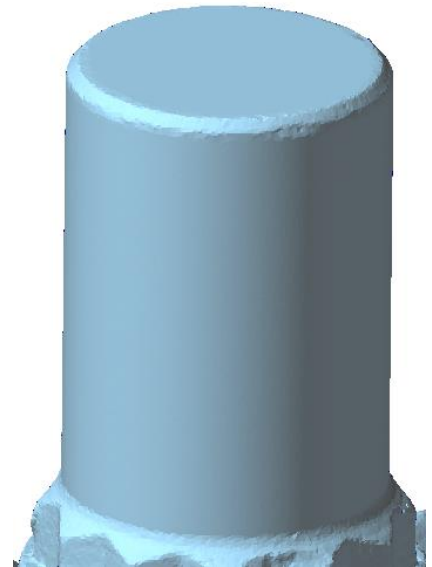
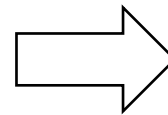
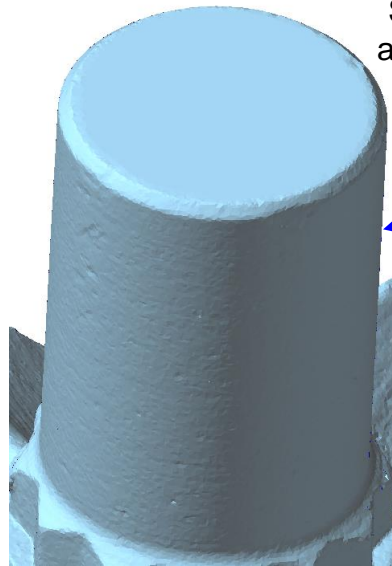
This command is effective when the partial area of the mesh derived from the measurement is shaped like a flat surface or a cylindrical surface.



Flatten the selected area



Shape the selected area like a cylindrical surface



# Make Flat / Cylindrical (2)

in the case of approximating to a plane

in the case of approximating to a cylinder

Select the area to be flattened or cylindrical.

If checked, the approximate plane or cylinder will be displayed in pink when the area is specified.

Click to change the viewing state so that the plane looks like a straight line.

It is possible to fine-tune the position of the approximate plane. "Height" is the amount by which the plane moves in the normal direction. Click the triangle icon to change the height by the amount of "Step".

Specify the normal direction of the plane.

**Make Flat / Cylindrical** How to use

Specify a target area.  
 Automatic Selection

Approximate Surface Shape ?  
 Plane  Cylinder

Remesh the specified Area ?  
 Display approximate shape

Advanced Setting

**Make Flat / Cylindrical** How to use

Specify a target area.  
 Automatic Selection

Approximate Surface Shape ?  
 Plane  Cylinder

Remesh the specified Area ?  
 Display approximate shape

Diameter

Advanced Setting

Advanced Setting

Height

Step

Plane Normal

Advanced Setting

Circle center (X, Y, Z)

Cylinder Axis

If checked, the specified area will be remeshed after being flattened or cylindricalized. It is effective when making a non-uniform faces into a cylindrical surface.

Specify the diameter of the cylinder. As the default value, the value estimated from the shape is set.

Click to change the viewing state so that you can see the cylinder from the axial direction.

Click this mark to show / hide the following input fields.

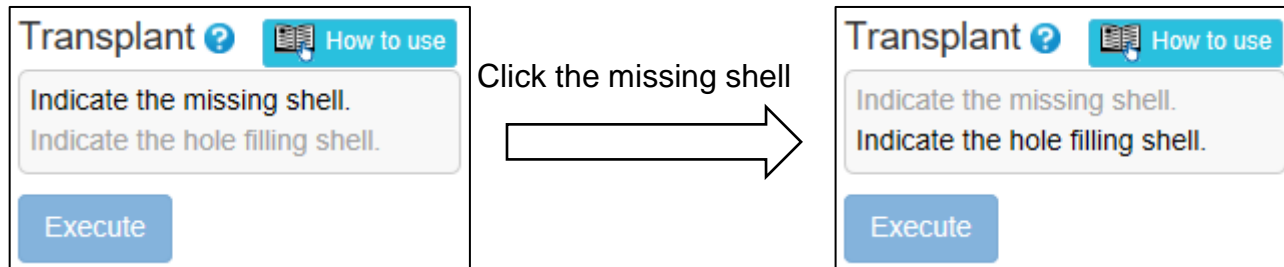
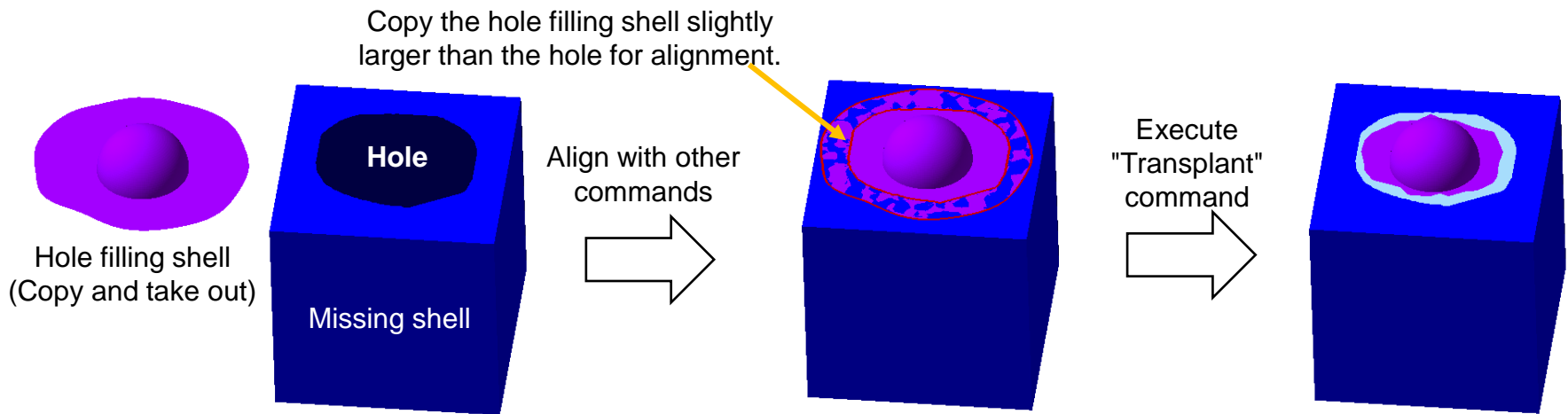
Specify the passing point and axial direction of the axis of the cylinder.

# Transplant

This command fills the holes in the shell with another shell.

This command can be used to make up for holes in the mesh due to lack of measurement data by copying other parts of the mesh.

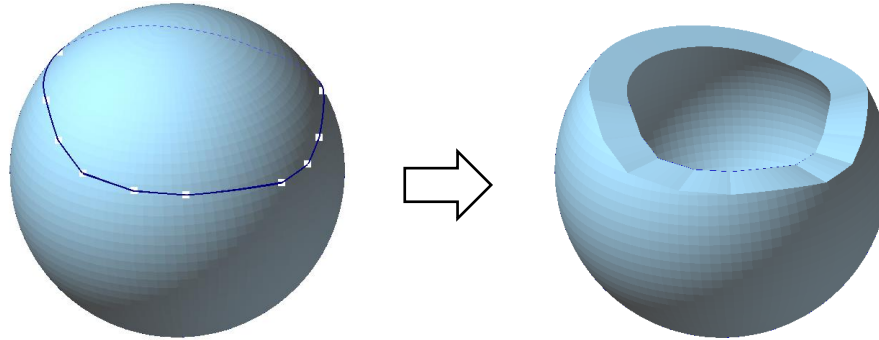
In advance, create a shell by copying the part to fill the hole (use the “Move/Copy” command etc.), and align it with the hole to be filled (use the “Arrange Shells” command etc.).



Click the “Missing shell” (the shell with the hole to fill) first, and then click on the “Hole filling shell” (the shell used to fill the hole).

## Loop Cut (1)

Draw a closed line on the mesh and use that line to cut the mesh. If there is hollow shells inside, they can be cut together.



Loop Cut [How to use](#)

**Cutting Patterns ?**

Cut Off  Split

**Cutting Line Specification ?**

Specify a detachment surface and the surviving plane.

[Reset](#)

**Cross Section Processing ?**

Filled  Unfilled

With Color

[Execute](#)

After cutting, select whether to leave only one side or both sides.

If "Split" is selected, an input field for "Clearance" will be added. Specify the width of the gap in the cut part.

Specify whether to fill or not fill the cut end of the cross section.

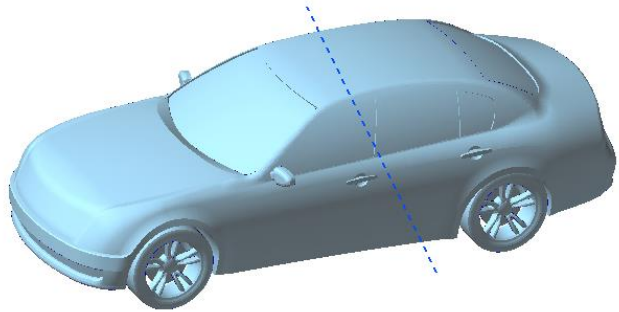
Draw a line to specify the cut surface. An operation example is explained on the next page.

Click "Reset" to return to the state before the section line was entered.

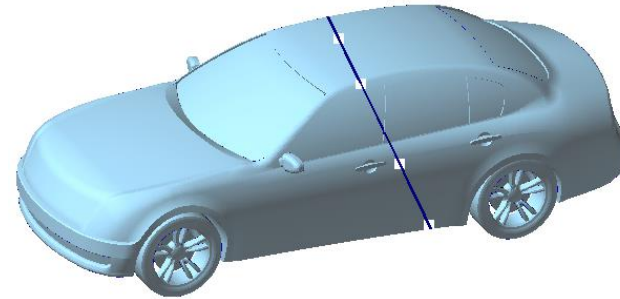
If checked, the color of the faces that fill the cut surface will be colored with reference to the surrounding colors. If unchecked, it will not be colored.

## Loop Cut (2)

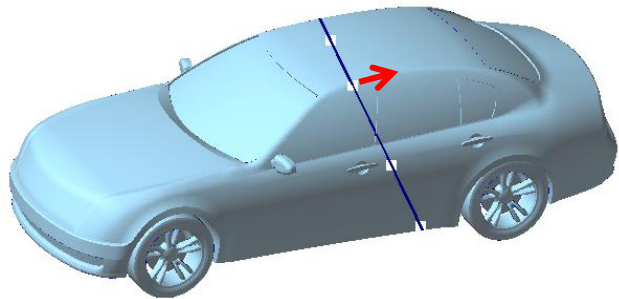
An operation example for creating a "cut surface".



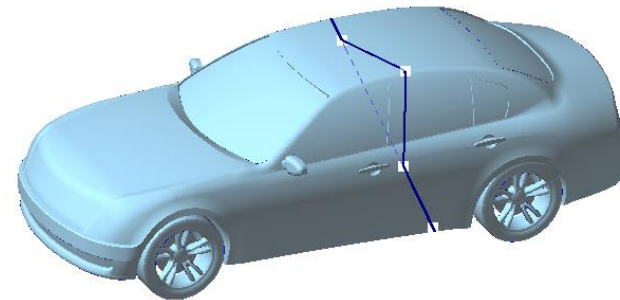
Drag the mouse to draw a straight line.



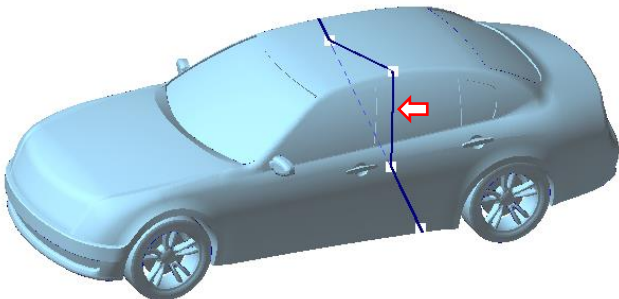
The initial Loop Polyline is created.



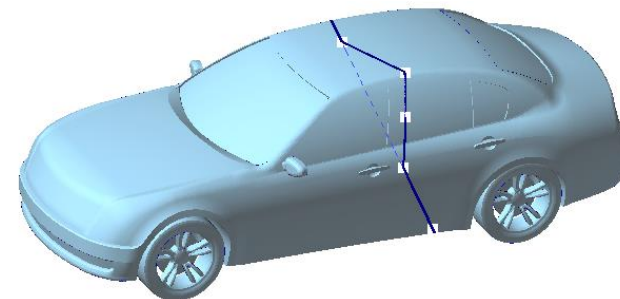
Grab a Control Point and drag it.



The Control Point is relocated to transform the Loop Polyline.



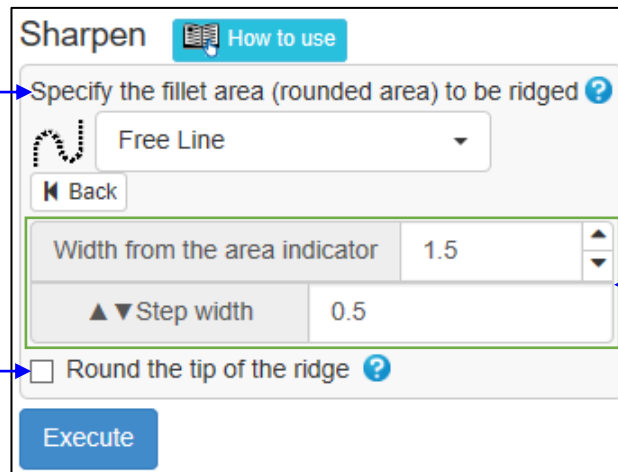
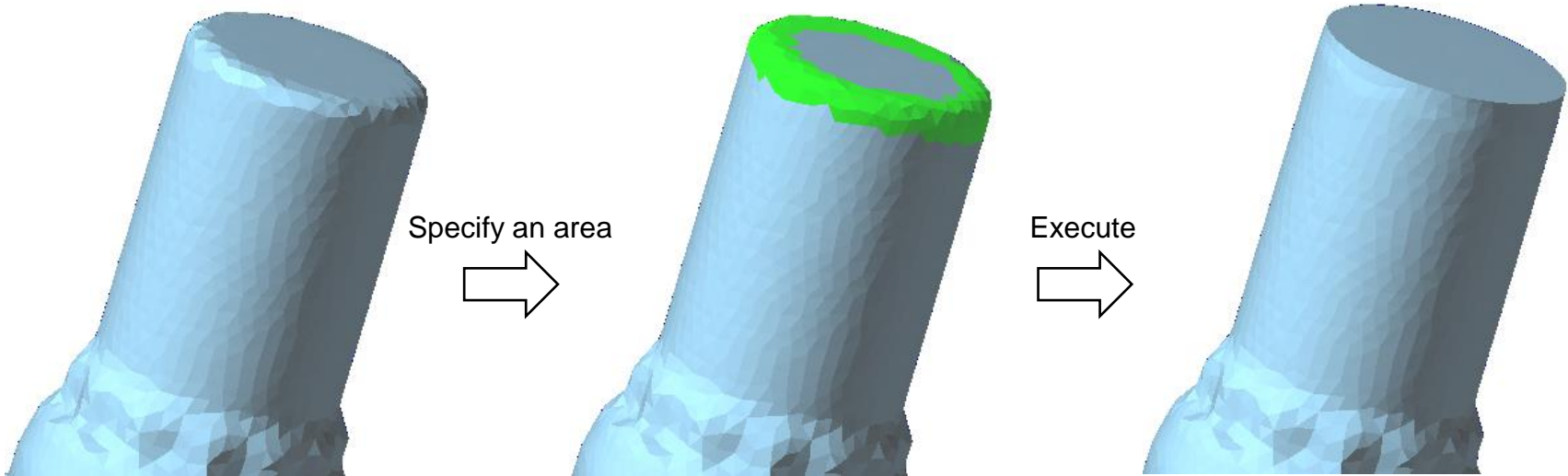
Click on the Loop Polyline.



A Control Point is added.

# Sharpen

This command makes the rounded ridge area a sharp edge.



Draw a line by dragging the center of the area to be sharpened. Faces within the "Width from the area indicator" from the drawn line are displayed in green. This area is the target of "Sharpen" process.

If checked, the ridgeline will not be sharp corners, but will be slightly rounded.

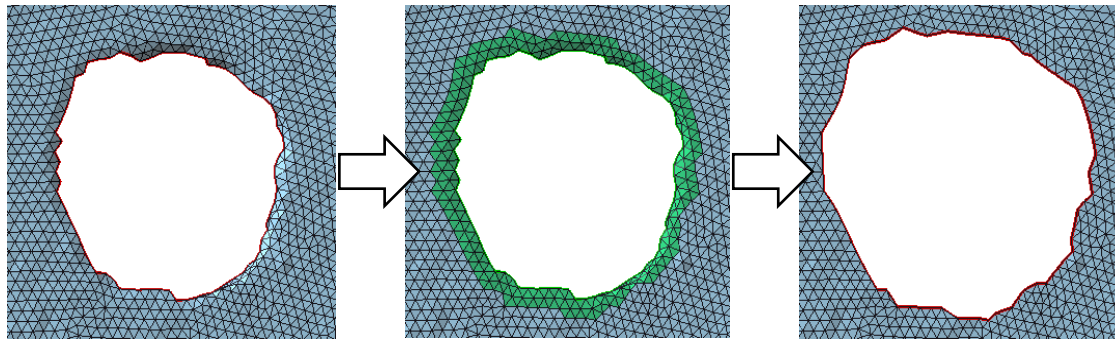
You can directly enter a value in "Width from the area indicator", or you can increase or decrease the width by the amount of "▲▼Step width" by clicking "▲" or "▼".

## Delete Boundary Faces (1)

Delete faces around shell boundaries. The range to be deleted can be specified by an amount such as the distance from the shell boundary.

Meshes derived from optical measurements can have unwanted irregularities near the shell boundary. This command is useful for deleting faces near shell boundaries like this.

Method to specify end range = [Shell Boundary] (The entire area of the specified shell boundary is targeted)

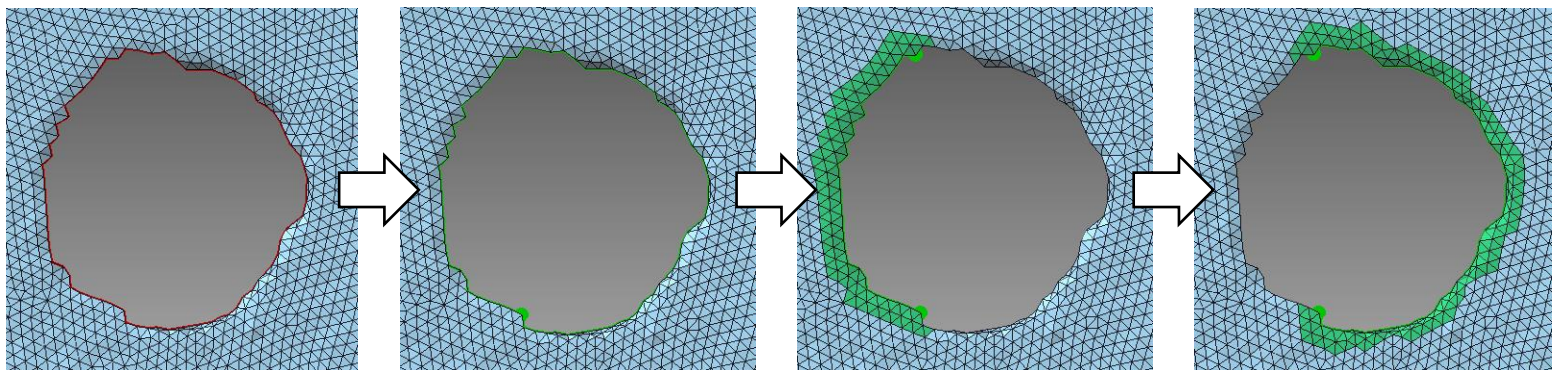


Original mesh

When a hole is specified, the range to be deleted is displayed.

After deletion

Method to specify end range = [Specify Range] (Targets the shell boundary between the two specified vertices)



Original mesh

The first vertex is clicked

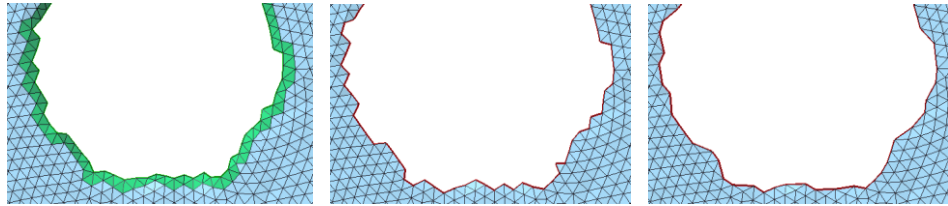
The second vertex is clicked

"Reverse" is clicked



# Delete Boundary Faces (2)

“Reduce irregularity of shell boundary” (checkbox)

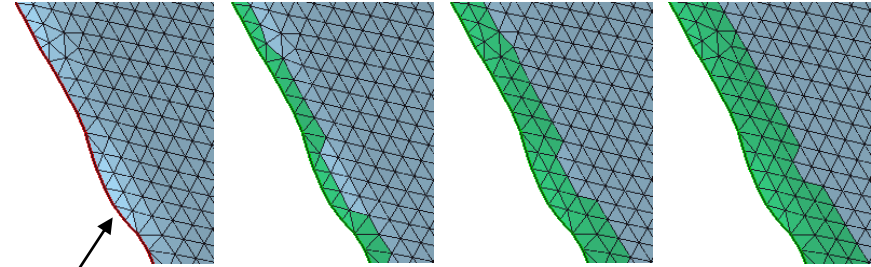


Original mesh

unchecked

checked

“Number of Edges” in “Method to specify removal range”



Shell boundary

Number of edges =1

=2

=3

Select whether to delete the entire shell boundary or between the two vertices. (See the figure on the previous page)

Click on the shell boundary to be processed. Click “Select All” button to process all shell boundaries. Click “Cancel All” button to exclude all shell boundaries.

Click to smooth the shell boundaries after deletion. (See the figure above)

**Delete Boundary Faces** How to use

Method to specify end range

Shell Boundary  Specify Range

Specify shell boundary

Select All Cancel All

Method to specify removal range

Distance  Number of Edges

Number of Edges

Preview

Reduce irregularity of shell boundary

Execute

**Delete Boundary Faces** How to use

Method to specify end range

Shell Boundary  Specify Range

Specify range by two vertices.

Reverse

Method to specify removal range

Distance  Number of Edges

Number of Edges

Preview

Reduce irregularity of shell boundary

Execute

Click on the two vertices that specify the range to be processed. (See the figure on the previous page)  
The selected vertices can be moved by dragging the mouse.

If checked, the valid range of shell boundaries will be inverted. (See the figure on the previous page)

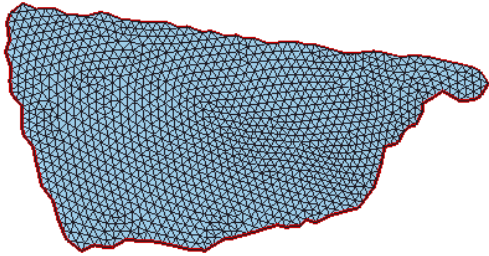
The amount to be deleted from the shell boundary can be specified by "Distance" or "Number of Edges (see the figure above)".

If checked, the face to be deleted will be displayed in green.

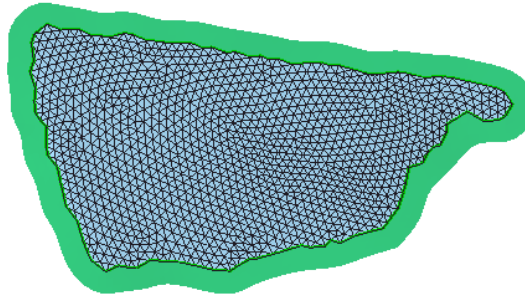
# Extension

This command extends the mesh smoothly at shell boundaries.

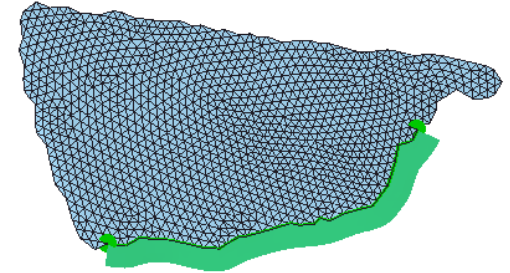
It is possible to extend the entire specified shell boundary or the specified range of shell boundaries.



Original mesh



Extend the entire shell boundary  
"Shell boundary"



Extend the specified range  
"Range indication"

Specify whether the range of the shell boundary to be extended is the entire range of the specified shell boundary ("Shell boundary") or between two vertices ("Range indication"). (See the figure above)

Click on the shell boundary to extend. Click "Select All" button to process all shell boundaries, and click "Cancel All" button to exclude all shell boundaries from processing.

The dialog box is titled "Extension" and includes a "How to use" icon. It has two radio buttons under "Instruction method of Extension range": "Shell boundaries" (selected) and "Range indication". Below this is a section "Indicate shell boundaries." with "Select All" and "Cancel All" buttons. There is an "Extension Amount" field set to "5.0" and a checked "Preview" checkbox. An "Execute" button is at the bottom.

The dialog box is titled "Extension" and includes a "How to use" icon. It has two radio buttons under "Instruction method of Extension range": "Shell boundaries" and "Range indication" (selected). Below this is a section "Specify the processing range at two vertices." with an unchecked "Inversion" checkbox. There is an "Extension Amount" field set to "5.0" and a checked "Preview" checkbox. An "Execute" button is at the bottom.

Click on the two vertices that specify the range you want to process. You can move the clicked vertex by dragging.

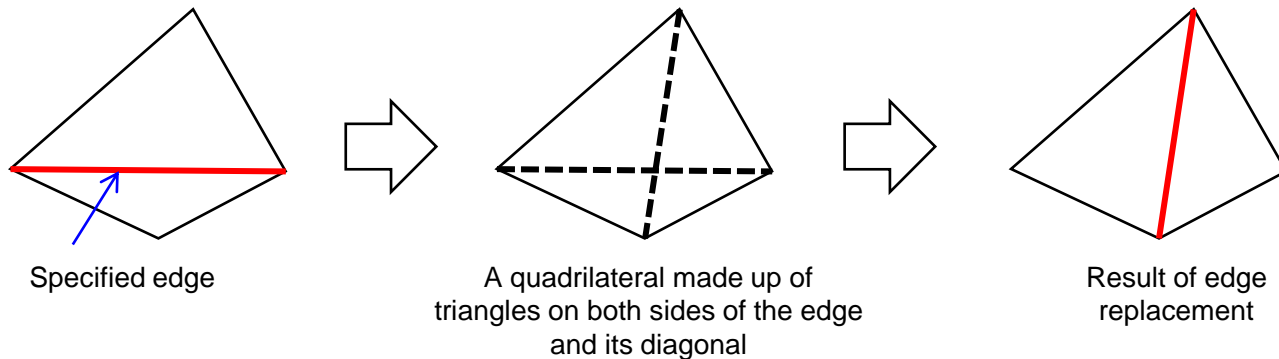
If checked, the valid processing range is inverted.

specify the length to extend.

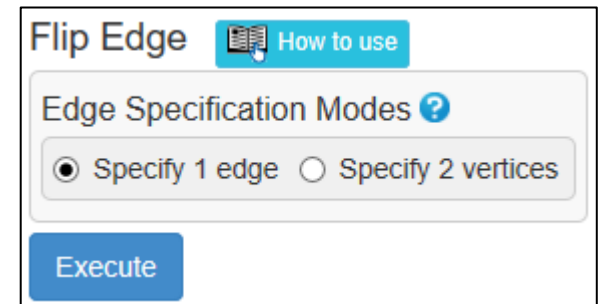
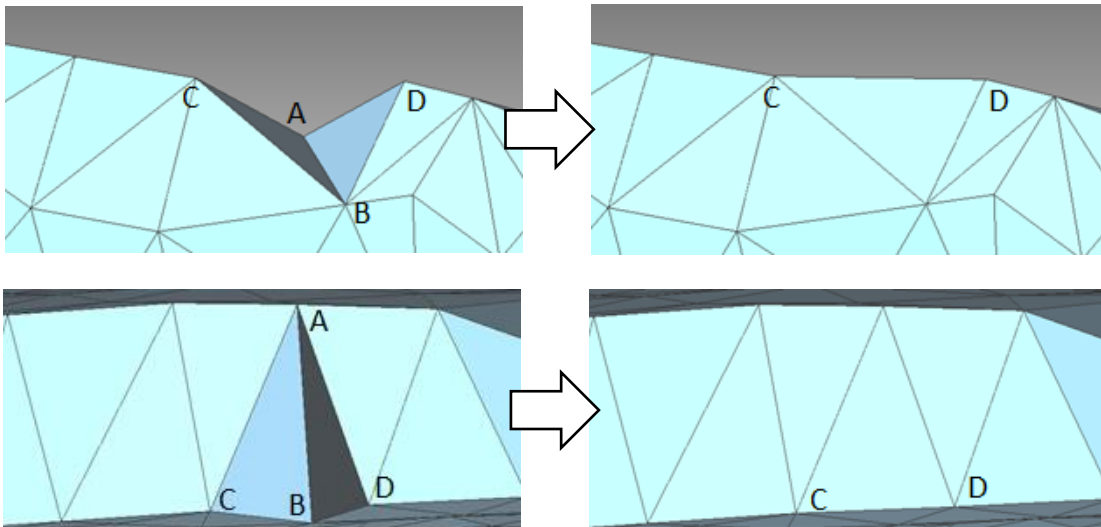
If checked, the extended face will be displayed in green before execution. (See the figure above)

# Flip Edge

Imagine a quadrilateral made up of triangles on either side of the specified edge. This quadrilateral has two diagonals, one of which is the specified edge. Deletes the specified edge and makes the other diagonal an edge.



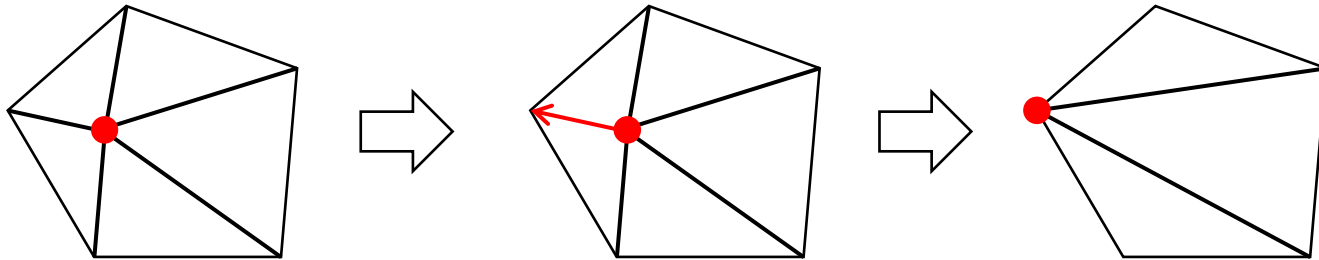
Example of replacing the edge connecting vertices A and B with the edge connecting vertices C and D



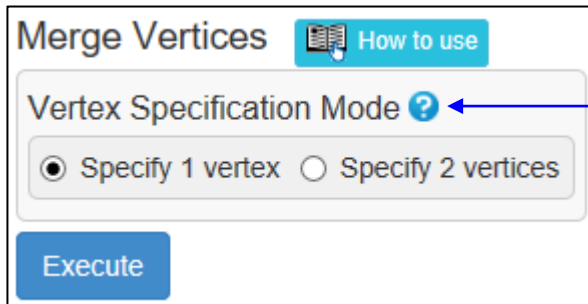
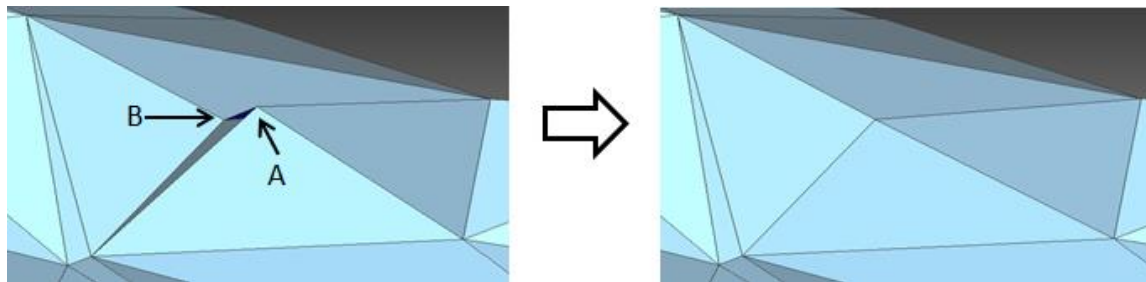
Click an edge to be replaced.  
If it is difficult to click the edge because it is behind faces, you can select "Specify 2 vertices" to specify the two vertices at both ends of the edge.

## Merge Vertices

This command moves the specified vertex to the position of one of the vertices adjacent to the vertex and combines them into a single vertex.



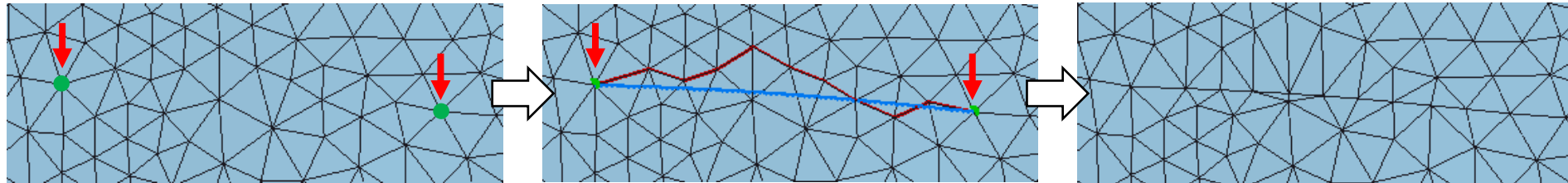
An example of moving vertex A to vertex B and making it one vertex.



If “Specify 1 vertex” is selected, the clicked vertex will be moved to the position of the nearest vertex among the vertices adjacent to that vertex, and will be combined into one vertex.  
If “Specify 2 vertices”, the first clicked vertex will be moved to the position of the next clicked vertex and combined into one vertex. However, the two vertices have to be adjacent.

## Align Vertices

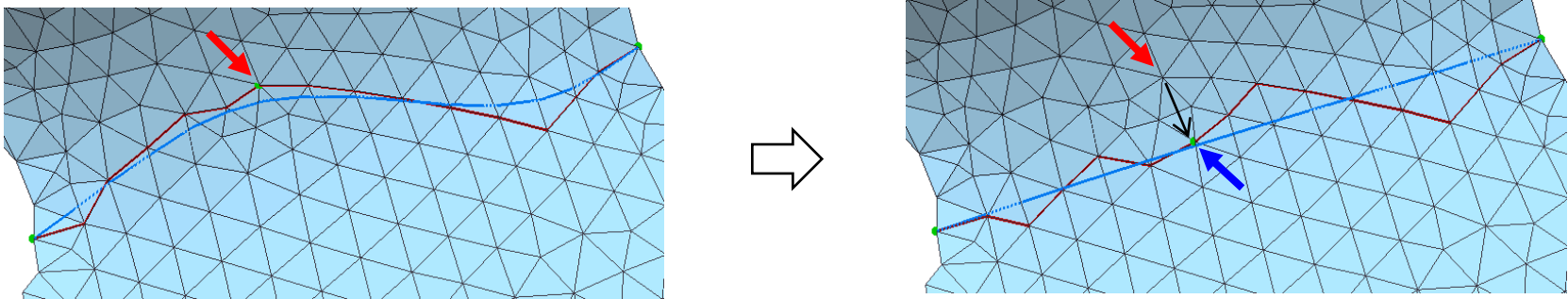
Adjust the position of the vertices so that the edge line connecting the specified two vertices connects smoothly.



Click on two vertices. The clicked vertex (the tip of the red arrow on the left in the above figure) is displayed in green, the edge line connecting the two vertices is displayed in red, and the position of the edge line after alignment processing is displayed in blue.

Click Execute to move the vertices to the position of the blue line.

The path connecting the specified two vertices (red edge line) can be changed as follows.

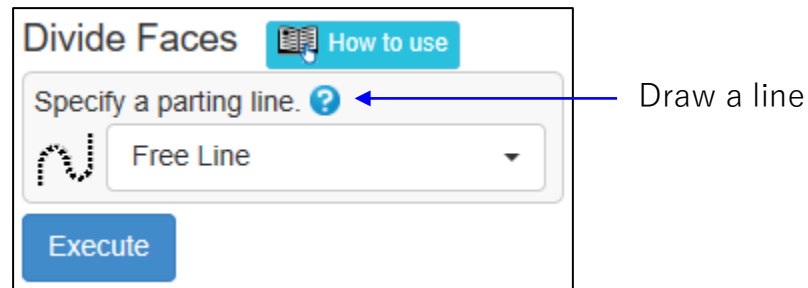
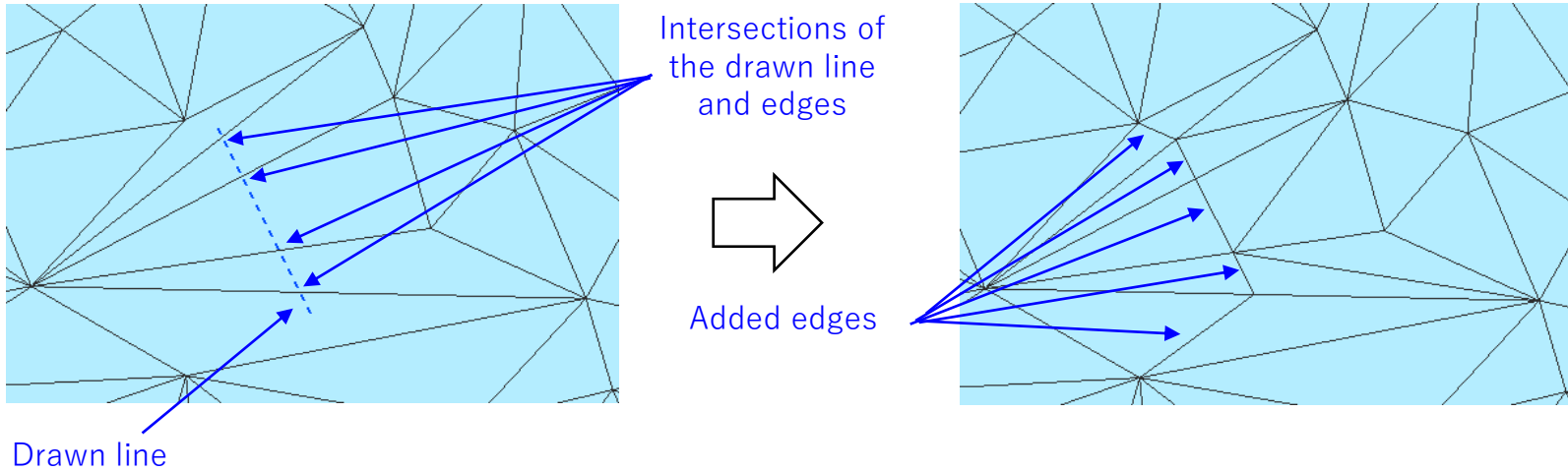


Click on the vertex at the position you want to change on the red edge line. The vertices (the tip of the red arrow) are displayed in green.

You can change the red edge line by dragging the green vertex (from the position of the red arrow to the position of the vertex of the blue arrow). The blue edge line will change accordingly.

## Divide Faces

This command divides faces at the intersections of the drawn line and the edges.  
(Use the "Subdivide" command if you want to divide the surface in consideration of the bulge.)



## 9. Evaluation Tab

# Property

This command displays the properties of the entire mesh or subregion, and display a list of properties for each shell and output a CSV file. In the figure display area, the box containing the mesh and the center of gravity are temporarily displayed in red.

The center of gravity of an open shell is the center of gravity of the faces that compose the shell.

Properties of the entire mesh

Properties of a Subregion

Property list for each shell

Properties of the selected shell

Property [How to use](#)

Calculation subject ?

Entirety  Part  Shellwise

Item Name	Value
Volume	27941.2
Surface Area	13556.4
Center of Gravity ●	X: -0.493392
	Y: 0.199535
	Z: 12.7567
Size	X: 59.743
	Y: 59.9232
	Z: 56.459
Shell Qty	13579
Shell Boundary Qty	0
Average Edge Length	0.153373
Maximum Edge Length	0.826619
Minimum Edge Length	9.83869e-005

Property [How to use](#)

Calculation subject ?

Entirety  Part  Shellwise

Specify region to compute properties ?

Shell

- Free-form
- Rectangular
- Shell
- Color

Volume	X: 59.743
Surface Area	Y: 59.9232
Center of Gravity ●	Z: 56.459
Size	X: 59.743
	Y: 59.9232
	Z: 56.459
Shell Qty	1
Shell Boundary Qty	0
Average Edge Length	0.172831
Maximum Edge Length	0.826619
Minimum Edge Length	9.94043e-005

Specify the area to calculate the properties.

Check the shell that calculates the properties. Click "Details of Selected Shells" to calculate the properties of the selected shell totals.

Property [How to use](#)

Calculation subject ?

Entirety  Part  Shellwise

Shell List Details of Selected Shells

	No. of Faces	Volume	Surface Area	
<input checked="" type="checkbox"/>	1039418	27961.7	12544.8	0
<input checked="" type="checkbox"/>	590	-0.139459	2.64783	0
<input checked="" type="checkbox"/>	366	-0.0712289	1.8093	0
<input type="checkbox"/>	326	-0.0603872	1.40256	0
<input type="checkbox"/>	324	-0.113037	1.77785	0
<input type="checkbox"/>	274	-0.0410314	1.02945	0
<input type="checkbox"/>	226	-0.0477067	1.18076	0
<input type="checkbox"/>	220	-0.0376599	0.855696	0
<input type="checkbox"/>	198	-0.0179191	0.778868	0
<input type="checkbox"/>	168	-0.0250017	0.679094	0
<input type="checkbox"/>	162	-0.0615835	1.00745	0
<input type="checkbox"/>	158	-0.0159039	0.529491	0
<input type="checkbox"/>	156	-0.0156503	0.557968	0
<input type="checkbox"/>	150	-0.0482379	0.901265	0
<input type="checkbox"/>	150	-0.0208409	0.567364	0

CSV Output

Click "CSV Output" button to output a list of properties for all shells to a CSV file.

Property [How to use](#)

Calculation subject ?

Entirety  Part  Shellwise

Shell List Details of Selected Shells

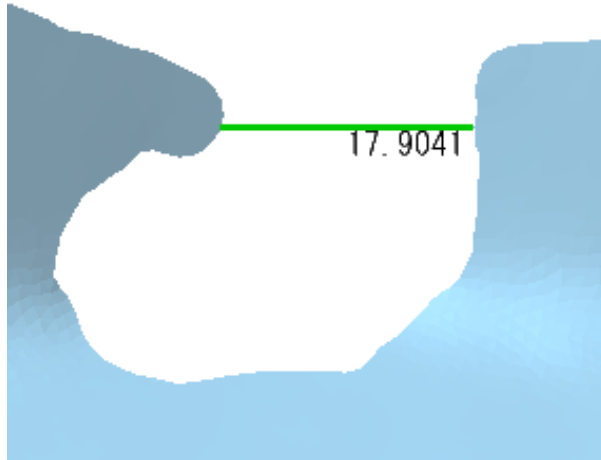
Item Name	Value
Volume	27961.4
Surface Area	12549.2
Center of Gravity ●	X: -0.493379
	Y: 0.199552
	Z: 12.7537
Size	X: 59.743
	Y: 59.9232
	Z: 56.459
Shell Qty	3
Shell Boundary Qty	0
Average Edge Length	0.172774
Maximum Edge Length	0.826619
Minimum Edge Length	9.94043e-005



# Distance

Measure the distance between the two vertices.

It is also possible to find the distance of the polygonal line by connecting the clicked vertices in order.



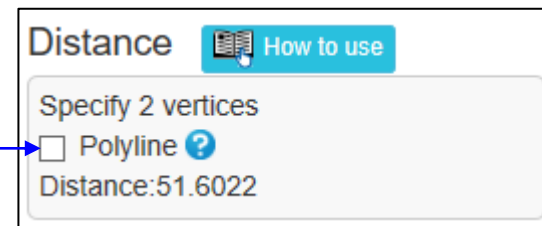
Distance between two vertices



Length of polygonal line (each line segment)

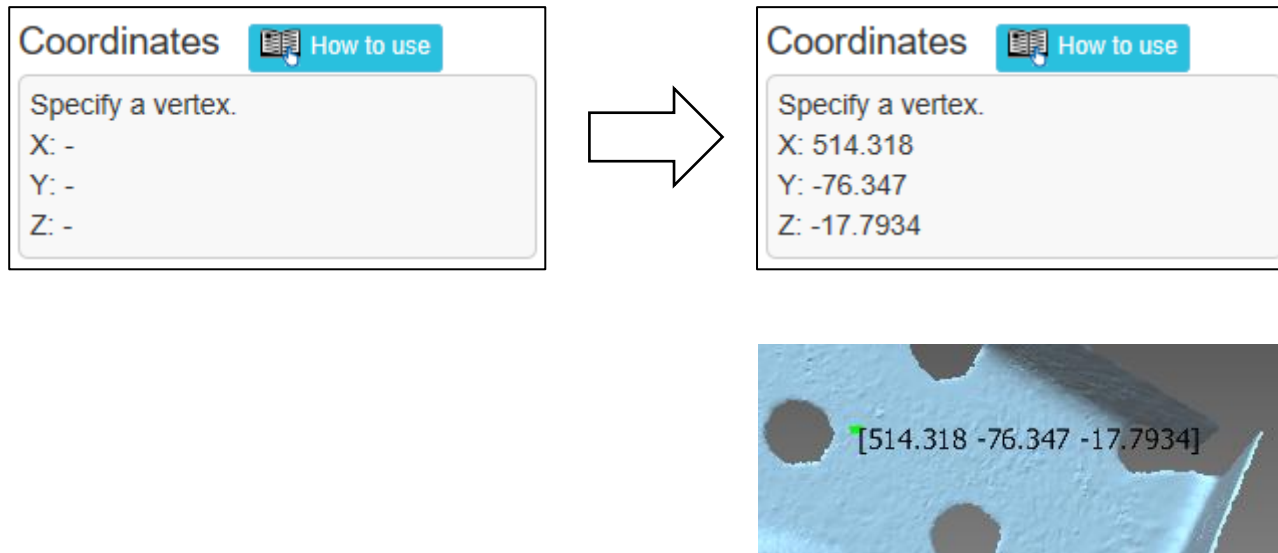
If unchecked, the distance between the two vertices clicked will be displayed in both the figure display area and the command dialog.

When checked, the length of each line segment connecting the clicked vertices is displayed in the figure display area, and the total value of the lengths (polygonal line length) is displayed in the command dialog.



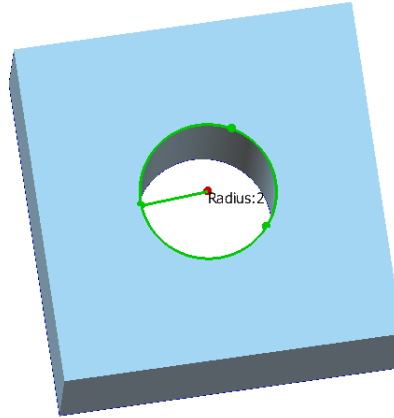
# Coordinates


Displays the coordinate values of the specified vertices in the command dialog and the figure display area .



## Circle radius

The radius, diameter, and center coordinate values of the circle passing through the specified three vertices are calculated and displayed in the command dialog and the figure display area.



**Circle Radius**  How to use

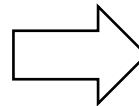
Specify 3 vertices on the circle to measure.


Radius :

Diameter:

Center :

**Clear**



**Circle Radius**  How to use

Specify 3 vertices on the circle to measure.

Radius :2

Diameter:4

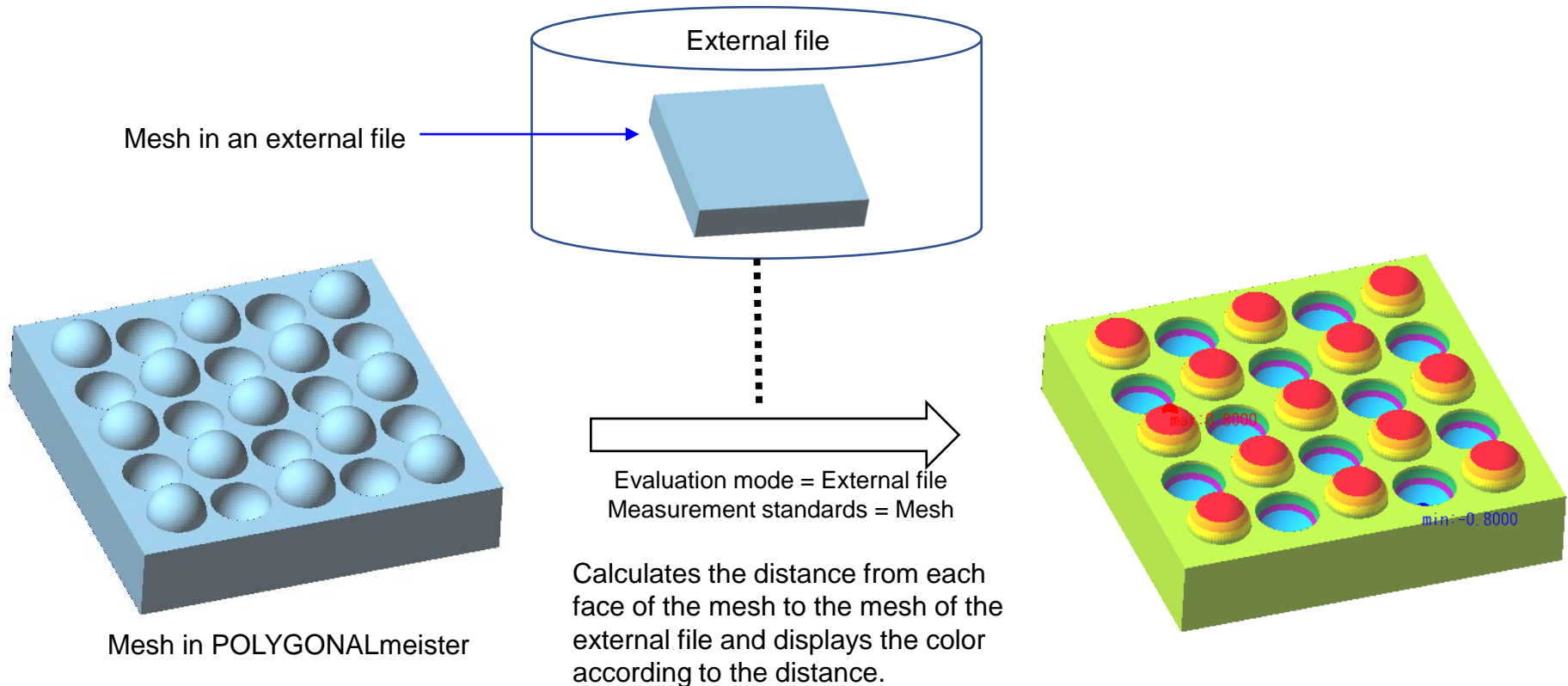
Center :6.66134e-016 -5.13478e-015 2.5

**Clear**

## Distance Contour (1)

The difference between the following two shapes is filled in with a color according to the degree of separation.

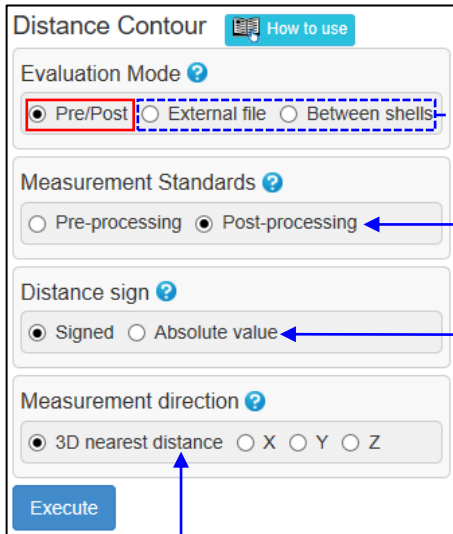
- Difference between the mesh before processing the command and the mesh after processing
- Difference between mesh and mesh of external file
- Difference between two shells



# Distance Contour (2)

[Pre-execution parameter settings]

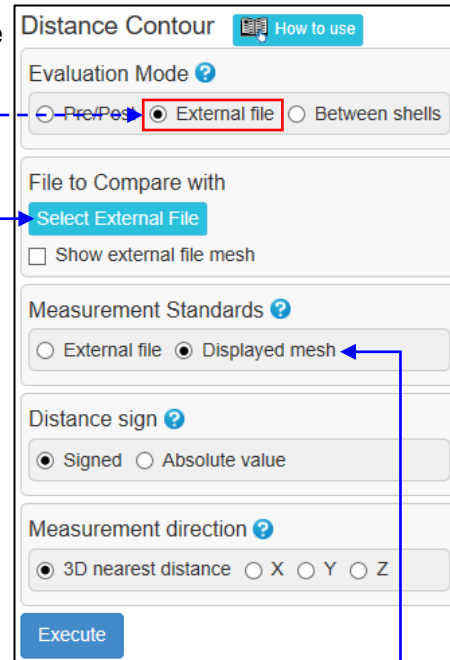
## Distance of mesh before/after processing



Click the button and select the external file.

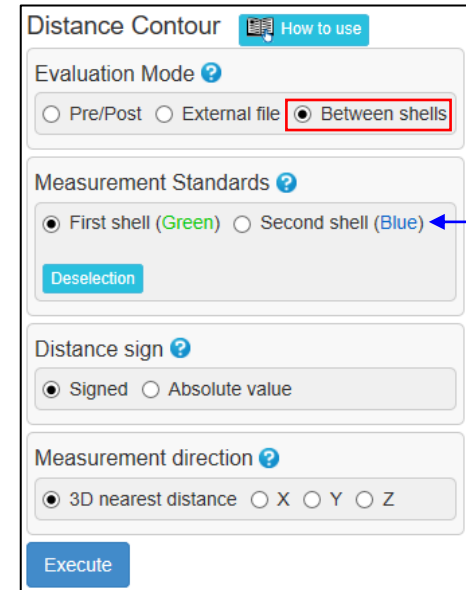
Select whether the mesh displayed after calculating the distance is "Pre-processing" (before execution) or "Post-processing" (current state).

## Distance between mesh and external file



Select whether the mesh displayed after calculating the distance is the mesh in "External file" or "Displayed mesh".

## Distance between 2 shells



The shell you select first is called the "first shell" and the shell you select next is called the "second shell". Select which shell to calculate the distance from and display the color.

"3D nearest distance" = the latest distance from the point on the measurement standard to the other figure.

"X" = Find the intersection of a straight line that passes through a point on the measurement standard and is parallel to the X axis and the other figure. The distance from the point on the metric to the nearest intersection. "Y" and "Z" are the same as "X".

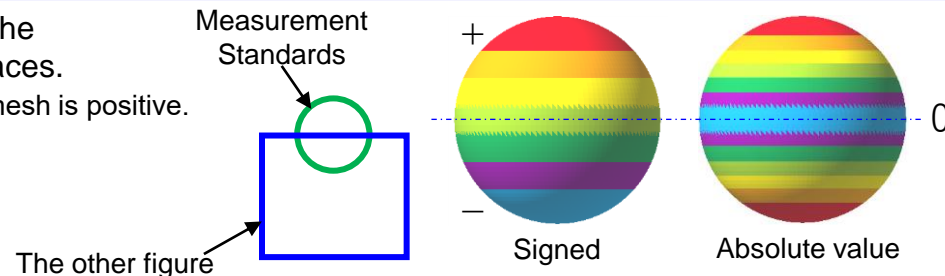
"Signed" = Distance is shown as follows to distinguish between the distance on the front side and the distance on the back side of faces.

"Pre/Post-processing" : The front direction of "Pre/Post-processing" mesh is positive.

"External file": The front side of the other is plus.

"Between shells": The front side of the other shell is plus.

"Absolute value" = The distance is shown as a positive value without distinguishing between the front and back.



# Distance Contour (3)

[Post-execution parameter settings]

Click to change the color.

Click this icon to turn the color of the color bar upside down.

Shows the correspondence between the distance range and the colors painted on the colormap.

You can change the value of "Lower limit". Click the "Apply to Contour" button to recreate the colormap with the changed values.

"Ratio" is the ratio of the sum of the surface areas of the faces belonging to the distance area to the surface area of the measurement standard.

If checked, a red mark is displayed on the color map for the vertex with the maximum value, and a blue mark is displayed for the vertices with the minimum value.

Click the mesh to see the distance at the clicked position (not the face distance). Click the "Clear distance display" button to turn off the distance display.

The maximum, minimum, mean, standard deviation of the distance at each vertex of the metric and the number of vertices for which the distance was calculated are displayed.

Lower Limit	Upper Limit	Ratio(%)
0.65	-	0.53
0.39	0.65	2.84
0.13	0.39	2.28
-0.13	0.13	86.61
-0.39	-0.13	2.51
-0.65	-0.39	2.54
-	-0.65	2.70

Maximum distance	0.7
Minimum distance	-0.9
Number of measurement points	17491
Average distance	-0.0796753
Standard deviation	0.498088

"Apply to contour" = Click to recreate the colormap with the values and colors you set for the distance range.

"Intermediate threshold automatic setting" = Click to distribute the middle thresholds at equal intervals based on the top and bottom thresholds.

"Set the previous lower limit" = Click to change the range of the interval so that it is the same as the threshold used last time.

"CSV output" = Click to output a CSV file in which the coordinate values and distances are written for each vertex. (x, y, z, distance of one vertex per line)

"Set face color" = Click to save the color map as a face color. This button does not appear if the "Measurement Standards" are "Pre-processing" or "External File".

# Display Section Lines

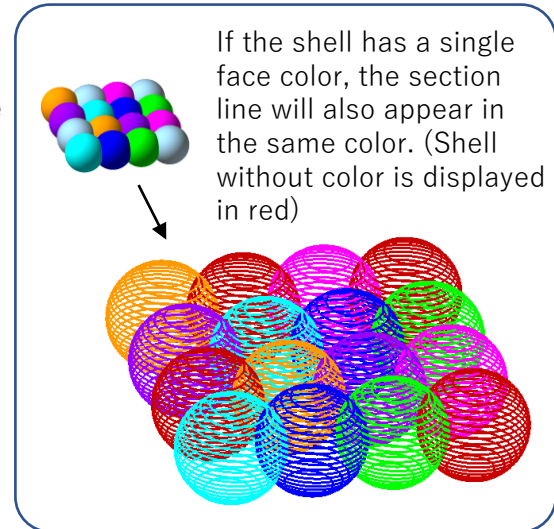
This command shows the line of intersection between the mesh and planes. There are two ways to set the planes.

(1) Arbitrary section: A plane that includes a straight line to be dragged and the line-of-sight direction (direction perpendicular to the screen).

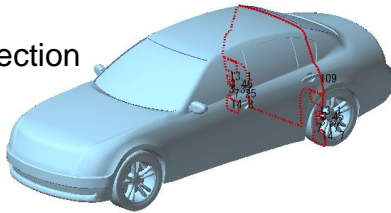
(2) Equidistant section: Equally spaced planes perpendicular to the specified coordinate axis.

In case of (1), the length of the cross section can be displayed.

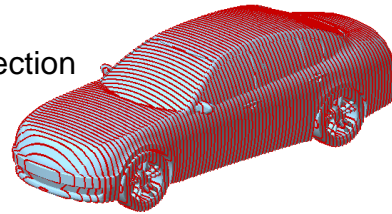
You can export the section line as polyline to an IGES file.



(1) Arbitrary section



(2) Equidistant section

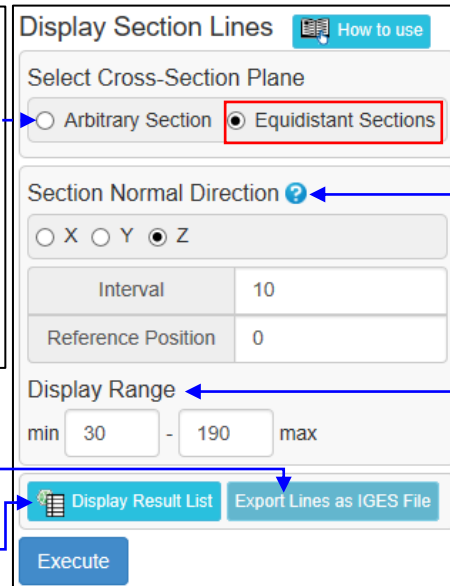
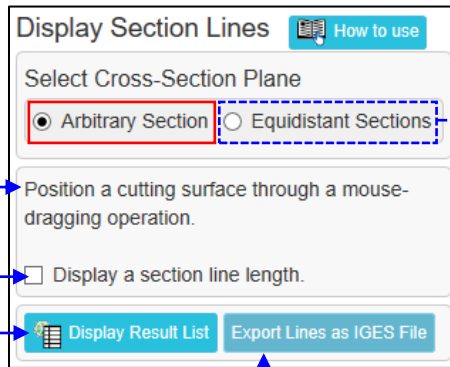


Draw a straight line while holding down the left mouse button at the position where you want the section line.

If checked, the length of the section line will be displayed.

Click "Export Lines as IGES File" button to output the section line as a polygonal line to an IGES file.

Click the "Display Result List" button to save the temporary figure of the section line and open the result list window.

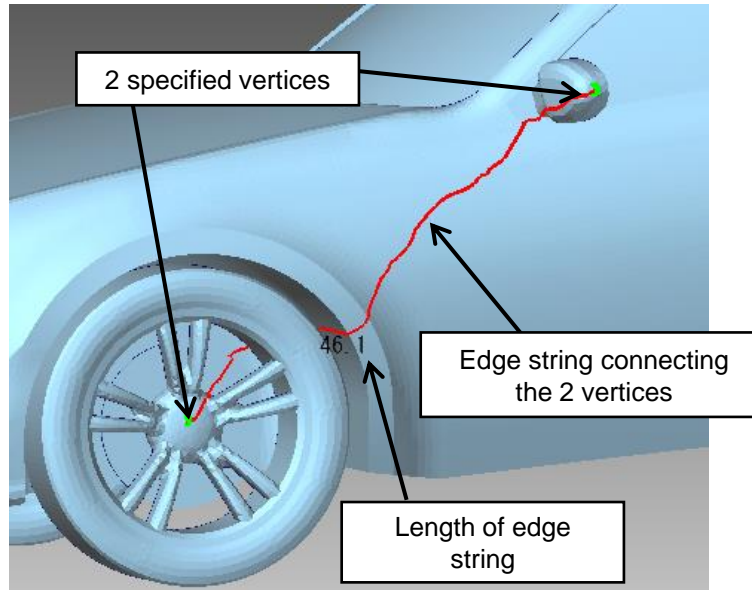


Shows the line of intersection between the mesh and the evenly spaced planes perpendicular to the axis. Select the axis and specify the plane interval and reference position. Planes are created at a position that is a multiple of the "Interval" from the position of the coordinate value of the "Reference Position".

The range for displaying the section line. Enter the coordinate values for "Section Normal Direction".

# Find Path

This command displays the path (edge string) that connects the two specified vertices by tracing edges. There are many routes connecting two vertices, but the route with the shortest route length (edge string length) is displayed. The route length can be displayed.



Specify the two vertices to route. →

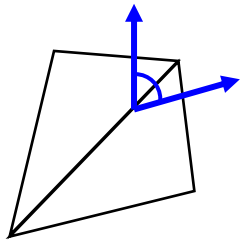
If checked, the length of the route will be displayed. →



# Crease Detection

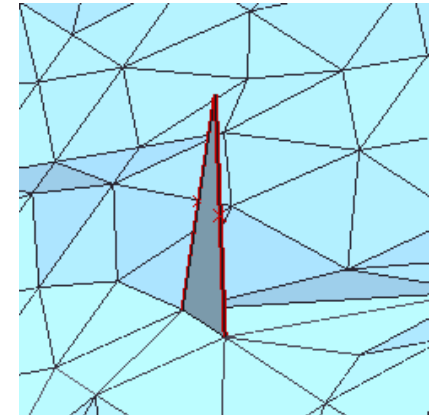
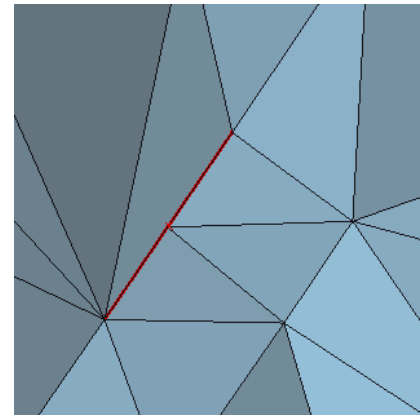
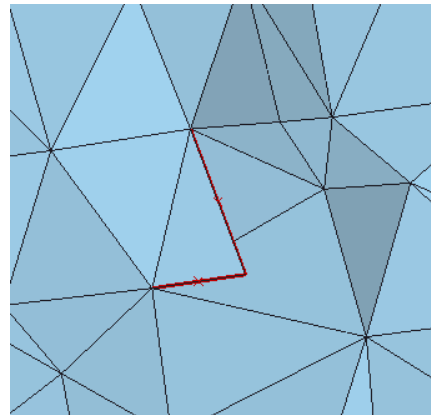
This command detects bent edges and displays them in red. Also, the number of detected edges is displayed in the command dialog.

It detects edges where the angle between the ostensibly normal vectors of both faces is greater than or equal to the specified value.



The angle between the two normal vectors

[ Detection examples ]



Detects edges where the faces on both sides are bent by this angle or more.

**Crease Detection** [How to use](#)

Angle deemed to create a crease ?  
0.0 <  <= 180.0

Detection Edge Qty :-  
[Display Result List](#)

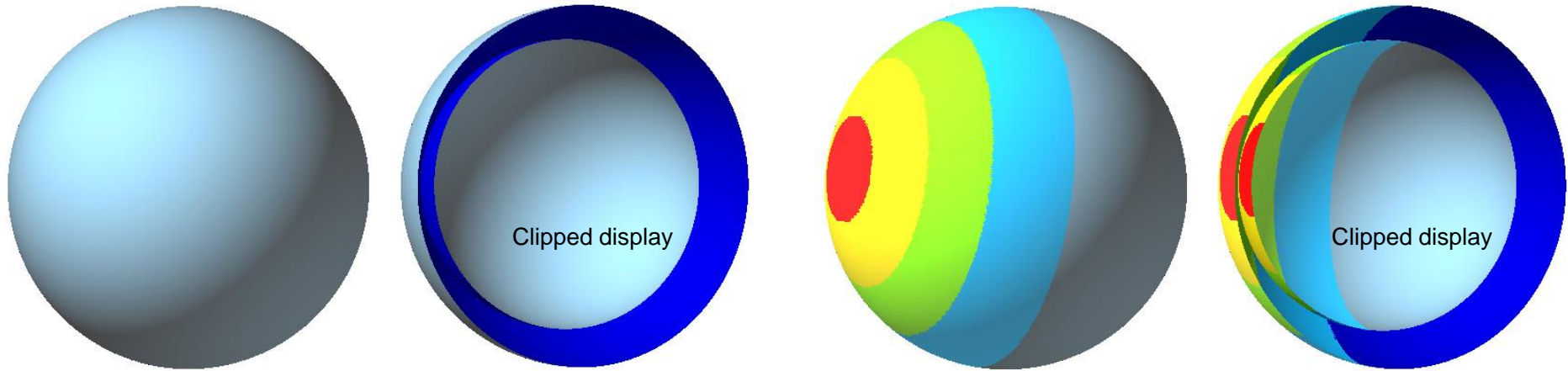
**Detect**

Shows the number of detected edges.

Click the “Display Result List” button to save the temporary figure of the detected edges and open the result list window.

# Thickness Inspection (1)

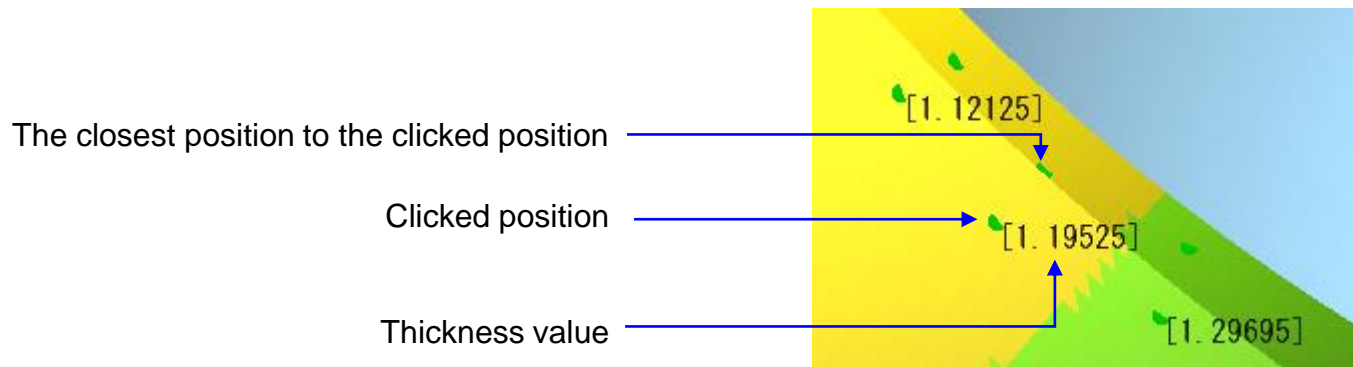
This command detects where the mesh thickness is less than or equal to the specified value. The detected part is painted with colors according to the thickness of 4 steps (displays a color map). The threshold of the color to be painted can be changed after the color map is displayed.



This is an example of a mesh with a hollow.

Detects thin areas and paints with a color according to the thickness.

If you click on the detected area, the thickness value of the clicked position will be displayed.

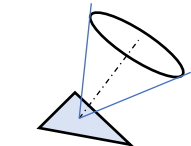


# Thickness Inspection (2)

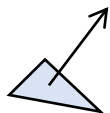
There are two ways to find the distance between one face and the others. With the center of gravity of the face as the end point, make a half straight line perpendicular to the face toward the back side of the face.

"Cone method" = Find the face group located inside the cone with the specified value as the apex angle, and calculate the recent distance between the center of gravity of the face and the face group.

"Ray method" = Find the group of intersections between a half line and another face, and calculate the recent distance between the center of gravity of the face and the group of intersections.



Cone Method



Ray Method

When you click on the mesh in the area where the color map is displayed, the thickness of the clicked position (not the thickness of the face) is displayed, and a green mark is displayed at the clicked position and its nearest point. Click the "Clear Thickness Display" button to remove the thickness and mark display.

**Thickness Inspection** [How to use](#)

Range to measure thickness ?

Method ?  Cone  Ray

Cone Apex Angle ?

**Execute**

Only the part thinner than the specified value is detected.



After execution, the area shown below will be displayed below the "Execute" button.

**Inspection Result**

Lower Limit	Upper Limit	Surface Area
<input type="text" value="3.75"/>	- 5	534.124
<input type="text" value="2.5"/>	- 3.75	502.683
<input type="text" value="1.25"/>	- 2.5	495.768
0	- 1.25	140.684

**Apply to Contour** **Set face color** **Display Result List** ?

In the colored area, the thickness is calculated and displayed at the clicked position.

**Clear Thickness Display**

Shows the correspondence between thickness and color. The values of "Lower limit" can be changed. Click "Apply to Contour" button to recreate the color map with the changed values. In addition, the sum of the surface areas of the corresponding faces is displayed at the right end of the table.

Click the "Display Result List" button to save the temporary figure of the color map and open the result list window.

Click the "Set face color" button to save the displayed contour as a face color.

# Unevenness Test (1)

The command approximates a finely-uneven plane or free-form surface (composed of meshes and point clouds) as a smooth plane or free-form surface.

The uneven surface is referred to as a test subject.

The command uses the color distributions below that indicate degrees of fine unevenness of a test subject in accordance with distances between the test subject and an approximate surface shape.

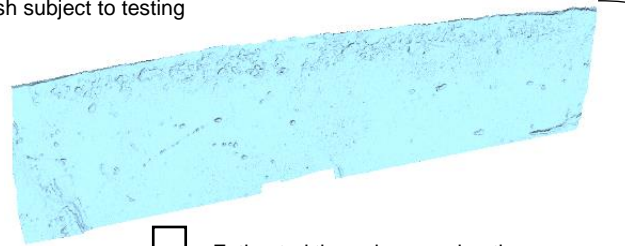
- Color Map (where individual faces are colored in accordance with distances)
- Heat Map (where each cell in a 2D grid composed of x and y coordinates is assigned with a color corresponding to the distance)
- Cloud of colored points (where each cell is assigned with its corresponding color points)

Distances of positive value indicate convex positions.

Distances of negative value indicate concave positions.

The command enables users to know a distance by indicating a plane at a fixed value of z axis, rather than estimating an approximate surface shape.

Mesh subject to testing

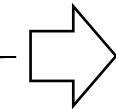


Estimated through approximation

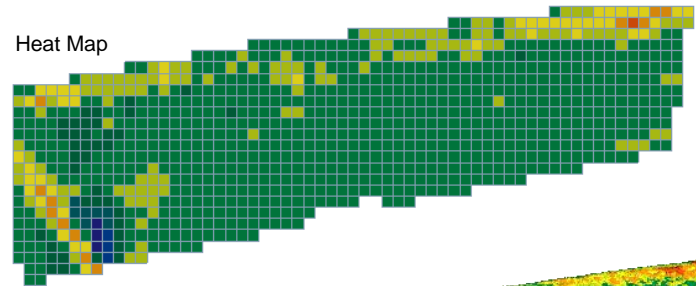
Estimated mesh



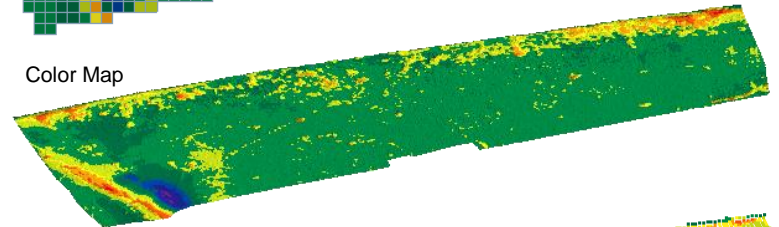
Differences



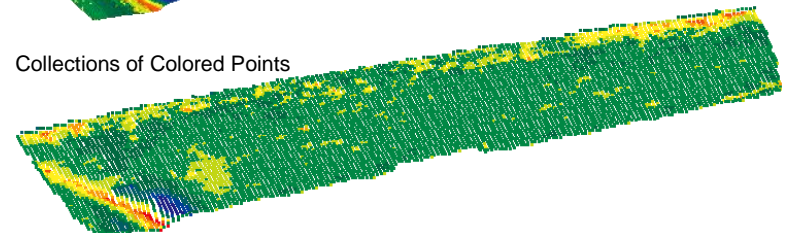
Heat Map



Color Map



Collections of Colored Points



# Unevenness Test (2)

Specify the type of shape subject to testing, '(imported) Mesh' or 'Point Clouds'.

Choose 'Point Clouds' in order to show the field below. Click the button 'Choose File' in order to specify a file subject to testing.

Tick 'Display Data of Point Clouds' in order to display imported point clouds.

If you choose 'Mesh', further choose the 'Entirety' or 'Part' of mesh. If you choose 'Part', please specify an area subject to testing.

The system creates an evaluation point in a cell and estimates a standard shape at the evaluation point. The system calculates the distance between the standard shape and the test-subject shape. The system uses the distance in order to calculate a Cell Distance (distance between a cell and the standard shape), as indicated below. It assigns a color to each cell depending upon the Cell Distance.

Tick 'Average in the Cell' in order to use the distance average as the Cell Distance.

Tick 'Maximum in the Cell' in order to use the maximum absolute value of distance as the Cell Distance.

Tick 'Minimum in the Cell' in order to use the minimum absolute value of distance as the Cell Distance.

Tick 'Left Upper Corner of the Cell' in order to use a distance at the left upper corner of a cell (X: min., Y: max.) as the Cell Distance.

Click the button 'Enter' in order to begin computation. A specified mode of color distribution appears after a computation is finished. The field (on the next page) appears under the button 'Enter'.

Choose either 'Estimate' or 'Specify' in the section of a standard shape (shape without unevenness on the surface). 'Estimate' means to use approximation. 'Specify' means to specify a position of plane.

If you choose 'Estimate', further choose a 'Plane' or 'Free-form Surface' as the estimation subject.

If you choose 'Specify', the system enables a standard shape as a plane that has a fixed z value. Specify a z value in the field below that appears.

Tick in order to prevent uneven portions from meddling with a standard shape estimation.


Tick in order to specify x coordinate value and y coordinate value that indicate a Grid Passing Point (a point at a corner of the cell).

Specify a cell width (Cell Size) of Heat Map.

If 'Assign 0 to the center of color distribution' is unticked, the system equally divides a Cell Distance (from the Value Max. to the Value Min.) and assigns a color to each segment.

Tick the option in order to equally divide a distance between a Cell Distance (an absolute value Max. or an absolute value Min. whichever is the larger) and the negative absolute value commensurate with the positive absolute value. The center of the distance is set to be zero.

# Unevenness Test (3)

Click  in order to show the figure below.

This shows average value, maximum value, minimum value, and standard deviation of evaluation point, as well as colored cell quantity.

Basic Statistics Q'ty	
Average Distance	-0.0481345
Max Distance	0.184739
Min Distance	-0.545161
Standard Deviation	0.0706502
Number of Cells	149

The scale indicates Cell Distances corresponding to cell colors.

Cells are assigned with colors corresponding to a range segment of Cell Distances.

Percentages of cells belonging to each range segment are indicated at the right edge.

Faces that constitute a Color Map are assigned with colors corresponding to how far they are positioned from the standard shape.

The system accepts revisions in the maximum threshold value and the minimum threshold value. Threshold values in-between are automatically equally re-divided.

Click the button 'Apply to Contour' if the values are changed.

The changes will be reflected onto color distributions (heat map, color map, or clouds of colored points).

Tick or untick in order to display or hide a Heat Map and its grids as well as a Color Map. If Point Clouds are the test subjects, the checkbox 'Display Point Clouds' appears.

Basic Statistics Q'ty

Basic Statistics Q'ty

Category Range Setting ?

Preset

Lower Limit	Upper Limit	Ratio(%)
0.45	-	0.00
0.35	- 0.45	0.00
0.25	- 0.35	0.00
0.15	- 0.25	0.00
0.05	- 0.15	0.00
-0.05	- 0.05	55.70
-0.15	- -0.05	32.21
-0.25	- -0.15	10.74
-0.35	- -0.25	1.34
-0.45	- -0.35	0.00
-	-0.45	0.00

Apply to contour

Display settings

Display heatmap

Display heatmap lines

Display colormap


Heatmap Distance Value Export

CSV output  Inversion of sign


Reflect Color on mesh

Use face color for colormap

Paste heatmap as texture

Click  in order to flip vertically colors of the color scale.

It is necessary to click the button 'Reflect on the Contour Map' in order to reflect the change onto color distributions (heat map, color map, or clouds of colored points).

Click  in order to display the Command Window below.

Edit preset contents and save threshold values and colors in the Command Window as needed.

Preset

Save Presetting ?

Presetting Name

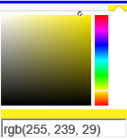
Save

Load Presetting ?

No Presetting

Click a color square in order to display the Window (right).

Change color in the Window as needed.



Click the button 'Apply to Contour' in order to re-create color distributions through the use of values and colors set for segments of a Cell Distance (distance between a cell and the standard shape).

Click the button 'CSV output' in order to output in a CSV file Cell Distance values of Heat Map in the same sequence as of Heat Map. Tick 'Invert Sign' in order to invert Distance Value from positive to negative or vice versa.

Click the button 'Use face color for colormap' in order to assign faces with colors indicated by the temporary figures of color map.

Click the button 'Paste heatmap as texture' in order to paste a Heat Map as a texture on a mesh.

# Display Enlarged Section Lines (1)

This command computes and identifies Section Lines of a finely uneven mesh (test subject) and Section Lines of a smooth mesh of close shape (standard shape). Zoom in differences (or discrepancies between the two).

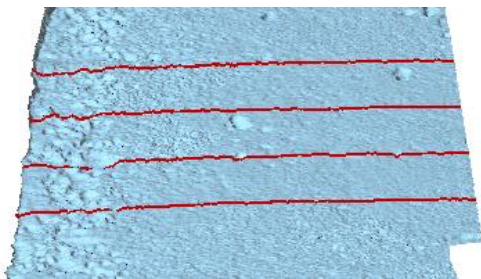
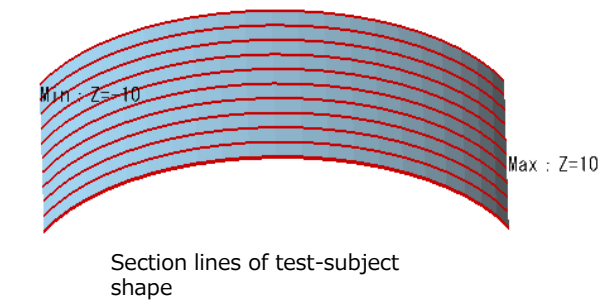
A test subject (mesh) is imported.

Choose as a standard shape, a smooth shape (plane, free-form surface, or cylindrical surface) computationally estimated on the basis of a test-subject shape, a plane of a fixed Z axis value, a mesh of external file and a cylindrical surface of a specified radius.

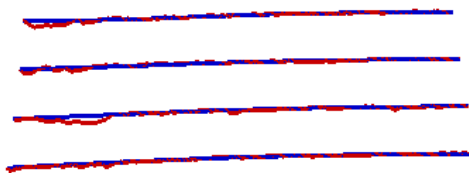
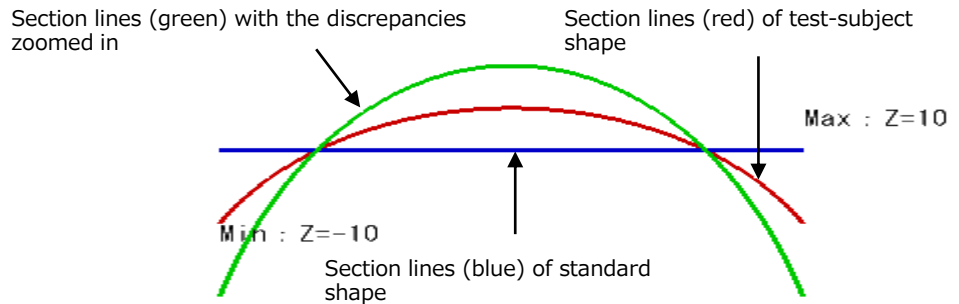
Calculate Section Lines by using planes positioned at even intervals perpendicular to a coordinate axis.

An area subject to calculating Section Lines can be changed after drawing Section Lines.

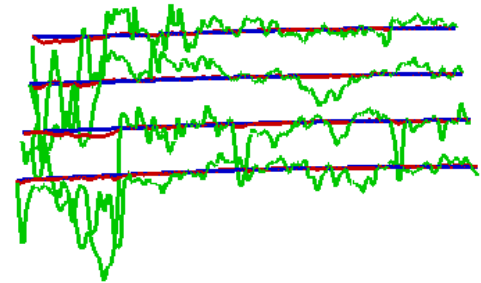
Section Lines can be converted into lines composed of polygonal lines. The conversion results can be exported into IGES files.



Section lines of test-subject shape



Section lines of test-subject shape and section lines of standard shape



Section lines (green) with the discrepancies zoomed in

# Display Enlarged Section Lines (2)

Choose a standard shape (see previous page).

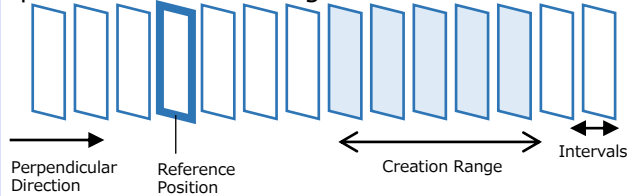
Specify a direction of a group of planes including the subject planes (cross-sections) where Section Lines will be drawn. Also, specify at what intervals the planes are arrayed and a position of the subject planes (cross-sections).

'Perpendicular Direction' refers to a direction of normal line of planes including the subject planes (cross-sections). Choose a direction of coordinate axis.

A 'Reference Position' will be set at the value null (0) of the coordinate axis chosen at the section 'Perpendicular Direction'.

A group of planes will be arrayed at intervals of a value specified at the section 'Intervals' from the Reference Position.

The subject planes (cross-sections) will be created within the 'Creation Range' to the extent that the value of a coordinate axis (specified in the section 'Perpendicular Direction') attributable to each of the subject planes falls in the Range.



Change a magnification rate, and click the button 'Use magnifying factor', in order to revise a magnification rate of differences and re-create a green line.

Click the button in order to save a temporary figure of Section Lines and open a Window that indicates a results list.

**Display Enlarged Section Lines** How to use

**Standard Shape**

Estimate on the basis of mesh  Specify plane  
 Specify cylinder  External file

**Select External File**

Display meshes of external file

**Cross-Section Setting** ?

**Perpendicular Direction**  X  Y  Z

Intervals	10
Reference Position	0

**Creation Range**

min  -  max

If you choose 'Estimate on the basis of mesh', choose as the estimation subject Plane, or Smooth Free-form Surface, or Cylinder. (See Figure A below)

If you choose 'Specify plane', specify a Z value for a plane that has a fixed value of Z axis. (See Figure B below)

If you choose 'Specify cylinder', specify a radius value and a cylindrical axis. (See Figure C below)

If you choose 'External file', specify an external file. Tick the option 'Display meshes of external file' in order to display a specified mesh. (Left Figure)

**Estimated Shape**

Plane  Free-form Surface  Cylinder

A) Estimate on the basis of a mesh

**Cylinder Setting**

Radius	10
--------	----

**Z value of the XY plane**

B) Specify a plane

**Axial Direction** ?

X  Y  Z

C) Specify a cylinder

Click the button 'Execute' in order to display Section Lines and the fields below under the button 'Execute'.

• Display section lines with the mesh  
 • Display section lines with the standard shape  
 • Display enlarged section lines

**Display Range**

min  max

**Magnifying factor**

Tick/untick the checkboxes in order to display/hide individual Section Lines.

Drag the circle ● of the sliding scale in order to limit the display area of Section Lines.

Click the button 'Export Lines as IGES File' in order to export individual Section Lines into IGES files.



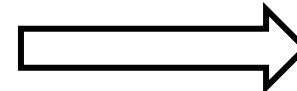
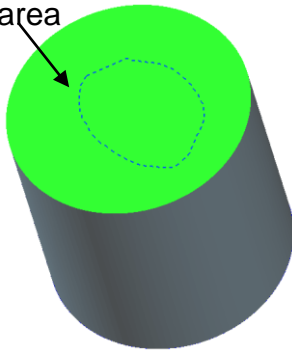
# 10. Color Tab

## Fill Color (1)

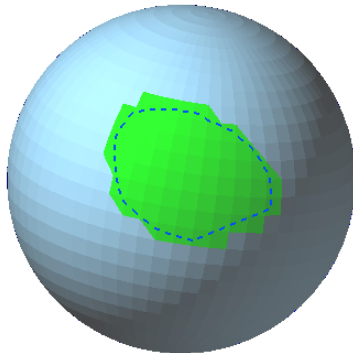
This command performs the following coloring operations on the faces.

- Apply texture
- Add face color
- Add a vertex color (to the vertices of the face group)
- Delete color information

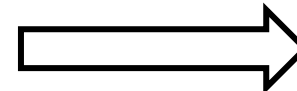
Specified area



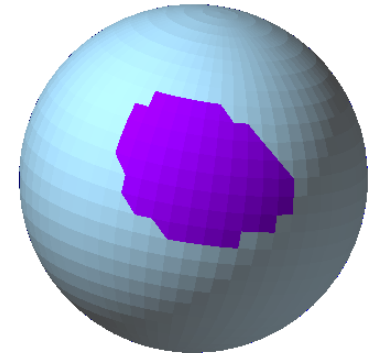
Paste the image data as a texture in the specified area.



Selected color



Add face color to the specified area.



## Fill Color (2)

Specify the area to be colored.

If checked, invisible faces will not be selected.

If checked, all faces in the specified area will be processed. If unchecked, the face connected to the foremost face in the specified area will be processed.

The screenshot shows the 'Fill Color' dialog box with the following settings:

- Specify areas to fill:** Free-form
- Visible Faces:**
- Target all faces in the specified area:**
- Color Type:**  Texture (Image),  Face Color,  Vertex Color,  Delete Color
- Texture Image Reference:** Eyedropper icon (1), Read Image button (2)
- Texture Pasting Method:**  Projection,  Unwrapping
- Execute** button

"Projection" projects an image in the direction of the line of sight and pastes it as a texture.

"Unwrapping" expands the area to a plane and pastes the texture.

"Projection" is faster and more stable than "Unwrapping", but the image becomes grainy on faces that are close to parallel to the line-of-sight direction.

The screenshot shows the 'Fill Color' dialog box with the following settings:

- Specify areas to fill:** Free-form
- Visible Faces:**
- Target all faces in the specified area:**
- Color Type:**  Texture (Image),  Face Color,  Vertex Color,  Delete Color
- Color Palette:** OFF, Eyedropper icon (1), Color palette grid (2), RGB input field (3) containing 'rgb(255, 0, 0)'
- Execute** button

Select the type of color painting. "Delete Color" leaves the color uncolored.

There are three ways to select the color to paint on the area.

1) Copy of color on mesh  
Click the eyedropper button, then click the dot on the colored mesh. The color of the clicked position is sucked up by the dropper and can be used as the face color of the specified area.

2) Use of color palette  
Click the color to paint from the palette.

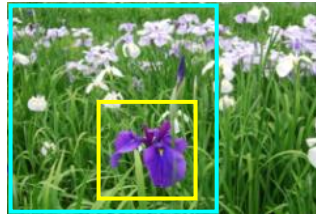
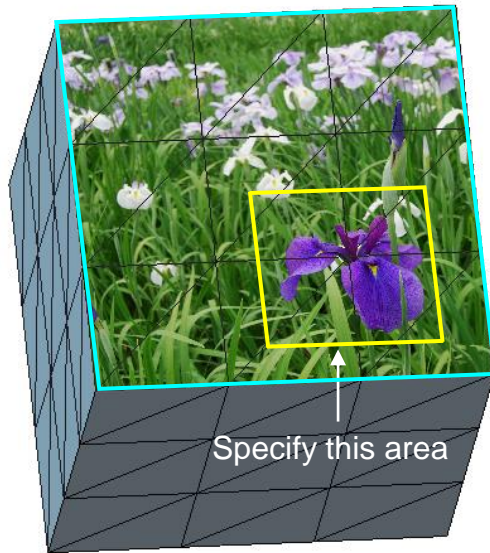
3) RGB input  
Enter the RGB value directly in parentheses.

There are two ways to refer to the image to be pasted.

- 1) Copy the texture image  
Click the eyedropper button, then click the texture you want to copy. The texture image is sucked up by the dropper and can be used as a texture to be attached to the specified area.
- 2) Use of image files  
Click the "Read Image" button and select an image file. The image is pasted as a texture in the specified area.

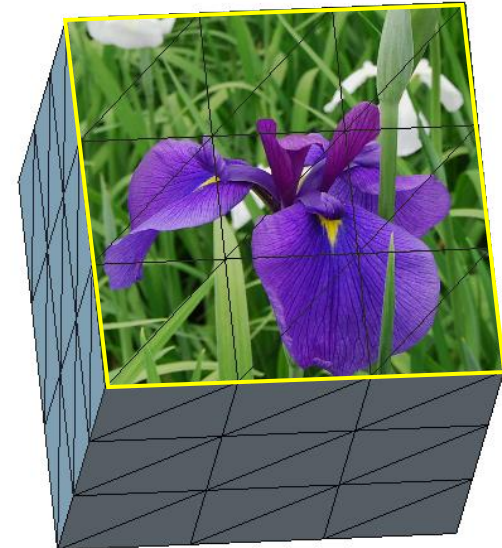
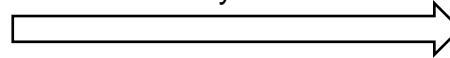
# Texture Alignment (1)

Changes the area of the texture image referenced by the faces.



Referenced image data

Change the reference range within the yellow frame



Texture Alignment [How to use](#)

Select the face with texture image to align. [?](#)

Material Name:

Specify two points on the model. [?](#)

Coordinates of the 1st Point:

Coordinates of the 2nd Point:

Specify two points on the texture image which correspond to the points on the model. [?](#)

Coordinates of the 1st Point:

Coordinates of the 2nd Point:

The operation method is explained on the next page.

# Texture Alignment (2)

The blue character string represents the operation performed by users.

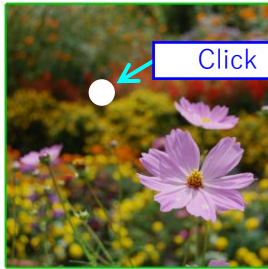
Step-1

Step-2

Step-3

Execute

Mesh



When the textured face is clicked, the boundary of the target area is displayed in green.



Click two points on the mesh.

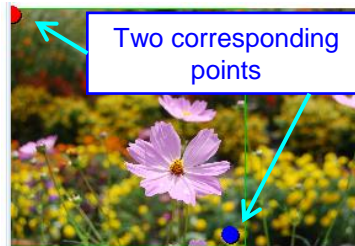


The Texture that is assigned to the mesh is updated.

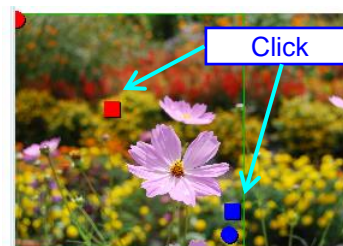
Texture image (separate window)



The texture image will be displayed in a separate window



The Texture Image shows two points corresponding to the clicked points.



Click two points on the Image.

The window disappears.

Command dialog

Select the face with texture image to align. ?  
  
 Material Name: createdByBlottingOut

The name of the Texture Image is displayed.

Specify two points on the model. ?  
  
 Coordinates of the 1st Point:  
 Coordinates of the 2nd Point:

The XYZ coordinate values for the clicked points are displayed.

Specify two points on the texture image which correspond to the points on the model. ?  
  
 Coordinates of the 1st Point:  
 Coordinates of the 2nd Point:

The coordinate values for the clicked points on the Texture Image are displayed.

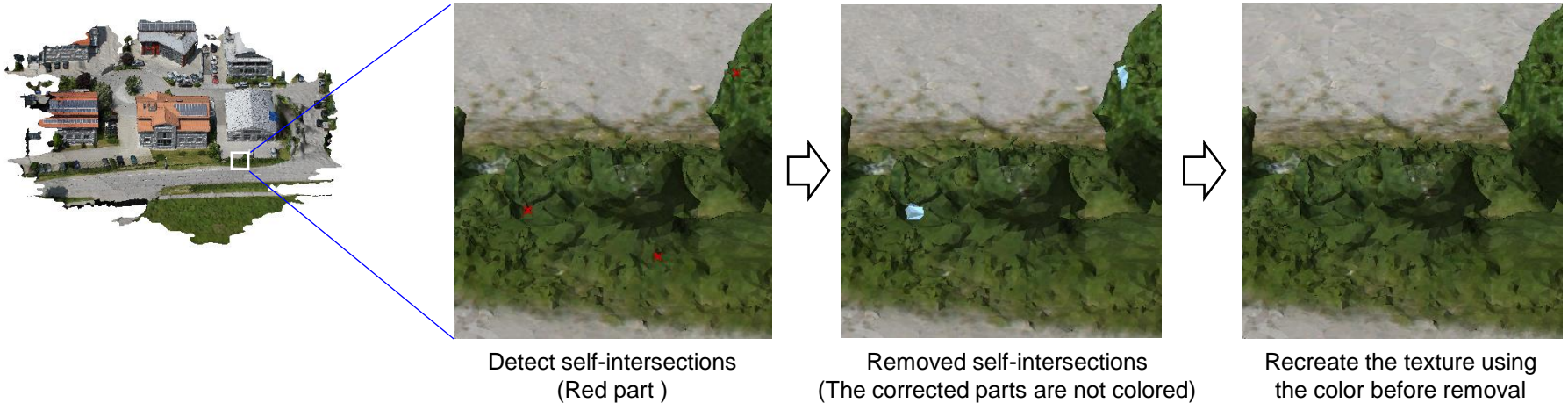
Click the "Execute" button.

# Transfer Color

If the external file has a mesh that has almost the same shape and position as the mesh in POLYGONALmeister, by referring to the mesh color of the external file, the mesh being operated is colored (texture or vertex color).

[Example] Faces without texture may be created by correcting the cleaning error part, etc.

It is possible to paste the texture by referring to the texture of the mesh before correction.



**Transfer Color** [How to use](#)

Transfer Source

Transfer Target  
 Entirely  Part

Transfer Destination  
 Texture (Image)  Vertex Color

Resolution of New Texture ?  
512 1024 2048 4096 8192

Click to specify the mesh of the external file to reference the color.

Specify the number of pixels of the texture created by transfer.  
For example, if you select 512, an image file of 512 x 512 pixels will be created to express the texture pattern.

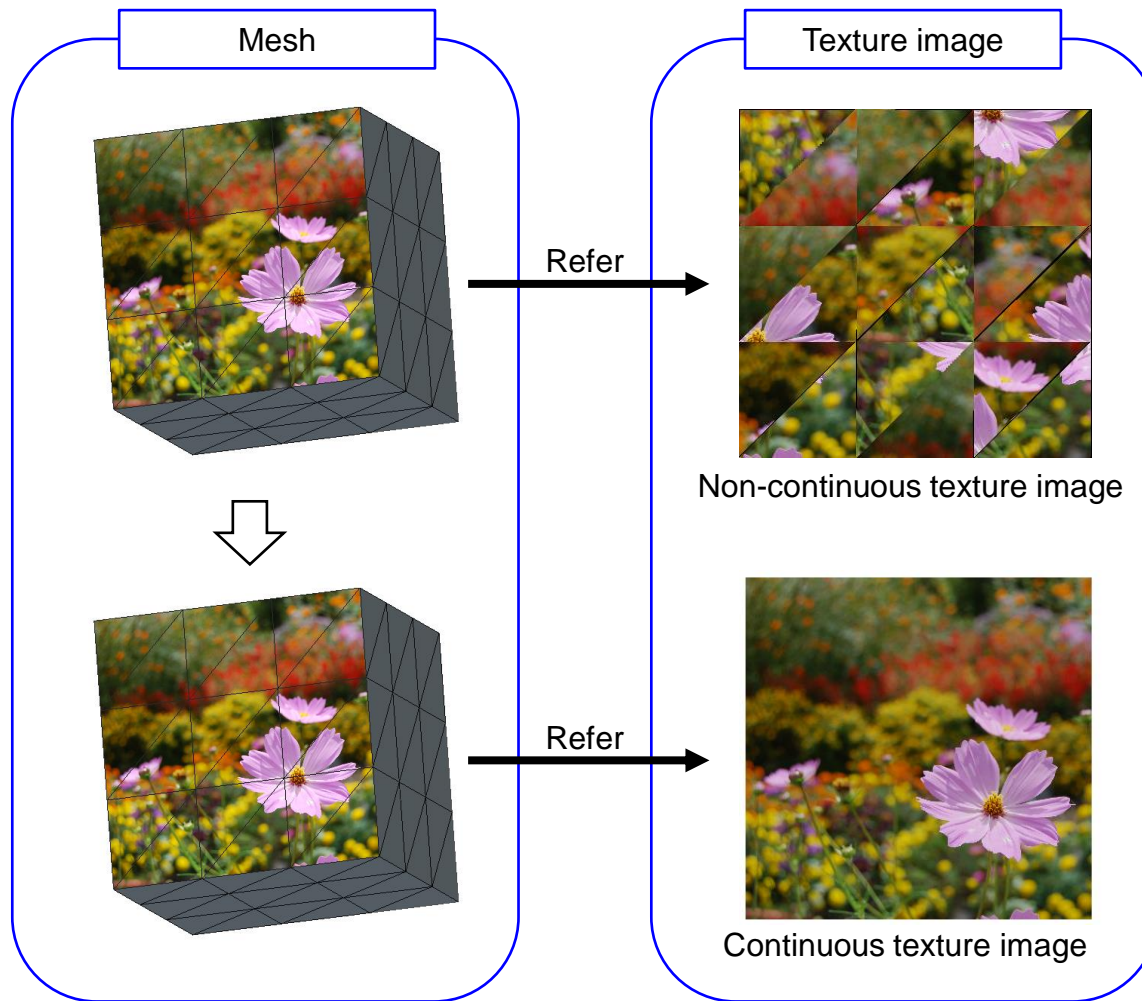
The system automatically sets the resolution close to the number of pixels of the image file used by the mesh of the external file as the initial value.

Select to recolor the entire mesh or recolor some of the faces.

Select either texture or vertex color to paint.

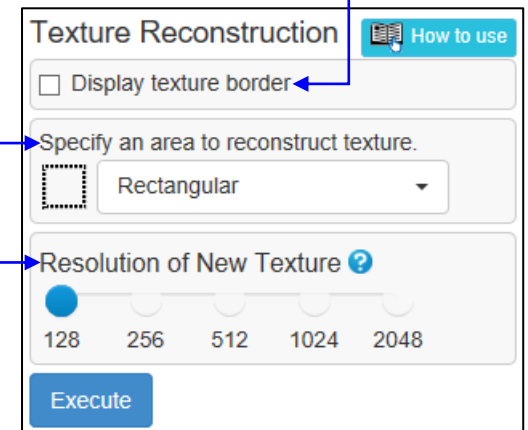
# Texture Reconstruction

This command ensures that there are no discontinuous texture boundaries inside the specified area (the adjacency of faces and the adjacency of the corresponding triangles in the texture image are the same).



Specify the area to recreate the texture.

If checked, a non-continuous texture boundary (where the faces on both sides of the edge refer to the texture image at a distance) is displayed as a blue line.



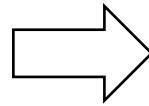
The number of pixels in the vertical and horizontal directions of the texture image created by the texture reconstruction process.

## Convert to Vertex Color

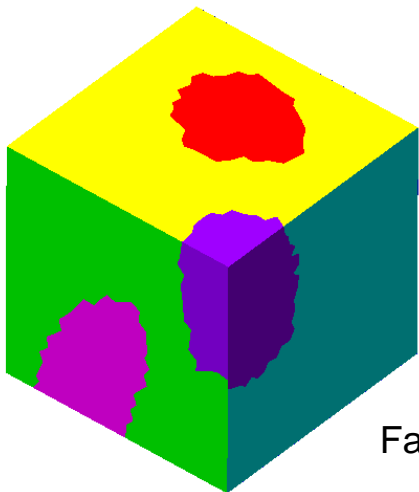
Recolor the mesh colored with "Texture" or "Face color" with "Vertex color".  
This function is effective when the software in the post-process doesn't handle textures.  
(There are no parameters to set in the command dialog)



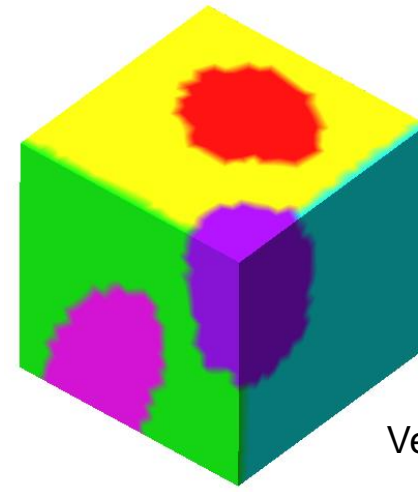
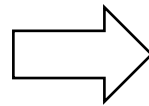
Texture



Vertex Color



Face color



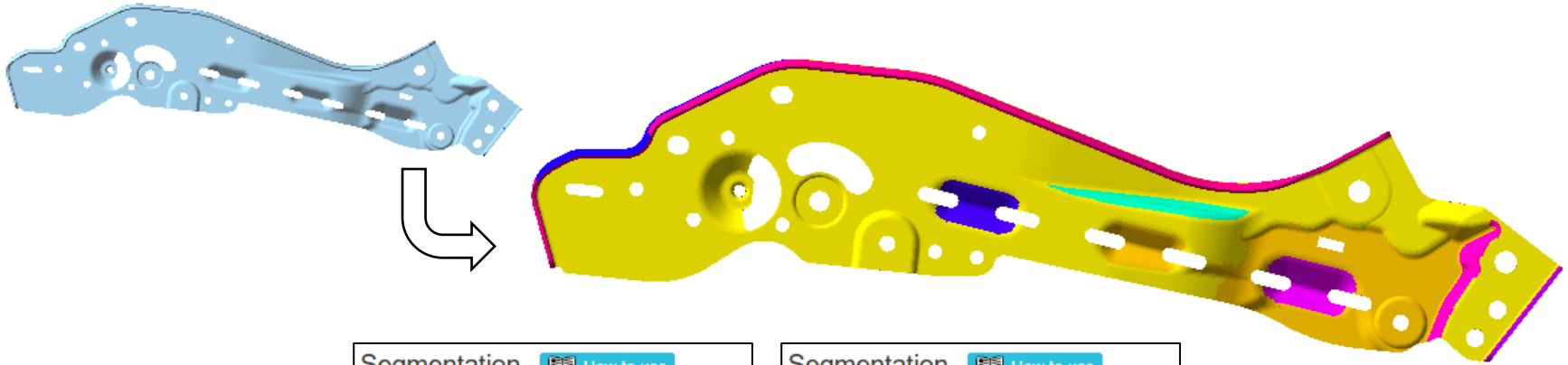
Vertex color



# Segmentation

The mesh is automatically divided into plural areas (called segments) based on the shape characteristics, and the face color is set for each area.

If you set "Color" to the area specification method with various commands, you can select the faces for each segment.



**Segmentation** [How to use](#)

Segmentation Subject  
 Entirety  Part

Segmentation method  
 Automatic  
 Indicate the number of Segments

Shape Type  
 CAD data  Measurement data

Approximate Number of Segments ?  
Few  Many

Execute

**Segmentation** [How to use](#)

Segmentation Subject  
 Entirety  Part

Segmentation method  
 Automatic  
 Indicate the number of Segments

Number of Segments ?  
24

Execute

Select whether to divide the entire mesh or the specified part.

Select whether the mesh is derived from CAD or measurement.

Specify in 3 steps whether to divide into a small number of areas or a large number of areas.

Select "Automatic" to determine the number of areas the system will divide. If you select "Indicate the number of Segments", the user can specify the number of areas to divide.

Specify the number of areas to be divided.

## Edit Segment Boundary (1)

You can do the following for segments (a segment is an area divided by the “Segmentation” command):

- Edit: Smooths the edge row of the segment boundary. You can automatically smooth all boundaries, smooth only the specified boundary, or change the position of the specified boundary.
- Area Integration: Combine plural adjacent segments into one. (You can also combine segments using the eyedropper function with the “Fill Color” command.)
- Area Division: One segment is divided into two at the specified position.

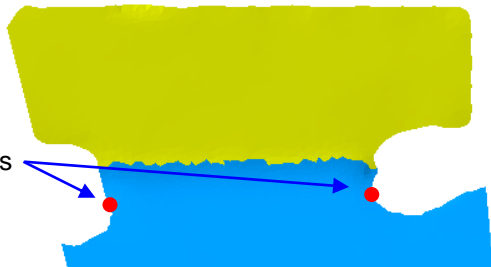


Edit  
→



Select two segments

Area Integration  
→



Specify two vertices

Area Division  
→



# Edit Segment Boundary (2)

If you select "Edit", the segment boundaries will be displayed as green lines. Green marks appear at the endpoints of the boundaries. Follow the explanation in the command dialog.

Passage points are created automatically, but you can adjust the number.

Click the "Align" button to change the segment boundary to the position of the displayed blue line.

**Edit Segment Boundary** How to use

**Edit** Area Integration Area Division

Specify the edge of the segment boundary.

- When you specify an edge, ● is displayed at start and end of boundary, ● is displayed at the passing point on boundary. The route before alignment (sequence of vertices connecting start and end with shortest distance) is displayed in red, and the route after alignment is displayed in blue.
- Click the 'Align' button to change the boundary to the route after alignment.
- You can move the end points on the boundary by dragging ●.
- Drag ● to change passing point and the route.
- When you click a vertex on the red route, clicked vertex becomes passing point of the route and ● is displayed.
- Clicking ● again will make it no longer a passing point.

Number of Passing Points

Few ● Many

Clear

Align

Automatically align all boundaries

Combine adjacent segments into one.

**Edit Segment Boundary** How to use

Edit **Area Integration** Area Division

Specify concatenated segments to be merged. When the "Integrate" button is clicked, all specified segments will be merged and has same color.

Clear

Integrate

Click plural adjacent segments and click the "Integrate" button to combine them into one area.

Click the "Automatically align all boundaries" button to automatically smooth all segment boundaries.

**Edit Segment Boundary** How to use

Edit Area Integration **Area Division**

Specify one segment.

- When you specify a segment, its boundary is displayed.
- When 2 vertices on boundary are specified, ● is displayed at the specified points, sequence of vertices connecting specified points with shortest distance is displayed in red, and the route after alignment is displayed in blue.
- You can move the end point of new boundary by dragging ●.
- When you click a vertex on the red route, clicked vertex becomes passing point of the route and ● is displayed.
- Clicking ● again will make it no longer a passing point.
- Drag ● to change passing point and the route.
- Clicking "Split" button to split into 2 segments.

Clear

Split

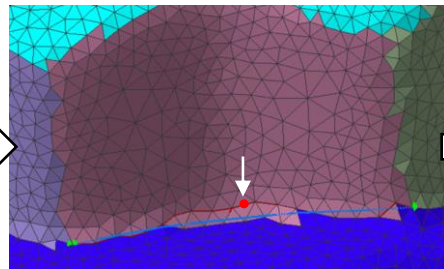
Click on the segment you want to split. The boundary of the clicked segment turns green. Then click on the two vertices on the green edges. Divide the segment by the line connecting the two vertices. Follow the explanation in the command dialog.

You can reselect the boundary by clicking the "Clear" button.

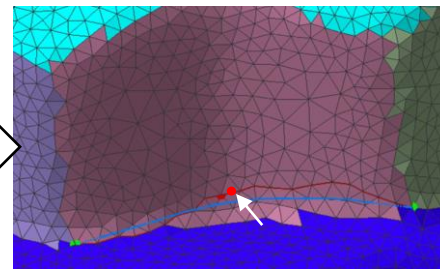
Click the "Split" button to split the segment.



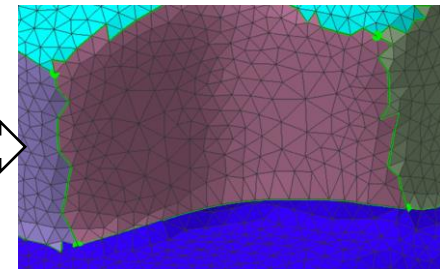
Click on a segment boundary to see a red edge row and a blue line. The blue line is the edge row after alignment. Smooth the red line to create a blue line.



To change the shape of the blue line, change the path of the red edge row connecting the green vertices (end points of the segment boundary). Click a vertex of the red edge row to add a red mark.



Dragging the red mark changes the path of the red edge row and changes the shape of the blue line.



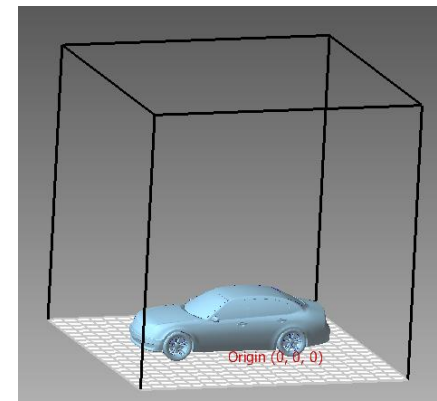
Click the "Align" button to move the segment boundary to the position of the blue line.

# 11. 3D Print Tab

# Display Forming Table

This command displays the image of the 3D printer's forming table in the figure display area (right figure).

You can register a forming table that matches the specifications of your 3D printer.



**Display Forming Table** How to use

Display | Edit Settings

Preset Selection ?  
Default

Printing Space Size  
X:200.000000  
Y:200.000000  
Z:200.000000

Display Grid ?

Grid Spacing ?  
10

Display | Hide

**Display Forming Table** How to use

Display | Edit Settings

Registered Presets ?  
Default

Preset Name ?  
Default

Origin Position ?  
 Bottom Left  Center

Printing Space Size ?

X	200
Y	200
Z	200

Grid Spacing ?  
10

Update / Add | Delete

Click the button to show / hide the forming table.

The preset name of the registered modeling space is set in the pull-down menu. Select the preset to use.

The size of the forming space selected in "Preset Selection".

If checked, the grid will be displayed on the forming table.

The spacing between the grid lines displayed on the forming table. The registered value is displayed, but change it if necessary.

The spacing between the grid lines displayed on the forming table.

Click to change the parameters of the forming table registered in the preset to the displayed contents. If it is an unregistered preset name, it will be additionally registered.

When changing / deleting the forming table registered in the preset, select the forming table.

To add a forming table to a preset, enter the forming table name.

Place the forming table so that the next position coincides with the origin of the XYZ coordinate system.

"Lower left" = lower left corner of the table

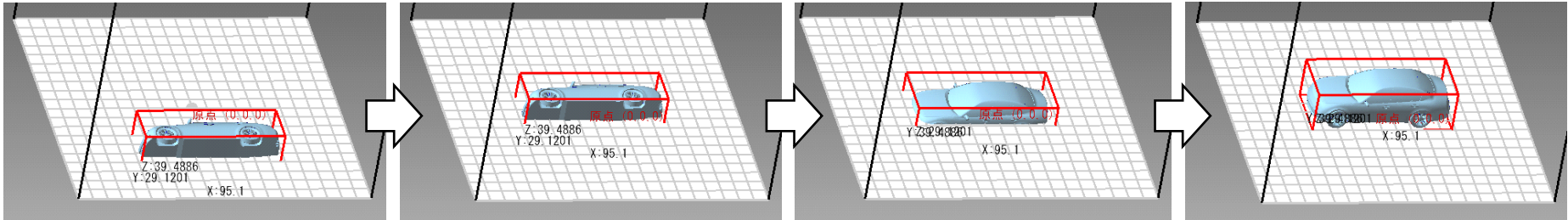
"Center" = table center

The vertical and horizontal lengths of the table (X, Y) and the height of the forming space (Z).

Click to remove the selected forming table from the preset.

# Forming Arrangement

This command simply places a mesh or shell on the build table.

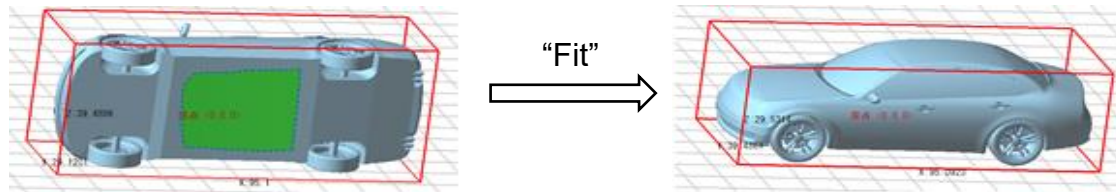


Original mesh

“Center”

“Rotate 90 deg.(X)”

“Z0”



Select whether to place the entire mesh or shells.

specify the area. Rotate the mesh so that the least squares plane of the region is parallel to the XY plane.

### Forming Arrangement How to use

**Target**

Mesh  Shell

**Simple Arrangement ?**

Center
Rotate 90 deg.(X)
Rotate 90 deg.(Y)
Z0

**Fit ?**

Specify region to make parallel with XY plane.

Specified Region Back

Execute Fit

“Center” = Translates the target so that the box center of the target coincides with the center of the forming table.

“Rotate 90 deg.(X)” = Rotate the target 90 degrees in the direction from the Y-axis to the Z-axis around a straight line that passes through the center of the target box and is parallel to the X-axis.

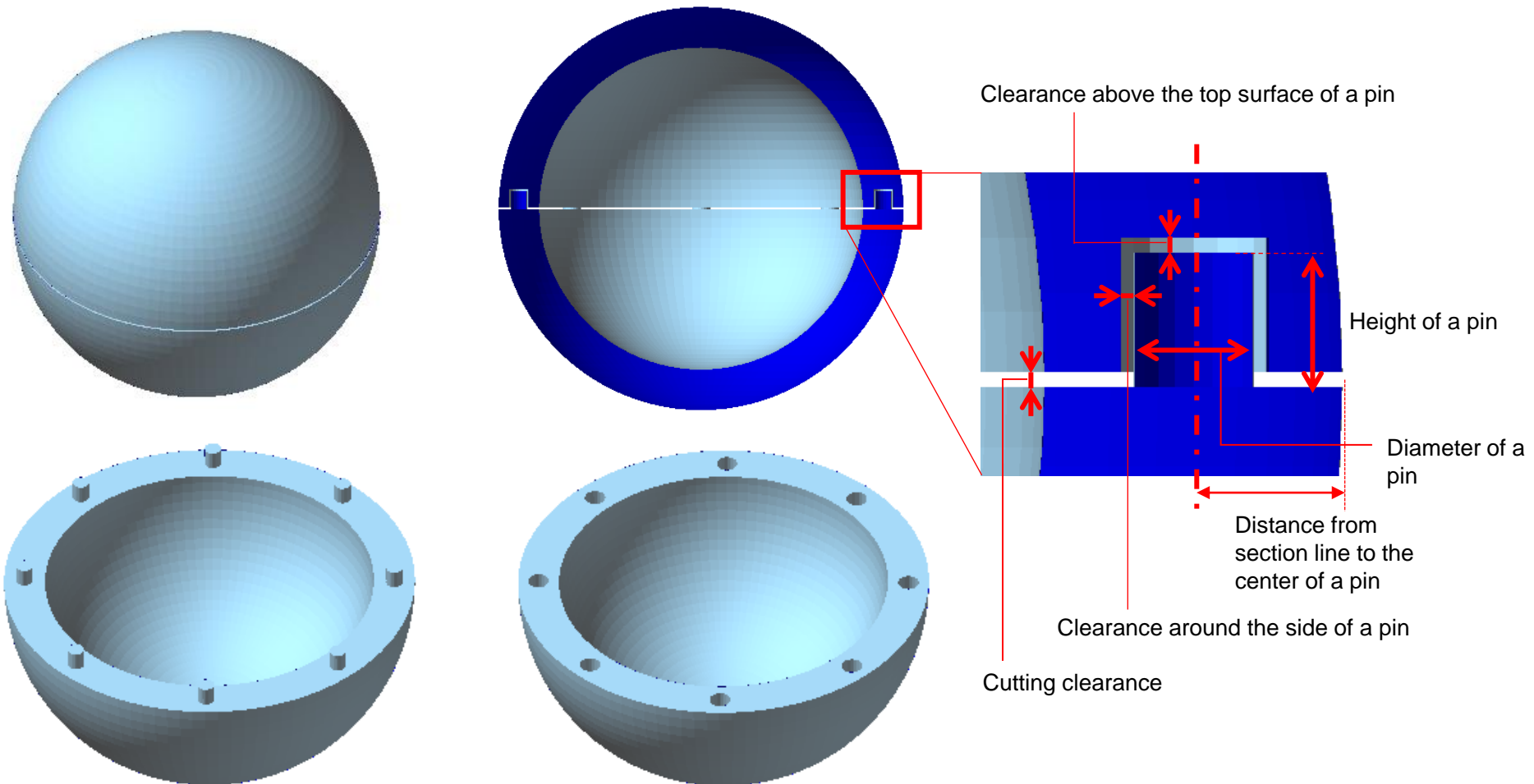
“Rotate 90 deg.(Y)” = Rotate the target 90 degrees in the direction from the Z-axis to the X-axis around a straight line that passes through the center of the target box and is parallel to the Y-axis.

“Z0 placement” = Translate the target in the Z direction so that the minimum Z coordinate value of the target is 0.

## Pinned Cut (1)

Due to 3D printer restrictions such as the size that can be formed, 3D printing may be performed by dividing into plural parts. In that case, a hole corresponding to a cylindrical protrusion (pin) may be provided on the contact surface to fit the parts together.

This command creates protrusions and holes at the same time as cutting.



# Pinned Cut (2)

**Pinned Cut** [How to use](#)

**Cutting Plane** ?  
Specify cutting plane.  
Line

**Select Cutting Line**  
Select All Cancel All Reset

**Cross Section Processing** ?  
Clearance 0.1  
 Color the lid surface

**Pin Setting** ?

Qty	10
Diameter	2.5
Height	5.0

Distance From Section Line To Pin Center  
4.0

**Clearance**

Pin Top	0.1
Pin Side	0.1

Execute

Specify the line to cut.

Specify the width of the gap on the cut surface.

"Qty" = number of pins to be made in the cross section

"Diameter" = diameter of pin (cylinder)

"Height" = height of pin (cylinder)

"Pin Top" = gap between the top surface of the pin and the bottom surface of the hole

"Pin Side" = gap between the pin side surface (cylindrical surface) and the hole side surface

When there are plural intersection lines, the intersection lines to be used for the cutting process can be specified. Click the line of intersection to change the color. The cutting process is performed using the green line of intersection.

"Select All" = Click to turn all lines of intersection green.

"Cancel All" = Click to turn all lines of intersection red.

"Reset" = Click to return to the state before the section line was entered.

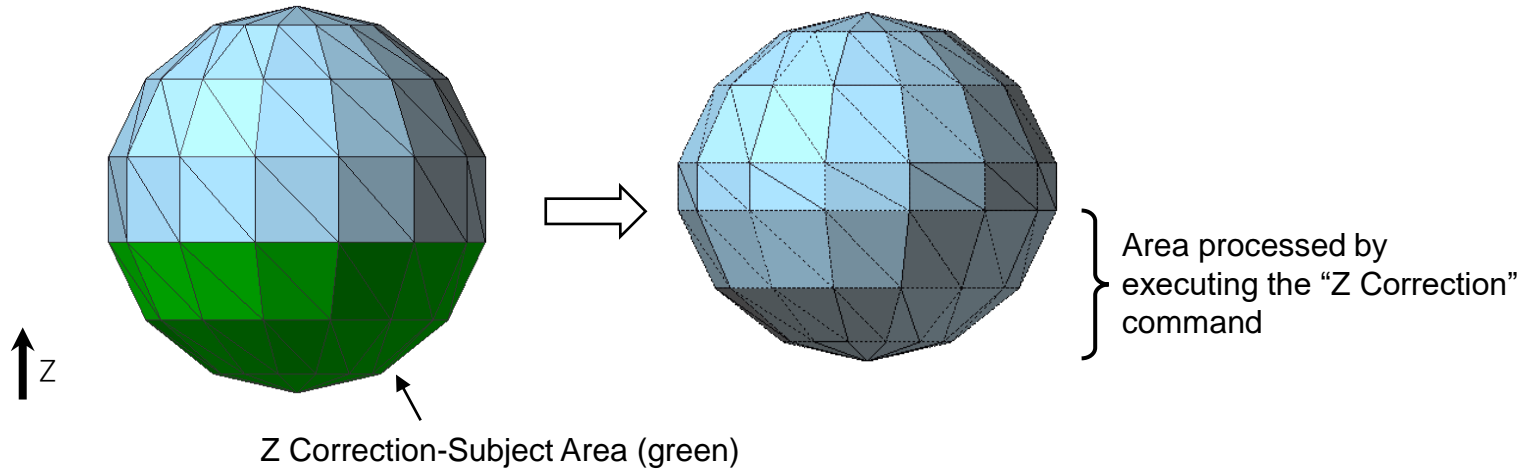
If checked, the cut surface will be colored based on the color of the surrounding face.

The distance from the outer circumference of the cross section to the center of the pin (cylinder axis).



# Z Correction

This command moves a group of faces on the obverse side in a Z-axis negative direction (as indicated by a normal vector pointing at the obverse direction with a negative z element) into the Z-axis positive direction by a specified amount. This function can predict computationally a 3D printing strain to be applied on a mesh, and previously impart the strain amount to the mesh in the opposite direction in order to offset the strain impact.



Distance of movement into the Z-axis direction

Click the button in order to indicate the command-subject area in green, as above.

### Z Correction

[How to use](#)  
 0.1

# 12. Geography Tab

# Terrain Meshing

Read a GeoTIFF file that is widely used as terrain information, and create a mesh with undulations according to the height value. The XY coordinates of the vertices that make up the mesh are arranged in a grid pattern like the GeoTIFF pixel array, and the Z coordinates represent the height.

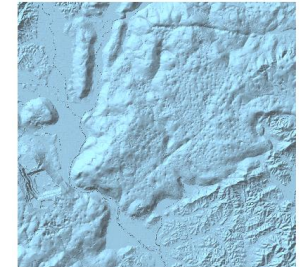
Height information may be missing in the GeoTIFF file at positions that cannot be measured, such as the water surface. Select whether to create vertices at such heights or not.

"Set the Height to 0.0" = Create a vertex with height 0 at the corresponding position.

"Do not Create" = Do not create a vertex (and faces around the vertex) at the corresponding position.



GeoTIFF



Mesh

Click to specify the GeoTIFF file to read.

Select whether to create a vertex at the position where the height information is written as 0.0 in the GeoTIFF file.

"Set the Height to 0.0" = Create a vertex with height 0. "Do not Create" = Do not create a vertex (and a face around the vertex) at the corresponding position.

Terrain Meshing [How to use](#)

**File Read**

How to create a vertex at a position without measured value

Set the Height to 0.0  Do not Create

How to create a vertex at the position of measured value 0.0

Set the Height to 0.0  Do not Create

The interval between pixels to be used ?

**Execute**

The pixel spacing used to create the mesh. For example, if this value is 1, all pixels are used.

GeoTIFF file read

Terrain Meshing [How to use](#)

**File Read**

File name: akiyoshi\_clip.tif

X direction length :8148.49

Y direction length :7834.86

Z min value :70.65

Z max value:430.98

Coordinate System Information

JGD2000 / Japan Plane Rectangular CS III

Longitude Latitude

Upper Left:131d15'04.87"E, 34d16'28.54"N

Lower Left:131d15'07.62"E, 34d12'14.09"N

Upper Right:131d20'23.65"E, 34d16'30.81"N

Lower Right:131d20'26.14"E, 34d12'16.35"N

Center:131d17'45.57"E, 34d14'22.48"N

How to create a vertex at a position without measured value

Set the Height to 0.0  Do not Create

GeoTIFF file name

The size of the mesh to be made

Coordinate system information

Latitude and longitude values of the four corners and the center

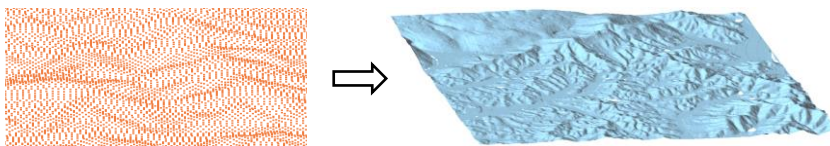


# Create Grid Mesh

This command imports a file of point clouds, and creates a mesh composed of vertices arrayed to form a grid as seen from the z axis direction on the basis of data therein.

This command uses vertices of other meshes or vertices drawn in a PLY file used as point clouds, and creates a mesh based on the point clouds.

(The system processes the point clouds on the premise that the points do not make an overhang in the z axis direction.)



Point Clouds

**Create Grid Mesh** How to use

Target Data ?

Mesh  Point Clouds

---

Point Cloud Data Selection ?

Select File

---

Reference Point ?

0.0,0.0

---

Grid Width ?

5.0

Recommended value(small)

Recommended value

Recommended value(large)

---

Representative Value in the Grid ?

Min.  Average  Max.

Fill in tiny holes and gaps ?

Create a mesh near the origin ?

Execute

Choose as the subject data, either vertices of the present mesh or a file of point clouds.

If you use point clouds, choose a file of point clouds. It is a CSV file or text file (describing x/y/z axis values of a point per line) or a PLY file of vertices data.

Please note that the undo command is disabled if the "Grid Mesh Creation" command is executed for a file of point clouds.

A mesh will be created with the vertices positioned in a grid formed by the axes (of x direction and y direction), as seen from the z direction.

A 'Reference Point' is an arbitrary grid point. Specify the x/y coordinate axis values that can be away from a mesh that will be created.

A 'Grid Width' is a grid interval on an x-y plane.

Buttons of Recommended Value for a grid cell width

The z coordinate value of a grid point can be the minimum, average, maximum of the z coordinate value for points around each grid point.

If checked, the system will fill in tiny holes and gaps generated in a mesh before it outputs the mesh.

If checked, the system will relocate a mesh that it created to a position where the minimum X coordinate value is null and the Y coordinate value is null.

## 【About Recommended Value of Grid Cell Width】

Grid Width ?

5.0

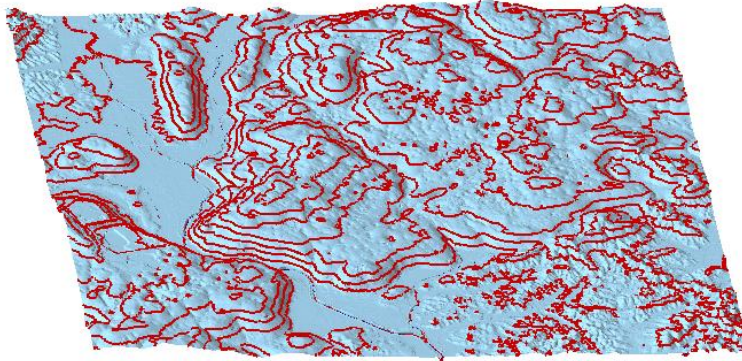
Recommended value(small)   Recommended value   Recommended value(large)

Import a file of point clouds, or select a mesh, or click one of the buttons of recommended value, in order to estimate a recommended value of grid cell width on the basis of the subject data. The recommended value appears in the field.

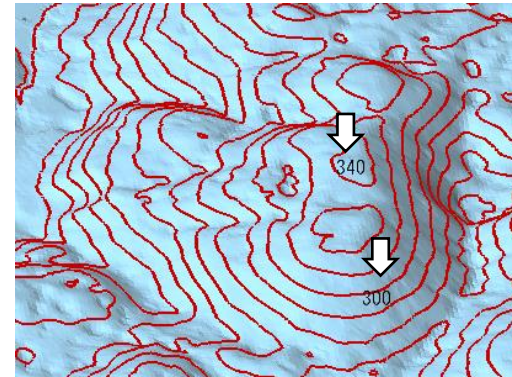
Use the recommended value as a reference and tweak it if an appropriate grid cell width is not known.

# Display Contour Lines

This command displays the contour lines in the specified axis in red. Click the displayed contour line to display the height value.



Display contour lines



Display height at clicked position

Select the height direction of the contour line from "X", "Y", and "Z".

The lowest position and the highest position to display contour lines. Use this to limit the height range and display contour lines.

Click the "Display Result List" button to save the contour lines and open the result list window.

### Display Contour Lines [How to use](#)

Height Direction ?

X  Y  Z

Interval	2
Reference Position	0

Display Range

min  -  max

"Interval" = Contour height spacing.  
"Reference Position" = A representative value of the height for which contour lines are to be obtained.

Contour lines are drawn at the height obtained by adding or subtracting a multiple of the "Interval" to the "Reference Position".

Click to output contour lines as polygonal lines to an IGES file.

# Soil / Space Volume

This command displays the amount of soil or space volume (volume required to fill soil, etc.) in the specified area. It also displays the surface area and the projected area in the Z direction.

	Amount of Soil	Space Volume	
Horizontal Plane			<p>Horizontal Plane(Boundary average): Calculates the projected solid on the Z constant plane that passes through the average of boundary vertices of the specified area.</p> <p>Horizontal plane(min/max): Calculates the projected solid on the Z constant plane that passes through the minimum (amount of soil) / maximum (space) vertex of the specified area.</p>
Slope			<p>Calculate the volume of the projected solid on a plane that roughly passes through the outer circumference of the specified area.</p>

specify the amount of soil and the area to be calculated.

Select either "Soil volume" or "Space volume", "Horizontal Plane(Boundary average)", "Horizontal plane(min/max)" or "Slope". See the figure above for the meaning of each.

When the area is specified, the approximate volume, surface area, and projected area in the Z direction are displayed in the command dialog, and the volume is also displayed in the figure display area.

**Soil/Space Volume** How to use

Specify Region.

Free-form

Back Clear

---

Method of Calculation ?

Amount of Soil  
  Space Volume

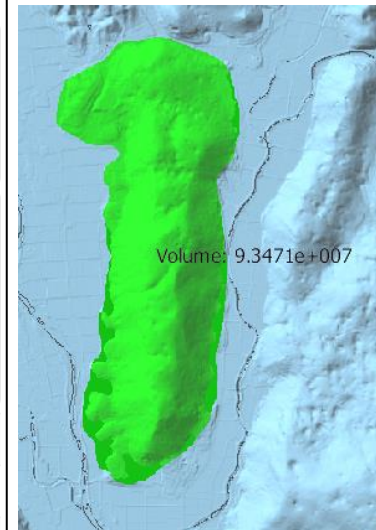
Horizontal Plane(Boundary average)  
 Horizontal Plane(min/max)  
 Slope

---

Approximate Volume      9.3471e+007

Surface Area              1.90874e+006

Projected Area in Z Direction   1.74405e+006



# GPS Track Log Display (1)

The command imports a GPS track data file and delivers temporary figures composed of line arrays or point sequences.

The command accepts GPX, KML, CSV and TXT files.

A CSV file contains comma-separated data of longitude, latitude and altitude for one point per line.

A TXT file contains space-separated data of the same contents.

Choose the “GPS Track Log Display” command in order to show the command dialogue below.

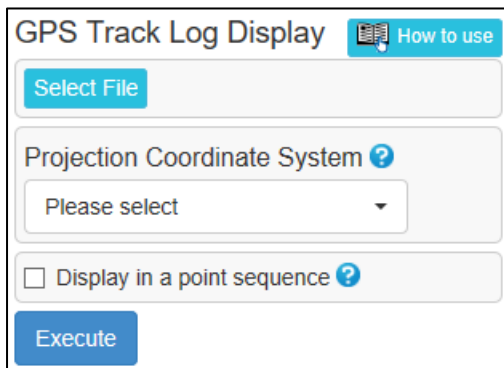
Use the “Choose File” button and choose a GPS Track Log data file, subject for view on the display.

Choose a coordinate system in the pull-down menu “Projection Coordinate System” and click the button “Enter”. (The Projection Coordinate System will be described later.)

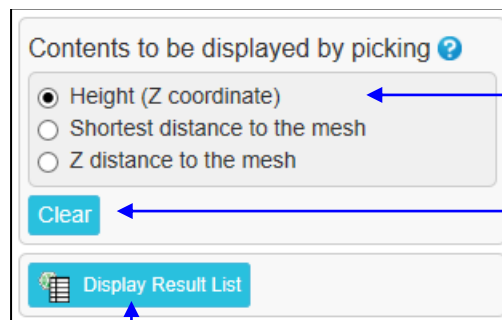
A temporary figure will show track data.

After the temporary figure appears on the display, click on it in order to display values selected by using a radio button.

【Before Executing the Command】



【Display after Executing the Command】



Choose which contents need to be displayed by picking a position on the track data.

If a mesh exists, it is possible to choose the shortest distance up to the mesh and distance of the z direction’.

Delete contents displayed by picking

Click the button “Display Result List” in order to save temporary figures of track data, and open a Results List window. (This is possible if there are meshes.)

## GPS Track Log Display (2)

The UTM zones from 51N to 55N or the Japan Plane Rectangular CS from 1 to 19 can be chosen as a Projection Coordinate System.

JGD2011 is the basis for the zones and the systems.

The tables below show standards of areas where the Projection Coordinate Systems can be used.

Please confirm the official information published mainly by the Geospatial Information Authority of Japan for details.

### 【UTM】

zone	Longitude (East Longitude)	Standard Areas for Use
51N	120°-126°	Part of Okinawa Pref.
52N	126°-132°	Part of Okinawa Pref. Kyushu Region
53N	132°-138°	Shikoku Region, Chugoku Region, Kinki Region
54N	138°-144°	Chubu Region, Kanto Region, Tohoku Region, Part of Hokkaido
55N	144°-150°	Part of Hokkaido

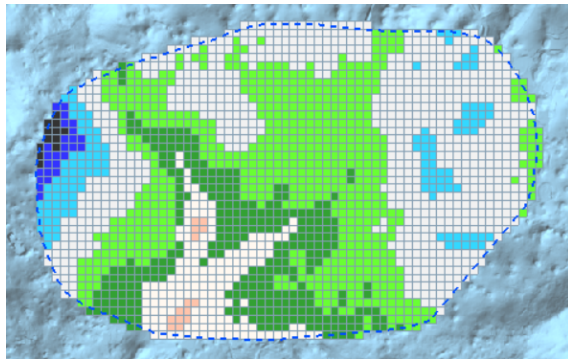
### 【 Japan Plane Rectangular CS】

System Number	Coordinate System Origin (Longitude/Latitude )		Standard Areas for Use
	Longitude (East Longitude)	Latitude (North Latitude)	
1	129°30'	33°0'	Nagasaki Pref. Part of Kagoshima Pref.
2	131° 0'	33°0'	Fukuoka Pref. Saga Pref. Kumamoto Pref. Oita Pref. Miyazaki Pref. Kagoshima Pref.
3	132°10'	36°0'	Yamaguchi Pref. Shimane Pref. Hiroshima Pref.
4	133°30'	33°0'	Kagawa Pref. Ehime Pref. Tokushima Pref. Kochi Pref.
5	134°20'	36°0'	Hyogo Pref. Tottori Pref. Okayama Pref.
6	136° 0'	36°0'	Kyoto Pref. Osaka Pref. Fukui Pref. Shiga Pref. Mie Pref. Nara Pref. Wakayama Pref.
7	137°10'	36°0'	Ishikawa Pref. Toyama Pref. Gifu Pref. Aichi Pref.
8	138°30'	36°0'	Niigata Pref. Nagano Pref. Yamanashi Pref. Shizuoka Pref.
9	139°50'	36°0'	Part of Tokyo, Fukushima Pref. Kanto Region
10	140°50'	40°0'	Aomori Pref. Akita Pref. Yamagata Pref. Iwate Pref. Miyagi Pref.
11	140°15'	44°0'	Part of Hokkaido
12	142°15'	44°0'	Part of Hokkaido
13	144°15'	44°0'	Part of Hokkaido
14	142° 0'	26°0'	Part of Tokyo
15	127°30'	26°0'	Part of Okinawa Pref.
16	124° 0'	26°0'	Part of Okinawa Pref.
17	131° 0'	26°0'	Part of Okinawa Pref.
18	136° 0'	20°0'	Part of Tokyo
19	154° 0'	26°0分	Part of Tokyo

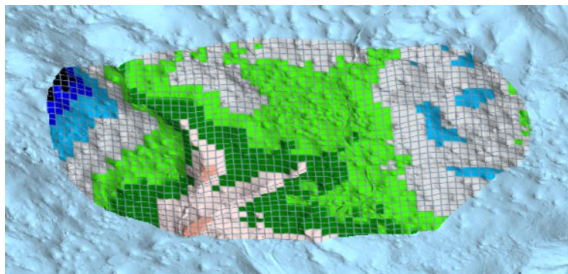


# Dekigata Heat Map (1)

The difference between the coordinate values in the Z-axis direction between the mesh or point cloud of the external file and the mesh being operated is calculated by the evaluation points arranged in a grid pattern. The calculated difference distribution is displayed as a heat map (a diagram showing the value distribution in colors corresponding to the values of individual cells in the 2D grid). You can also paste the heatmap as a texture on the mesh.

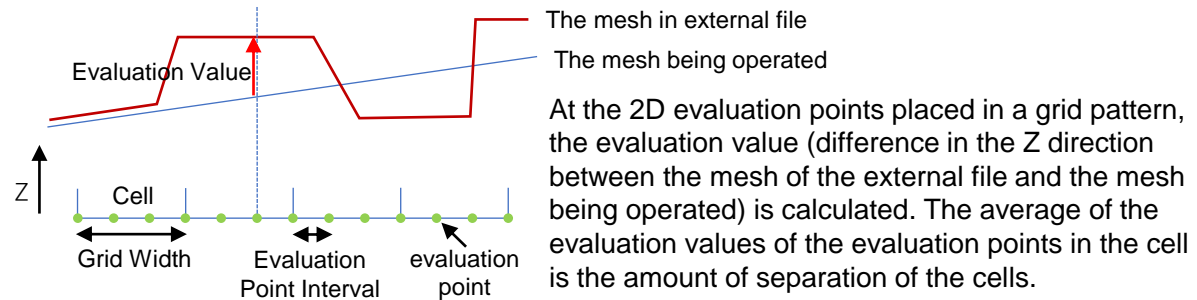


Heatmap

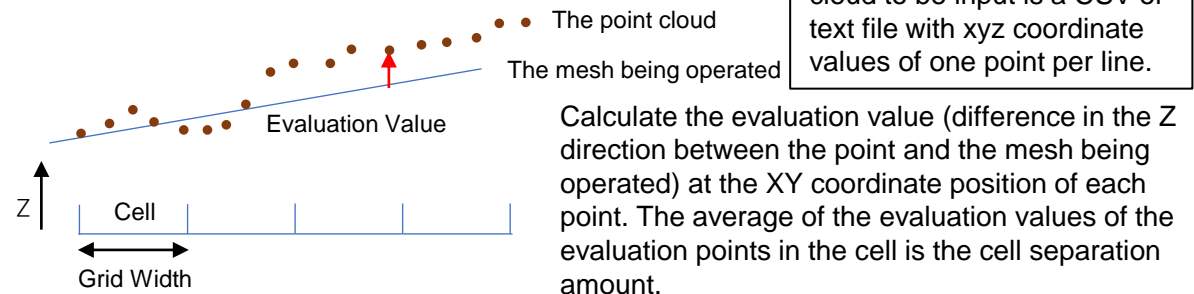


Paste the heatmap as a texture

## [ Comparison with mesh in external file ]



## [ Comparison with point cloud in external file ]



The external file of the point cloud to be input is a CSV or text file with xyz coordinate values of one point per line.

## [ Cell Color ]

The color to be painted on the cell is determined by whether the ratio of the cell separation amount to the "Distance Reference Value" belongs to the section whose threshold 0%,  $\pm 20\%$ ,  $\pm 50\%$ ,  $\pm 80\%$ , or  $\pm 100\%$ .

# Dekigata Heat Map (2)

**Dekigata Heat Map** [How to use](#)

**Data Types** ?

Point Clouds  Mesh

---

**Point Cloud File Selection** ?

[Select File](#)

Display point cloud data

**Interpolation of Heatmap** ?

Interpolate

**Reference Point** ?

0,0

**Distance Reference Value** ?

10

**Creation Area**

Free-form

[Back](#)

---

**Dekigata Heat Map Grid Width** ?

2

[Execute](#)

**Dekigata Heat Map** [How to use](#)

**Data Types** ?

Point Clouds  Mesh

---

**Data selection for evaluation** ?

[Select File](#)

Display external file mesh

**Evaluation Point Interval** ?

1.0

Specify the external file that contains the mesh you want to compare.

If you check here, the mesh of the specified file will be displayed.

The interval between evaluation points (points that evaluate distance) in the XY coordinate system. The evaluation points are made in a grid pattern.

It is one of the intersections of the of cell grid and the evaluation points grid.

Information about evaluation points.

Click to change the color.

For each parcel, the sum of the volumes between the shape and mesh in the external file of the cells contained in the parcel and the number of cells are displayed.

If checked, the heatmap will have cell borders.

If checked, each cell in the heatmap will appear as a 3D point at the location of the shape in the external file.

Click to reflect the color change in the heatmap.

**Dekigata Management Information**

Average Value	0.927172
Maximum Value	7.12176
Minimum Value	-10.8637
Number of Points	4023
Volume	18190

Volume	Number of Cells
0	0
0	0
7000	278
4500	298
400	87
150	44
4000	275
2000	98
100	3
40	1

Put a Ruled Line ?

Display Colored Point Cloud

[Apply to contour](#)

[Paste Texture](#) ? [CSV output](#)

Select the type of data in the external file you want to compare.

Specify the external file containing the point cloud to be compared.

If you check here, the point cloud of the specified file will be displayed.

If checked, when there are no points in a cell due to the sparse point cloud, the values in the surrounding cells will be used for interpolation.

The ratio of the cell separation amount to the "Distance Reference Value" determines the color to be painted on the cell.

Specify the area to create the heatmap.

The grid width of the heatmap.

Click to start the calculation, and when it finishes, the heat map will be displayed, and the "Dekigata Management Information" will be displayed under the "Execute" button.

Click "Paste Texture" to paste the heat map as a texture in the indicated area.

Click "CSV output" to output the xyz coordinate value (z value is the position on the mesh) of the cell center point of the heat map and the distance value of the cell. (x, y, z, cell-separation-amount in one cell per row)

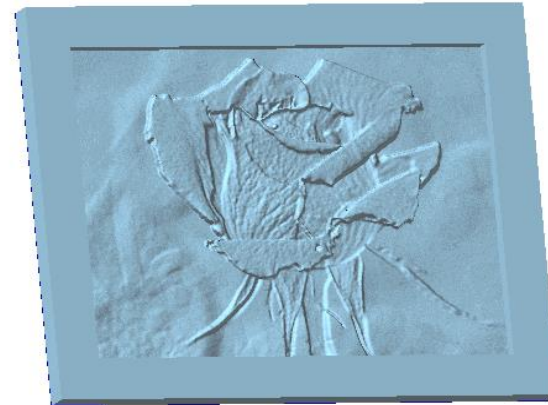
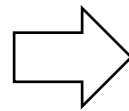
# 13. Relief Tab

## Relief (1)

Read the image file and create a relief-shaped 3D mesh with an uneven pattern according to the grayscale value of the image. The read image can be pasted as a texture.



Image file



With no texture



With texture

# Relief (2)

Click to read an image file. The extensions of readable files are "bmp", "jpeg", "jpg" and "png".

If checked, the relief will be created by dividing the brightness into only two, white and black, without using the intermediate value.



"Binarize" = OFF



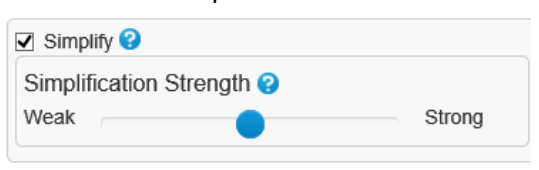
"Binarize" = ON

If checked, bright areas of the image will be raised, and if unchecked, dark areas will be raised.

Extruded parts that are higher than the specified height will be at the specified height, and the high relief parts will be flattened. Set a value smaller than "Extrusion Amount".

If checked, the read image will be pasted as a texture on the mesh. (See previous page)

If checked, the "Simplification Strength" field shown below will be added to the command dialog, so specify the simplification strength in three levels. A simplified mesh is created.



Relief ? How to use

**Import Images**

Smooth ?  Binarize ?

---

Mesh Resolution ?

Coarse  Fine

Raise bright pixels.

---

Relief Breadth ?

10.0 <=  <= 300.0

100.0

Extrusion Amount ?

0.0 <=  <= 30.0

6.0

Cut height ?

0.0 <=  <= 30.0

30.0

---

Frame Width ?

0.0 <=  <= 30.0

10.0

Distance between Relief and Frame ?

1.0 <=  <= 50.0

1.0

Frame Thickness ?

1.0 <=  <= 50.0

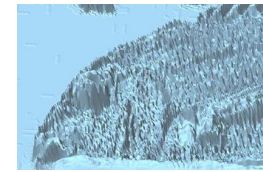
3.0

---

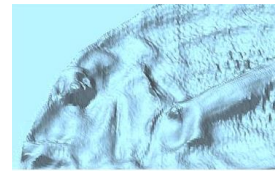
Paste image as texture ?

Simplify ?

**Execute**



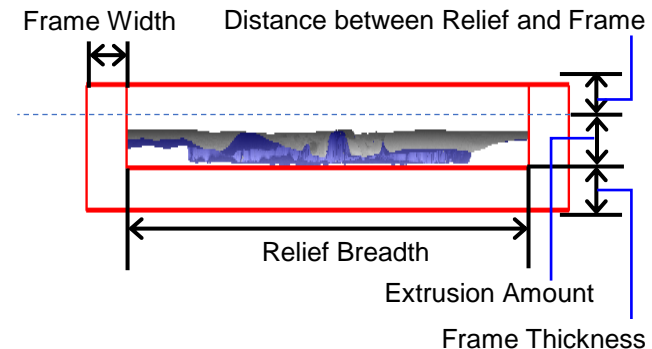
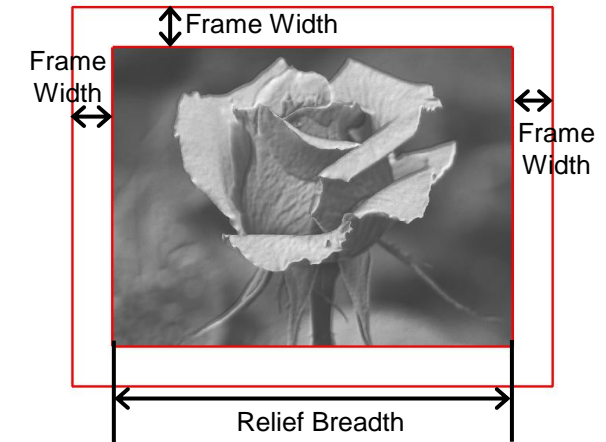
"Smooth" = OFF



"Smooth" = ON

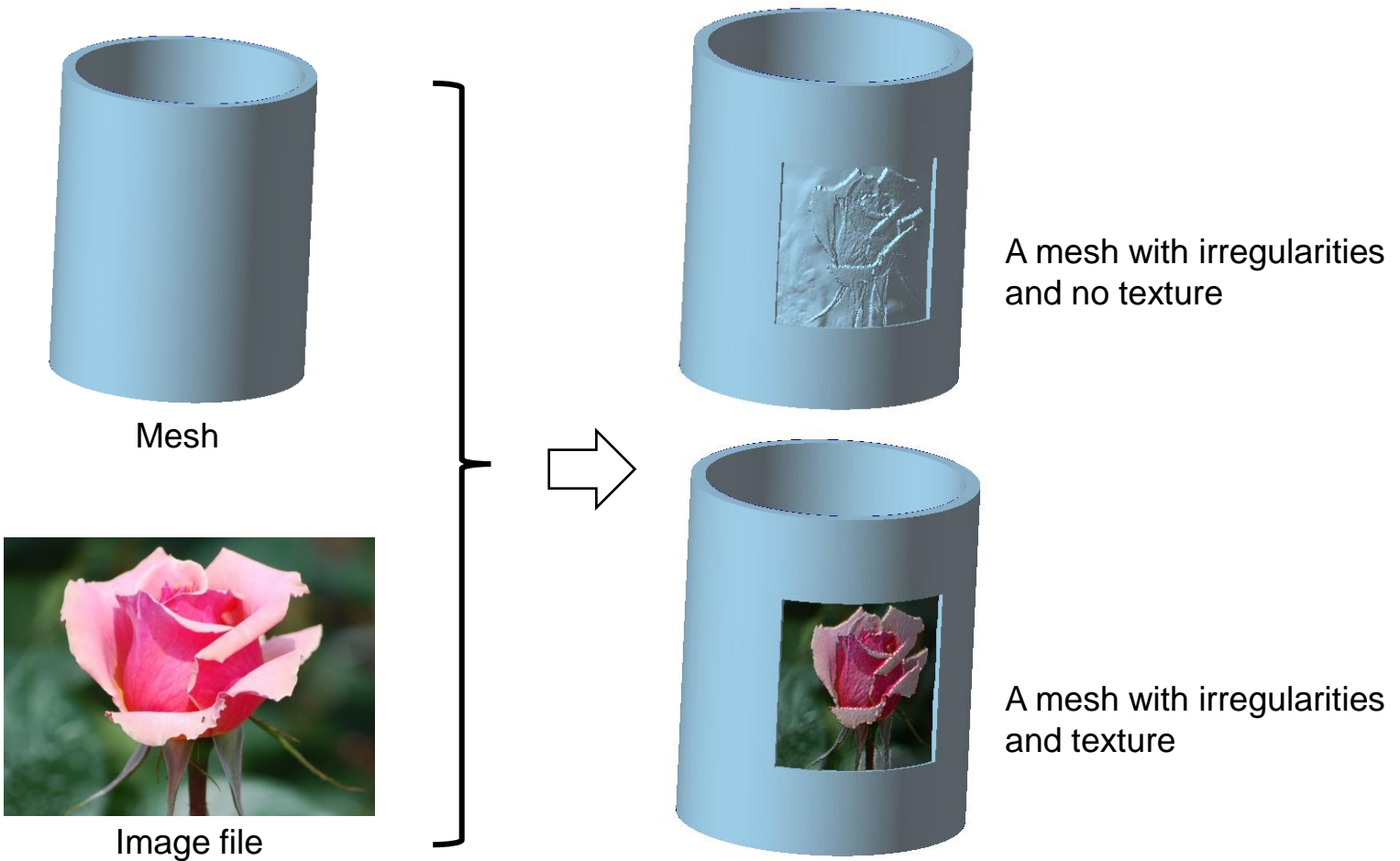
If checked, the unevenness of the created mesh will be smooth.

The fineness of the mesh to be made. The finer the size, the larger the data size of the mesh.

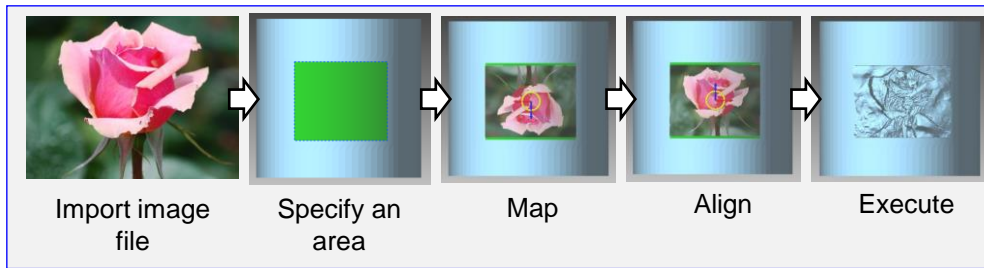


## Embed Relief (1)

Read the image file and add an uneven pattern according to the grayscale value of the image to the specified area of the mesh. (Move the vertices in the area to be patterned in each normal direction by the amount corresponding to the grayscale value of the image.) You can paste the read image as a texture.



# Embed Relief (2)



Click to read an image file. The extensions of readable files are "bmp", "jpeg", "jpg" and "png".

If checked, the uneven pattern will be added by dividing the brightness into only two brightness, white and black, without using the intermediate value.

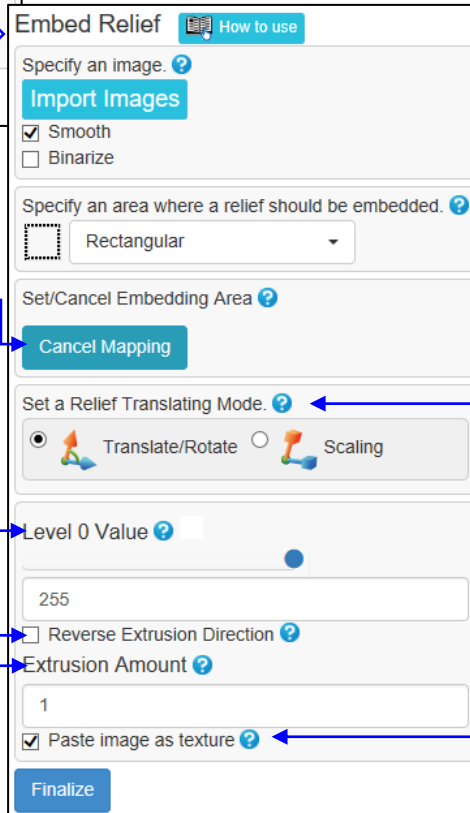
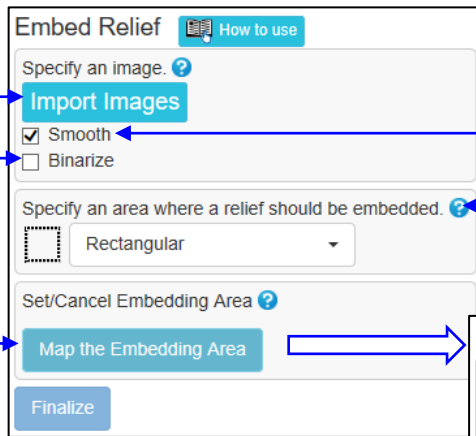
When clicked, the imported image is displayed in the specified area, and the command dialog changes as shown in the figure on the right.

Click to return the command dialog to the state on the left.

The specified value is a grayscale value with no unevenness. For example, if set to 0 (corresponding to black), the black area of the image will not move, and the closer it is to white, the larger it will move.

If unchecked, the white part will be convex and the black part will be concave. If checked, the black part will be convex and the white part will be concave.

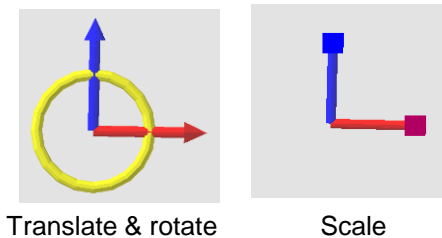
The difference in height between pure white and pure black on a gray scale. Adjust the degree of unevenness of the relief with this value. (Note that this is not the difference between the brightest part and the darkest part of the image.)



If checked, the unevenness of the created mesh will be smooth.

Specify the area to be uneven.

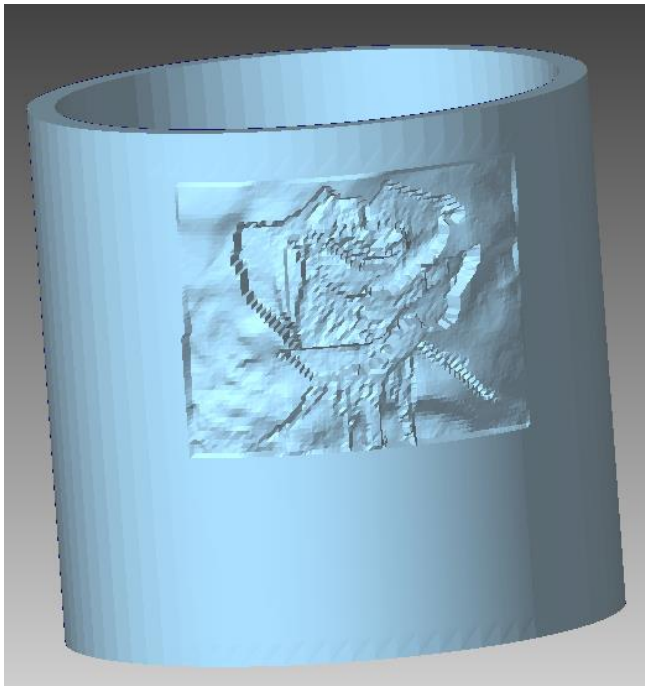
Use the mouse to move, rotate, and scale the imported image. "Translate / Rotate" and "Scale" can be switched with the radio button. In the translate / rotate state, drag the arrow with the mouse to translate the image. Drag the ring to rotate it. In the scaling state, drag the bar to scale in that direction. (See the figure below)



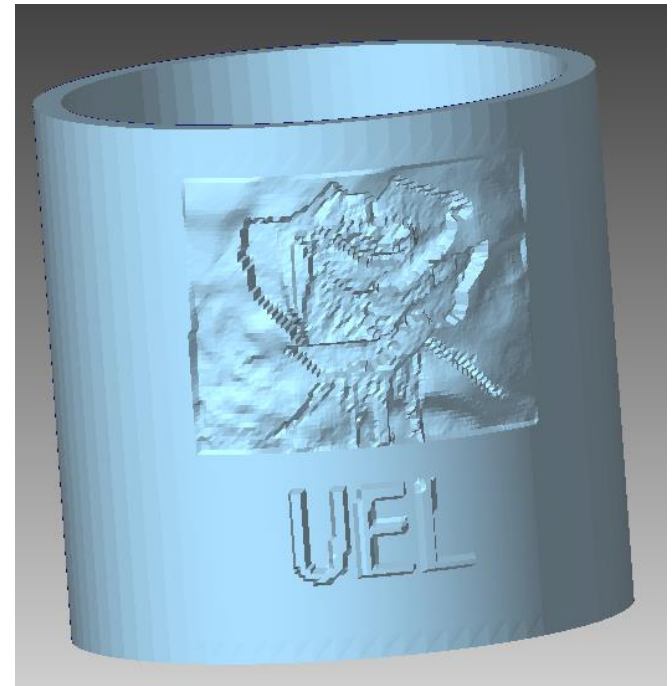
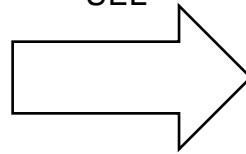
If checked, the imported image will be pasted as a texture on the area. (See previous page)

## Stamp (1)

Add a pattern to the mesh with input characters or handwritten lines.



Embed the letters  
"UEL"



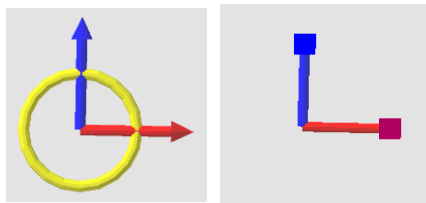


# Stamp (2)

Click to return the command dialog to the state on the left.

Specify the area to embed characters and handwritten lines. The vertices in the area are moved along the plane normal depending on the grayscale value at the corresponding position on the canvas. Make the faces of the area finer in advance with a remesh command or the like.

Use the mouse to move, rotate, and scale the characters and lines in the canvas. "Translate / Rotate" and "Scale" can be switched with the radio button. In the translate / rotate state, drag the arrow with the mouse to translate the image. Drag the ring to rotate it. In the scaling state, drag the bar to scale in that direction. (See the figure below)



If checked, a grayscale color according to the darkness as pasted as a texture. The background color of the canvas is white.

**Stamp** How to use

Specify an area where figures drawn through the use of the Canvas function are to be embedded. ?

Rectangular

Set/Cancel Embedding Areas ?

**Map the Embedding Area**

Canvas ?

Thin  Thick

Low  High

**Stamp** How to use

Specify an area where figures drawn through the use of the Canvas function are to be embedded. ?

Rectangular

Set/Cancel Embedding Areas ?

**Cancel Mapping**

Set the Embedding Area Translating Mode. ?

Translate/Rotate  Scale

Canvas ?

Thin  Thick

Low  High

Reverse the extrusion direction ?

Extrusion Amount ?

Paste image as texture ?

[When entering characters]  
Click the **A** button, then click where you want to write on the canvas. A text box is displayed. Enter a character string and change the font size and font type next to the **A** button if necessary. Press the "Enter" key on your keyboard to display the mesh with the characters added.

[When drawing a line]  
You can draw a line by holding down the left mouse button and dragging to the canvas while the **A** button is not pressed.

The thickness of the line to draw. Please adjust before drawing.

The darkness of characters and lines. The darker it is, the larger the unevenness.

If unchecked, a convex pattern will be created, and if checked, a concave pattern will be created.

White to black colors are pasted as textures according to the darkness ("Thickness"). The background color of the canvas is white.

Click to display the handle and change to the command dialog on the right.

# Letters Relief

The command embosses letters in a specified area of mesh, even if the area is bent.



Specify an area where letters are to be embedded.  
The area type is rectangular only.  
The system aligns letters horizontally/laterally on the display.

**Letters Relief** 📄 使い方の説明

Specify the area to embed letters. ?

Rectangular

---

Font Size

Fit to area  Specify

---

Letter String ?

Font MS Gothic

---

Protrusion Amount ?

0.1

---

Direction of Protrusion ?

View  Average Normal  X  Y  Z

Front side  Rear side

---

**Finalize**

Input a string of characters to be embedded and press the key "Enter".  
Letter fonts can be chosen, as well.

Specify a protruding length.

Input parametric data in order to display a temporary figure that indicates a form of letters to be embossed. Check the temporary figure shape and click the button "Finalize".

Choose "Fit to area" in order to create a pattern of characters composed of the area-long maximum letters.

Choose "Specify" in order to show a field where a height of letter (or longitudinal letter size) is input. The system creates a pattern of letters according to a specified letter size.

Specify a direction of protrusion.

Choose protruding to the obverse (convex pattern) or protruding to the reverse (concave pattern).

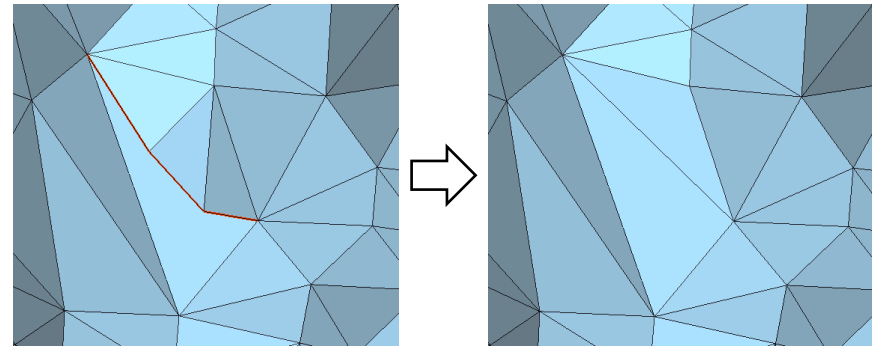
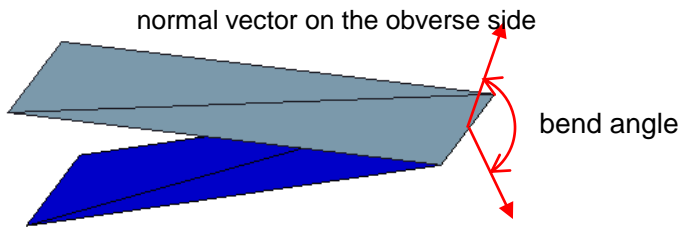
# 14. Beta version Tab

# Creased Faces Removal

This command detects a crease (where two faces make an angle) more acute than a specified angle, and adjusts it through transformation if the crease is curable.

The command recognizes a concave fold (valley fold) and a convex fold (mountain fold), and displays temporary figures of concave fold in blue or convex fold in red. It also displays the quantity of edges thus detected in a command dialogue.

The command is effective in detecting and curing folds that are found in a mesh made of measurement values.



Specify as subject to detection and correction, concave fold, convex fold or both.

**Creased Faces Removal** [How to use](#)

Crease Types  
 Concave  Convex  Both

Crease Angle [?](#)  
0 <  < 180  
150

Qty of edges of crease:  
[Display Result List](#)

[Detect](#) [Correct](#)

Detect and correct edges that are more acute than a specified value.

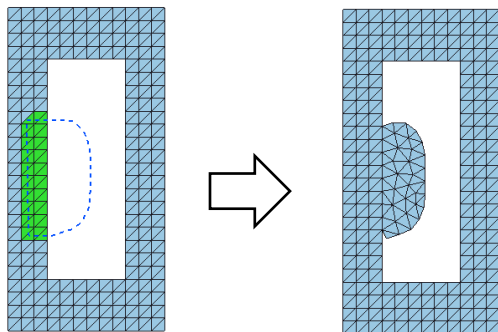
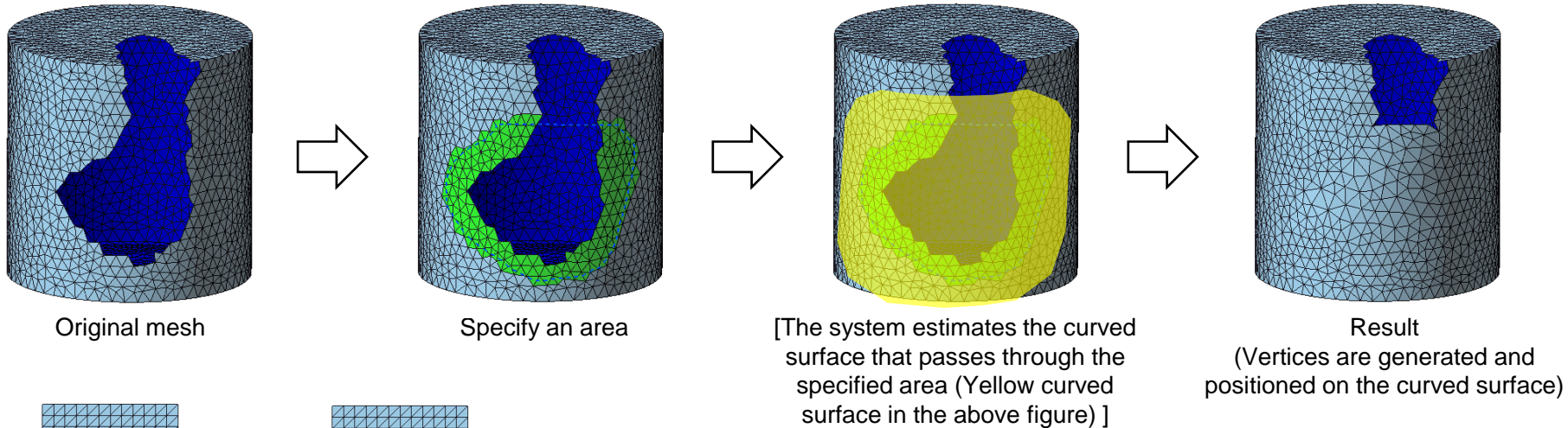
Display the quantity of detected edges/creases. The quantity is updated after correction.

Click the button in order to perform processes for 'detection' and 'correction' for a fold.

# Fill Region

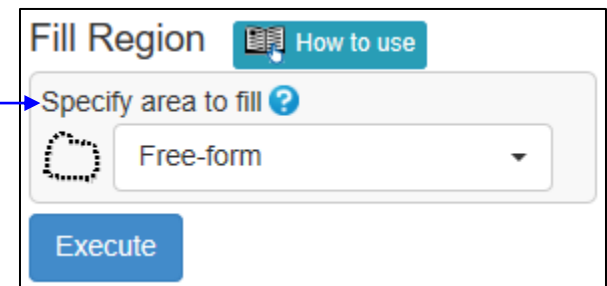
This command estimates the curved surface that passes through the shape around the hole and creates faces that fill the hole so that the faces ride on the estimated curved surface. Since the curved surface that fills the hole is created from the shape around the hole, it fits in the surrounding area.

The range in which the surface to fill the hole is created is determined by the trajectory of dragging the outside of the mesh (inside the hole).



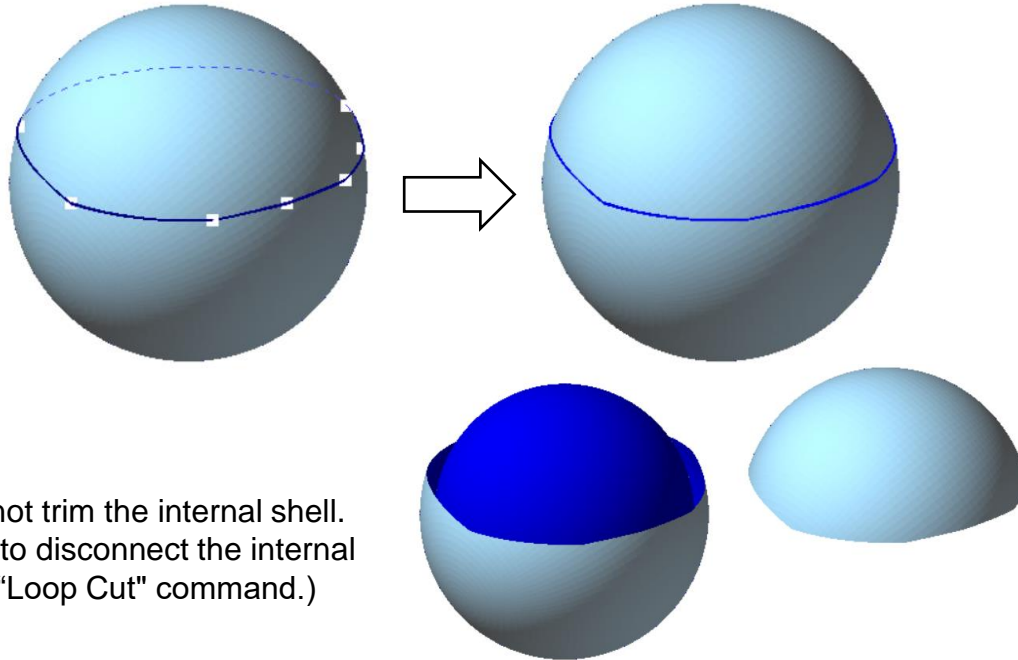
The range to create faces is determined by the trajectory of dragging outside the mesh.

Specify the area around the hole including the hole by dragging.



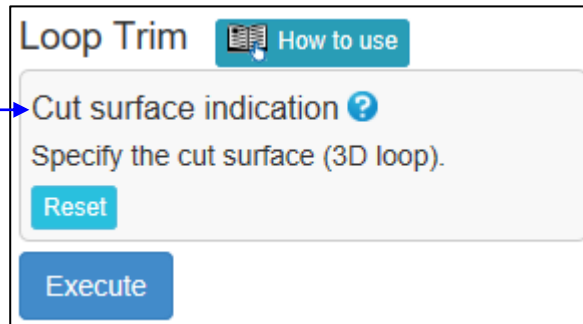
## Loop Trim (1)

Divide the mesh by the drawn closed lines.



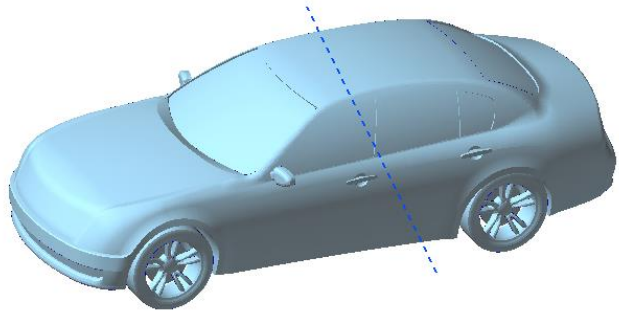
Loop trim does not trim the internal shell.  
(If you also want to disconnect the internal shell, use the "Loop Cut" command.)

Draw a line to specify a cut surface. An operation example is explained on the next page.

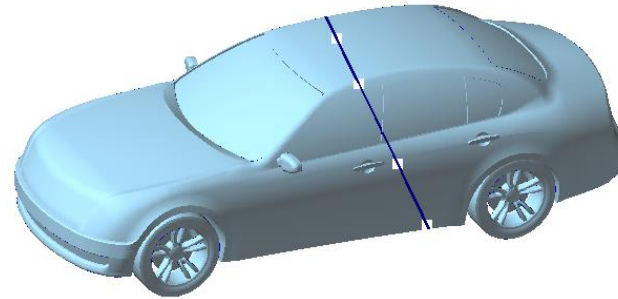


## Loop Trim (2)

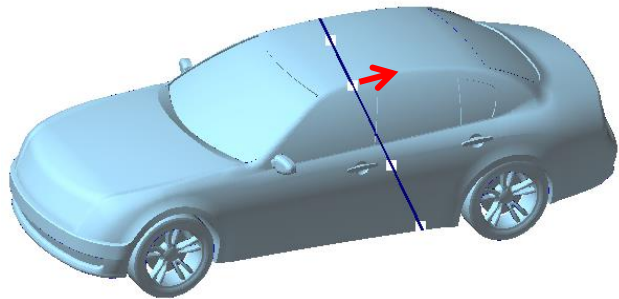
An operation example for creating a "cut surface".



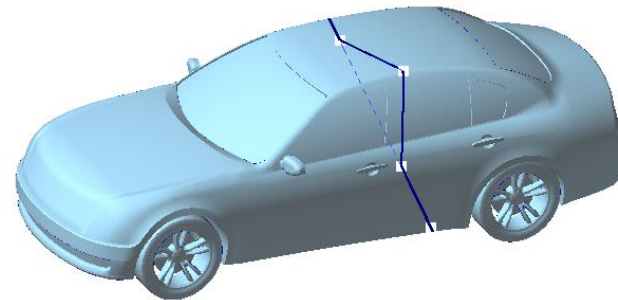
Drag the mouse to draw a straight line.



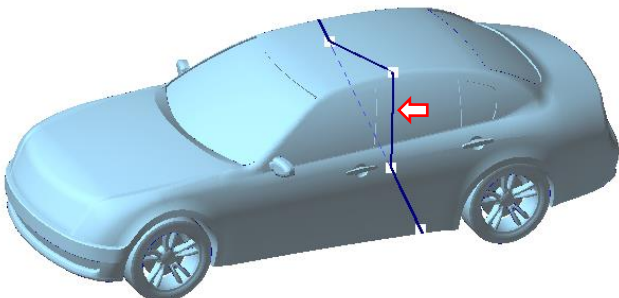
The initial Loop Polyline is created.



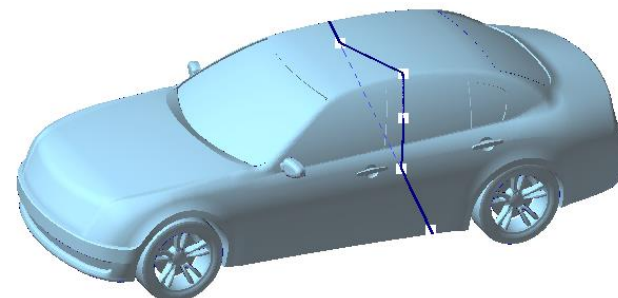
Grab a Control Point and drag it.



The Control Point is relocated to transform the Loop Polyline.



Click on the Loop Polyline.

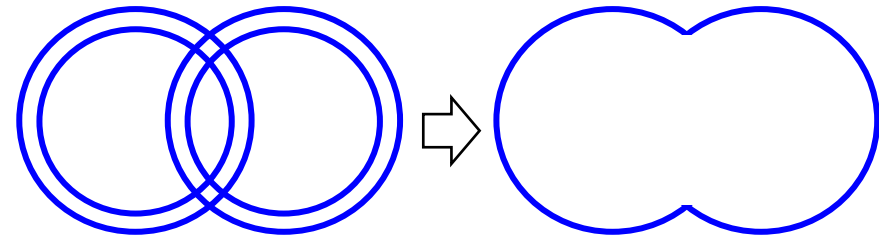
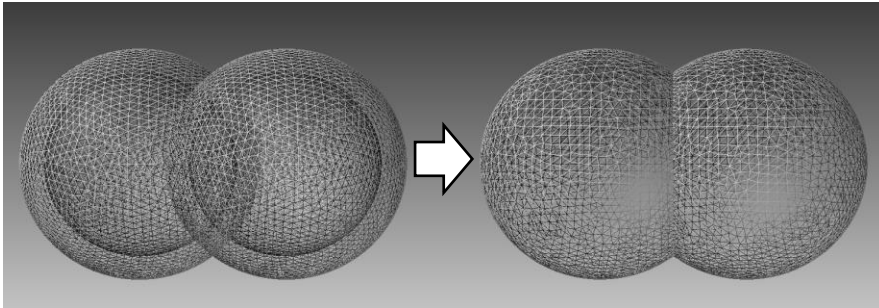


A Control Point is added.

# Wrapping

This command takes only the outside of the mesh and creates a new mesh.  
This command assumes that the mesh is solid.

An example of extracting only the outside from a mesh consisting of two hollow spheres.



**Wrapping** [How to use](#)

Edge Length ?  
0.09

Reproduce sharp edges

Run cleaner as post process

Execute

Guideline for the average edge length of the mesh created by wrapping.

Wrapping may round sharp edges.  
If "Reproduce sharp edges" is checked, the ridgeline will be made to be a sharp edge.

If checked, the mesh will be cleaned automatically after wrapping.



# 15. Help Tab

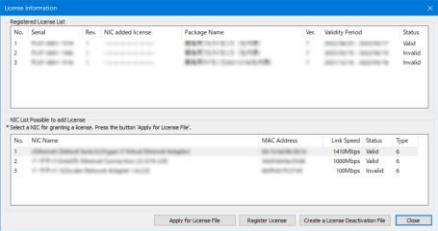
# Website

Show the POLYGONalmeister webpage.  
The Japanese web page will be displayed, so switch to English with the button at the top right of the screen.



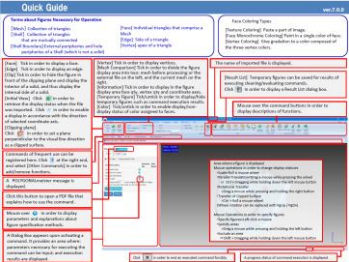
# License

Register the POLYGONalmeister license.



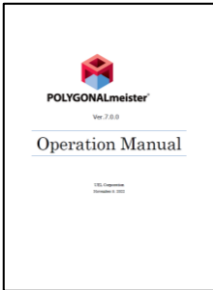
# Quick Guide

Show the minimum information required to operate POLYGONalmeister.



# Manual

Show the operation manual for POLYGONalmeister.



# License Guide

Show the POLYGONalmeister license guide.

