



3DxSUITE Editor

Tutorial -Standard function-

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Elysium Co. Ltd.

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Shortened Names for 3DxSUITE Products

In this document, the 3DxSUITE product names are referred to as follows:

- 3DxSUITE Components → Components
- 3DxSUITE Viewer → Viewer
- 3DxSUITE Editor → Editor
- 3DxSUITE SmartLauncher (Standalone) → SmartLauncher (Standalone)
- 3DxSUITE SmartLauncher (Plug-in) → SmartLauncher (Plug-in)
- 3DxSUITE SmartController → SmartController
- 3DxSUITE SmartController Pro → SmartController Pro
- 3DxSUITE TransServer → TransServer
- 3DxSUITE WorkerNode → WorkerNode
- 3DxSUITE ScenarioEditor → ScenarioEditor
- 3DxSUITE Data Package Studio → Data Package Studio
- 3DxSUITE Validation Configurator → Validation Configurator
- 3DxSUITE PDQ Checker Configurator → PDQ Checker Configurator
- 3DxSUITE Setting Utility → Setting Utility

1. Preface

1.1. About this Tutorial

This tutorial consists of four parts: Beginner, Intermediate, Advanced, and More Advanced. It is designed to help you learn how to use Editor step by step. Each content will be explained in detail.

2, *Beginner*

This level explains the basic operation for translating data in Editor. If the quality of data are not so bad, translation can be done just with the knowledge of Beginner level.

3, *Intermediate*

This level explains how to correct errors by using Interactive Healing. Most of the data can be translated properly with the knowledge of Intermediate level.

4, *Advanced* / 5, *More Advanced*

For errors which could not be corrected with the knowledge of Intermediate level are explained in the form of case studies. Please refer to these case studies whenever necessary.

Furthermore, the functions described in this tutorial are just a part of Editor. Please refer to the help for other functions.

About Help


For help, select [Help] > [Help Index] from Editor menu. The help provides details about the content, how to operate, options, and things to keep in mind.

Another way to open the corresponding page of help, select [Help] > [Context Help], and a question mark appears next to the cursor so either double-click the menu or just click the icon.

1.2. About the Notations of Menus and Icons

Each menu item button or dialog is represented by [Menu Name] and icon image. Right angle bracket (>) is used in sub menu.

Example:

For example, the "Fit" function is described as [View] > [Fit] ().

In this tutorial, the folder containing sample data is referred to as <tutorial>.

1.3. About Explanatory Notes

Explanatory notes are categorized as follows:



Contains important information. Be sure to check and set this item.



Warnings are listed. Be sure to check these items.



This information is useful when using the product.



Contains information that can be used for reference.

1.4. About Sample Data

The sample data to be used is located in the folder " \document\tutorial_models\standard" inside the folder where Editor is installed.

1.5. About Images

The images in this document may include slight differences from the ones actually displayed on your Editor depending on your specific computer hardware and the Editor version.

2. Beginner

This section will explain the basic operation for translating data in Editor.

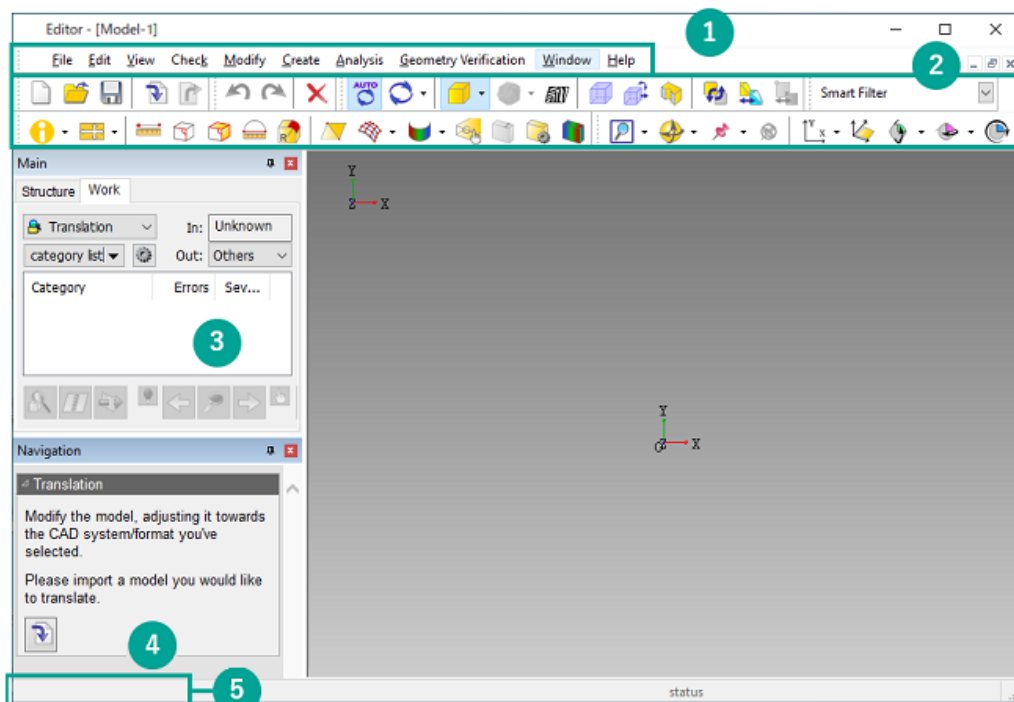
If the quality of the data are not so bad, translation can be done just with the knowledge of Beginner level.

Firstly, this chapter will explain about "Basic Operation" and "File Management." Then to [2.3, "General Data Translation"](#) which is about the basic translation method, and [2.3, "General Data Translation"](#) which compares before and after modification of geometries.

2.1. Basic Operation

This section will explain the basic operation of Editor.

2.1.1. User Interface

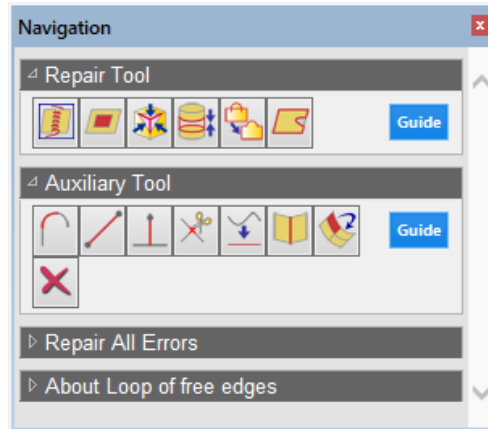


① Menu bar	Displays various commands.
② Toolbar	Displays frequently used icons from the menu bar.
③ Main panel	Used mainly for basic operations. Run [Check] and [Auto Heal] for check items displayed in the list.
④ Navigation panel	Displays the most suitable tool to modify the check item. Also, when switching to [Fit], the fitting tools will be displayed.
⑤ Status bar	Displays information such as the current status and a guide to the next operation.
⑥ "3D View" window	Displays the imported model.

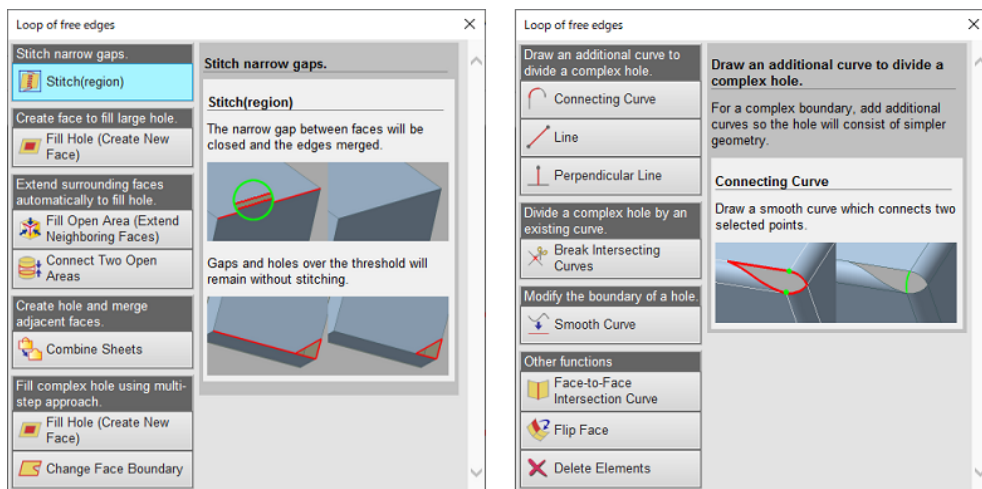
■ Navigation / Guide

Advises the best way to modify the error items.

- Navigation



- Guide (Repair Tool / Auxiliary Tool)



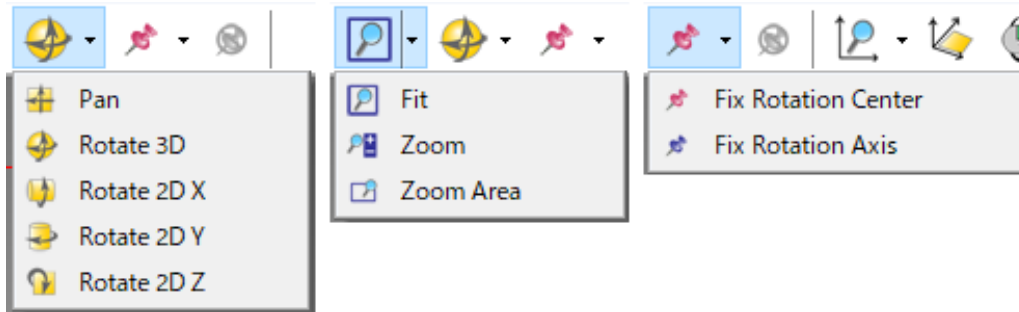
The operation procedure is as follows.

1. In [Main (Work)] panel, select the error item from the category list.
2. The best modification method is displayed on Navigation panel. Press and use the appropriate one.
 - Repair Tool: Effective for error modification.
 - Auxiliary Tool: Creates reference lines, etc. needed for repair tools.
 - Guide: Explains in detail Repair Tool / Auxiliary Tool on the Navigation panel.

2.1.2. View Operation

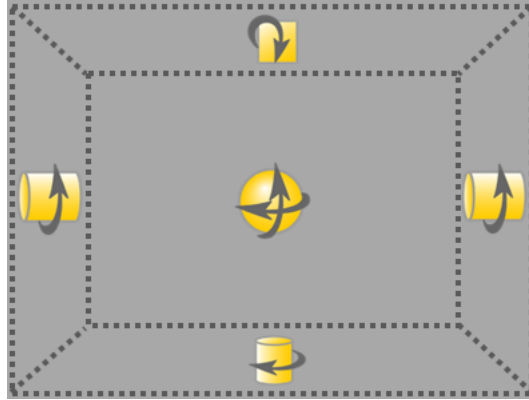
■ Mouse Operation

This section explains how to use the mouse on "3D View" window.



Command	Description	Mouse / Keyboard Shortcut
Pan	Move the model parallel.	Right mouse button + drag
Rotate 3D	Rotate the model in 3D according to the mouse movement.	Left mouse button + drag
Rotate 2D X	Rotate around the horizontal axis of "3D View" window.	(Left and right of the screen) Left mouse button + drag
Rotate 2D Y	Rotate around the vertical axis of "3D View" window.	(At the bottom of the screen) Left mouse button + drag
Rotate 2D Z	Rotate around the depth axis of "3D View" window.	(At the top of the screen) Left mouse button + drag
Fit	Automatically adjust all elements to fit in "3D View" window.	Ctrl + F
Zoom	The view is zoomed in and out. (When zoomed in, the vertical height doesn't change, but moves in the depth direction and the opposite direction.)	<ul style="list-style-type: none"> • Mouse scroll wheel • Middle mouse button + drag • Both mouse button + drag
Zoom Area	Zoom into the area enclosed by a rectangle.	None
Fix Rotation Center	Set the specified point as the center of 3D rotation.	Ctrl + Q
Fix Rotation Axis	Specify the rotation axis when rotating the plane.	None

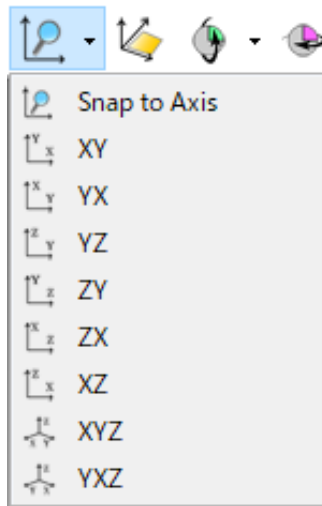
Please note that the rotation method in "3D View" window will change depending on where you drag from.



- Drag at the top of "3D View" window: Rotate 2D Z (🔄)
- Drag at the bottom of "3D View" window: Rotate 2D Y (🔄)
- Drag at the left of "3D View" window: Rotate 2D X (🔄)
- Drag at the right of "3D View" window: Rotate 2D X (🔄)
- Drag at the center of "3D View" window: Rotate 3D (🔄)

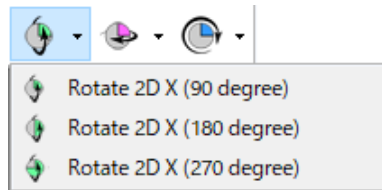
■ Change View

This section will explain how to change the view using the toolbar icons.

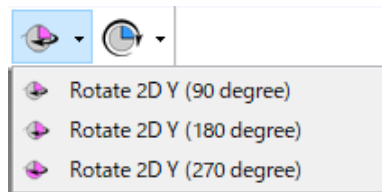


Command	Description
Snap to Axis	Change the current view to the closest basic axis view, then fit the view.
XY	Change the viewing angle toward XY
YX	Change the viewing angle toward YX
YZ	Change the viewing angle toward YZ

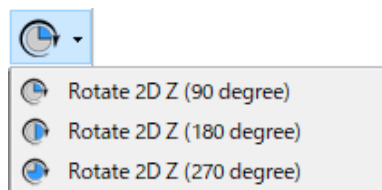
Command	Description
ZY	Change the viewing angle toward ZY
ZX	Change the viewing angle toward ZX
XZ	Change the viewing angle toward XZ
XYZ	Change the viewing angle toward XYZ
YXZ	Change the viewing angle toward YXZ



Command	Description
Rotate 2D X (90 degree)	Rotate 90 degrees vertically.
Rotate 2D X (180 degree)	Rotate 180 degrees vertically.
Rotate 2D X (270 degree)	Rotate 270 degrees vertically.



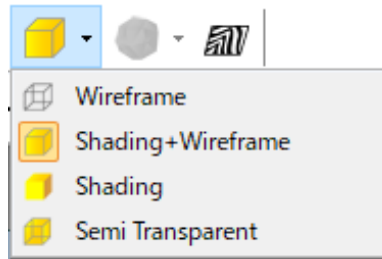
Command	Description
Rotate 2D Y (90 degree)	Rotate 90 degrees horizontally.
Rotate 2D Y (180 degree)	Rotate 180 degrees horizontally.
Rotate 2D Y (270 degree)	Rotate 270 degrees horizontally.



Command	Description
Rotate 2D Z (90 degree)	90 degrees plane rotation.
Rotate 2D Z (180 degree)	180 degrees plane rotation.
Rotate 2D Z (270 degree)	270 degrees plane rotation.

■ Display Type

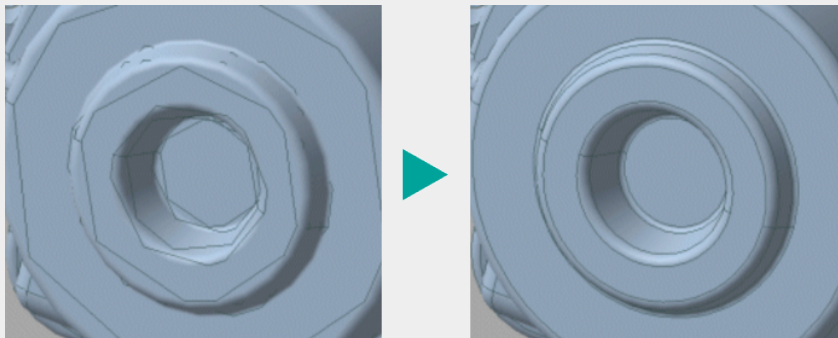
This section will explain how to change the display type using the icons on the toolbar.



Command	Description
Wireframe	Display elements in wireframe mode.
Shading + Wireframe	Display elements in shading and wireframe mode.
Shading	Display elements in shading mode.
Semi Transparent	Display elements in semi-transparent mode.

When "3D View" window is coarse

Zooming in and out of "3D View" window may result in a coarse appearance of the model. In such a case, select [View] > [Regen] from the menu or click [Regen] (🔄) on the toolbar. "3D View" window will be updated.

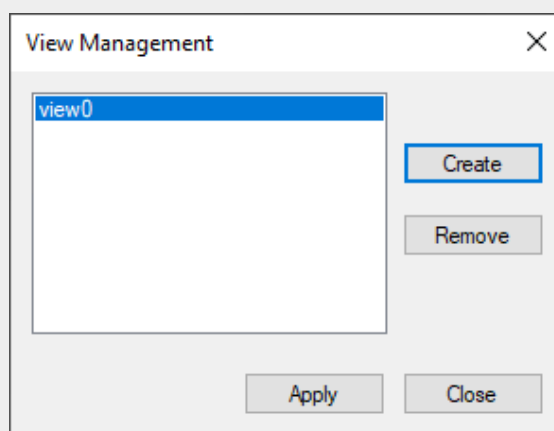


Saving View Direction

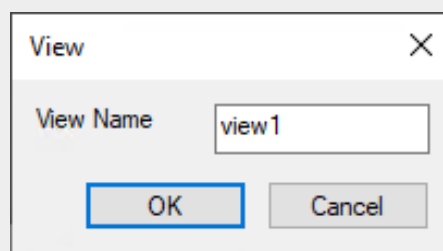
With Editor View Configuration function, it is possible to register any view direction and display it again later. View information is held by Editor, not by each model. Therefore, the registered view direction can be shared between different models.

■ How to Save View Direction

1. Display the model by the view direction that you want to save.
2. Select [View] > [Change View] > [View Config] from the menu.
3. "View Management" dialog will appear. A list of previously saved view names is displayed. Click [Create] to save new view.



4. "View" dialog will appear. Enter the name to register, then click [OK]. The newly created view direction is registered.



■ Redisplay with the Saved View Direction

1. Display "View Management" dialog in the same way as you did in "Saving View Direction".
2. Select the preferred view to display from the list and click [Apply]. Selected view information is reflected on the screen.

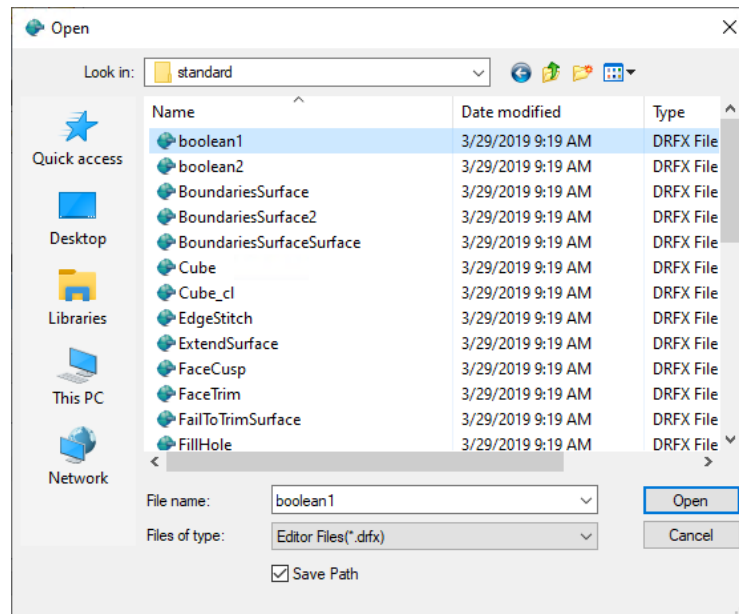
If you save the errors in Editor with this function, the viewpoints can easily be redisplayed for checking after modification.

2.2. File Management

2.2.1. Open Files

Open .drfx file that contains the working state of Editor.

1. Select [File] > [Open] from the menu or click [Open] (📁) on the toolbar.
2. "Open" dialog will appear. Specify .drfx file and click [Open].

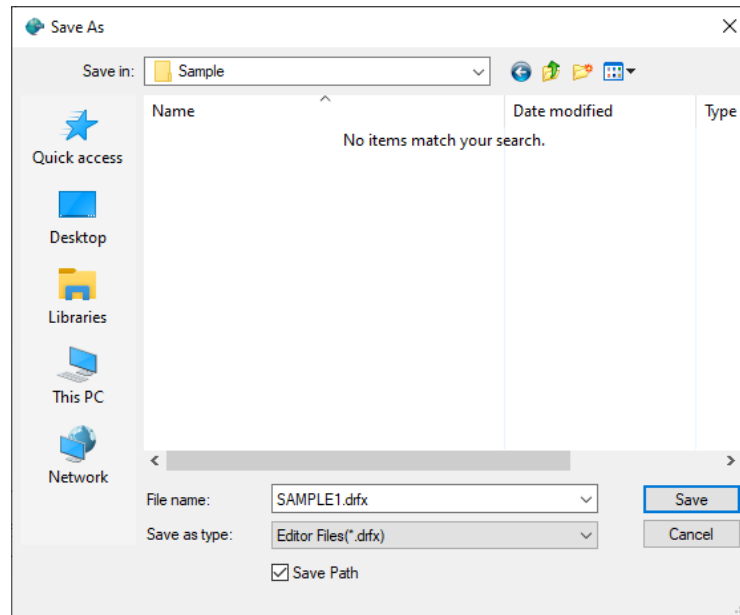



The sample data used in the tutorial is located in "document" > "tutorial_models" folder inside the folder where Editor is installed.

2.2.2. Save Files

With Editor, the current working state can be saved as a .drfx file.

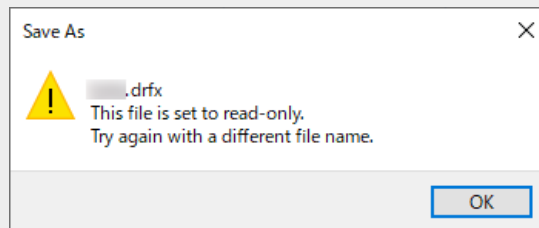
1. Select [File] > [Save As] from the menu.
2. "Save As" dialog will appear. Specify the location and file name to save, and then click [Save].



When editing an existing .drfx file, click [Save] () to overwrite the currently opened .drfx file.

About the sample data in the tutorial

The sample data in the tutorial are "read-only" and cannot be saved.



To save the sample data you are editing, select [Save As] to save it under a different file name.

2.3. General Data Translation

When translating data, errors caused by the differences in tolerance and data structure between the source and target CAD systems need to be corrected.

With Editor, you can correct these errors and translate the data with the following operation flow.

1. Import file from source CAD system
2. Check and detect errors that may cause problem during translation
3. Correct the detected errors
 - Auto Stitching
 - Auto Healing
 - Interactive Healing
4. Export file to target CAD system

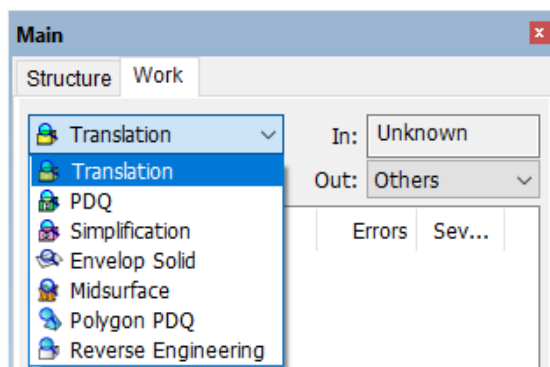
Beginner level follows the above operation flow to translate the sample data.

2.3.1. Mode

Editor has multiple modes that can be switched depending on your purpose. The best choice for data translation, for example, is "Translation" mode. Modes can be switched on [Main] panel.

In this case, use "Translation" mode which is most suitable for data translation.

1. Click "Translation" from the drop-down list at the upper left of [Main (Work)] panel.




Please note that each mode requires an optional license when switching to the mode, for example, Simplification mode.

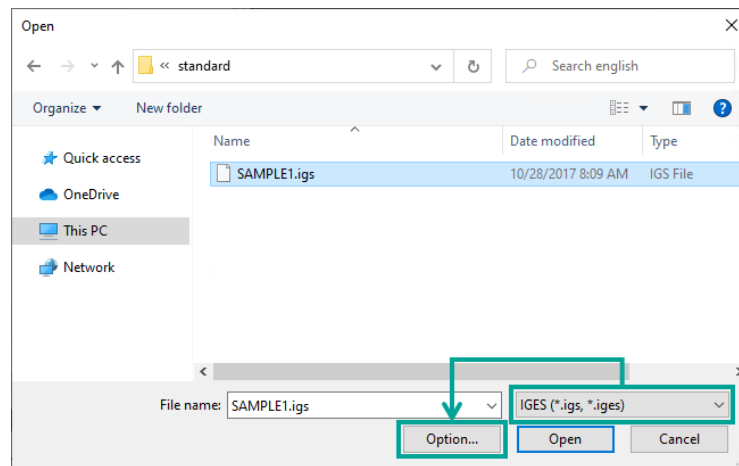
2.3.2. Import

Import the sample model (SAMPLE1.igs) into Editor.

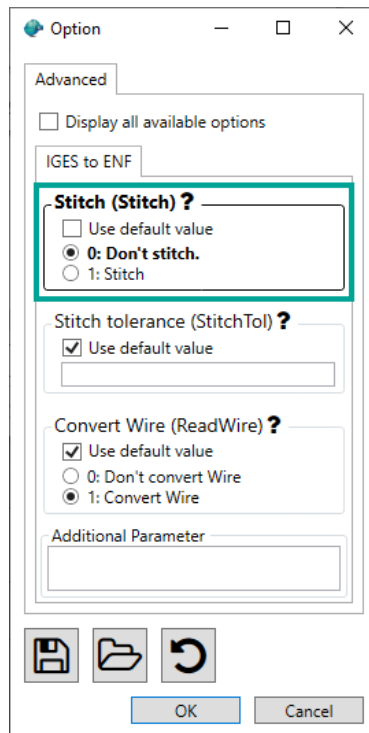


In some parts of this manual, IGES format files are used to explain the procedures. To import IGES format files, a dedicated license is required. If you do not have an IGES license, open a file with the same file name but with a .drfx extension.

1. Select [File] > [Import] from the menu or click [Import] () on the toolbar.
2. "Open" dialog will appear. Change the type of file to "IGES (*.igs, *.iges)" and click [Option].



3. "Option" dialog will appear. In "Stitch (Stitch)?", enable "0: Don't stitch." and click [OK].





Please note that "Stitch" is disabled intentionally at this point because it is necessary in the operation explanation that follows. Normally, you do not need to disable "Stitch".

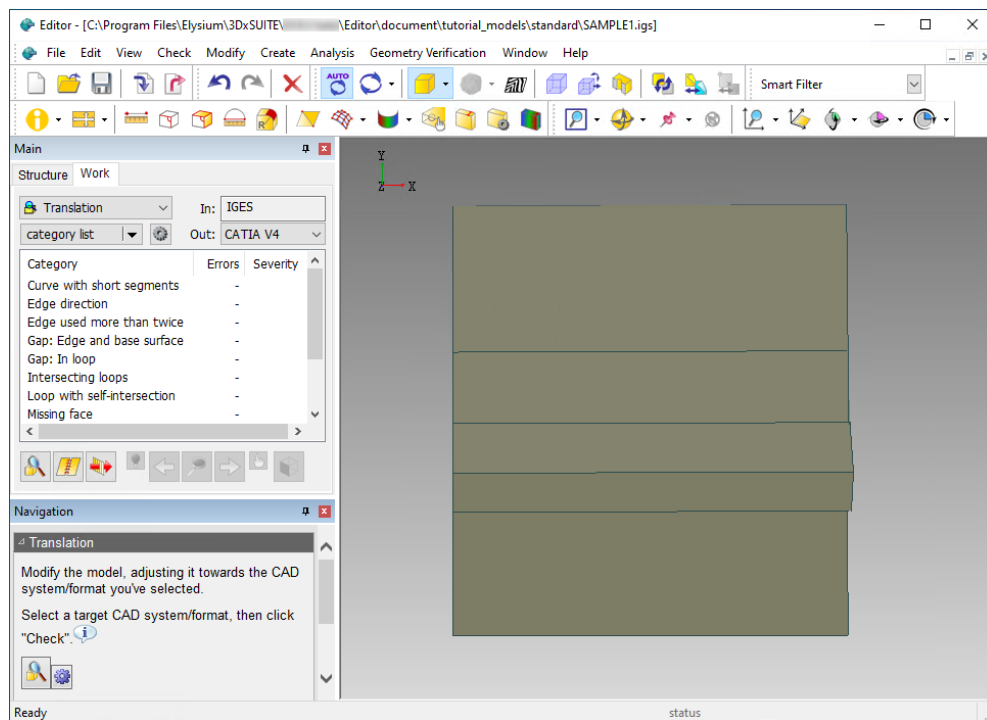


In "Open" dialog, click [Option]. Set input option in the format specified in "Files of type".



Please note that the type of files you can import depends on the licenses you have.

- "Open" dialog will appear. Specify **SAMPLE1.igs** file from the <tutorial> folder and click [Open]. Specified file is imported and displayed on "3D View" window.

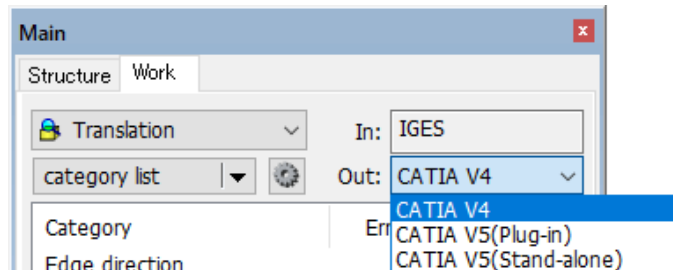


2.3.3. Data Check

This function checks where the errors exist in the imported data.

[1] Target CAD System

1. In [Main (Work)] panel, specify the output target in "Out". In this case, specify "CATIA V4".



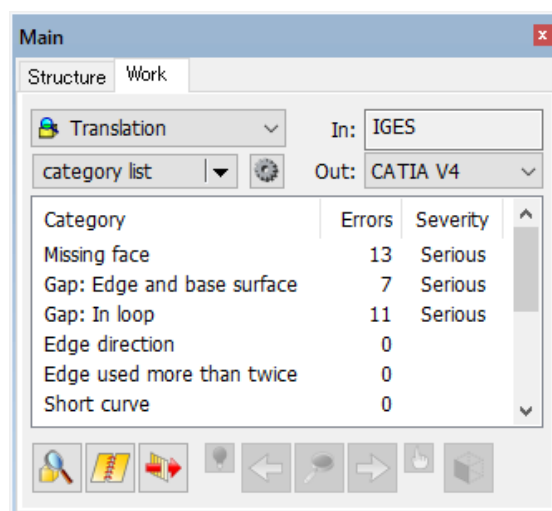
With Editor, it is recommended that you specify the target CAD system at the time of import. Main options are automatically set for the source / target CAD system at the time of import. Target CAD system can be changed before Auto Heal.



Please note that when importing an Elysium Neutral File (*.enf, *.drx, *.armo), the target CAD system may be selected automatically for you. In such a case, you do not need to explicitly specify the target CAD system here.

[2] Check

1. In [Main (Work)] panel, press [Check] (🔍).
Check is executed and the check result will be displayed.

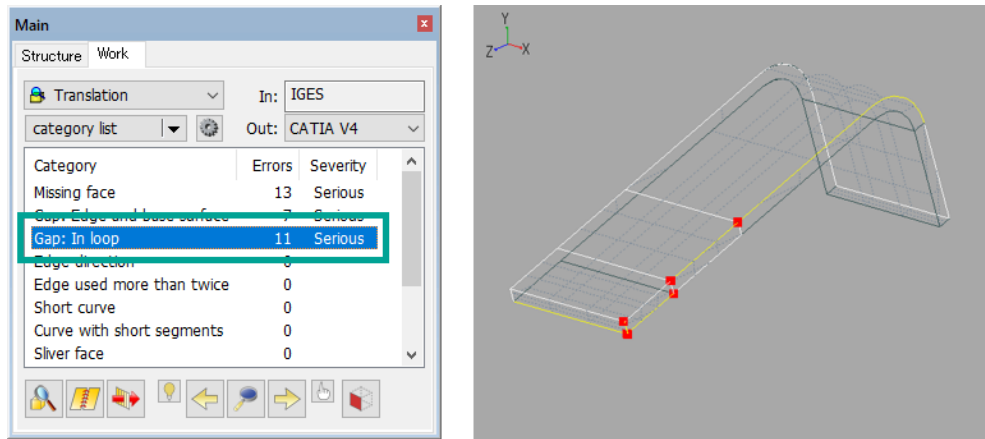


[3] Check for Errors

Confirm the detected errors in order.

1. In [Main (Work)] panel, select "Gap: In loop" in the category list.
The errors are highlighted in red on "3D View" window.

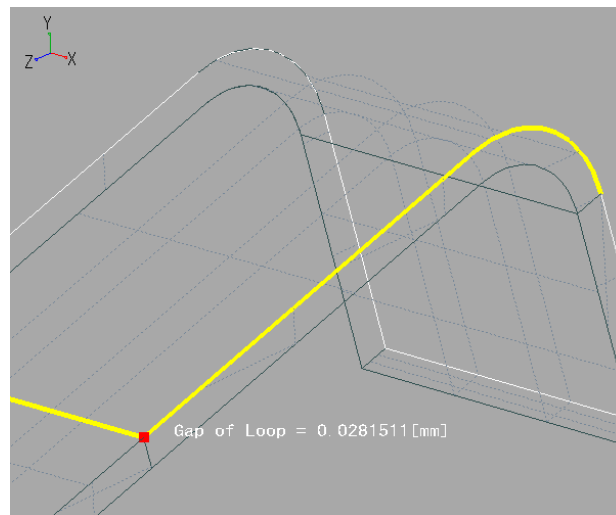
* To make the errors easier to see, the view has been switched to wireframe mode.



Please refer to [A.1, "Error Items"](#) in the Appendix for details about error items.

Confirm the detected errors one by one.

2. In [Main (Work)] panel, press [Zoom current target] ().
One of the errors is zoomed up and highlighted.




Press [Next] () to highlight the next error. Also, press [Previous] () to go back to the previous error.

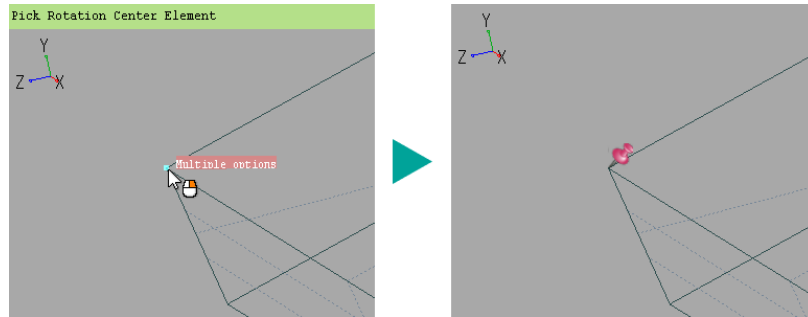


Use [2.1.2, "View Operation"](#) to make the errors easier to check.



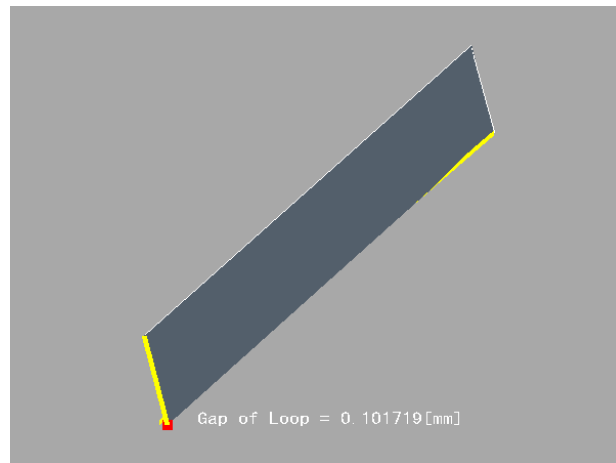
If you have difficulty moving the model in "3D View" window, you can specify the center position of 3D rotation.


Press [Fix Rotation Center] () and pick a spot for the center.
Alternatively, right-click on "3D View" window and specify from the context menu.

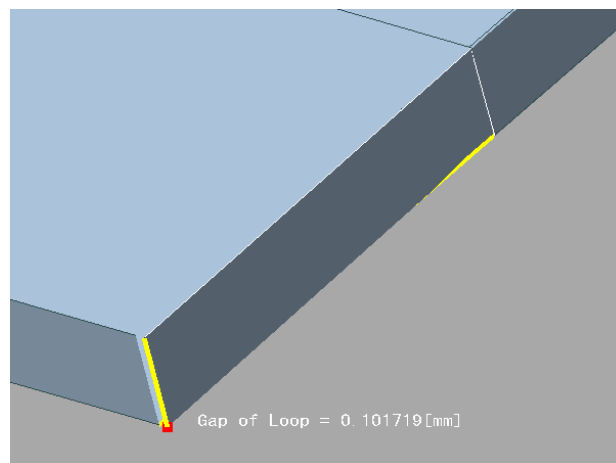


Press [Deactivate Rotation Setting] () to release the rotation center.


3. In [Main (Work)] panel, press [Show Surrounding] (). Only the elements related to the error are shown in "3D View" window.



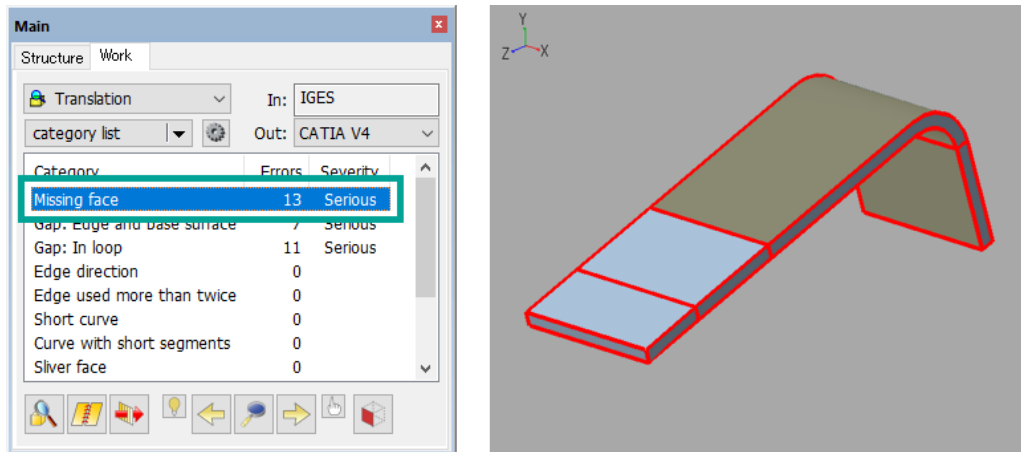
Again, press [Show Surrounding] () to undo. This function is useful when there are numerous elements and it is difficult to see the errors.




2.3.4. Auto Stitch

In [Main (Work)] panel, press [Auto Stitch] (). If there is an element in a model that does not have adjacency information (topology), the adjacent faces can be combined to have a topology. This operation is called "Stitch".

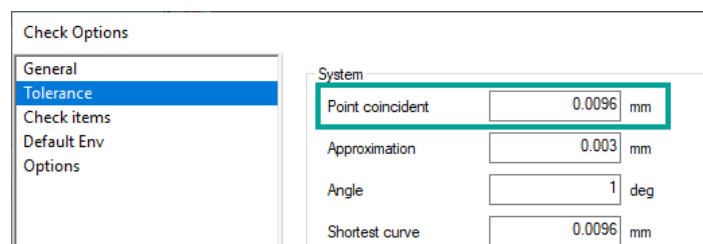
By stitching, "Missing face" can be modified.



1. In [Main (Work)] panel, press [Auto Stitch] ().
2. "Auto Stitch" dialog will appear. Because the default value will be used in this case, click [Try].

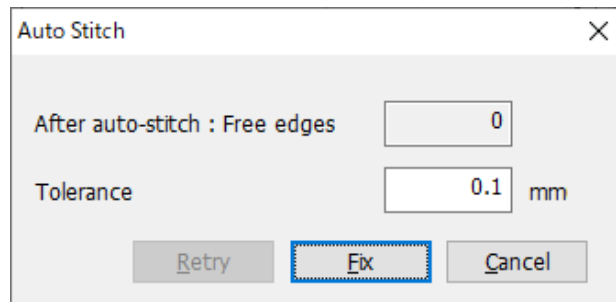


Please note that the tolerance described in IGES file will be displayed as the initial value in "Tolerance". For this sample model (SAMPLE1.igs), 0.1mm will be displayed. If the tolerance is not described in the model, the specified "Point coincident" value in "Check Options" dialog, which can be called from [Check] > [Options], will be displayed.



Tolerances specified here is used only for Auto Stitch. It will not affect other tolerances during modification or check.

3. The number of free edges is updated. In "Auto Stitch" dialog, confirm that the number of free edges is "0", then click [Fix].



If free edges still remain after clicking [Fix], loosen the tolerance and try again. However, note that if the tolerance is loosened too much, an error such as "Gap: Edge and base surface" may occur.



For topological reasons, there are models whose unused edge does not become "0". If so, check the location of the free edge and determine whether you need to retry.

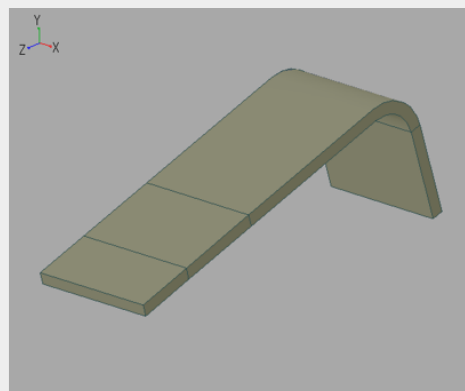
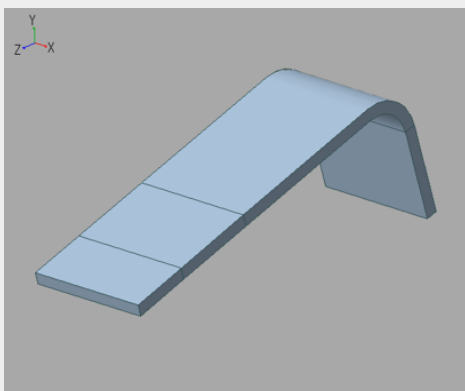
Please refer to 4.1, "Modifying Free Edge" for details about free edge.

Data is updated. At the same time, the check result in [Main] panel is updated.

Flip Face

Orientation of the faces are automatically consistent after stitching; however, in some cases, the front and back of the faces are reversed.

If the face is in gray, as shown in the lower left figure, the direction of the face is correct and no modification is necessary. If the face is in khaki, as shown in the lower right figure, the face is reversed.

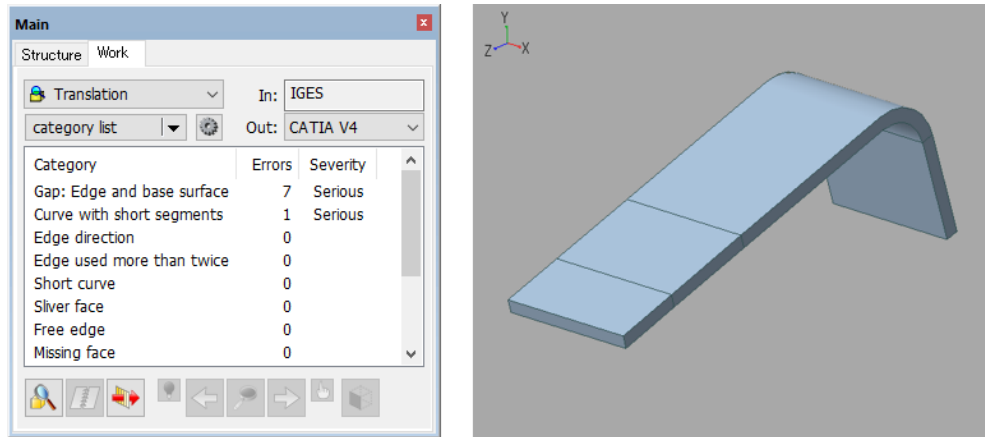


In such a case, select [Modify] > [Repair Solid] > [Flip face] from the menu or click [Flip face] (🔄) on the toolbar to correct the direction of the faces.

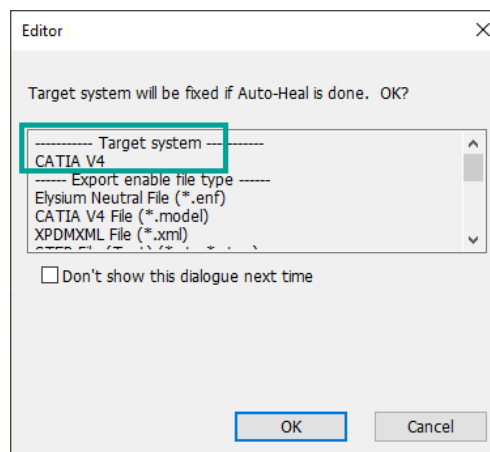
2.3.5. Auto Healing

This function can be used to automatically heal errors.

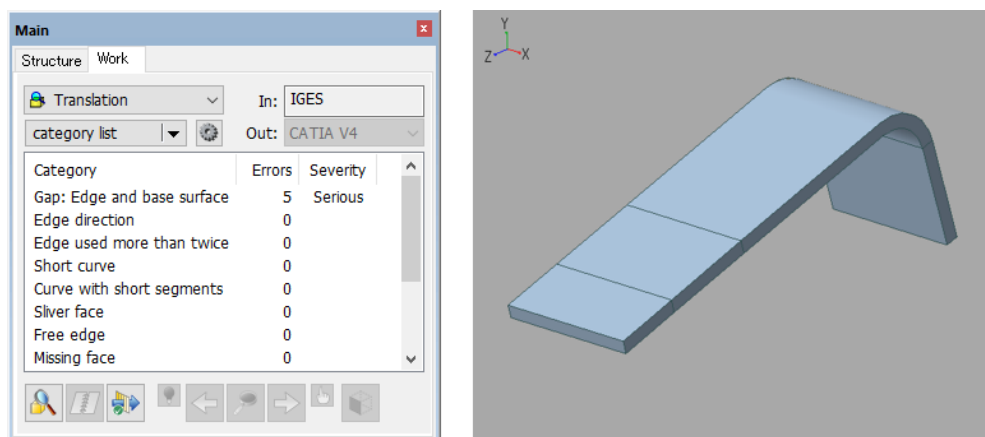
1. In [Main (Work)] panel, press [Auto Heal] (🔧➡️).



2. A confirmation dialog will appear. Confirm that the target CAD system is set properly, then click [OK].



Automatically heal a portion of the errors and the category list is updated.





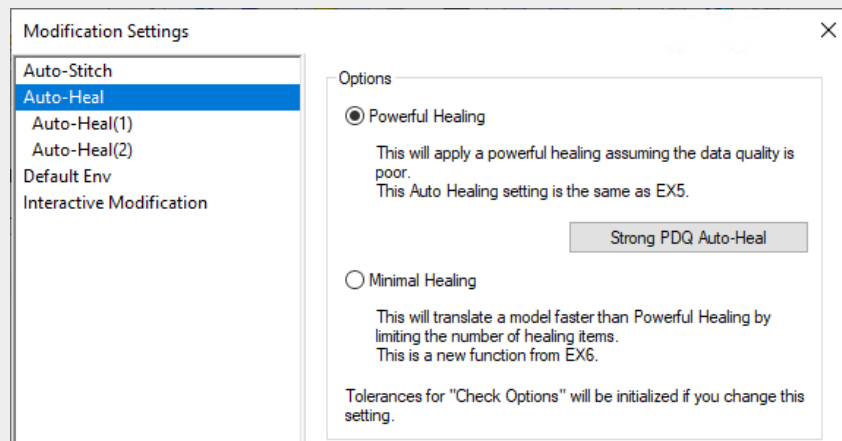
Please note that the icon color of [Auto Heal] will change from "before" (🔴🔵) to "after" (🟢🔵).

Options for Auto Heal

There are two options for Auto-Heal, "Powerful Healing" and "Minimal Healing".

- With Powerful Healing, the usual powerful healing is applied.
- With Minimal Healing, which can heal faster than Powerful Healing, the healing items are limited to ones that are required to keep the solid model.

Select [Modify] > [Options] from the menu. "Modification Settings" dialog will appear. Switch the setting in [Auto-Heal] tab.



2.3.6. Interactive Healing

When errors still remain after automatic healing, perform interactive healing to modify the errors.

"Serious" and "critical" level errors may cause missing faces, etc. during data translation.

In this case, the serious level error "Gap: Edge and base surface" will be modified.

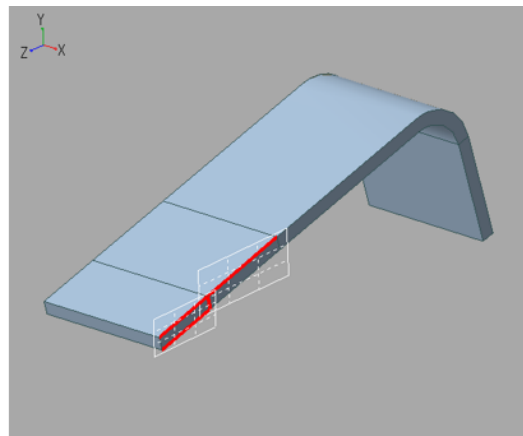
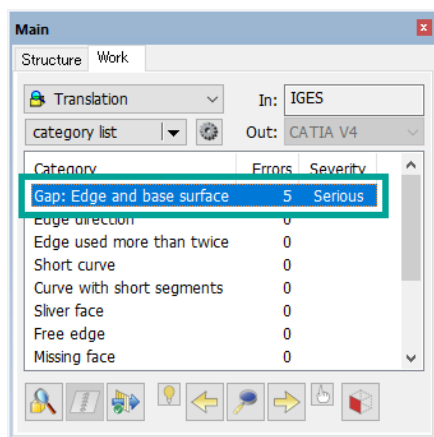


Depending on the source and target CAD systems, automatic healing may modify all errors. This is due to the difference in options, etc. If all errors have been modified, then skip to [2.3.7, "Export"](#).

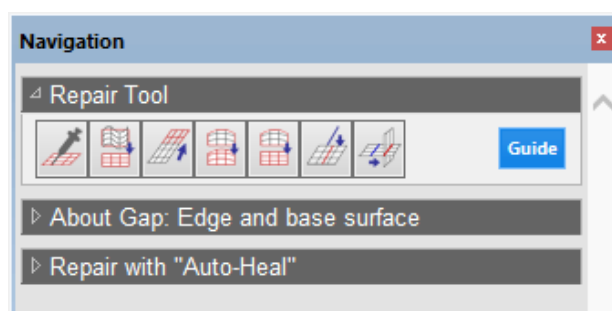
In some cases, errors with a severity of "Minor" may remain, which is a warning level error. You do not have to modify.

Please refer to [A.1, "Error Items"](#) in the Appendix for details about error items.

1. In [Main (Work)] panel, select "Gap: Edge and base surface" from the category list. On "3D View" window, the position of the error "Gap: Edge and base surface" is highlighted in red.

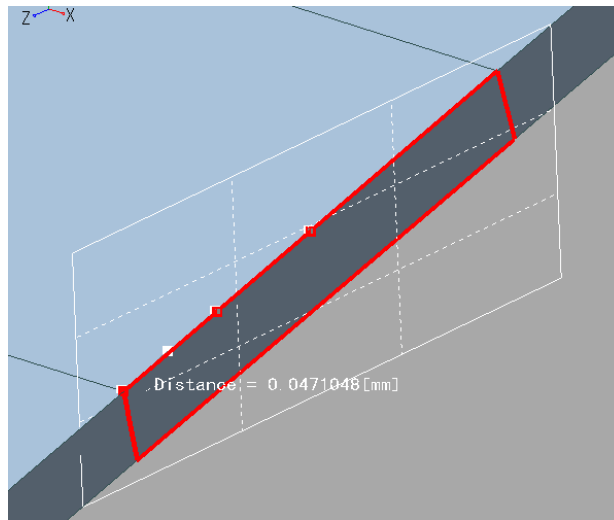


Also, Repair icons will appear on Navigation panel. These repair icons can be used to modify the error "Gap: Edge and base surface".



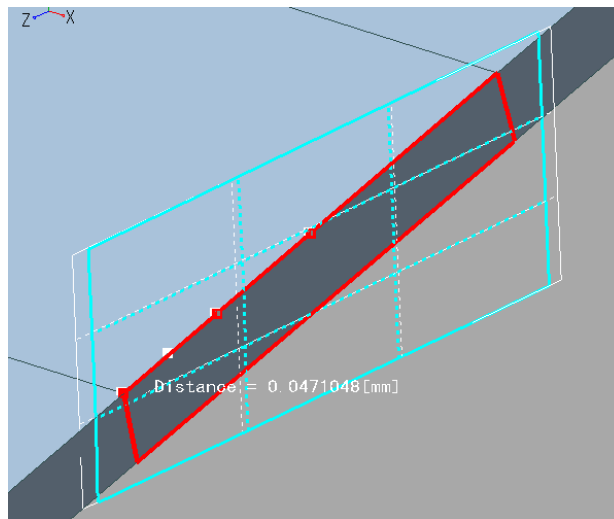
Repair icons are listed from the left side in the order of highest frequency of use.

2. In [Main (Work)] panel, press [Zoom current target] ().
On "3D View" window, the current target error is zoomed up.

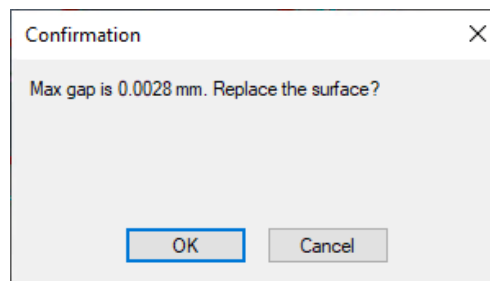


3. Press [Fit Face to Loops] () located at the farthest left of Repair Tool on Navigation panel.



A new surface is calculated, and the geometry of the surface is displayed and highlighted in light blue on "3D View" window.

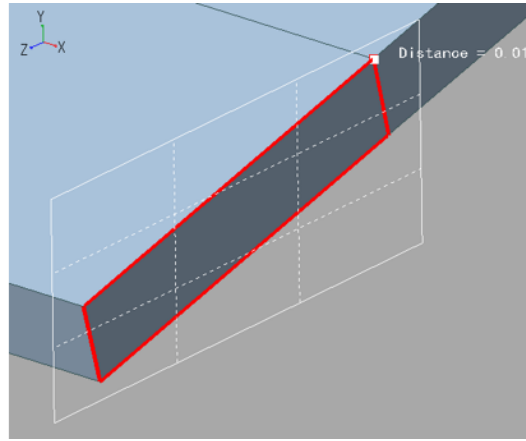
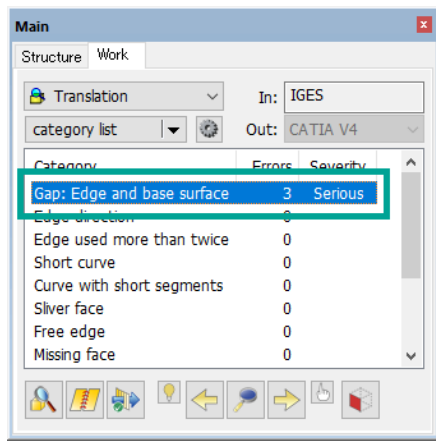


4. A confirmation dialog will appear. Click [OK].



The error is modified, and the category list is updated.

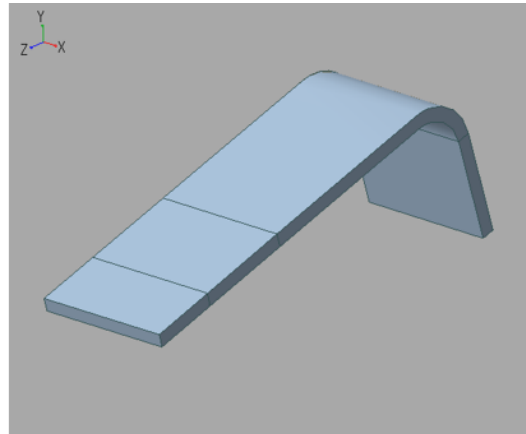
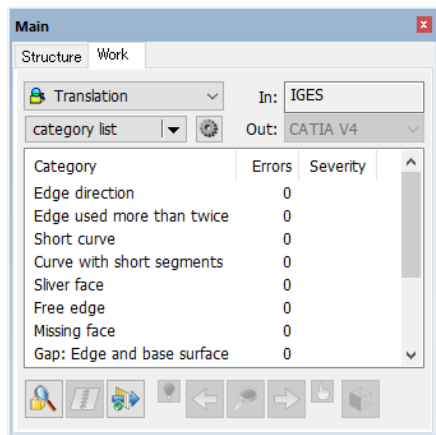
5. Again, press [Zoom current target] () to zoom up the next error.
Modify the remaining "Gap: Edge and base surface" errors with [Fit Face to Loops] ().



The function [Fit Face to Loops] () is highly versatile, and can be used to modify almost all "Gap: Edge and base surface".


Please refer to Intermediate: [3.1, "Interactive Healing"](#) for details about modification functions.

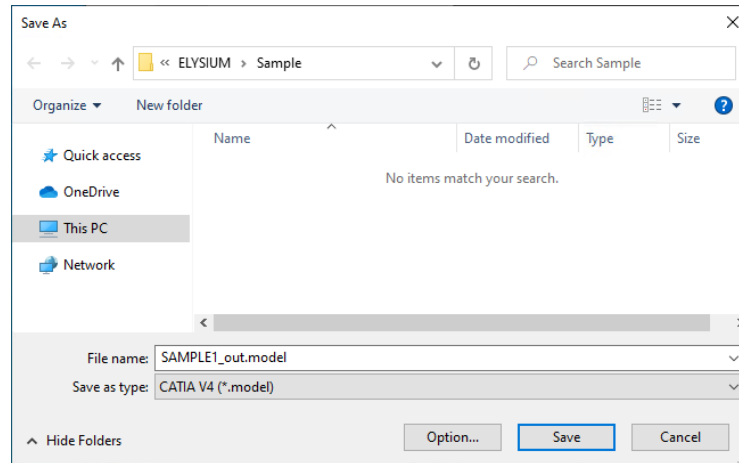
In [Main (Work)] panel, all errors are now "0" in the category list.



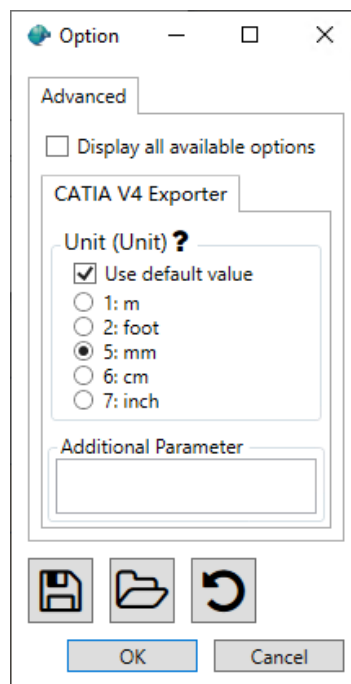
2.3.7. Export

In this section, the sample model (SAMPLE1.igs) will be output in CATIA V4 file format.

1. Select [File] > [Export] from the menu or click [Export] () on the toolbar.
2. "Save As" dialog will appear. Specify "CATIA V4 (*.model)" as the type of file. Enter the file name and location to save the file, then click [Save].



In "Save As" dialog, click [Option] to set the output options for the format specified in "Save as type".

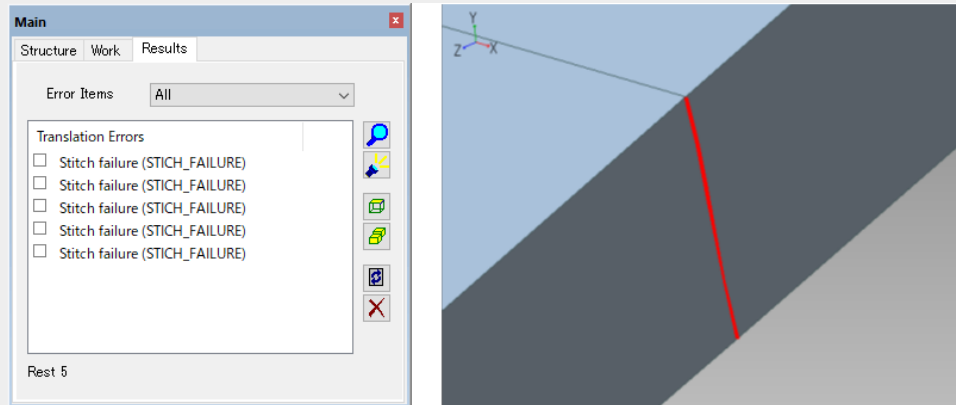


Please note that the type of file that can be exported depends on the target CAD system selected at the time of import and the license you have.

Check the result

When exporting, if your file type is either "CATIA V4" or "Parasolid", you can confirm the result from [Main (Result)] panel.

If the data cannot be translated into solid, the [Results] tab becomes active, and a list of translation errors will appear. (See below figure)



In such a case, confirm the areas around the translation errors on "3D View" window, and repeat interactive healing on the errors. In addition, the failed errors might only be identified on a per volume basis.

Nothing will appear in the list of translation errors if you were able to translate into a solid. The sample model used in this case also does not show any errors in the list; therefore, you can confirm that it was translated into a solid.



Please refer to the help for more details about Results tab.

Translating Large Files

When translating a large file, set the "Reduce Memory Mode". When this mode is set, a part of the information on memory is saved to a file, so memory usage can be suppressed.

■ How to Set Up

1. Select [File] > [System Preference] from the menu.
2. "System Preference" dialog will appear. In [Miscellaneous] tab, enable "Reduce Memory Mode".
3. In "System Preference" dialog, specify "Work" from [Path] tab.



Use "Reduce Memory Mode" even if "Work" is not specified, but in such a case, the default temporary area of Windows will be used. Therefore, it is recommended that you specify "Work".

4. Click [OK] in "System Preference" dialog. When changing the Work path, reboot the application to reflect the changes.



- Please note that a warning dialog might appear when Work folder has only 500MB or less storage capacity.

2.4. Geometry Verification Mode

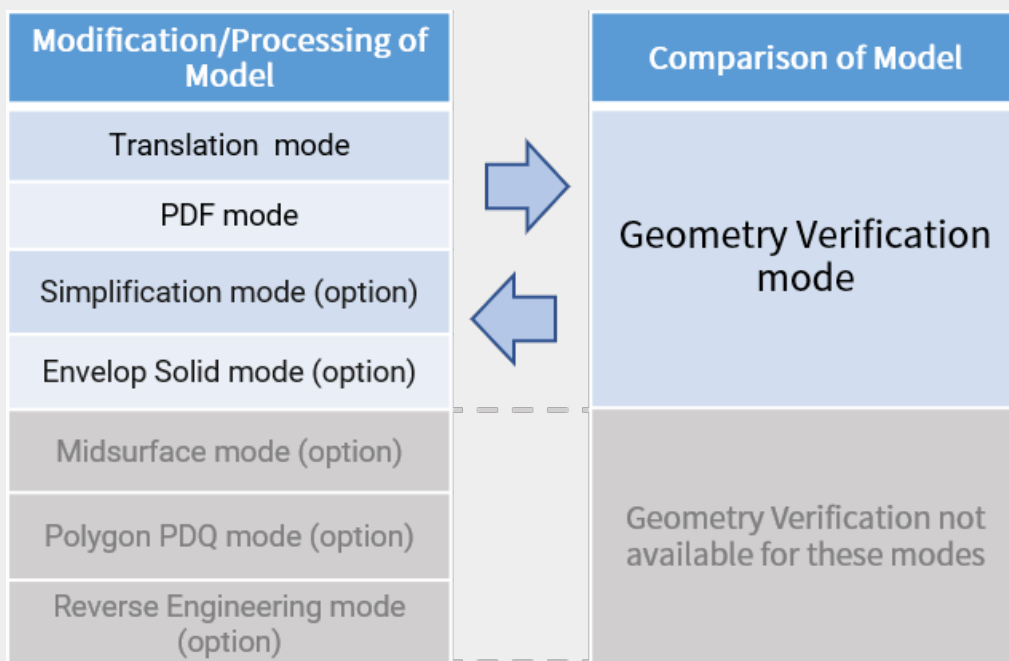
With Geometry Verification function, you can visualize the differences geometrically and topologically by comparing the two models in such ways as before and after data translation, and before and after modification.



Geometry Verification Mode, in which geometry comparison is performed, is treated as an independent function of Analysis menu. Therefore, operations that involve modifying elements or models cannot be done in Geometry Verification Mode. In addition, menus other than view operations to check the comparison results are restricted.

It is possible to switch to Geometry Verification Mode from any mode. Just select [Geometry Verification Mode] (🔍) from Geometry Verification menu.

Position of Geometry Verification Mode




Geometry Verification Mode has two functions, "Detailed Comparison" and "Quick Comparison", which can be used according to your purpose.

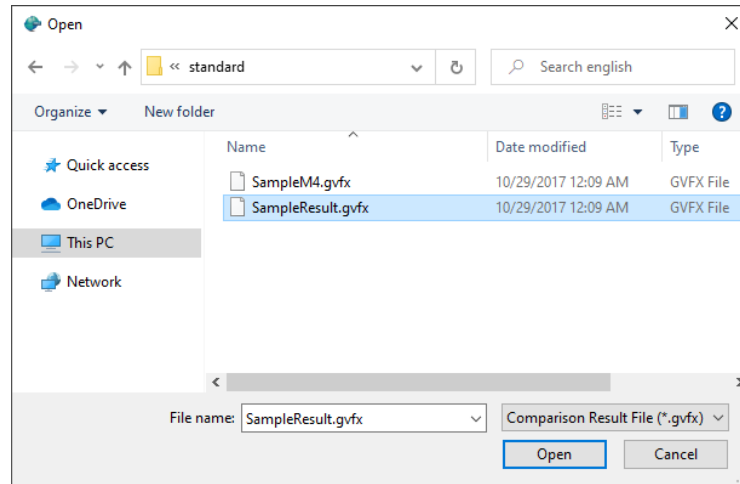
"Example of Usage"

Quick Comparison: Comparing the design before and after

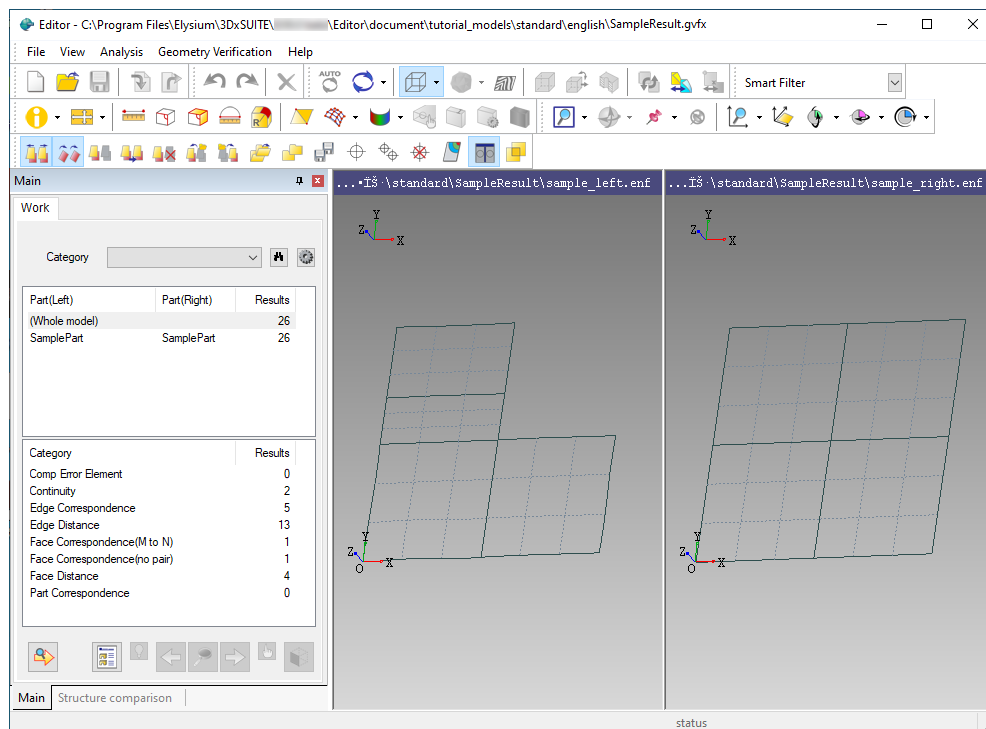
Detailed Comparison: Comparing the translated model before and after modification

2.4.1. Operation

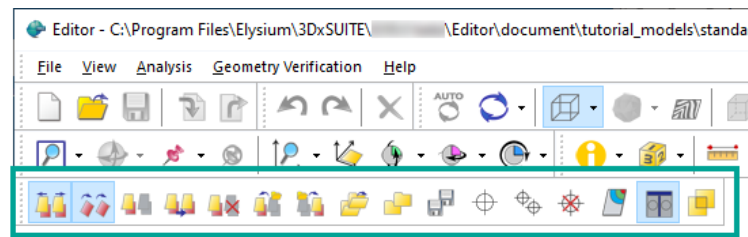
1. Select [Geometry Verification] > [Open Comparison Result] () from the menu.
2. "Open" dialog will appear.
Specify **SampleResult.gvfx** in the <tutorial> folder and click [Open].



This will switch automatically to Geometry Verification Mode. In "3D View" window, the left sample model, which is before modification, has a soft wavy surface. The right sample model, which is after modification, is a plane.



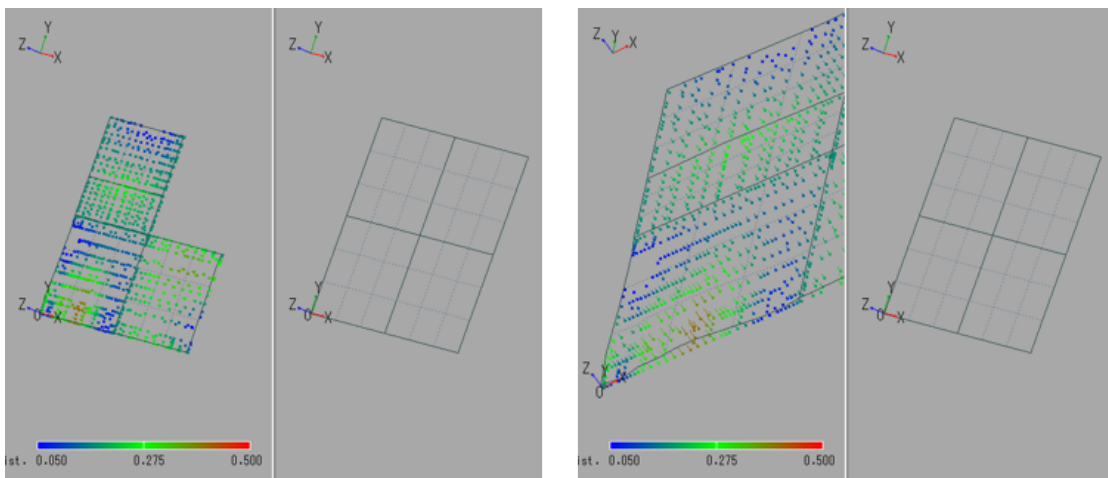
Geometry Verification toolbar will appear in the toolbar.



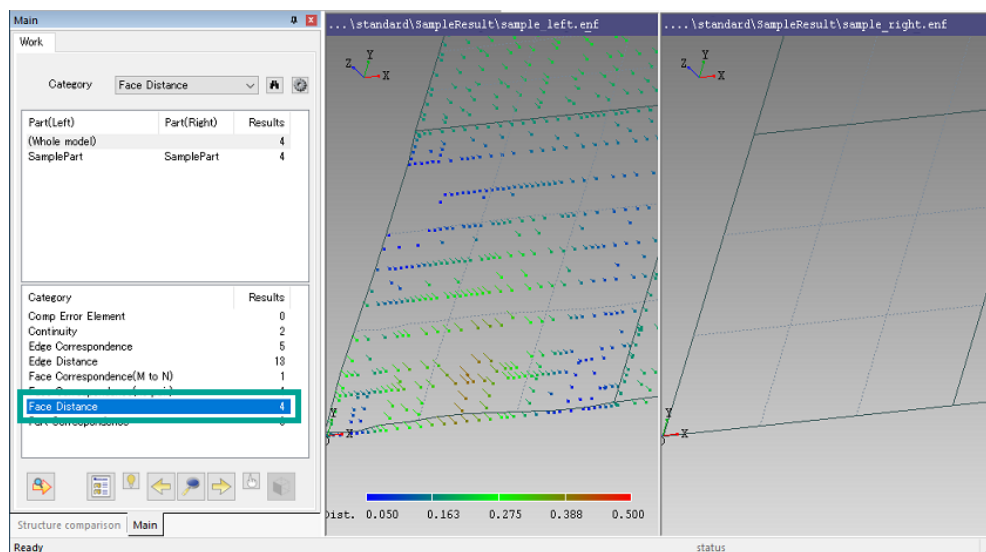
Geometry Verification toolbar

Immediately after switching to Geometry Verification Mode, since [Synchronize Views] (🔄) is enabled, all view operations are synchronized on the left and right models as shown in the below left figure.

Press [Synchronize Views] (🔄) to disable the function. Left and right view windows can be operated independently as shown in the below right figure.



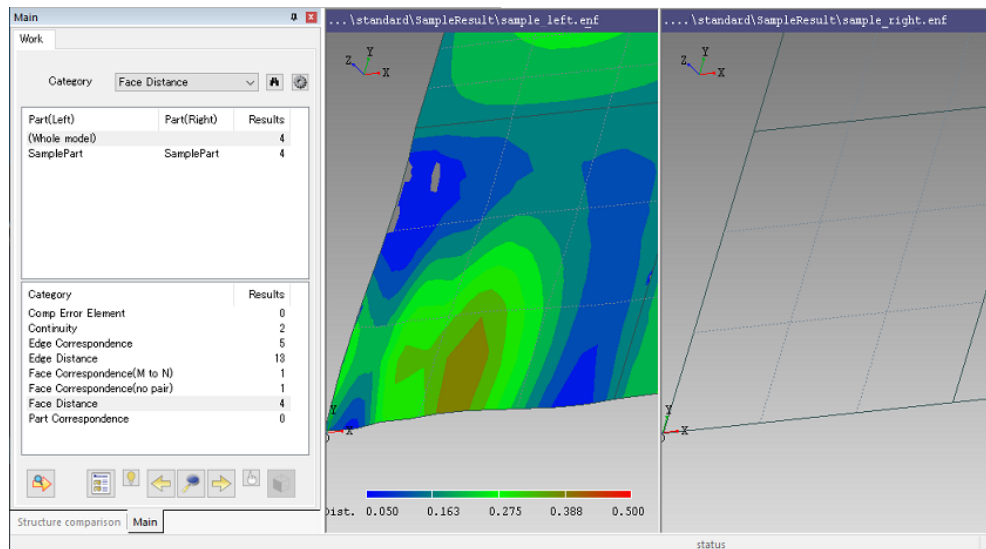
- In [Main (Work)] panel, select "Face Distance" from the category list. Distance between the left and right models is displayed in different colors according to the gap between faces.



By default, "Nearest Point Direction" is set to display by needles the nearest point direction per point. Small deviations are shown in blue, and large deviations in red.

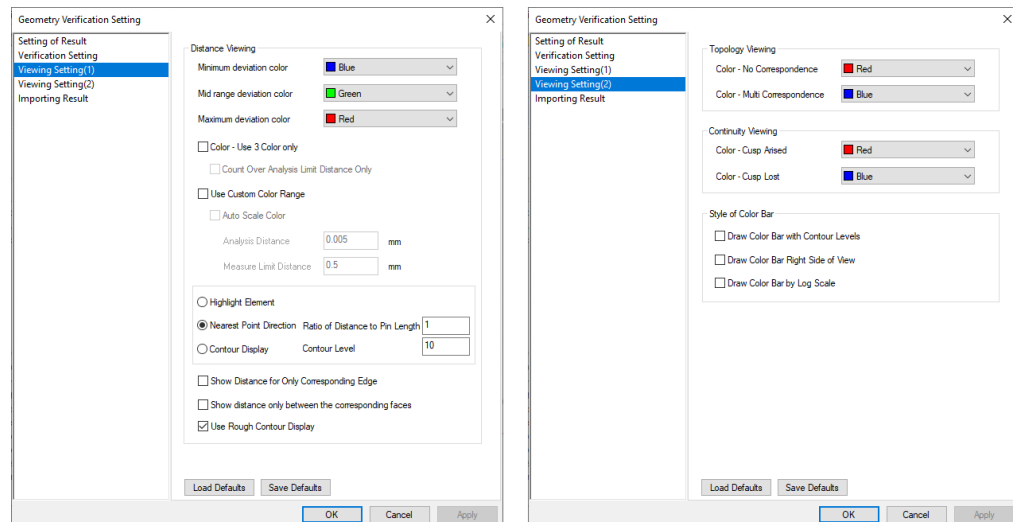
- Press [Switch Direction / Contour] (🔄) on the toolbar.

Toggle the viewing mode to contour. [Contour View] will fill in the gap and display the data based on [Nearest Point Direction].

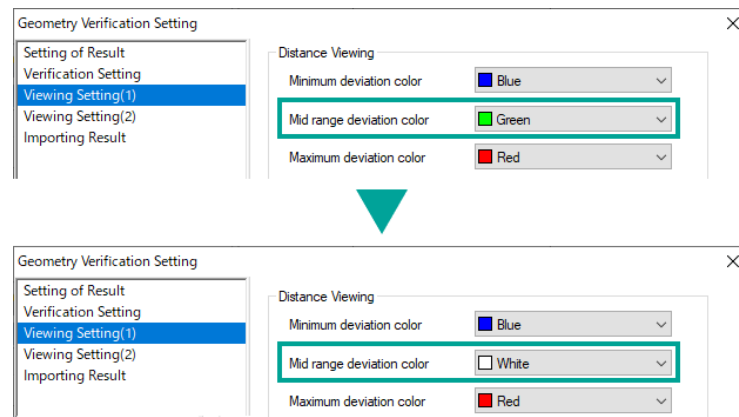


[Contour View] may take longer and consume more memory than [Nearest Point Direction]. Therefore, please ensure to have sufficient memory when running [Contour View] on a large model.

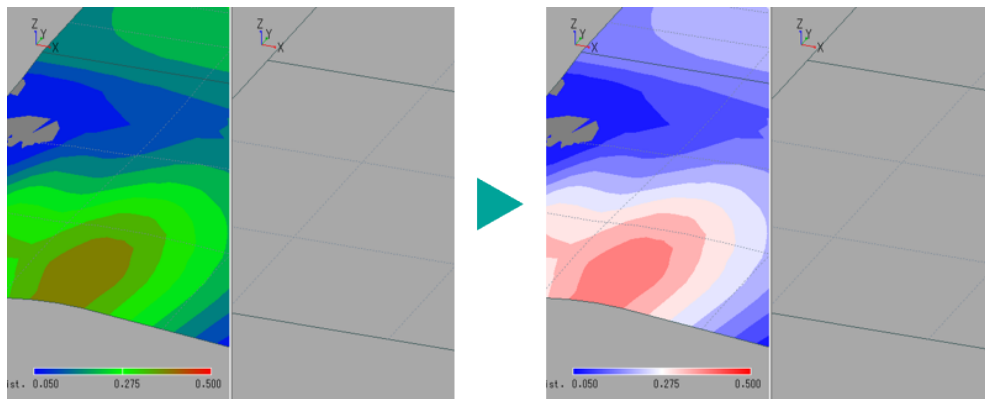
To switch direction / contour or set the displayed color result, press [Settings] (⚙️) located at top right of [Main (Work)] panel. "Geometry Verification Setting" dialog will appear. Specify the preferred setting in [View Setting (1), (2)] tabs.




For example, change "Mid-range deviation color" from "Green" to "White", then click [Apply].



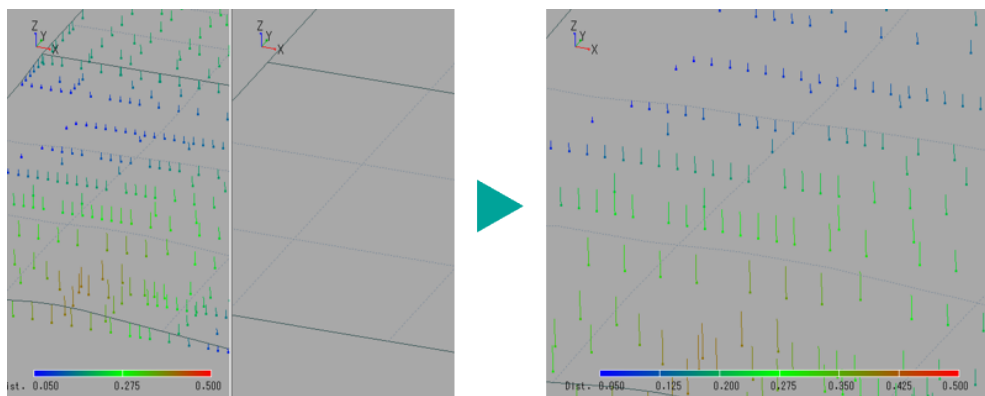
The display color will change as shown below.




Again, set "Mid-range deviation color" to "Green", and press [Switch Direction / Contour] () on the toolbar to switch to [Nearest Point Direction].

5. Press [2View/1View] () on the toolbar.

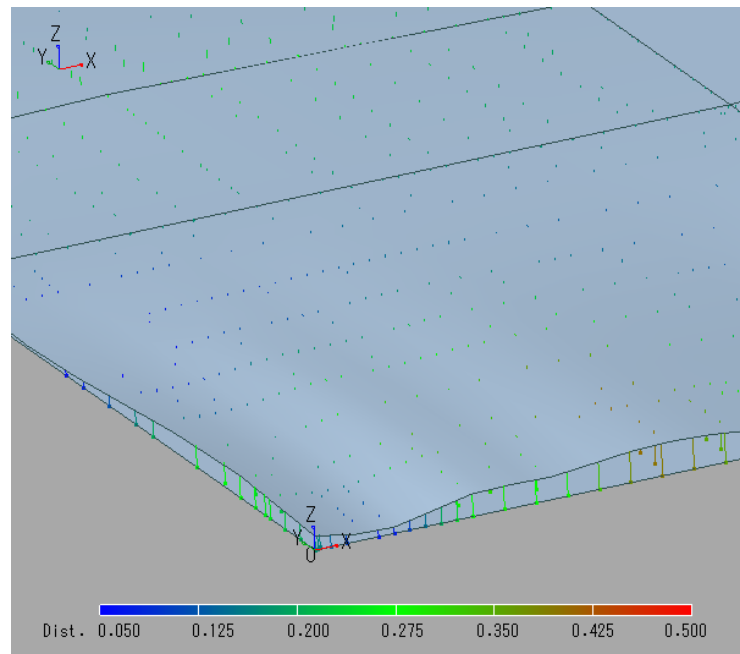
As shown below, only the left model and the differences are shown in one big view (1View). [1View] is useful when there is no need to look at the right model to compare with or when you want to temporary use the full screen to check the left model.



When [2View/1View] () is enabled, [2View] is enabled. When disabled, then [1View] is enabled. Please note that this function only appears on the toolbar.


6. Press [Draw 2Model in 1View] () on the toolbar.

Left and right models are displayed overlapping each other. Depending on the geometry of the models being compared, it may be easier to see the differences by overlapping them.




Above figure is switched to [Shading + Wireframe] () for easier viewing on "3D View" window.

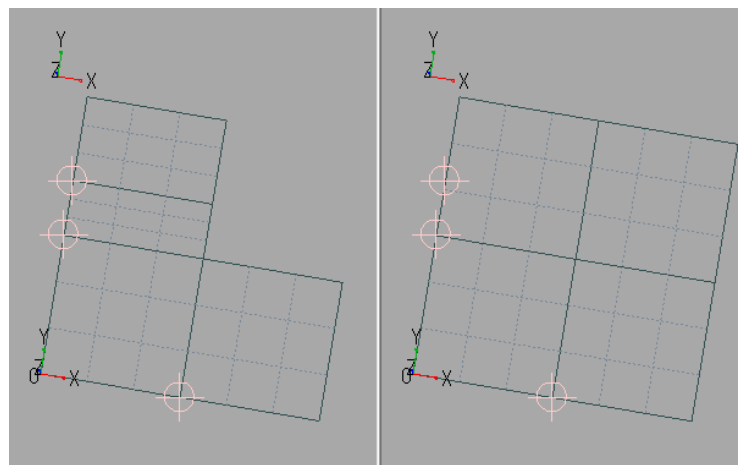


When [Draw 2Model in 1View] () is enabled, the models will overlap each other. When disabled, the models are independently displayed. Please note that this function only appears on the toolbar.

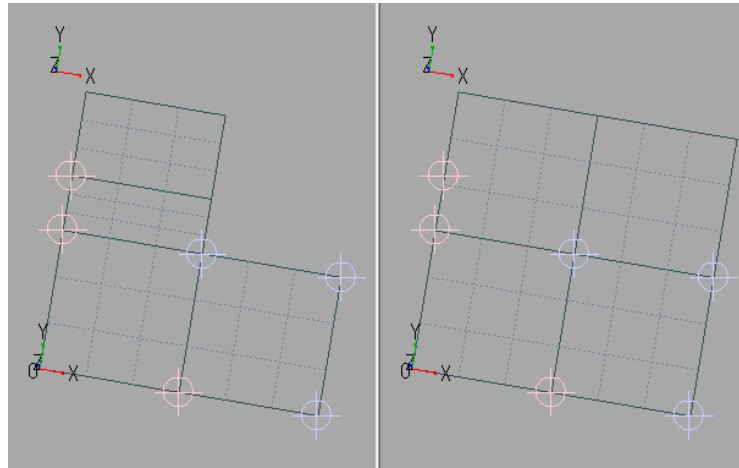
Besides toggling this function for easier comparison, you can insert indicators as markers.

■ Add and Remove Point Indicator

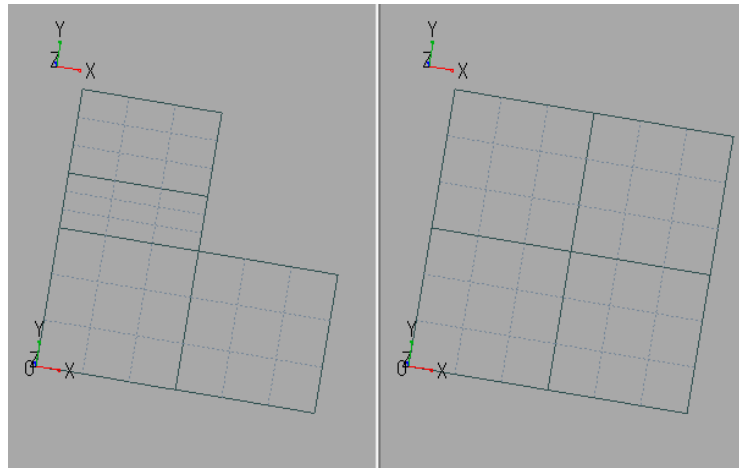
1. Press [Add Point Indicator] () on the toolbar.
2. Select elements such as vertices and edges, etc. in the left model. Red indicators will be created at the specified position and likewise for the right model.



Same operation can be done with the right model except the indicators will be blue. Indicators will be created in the same position for the left model.



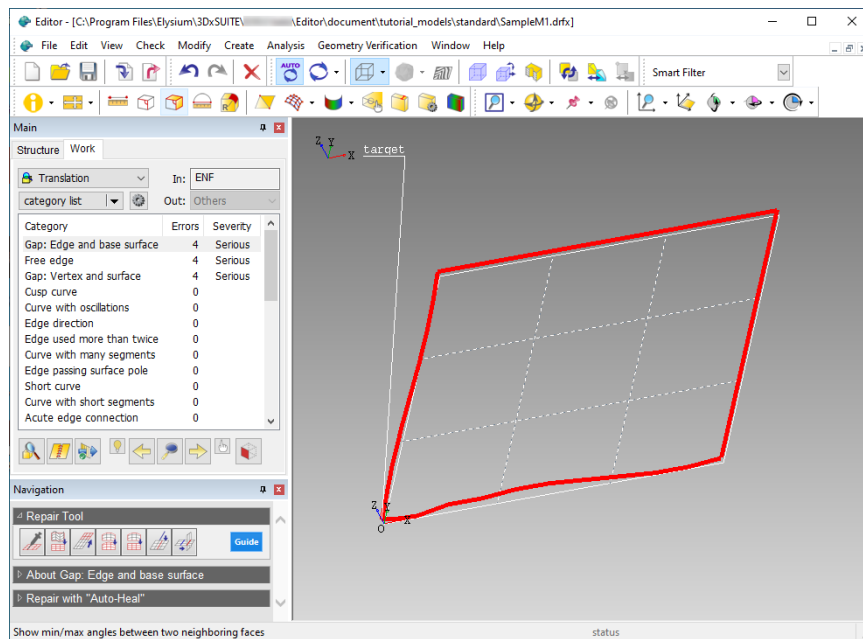
3. Press [Remove All Point Indicators] () on the toolbar to remove all indicators.



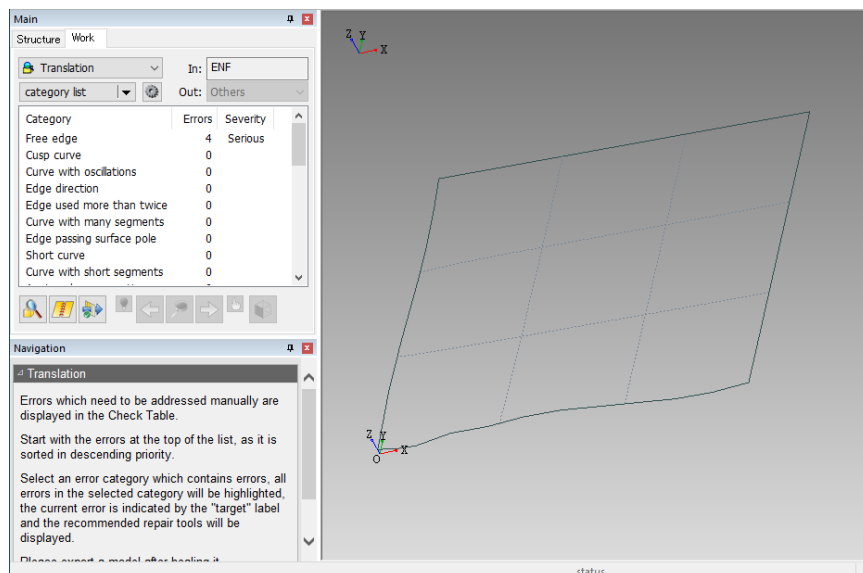
2.4.2. Start Geometry Verification

Using the sample model which has the error "Gap: Edge and base surface", compare the geometry before and after modification and check how the surface has changed.


1. Refer to 2.2.1, "Open Files" and open **SampleM1.drfx** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Gap: Edge and base surface" from the category list. Target error is highlighted in red.



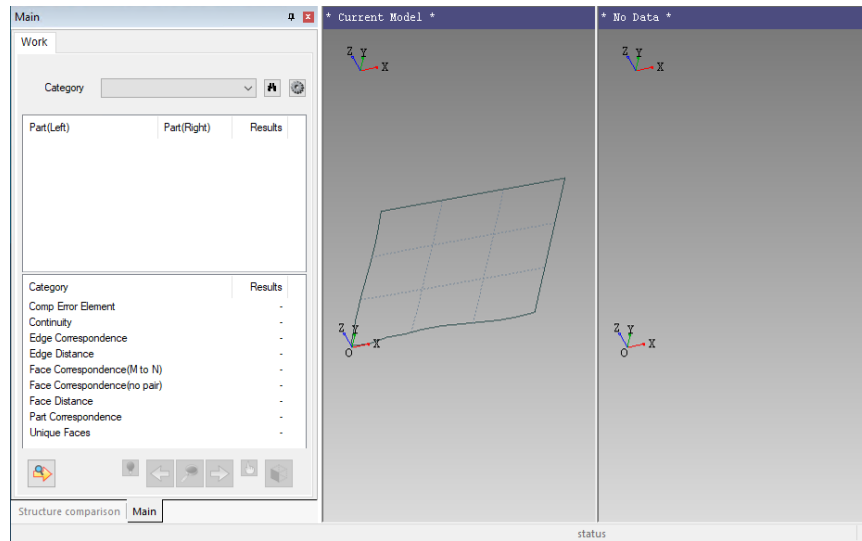
3. Click [Fit Face to Loops] () on Navigation panel to modify the error.




Please refer to 3.1.2, "Fit Face to Loops" in Intermediate for details about operation.

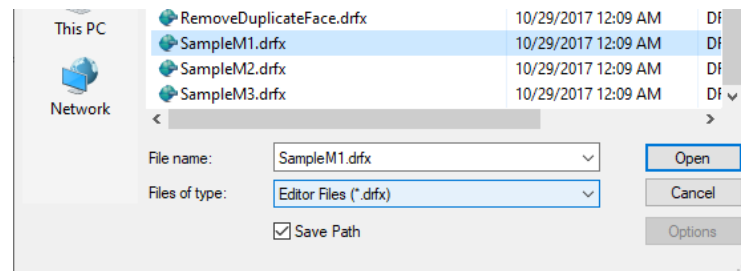
4. Select [Geometry Verification] > [Geometry Verification Mode] () from the menu to switch to "Geometry Verification Mode".

Current model is displayed on the left screen as the comparison reference model.



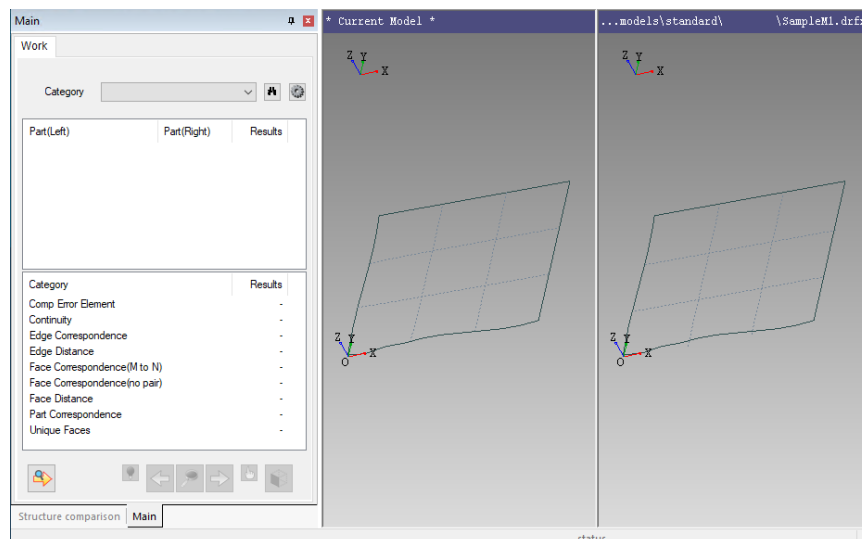
Next, open the model that you want to compare.

5. Press [Open Right Model] () on the toolbar.
6. "Open" dialog will appear. Change the file type to "Editor Files (.drfx)", then open **SampleM1.drfx** in the <tutorial> folder.



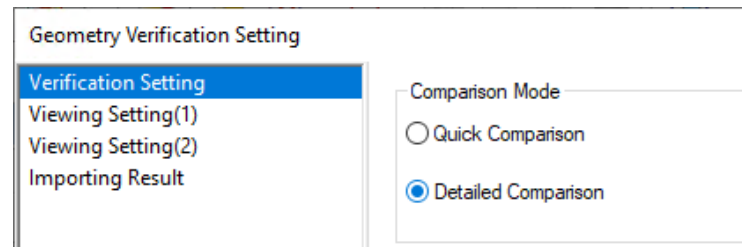
Also, to import a model, just drag-and-drop the model onto the right screen.

Sample model will appear on the right screen.



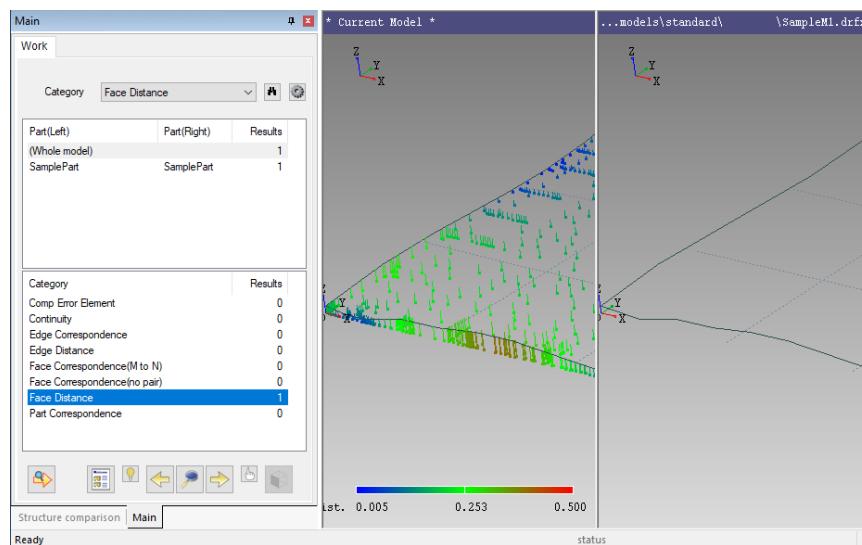
Next, set the options for Geometry Verification.

7. Select [Geometry Verification] > [Settings] from the menu or press [Settings] (⚙️) at the upper right of [Main (Work)] panel.
8. "Geometry Verification Setting" dialog will appear. From [Verification Setting] tab, select "Detailed Comparison" in "Comparison Mode", and click [OK].



Next, run Geometry Verification.

9. Press [Begin Verification] (🔍) at the lower left of [Main (Work)] panel. Verification between the modified model on the left screen (current model) and the unmodified model on the right screen is executed.
10. Select "Face Distance" from the check item in [Main (Work)] panel. Confirm that the surface has been modified to fit along the curved shape.



Click [Save Comparison Result] (💾) on the toolbar to save the comparison result.



The saved comparison result file (.gvfx) can be opened by clicking [Open Comparison Result] (📁) on the toolbar.

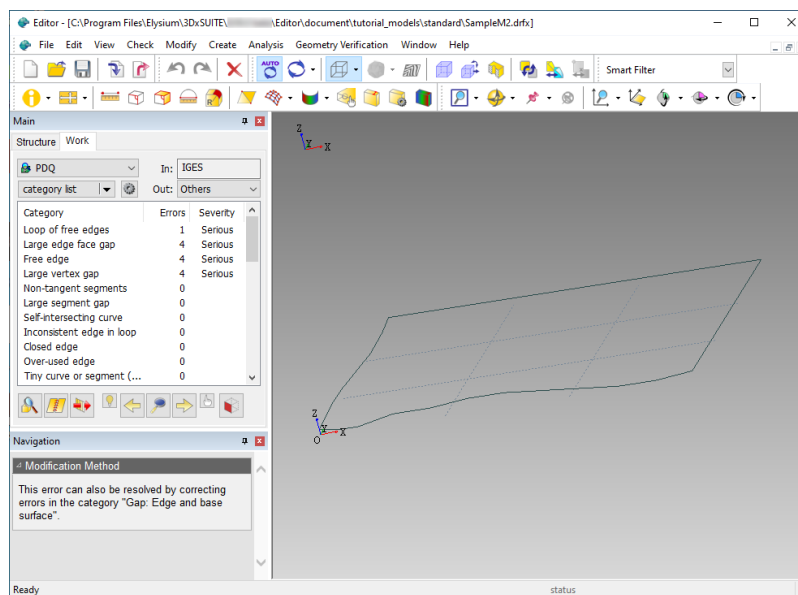
11. After confirming the comparison result, select [Geometry Verification] > [Geometry Verification Mode] (🔍) again.
You will return to the state just before entering Geometry Verification Mode.

2.4.3. Geometry Verification of Whole Model Before and After Auto Heal

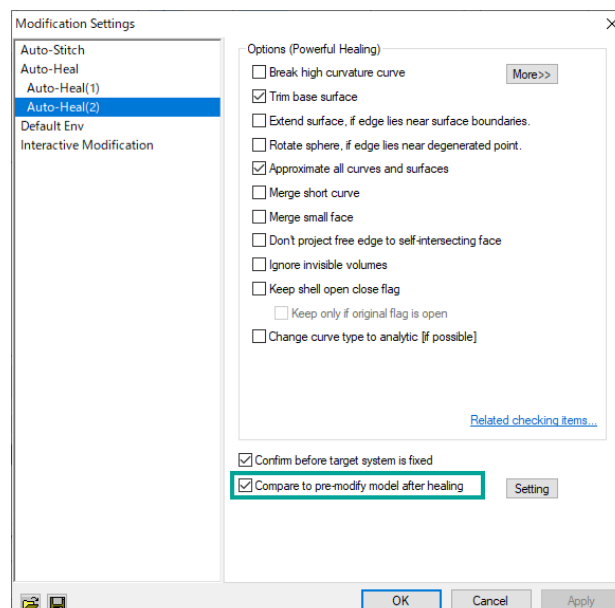
This section will explain how to automatically perform Geometry Verification for models before and after Auto Heal.

Advantage of this function is that you can shorten the running time rather than if you were to manually save the drfx file before auto healing and manually compare the geometry afterwards.

1. Press [Open] (📂) on the toolbar. Open **SampleM2.drfx** from the <tutorial> folder.



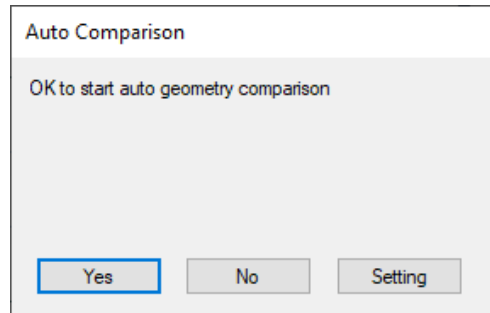
2. Select [Modify] > [Options] from the menu. "Modification Settings" dialog will appear. In [Auto-Heal(2)] tab, enable the option "Compare to pre-modify model after healing" and click [OK].



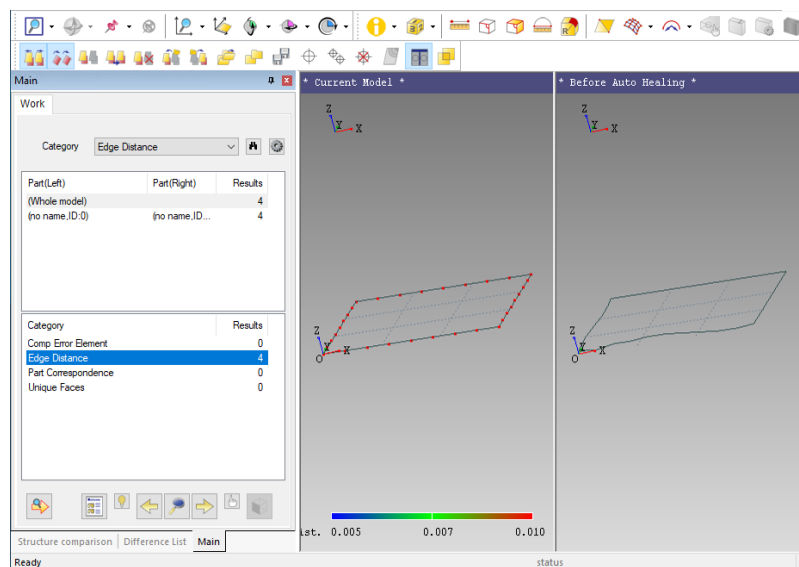


Please note that [Setting], which is located on the right of this option, must be set before auto healing. Settings will not be changed here because the default values will be used.

3. In [Main (Work)] panel, press [Auto Heal] ().
4. After auto healing is completed, "Auto Comparison" dialog will appear. Click [Yes].



When the verification is completed, it automatically switches to Geometry Verification Mode and the comparison result is displayed.





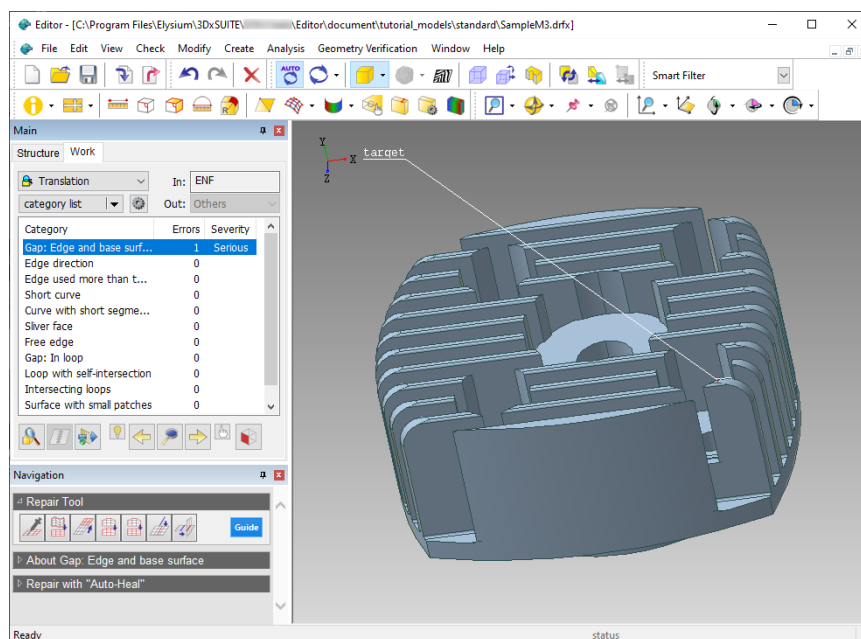
5. After confirming the check result, once more select [Geometry Verification] > [Geometry Verification Mode] () from the menu.
Close Geometry Verification Mode and return to where you left off after auto healing.

2.4.4. Partial Geometry Verification of Interactive Healing (Before and After)

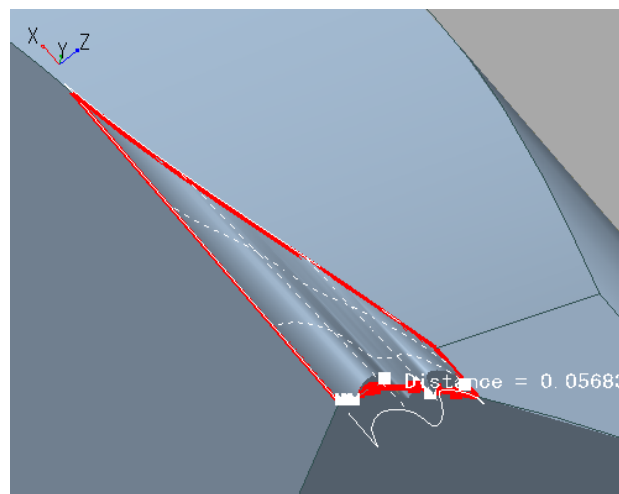
For a model before and after interactive healing, you can perform partial geometry comparison on a partially modified part. However, this section will explain how to perform geometry verification without saving the unmodified drfx file.

Detected error "Gap: Edge and base surface" will be interactively healed, and the geometry before and after modification will be compared.

1. Press [Open] () on the toolbar. Open **SampleM3.drfx** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Gap: Edge and base surface" from the category list. Press [Zoom current target] ().



The error is zoomed up. In this error, the surface is distorted, so modify the error by replacing the surface.



First of all, store the "before modification" model internally.



With Editor, the currently displayed model in "3D View" window is stored as a geometry data, but a second model can be internally stored as a separate model. This internally stored data is displayed as "Target model (on right)" when switching to [Geometry Verification Mode].

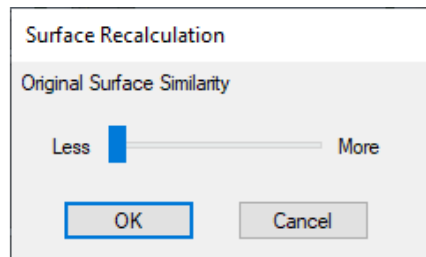
3. Select [Geometry Verification] > [Target Model] > [Copy Current Model] (📄➡️) from the menu.



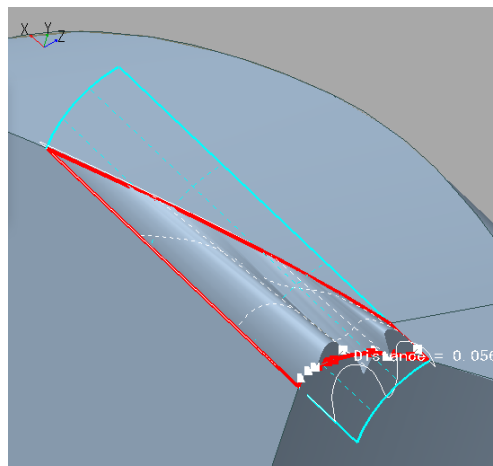
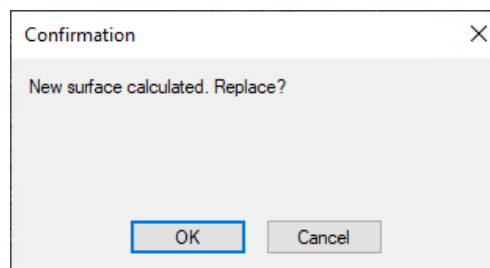
No change can be seen on Editor, but the displayed model (before modification model) is stored internally. Also, when selecting [Geometry Verification] > [Target Model] > [Delete Target Model] (🗑️❌), the internally stored model will be deleted. (*So DON'T select here.)

Next, modify the error.

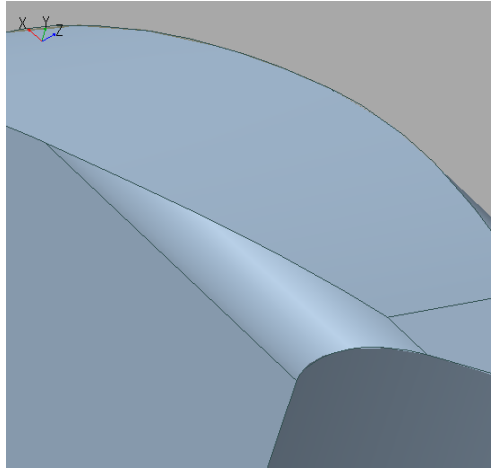
4. Press [Recalculate Surface] (📐➡️) from Repair Tool on Navigation panel.
5. "Surface Recalculation" dialog will appear. To avoid reflecting the original surface in this model, set "Original Surface Similarity" to "Less", and click [OK].



6. Recalculated surface will be displayed and a confirmation dialog will appear. Click [OK].

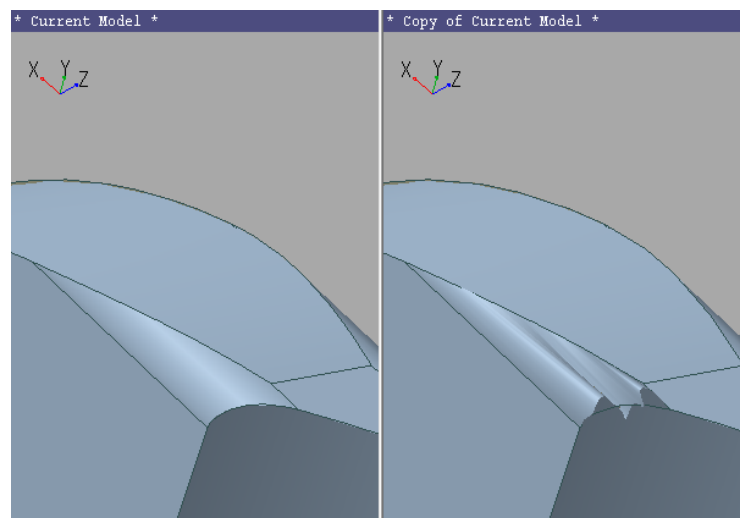


After completing interactive healing, compare the modified areas.



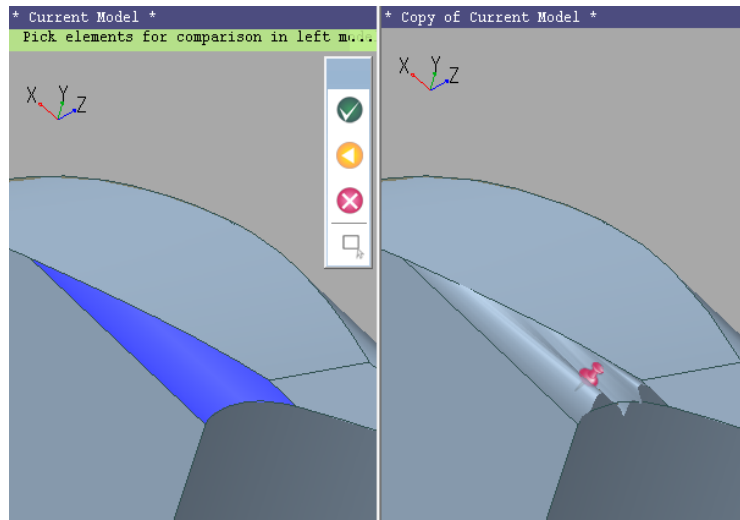
7. Select [Geometry Verification] > [Geometry Verification Mode] (🔍) from the menu.

Switch to Geometry Verification Mode. The model after interactive healing is displayed (on left), and the unmodified model that was saved internally beforehand is displayed (on right).

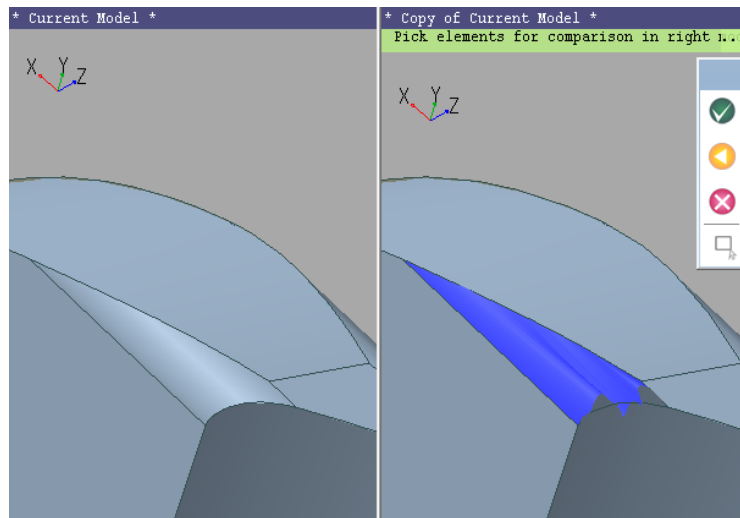


Next, specify the face modified by interactive healing as the part to be compared.

8. Select [Geometry Verification] > [Partial Geometry Verification] from the menu.
9. Pick the face for comparison from the left model and press [Done] (✅).

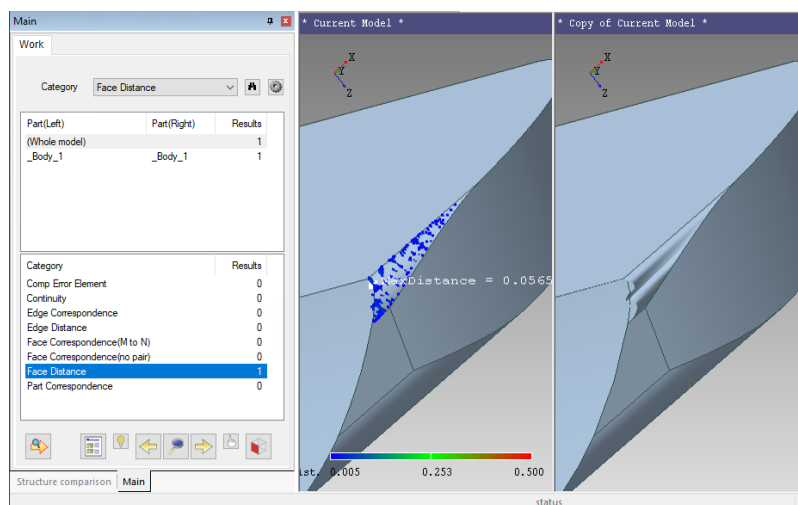


10. Next, pick the face for comparison from the right model and press [Done] (✓).

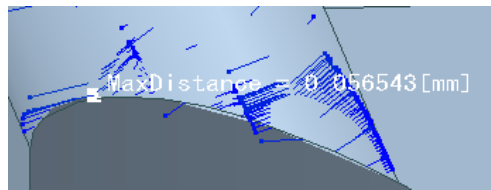


Comparison of the two selected faces is executed.

11. In [Main (Work)] panel, select "Face Distance" and press [Zoom current target] (🔍).



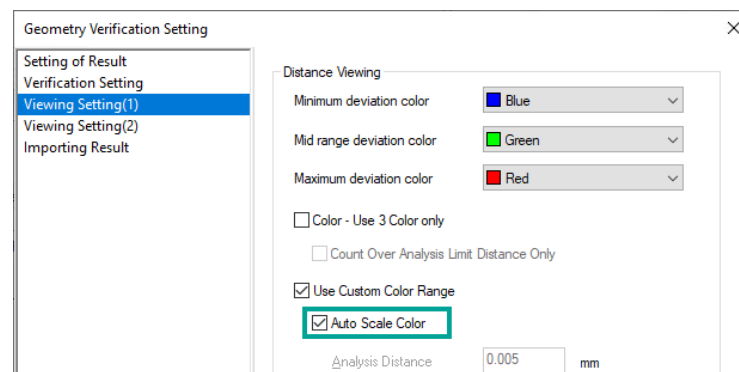
When checking the differences, it can be confirmed that a maximum of about 0.05 mm of geometry deformation has occurred.



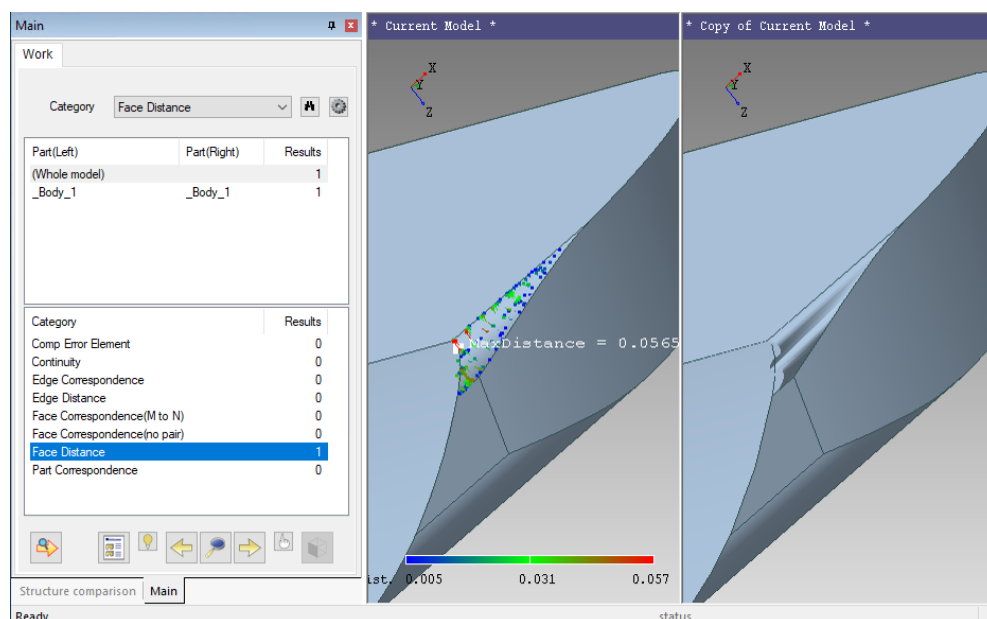
By default setting, the minimum value is 0.005 mm and the maximum value is 0.5 mm for the difference distance shown by the color bar. Minimum and maximum values of the color scheme can be changed according to the range of difference in distance in the displayed area.

The displayed color difference is not set properly, so change the setting.

12. Select [Geometry Verification] > [Settings] from the menu or press [Settings] (⚙️) located at top right of [Main (Work)] panel. "Geometry Verification Setting" dialog will appear. In [Viewing Setting (1)] tab, enable both "Use Custom Color Range" and "Auto Scale Color" options, and then click [OK].



Color scheme of the color bar changes depending on the range of the difference distance.



13. After confirming the comparison result, once again select [Geometry Verification] > [Geometry Verification Mode] (🔍) from the menu.

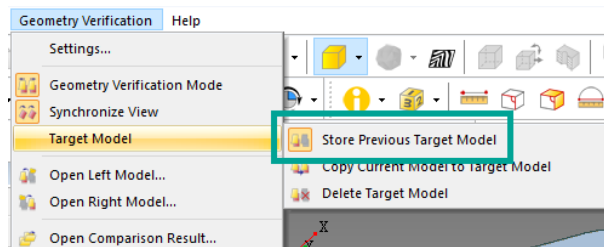
Geometry Verification Mode ends, and you will return to the state after interactive healing.



Please note that when exiting Geometry Verification Mode, the model to be compared displayed on the right screen is deleted, and at the same time, the model stored internally is also deleted.

In case you want to keep the internally stored model after exiting Geometry Verification Mode, select [Geometry Verification Mode] > [Target Model] > [Store Previous Target Model] (📁📄).

When exiting Geometry Verification Mode with [Store Previous Target Model] (📁📄) enabled, the internally stored model will not be deleted.



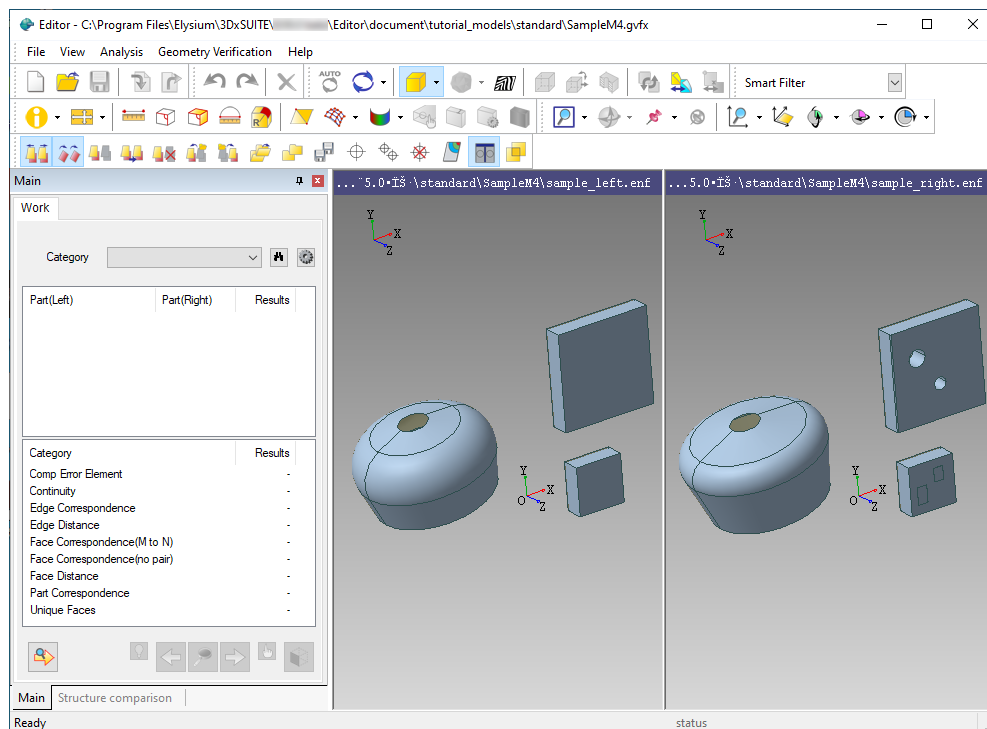
2.4.5. Quick Comparison of Whole Model


By using the function "Quick Comparison" in Geometry Verification, you can quickly see the differences in the characteristics between models.

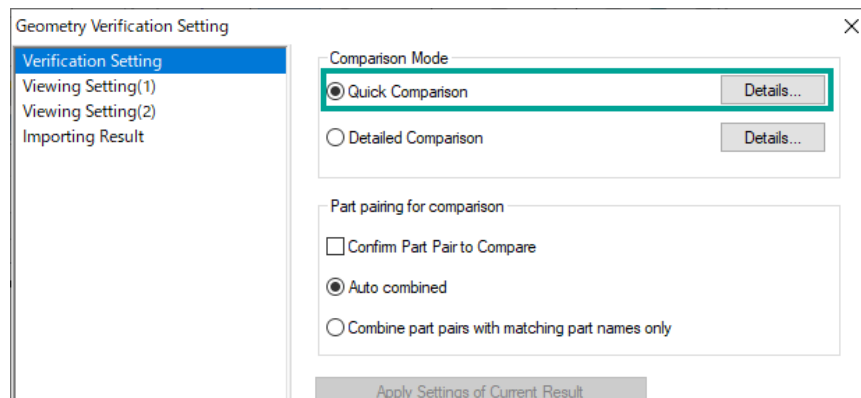


Please note that the function "Quick Comparison" compares fewer check items than "Detailed Comparison".

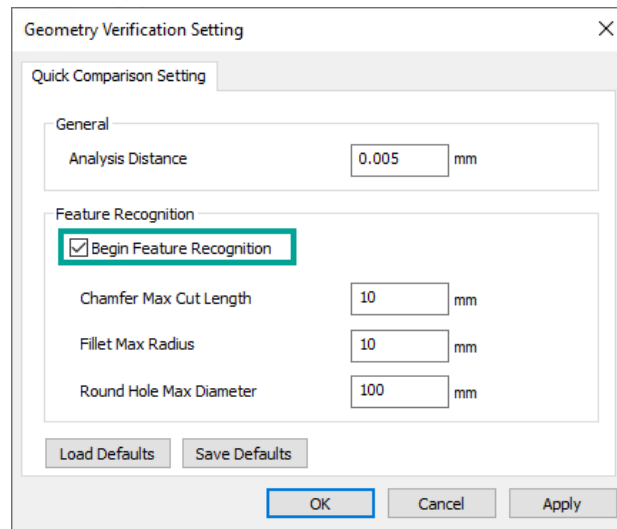
1. Select [Geometry Verification] > [Open Comparison Result] () from the menu.
2. "Open" dialog will appear. Specify **SampleM4.gvfx** from the <tutorial> folder, and click [Open].



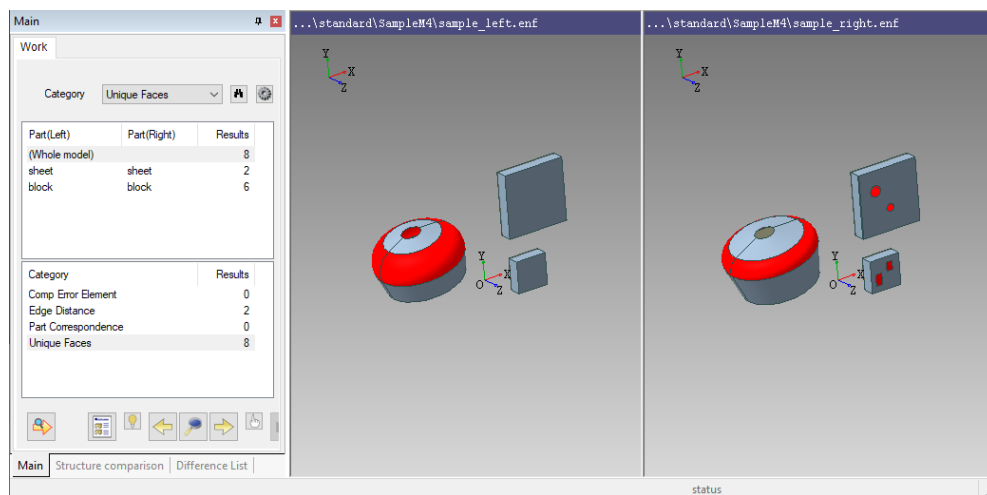
3. Select [Geometry Verification] > [Settings] from the menu or press [Settings] () located at top right of [Main (Work)] panel.
4. "Geometry Verification Setting" dialog will appear.
In [Verification Setting] tab, select "Quick Comparison" in Comparison Mode, then click [Details] located on the right.



5. [Quick Comparison Setting] tab will appear in "Geometry Verification Setting" dialog. Make sure that "Begin Feature Recognition" is enabled and click [OK].

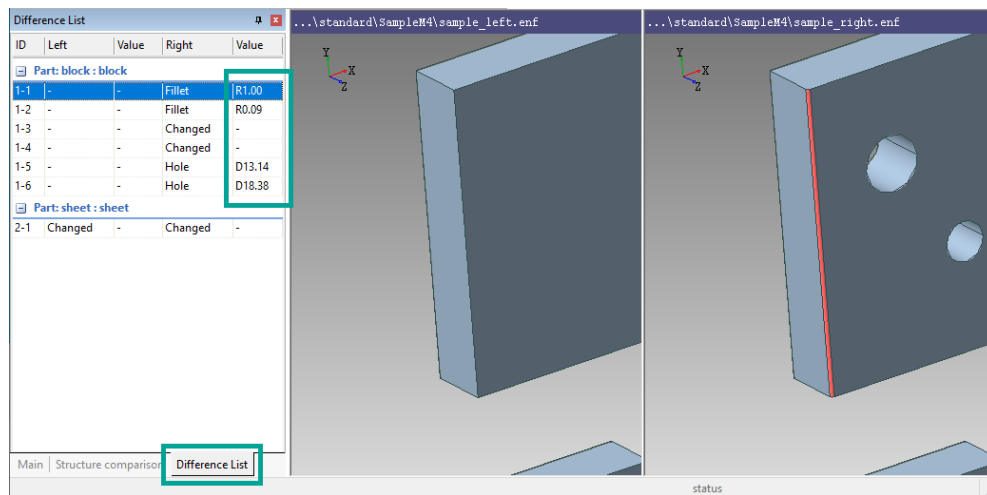


6. Select [Geometry Verification] > [Begin Verification] from the menu or select [Begin Verification] (🔍➡️) in [Main (Work)] panel.
7. In [Main (Work)] panel, select "Unique Faces" from the category list. The differences between the two models are highlighted in red.



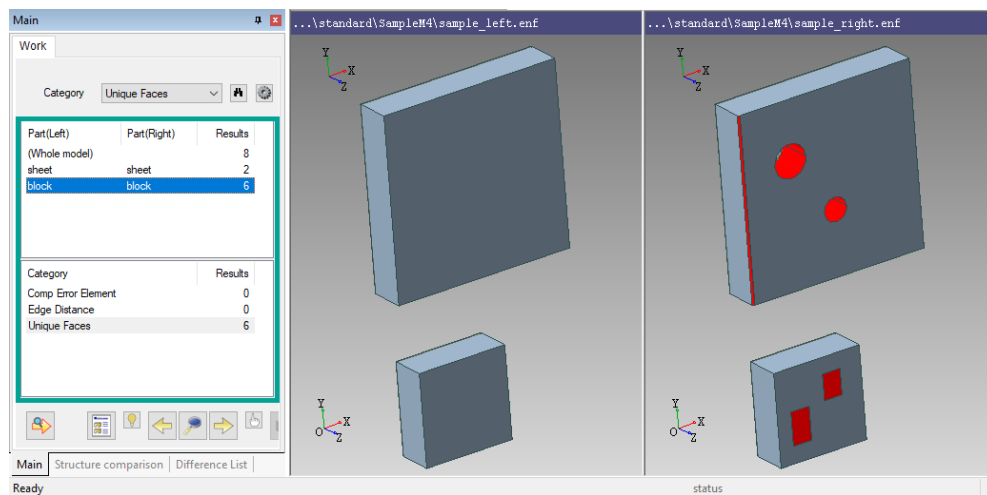
Geometry differences can be confirmed using the icon such as [Zoom current target] (🔍) located at the bottom of [Main (Work)] panel as in the case of the normal comparison function.

Because "Begin Feature Recognition" was enabled in "Quick Comparison Setting" dialog when running Geometry Verification, "Difference List" was added to [Main] panel. In Difference List, you can confirm the difference for each recognized feature.

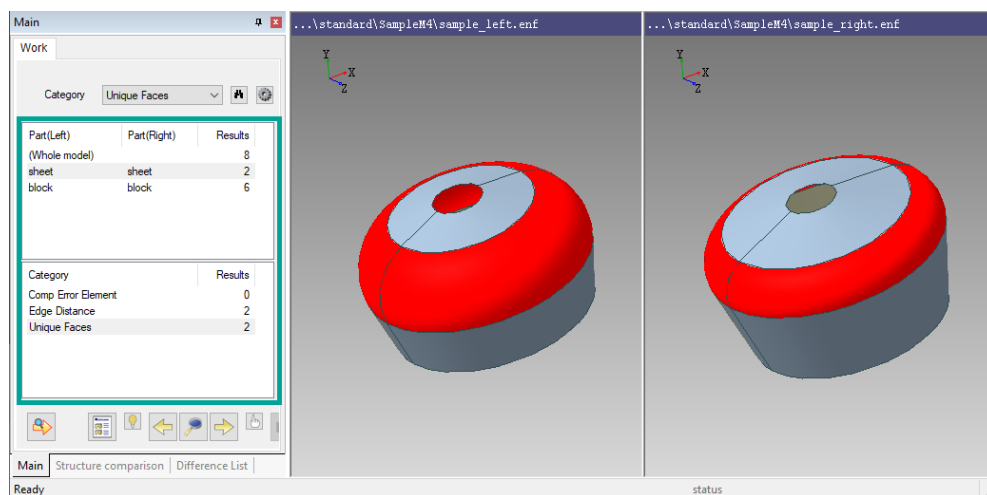


Furthermore, models that have multiple parts can have their comparison results displayed according to each part.

- In [Main (Work)] panel, double-click the part named "block" from the Difference List. The display switches from the whole model to an individual part. Confirm the difference in each part.

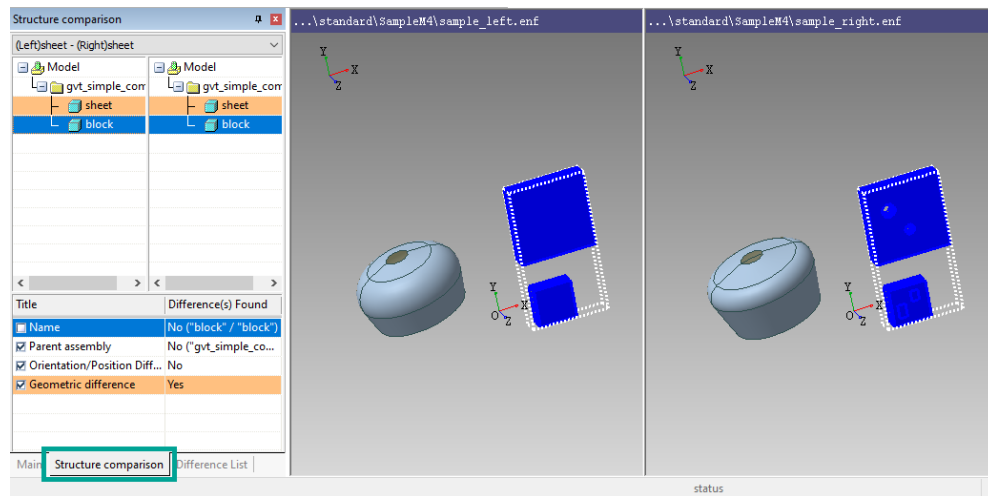


- In [Main (Work)] panel, double-click the part named "sheet" from the Difference List. Just like with the "block", you can quickly see the difference between the parts.



When confirming [Structure comparison] panel, the background color of the parts with differences in the assembly tree changes in the list.

Also, when selecting a part in [Structure comparison] panel, the corresponding part is highlighted in blue on "3D View" window.



By using the function "Quick Comparison", you can easily detect geometric differences in assembly models.

3. Intermediate

This chapter will explain in detail how to modify errors using Interactive Healing functions. With the knowledge you have gained up to this point, Editor will let you translate most of the data adequately.

3.1. Interactive Healing

This section will explain how to modify the error, "Gap: Edge and base surface", which tends to occur quite often.

Which Category to Modify From

When errors "Loop with self-intersection" and "Surface with self-intersection" are detected, these two errors must be modified first before any other error items. If you modify other error items while these two types of errors exist, the modification function may not work properly.

When "Loop with self-intersection" and "Surface with self-intersection" are not detected, you can start modifying from any error items.

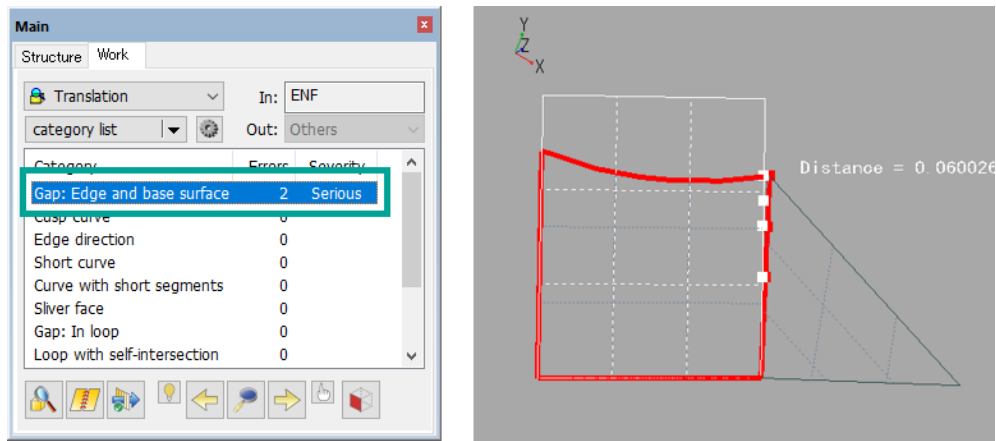
"Gap: Edge and base surface" is an error that often occurs during data translation, and there are many modification functions to use from to modify the error. The characteristics of each function will be explained so that you can select the most appropriate one for you.





- Please refer to "[Check for Errors](#)" in [Beginner] for more details about how to check detected errors.
- Also, refer to "[Interactive Healing](#)" in [Beginner] for more details about the general operation of interactive healing.

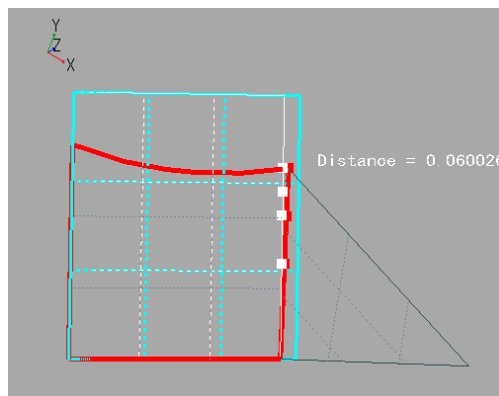
3.1.1. Extend Surface

1. Refer to [2.2.1, "Open Files"](#) and open **ExtendSurface.drfx** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Gap: Edge and base surface" from the category list. Target position is highlighted in red on "3D View" window.

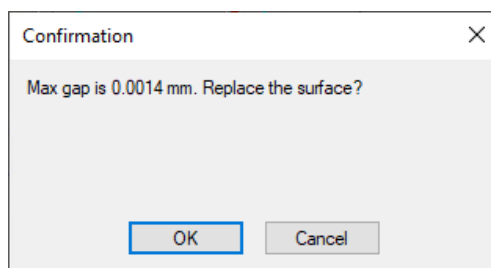


Confirm the face (red) protruding from the surface (white). In this case, use the function [Extend Surface].

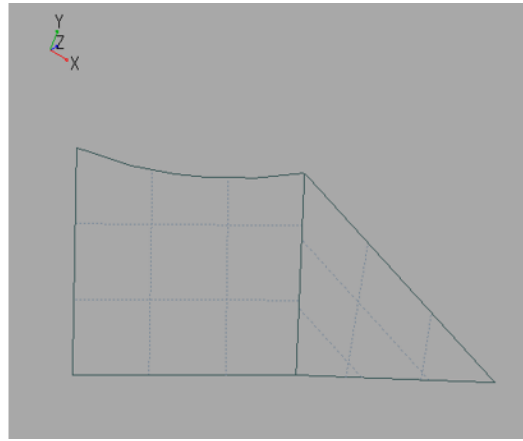
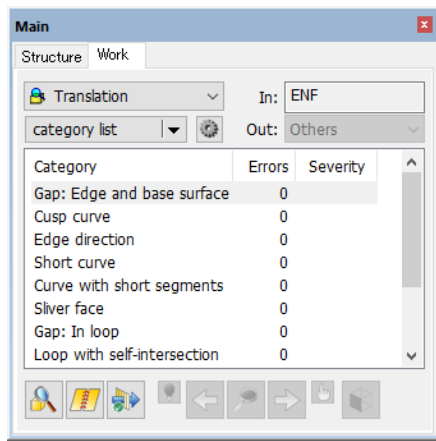
3. Select [Modify] > [Divide/Extend] > [Extend Surface] from the menu or click [Extend Surface] () from Repair Tool on Navigation panel. In this case, press [Extend Surface] () on Navigation panel.
4. A new surface is calculated and its boundary is highlighted in light blue on "3D View" window.



In addition, a "Confirmation" dialog will appear. The maximum value of "Gap: Edge and base surface" on the new surface will be displayed. Make sure you have made the appropriate modification, and click [OK].



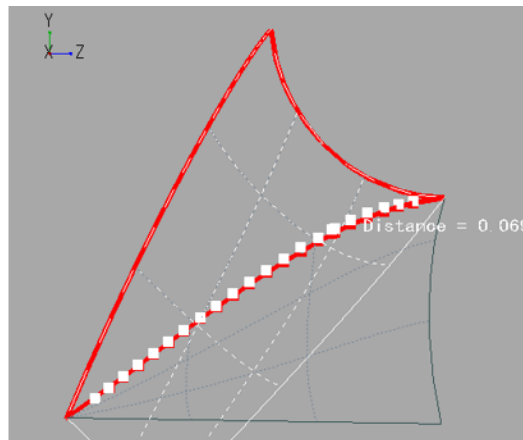
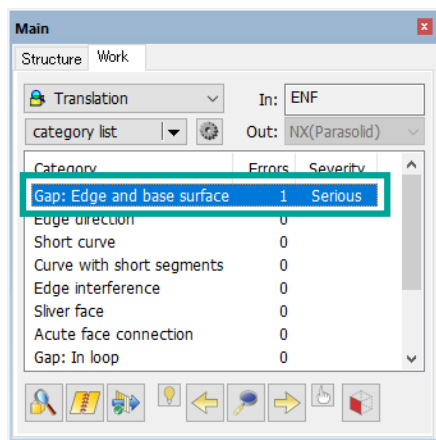
The error is modified and the category list is updated.



Please note that [Extend Surface] () can be used to extend cylindrical and other analytical surfaces appropriately. Refer to [A.2, “Analytic Surfaces”](#) in the Appendix for more details.

3.1.2. Fit Face to Loops



1. Refer to [2.2.1, “Open Files”](#) and open **FitFaceToLoops.drfx** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Gap: Edge and base surface" from the category list. Target position is highlighted in red on "3D View" window.



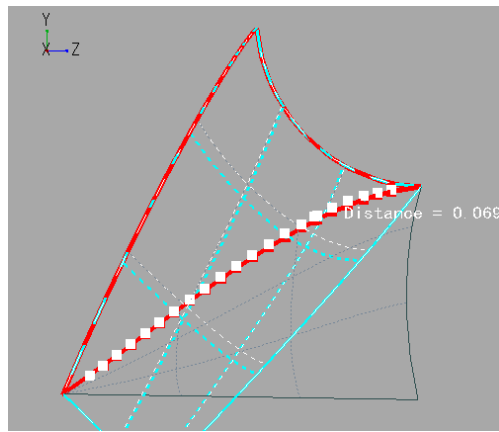
In this case, use [Fit Face to Loops] to modify the error.



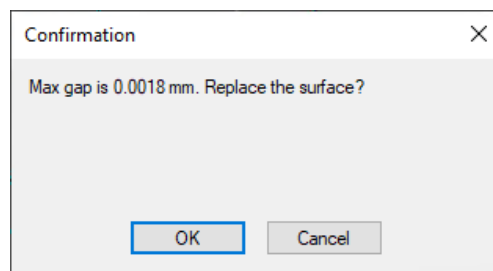
The function [Fit Face to Loops] finely adjusts the position of control points on the surface or increases the number of control points. This function is versatile, and it can modify almost all "Gap: Edge and base surface".

3. Select [Modify] > [Gap between Edge and Surface] > [Fit Face to Loops] or click [Fit Face to Loops] () from Repair Tool on Navigation panel. In this case, press [Fit Face to Loops] () on Navigation panel.

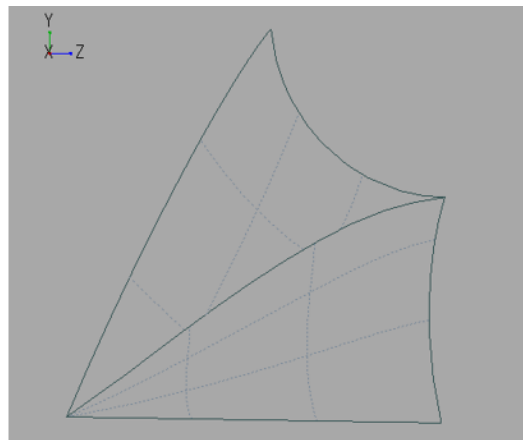
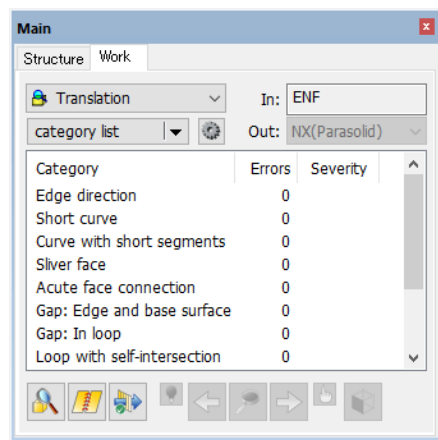
A new surface is calculated and its boundary is highlighted in light blue on "3D View" window.



4. A "Confirmation" dialog will appear. The maximum gap of the new surface will be displayed. Click [OK].



The error is modified and the category list is updated.



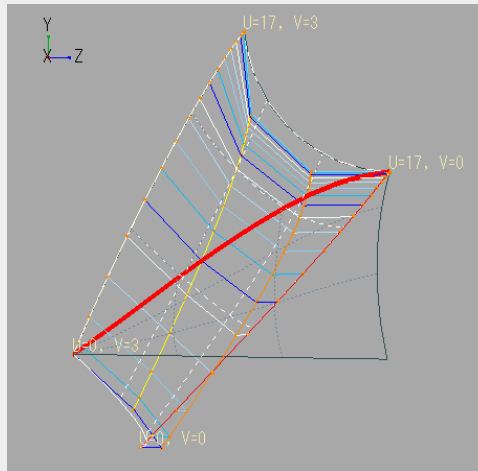
Most of the errors in "Gap: Edge and base surface" can be modified with [Fit Face to Loops]. However, if you want to keep the shape of design intentional surface or a cylindrical / spherical surface, use the functions [[Extend Surface](#)] or [[Project Edge on Surface](#)] to modify.

Fit Face to Loops and Control Points

To better understand the function [Fit Face to Loops], compare the control points before and after modification.

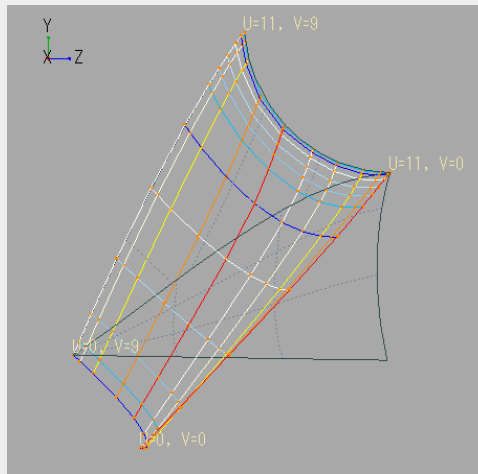
To undo the modification after executing [Fit Face to Loops], select [Edit] > [Undo] (↶).

In this case, select [Analysis] > [Control Points] > [Surface] (📐) and pick the surface with an error. The control points before modification will appear.



Next, select [Edit] > [Redo] (↶) to go to the modified state.

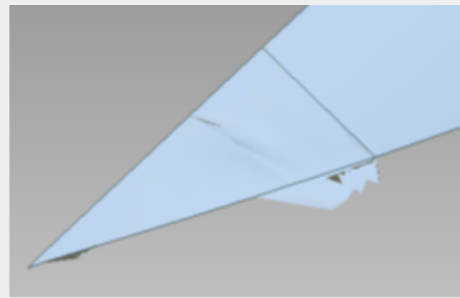
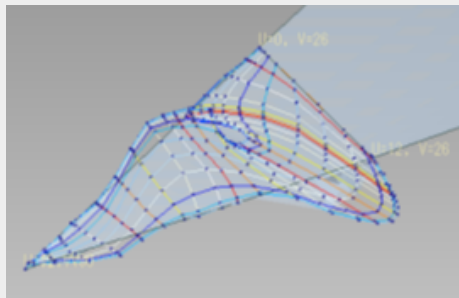
Likewise, select [Analysis] > [Control Points] > [Surface] (📐). Pick a modified surface to show its control points.




When comparing the surfaces before and after modification, you can see that the number of control points has increased to modify "Gap: Edge and base surface".


Since [Fit Face to Loops] is performed by moving or increasing the number of control points, it is not suitable for surfaces with poor quality control points.

(Reference) Surface with poor quality control points (Left: Control points + Semi Transparent, right: Shading)

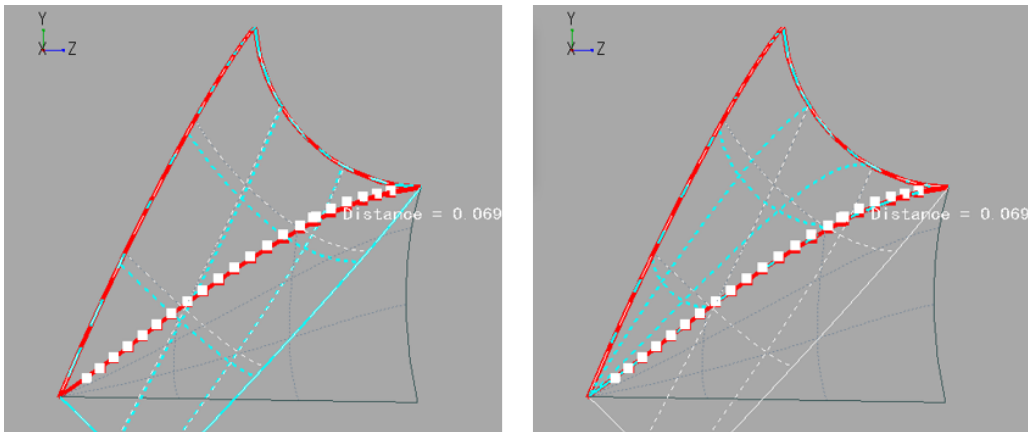


3.1.3. Boundaries, Surface → Surface

The function [Boundaries, Surface → Surface] () is used to create a new surface based on the boundary edges and original surface.

This function is recommended when [Fit Face to Loops] () does not work. Here is just a brief explanation about the function because it is not used very often.


Below is a comparison result between [Fit Face to Loops] (lower left) and [Boundary, Surface → Surface] (lower right). The modified surface is shown in light blue and can be previewed.




Since [Fit Face to Loops] manipulates and modifies the control points, the geometry of the surface hardly changes. On the other hand, [Boundary, Surface → Surface] recalculates to define a new surface. If the quality of the control points is poor, it may be more appropriate to use [Boundaries, Surface → Surface].

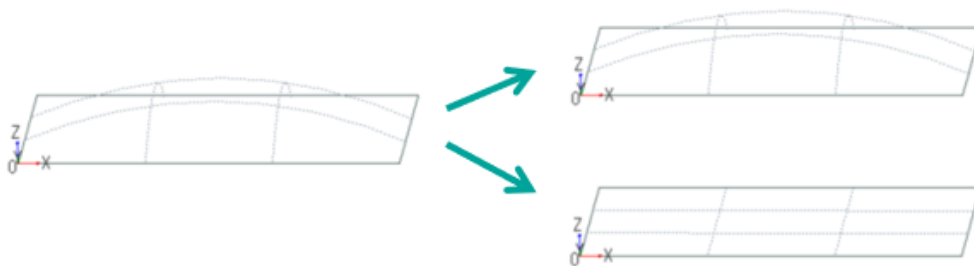
Furthermore, the function [Boundaries, Surface → Surface] cannot be used for faces with complex boundary edges.

3.1.4. Boundaries → Surface

The function [Boundaries → Surface] () is used to recreate the surface based on the geometry of the boundary edges.

Therefore, unlike [Boundaries, Surface → Surface] (), the geometry of the original surface is not referenced at all.

Following figures show the result of modifying the surface with [Boundaries, Surface → Surface] (upper right) and [Boundaries → Surface] (lower right) for the surface with bulged center (left).




This function is recommended for modifying the surface with invalid geometry such as

undulation and self-intersection.

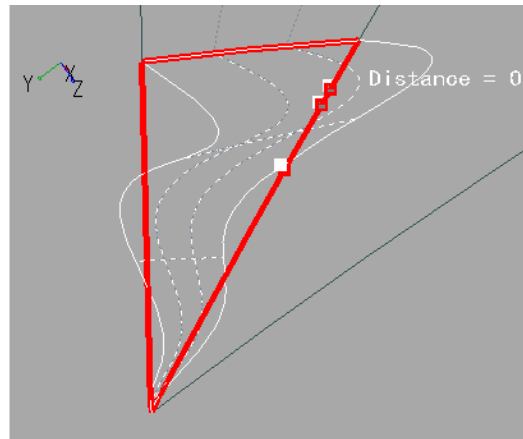
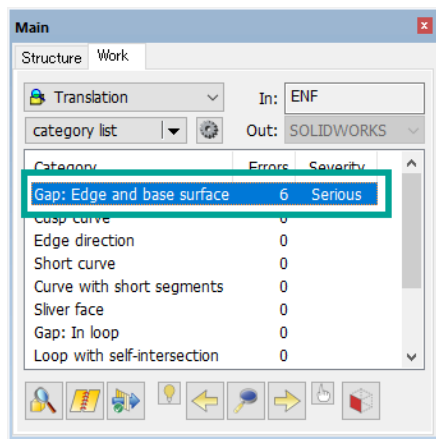
Furthermore, like in the case with [Boundaries, Surface → Surface], the function cannot be used for faces with complex boundary edges.

3.1.5. Recalculate Surface


The function [Recalculate Surface] () is used to recreate a surface based on a plane or a cylindrical surface. Therefore, a surface which is closer to a plane or cylinder is better suited for modification.

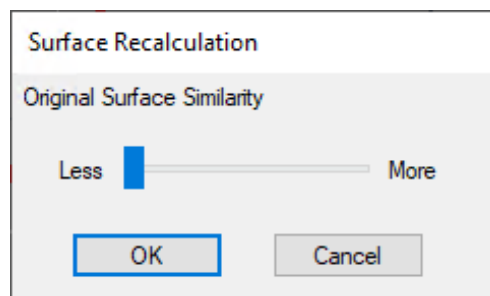
This function is mainly used to modify the errors "Surface with self-intersection" and "Surface with singular point". It is recommended to use on the error "Gap: Edge and base surface" when other functions cannot modify it.

1. Refer to 2.2.1, "Open Files" and open **RecalculateSurface.drfx** in the <tutorial> folder.
2. In [Main (Work)] panel, select "Gap: Edge and base surface" from the category list. Target position is highlighted in red on "3D View" window.



The surface (in white) is oscillating greatly. In such a case, use [Recalculate Surface].

3. Press [Recalculate Surface] () on Navigation panel.
4. "Surface Recalculation" dialog will appear. To avoid reflecting the original abnormal surface, move the slider toward "Less" for "Original Geometry Similarity", and click [OK].



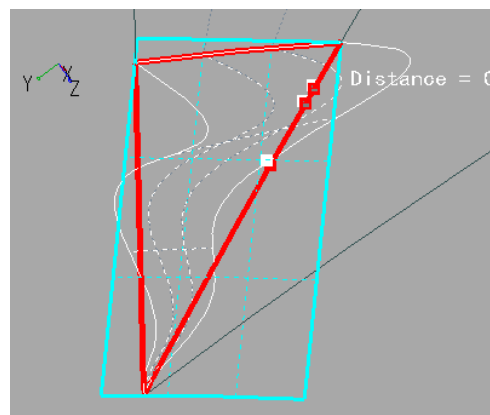
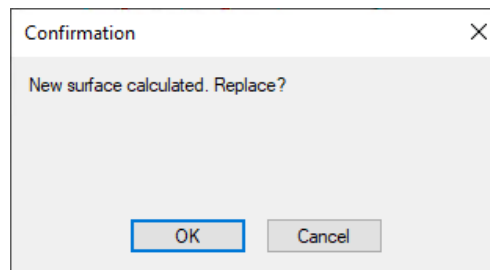


The higher the dependency, the more the geometry of the original surface is reflected, but the time required for calculation increases as well. Normally, use the default settings, and if you want to reflect the geometry of the original surface more accurately, increase the dependency.

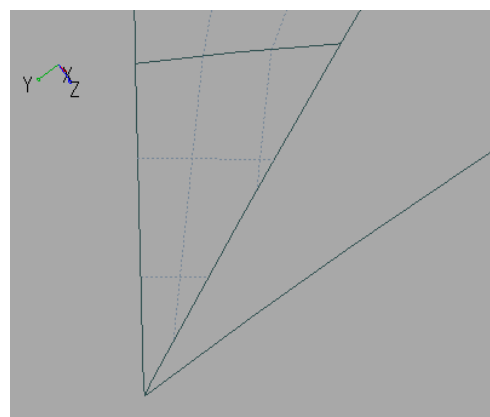
On the other hand, if you don't want to reflect the geometry because the original surface is an invalid geometry, set the dependency toward "Less".

Also, if the quality of the generated surface is poor, adjusting the dependency may improve the quality.


5. A confirmation dialog will appear. Click [OK].



Faces with "Surface with self-intersection" are replaced and all errors have been modified.



3.1.6. Project Edge on Surface

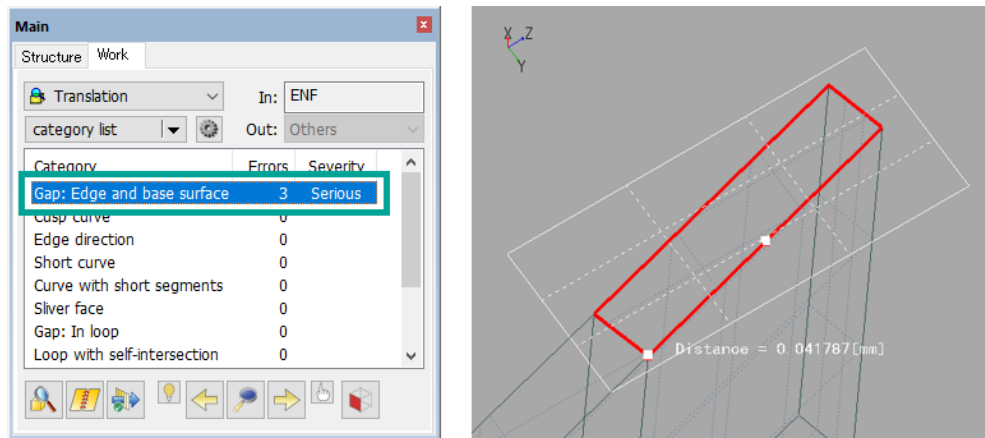
While other modifying functions for "Gap: Edge and base surface" modifies "Surfaces", the function [Project Edge on Surface] () only modifies "Edges". Therefore, please note that this may cause errors in the related elements.

Mainly used for surface such as design intentional surface, cylindrical surface, and spherical surface where you want to strictly maintain the geometry.

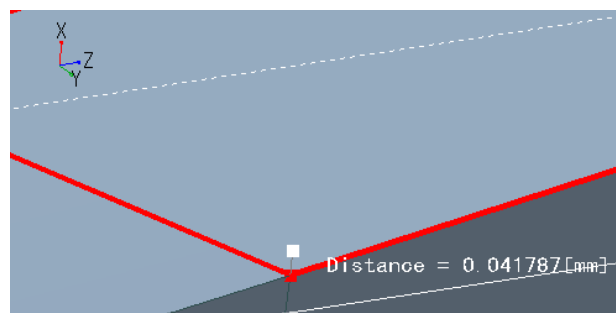
In the following, the characteristics and notes on [Project Edge on Surface] will be explained along with examples of usage.


■ Example of "Project Edge on Surface" being used (1)

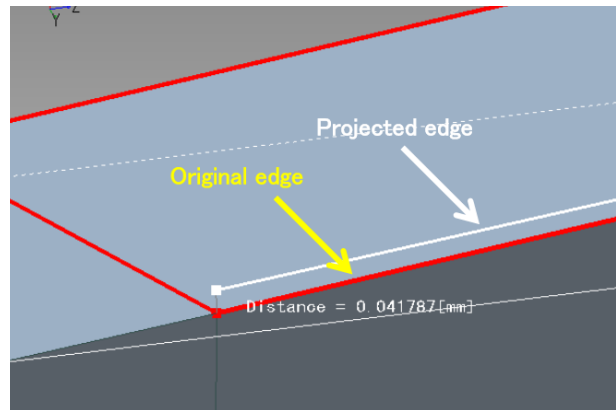
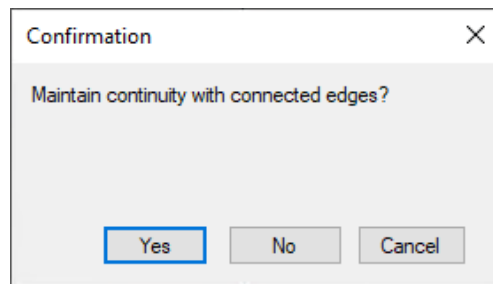
1. Refer to 2.2.1, "Open Files" and open **Projection.drfx** in the <tutorial> folder.
2. In [Main (Work)] panel, select "Gap: Edge and base surface" from the category list. Target position is highlighted in red on "3D View" window.



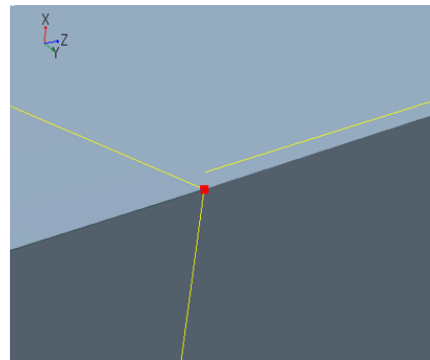
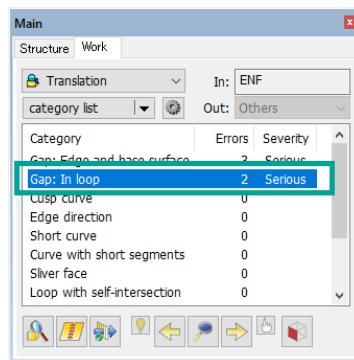
It is possible to confirm that the surface (white) and the curve (red) are apart. In such a case, use [Project Edge on Surface] to project the curve onto the surface.



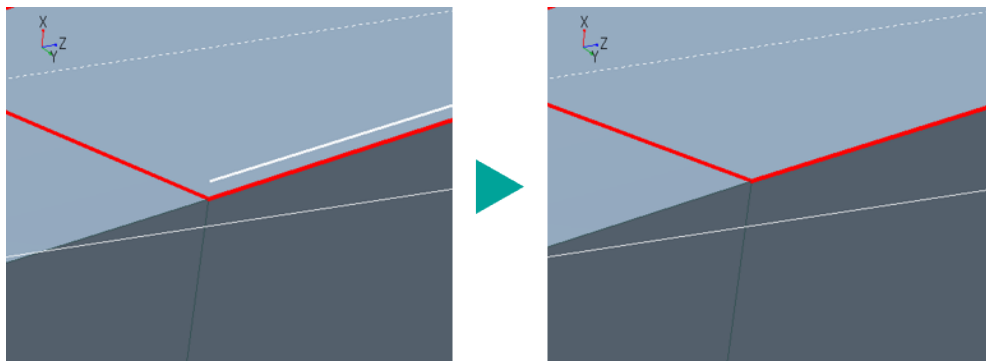
3. Press [Project Edge on Surface] () on Navigation panel.
4. Result of the projection can be previewed in white on "3D View" window, and a dialog will appear to confirm the update of the relevant edges. Click [Yes].



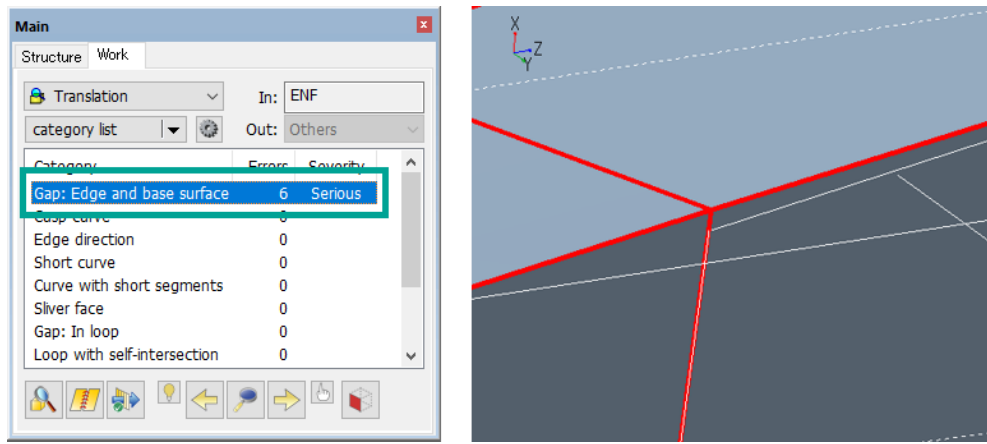
Please note that when you click [No] in the confirmation dialog, only the curves to be modified among the boundaries will be moved. This causes a separation in the boundary and detected as "Gap: In loop". When the relevant edges are not updated, the error "Gap: In loop" may occur.



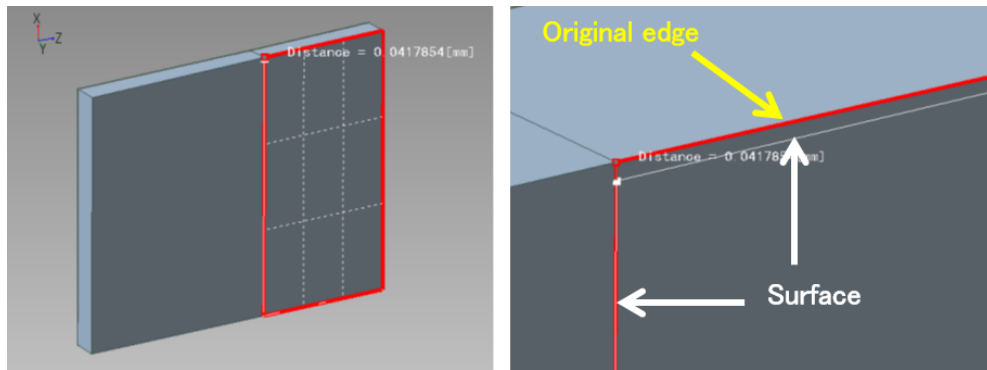
The curve to be modified and its relevant edges are moved.



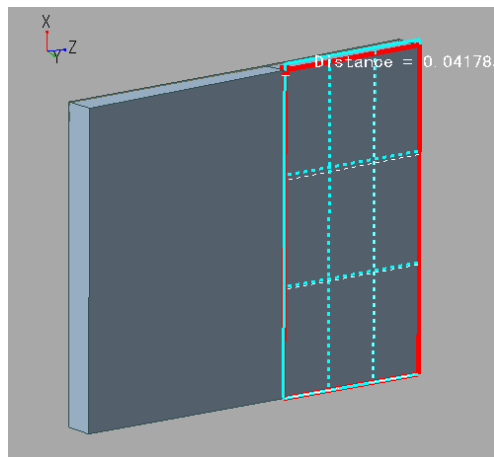
Because the surface moved upward, there is a new gap between the adjacent edge and the surface, increasing the number of errors in "Gap: Edge and base surface".



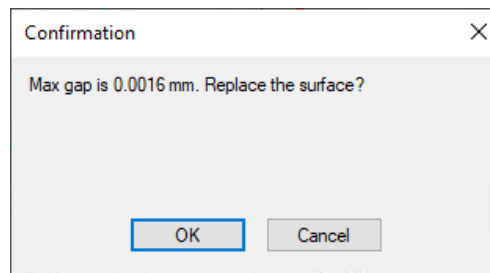
5. In [Main (Work)] panel, use [Next] (→) and [Previous] (←) to switch the following errors.



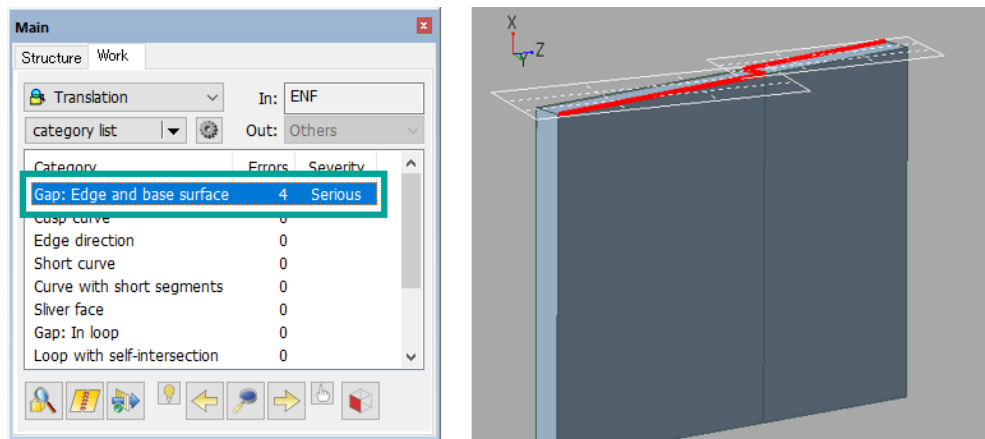
6. Press [Extend Surface] (↗) on Navigation panel.
7. A new surface is calculated and its geometry is highlighted in light blue on "3D View" window.



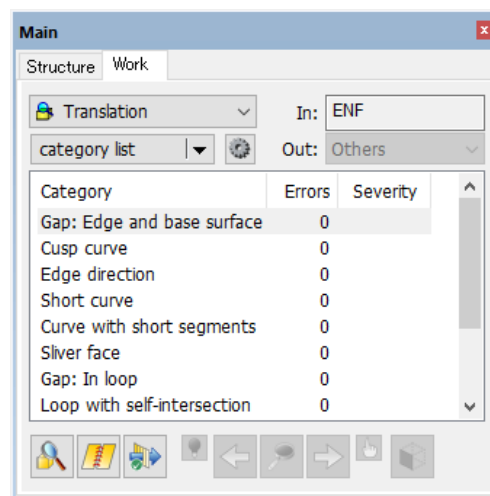
Also, a confirmation dialog will appear. Maximum value of "Gap: Edge and base surface" on the new surface is displayed. Make sure you have made the appropriate modification and click [OK].



The error is modified and the category list is updated.

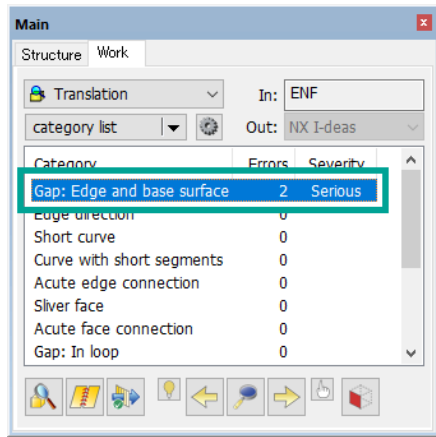



To modify the remaining errors in "Gap: Edge and base surface", use [Fit Face to Loops] () on Navigation panel.

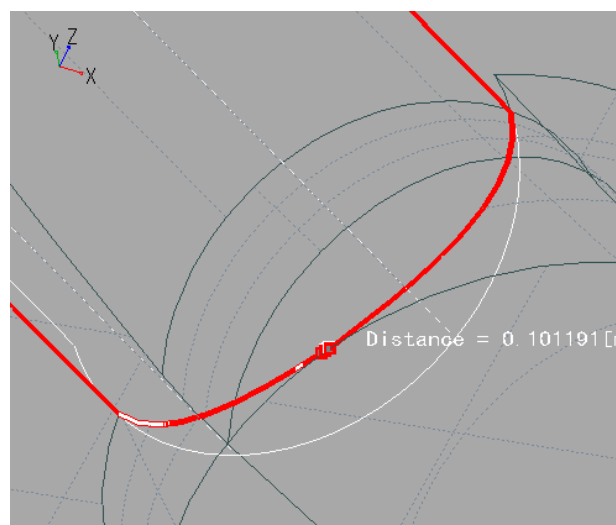
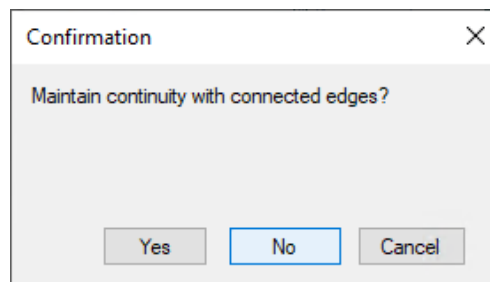


■ Example of "Project Edge on Surface" being used (2)

1. Refer to 2.2.1, "Open Files" and open **Projection2.drfx** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Gap: Edge and base surface" from the category list. Target position is highlighted in red on "3D View" window.



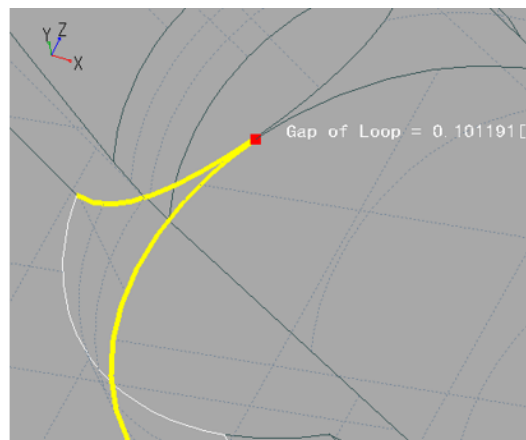
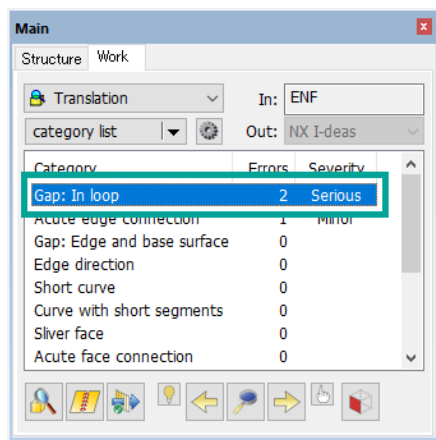
3. Press [Project Edge on Surface] () on Navigation panel.
4. The projection result is previewed in white on "3D View" window, and a dialog confirming the update of relevant edges will appear. Click [No] in this case.



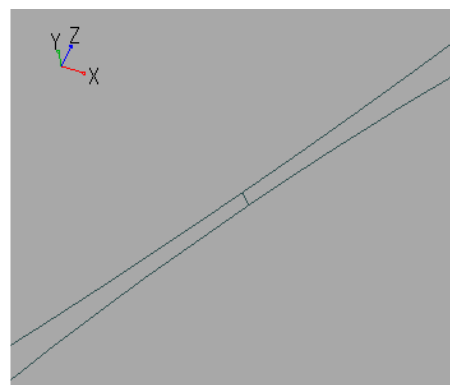
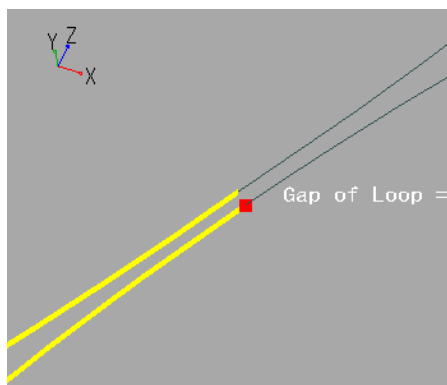
Please note that when an adjacent edge is close, as in the sample model, and updated together, there is a possibility that the modification is not adequate.



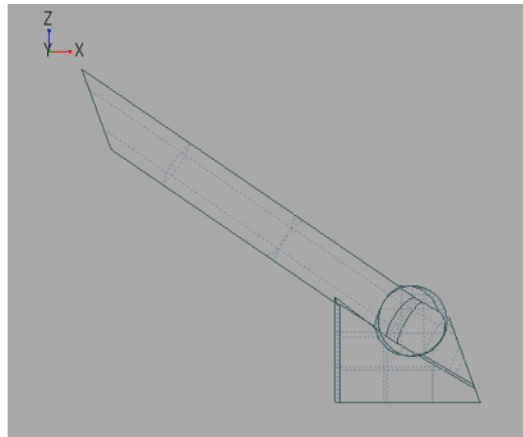
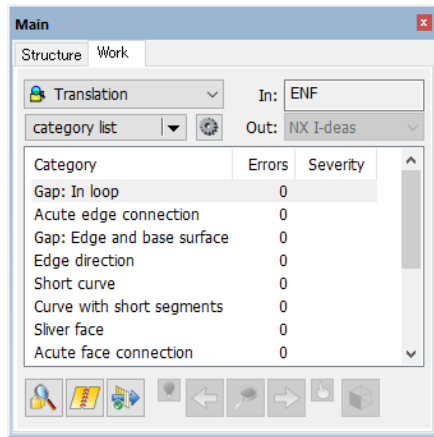
- Similarly, modify one more "Gap: Edge and base surface" with [Project Edge on Surface] () without updating the relevant edges.
All "Gap: Edge and base surface" will be modified; however, because the relevant edges have not been updated, a new error "Gap: In loop" is detected.
- In [Main (Work)] panel, select "Gap: In loop" from the category list. Target position is highlighted in red on "3D View" window.





- Press [Merge Edge Terminals] () on Navigation panel. A curve is inserted into the long and narrow face to modify the error "Gap: In loop".




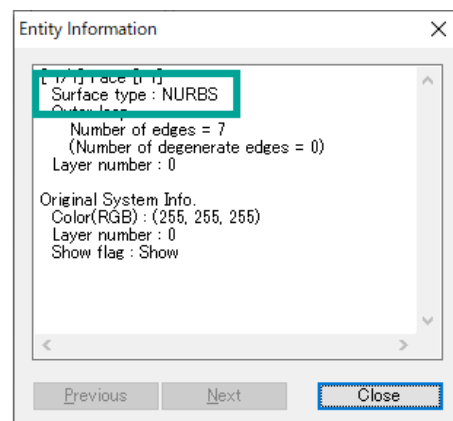
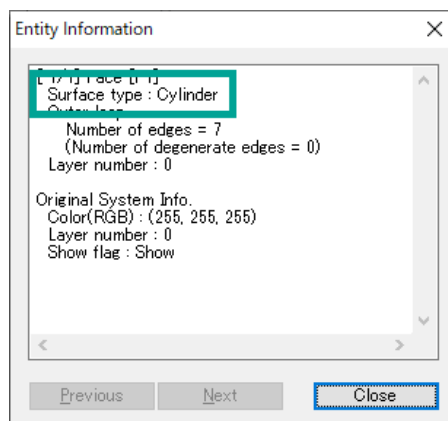
Now all the errors have been modified.



The error "Gap: Edge and base surface" can be modified by the function [Fit Face to Loops] () as well. However, when you modify with [Fit Face to Loops], the surface may deform slightly and may not be a cylindrical surface.

To keep the cylindrical surface as in the sample model, use [Project edge on Surface] () to modify the geometry of the surface.

When confirming from [Analysis] > [Info] (), you can see that the surface modified with [Project Edge on Surface] (lower left) has not changed from the "Cylindrical Surface", while the surface modified with [Fit Face to Loops] (lower right) has changed to "NURBS".




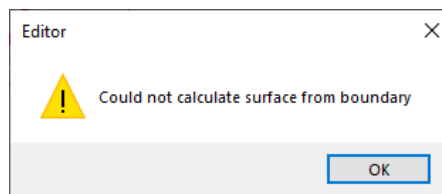
3.2. Suitable Method

The following three ways allow you to check whether the appropriate modification method had been applied to an error.


- Error message during modification
- Surface geometry after modification
- Increase or decrease of detected errors

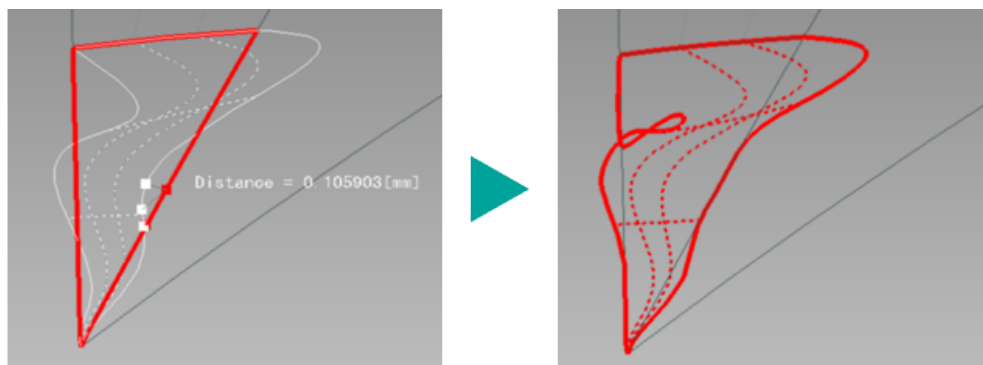
■ Error message during modification

For example, when using [[Boundaries, Surface → Surface](#)] () to modify a face with complex boundaries, the following error message will appear. You will notice that the method is inappropriate.



■ Surface geometry after modification


For example, as a result of executing [[Fit Face to Loops](#)] (), a large oscillated surface may be calculated, as shown in the figure below. In such a case, cancel the modification and execute other modification functions.



■ Increase or decrease of detected errors

Since the error caused by modification is automatically detected, confirm whether the modification was appropriate by looking at the increase or decrease in the number of errors in the category list on [Main (Work)] panel.



Please note that just because the number of errors increased does not necessarily mean that the function is inappropriate. For example, as seen in [[Project Edge on Surface](#)] (), an error "Gap: In loop" may be detected after execution due to the nature of modification.

When the repair icon does not appear

Although an error item is selected from the category list, the repair icon may not appear on Navigation panel for the following reasons:

- Before performing Auto Healing

With the exception of some functions, you cannot perform interactive healing before auto healing.

- Severity is "Minor"

For some minor error items, there aren't any dedicated modification functions because the data can be successfully passed to the output system without modifying the error.

However, try to design the product in such a way as to prevent errors from occurring because they may cause problems in the post processes such as analyzing, manufacturing, etc.



Please refer to [A.1, "Error Items"](#) in the Appendix for details about the severity level of error items.

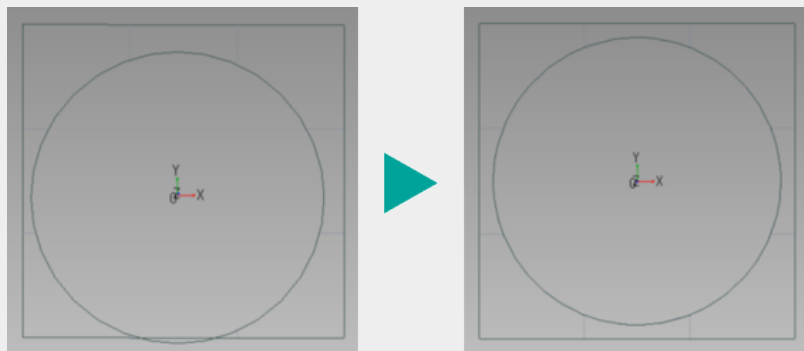
- Errors that can be modified by modifying other error items.

Repair icon may not appear because it is possible to modify an error by modifying another error. An example would be "Gap: Edge and base surface" which can be modified by modifying "Gap: Vertex and surface".

- Items that cannot be modified

If there are no appropriate modification function, the repair icon will not appear. Check the errors and modify them with the source CAD system if necessary. For example, Editor cannot modify the error where the inner loop protrudes over the outer boundary. (Below left figure)

Such an error is detected as "Intersecting loops", so make modification such as changing the size and position of the inner loop in the source CAD system.



3.3. Useful Tools

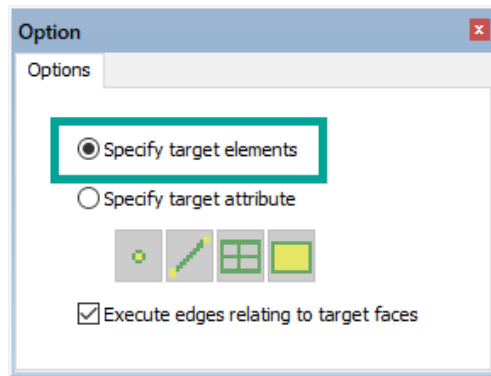
This section will show you some useful tools available for Interactive Healing with Editor.

3.3.1. Pick Filter

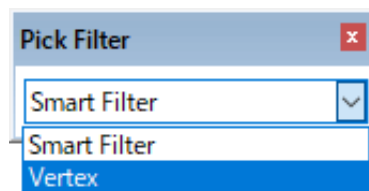
Pick Filter will allow you to filter the target elements to be selected on "3D View" window. To multiselect the target elements, it will be easier to select by limiting the target elements in Pick Filter.

(Example) When deleting unnecessary single points

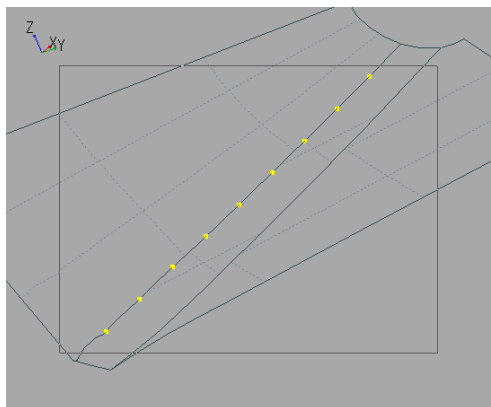
1. Select [Edit] > [Delete] from the menu or select [Delete] (✖) on the toolbar.
2. In [Option] panel, select "Specify target elements".



3. Switch the Pick Filter to "Vertex".



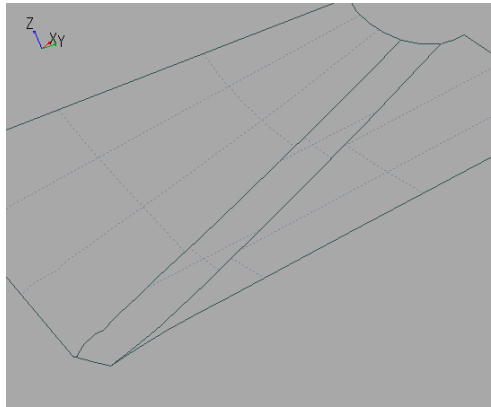
4. At the upper right of "3D View" window, press [Area Selection Mode] (☐). Left-click and drag to enclose the points you want to delete.





Also, select elements by left-clicking while holding down [Ctrl] key and dragging to enclose with a rectangle.

5. Press [Done] (✓). Only points within the selected area will be deleted.



3.3.2. Clipping Box

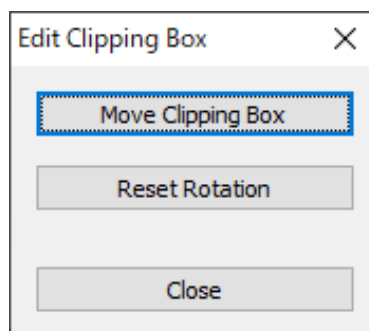
Editor Clipping Box will allow you to display only those elements that are contained in a rectangle (clipping box) centered at a specified position.

When working with only the elements inside an intricate model, you can easily do so by hiding the elements outside the target position. The position, shape and angle of Clipping Box can be adjusted freely.

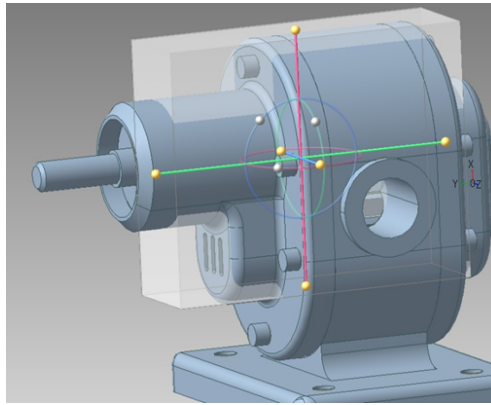
■ How to Use Clipping Box

1. Select [View] > [Edit Clipping Box] from the menu or select [Edit Clipping Box] (📏) on the toolbar.

The Clipping Box and "Edit Clipping Box" dialog will appear on "3D View" window.



2. In "Edit Clipping Box" dialog, click [Move Clipping Box] and pick any element on the model. The element you have picked becomes the center of Clipping Box.




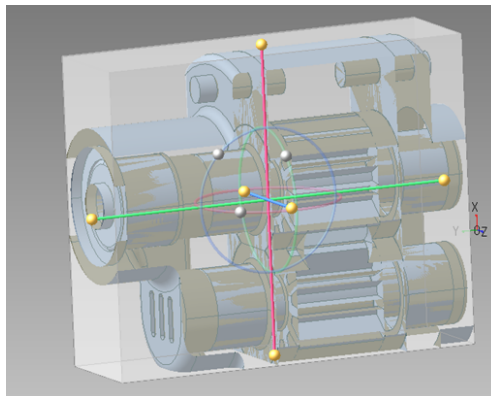
Please note that the size of Clipping Box is automatically determined by the size of "3D View" window.

3. Move the handles inside Clipping Box to adjust the size and orientation of the box.

- Yellow handle: Handle for changing the size
- Gray handle: Handle for adjusting the angle

After editing, click [Close] in the "Edit Clipping Box" dialog to finish the positioning.

4. Select [View] > [Clipping Box (On/Off)] from the menu or enable [Clipping Box (On/Off)] () on the toolbar. Only the elements inside Clipping Box are displayed.




While the displayed area is limited, perform such actions as deleting elements inside the model.

4. Advanced



Most CAD data can be properly translated using the modification methods you have learned up to the "3, *Intermediate*" level. However, more sophisticated operations may be required depending on the state of the source data.

This chapter will explain how to modify errors that require technical operations in the form of case studies.

4.1. Modifying Free Edge

When importing data that do not have connection information between faces, it is possible to combine free edges by [Auto Stitch] (). However, it may not be possible to combine all free edges.

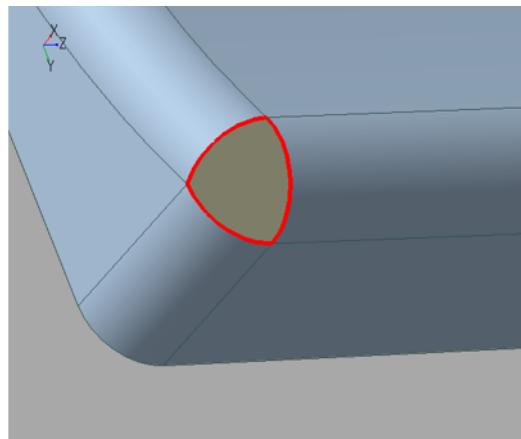
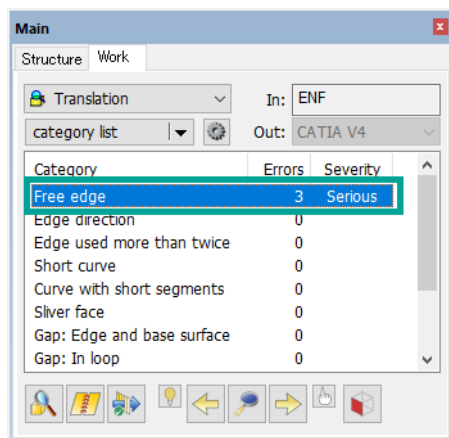


In each case, [Auto Stitch] () and [Auto Heal] () have already been performed beforehand.


4.1.1. Case: Missing Face

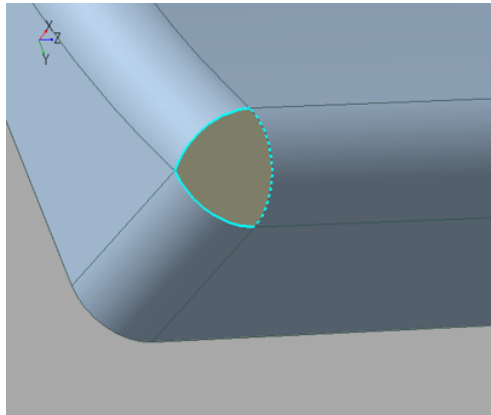
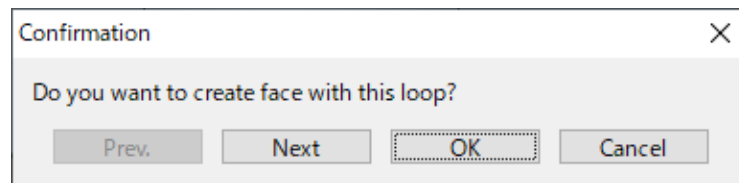
When importing a file such as IGES, the face may be missing at the fillet position, etc. If the face is missing, the boundary is recognized as "Free edge".

1. Refer to 2.2.1, "Open Files" and open **FillHole.drfx** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Free edge" from the category list. Target position is highlighted in red on "3D View" window.

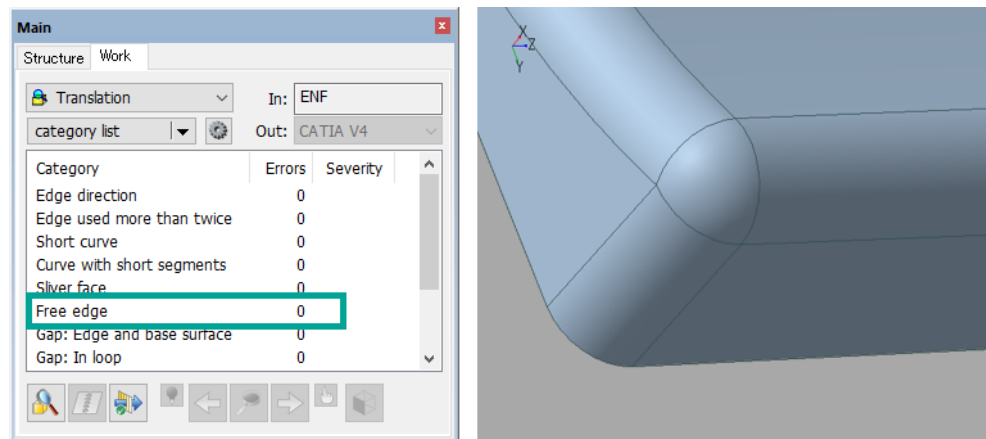


In this sample model, the face is missing in the fillet position. Fill in the face to modify the error.

3. Press [Fill Hole (Create New Face)] () on Navigation panel.
4. A confirmation dialog will appear. Confirm that the edges you want to fill are highlighted correctly on "3D View" window, and click [OK].



A new face is created in the target position, and the category list is updated.

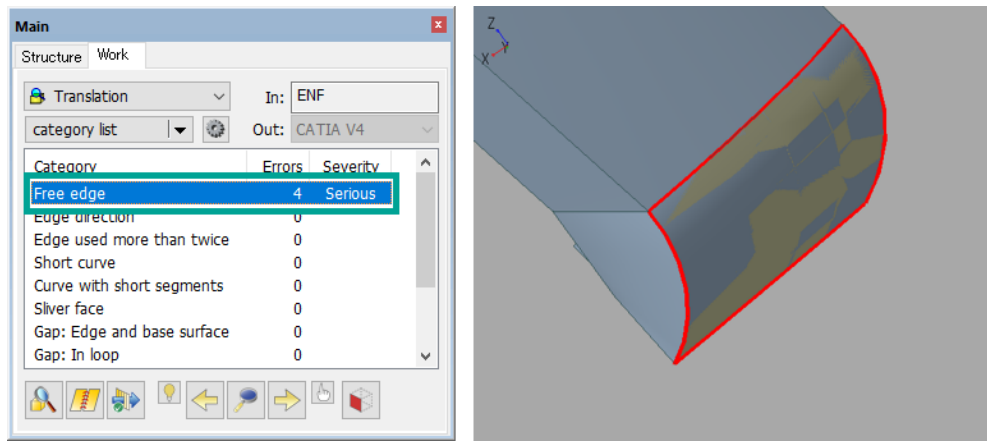


4.1.2. Case: Duplicate Faces

When importing such files as IGES file, the faces may be duplicated.

When the faces are duplicated in this way, the boundaries without adjacent faces are recognized as free edges.

1. Refer to 2.2.1, “Open Files”, and open **RemoveDuplicateFace.drfx** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Free edge" from the category list. Target position is highlighted in red on "3D View" window.

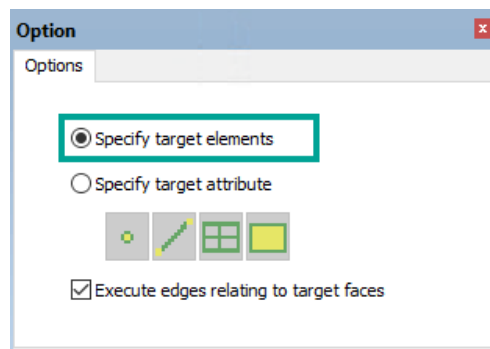


Faces are duplicated in this position. In this case, delete one of the duplicated faces, and update the connection information between the faces with [Auto Stitch].



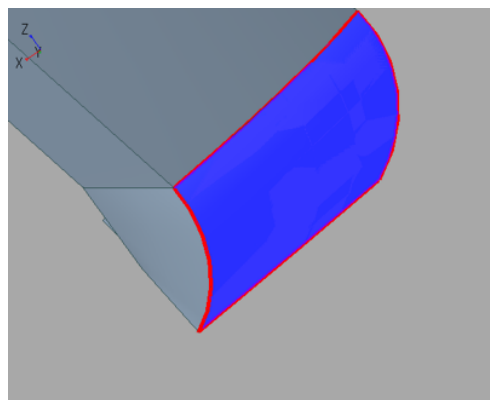
When finding the duplicated position is difficult, select [View] > [Regen] (🔄) from the menu to update the "3D View" window.

3. Select [Edit] > [Delete] from the menu or click [Delete] (✖) on the toolbar.
4. In [Options] panel, select "Specify target elements".

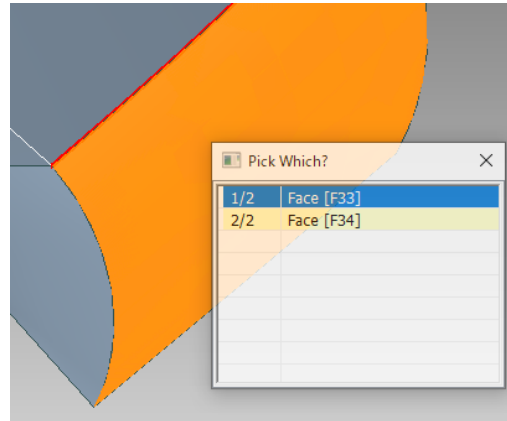


Pick the duplicated face on "3D View" window, and press [Done] (✔).

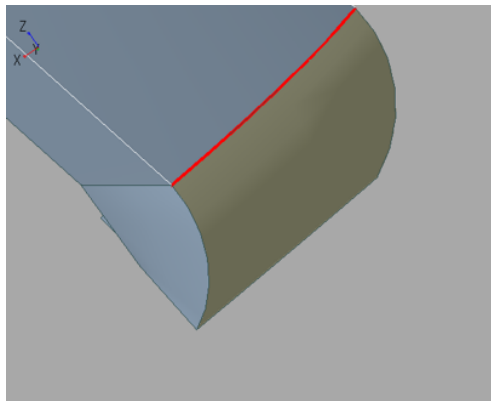
* In this case, delete either of the face.



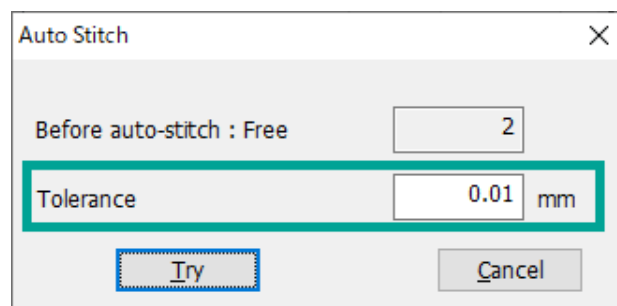
To specify a face to delete, right-click on the duplicated face and "Pick Which?" dialog will appear. Specify the candidate in the dialog.



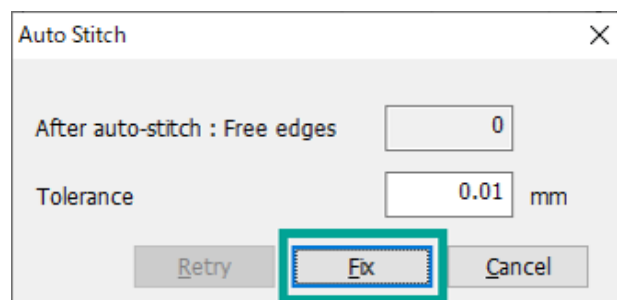
One face has been deleted. Press [Quit (Esc)] (✖) to exit the command.



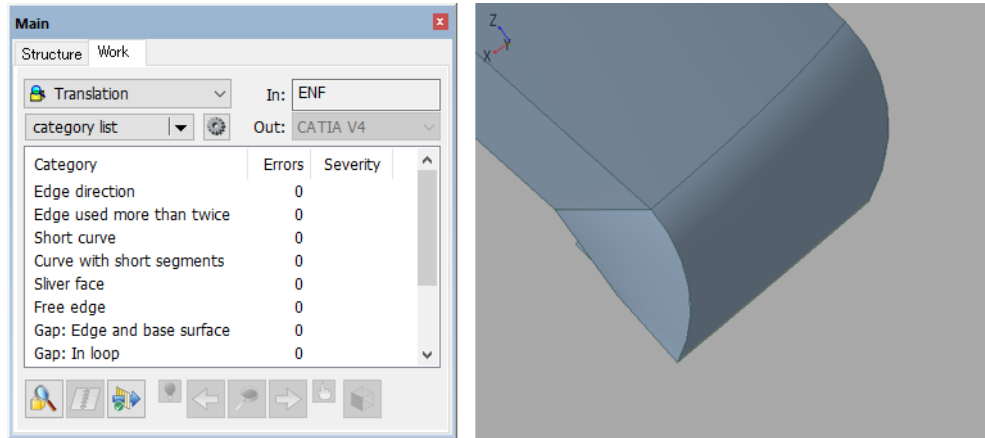
5. In [Main (Work)] panel, press [Auto Stitch] (🔧).
6. "Auto Stitch" dialog will appear. Specify "Tolerance" to 0.01mm and click [Try].



7. Make sure that "After auto-stitch: Free edges" is "0", and click [Fix].



Auto stitching is executed, and the category list is updated.

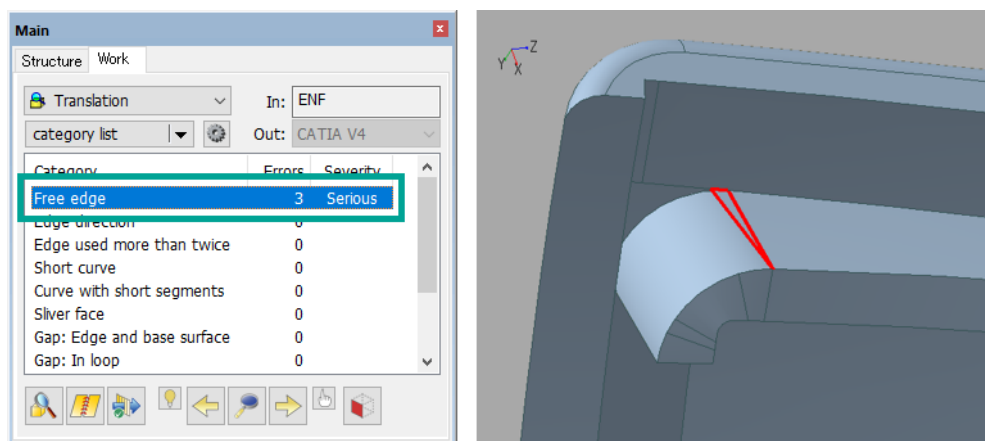


After stitching, if the face direction needs to be modified, select [Modify] > [Repair Solid] > [Flip Face] (🔄) to reverse the face. Please refer to "[Flip Face](#)" for more details.

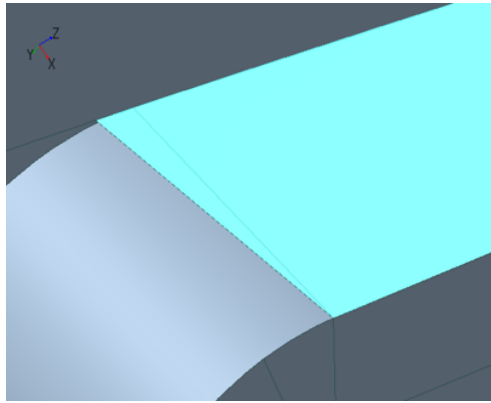
4.1.3. Case: Fail to Trim

When importing files such as IGES, faces which fail to be trimmed may exist. In this case, the boundary of the face that failed to be trimmed is recognized as a free edge.

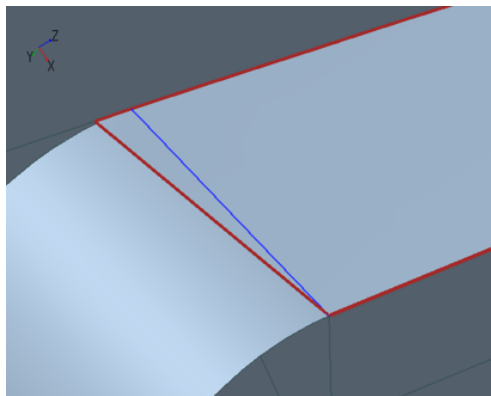
1. Refer to 2.2.1, "Open Files" and open **FailToTrimSurface.drfx** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Free edge" from the category list. Target position is highlighted in red on "3D View" window.



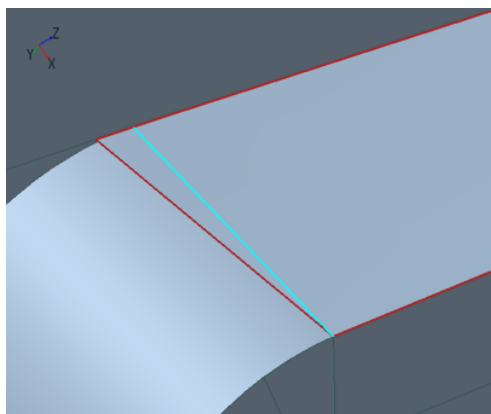
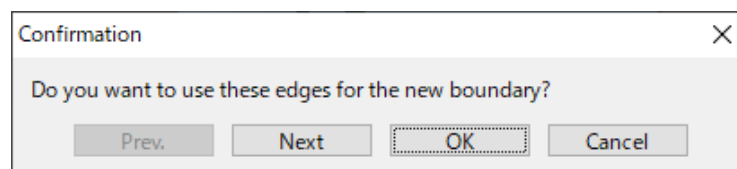
3. Select [Modify] > [Repair Solid] > [Change Face Boundary] from the menu or click [Change Face Boundary] (🔍) on the toolbar.
4. Pick the face that has failed trimming on "3D View" window.



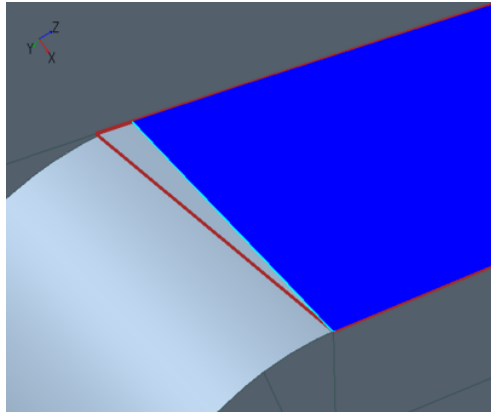
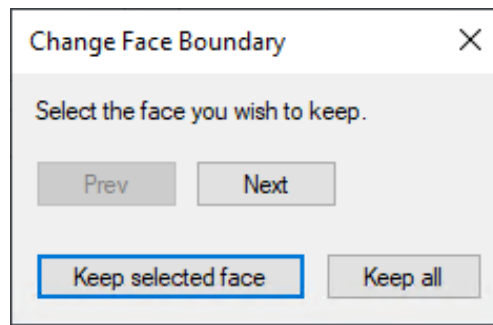
5. Pick the edges to use as the new boundary, and press [Done] (✓).



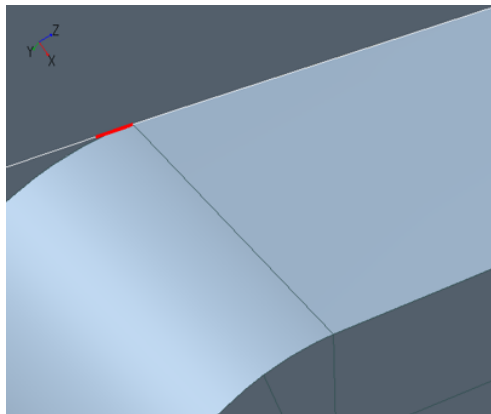
6. A confirmation dialog will appear. Confirm that the edges highlighted on "3D View" window are correct, and click [OK].



7. "Change Face Boundary" dialog will appear. Click either [Next] or [Previous] to switch to the state where the face you wish to keep is highlighted in blue. Click [Keep selected face].

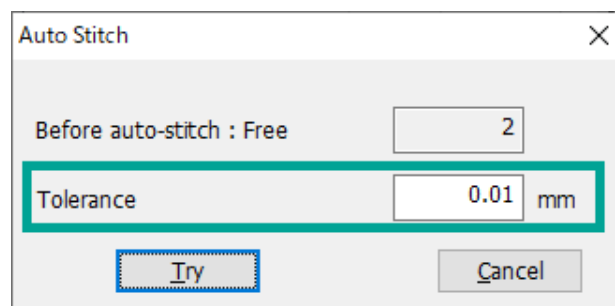


The boundary has been modified. Press [Quit (Esc)] (✖) to exit the command.

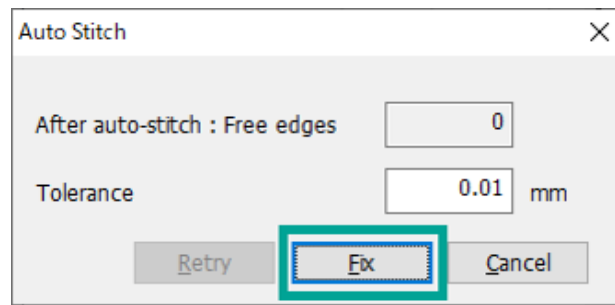


Finally, update the new boundary's connection information.

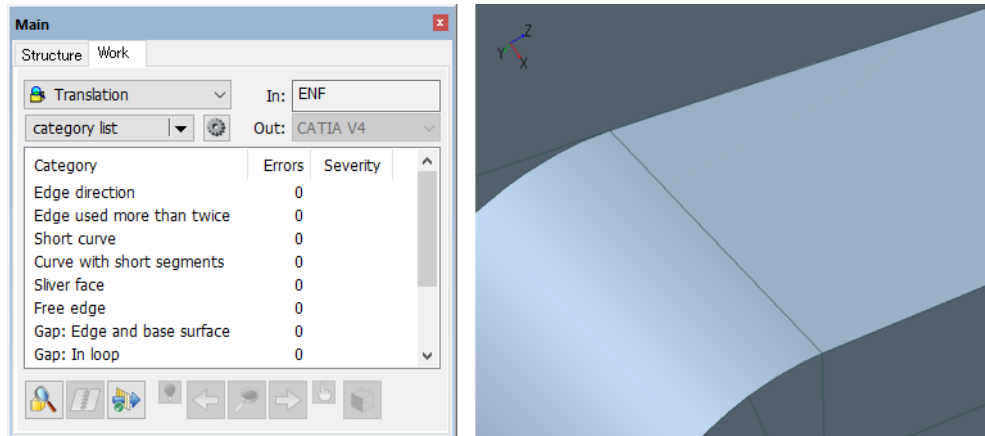
8. In [Main (Work)] panel, press [Auto Stitch] (🧵).
9. "Auto Stitch" dialog will appear. Specify "Tolerance" to 0.01mm and click [Try].



10. Confirm that "After auto-stitch: Free edges" is "0", and click [Fix].



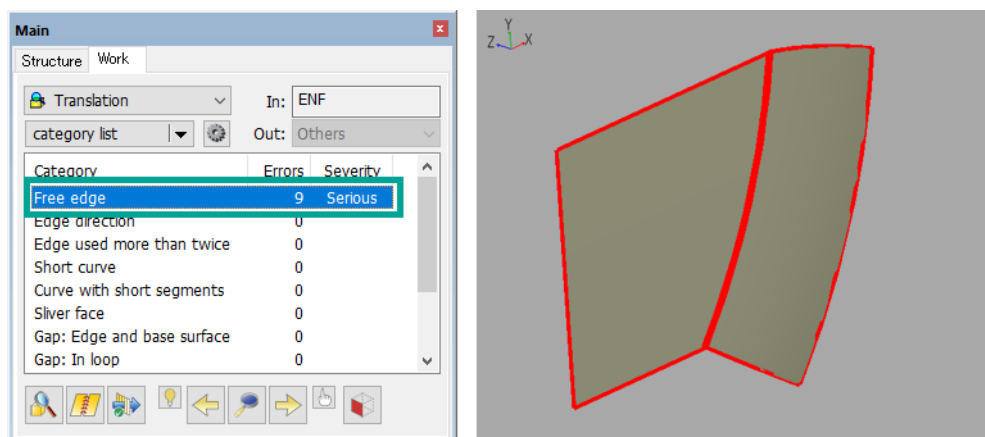
Auto stitching is executed and the free edges are modified.



4.1.4. Case: Huge Gap between Edges

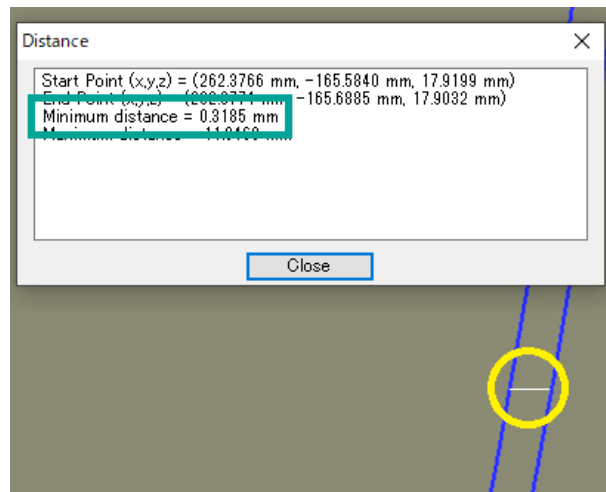
When faces cannot be stitched by auto stitching due to the large gap between the edges, the position is recognized as a free edge.

1. Refer to 2.2.1, "Open Files" and open **EdgeStitch.drfx** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Free edge" from the category list. Target position is highlighted in red on "3D View" window.



Auto stitching cannot be used because the gap is too far apart between the edges of two faces.

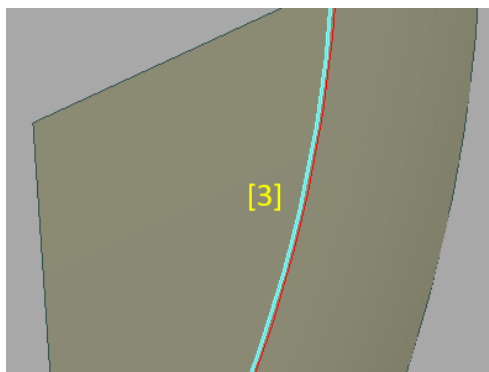
Shortest distance between the edges is approximately 0.3mm. Select [Analysis] > [Distance] (📏) to measure the distance between the two edges.



3. Select [Modify] > [Repair Solid] > [Stitch (pick)] from the menu or click [Stitch (pick)] (🔗) on the toolbar.
4. For the first edge, pick both [1] and [2] on "3D View" window.



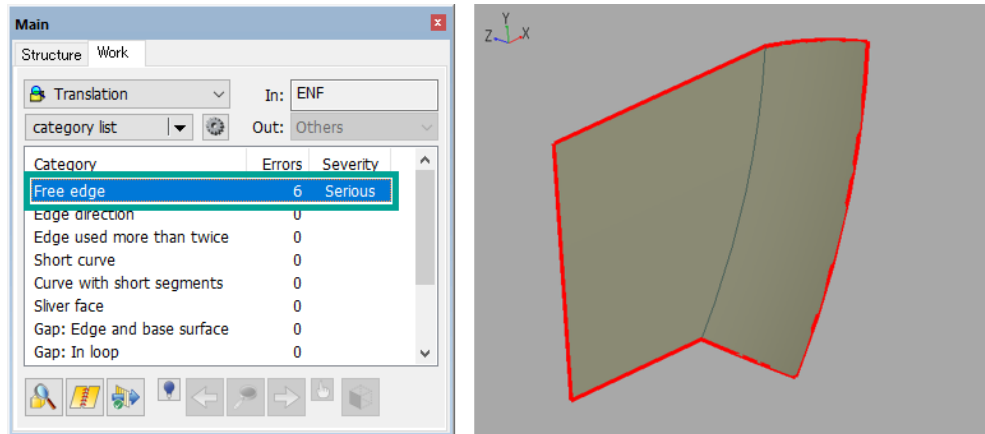
Next, for the second edge, pick [3] on "3D View" window. Press [Done] (✅).





It is also possible to stitch edges that are far apart by loosening the tolerance and performing auto stitching; however, because auto stitching covers all edges that have no connection, if the tolerance is too loose, the edges may be stitched together with the wrong edges than what the designer intended.

Edges [1]~[3] are stitched together, and the category list is updated.

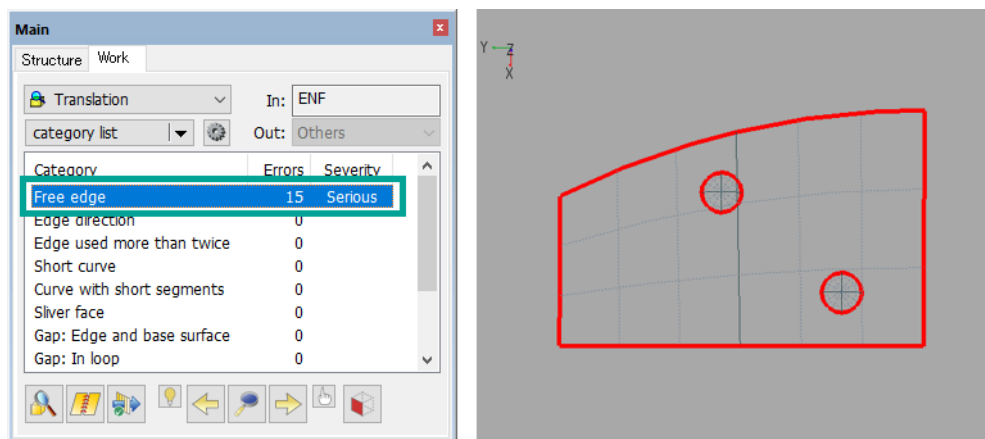


Please note that for models that are not originally solids, such as an IGES file output from a surface model, the outer free edges will remain after auto stitching, but this is not an error and does not need to be modified.

4.1.5. Case: No Adjacent Faces

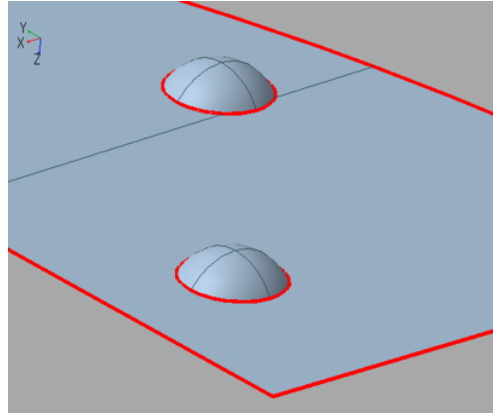
When a model is not originally a solid, such as an IGES file output from a surface model, the outer free edges may remain because there are no adjacent faces.


1. Refer to 2.2.1, "Open Files" and open **FaceTrim.drfx** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Free edge" from the category list. Target position is highlighted in red on "3D View" window.



Because the flat face is not adjacent to the hemispherical protrusions, the position around the two hemispherical protrusions are recognized as "Free edge". Errors can be modified by

retrimming the faces so that they are adjacent to the hemispherical protrusions.



Select [Analysis] > [Related Element] from the menu or click [Related Element] () on the toolbar. Confirm the element related to the specified element. After modification, you can confirm that both faces are adjacent to the hemispherical protrusion.

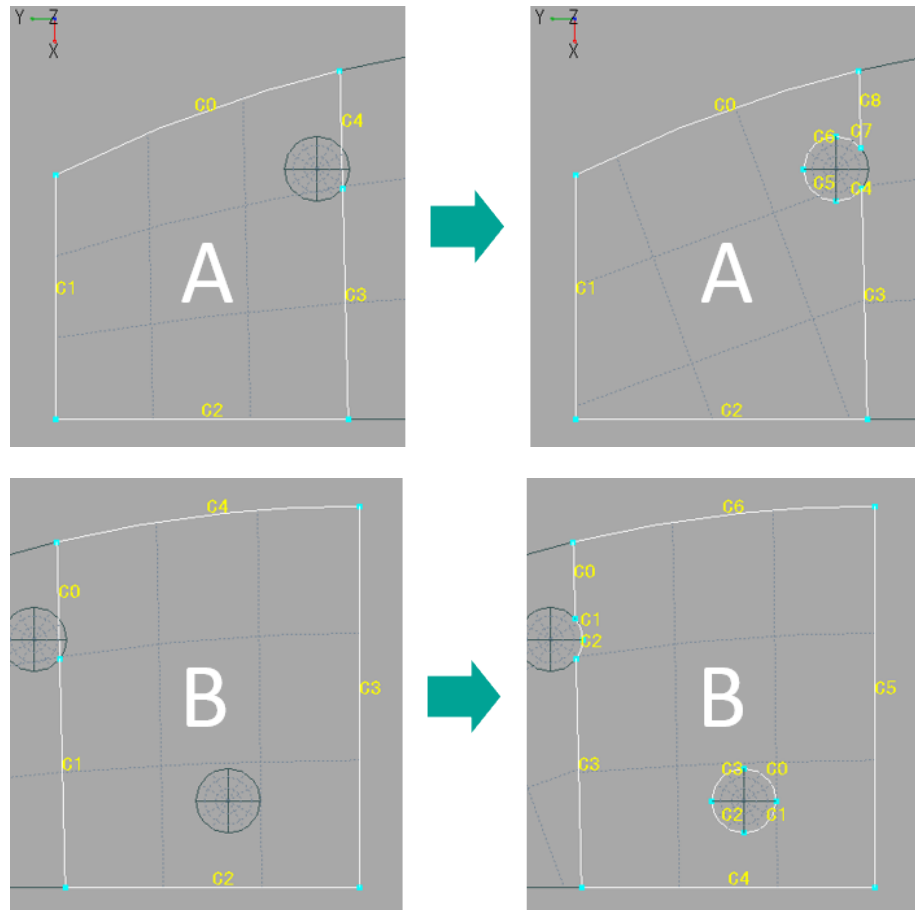



Figure 1. Adjacent relation between face and hemispherical protrusions (Before > After)


How to modify Face A and B.

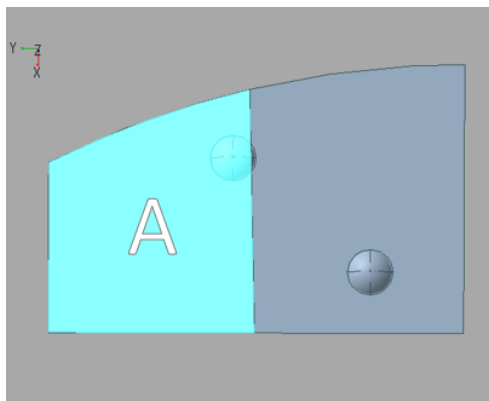


When the free edge is highlighted on "3D View" window, it is difficult to see the modification, so press [Zoom current target] () on [Main (Work)] panel. Detected errors are not selected and highlight will be canceled on "3D View" window.

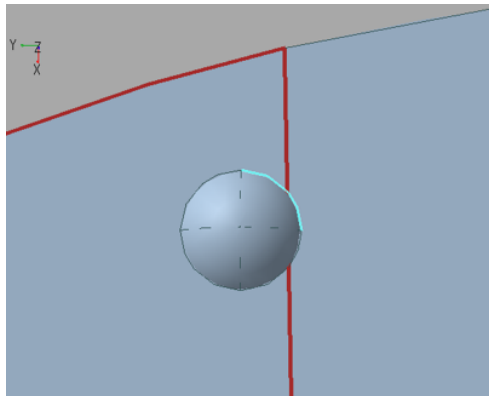
■ Change Boundary of Face A

Modify the boundary of Face A according to the geometry of hemispherical protrusion.

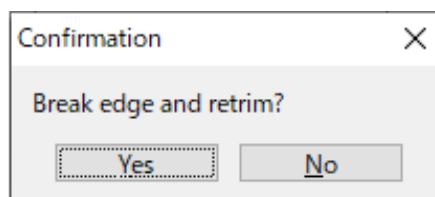
1. Select [Modify] > [Repair Solid] > [Change Face Boundary] from the menu or click [Change Face Boundary] () on the toolbar.
2. Select Face A on "3D View" window.

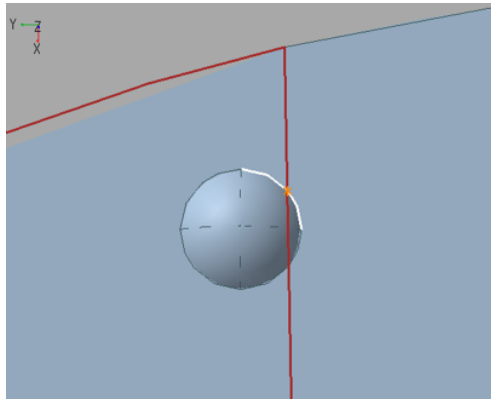


3. Pick a curve (light blue edge) that intersects the existing boundary, and press [Done] ().

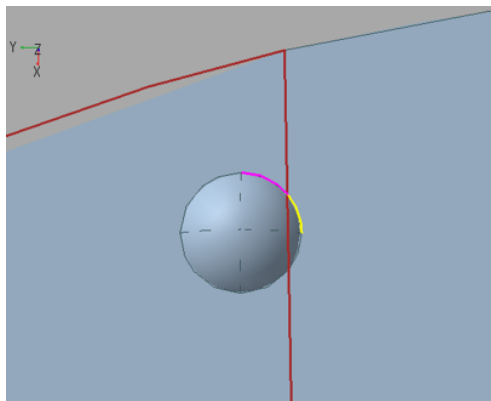


4. A confirmation dialog will appear. Confirm that the trimming position is the same as the image below, and click [Yes].

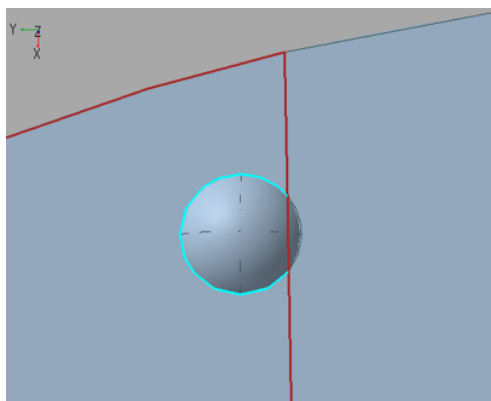
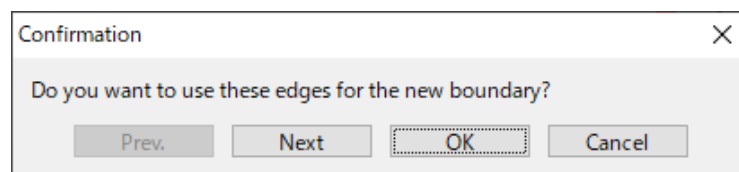




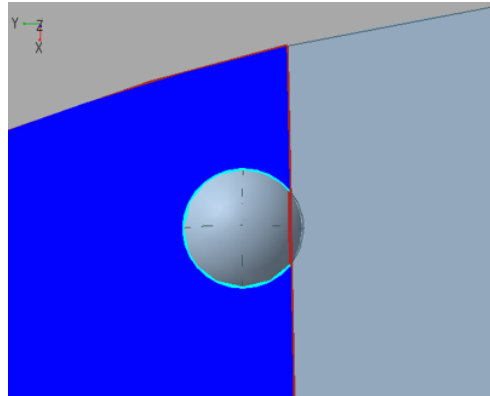
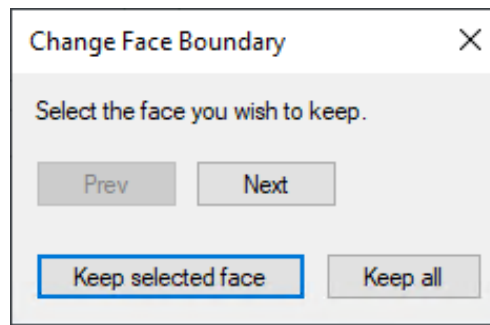
5. Select the edge that will be the boundary of the new face (pink edge).



6. A confirmation dialog will appear. Confirm that the edges highlighted on "3D View" window are correct, and click [OK].



7. "Change Face Boundary" dialog will appear. Click either [Next] or [Previous] to switch to the state where the face you wish to keep is highlighted in blue. Click [Keep selected face].



Face A boundary has been modified. Press [Quit (Esc)] (⌫) to exit the command.

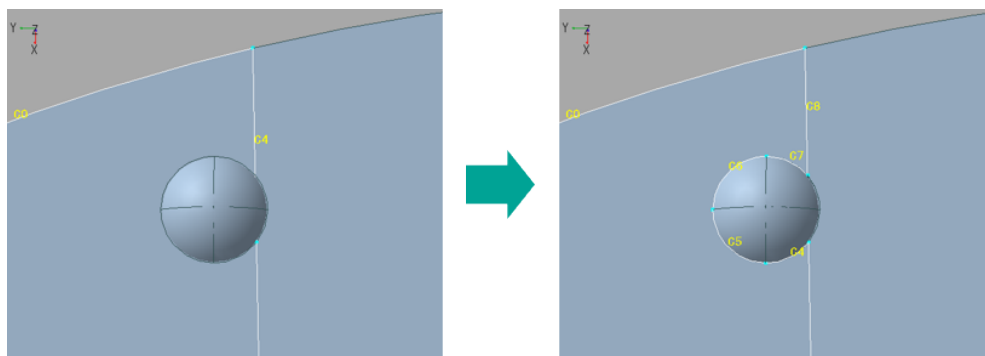
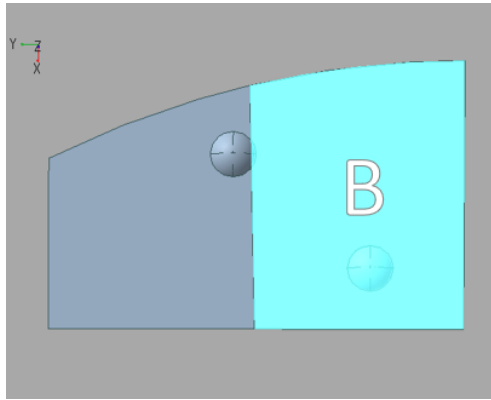


Figure 2. Adjacent relation of Face A (Before > After)

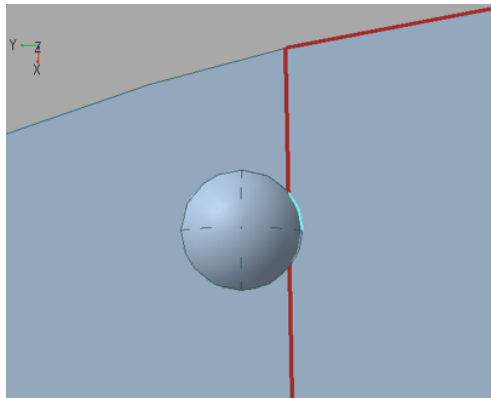
■ Change Boundary of Face B

As with Face A, modify the boundary of Face B to match the geometry of the hemispherical protrusion.

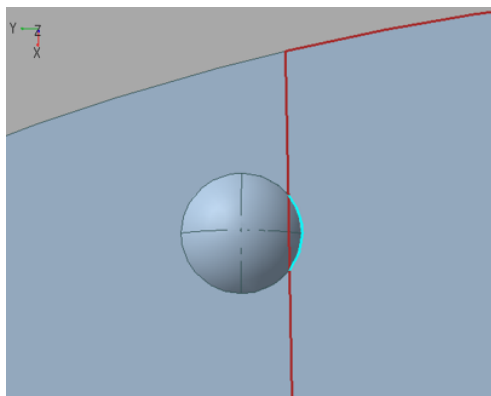
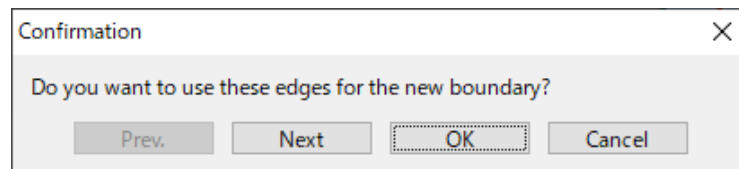
1. Select [Modify] > [Repair Solid] > [Change Face Boundary] from the menu or click [Change Face Boundary] (🔧) on the toolbar.
2. Select Face B on "3D View" window.



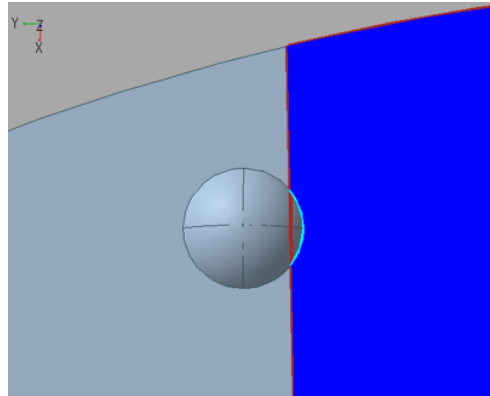
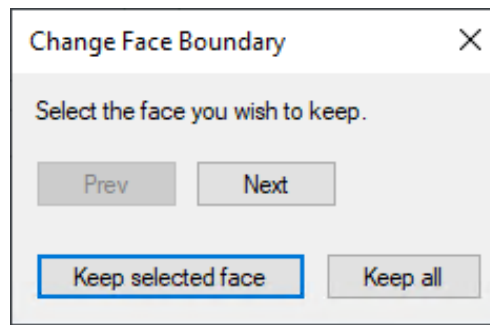
3. Pick a curve (light blue edge) which intersects with the existing boundary, and press [Done] (✓).



4. A confirmation dialog will appear. Confirm that the edge highlighted on "3D View" window is correct, and click [OK].



5. "Change Face Boundary" dialog will appear. Click either [Next] or [Previous] to switch to the state where the face you wish to keep is highlighted in blue. Click [Keep selected face].



Boundary of Face B has been modified.

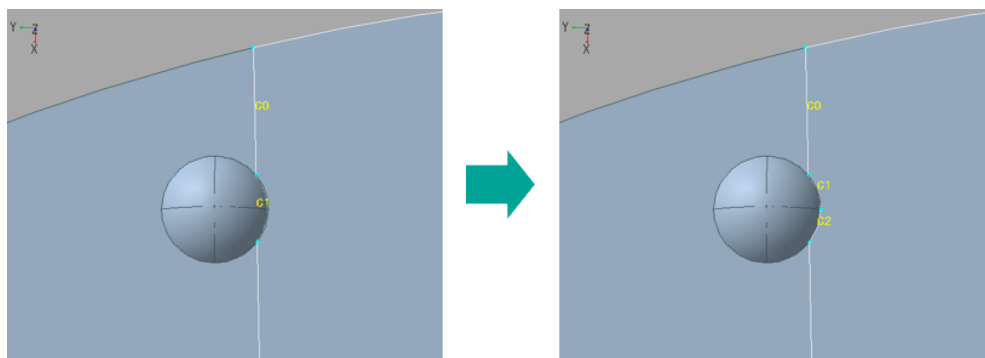
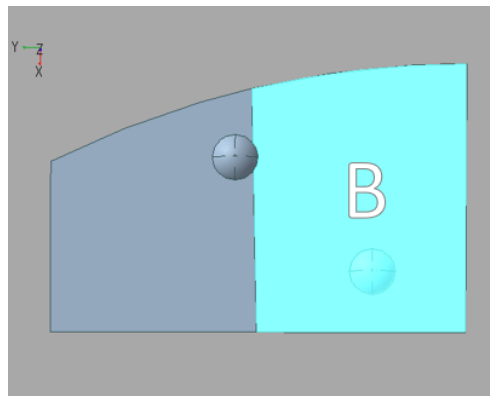


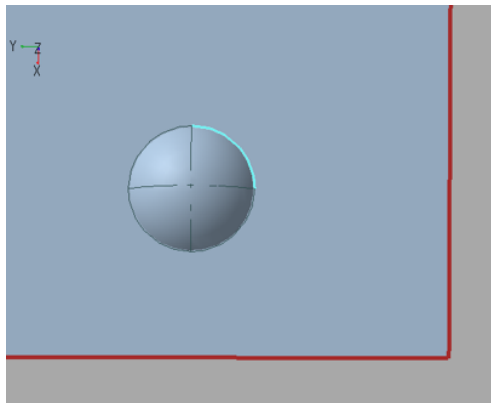
Figure 3. Adjacent relation of Face B (Before > After)

Finally, use the same procedure to modify Face B so that the hemispherical protrusion at the bottom right connects with the face.

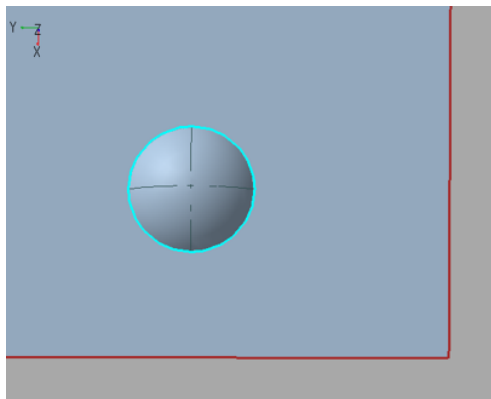
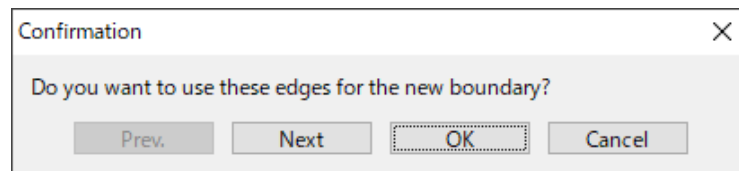
6. Again, select Face B on "3D View" window.



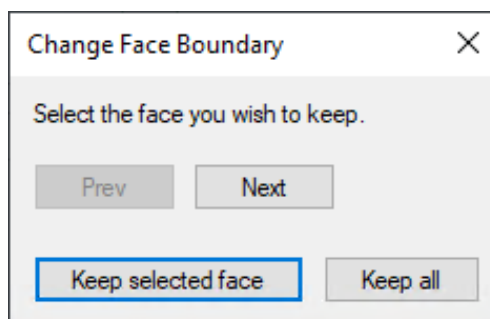
7. Pick an edge from the outer boundary of the hemispherical protrusion at the bottom right of the model, and press [Done] (✔).

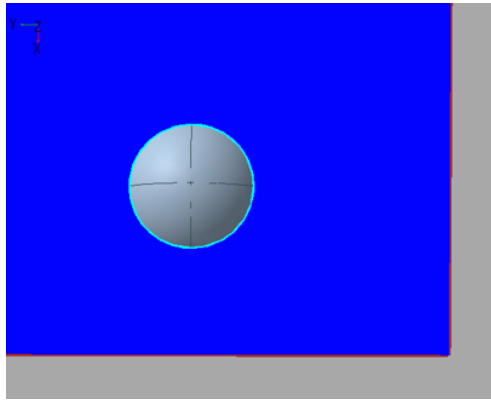


8. A confirmation dialog will appear. Confirm that the edge highlighted on "3D View" window is correct, and click [OK].



9. "Change Face Boundary" dialog will appear. Click either [Next] or [Previous] to switch to the state where the face you wish to keep is highlighted in blue. Click [Keep selected face].





Boundary of Face B has been modified.

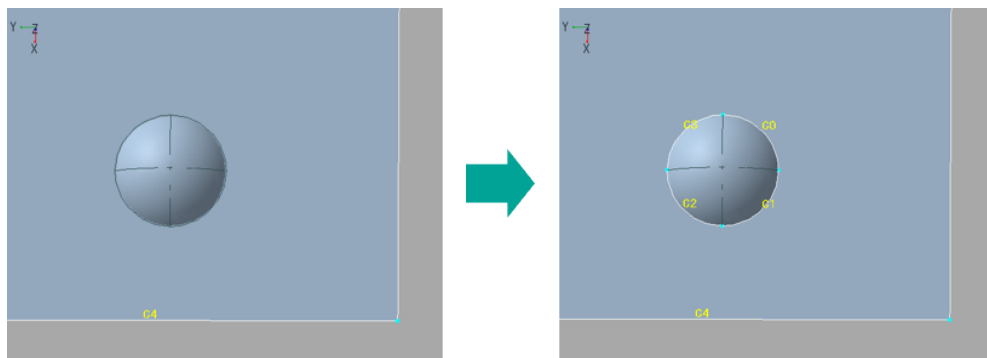


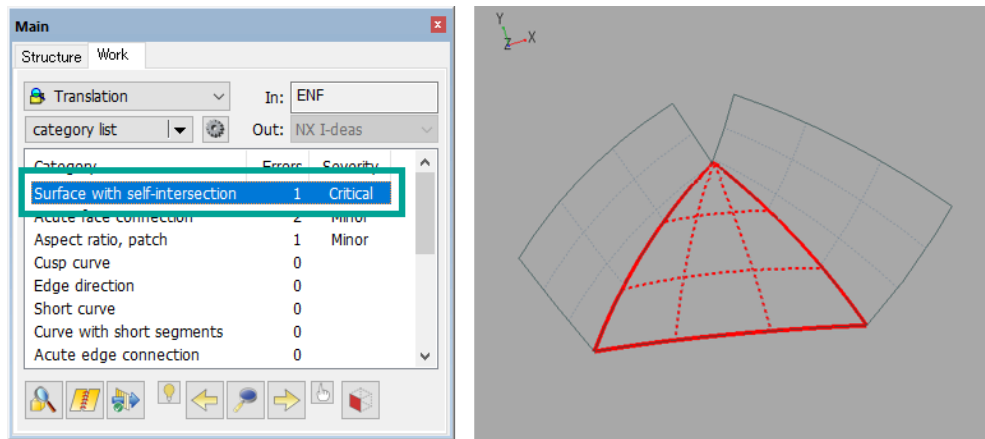
Figure 4. Adjacent relation of Face B (Before > After)

4.2. Modifying Invalid Geometry

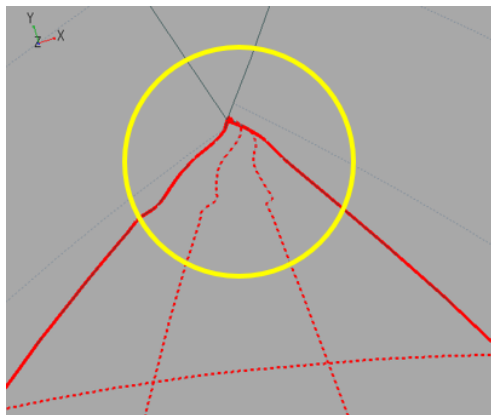
In some cases, the imported CAD model may have unpreferable geometry such as wavy curves. This section will explain how to modify them.

4.2.1. Modify Curve with Oscillation (1)


1. Refer to 2.2.1, “Open Files” and open **InvalidGeometry1.drfx** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Surface with self-intersection" from the category list. Target position is highlighted in red on "3D View" window.

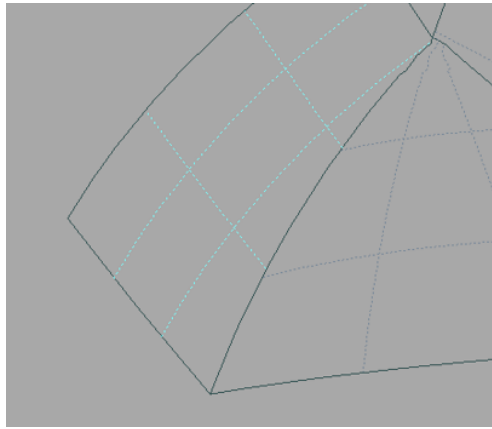


The face in the center of "3D View" window intersects itself. Furthermore, when you zoom up the tip of that face, you can see the curves oscillating. Such oscillations are undesirable for data quality.

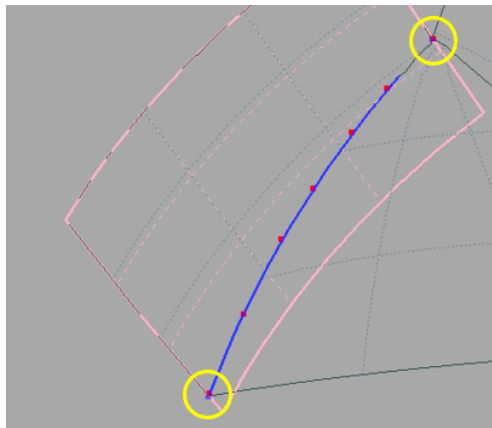


Create a curve which replaces the oscillating curve on both sides of the surface.

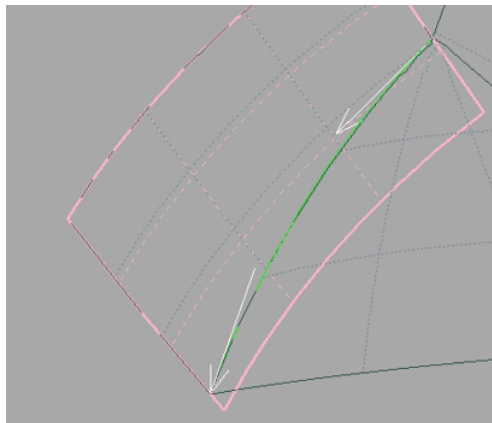
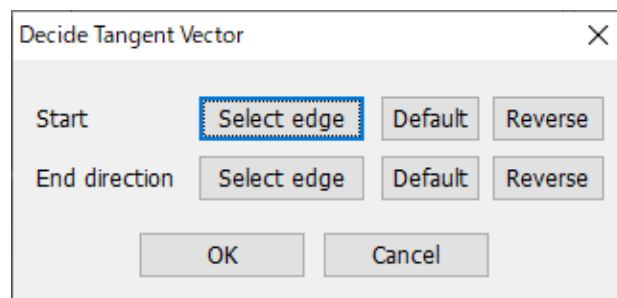
3. Select [Create] > [Curve] > [On Surface] from the menu or click [On Surface] () on the toolbar.
4. Select the face on the left.



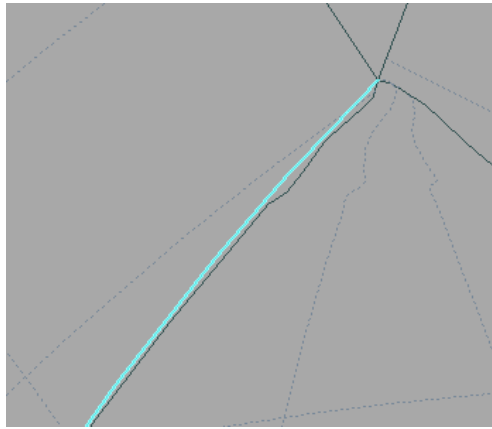
5. Pick two places on the curve as shown below, and press [Done] (✓).



6. "Decide Tangent Vector" dialog will appear. Accept the default setting and click [OK].

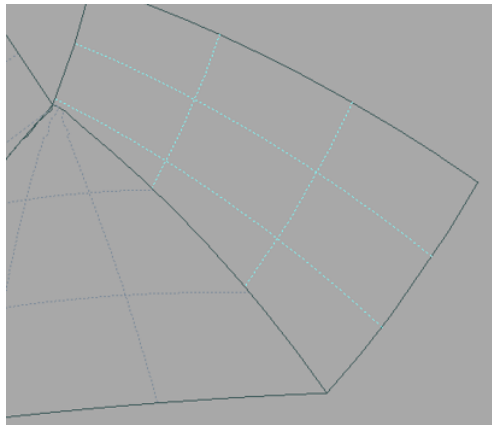


A new curve is created. (The side highlighted in light blue)

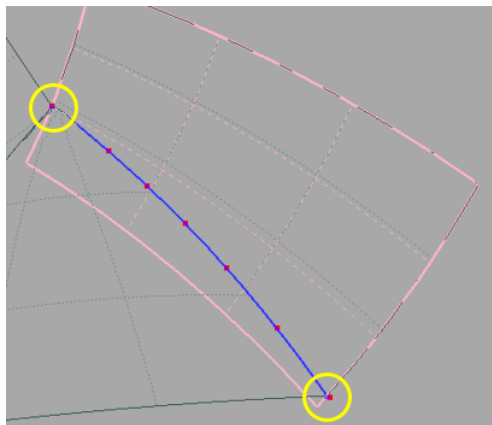


Next, create a new curve on the right side of the face as well.

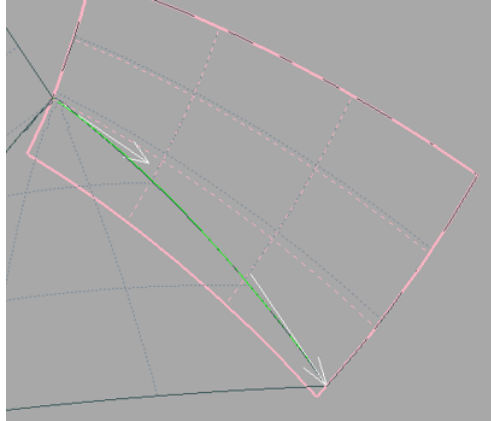
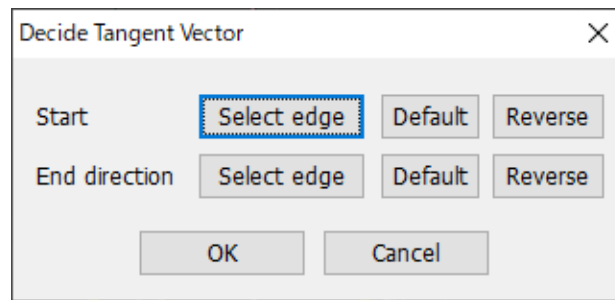
7. Select the face on the right.



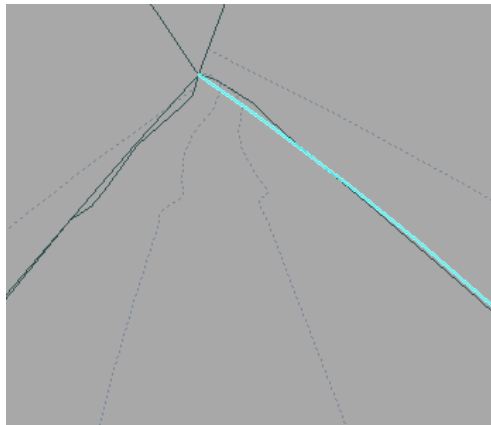
8. Pick two places on the curve as shown below, and press [Done] (✔).




9. "Decide Tangent Vector" dialog will appear. Accept the default setting and click [OK].

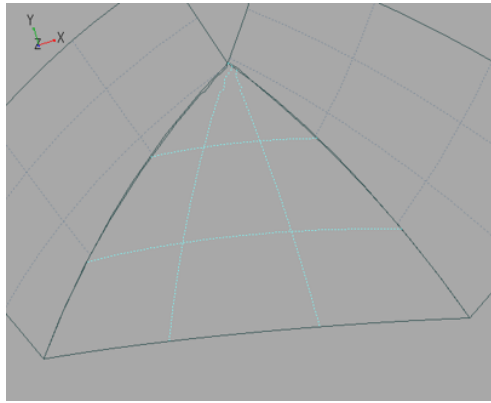


A new curve is created. (The side highlighted in light blue)

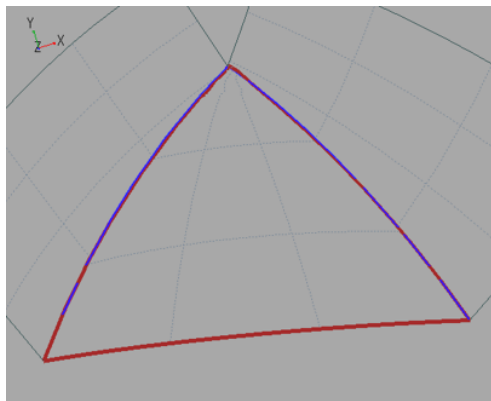


Next, the two newly created curves will be used to replace the boundaries of adjacent faces.

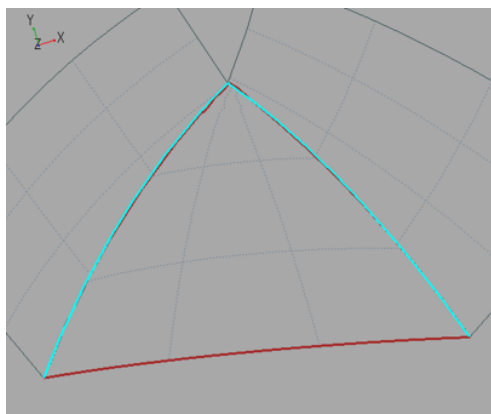
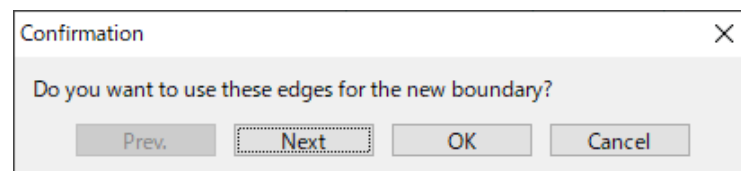
10. Select [Modify] > [Repair Solid] > [Change Face Boundary] from the menu or click [Change Face Boundary] () on the toolbar.
11. Select the center face on "3D View" window.



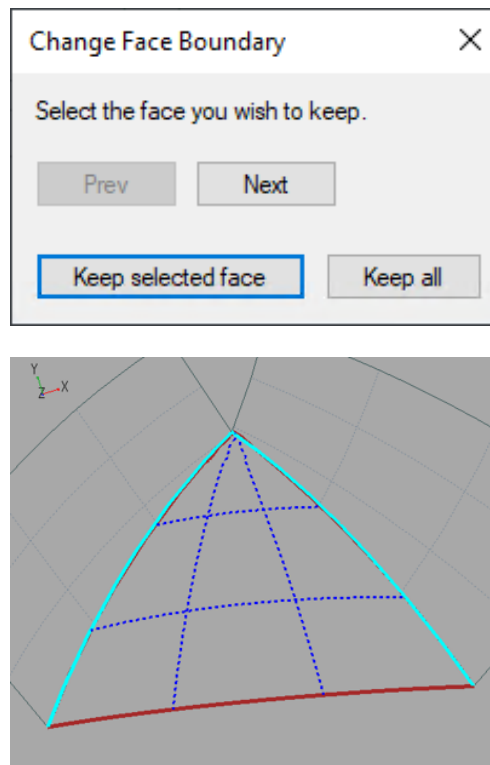
12. Pick the two newly created curves, and press [Done] (✔).



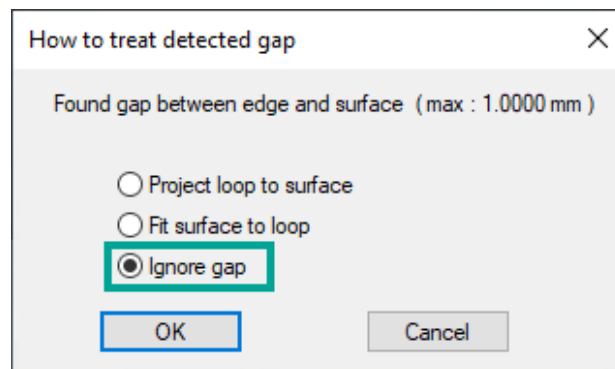
13. A confirmation dialog will appear. Confirm that the edges highlighted on "3D View" window are correct, and click [OK].



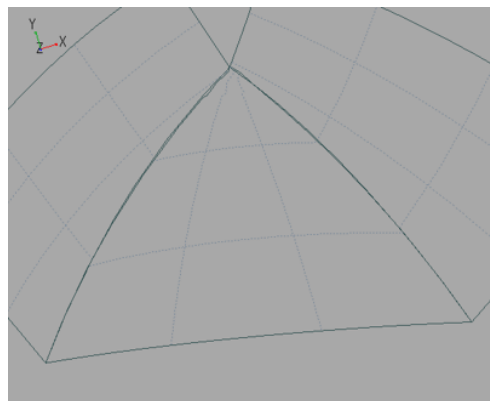
14. "Change Face Boundary" dialog will appear. Click either [Next] or [Previous] to switch to the state where the face you wish to keep is highlighted in blue. Click [Keep selected face].




15. "How to treat detected gap" dialog will appear. Here, select "Ignore gap" and click [OK].

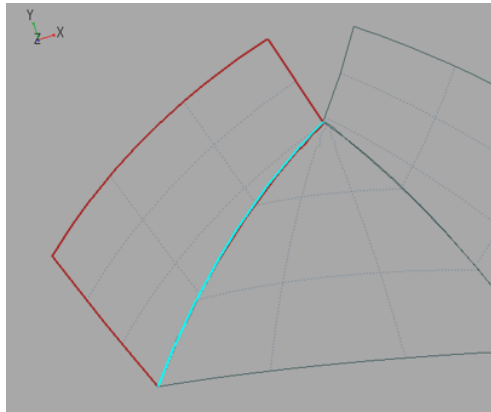


Boundary of the center face has been replaced by the two curves you created.
With the same procedure, replace the two newly created curves as the boundaries of the adjacent left and right faces.

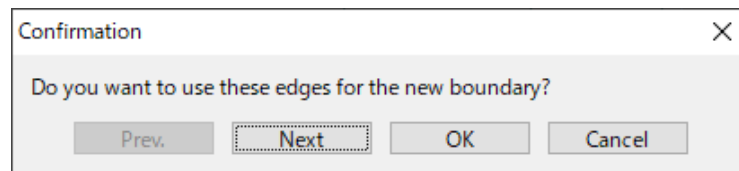


16. Select [Modify] > [Repair Solid] > [Change Face Boundary] from the menu or click [Change Face Boundary] () on the toolbar.

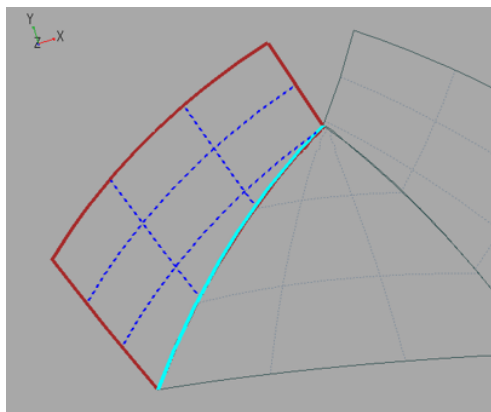
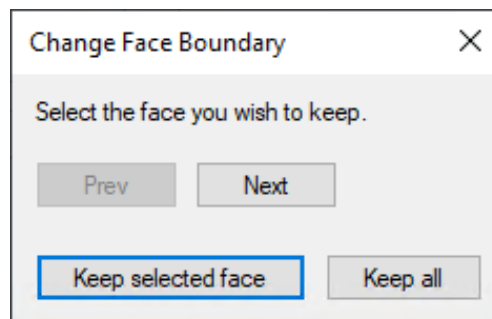
17. Select the left face on "3D View" window.
18. Pick the edge which will be the boundary of the new face, and press [Done] (✓).



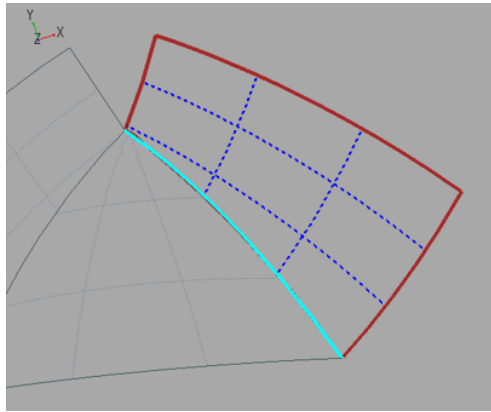
19. A confirmation dialog will appear. Confirm the edges highlighted on "3D View" window are correct, and click [OK].



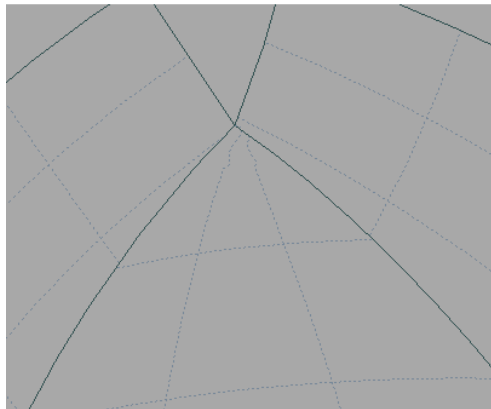
20. "Change Face Boundary" dialog will appear. Click either [Next] or [Previous] to switch to the state where the face you wish to keep is highlighted in blue. Click [Keep selected face].



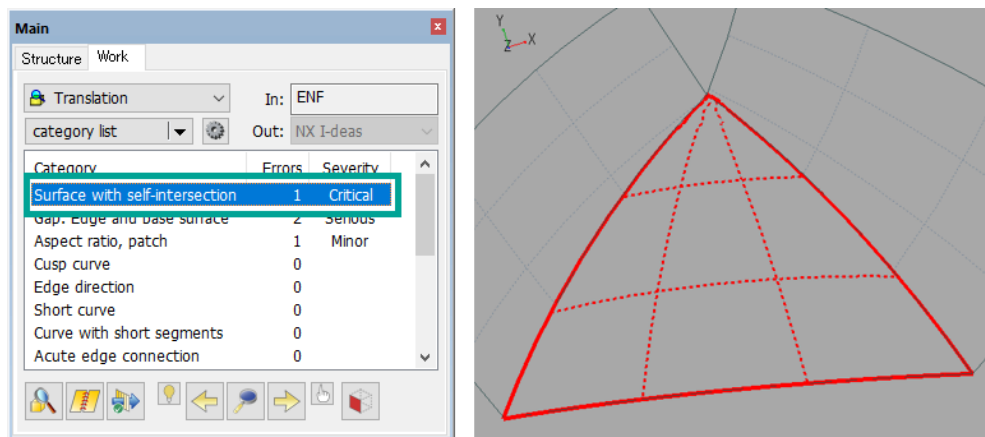
Boundary of the face will be changed. With the same procedure, replace the boundary of the right face.




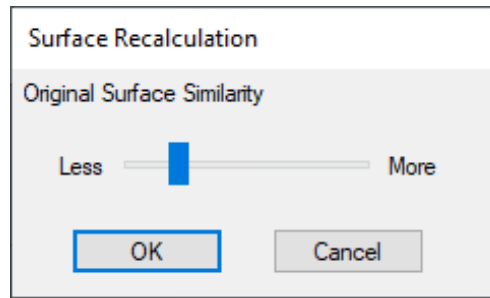
Two newly created curves have replaced the boundaries of the adjacent faces.



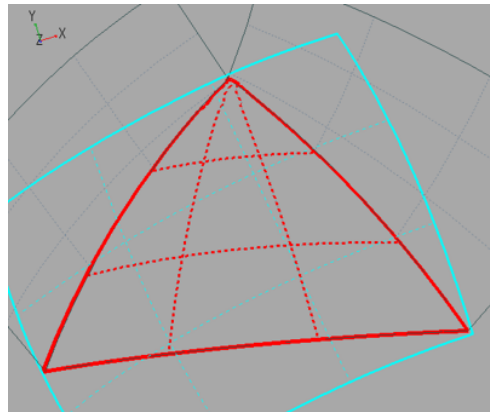
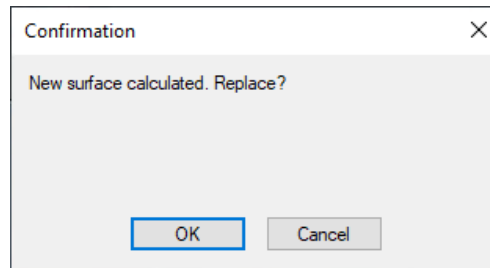
21. In [Main (Work)] panel, select "Surface with self-intersection" from the category list. Target position is highlighted in red on "3D View" window.



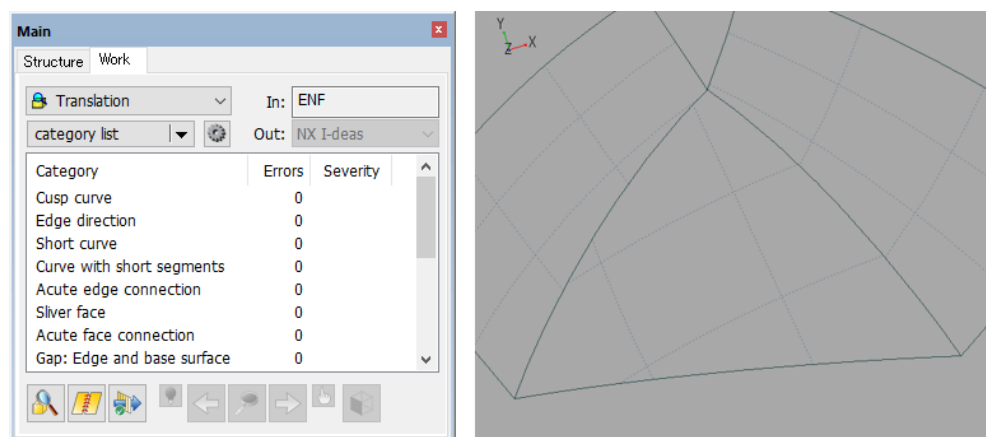
22. Press [Recalculate Surface] () on Navigation panel.
23. "Surface Recalculation" dialog will appear. Without changing "Original Surface Similarity", click [OK].



24. A confirmation dialog will appear. Click [OK] as is.




The face with "Surface with self-intersection" is replaced, and all errors have been modified.



4.2.2. Modify Curve with Oscillation (2)

1. Refer to 2.2.1, "Open Files" and open **InvalidGeometry2.drfx** from the <tutorial> folder.

Although this model does not have any detected errors in the category list, when visually observing the geometry on "3D View" window, you can see that some of the edges shared by

the left and center faces are oscillating. Also, the adjacent faces are oscillating so even if you create new curves with the function [On Surface] (), distorted curves will be created.

In such a case, create curves that replace the oscillating curves on both sides of the surface.

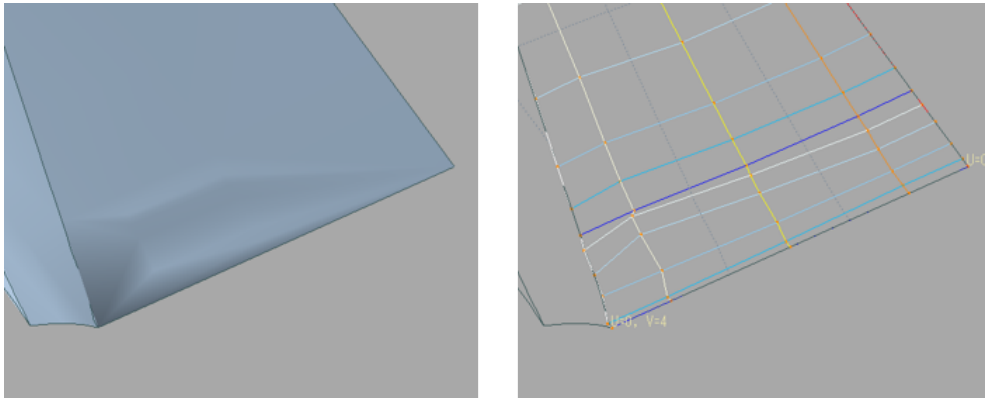

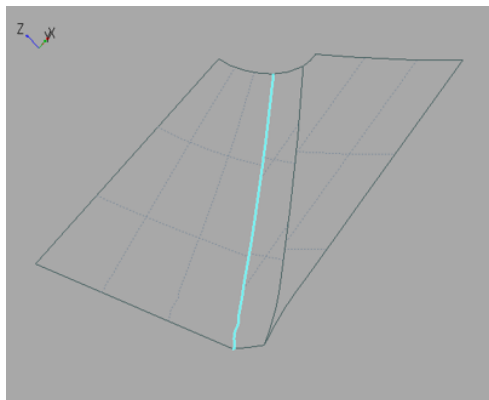
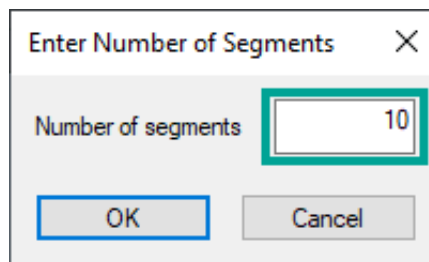


Figure 5. Curve oscillation (Shading and Control Points)

2. Select [Create] > [Point] > [Division] from the menu or click [Division] () on the toolbar.
3. Pick a curve (cyan color) with oscillation on "3D View" window.

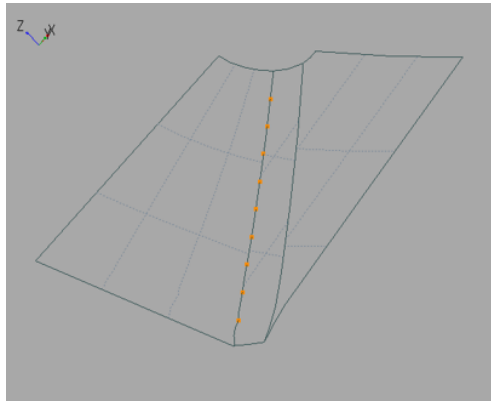




4. "Enter Number of Segments" dialog will appear. Enter "10" and click [OK].

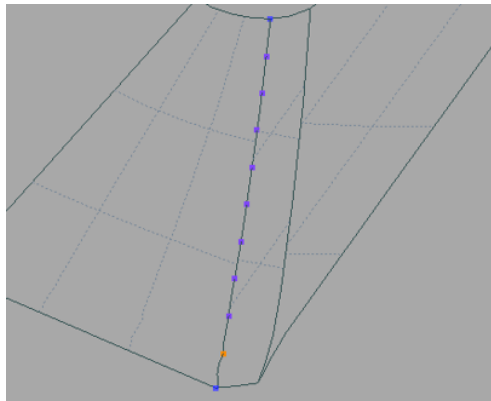


Division points are created on the specified curve.

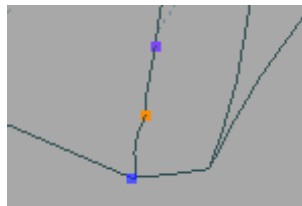
In this case, there are nine points created because "10" was specified as the number of segments.



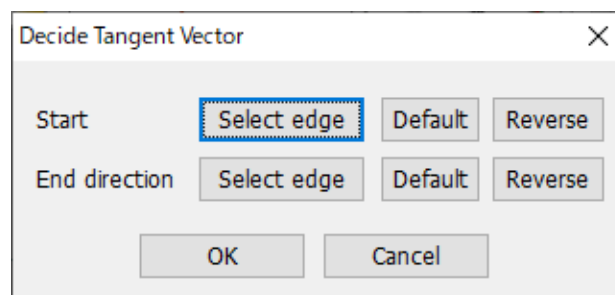
5. Select [Create] > [Curve] > [Interpolation] from the menu or click [Interpolation] () on the toolbar.
6. On "3D View" window, pick in the order of "Start" point > created points > "End direction" point, then press [Done] ().

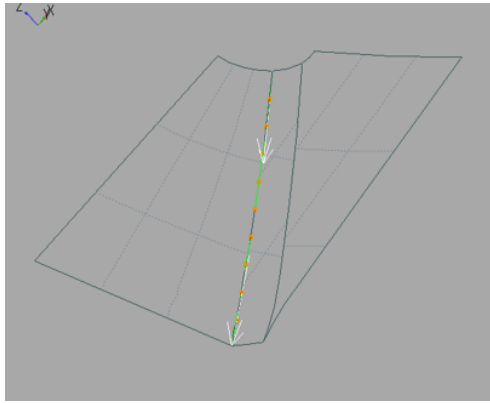


Please ensure to exclude the point where the curve is oscillating.

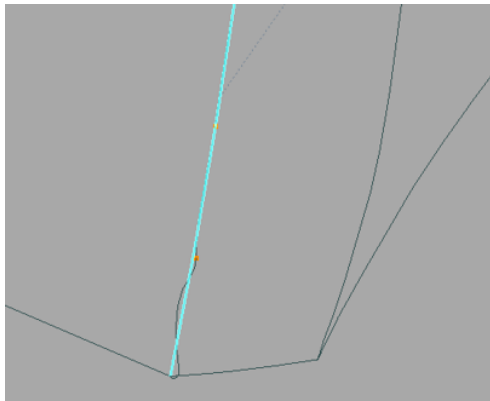


7. "Decide Tangent Vector" dialog will appear. Accept the default setting and click [OK].




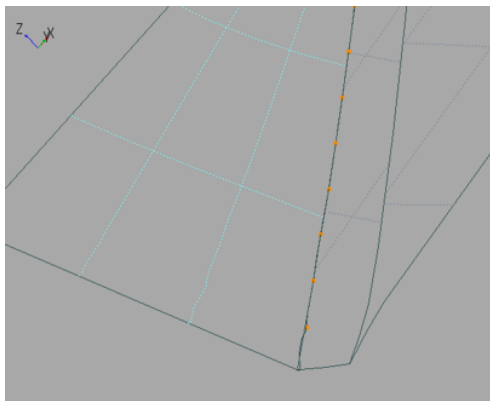


An interpolation curve is created. (Curve on the side highlighted in light blue)

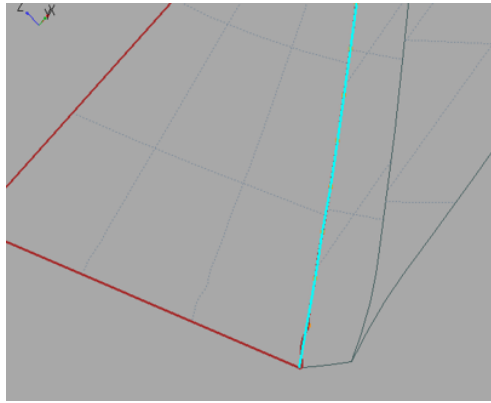


Next, replace the edge shared by the left and center faces with the created interpolation curve.

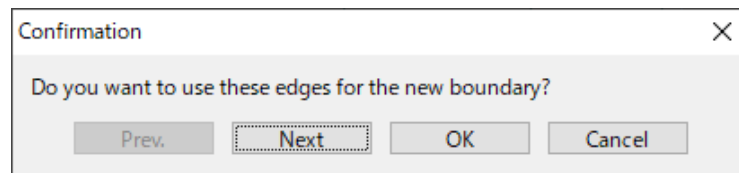
8. Select [Modify] > [Repair Solid] > [Change Face Boundary] from the menu or click [Change Face Boundary] () on the toolbar.
9. On "3D View" window, pick the left face.



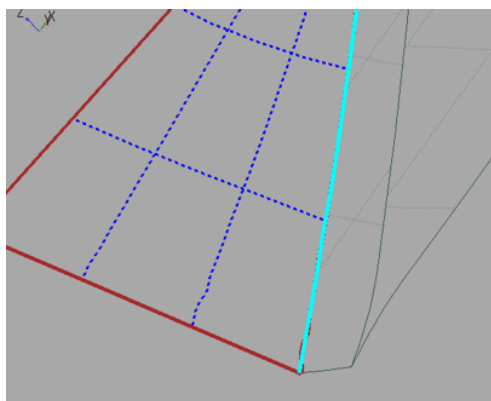
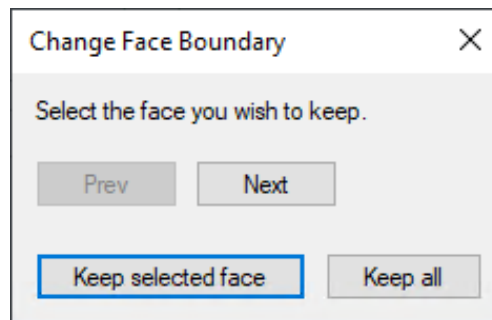
10. Pick the edge to use for new boundary, and press [Done] ()



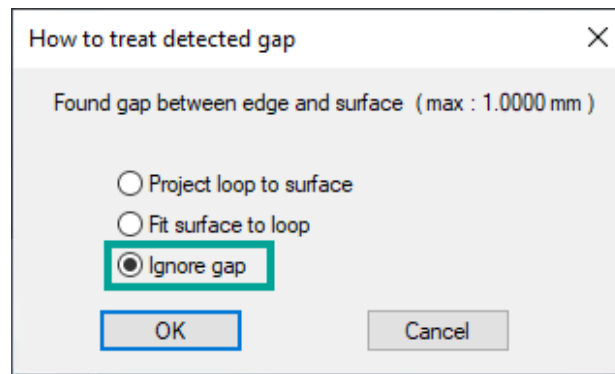
11. A confirmation dialog will appear. Confirm the curve highlighted on "3D View" window is correct, and click [OK].



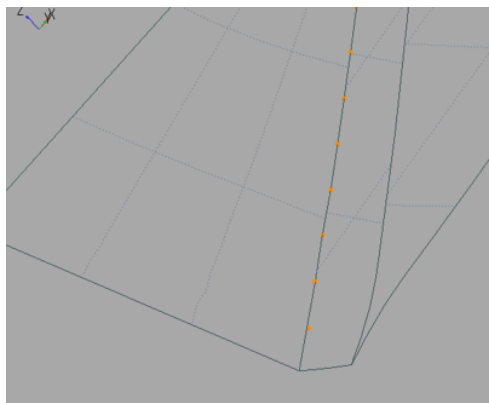
12. "Change Face Boundary" dialog will appear. Click either [Next] or [Previous] to switch to the state where the face you wish to keep is highlighted in blue. Click [Keep selected face].



13. "How to treat detected gap" dialog will appear. Here, select "Ignore gap" and click [OK].

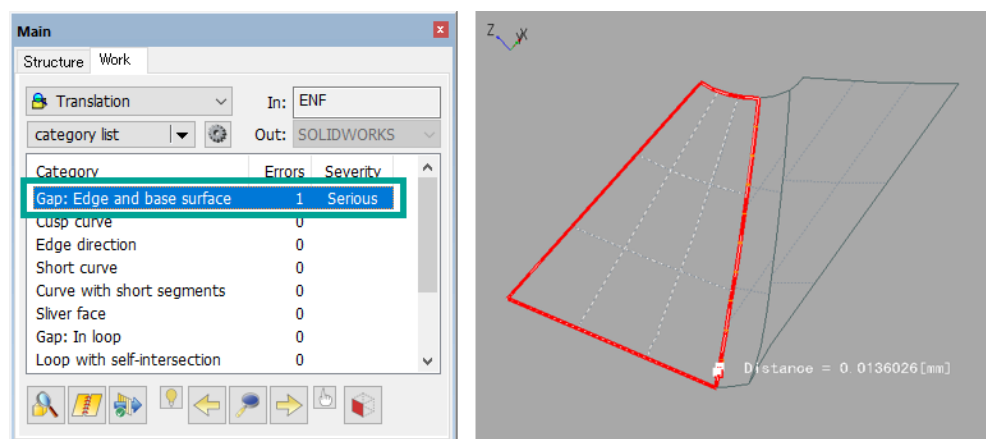



Boundary of the left face has been replaced by the interpolation curve you have created. With the same procedure, replace the created interpolation curve as the boundary of the center face.



Next, modify the error "Gap: Edge and base surface" which appeared when the face boundary was replaced.

14. In [Main (Work)] panel, select "Gap: Edge and base surface" from the category list. Target position is highlighted in red on "3D View" window.

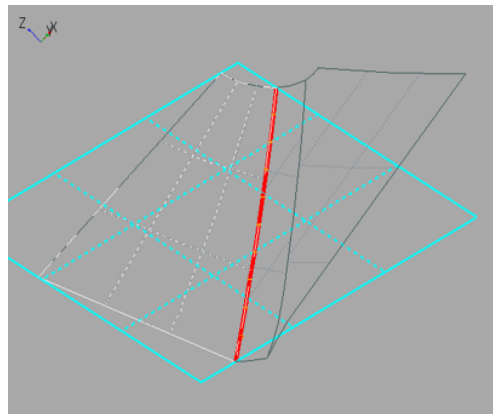
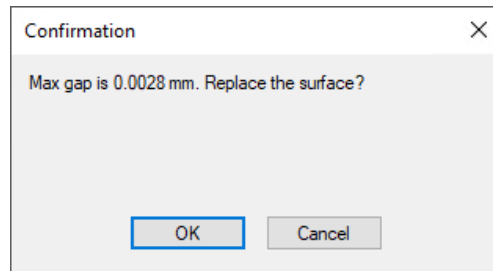


15. Press [Boundaries, Surface → Surface] () on Navigation panel.

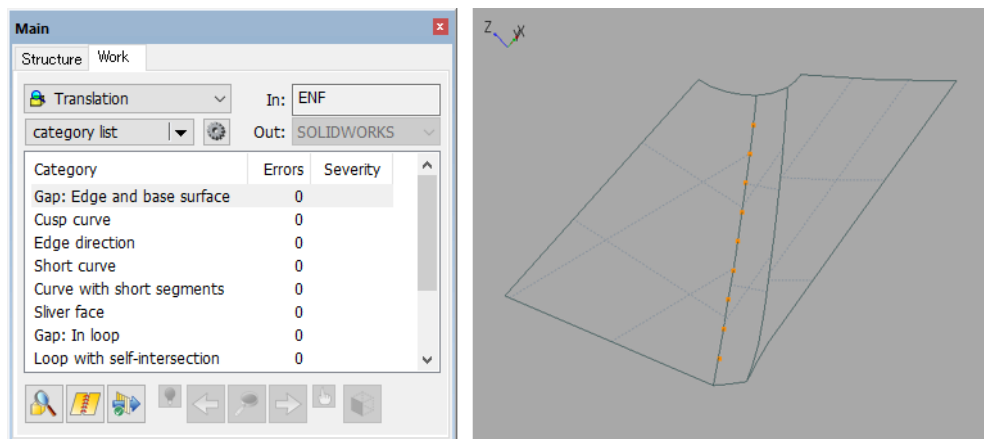


Please note that due to the poor quality of the original control points, the function [Boundaries, Surface → Surface] is more suitable than [Fit Face to Loops].

16. A confirmation dialog will appear. Click [OK].



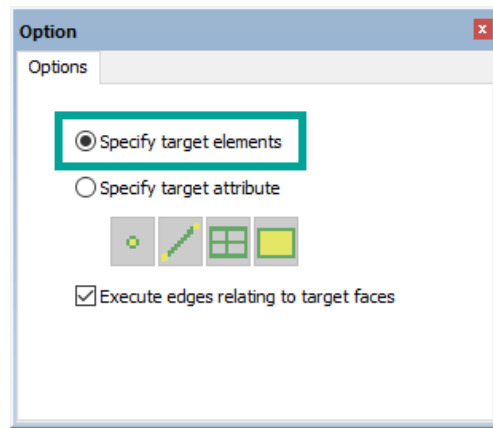
Surface is replaced, and the category list is updated.



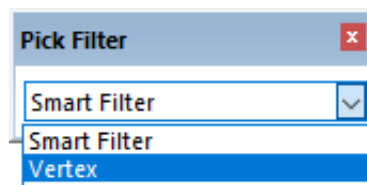
Finally, delete the unnecessary division points.

17. Select [Edit] > [Delete] from the menu or click [Delete] (X) on the toolbar.

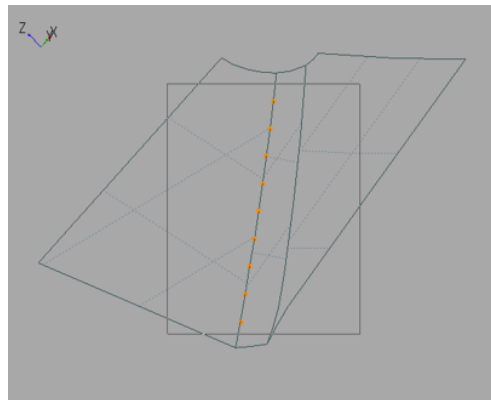
18. In [Options] panel, select "Specify target elements".



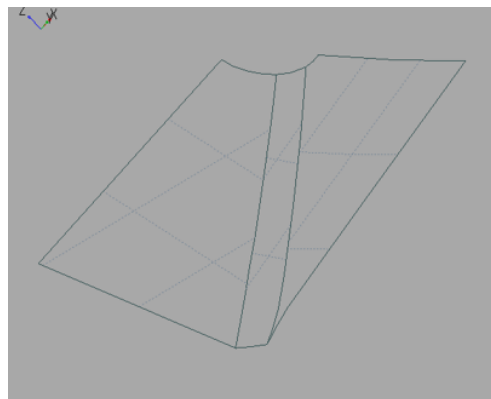
Switch the Pick Filter to "Vertex".



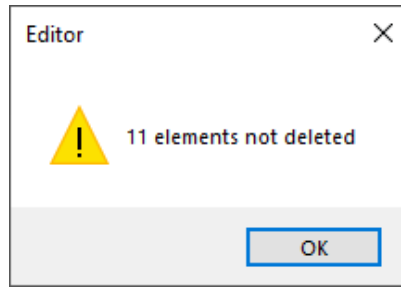
On "3D View" window, press down [Ctrl] key while left-clicking the mouse and drag around the points to be deleted. Press [Done] (✓).




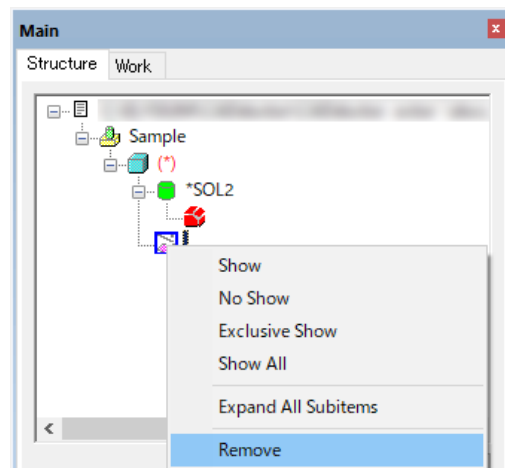
Points within the selected range have been deleted.



Please note that only single elements can be deleted. Following dialog will appear if the element includes non-single elements.



Elements other than solids can be deleted from the structure tree on [Main (Structure)] panel. For example, right-click on the single element () you wish to delete on [Main (Structure)] panel, and click [Remove] from the context menu.



This is the end of modifying curves with oscillation.

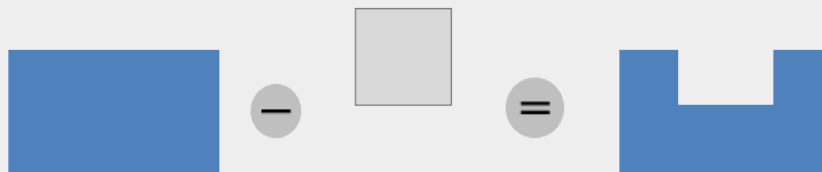
4.3. Modifying Invalid Topological Structure

Unintended topological structure model may be generated as a result when inappropriate Boolean operation is executed in the source CAD system. This section will explain how to modify the errors.

Example of Boolean Operation Generating Unintended Data

■ Logical Negation (NOT)

If there is an area where Solid A (blue) and Solid B (gray) overlap, a Boolean operation "logical negation (NOT)" produces the result as shown below.

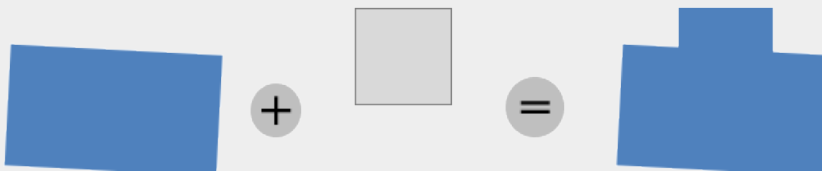


However, if Solid A (blue) and Solid B (gray) do not completely overlap, such as when they touch each other at the top, tiny areas like the one shown below may remain due to calculation errors.

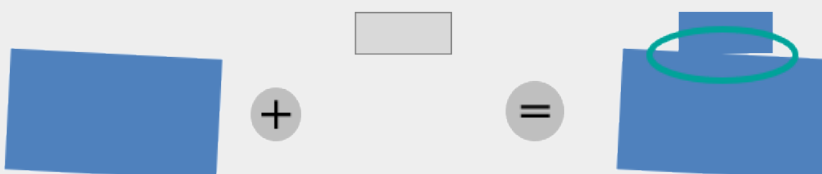


■ Logical Disjunction (OR)

If there is an area where Solid A (blue) and Solid B (gray) overlap, a Boolean operation "logical disjunction (OR)" produces the result as shown below.



However, if the size and position of Solid B (gray) are improper, Solid B may merge incompletely, resulting in a tiny gap as shown in the figure below.

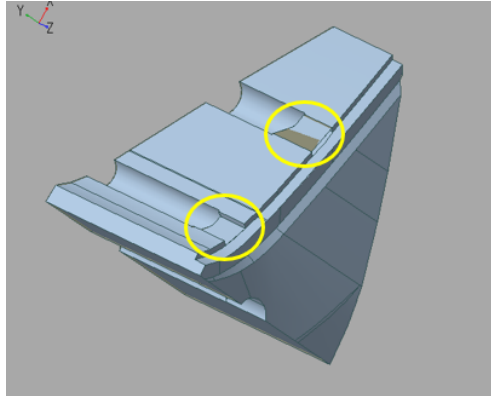


To avoid such a result, when using Boolean operation, the solids should have an overlapping area. Afterward, it is recommended that you visually observe the result.

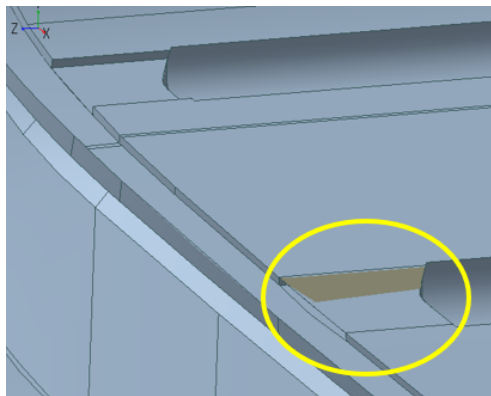
The following explains how to modify the geometry caused by the Boolean operation.

4.3.1. Modifying Invalid Geometry Caused by Negation


1. Refer to 2.2.1, “Open Files” and open **boolean1.drfx** from the <tutorial> folder.

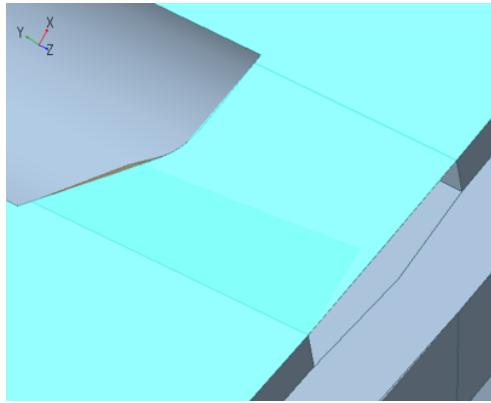


Zoom in to the area surrounded by the yellow circles to see where the Boolean operation was intended and where it failed. In the area surrounded by the yellow circle in the figure below, the result of the Boolean operation is not as intended and the unnecessary faces remain.

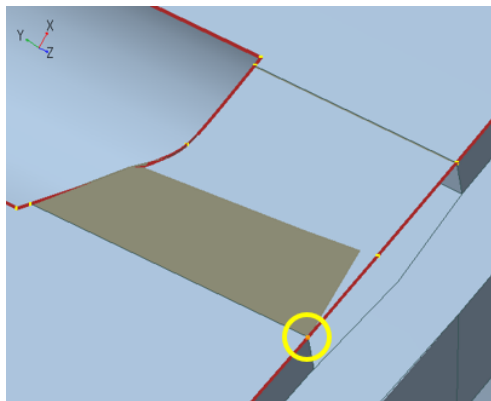


Remove the remaining faces at the point of failure.

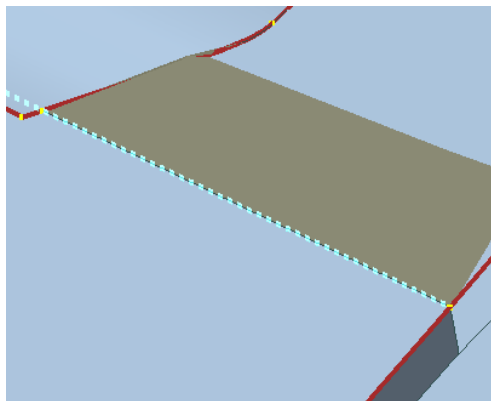
2. Select [Modify] > [Divide / Extend] > [Divide Face] from the menu or click [Divide Face] () on the toolbar.
3. Pick the target face on "3D View" window.



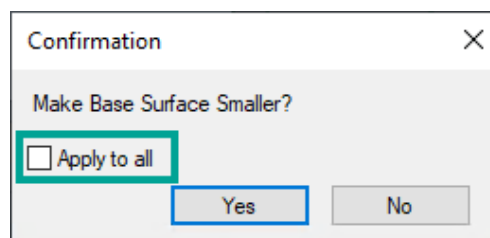
Left-click the position where you wish to divide on the face.

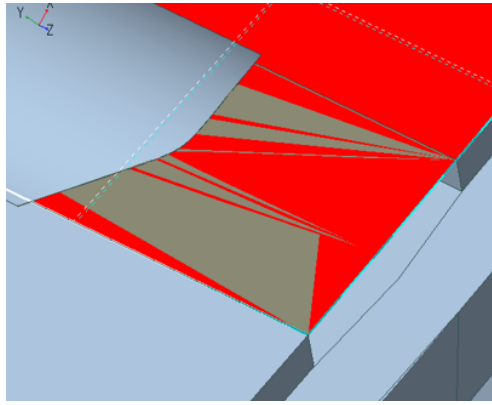


4. A guide to divide direction will appear. Click the guide at the position where you want to divide the face, and press [Done] (✓).

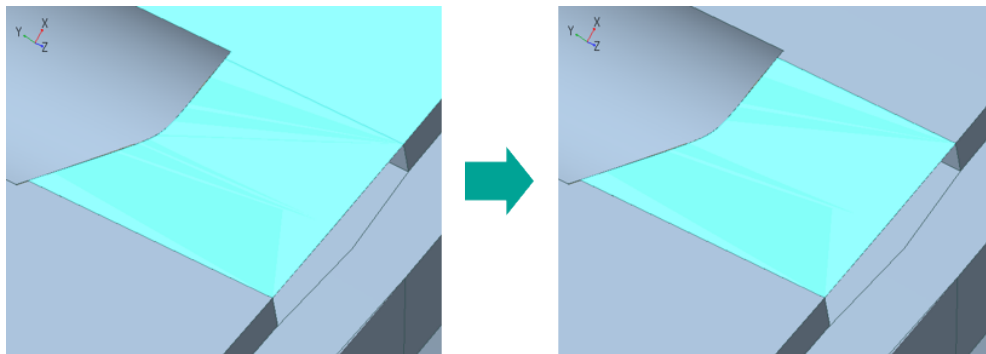


5. A confirmation dialog will appear. Enable "Apply to all" and click [Yes].



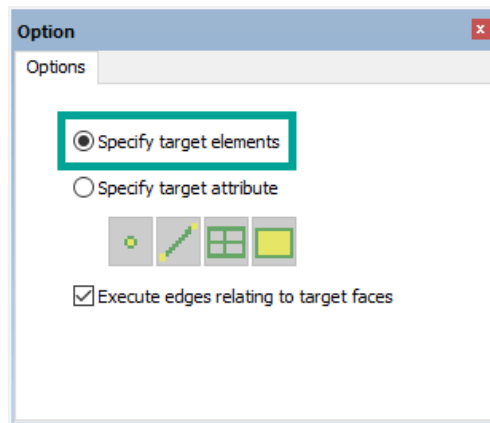


6. Repeat the same procedure to divide the other side of the face.

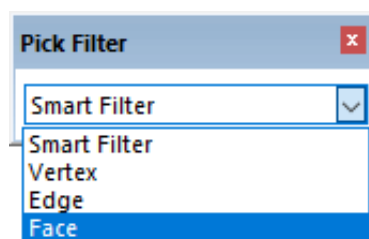


Next, delete the unnecessary face from the divided faces.

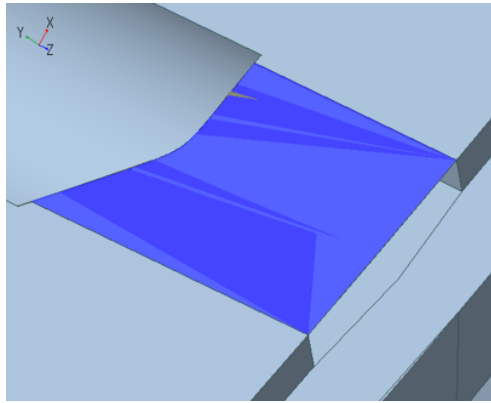
7. Select [Edit] > [Delete] from the menu or click [Delete] (✖) on the toolbar.
 8. In [Options] panel, select "Specify target elements".



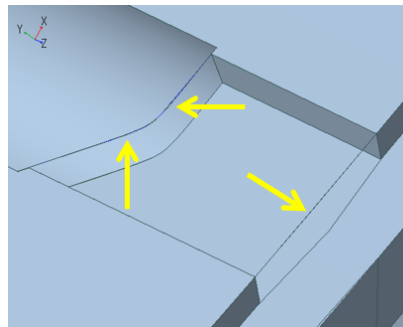
Switch the Pick Filter to "Face".



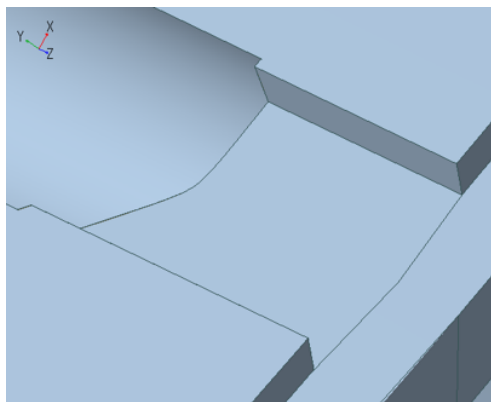
Pick the face to delete on "3D View" window, and press [Done] (✔).



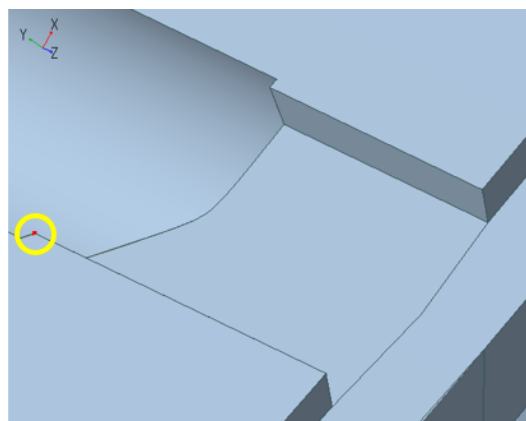
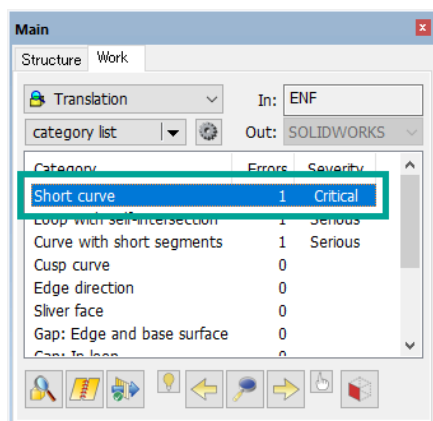
Please note that you need to delete the tiny faces hidden under the big face.



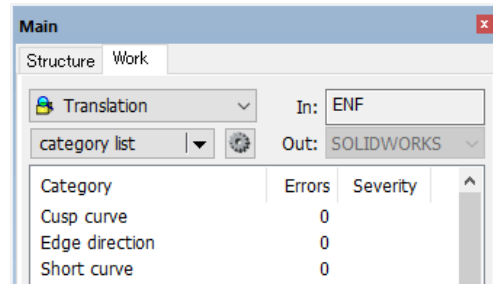
Faces are deleted. Press [Quit (Esc)] (✖) to exit the command.



- In [Main (Work)] panel, select "Short curve" from the category list. Target position is highlighted in red on "3D View" window.

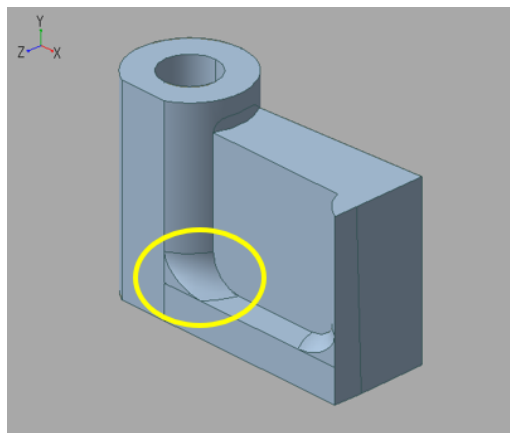


10. Press [Remove Short Edge] (🔧) on Navigation panel. All errors have been modified.

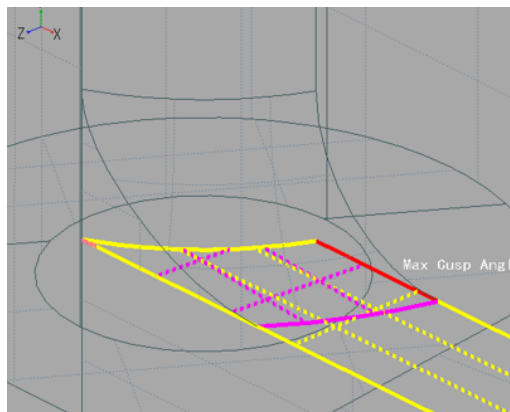


4.3.2. Modifying Invalid Geometry Caused by Logical Disjunction

Refer to 2.2.1, “Open Files” and open **boolean2.drfx** from the <tutorial> folder.



When zooming into the yellow circle, the result of the Boolean operation (logical disjunction) is not as intended and a tiny gap is created, so the face is extended into the gap.



Modify the position where the tiny gap was generated.

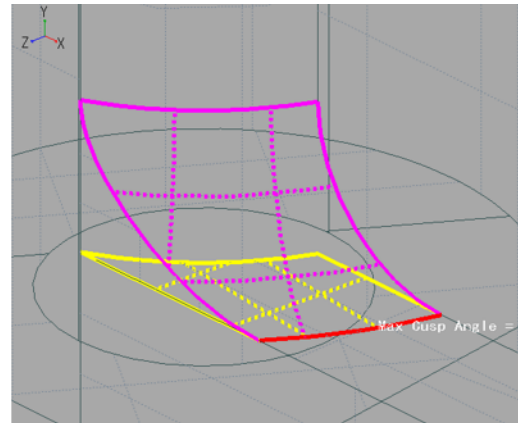
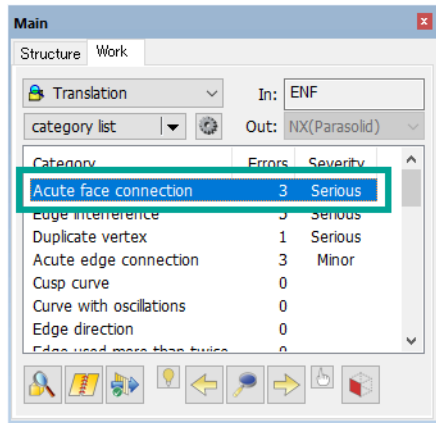
Detected error in this sample model can be modified by either [Remove Faces With Fix] (🔧) or [Change Face Boundary] (🔧).

■ When modifying with [Remove Faces With Fix] (🔧)

1. In [Main (Work)] panel, select "Acute face connection" from the category list.

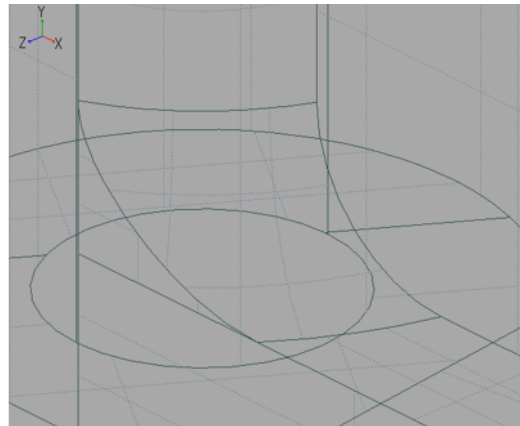
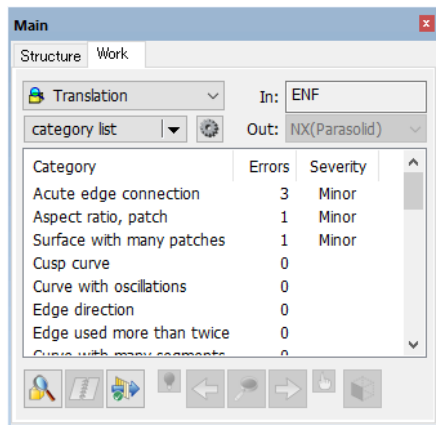
Press [Zoom current target] (🔍) to highlight the target position (below figure) on "3D View"

window.



2. Press [Remove Faces with Fix] (🔧) on Navigation panel.

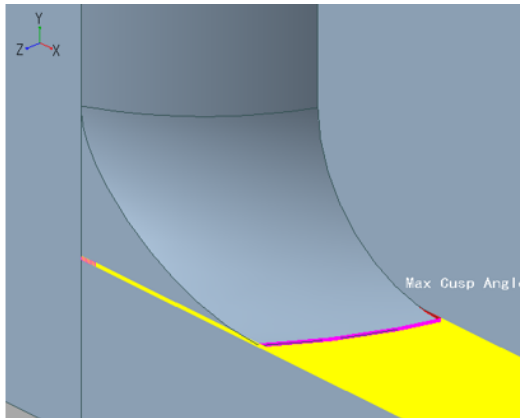
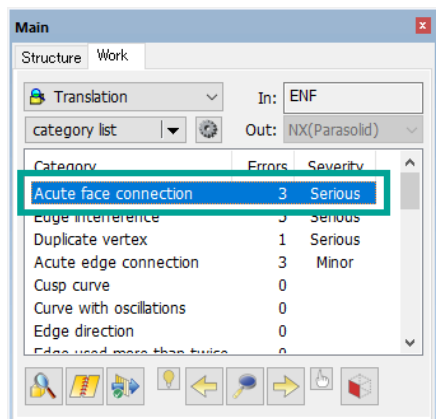
Tiny gaps have been filled in and unnecessary parts of extended faces have been removed.




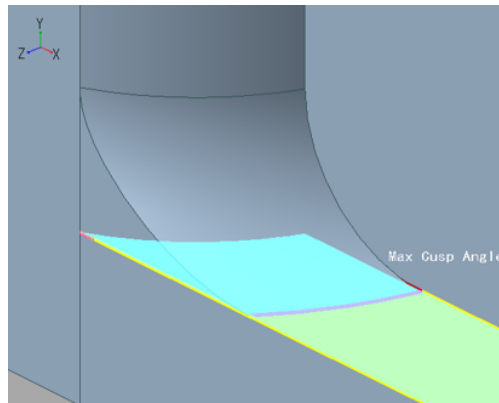
Please note that items marked as "Minor" in the category list are not errors and do not need to be modified.


■ When modifying with [Change Face Boundary] (🔧)

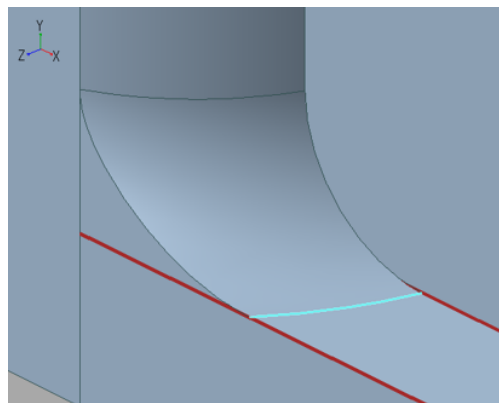
1. In [Main (Work)] panel, select "Acute face connection" from the category list.
Press [Zoom current target] (🔍) to highlight the target position on "3D View" window.



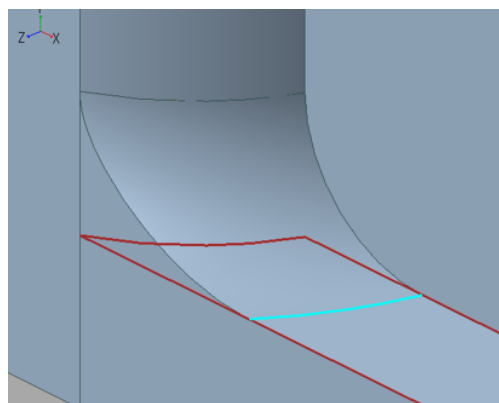
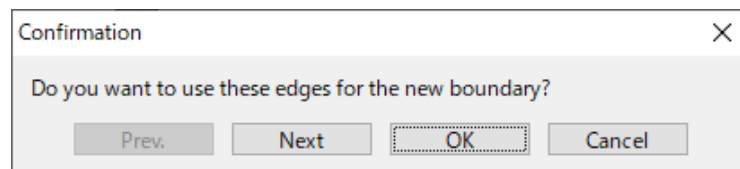
- Press [Change Face Boundary] () on Navigation panel.
Pick the face you wish to divide on "3D View" window.



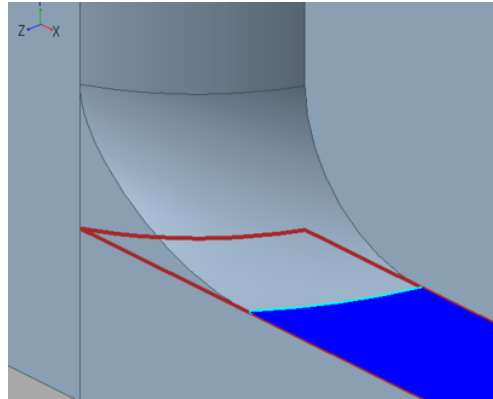
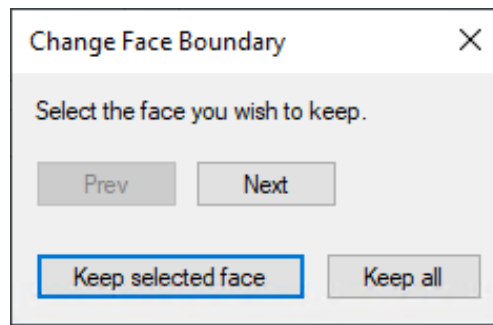
- On "3D View" window, pick the edge to use for the new boundary, and press [Done] ().



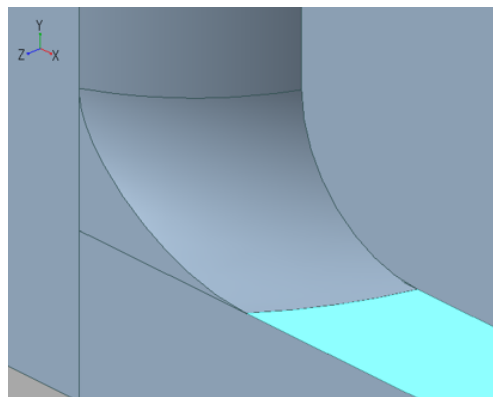
- A confirmation dialog will appear. Confirm that the edges highlighted on "3D View" window are correct, and click [OK].



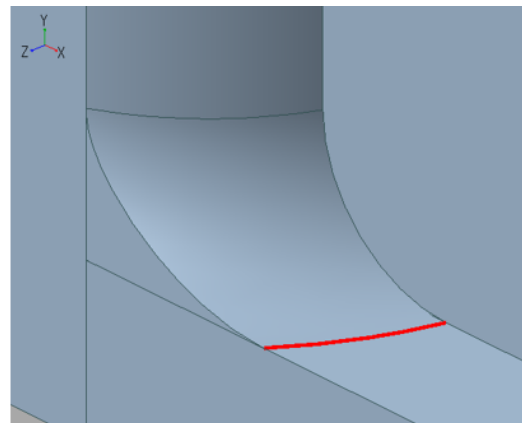
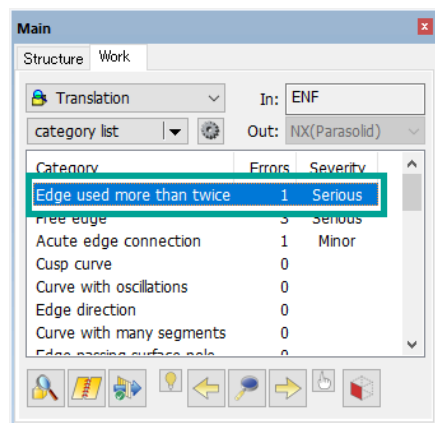
- "Change Face Boundary" dialog will appear. Click either [Next] or [Previous] to switch to the state where the face you wish to keep is highlighted in blue. Click [Keep selected face].



The boundary has been modified. Press [Quit (Esc)] (✖) to exit the command.

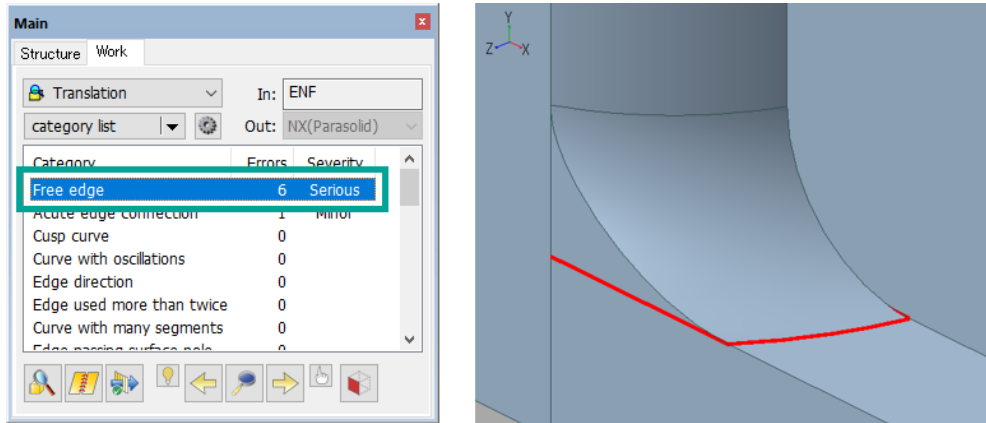


6. In [Main (Work)] panel, select "Edge used more than twice" from the category list. Target position is highlighted in red on "3D View" window.

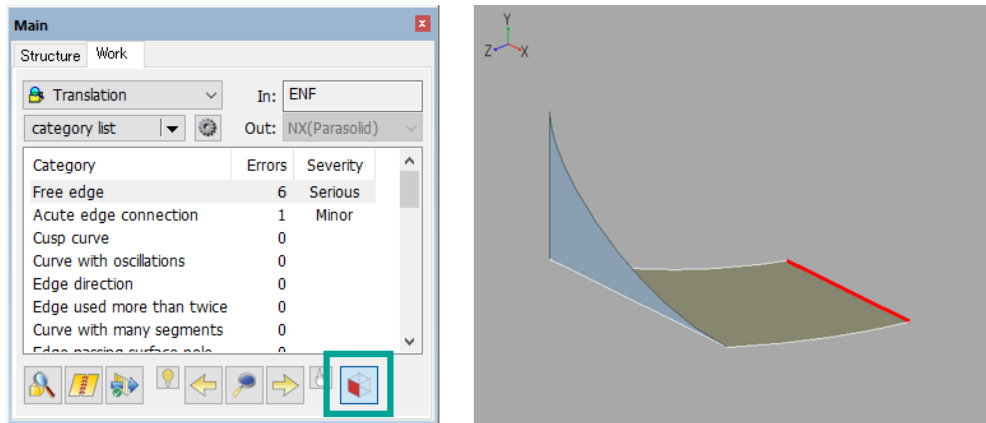


7. Press [Unstitch] (✂). Edges highlighted in red will be unstitched.

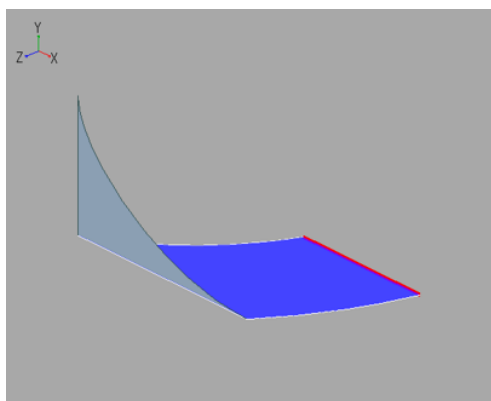
8. In [Main (Work)] panel, select "Free edge" from the category list.



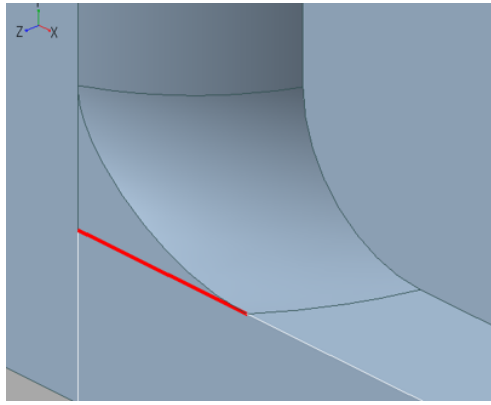
In [Main (Work)] panel, enable [Display Surroundings Only] (📦). To display the position of errors, use [Next] (➡) and [Previous] (⬅) to switch the currently displayed error position.



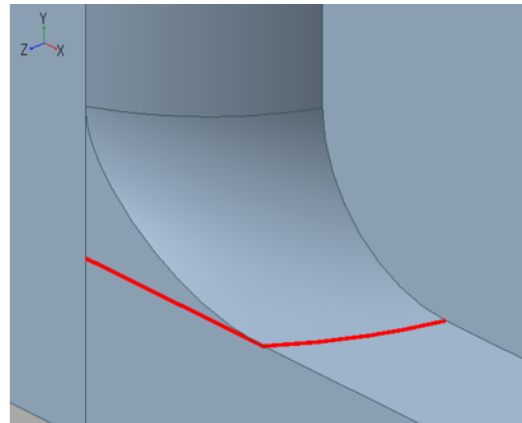
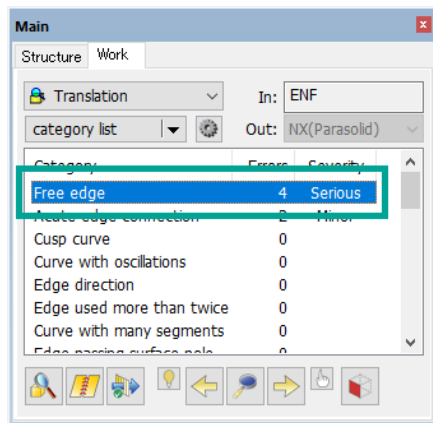
9. Press [Delete Elements] (✖). On "3D View" window, pick the unnecessary part of the face you just trimmed, and press [Done] (✅).




The face is deleted. In [Main (Work)] panel, disable [Display Surroundings Only] (📦).



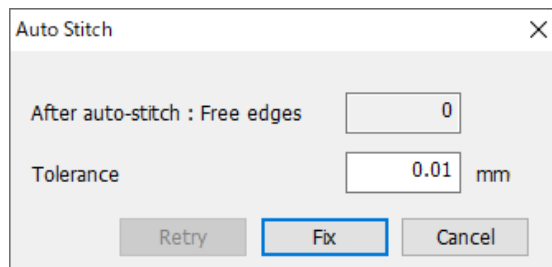
10. In [Main (Work)] panel, select "Free edge" from the category list.



11. In [Main] panel, press [Auto Stitch] ().
"Auto Stitch" dialog will appear. Specify the tolerance to "0.01mm" and click [Try].



Confirm that the number of free edges is "0" and click [Fix].



Connection information between faces has been modified, and all errors have been modified.

5. More Advanced

Most CAD data can be translated properly with the modification methods used up to the "3, *Intermediate*" level. In addition, even in cases where an element is missing and cannot be easily modified, you can modify it by combining several functions.

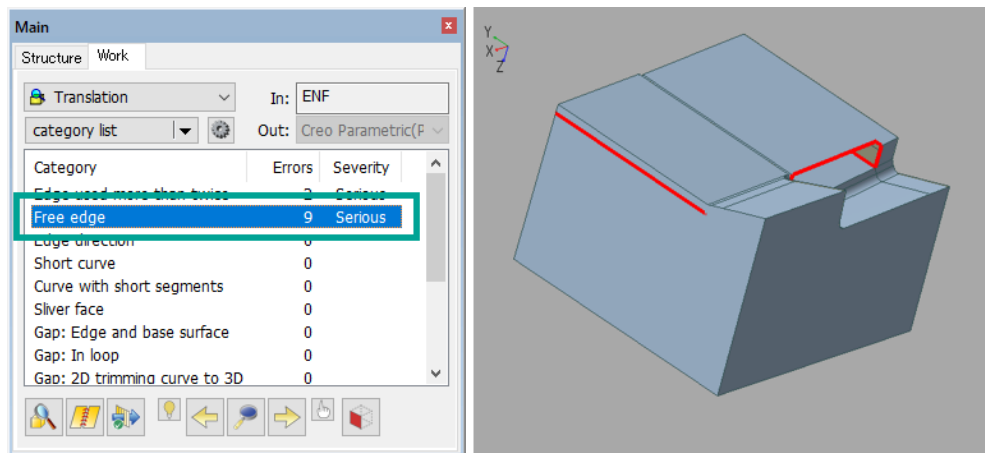
This chapter will explain how to modify errors that require higher technical operations than "4, *Advanced*" in the form of case studies.

5.1. Modifying Complex Errors

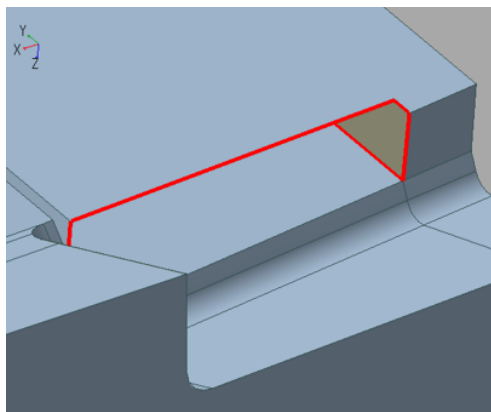
5.1.1. Case: Complex Face Missing

When importing a file such as IGES, there may be a face missing, and at the same time, there may be a trimming failure. This section will explain how to modify these errors to make them into solids.


1. Refer to 2.2.1, "Open Files" and open **FillHole2.drfx** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Free edge" from the category list. Target position is highlighted in red on "3D View" window.

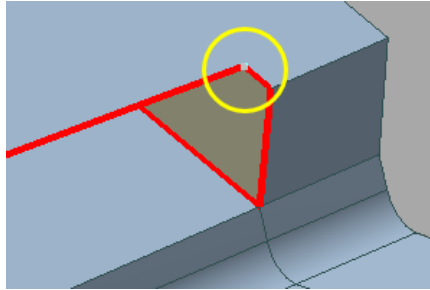


In this sample model, there is a missing face and an incorrect trimming in the step area. First of all, modify the area missing a face.

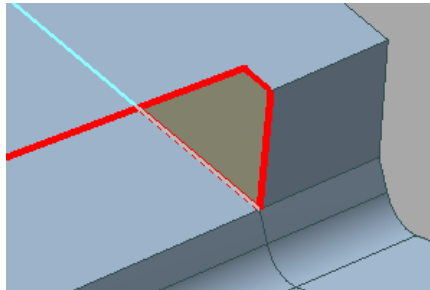


Begin by creating a curve (auxiliary line) necessary to create the face.

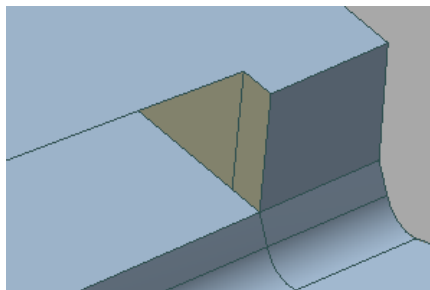
3. Select [Create] > [Curve] > [Perpendicular Line] from the menu or click [Perpendicular Line] () on the toolbar.
4. On "3D View" window, pick a point which will be the tip of the vertical line.




Pick the edge to where you want to extend down the vertical line.

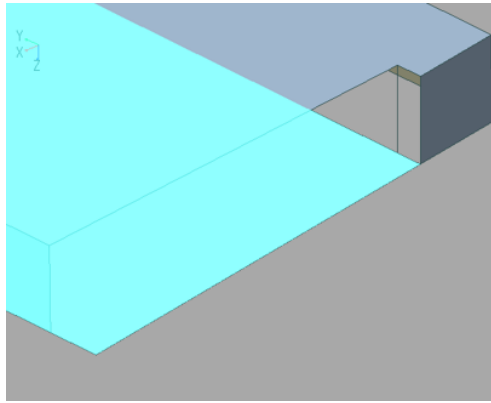


A vertical line is created and the edge to which the line is extended down is divided.

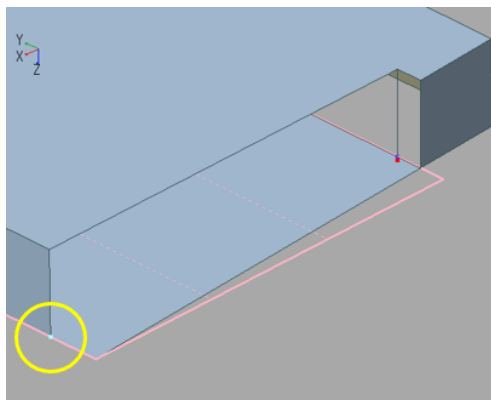
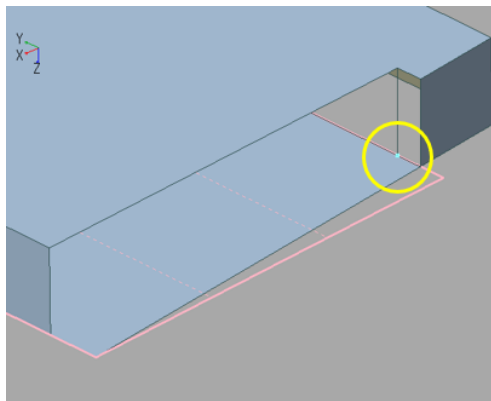


Next, create a curve to divide the face.

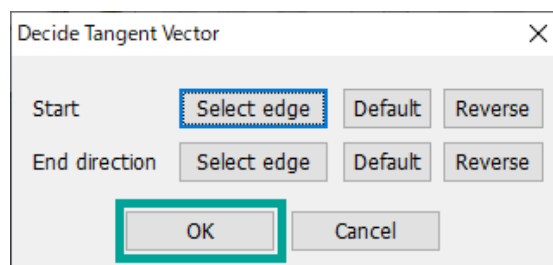
5. Select [Create] > [Curve] > [On Surface] from the menu or click [On Surface] () on the toolbar.
6. On "3D View" window, pick the face you want to divide. (*From here, some faces are hidden.

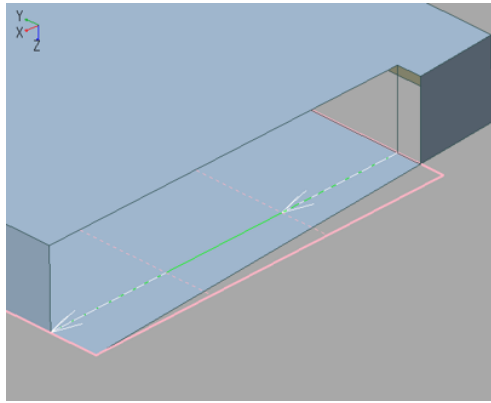


Pick two points that will be the end points of the line on surface, and press [Done] (✔).

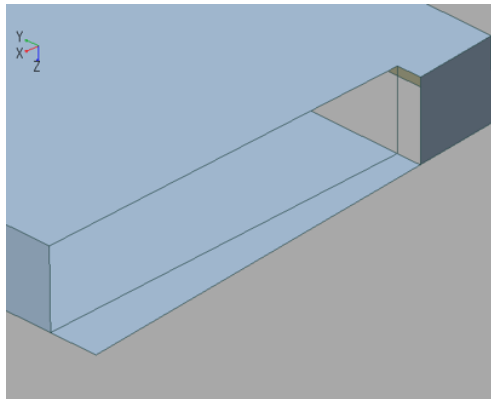


7. "Decide Tangent Vector" dialog will appear. Accept the default setting and click [OK].



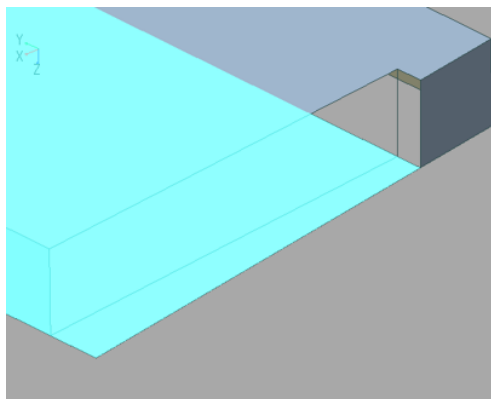


A new curve is created. Press [Quit (Esc)] (✖) to quit the command.

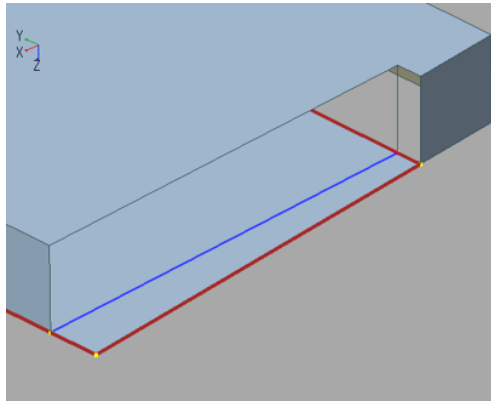


Next, divide the face with the new created curve.

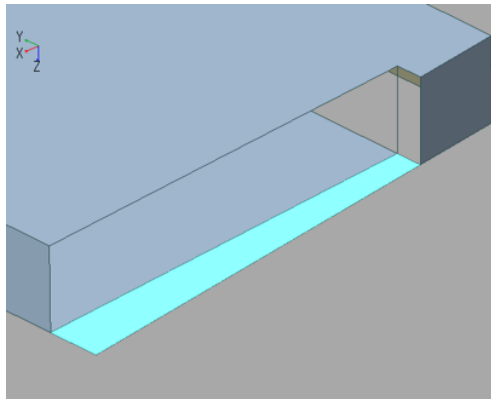
8. Select [Modify] > [Divide / Extend] > [Divide Face] from the menu or click [Divide Face] (📐) on the toolbar.
9. On "3D View" window, pick the face you wish to divide.



Next, pick the created line on surface, and press [Done] (✔).

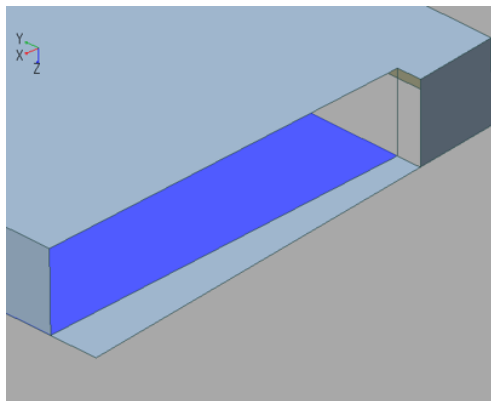


The face has been divided. Press [Quit (Esc)] (✖) to quit the command.

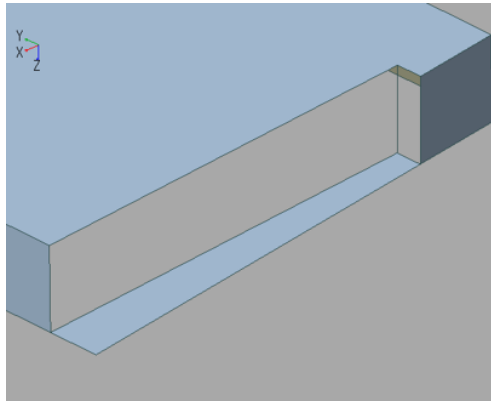


Next, delete the unnecessary face.


10. Select [Edit] > [Delete] from the menu or click [Delete] (✖) on the toolbar.
11. On "3D View" window, pick the face to delete, and press [Done] (✔).

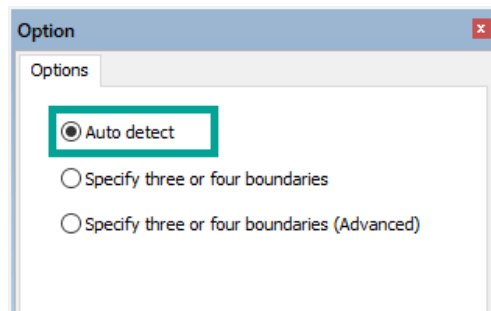


The face has been deleted. Press [Quit (Esc)] (✖) to quit the command.

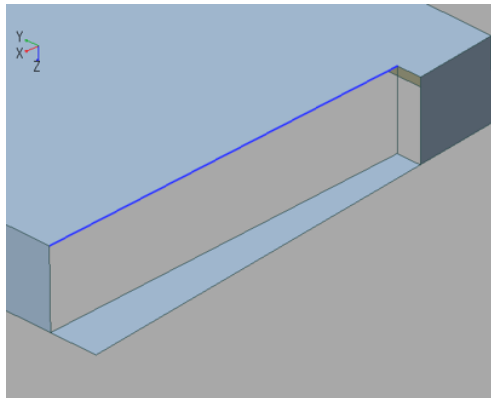


Finally, create a face where the face is missing.

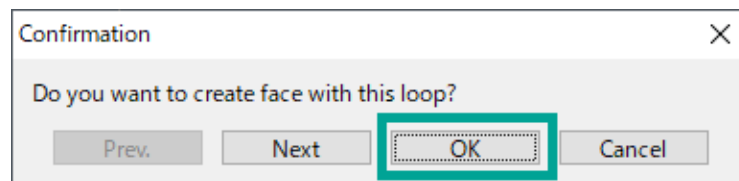
12. Select [Create] > [Face / Surface] > [Fill Hole (Create New Face)] from the menu or click [Fill Hole (Create New Face)] () on the toolbar.
13. "Option" dialog will appear. Select "Auto detect".

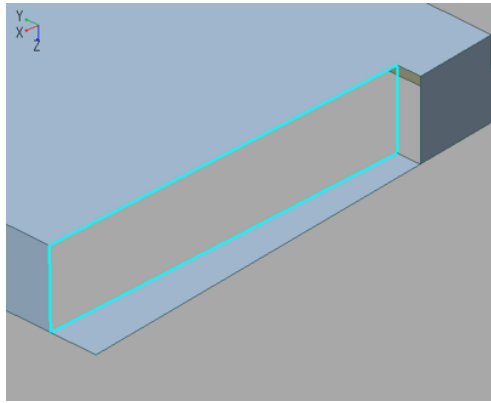


14. On "3D View" window, pick an edge where the face is missing, and press [Done] ().



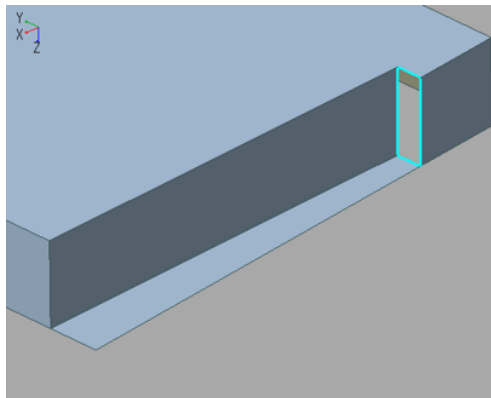
15. A confirmation dialog will appear. On "3D View" window, confirm that the edges you want to fill in are highlighted correctly, and click [OK].



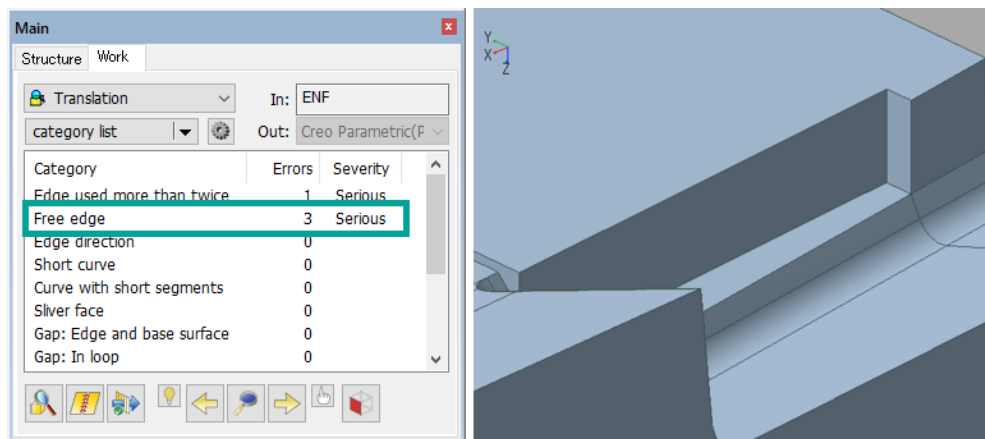


A new face is created in the target position.

Follow the same procedure to create a face for the other opening.



A new face is created in the target position, and the category list is updated.

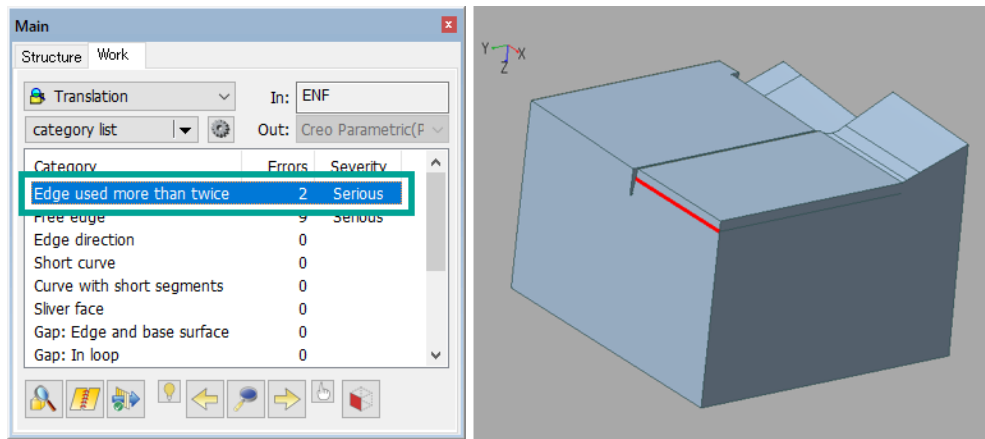


This concludes the modification of missing faces.

5.1.2. Case: Overly Shared Edges

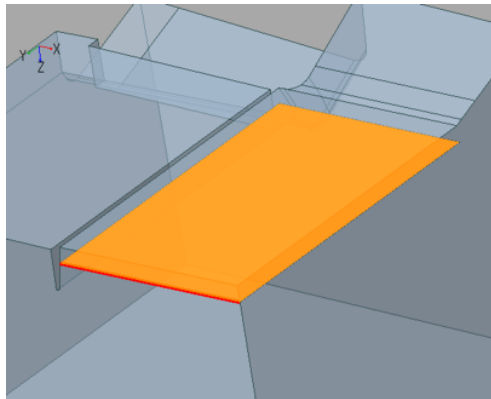
*Continue using the sample model **FillHole2.drxf** which you have just modified the missing face.

1. In [Main (Work)] panel, select "Edge used more than twice" from the category list. Target position will be highlighted in red on "3D View" window.

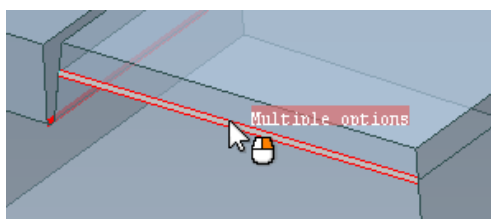


This edge is shared by three faces. One of these faces is inside the CAD model; therefore, the face inside the model is not needed.

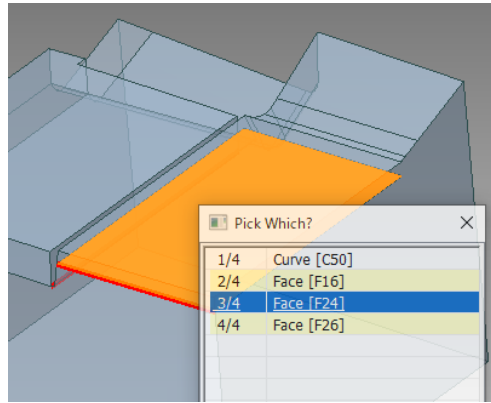
In this case, delete the unnecessary face existing inside the CAD model for modification.



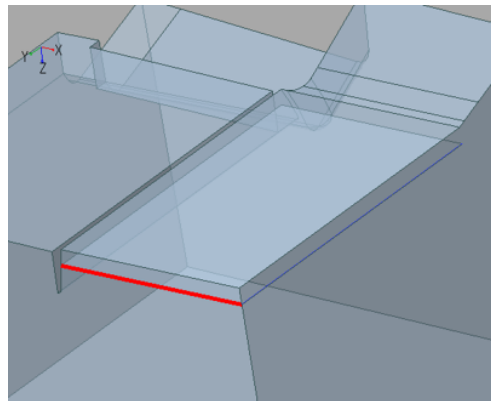
2. Press [Delete Elements] (✖) in Auxiliary Tool on Navigation panel.
3. If you move the mouse cursor over on the edge highlighted in red on "3D View" window, "Multiple options" will appear.



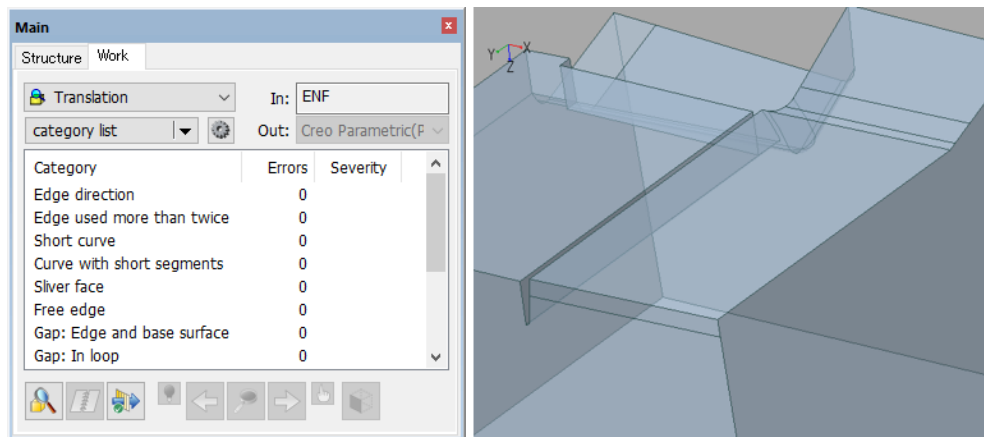
Right-click the mouse in that state to display "Pick Which?" dialog.



Specify the face existing inside the model in "Pick Which?" dialog, and press [Done] (✓).

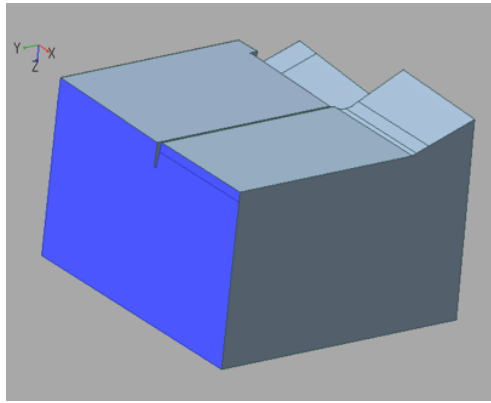


Specified face is deleted, and the category list is updated. Press [Quit (Esc)] (✗) to quit the command.

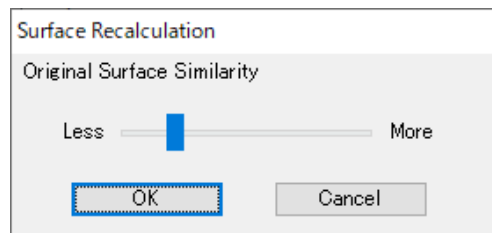


This concludes the modification of overly shared edges.
Finally, recreate the two divided faces into a single face.

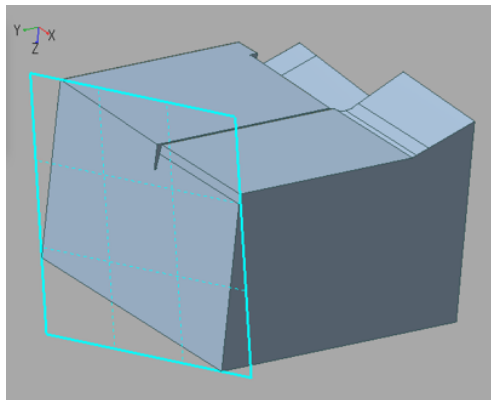
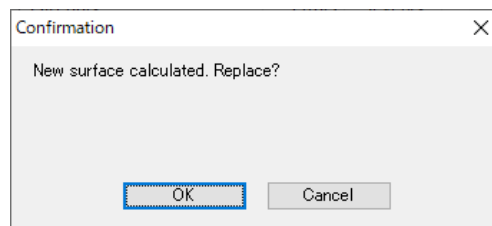
4. Select [Modify] > [Replace Surface] > [Recalculate Surface] from the menu or click [Recalculate Surface] (🔄) on the toolbar.
5. On "3D View" window, pick the two divided faces, and press [Done] (✓).



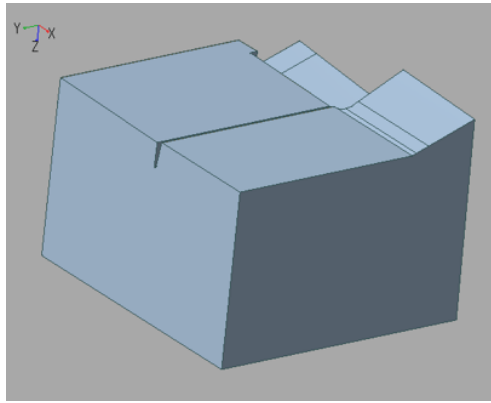
6. "Surface Recalculation" dialog will appear. Click [OK] without changing "Original Surface Similarity".



A confirmation dialog will appear. Click [OK].



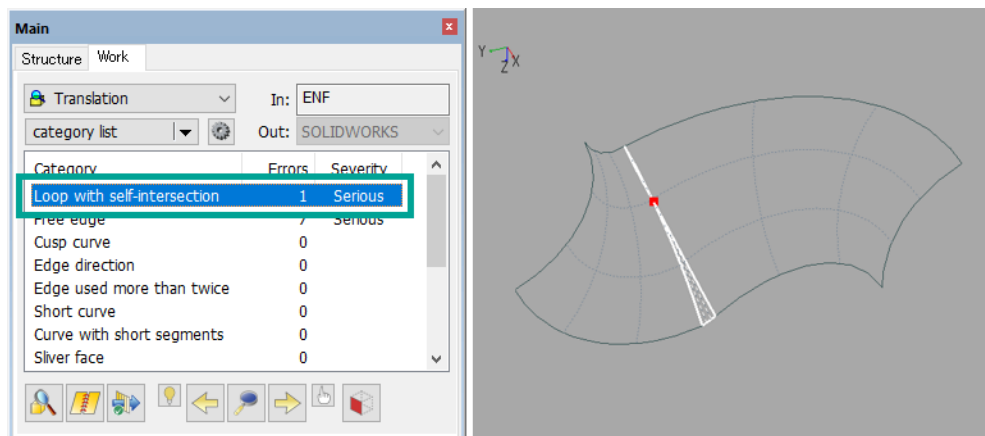
Surfaces are swapped, and two faces are replaced with one.



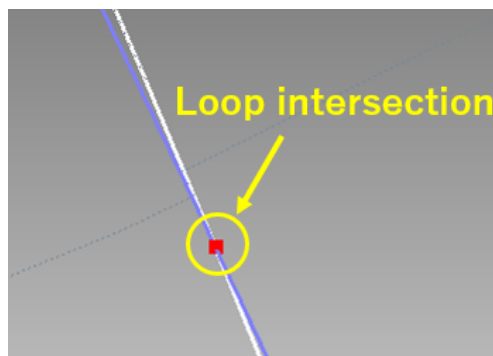
5.1.3. Case: Loop with Self-intersection


■ Modify with "Clean Self-intersection"

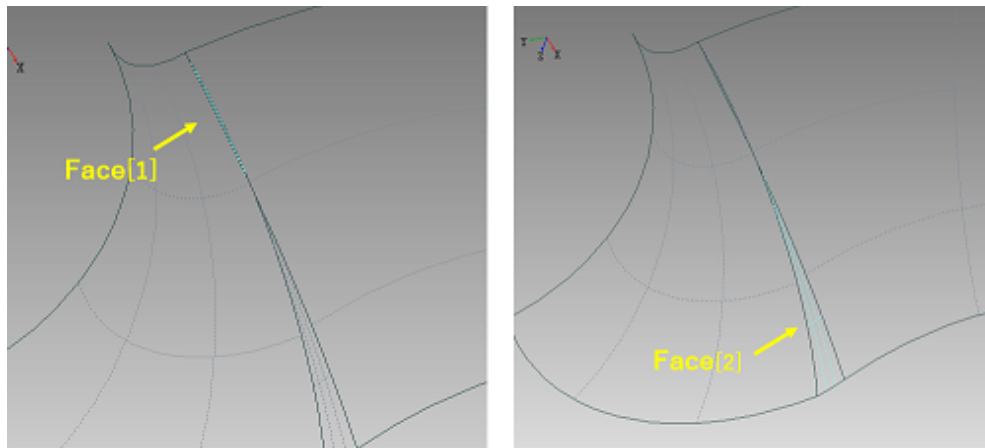
1. Refer to 2.2.1, “Open Files” and open **LoopIS.dr fx** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Loop with self-intersection" from the category list. Target position is highlighted in red on "3D View" window.



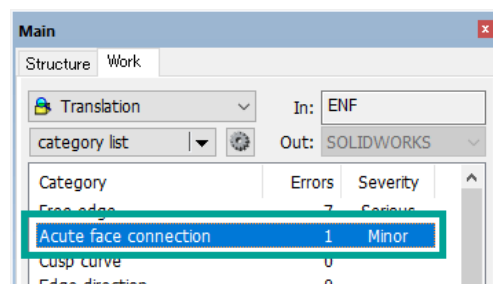
This model has an elongated face with an edge intersecting in the center, which has been detected as an error.



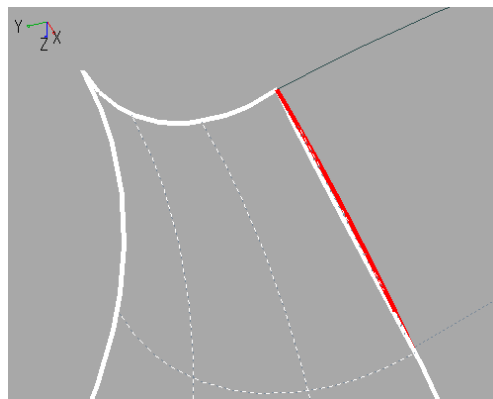
3. Press [Clean Self-intersection] () on Navigation panel.
In the position where the edges intersect, the face is divided into two faces, Face [1] and Face [2].



Detected error "Loop with self-intersection" has been modified, but another error "Acute Face Connection" has been detected.




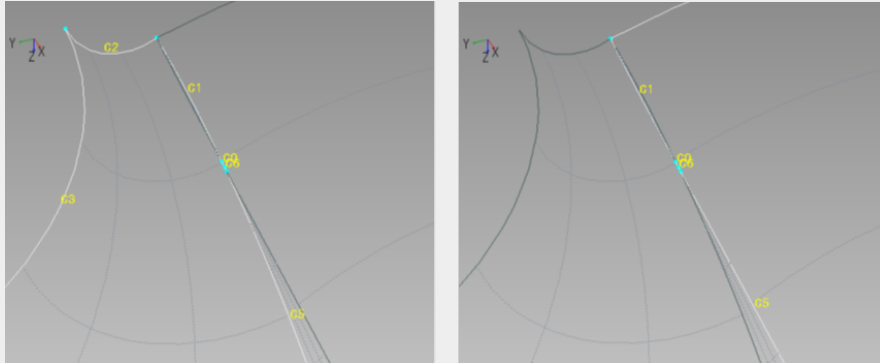
On "3D View" window, the corresponding position is highlighted in red. Confirm that the elongated face (Face [1]) and the large face on the left are connected at an acute angle.



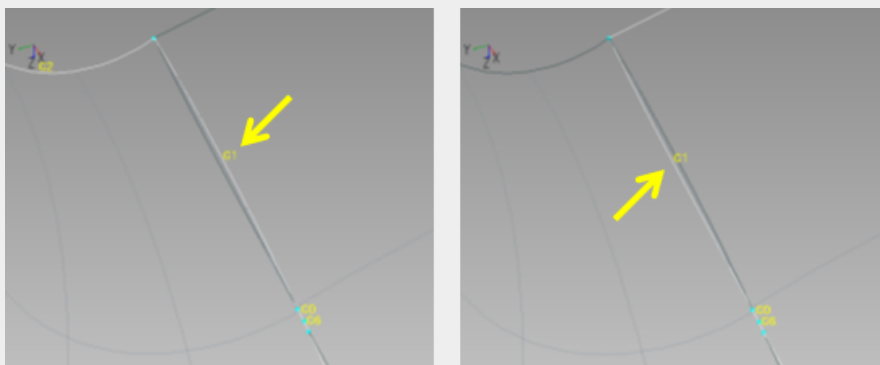
Delete the elongated face (Face [1]), and connect the large faces on both sides.

Confirm the positional relation of surrounding elements

- Select [Analysis] > [Related Element] from the menu or click [Related Element] () on the toolbar.
- By picking a face, the edges and vertices related to the picked face will be highlighted.




In the edge (C1) near the center of the figure below, you can see a large face each on the left and right which are trimmed so that they overlap each other.



When adding "Duplicate Faces" to Check items, you can detect where the faces overlap.

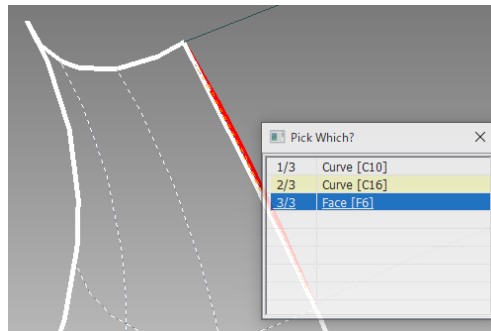


To add or delete an error in the category list, select [Check] > [Options] (in Data Translation Mode). In [Check items] tab, set the preferred setting.

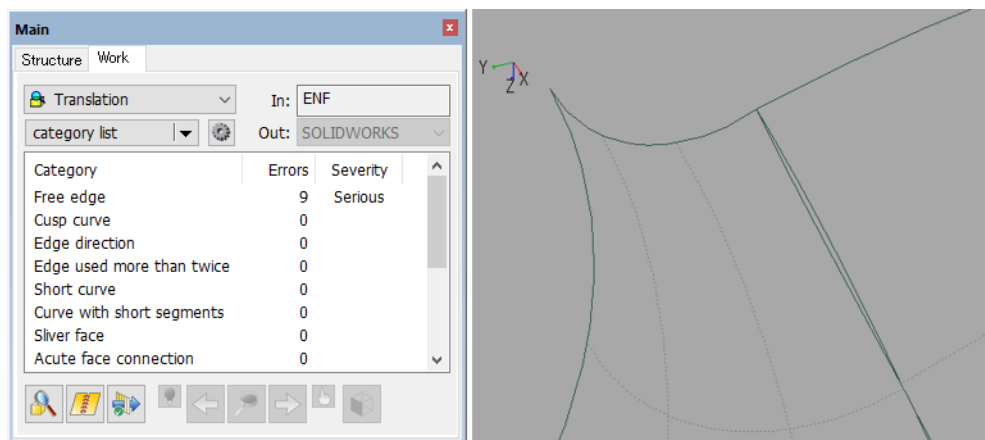
- Press [Delete Elements] () in Auxiliary Tool on Navigation panel.
- If you move the mouse cursor over on the edge highlighted in red on "3D View" window, "Multiple options" will appear.



Right-click the mouse in that state to display "Pick Which?" dialog. Specify (Face [1]) in "Pick Which?" dialog, and press [Done] (✓).



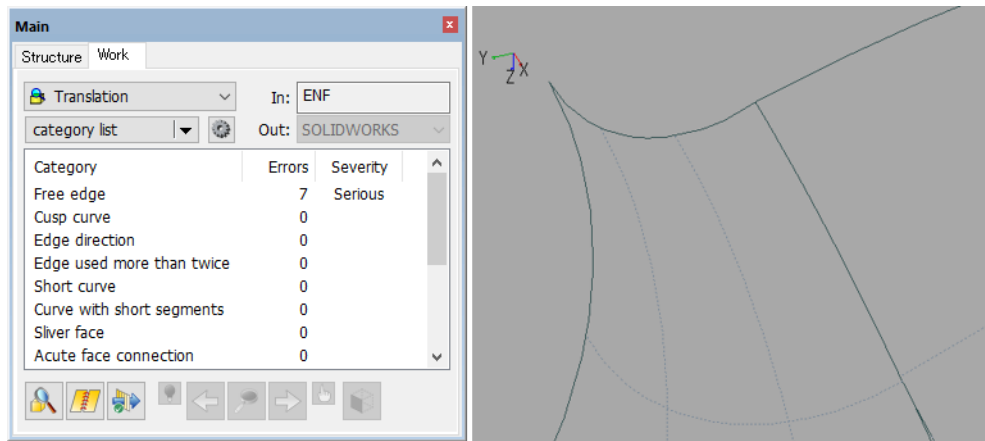
6. Specified face is deleted and the category list is updated. Press [Quit (Esc)] (✗) to quit the command.



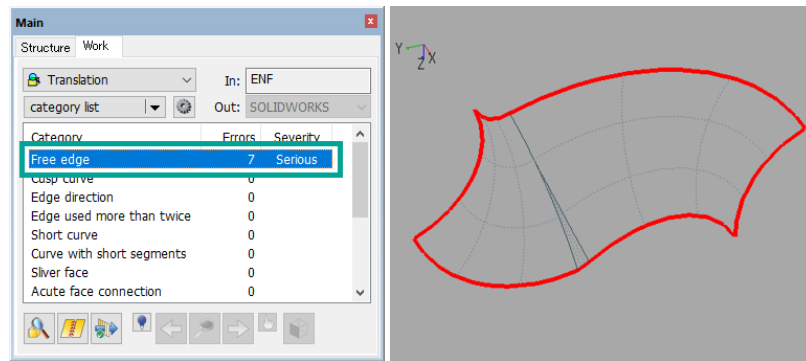
7. Select [Modify] > [Repair Solid] > [Stitch (pick)] (🔧) from the menu.
8. Pick one of the edges tangent to the deleted (Face [1]), and press [Done] (✓). Similarly, pick the other edge, and press [Done] (✓).



The edges are connected, and the category list is updated.

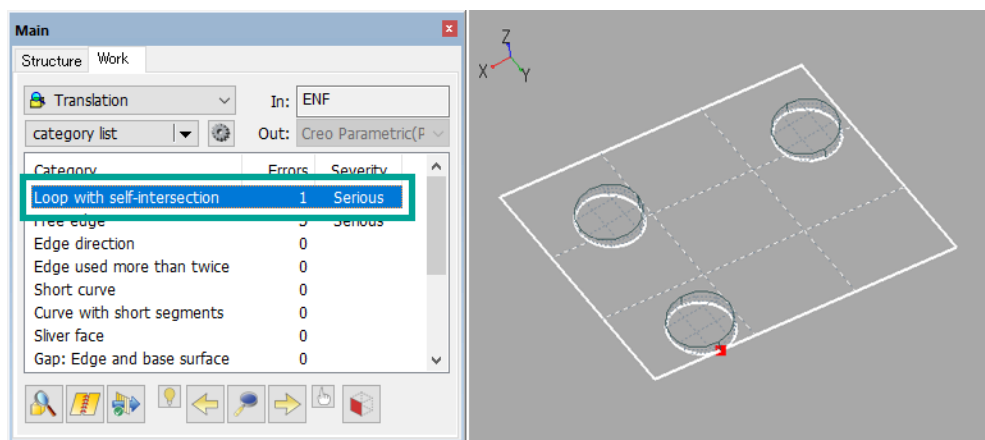


Please note that although several "Free edge" errors still remain, since they are the boundaries of the sample model (sheet), these errors do not need to be modified.



■ Modify with "Divide Face"

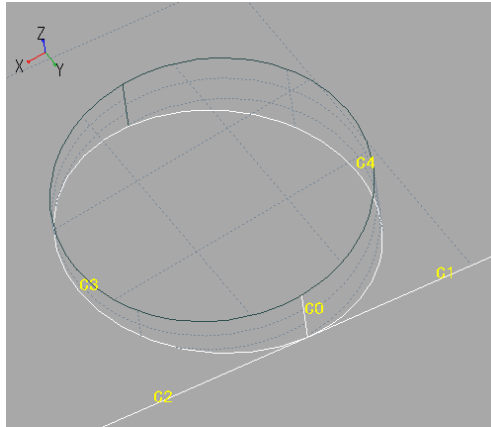
1. Refer to 2.2.1, "Open Files" and open **LoopIS2.drfx** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Loop with self-intersection" from the category list. Target position is highlighted in red on "3D View" window.




Confirm the positional relation of surrounding elements around this error.

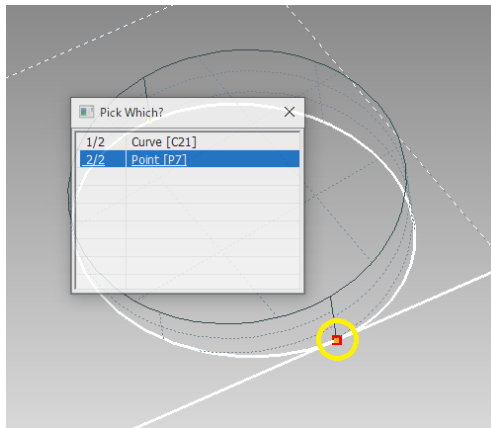
3. Select [Analysis] > [Related Element] from the menu or click [Related Element] (🔍) on the toolbar.


4. Pick the vertex that is highlighted in red. Five edges (C0-C4) related with that vertex are highlighted. Since this point shares the vertex of the edges that make up the loop, it is detected as "Loop with Self-intersection".

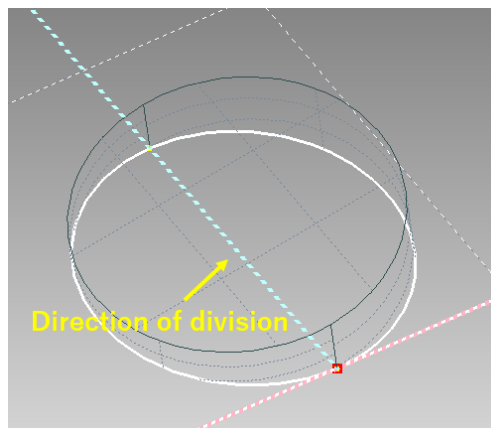


[Clean Self-intersection] () can be used to modify, but in this case, use [Divide Face] ().

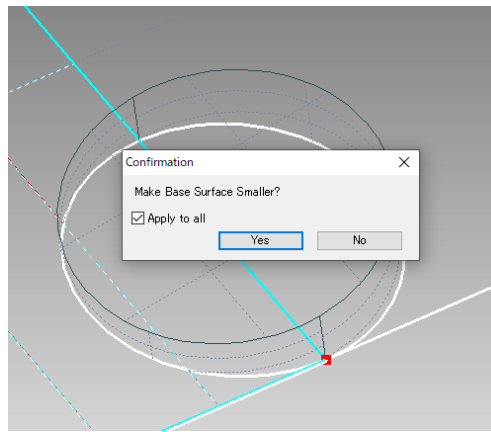
5. Press [Divide Face] () on Navigation panel.
6. On "3D View" window, pick the vertex highlighted in red.



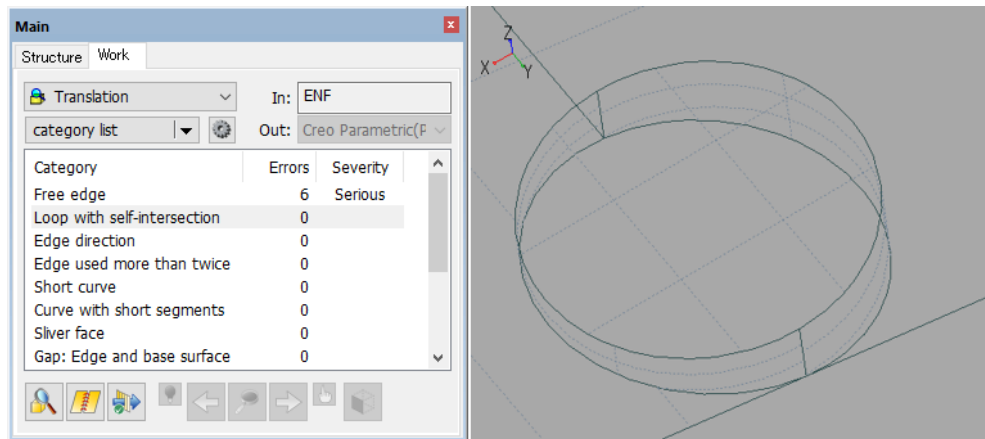
Two dotted lines are displayed around the picked vertex. Pick the dotted line in the direction you want to divide the face, and press [Done] ().



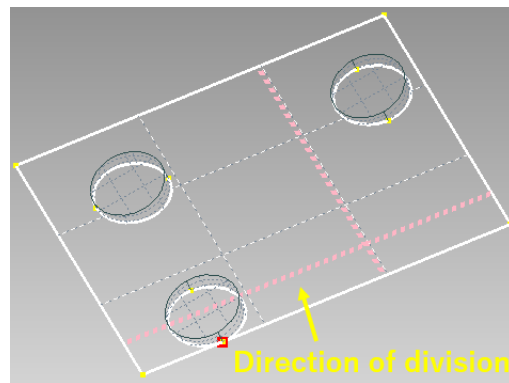
7. The face is divided by the dotted line, and a confirmation dialog will appear. Enable "Apply to all", then click [Yes].



The base surface becomes smaller, and the category list is updated.

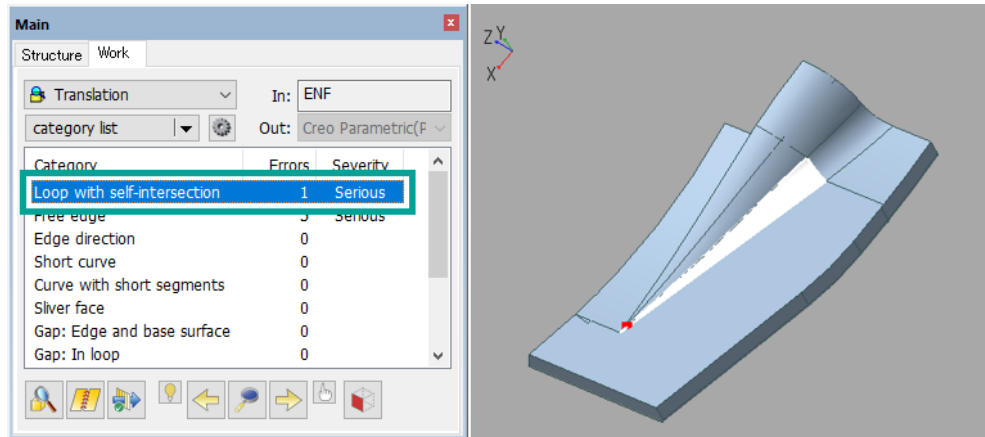


Please note that if the four edges that share a vertex result in having different faces, it's also possible to modify the position and direction of the face division as shown below.

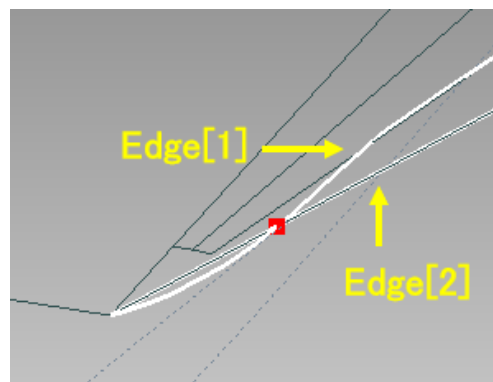


■ Modify with "Recalculate Surface"

1. Refer to 2.2.1, "Open Files" and open **LoopIS3.drfx** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Loop with self-intersection" from the category list. Target position is highlighted in red on "3D View" window.

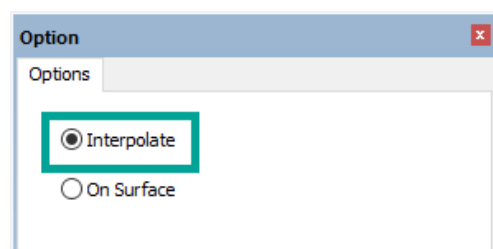


When zooming into the target, you can see that Edge [1] and Edge [2] are intersecting.



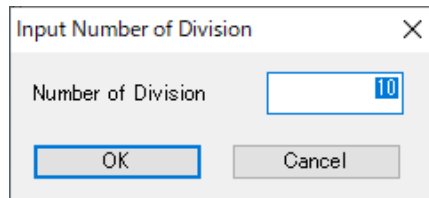
Self-intersection at this position is caused by a distortion of Edge [1]. In this case, Edge [1] will be modified by recalculation.

3. Press [Recalculate Surface] () in Repair Tool on Navigation panel.
4. In [Option] panel, select "Interpolate".

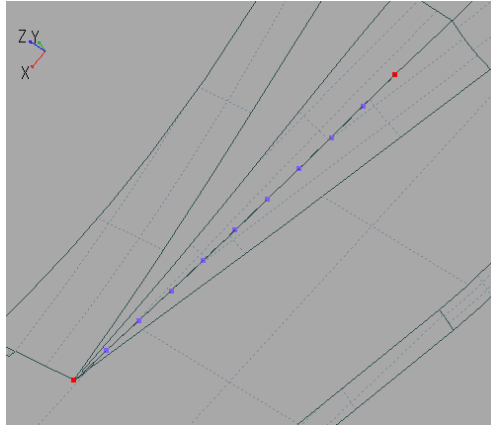


Pick Edge [1] on "3D View" window.

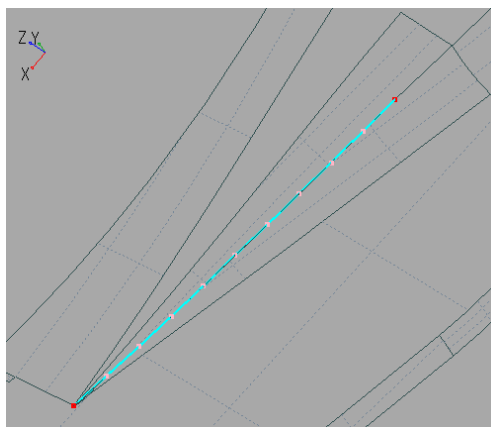
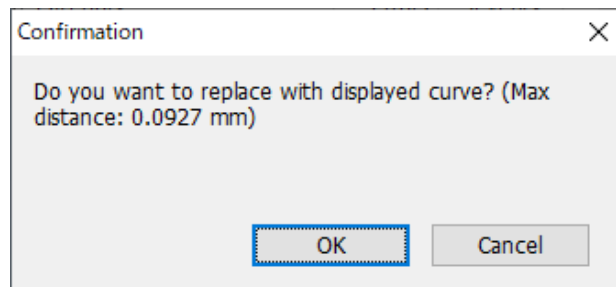
5. "Input Number of Division" dialog will appear. Use the default value "10", and click [OK].



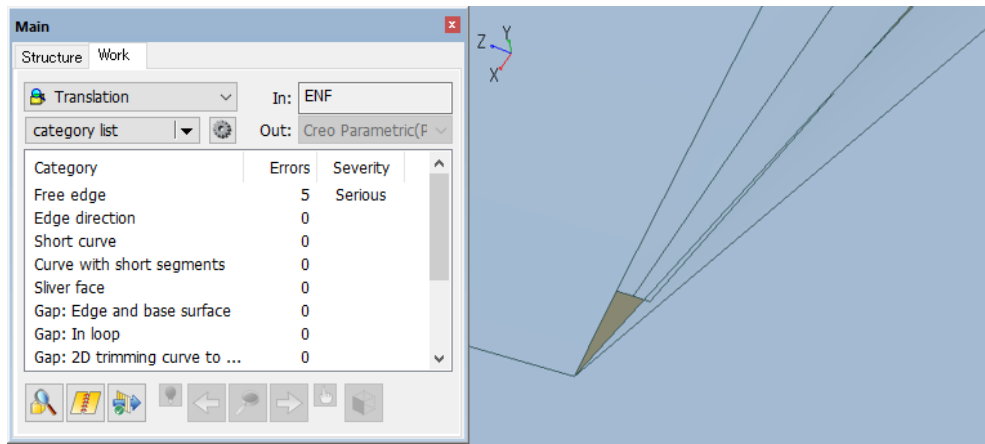
6. Nine passing points are shown, dividing Edge [1] into ten segments.
Pick all nine passing points or enclose the area range, and press [Done] (✔).





7. Recalculated Edge [1] is displayed in light blue and can be previewed. A confirmation dialog will appear. Click [OK].

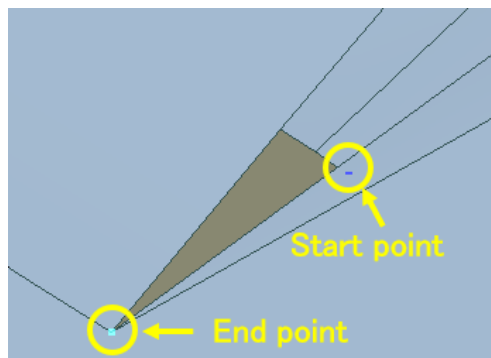


Edge [1] is replaced, and [Loop with self-intersection] is modified.

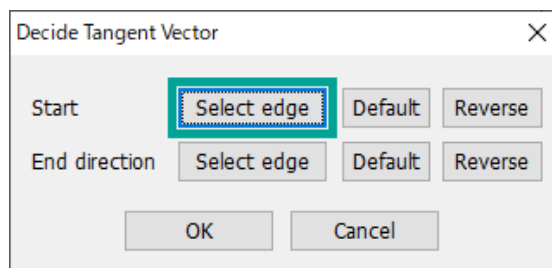


Next, modify the boundary of existing face and fill in the missing parts of the face.
First of all, create an edge that will be the new boundary of the face.

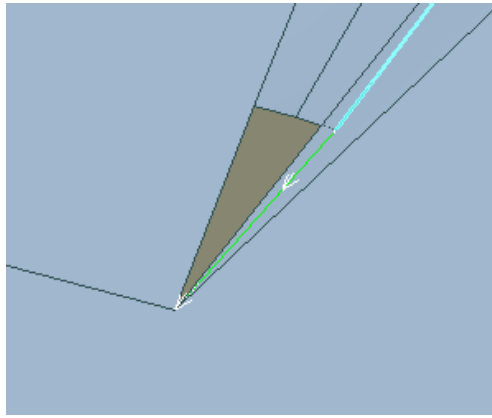
8. Select [Create] > [Curve] > [Interpolation] from the menu or click [Interpolation] () on the toolbar.
9. On "3D View" window, pick points (Start point / End point) at the position where you want to create a curve, and press [Done] ().



10. "Decide Tangent Vector" dialog will appear. Click [Select edge] next to "Start".



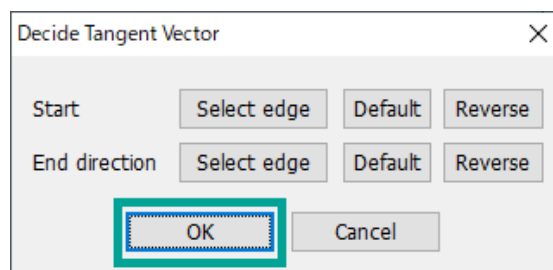
Pick an existing edge that connects to the curve you just created.




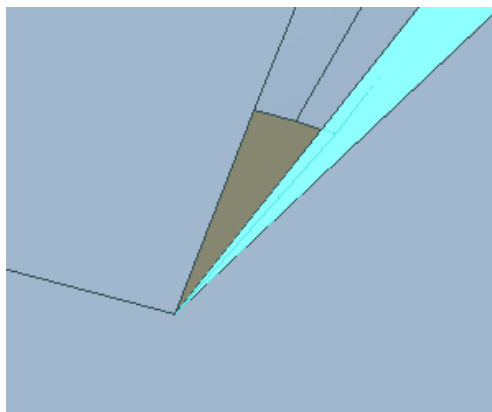
Tangent direction of the new curve is updated to connect smoothly with the existing edge.



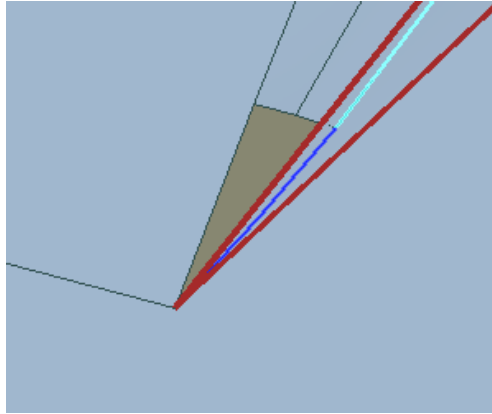
Click [OK] in "Decide Tangent Vector" dialog.



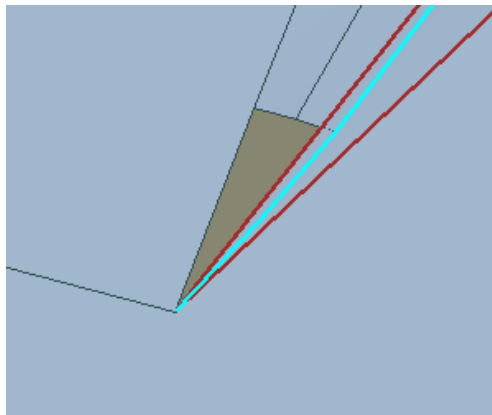
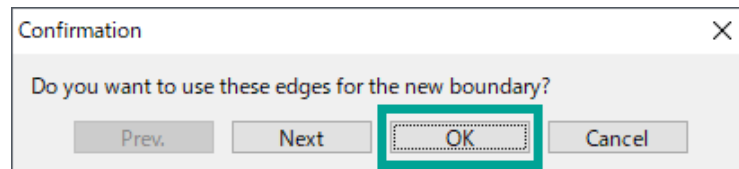
11. Select [Modify] > [Repair Solid] > [Change Face Boundary] from the menu or click [Change Face Boundary] () on the toolbar.
12. On "3D View" window, pick the face whose boundary you wish to change.



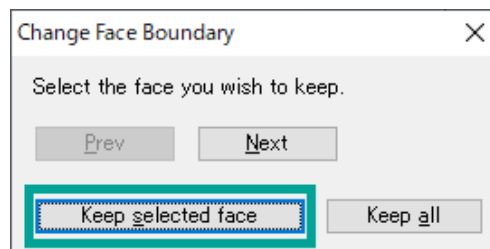
13. Pick the curve you just created and the edges that connect to it, and press [Done] (✓).

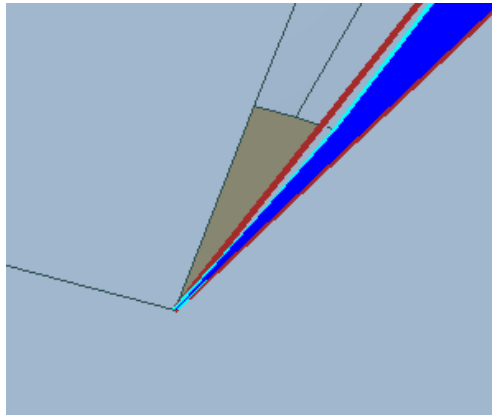


14. A confirmation dialog will appear. Confirm that the new boundary edges are highlighted on "3D View" window, then click [OK].

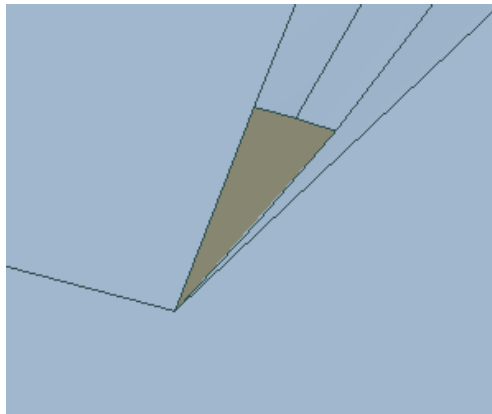




15. "Change Face Boundary" dialog will appear. Confirm that the face you want to keep is highlighted in blue on "3D View" window, and click [Keep selected face].

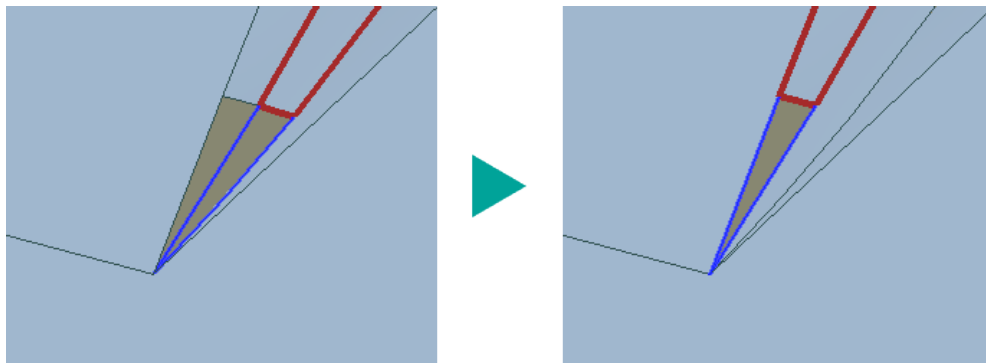




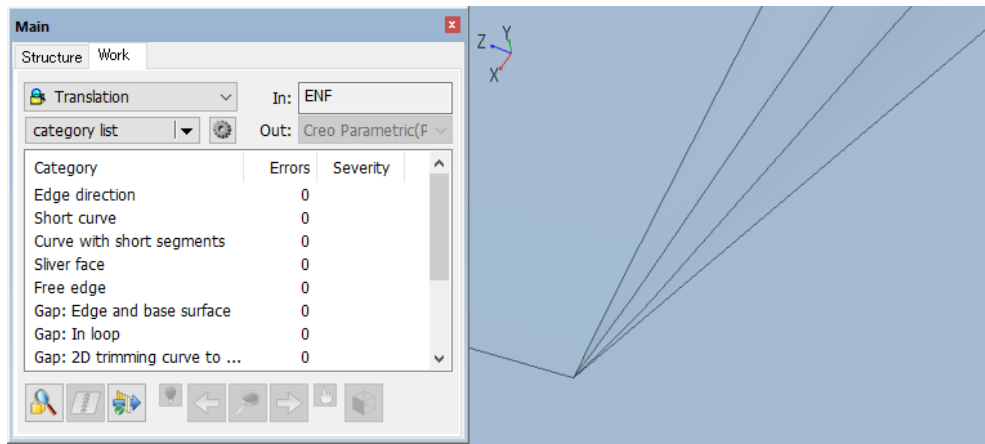
Boundary of the face has been changed.



16. With the same procedure, use [Interpolation] () and [Change Face Boundary] () to fill the holes.

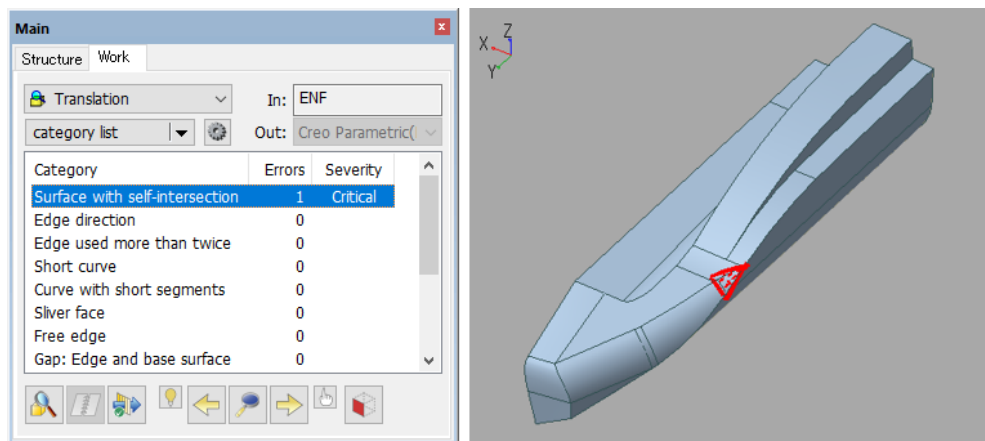


When all holes are filled, the number of detected errors is "0" in the category list.

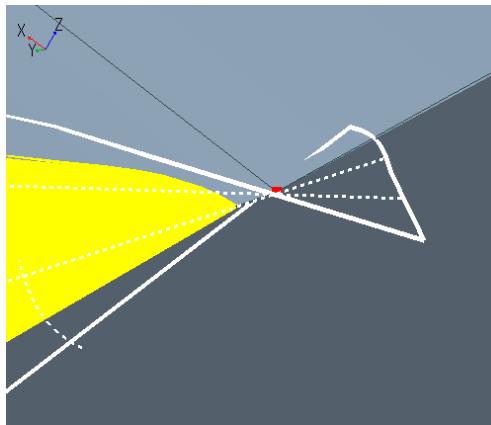



5.1.4. Modifying Surface with Self-intersection

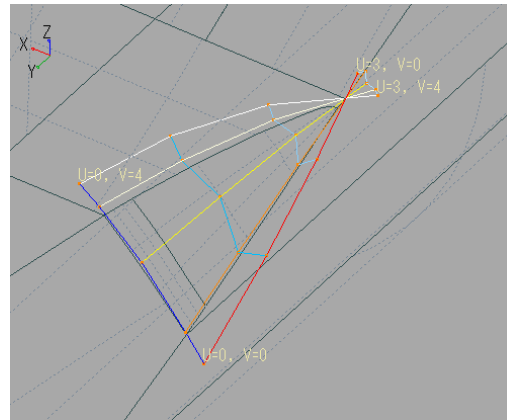
1. Refer to 2.2.1, “Open Files” and open **SurfaceIS.drfx** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Surface with self-intersection" from the category list. Target position is highlighted in red on "3D View" window.





Zoom into the target position and confirm the face self-intersecting.

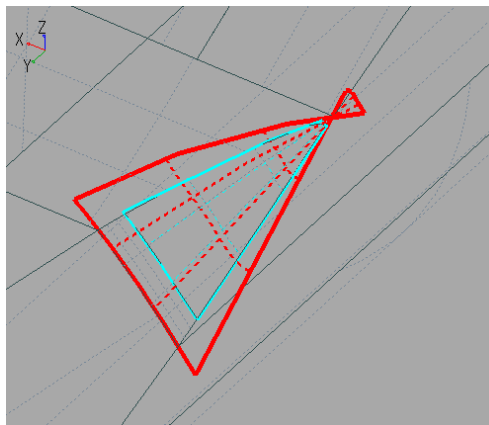
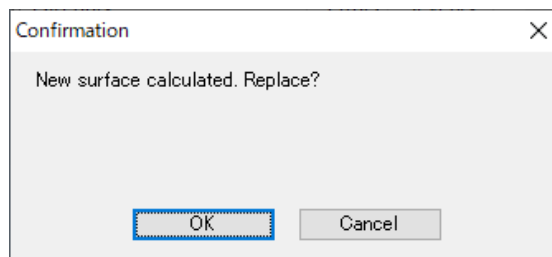


In the error "Surface with self-intersection", it might be easier to see where they intersect by checking the control points of the surface. Surface control points are displayed by selecting [Analysis] > [Control Points] > [Surface] () and picking a face.

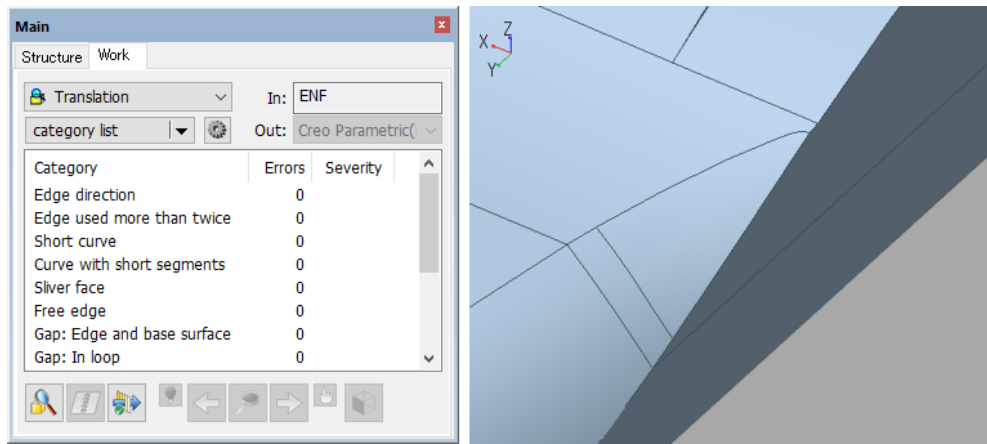


Main method to modify "Surface with self-intersection" is the function [Recalculate Surface] (); however, in this case, the error will be modified by [Make Base-Surface Smaller].

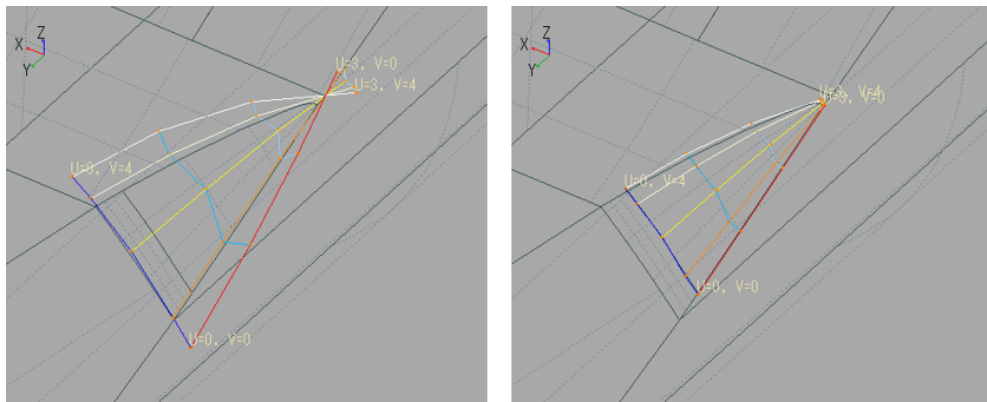
3. Press [Make Base-Surface Smaller] () in Repair Tool on Navigation panel.
4. On "3D View" window, the reduced base surface is highlighted in light blue. A confirmation dialog will appear.



Click [OK]. Self-intersection is modified and the category list is updated.

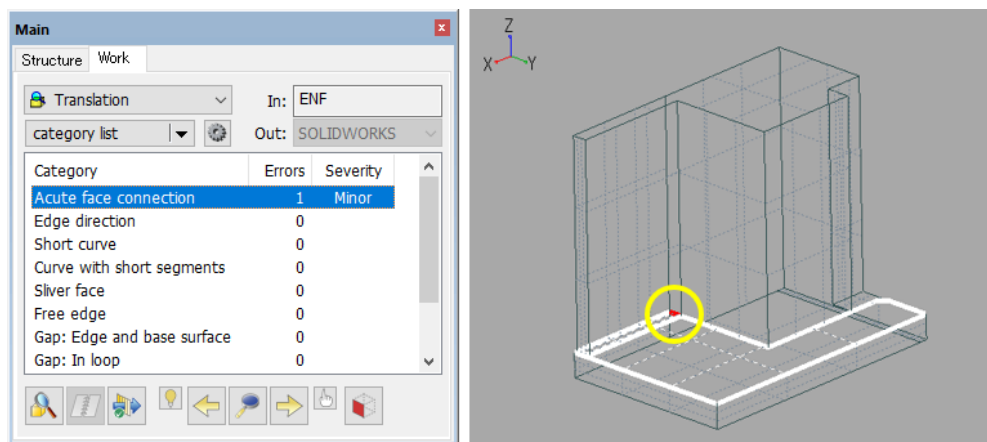



By reducing the base surface in this way, you can modify self-intersection without changing the geometry of face.

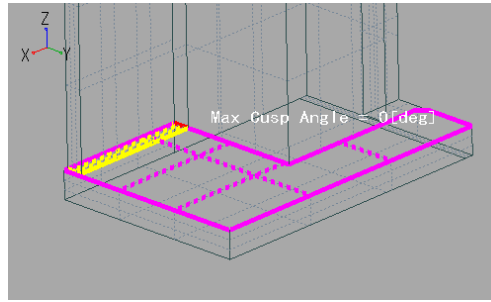


5.1.5. Modifying Acute Face Connection

1. Refer to 2.2.1, “Open Files” and open **FaceCusp.drxf** from the <tutorial> folder.
2. In [Main (Work)] panel, select "Acute Face Connection" from the category list. Target position is highlighted in red on "3D View" window.



3. In [Main (Work)] panel, click [Zoom current target] () to confirm the two faces (yellow and magenta) that are connected at an acute angle.



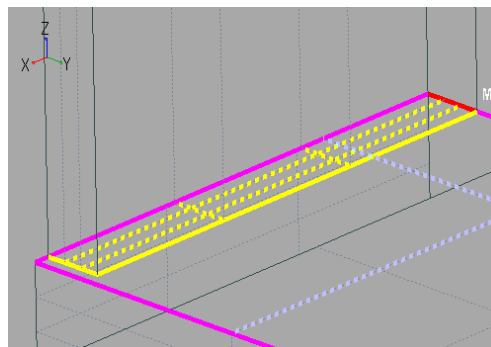
For this sample model, modify the error by deleting the invalid face (yellow) that exists inside the solid.

The functions [Remove Faces With Fix] (🔧) and [Change Face Boundary] (📐) can be used to modify. Here, use [Change Face Boundary] (📐) to modify.

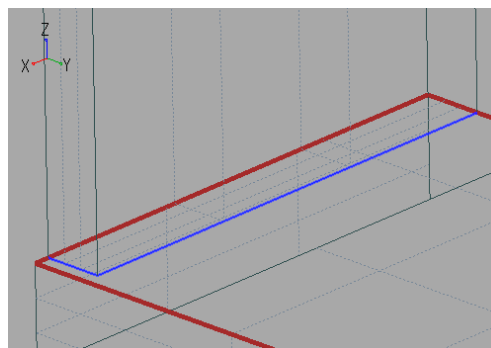


When using the function [Remove Faces With Fix] (🔧), the yellow face is deleted, and the magenta face boundary is automatically changed so that there are no unused edges in the boundary with the deleted face.

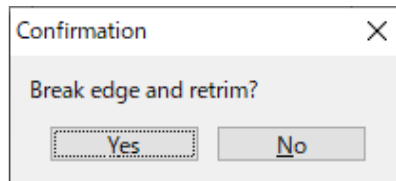
4. Press [Change Face Boundary] (📐) in Repair Tool on Navigation panel.
5. Pick the magenta face on "3D View" window.



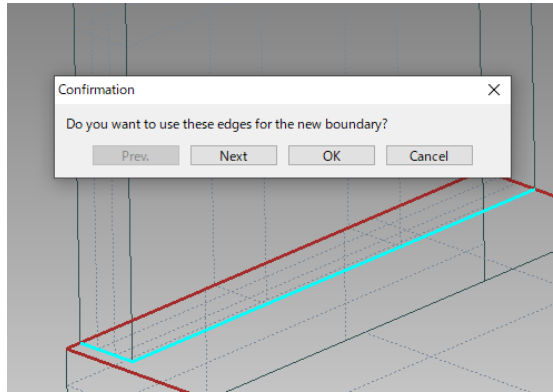
6. To change the boundary of the magenta face, pick the edge that will be the new boundary. Press [Done] (✅).



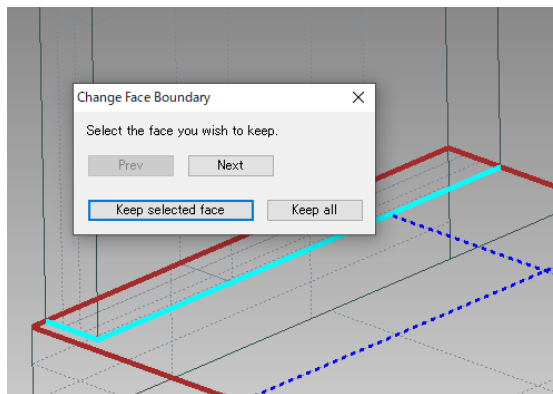
A confirmation dialog will appear. Click [Yes].



7. A confirmation dialog will appear. Confirm that the new boundary edges are highlighted on "3D View" window. Click [OK].

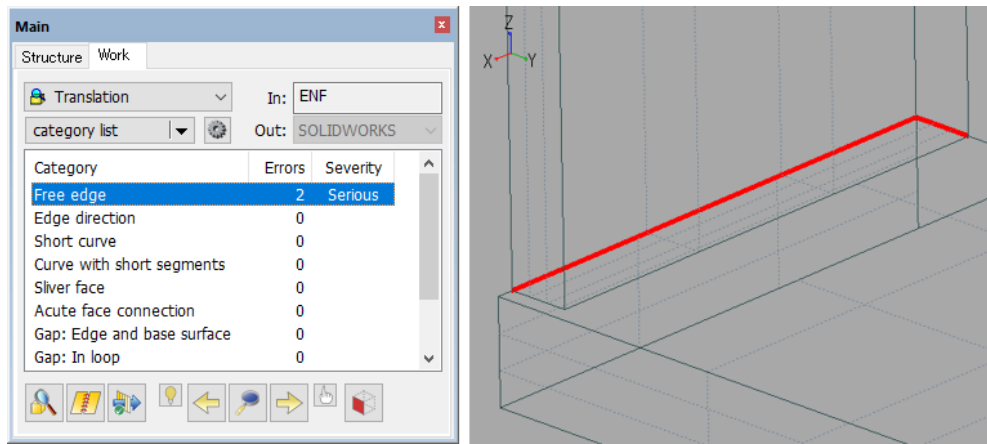


8. "Change Face Boundary" dialog will appear. Confirm that the face you wish to keep is highlighted in blue on "3D View" window, then click [Keep selected face].

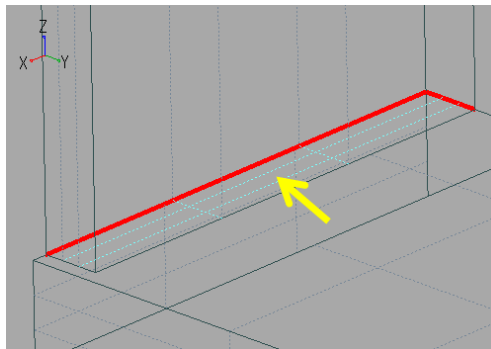


Face boundary is changed, and the category list is updated. Press [Quit (Esc)] (✖) to quit the command.

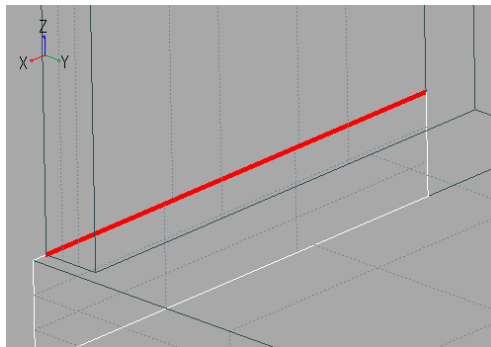
9. In [Main (Work)] panel, select "Free edge" from the category list. The part where you changed the boundary of the magenta face is highlighted in red.



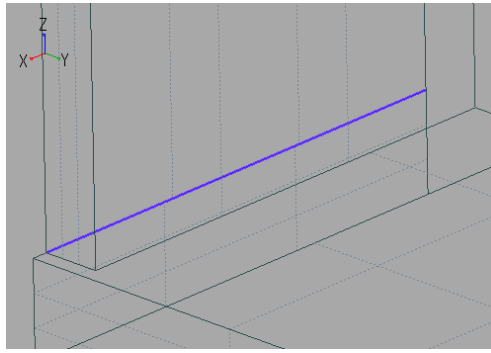
10. Select [Edit] > [Delete] from the menu or click [Delete] (✖) on the toolbar.
11. On "3D View" window, pick the face where the boundary of the face has been changed, and press [Done] (✔).



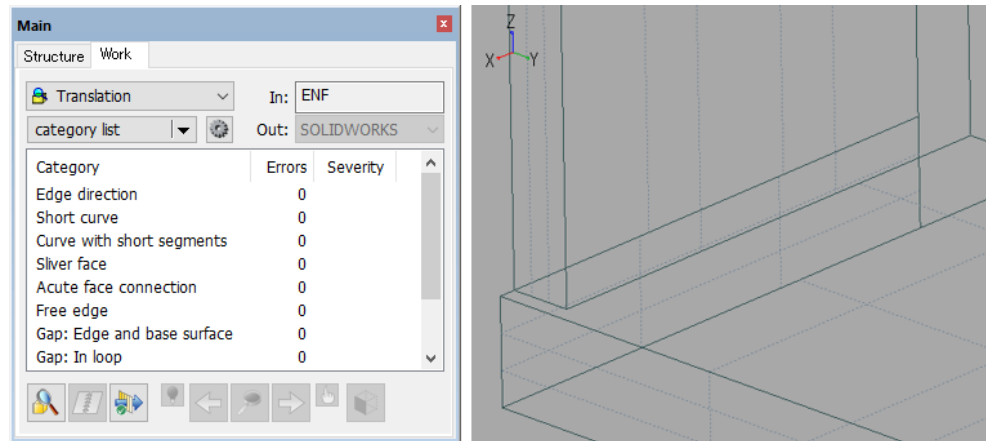
The face is deleted. Press [Quit (Esc)] (✖) to quit the command.



12. Press [Stitch (pick)] (🧵) in Repair Tool on Navigation panel.
13. Pick the edge highlighted in red on "3D View" window, and press [Done] (✔).



The edge is stitched and the category list is updated.



Appendix A: Appendix

A.1. Error Items

Category	Error Item	Description	Severity
Tiny Element	Short Curve	Detect a curve that is shorter than the tolerance in length.	Critical
	Sliver Face	Detect a face with maximum width less than the tolerance.	Critical
	Tiny Step Free Edge	Detect a step-like geometry with a tiny edge that has been used once on the face.	Minor
Topology	Edge Direction	Detect an edge whose direction does not match the direction flag of the belonging loop.	Critical
	Free Edge	Detect an edge which is used by a face only once.	Serious
	Missing face	Detect an edge loop which contains an edge used by a face once.	Serious
	Edge used more than twice	Detect an edge that is shared by faces more than twice.	Serious
	Isolated Curve	Detect a curve that do not have topological relation with any faces.	Minor
Gap around Vertex	Gap: Terminal Point of Edges	Detect a gap between end points of edges linked to the same vertex.	Serious
	Gap: In loop	Detect a gap between the edge endpoints in the loop.	Serious
	Gap: Edge and Vertex	Detect a gap between a vertex and an endpoint of the edge linked to that vertex.	Serious
	Gap: Vertex and Surface	Detect a gap between the vertex and the related UV curve at the actual distance.	Serious
Gap between Edge and Surface	Gap: Edge and Base Surface	Detect a gap between the edge used on the face and the base surface.	Serious
	Gap: Edge and UV Curve	Detect a gap between the edge used on the face and UV curve.	Serious
	Gap: 2D trimming curve to 3D	Detect the actual gap of UV curve associated with one edge.	Serious

Category	Error Item	Description	Severity
Invalid Element	Surface with Singular Point	Detect the surface where U and V directions are parallel at the corner of the curved surface and the normal line cannot be calculated.	Serious
	Edge crossing of Surface Pole	Detect the edges of face passing through the poles of the base surface.	Critical
	Intersecting Loops	Detect loops that intersect each other on the same face.	Critical
	Loop with Self-Intersection	Detect a self-intersection loop.	Serious
	Surface with Self-Intersection	Detect a self-intersecting surface.	Critical
Continuity	Cusp Curve	Detect a curve with G1 discontinuity.	Critical
	Creased Surface	Detect a surface with G1 discontinuity. (Check the discontinuity for both u and v directions.)	Critical
Quality	Curve with Many Segments	Detect curves with more than a certain number of segments (control points).	Minor
	Surface with Many Patches	Detect surfaces with more than a certain number of patches (control points).	Minor
	Surface with Small Patch	Detect small patches on a surface.	Serious (*1)
	Curve with Short Segment	Detect a curve with small segment.	Serious (*1)
	Aspect Ratio of Patch	Detect a patch with high aspect ratio.	Minor
	Aspect Ratio of Surface	Detect a surface with high aspect ratio.	Minor
	Curve with Oscillations	Detect a curve whose control points form oscillation.	Minor
	Surface with Oscillations	Detect a surface whose control points form oscillation.	Minor

Category	Error Item	Description	Severity
Others	Edge Interference	Detect interference within tolerance between edges which belong to the same volume.	Serious
	Duplicated Vertex	Detect duplicated vertices which are located closer to each other than the tolerance.	Serious
	Acute Edge Connection	Detect where the two edges connect at an acute angle in the loop.	Minor
	Acute Face Connection	Detect where the adjacent face and the base surface is connected at an acute angle in the loop.	Minor (*2)
	Duplicate Faces	Detect whether a face completely overlaps another face within the tolerance.	Minor
	Shells interference in a Part	Detect an intersection between two shells belonging to the same part.	Minor

(*1) Critical only for NX I-deas

(*2) Serious only for NX

Explanation of Severity Level

The following section describes how error items for each severity affect data translation.

Critical

Serious problems may occur, such as data not being passed to the target CAD system.

Serious

When passing data to the target CAD system, some data may not be passed, or an error may occur in the operation after being passed.

Minor

This is just a warning. This level hardly affects the data translation. However, it is recommended to look through the errors because they may cause problems in the post processes such as analyzing, manufacturing, etc.

A.2. Analytic Surfaces

In addition to NURBS, which represents general surface, there are surfaces called "Analytical surfaces," such as spheres and cylinders. Analytical surfaces have the following characteristics.

1. Relatively easy to change the geometry in post-process.
For example, the radius and the height of a cylinder can be changed quite easily.
2. Easy to process with CAM, etc.
3. The center of cylindrical or spherical surface can be determined accurately.



Please note that although analytical surfaces have such characteristics, depending on the modification function, the surface may slightly deformed after modification, ending up as a general surface. When passing analytical surface for post processing, the appropriate modification function must be used.

■ Relationships between modification functions and analytical surfaces before and after modification

Function Name	Surface Type
Project Edge on Surface	○
Extend Surface	○
Boundaries→Surface	×
Boundaries, Surface→Surface	×
Recalculate Surface	×
Fit Face to Loops	×
Divide Face	○
Approximate Surface	○ ^[1]
Make Base-Surface Smaller	○
Change Face Boundary	○

Surface Type

○ : Kept

× : Change to General Surface

*[1] If it's possible to approximate the surface as an analytical surface, the surface will be replaced with an analytical one.

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