FFCAM 2023
Description of New Functions

MAKINO
Promise of Performance

Preface
This manual describes the functions added to MAKINO FFCAM 2023 and how to use them.

Created on
March 2023

List of Functions Added to FFCAM 2023

1. Addition of the [Chamfering Route Machining] Function

FFCAM2023 adds a new machining type called [Chamfering Route Machining] to allow chamfering of the contour of the shape.
In 3D milling machining, chamfering is performed with a chamfering tool.
Chamfering can be processed in two modes: [Chamfer Machining] for chamfering on the modeled chamfer shape, and [Light Chamfer Machining] for the purpose of removing burrs from sharp corners.

- **Chamfer Machining Mode**

  This mode outputs the tool paths for the chamfer shape of the model.
  Specify the bottom edge (constant Z height) of the chamfering portion, and perform chamfering at a constant tilt with a chamfering tool of the same tilt (taper angle).
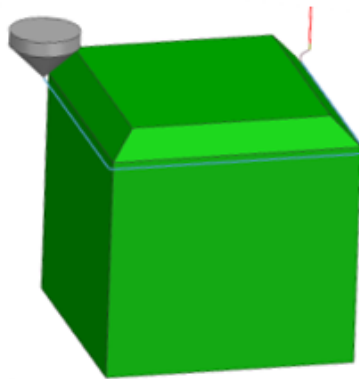
- **Light Chamfer Machining Mode**

  This mode outputs the tool paths of Light Chamfer Machining.
  Perform machining of the specified edge (constant Z height) with the set chamfering width using a chamfering tool.
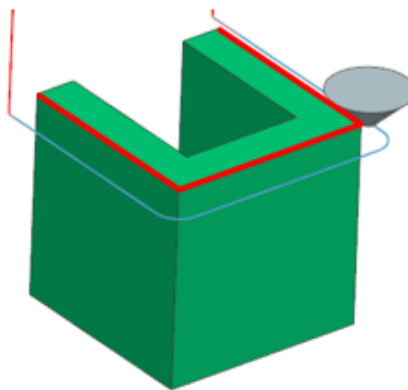
■ Machining Data Settings

Add machining data for [Chamfering Route Machining] from the templates of the User Database or from [Machining Addition] on the Machining Data Selection screen to set chamfering.

- Standard templates in User Database

- [Machining Addition] on the Machining Data Selection screen

■ Chamfer Machining Mode setting screen

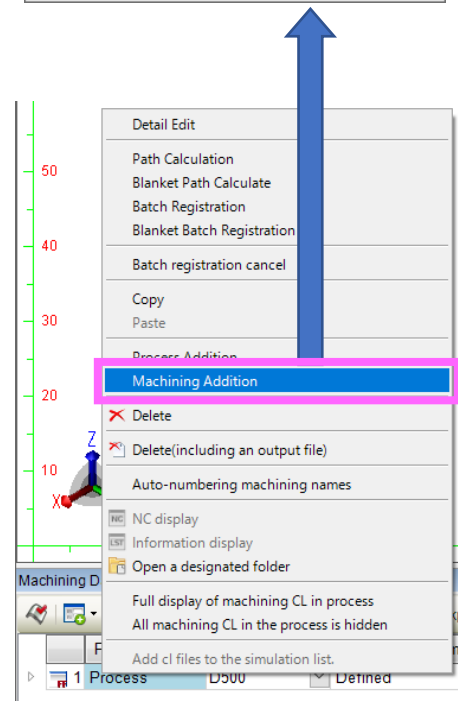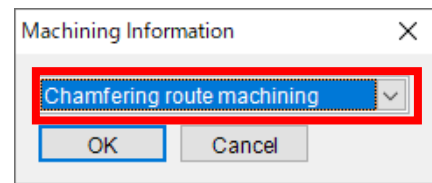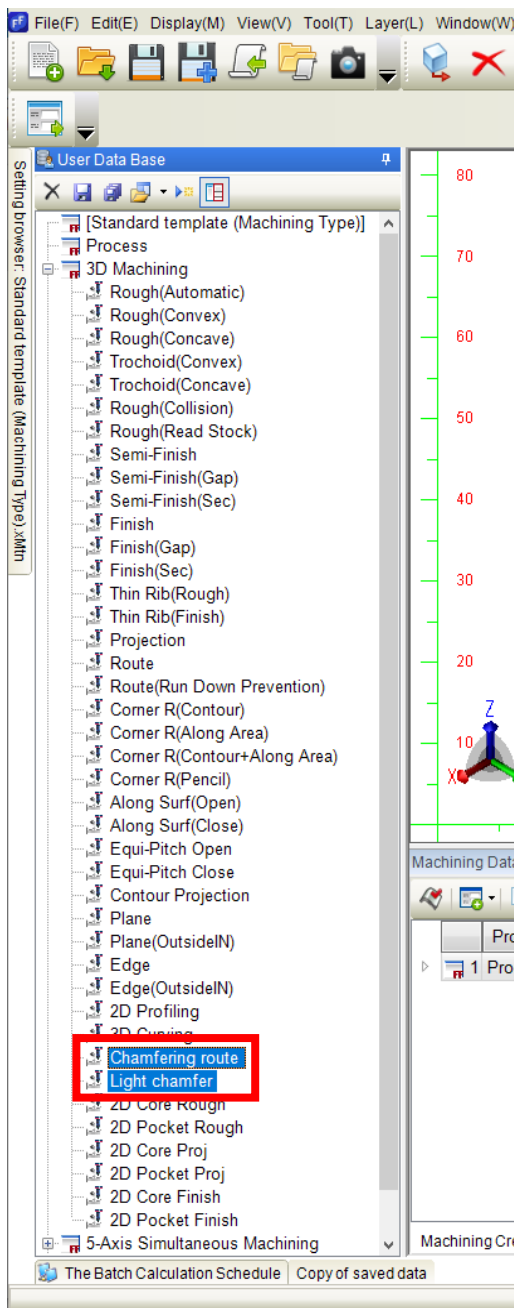The tool position is determined by specifying the bottom edge of the chamfering portion of the shape and setting the height from there to the top edge of the chamfering portion (Chamfering depth).
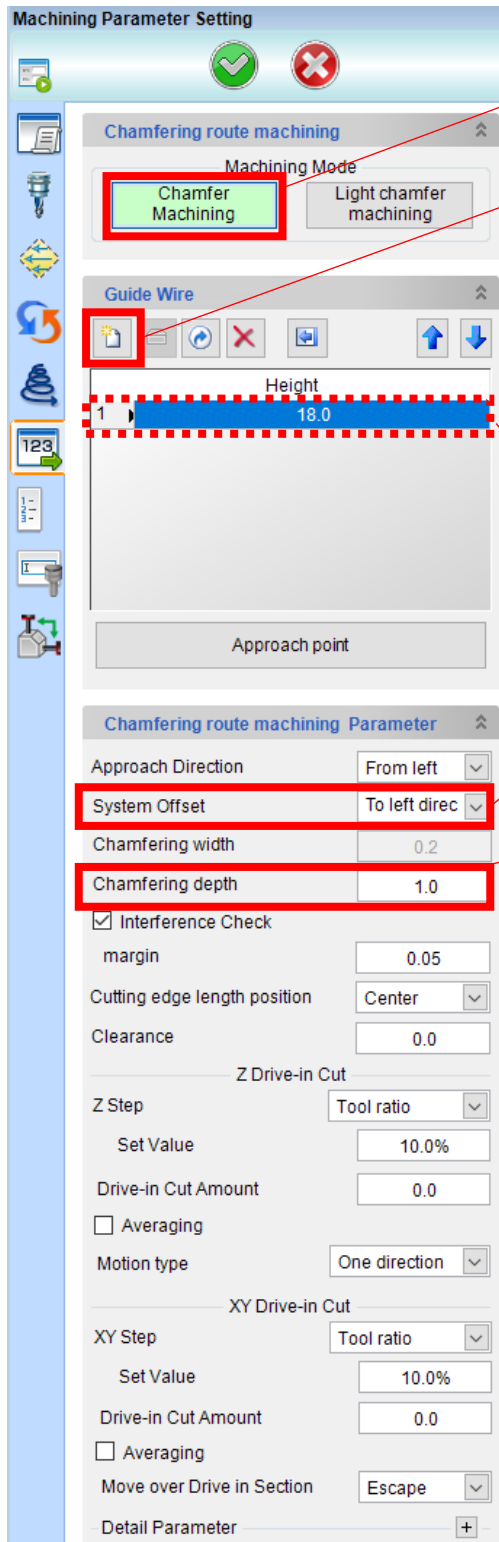
① Select [Chamfer Machining] under [Machining Mode].

② Click the [Wire Selection] button and select the bottom edge of the chamfering portion of the model geometry.
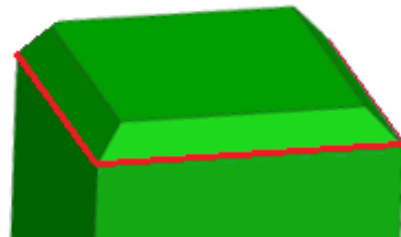
The list displays the height of the selected edge.

③ In [System Offset], select [To left direction] or [To right direction].

④ In [Chamfering depth], enter the height from the bottom to the top edge of the chamfering portion.
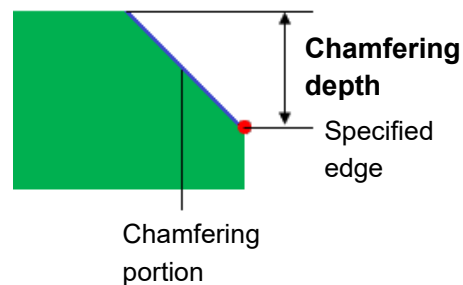
⑤ Set other parameters as necessary.

　* For details on other parameters, see the next page.

Interference Check

Performs interference checks at the time of path calculation. Inhibits the output of tool paths to interference location.
For more information, see "Interference Check" below.

Cutting edge length position

Specify which position of the chamfering tool is applied for cutting.
Select one of the following three levels within the cutting edge length range: the top position as "Top", the middle position as "Center", and the bottom position as "Bottom".

"Top"   "Center"   "Bottom"
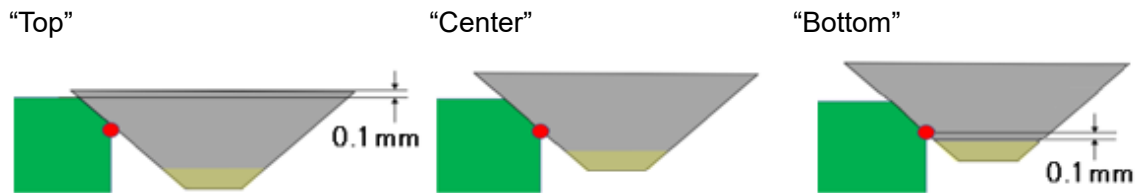
In the case of "Top" and "Bottom", the position is set with a margin of 0.1 mm from the top and bottom of the cutting edge length, respectively.

Z Drive-in Cut

In the [Drive-in Cut Amount], set the total amount of drive-in cuts in the Z direction.
The above [Drive-in Cut Amount] is used to drive and cut the tool path by the [Set Value] of [Z Step].
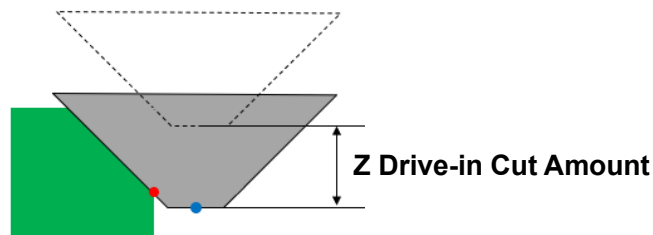
XY Drive-in Cut

In the [Drive-in Cut Amount], set the total amount of drive-in cuts in the XY direction.
The above [Drive-in Cut Amount] is used to drive and cut the tool path by the [Set Value] of [XY Step].
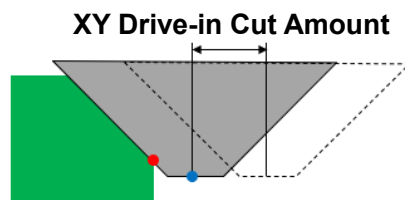
Supplement

The [Cutting edge length position], [Z Drive-in Cut], and [XY Drive-in Cut] are available only in the [Chamfer Machining] mode.

■ Light Chamfer Machining Mode setting screen

When the edge of the chamfering portion is specified, the tool is positioned so that the center of the cutting edge length touches the center of the chamfering portion.
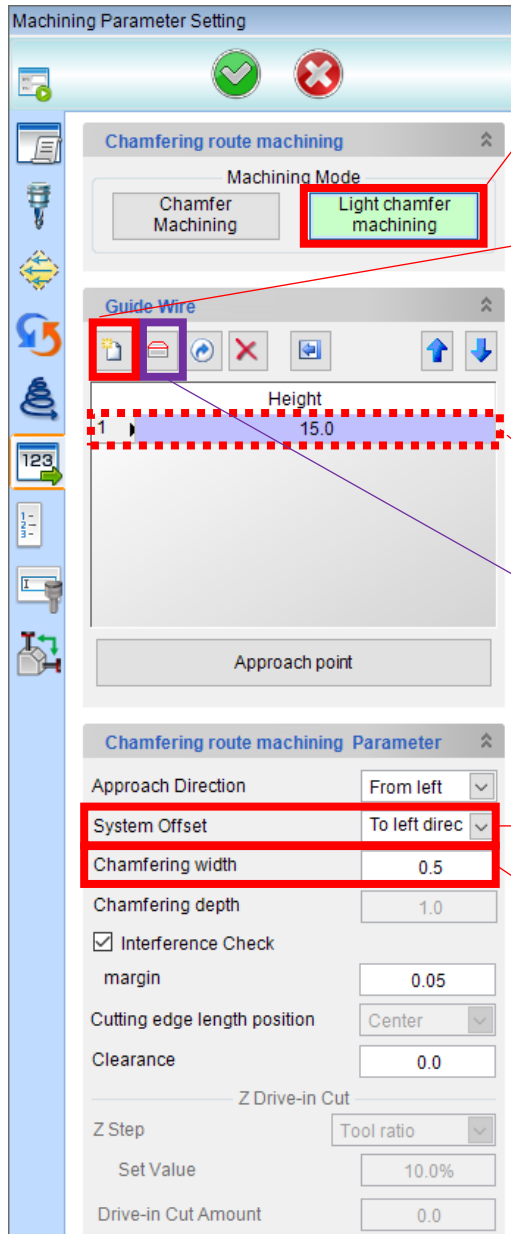
① Select [Light Chamfer Machining] under [Machining Mode].

② Click the [Wire Selection] button and select the edge of the chamfering portion of the model geometry.

The list displays the height of the selected edge.

* The edges to be light chamfered can be automatically obtained by clicking the [Automatic Acquisition of Light Chamfering Edges] button.
See next page for details.

③ In [System Offset], select [To left direction] or [To right direction].

④ Enter the width to be chamfered in [Chamfering width].

Chamfering width

Cutting edge length

Tool path output position
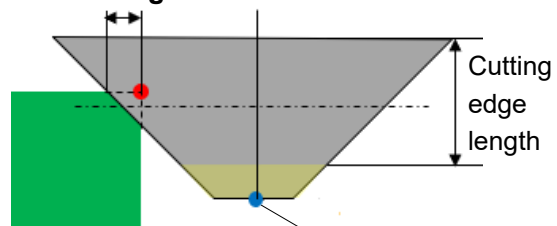
* The tool position is set so that the center of the cutting edge length touches the workpiece.

⑤ Set other parameters such as [Interference Check] as necessary.

* For more information about [Interference Check], see "Interference Check"

5

Automatic Acquisition of Light Chamfering Edges

This option is available when [Light Chamfer Machining] mode is turned to ON.
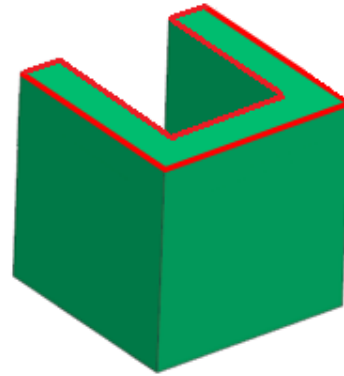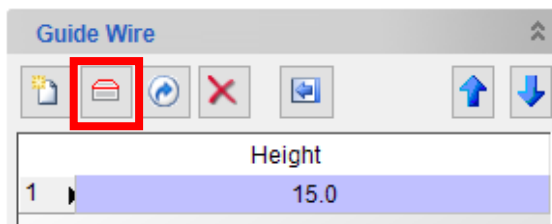When the button is clicked, the function automatically acquires the edges from the model that is the target of light chamfering.
If there is more than one target edge, multiple edges are acquired.

Notes

- The auto-acquisition function is available only for "Solid" models.

- The function automatically acquires the edges at the boundaries of flat surfaces and walls (vertical surfaces).
  Edges on the boundary with sloped faces are not acquired.
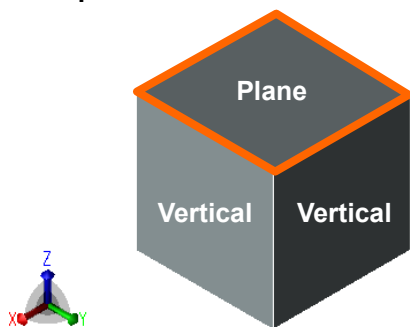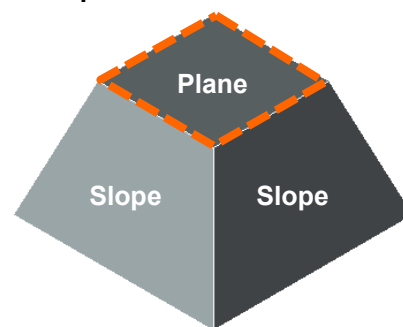
- As shown in the figure below, edges at positions where interference is not taken into account may be acquired.
  After automatic acquisition, check the edges and remove edges that should not be cut from the list.

Edge at position where interference is not taken into account

■ Interference Check

Performs interference checks at the time of path calculation. Inhibits the output of tool paths to interference location.
In [Margin], set the margin amount for the interference check portion.

The [Machining geometry] must be specified in the [Process Setting] to use the interference check function.
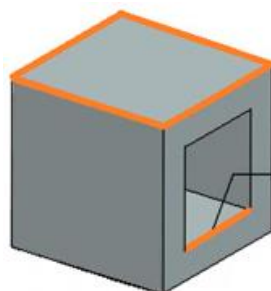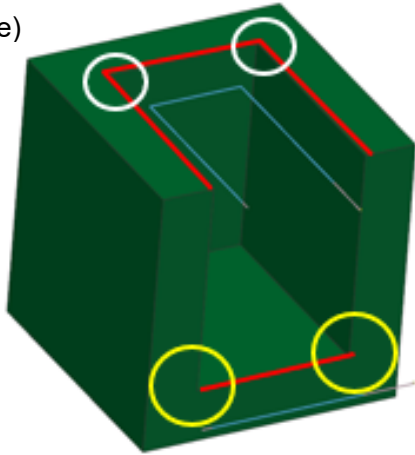
Example)

If the edges near the yellow circles are chamfered, the tool will interfere with the adjacent standing walls.
Use the [Interference Check] function in such cases so that the tool path is not output for the interfering portions.

* Because the tool will not reach the edges near the white circles, uncut areas may remain.
   Use a simulation to check if uncut areas remain, and use other machining methods.

The range of the interference check depends on the mode specified. Details are described on the following pages.
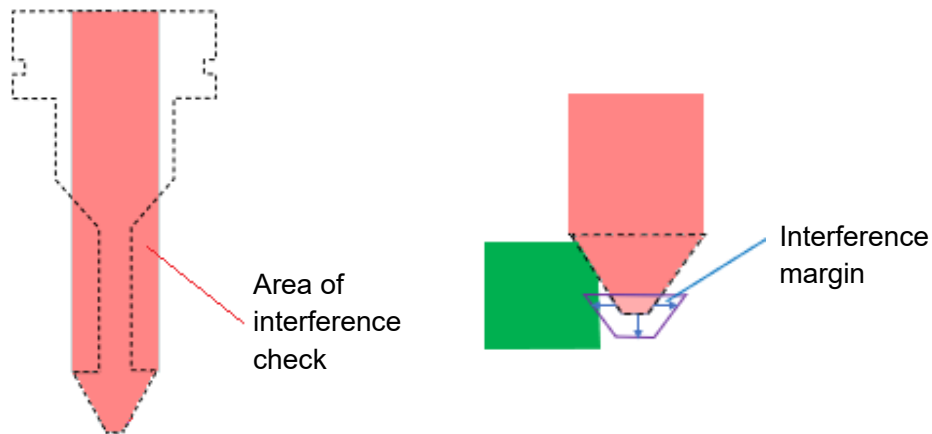
- **In [Chamfer Machining] mode**

  The following portions are checked for interference.

  - Tool (detects for the range assuming that a straight shank is present)
  - Tool bottom face or the tip point
  - If there is no blade at the tool tip, the tip part is targeted.

  The function checks for interference by assuming that the tool has a straight shank.
  Checks for interference at the tool bottom face (if the tip is sharp, for one point at the tip).
  If a part with no cutting edge is at the tip, that part is also targeted for interference checks.
  In [Margin], the amount of margin in the 3D direction can be set for the bottom face or tip of the tool (including parts without a blade).

  Area of interference check

  Interference margin

In [Light Chamfer Machining] mode

The following portions are checked for interference.

- Portion where a straight shank is assumed to be present directly above the cutter
- Tool bottom face or the tip point
- If there is no blade at the tool tip, the tip part is targeted.

The function checks for interference by assuming that a straight shank is present directly above the cutter.
The tool bottom face (if the tip is sharp, for one point at the tip) is also checked for interference.
If a part with no cutting edge is at the tip, that part is also targeted for interference checks.
In [Margin], the margin amount can be set in 3D directions for the interference check portions.


Area of interference check

Interference margin
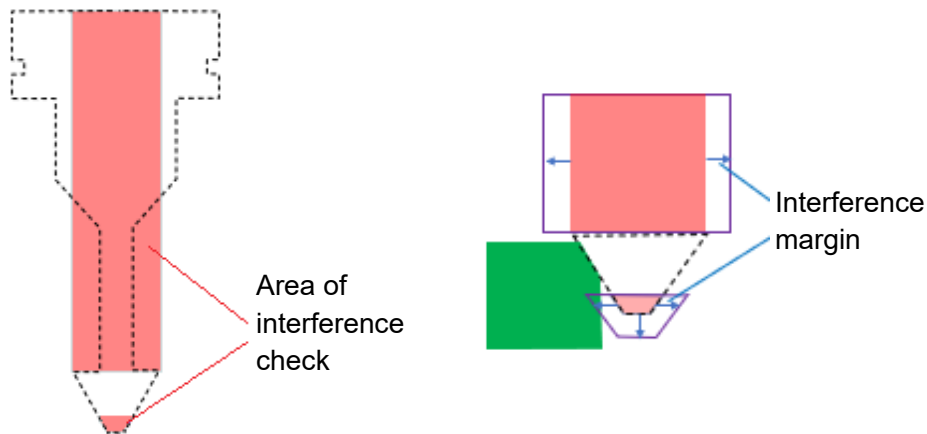
Note
Since portions other than the above cannot be checked, the cutting edge may contact areas other than the specified edges.
In the simulation, confirm the results with a color-coded display of uncut areas or gouging check of the machining geometry.
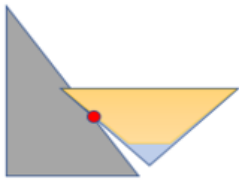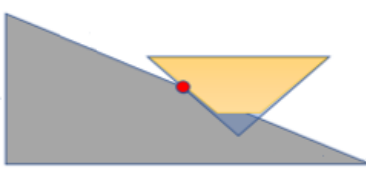Of the interference points, interference is more likely on the path approach, so setting [Motion Type] of [Path Approach Motion] to "Vertical" makes it easier to set the approach.

Example of the cutting edge contacting an unspecified edge)

Specified edge

Unspecified edge
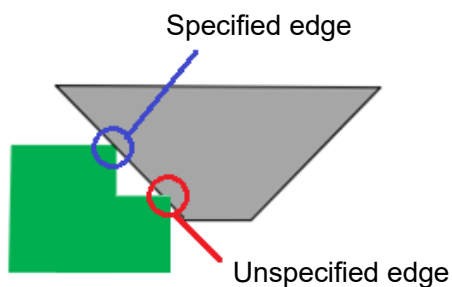
■ Precautions

● In [Chamfering Route Machining], [Toolpath Assist], [Uncut Model Setting], [Check Geometry/Finishing Allowance], and [5-axis simultaneous movement] in [Machining Parameter Setting] are not available.

● In [Chamfering Route Machining], only chamfering tools of 3-axis machines that can be registered in the tool DB are available.

● In 🔧 (Tool Display) of the chamfering tool, when 🔲 (Contact point) of ✛ (Tool position) is specified, the tool display position differs depending on the picked position on the model, as shown in the table below.

| Pick position	Tool display position (Red dot is pick position.)
Pick a top surface (flat surface).	
Pick a wall.	
Pick a tilted surface. (When the angle of the tilted surface and the taper angle of the tip are the same angle)	
Pick a tilted surface. (When the angle of the tilted surface is steeper than the taper angle of the tip)	
Pick a tilted surface. (When the angle of the tilted surface is less steep than the taper angle of the tip)	

● The [Tool Radius Offset] cannot be set for [Chamfering Route Machining].

● The [Machining Direction Calculation] in the index machining setting cannot be used for [Chamfering Route Machining].

● The [Tool Use Range Setting] in 4/5-Axis Simultaneous Machining cannot be used for [Chamfering Route Machining].

2. Addition of the [Output Plane Only] Function for the [Auto Recognize Horizontal Plane] Function of Contour Machining

In FFCAM2023, an [Output Plane Only] function has been added to the [Auto Recognize Horizontal Plane] function of contour machining.
This function can be used when [Contour Face Cut Machining] is specified in contour machining.
When the [Output Plane Only] function is used, the contour face cut machining path is output only in areas that include an automatically recognized plane. This function can reduce the machining time.
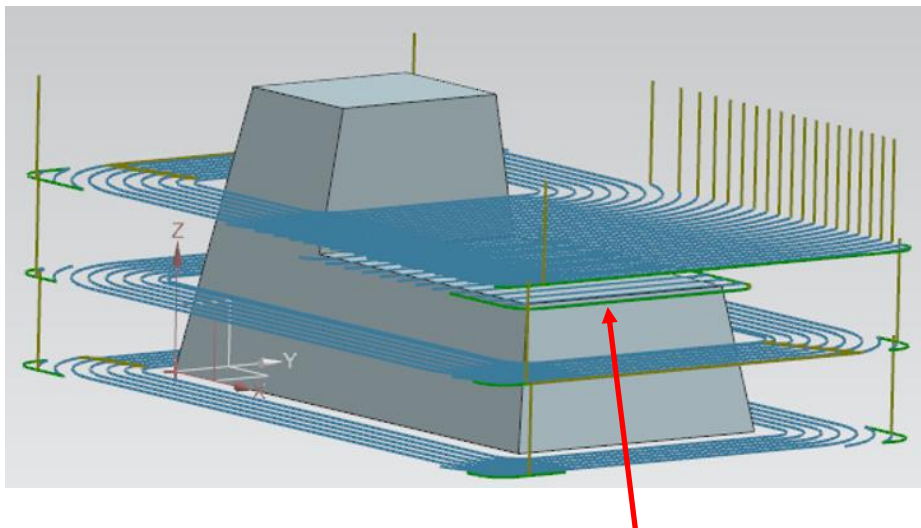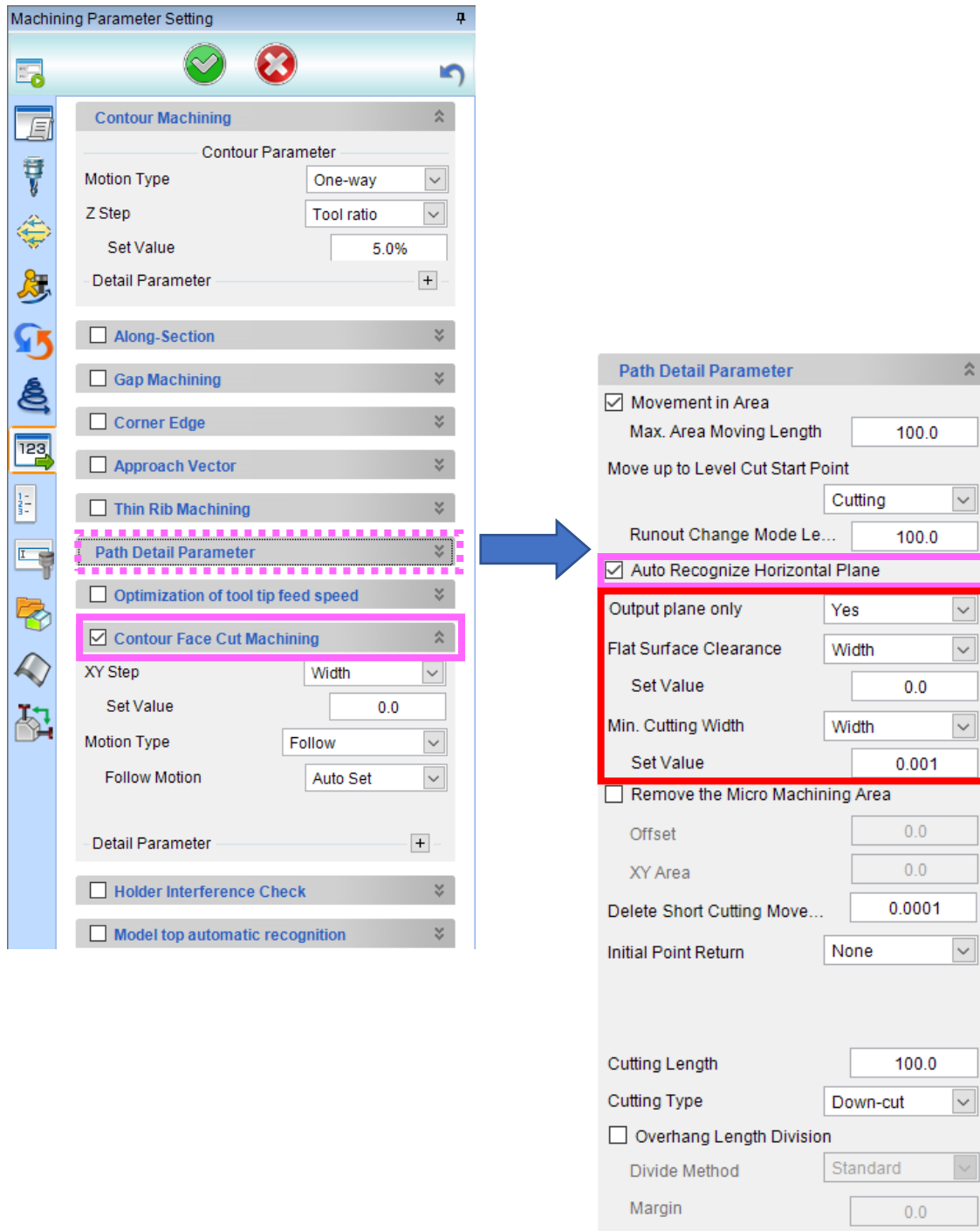

When the [Output Plane Only] function is used, the contour face cut machining path is output only for the surfaces that are recognized as planes.

The original contour face cut machining path is first output with a consistent step amount. The contour face cut machining path of [Output Plane Only] is output for the uncut areas after the entire original contour face cut machining path is output.
  * When machining planes that were automatically recognized, they are cut from the sides.

■ Setting Screen

When [Contour Face Cut Machining] is selected in Contour Machining and [Auto Recognize Horizontal Plane] is selected in the [Path Detail Parameter] setting, the parameters of [Output Plane Only] can now be set.

Output plane only	Yes
Flat Surface Clearance	Width
Set Value	0.0
Min. Cutting Width	Width
Set Value	0.001

Output Plane Only

Select whether the contour face cut machining path is to be output only for the surfaces that are recognized as planes.

No   Yes
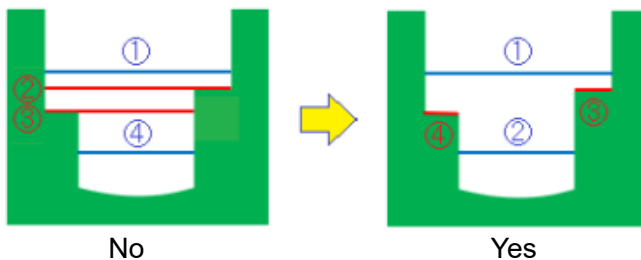
——— : Original contour face cut machining path

——— : Contour face cut machining path of the [Auto Recognize Horizontal Plane]

Numbers: Machining order

* If [Output Plane Only] is set to "Yes", the machining time can be reduced, since machining is done only where additional tool paths are required.

Flat Surface Clearance

This item is displayed when "Yes" is selected for [Output Plane Only].
It outputs a contour face cut machining path over an area wider than the plane by the input value.
Select from "Width" or "Tool Ratio (%)", and set the value according to the method selected.
For example, by entering a value greater than or equal to the tool radius, the side of the tool can be used to approach the uncut areas on the plane.

Plane   Side surface cutting

Clearance
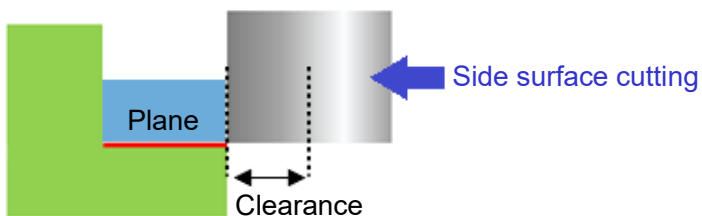
Min. Cutting Width

If the difference between the Z level of the original contour face cut machining and the Z level of a plane recognized by the [Auto Recognize Horizontal Plane] function is less than the specified value, then the contour face cut machining path of [Output Plane Only] will not be output.

Select from "Width" or "Tool Ratio (%)", and set the value according to the method selected.
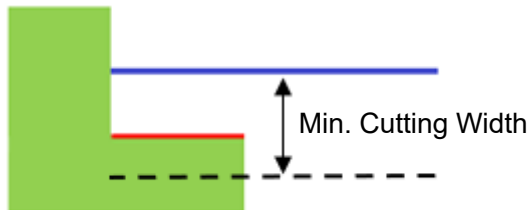
————— : Z level of the original contour face cut machining

————— : Recognized plane path

■ Precautions

● The [Output Plane Only] function cannot be used together with gap machining.

● If a gentle slope exists adjacent to a plane, as shown below, and there is a risk of interference with uncut areas if rapid feed is used, change the settings to perform tilt infeed.
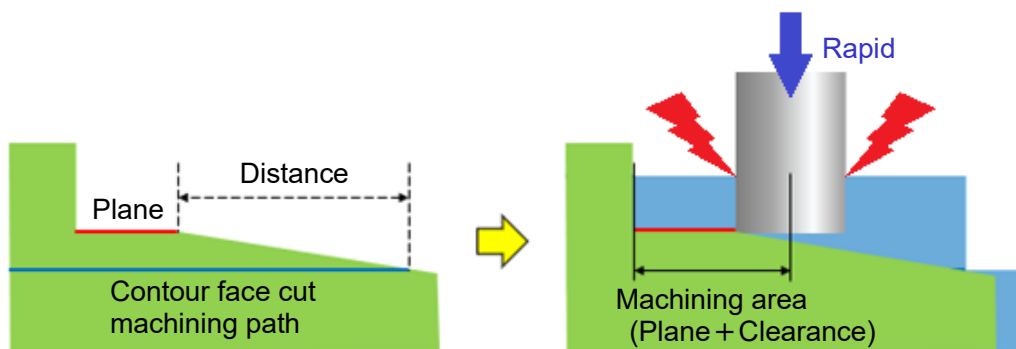
Increasing the clearance value more than necessary to increase the machining area is not recommended.
Unnecessary paths outside the plane (including planes within the model) may be output.
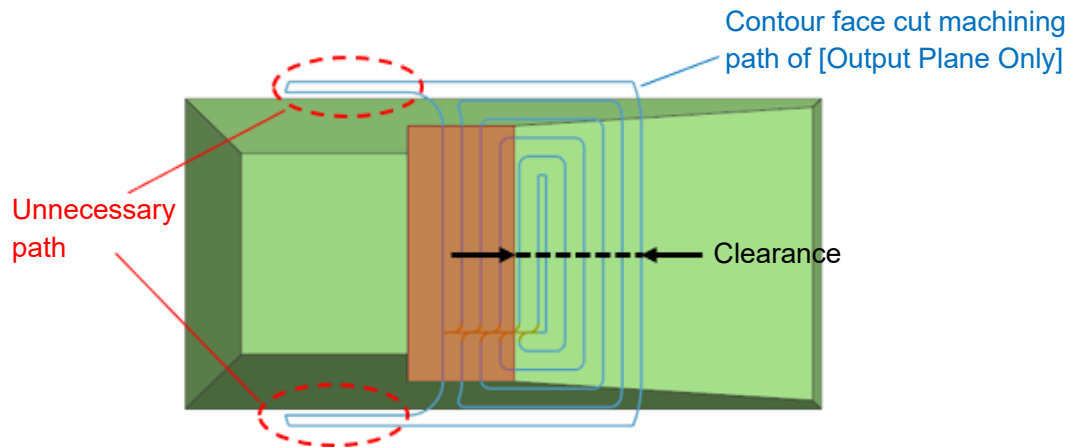
- For the uncut areas that remain after the workpiece was machined with a fixed step value, a plane machining path is output.
  Paths may be output for small uncut areas in some cases.
  Output of such paths can be avoided by decreasing the value of [Flat Surface Clearance] or increasing the value of [Min. Cutting Width].

- Machining time can be reduced with [Output Plane Only] in cases where horizontal planes are distributed across the geometry at various Z heights and the piece is machined with a large stock removal value. (Such as machining solid wood)

- Machining time may increase if the lead amount for tilt infeed is large.

- For the path settings for planes, motions are the same as for the following settings for safety.

    Tilt Infeed Out of Area: On
    Path Approach Motion
      - Circular (Infeed Start: Out-of-Area)
      - Contact Line (Mode: Width)
      - Vertical → Changed to Contact Line (Mode: Width)

3. [Drilling] Addition of the Function to Rearrange the Hole Positions in a Zigzag Order from the Edge

In previous versions of FFCAM, sorting of hole positions for drilling was limited to sorting by axis direction or manual reordering.

In FFCAM2023, the function to sort the hole positions in a zigzag order starting from the edge has been added.

Sorting can be done automatically in a zigzag direction based on the X axis or Y axis.

■ Setting Screen

The [Rearrangement of Hole Position] button has been placed in the [Drilling (Hole Position)] section of the [Hole position] setting screen.
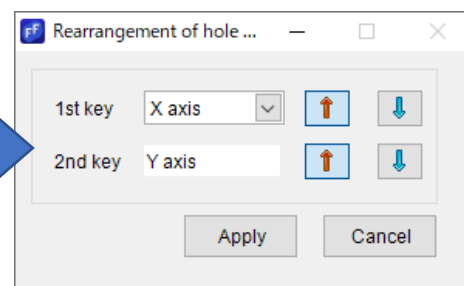
Upon selecting the hole positions to be sorted in the list and clicking the [Rearrangement of Hole Position] button, the hole positions can be set to be sorted automatically in a zigzag direction.

[Drilling (Hole Position)] screen

Select the hole positions to be sorted in the list, and click the 🔃 [Rearrangement of Hole Position] button.
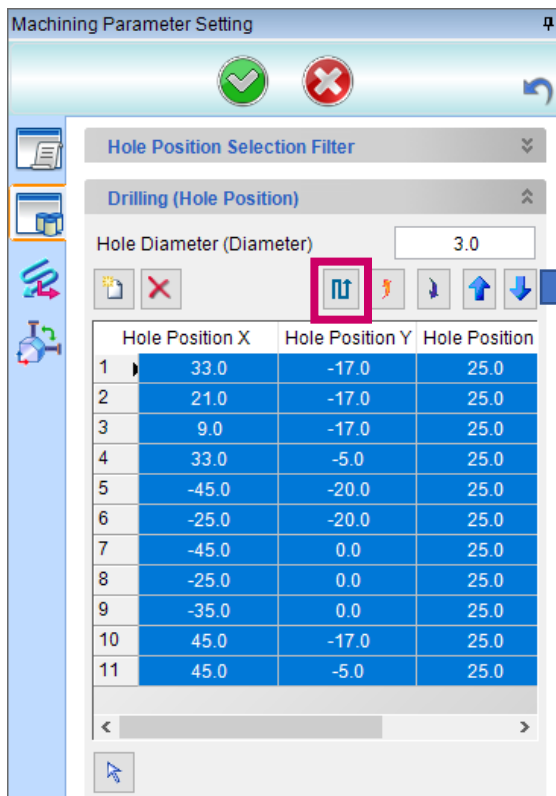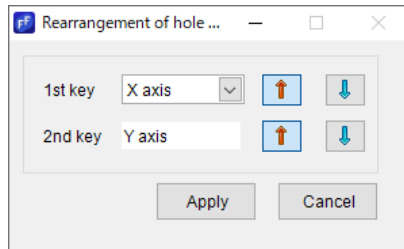
The [Rearrangement of Hole Position] screen is displayed.

In the [Rearrangement of Hole Position] screen, set the sorting based on X axis or Y axis.
After setting the sorting, when the [Apply] button is clicked, the hole positions are automatically sorted in a zigzag direction.

[Rearrangement of Hole Position] screen

Select the axis (X or Y axis) that will be referenced by [1st key], and select the axis direction (⬆ ascending or ⬇ descending).
Select the axis direction (⬆ ascending or ⬇ descending) of the other axis with [2nd key].
When [Apply] is clicked, the selected hole positions are automatically sorted in a zigzag direction according to the settings.
  * The axis for [2nd key] is automatically set according to the axis selected for [1st key].

Sorting example)

<Relative to X axis>

<Relative to Y axis>

Supplement

The axis direction is based on the machining coordinate system.
The X axis direction is ascending, and the -X axis direction is descending (same for Y axis).

4. Enhancement of Function for Copying Process Data Templates from User Database

FFCAM has a function to automatically select a model in [Select (Element)] of [Machining Workpiece (Type)] when a process data template is copied from [User Database] by dragging and dropping it to the [Machining Data Selection] list.

In FFCAM2023, the function to automatically select models when copying templates has been enhanced.

In previous versions of FFCAM, a model was automatically selected when a template whose [Form] under [Machining Workpiece (Type)] was set to [Box Area] or [Cylinder] was copied.

In FFCAM2023, the model is also automatically selected when a template whose [Form] under [Machining Workpiece (Type)] is set to [Model Specified] is copied.
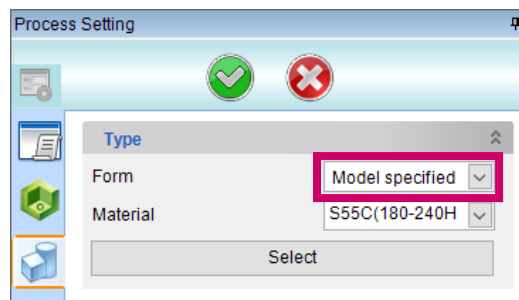
■ Supplementary Note

- When a template is copied, a confirmation screen is displayed asking whether to select the model automatically or not. Select [No] if you do not want the model to be automatically selected.
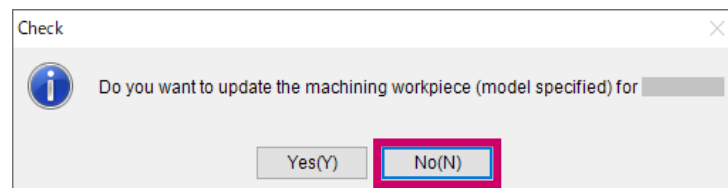
- When copying a template whose [Form] under [Machining Workpiece (Type)] is set to [Specify Curve and Height], the model is not automatically selected.

- Models that are not displayed on the screen (layer hidden or blank) are also automatically selected.

- If a model does not have any parts that meet the selection conditions, the model is not automatically selected.

5. Addition of the Function for Copying Machining Data Templates from User Database

In FFCAM2023, a function has been added to automatically select a model and automatically select the corresponding setting elements when a machining data template is copied from [User Database] by dragging and dropping it to the [Machining Data Selection] list.

When you copy the machining data registered as a template in [User Database] to the [Machining Data Selection] list by dragging and dropping it, the model and the setting elements are automatically selected.
Refer to the table below for the target machining data and the setting items for which elements are automatically selected.

Target machining data	Setting items whose elements are automatically selected
Route Machining	Route Wire (Edge)
Along-Surface Machining (Open)	Drive Surface
Along-Surface Machining (Close)	Drive Surface
2D Route Machining	Guide Wire

■ Setting Screen

[Route] screen in [Route Machining]

When the machining data of [Route Machining] that was registered in [User Database] is copied to the [Machining Data Selection] list, the selection of [Route Wire (Edge)] that was set in the template is automatically performed.
The selection is made based on the filter information for each element selection in the template.
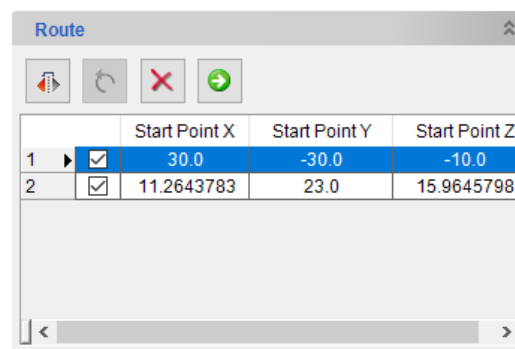
Notes

- If the first and second lines of the list contain the same filter information for element selection, the results of element acquisition for those first and second lines may be the same. (Duplicate)

- The "Path Progress Direction" is not set automatically.
- Even if an element has only one line in the template list, the results of element acquisition may be divided into multiple lines. In such a case, the element selection filter information for the divided lines is the same.

[Open] screen in [Along-Surface Machining (Open)]

When the machine data of [Along-Surface Machining (Open)] that was registered in [User Database] is copied to the [Machining Data Selection] list, the selection of [Face selection] of [Drive Surface] that was set in the template and the [Guide Registration] are automatically performed.
The selection is made based on the filter information for element selection in the template.

Note

- [Edge Selection] and [Control Points (4 points)] for [Outside Curve] are not set automatically.

[Close] screen in [Along-Surface Machining (Close)]

When the machine data of [Along-Surface Machining (Close)] that was registered in [User Database] is copied to [Machining Data Selection] list, the selection of [Face selection] of [Drive Surface] that was set in the template and the [Guide Registration] are automatically performed.

The selection is made based on the filter information for element selection in the template.

Note

- The [Curve] and [Control Points] for [Outside Curve] and [Inside Curve] are not set automatically.

[Guide Wire] screen for [2D Route Machining]

When the machining data of [2D Route Machining] that was registered in the [User Database] is copied to the [Machining Data Selection] list, the selection of [Guide Wire] that was set in the template is automatically performed.
The selection is made based on the filter information for each element selection in the template.

Notes

- If the first and second lines of the list contain the same filter information for element selection, the results of element acquisition for those first and second lines may be the same. (Duplicate)
- The "Path Progress Direction" and [Initial Point Return] are not set automatically.
- Even if an element has only one line in the template list, the results of element acquisition may be divided into multiple lines. In such a case, the element selection filter information for the divided lines is the same.

■ Supplementary Note

- When a template is copied, a confirmation screen is displayed asking whether or not to auto-select the model and element. Select [No] if you do not want the model and element to be automatically selected.
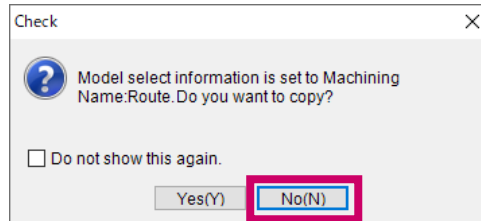
The confirmation screen can be set to be displayed or not in the [Option Set Screen].
※ The existing "Template with a machining area set" is integrated with the confirmation screen control for copying.
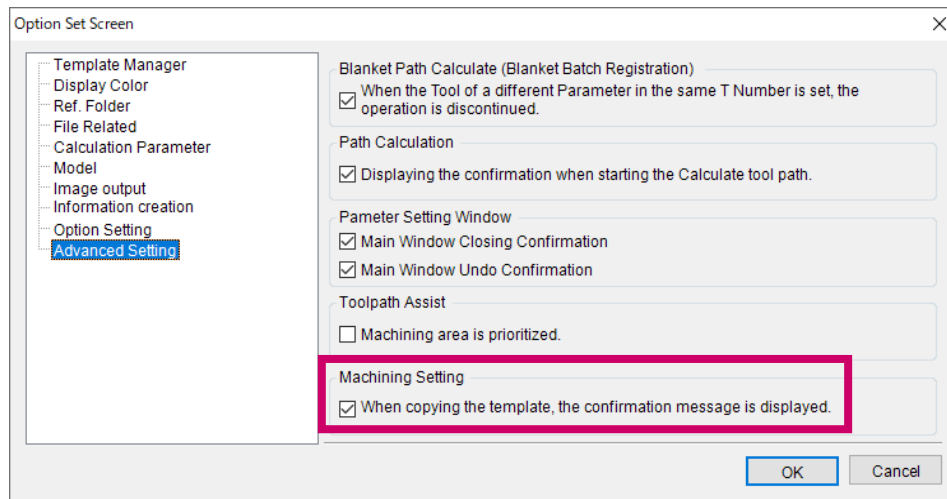
- Models that are not displayed on the screen (layer hidden or blank) are also automatically selected.
- If a model does not have any parts that meet the selection conditions, the model and element are not automatically selected.

6. Addition of the Function to Output Tool Lists in the Order of Machining in Excel Machining Information

When Excel machining information is output as a tool list, a function has been added to output the information in the order in which they are used in machining.

Specify the output method in the [Excel machining information settings (Sheet name)] screen of the option settings.

■ Setting Screen

In the [Excel machining information settings (Sheet name)] screen, the [Machining Order] checkbox is now located next to the [Tool List] checkbox.

[Excel machining information settings (Sheet name)] screen

When the [Tool List] checkbox is selected, the [Machining Order] checkbox becomes selectable.

The [Machining Order] output function is function attached to the [Tool List] output function.

When Excel machining information is output as a tool list, the function changes the order of entry to the order in which they are used in machining.

■ Excel machining information settings example

If the [Tool List] checkbox is not selected:
The [Machining Order] checkbox is not selectable.
Excel machining information is described in the machining list.

If the [Tool List] checkbox is selected and the [Machining Order] checkbox is not selected:
The Excel machining information is described in the tool list in the T-number order.
(Same as the old function)

They are listed in ascending order of the T number.

If the [Tool List] checkbox is selected and the [Machining Order] checkbox is selected: The Excel machining information is described in the tool list in machining order. (New function)

They are listed in machining order.

7. Addition of the Function to Output Simulation Results to Excel Machining Information

A function has been added to output the results of "Overhang Length Calculate" for CL simulation to Excel machining information.

Previously, FFCAM could output only values set by parameters to an Excel machining information sheet, but FFCAM2023 can now output the results of CL simulations to Excel machining information.
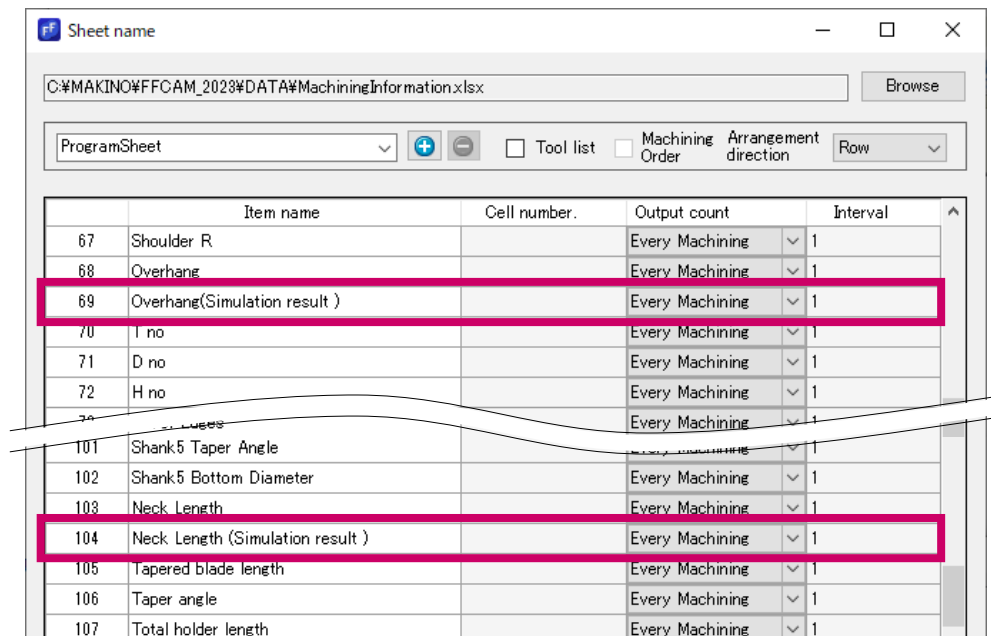
The values of "Overhang length" and "Neck length" calculated by the simulation are automatically entered in the Excel machining information.

■ Setting Screen

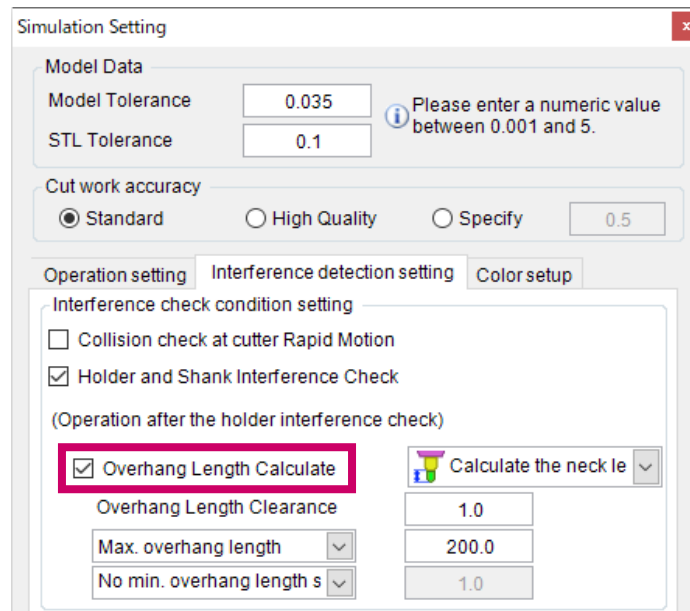### [Excel machining information settings (Sheet name)] screen

Set [Overhang (Simulation result)] and [Neck Length (Simulation result)] as output targets on the [Excel machining information settings (Sheet name)] screen.

When machining information is output during path calculation or in repost information, the "Overhang length" and "Neck length" calculated by simulation are included in the Excel machining information.

[Simulation Setting] screen

To output the values of "Overhang length (Simulation result)" and "Neck length (Simulation result)" to Excel machining information, [Overhang Length Calculate] must be selected in [Interference detection setting] under [Simulation Setting].

If [Overhang Length Calculate] is not selected, the values for "Overhang length (Simulation result)" and "Neck length (Simulation result)" are not included in the Excel machining information.

8. Addition of the Function to Output the Names of Cutter Manufacturer and Holder Manufacturer to Excel Machining Information

A function has been added to output the names of the cutter manufacturer and holder manufacturer in Excel machining information.
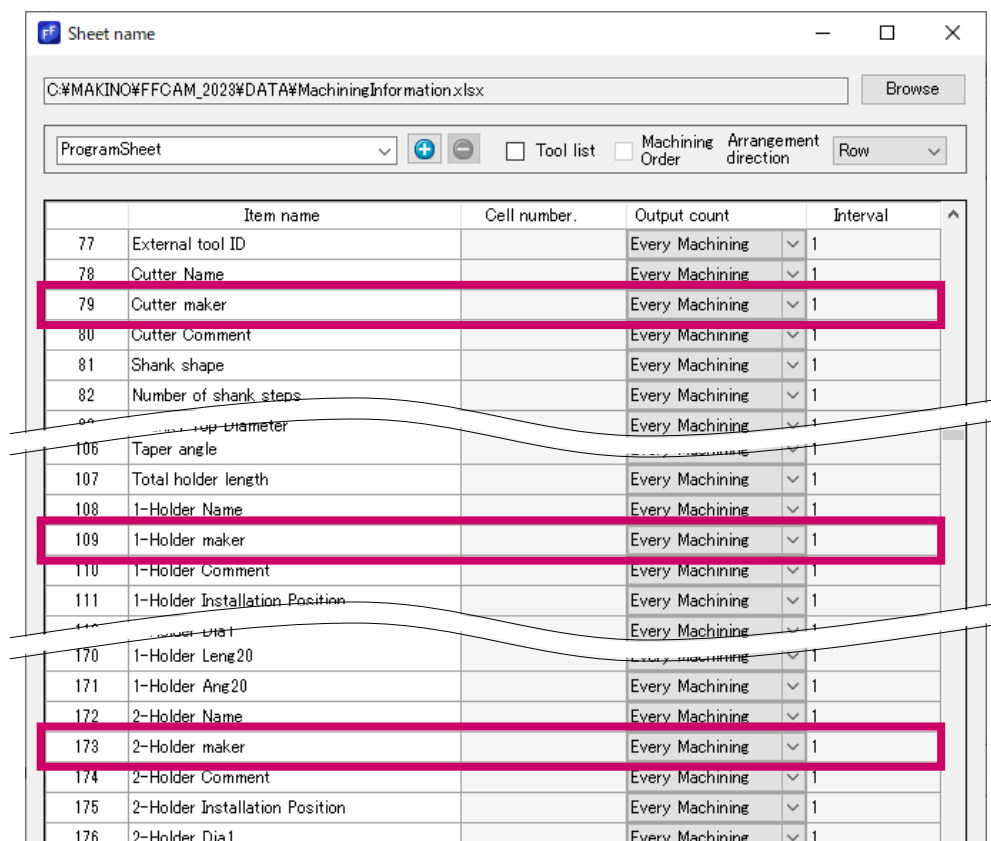
The [Cutter Maker] and [Holder Maker] entered in each [Tool Setting] screen are automatically entered in the Excel machining information.

■ Setting Screen

[Excel machining information settings (Sheet name)] screen

Set [Cutter Maker] and [Holder Maker] as output targets in the [Excel machining information settings (Sheet name)] screen.

When machining information is output during path calculation or in repost information, the "Cutter Maker" and "Holder Maker" that were entered in the [Tool Setting] screen are listed in the Excel machining information.

[Tool Setting] screen

Enter the name of the manufacturer in [Cutter Maker] or [Holder Maker] field.

The entered name of the manufacturer appears in the Excel machining information.

Supplement

- The [Cutter Maker] or [Holder Maker] field can also be checked and entered in the [Tool Setting] screen of [CL Edit] and [Overhang Length Division].

- In the [Tool Setting] screen of [CL Division], the [Cutter Maker] or [Holder Maker] field can only be checked.

9. Addition of the Function to Output the IDs for Tool Presetters to Excel Machining Information

A function has been added to output the IDs for tool presetters for Excel machining information.

The [External Tool ID] that was entered in the [Tool Database (Maintenance)] screen or each [Tool Setting] screen is automatically entered in the Excel machining information as an ID for the tool presetter.

■ Setting Screen

[Excel machining information settings (Sheet name)] screen

Set [External Tool ID] as an output target in the [Excel machining information settings (Sheet name)] screen.

When machining information is output during path calculation or in repost information, the [External Tool ID] that was entered in the [Tool Database (Maintenance)] screen or [Tool Setting] screen is included in the Excel machining information.

[Maintenance - Tool Database] screen

Enter the ID for the tool presetter in the [External Tool ID] field under the [Tool Edit] screen item.

The entered ID is included in the Excel machining information.

[Tool Setting] screen

Enter the ID for the tool presetter in the [External Tool ID] field under [Detail Setting] in the [Tool] section.

The entered ID is included in the Excel machining information.

Supplement

• The [External Tool ID] field can also be checked and entered in the [Tool Setting] screen of [CL Edit] and [Overhang Length Division].

• In the [Tool Setting] screen of [CL Division], the [External Tool ID] field can only be checked.

10. Addition of Tip Comments on the Settings Screen

Tip comments have been added to the [Contour Machining] and [Contour Face Cut Machining] setting screens to increase the amount of information.

Tip comments are also provided on the [Output Plane Only] item of [Contour Face Cut Machining], which has been added in FFCAM2023, and on the [Chamfering Route Machining] setting screen.

■ Screen example

Tip comments explain the function quickly and clearly by using diagrams, etc.

[Contour Machining] setting screen
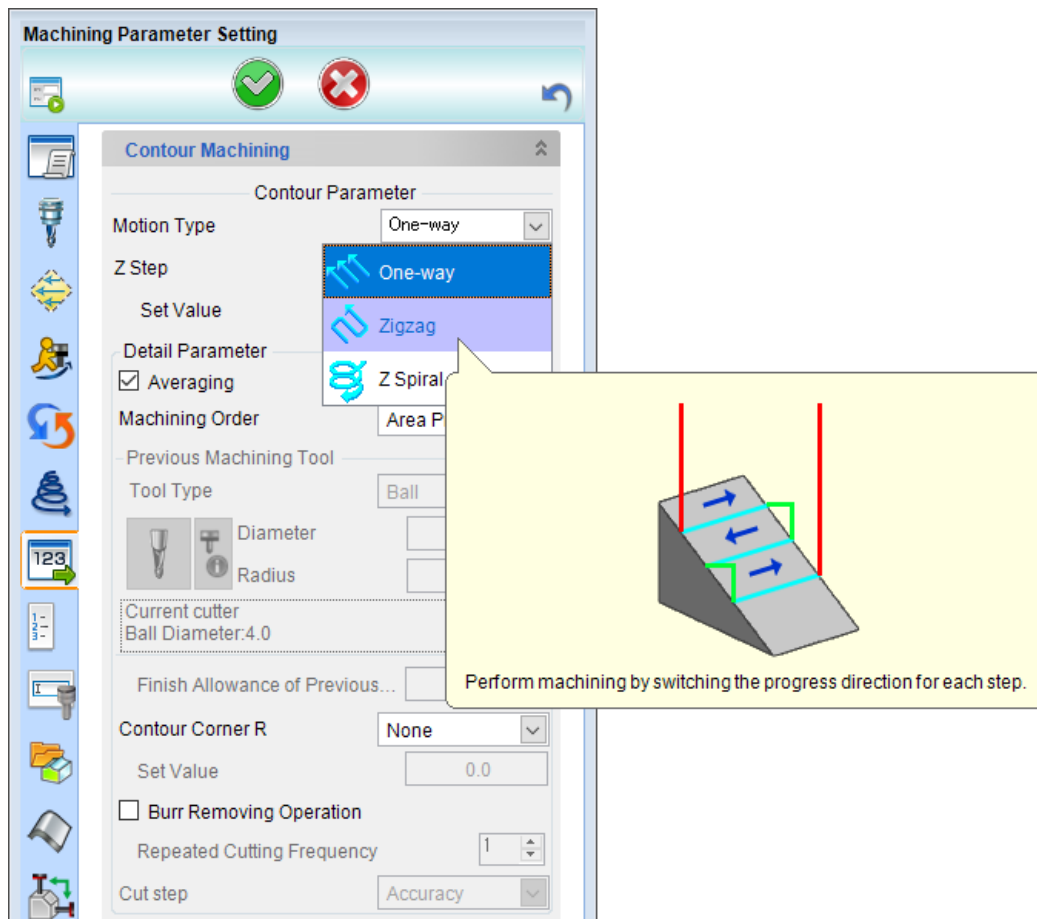
11. Addition of Parameter Macro Functionality (1)

Additional variables that can be defined in parameter macros are now available.

Because of the addition of tool parameter variables, a new selection tab has been provided on the [Custom Macro Edit] screen to make it easier to find tool variables.

■ Setting Screen

A [Tool] tab has been added to the [Custom Macro Edit] screen to consolidate tool variables into a single list.
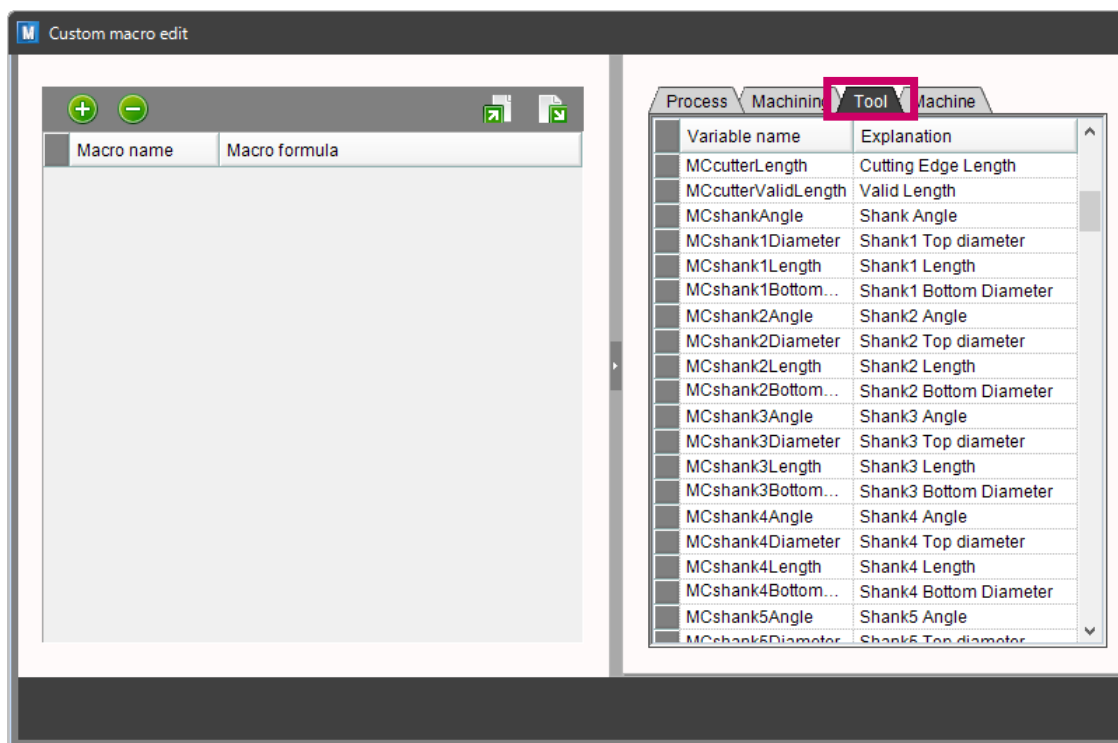
Variable items added in FFCAM2023

<Tool>
- Cutting Edge Length
- Valid Length
- Holder Diameter (1st block x 20 steps, second block x 20 steps)
- Holder Length (1st block x 20 steps, second block x 20 steps)
- Holder Taper Angle (1st block x 20 steps, second block x 20 steps)

12. Addition of Parameter Macro Functionality (2)

Additional items that allow macros to be specified in machining parameters are now available.

The [Area Offset - Set Value] in [Area Setting] and the [Inter-scan Retract Switching Length] in each [Machining Parameter Setting] can now be specified by a macro.

■ Setting Screen

[Macro Edit] screen in [Area Setting]
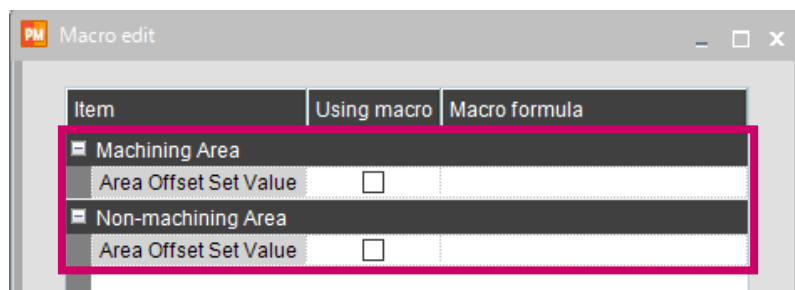
[Macro Edit] screen in [Machining Parameter Setting]

    <Screen example: Corner R Machining (Along-Area)>
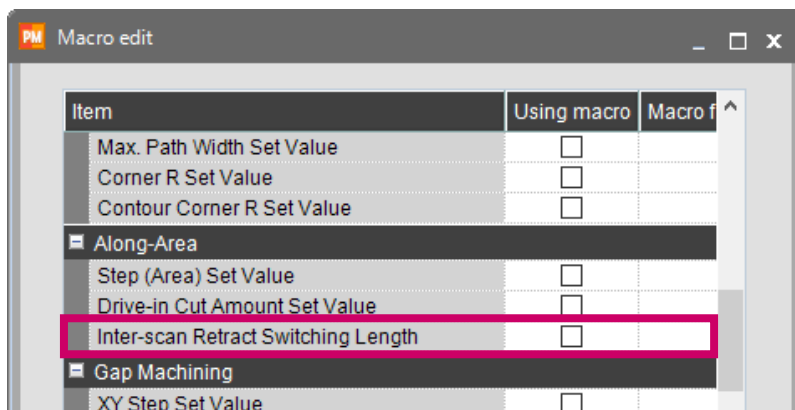
Target of macro designation added in FFCAM2023

Target	Setting Location
Area Offset - Set Value	Area Setting/Machining Area
	Area Setting/Non-Machining Area
Inter-scan Retract Switching Length	Corner R Machining (Along-Area/Contour + Along-Area)
	Along-Surface Machining
	Projection Machining
	Contour Projection Machining (Projection/Contour Projection)
	Route Machining
	3D Equi-Pitch Open Machining
	3D Equi-Pitch Close Machining

13. Addition of Parameter Macro Functionality (3)

Additional functions are now available for use in the parameter macros.

A function can now be created to display any message when a macro error occurs.

■ Description of function

"CUSTOMMSG" can be used as a function to display any message.

<Specifications of function>

CUSTOMMSG (macro expression, conditional expression, output message)

First argument (macro expression)	Specify a macro expression to obtain a value.
Second argument (conditional expression)	Specify a conditional expression that includes the value obtained in the first argument as the variable [VALUE].
Third argument (output message)	Specify the output message.

If the conditional expression of the second argument is False, the result of the macro expression is invalid, and an output message (the third argument) is displayed on the macro expression result screen.
If True, the result of the macro expression is correct, and the value of the macro expression (the first argument) is set to each item. (The message is not displayed.)
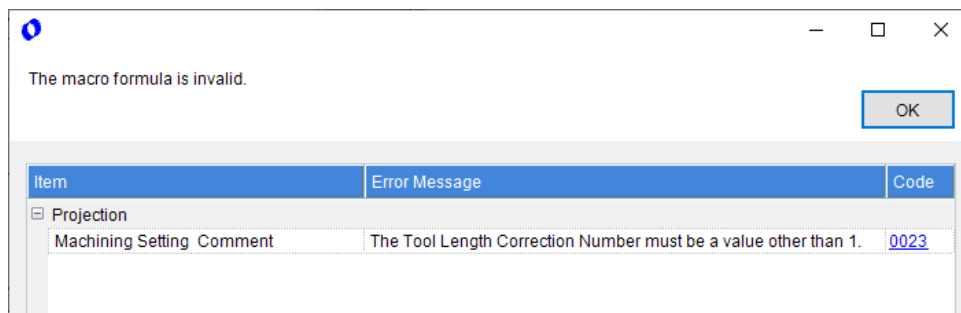
<Example of function setting>

When the Tool Length Correction Number is "1", an error message is displayed.

CUSTOMMSG(MChNo,VALUE<>1,"The Tool Length Correction Number must be a value other than 1.")

This function is executed during path calculation.
If the result is False, a message like the one shown below is displayed, and the calculation is canceled.

If the result is True, no message is displayed, and the calculation is performed as is.

Supplement

For more information on parameter macro functions, see "Machining Parameter Manual" included in FFCAM's Help.

14. [Machine Simulator] Addition of the Function to Change the Display Size of the Menu Icon/Toolbar Icon

A function has been added to change the display size of the menu and toolbar icons on the screen of the Machine Simulator between two different sizes (large and small).
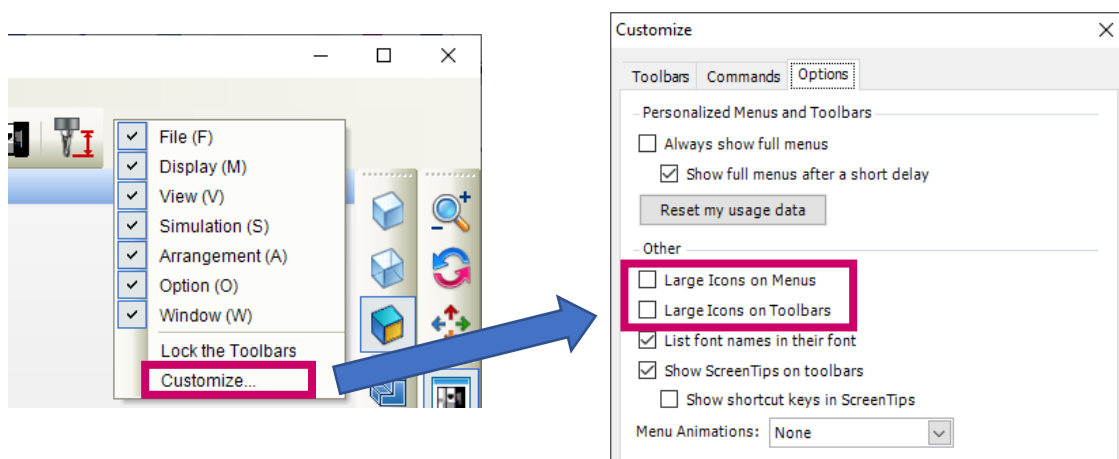
The function that was included in FFCAM has been introduced to the Machine Simulator as well.
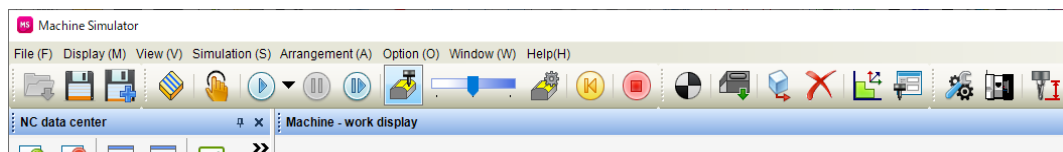
■ Setting Screen

The icon display size can be changed in [Customize].

Select [Customize] from the right-click menu of the menu section.
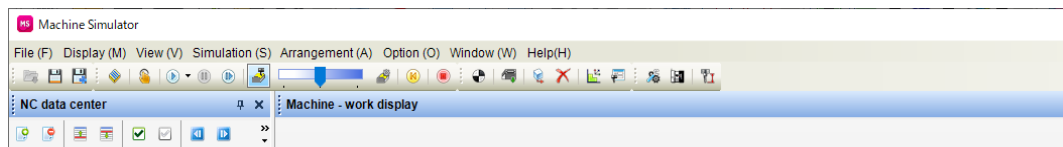
In [Options] tab of [Customize], set the [Large Icons on Menus] and [Large Icons on Toolbars] checkboxes to change the display sizes.

<Menu icons appear large> * Default state

<Menu icons appear small>

15.Support for Windows11

FFCAM2023 supports Windows11 version 22H2.

16.About Graphics Driver Updates

The "DirectX 9" graphics driver, which could be specified in previous FFCAM versions, can no longer be specified in FFCAM2023 due to a system update.
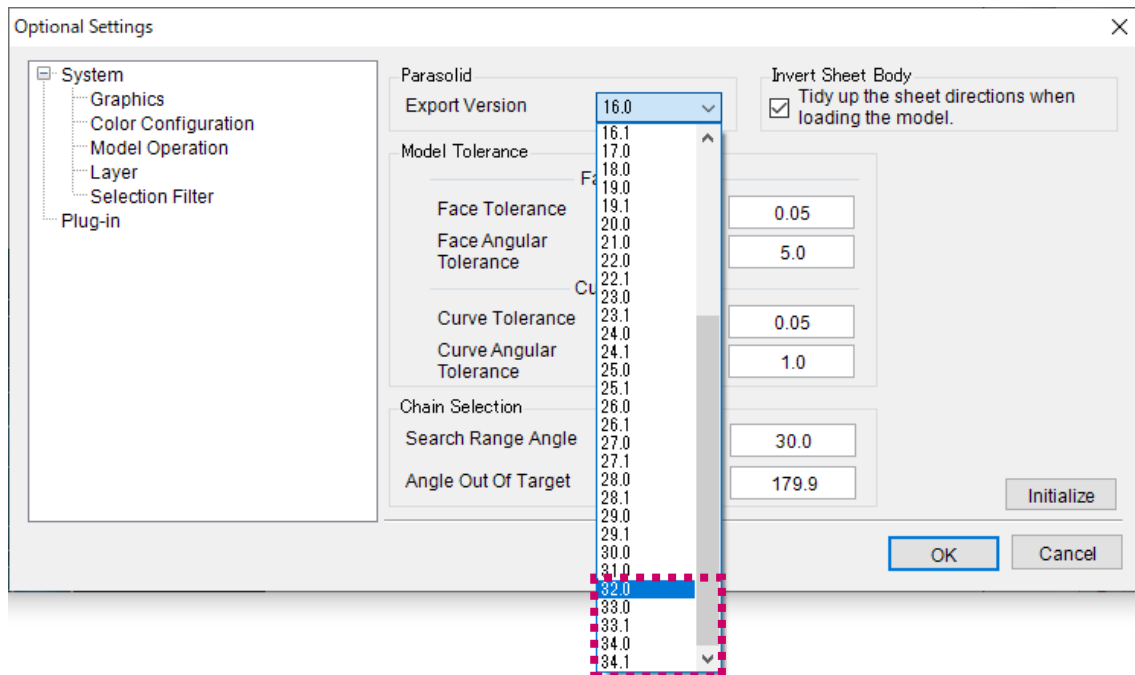
If an FFCAM file from an older version that had "DirectX 9" specified as the graphics driver is loaded in FFCAM2023, "DirectX11" is specified as the graphics driver.
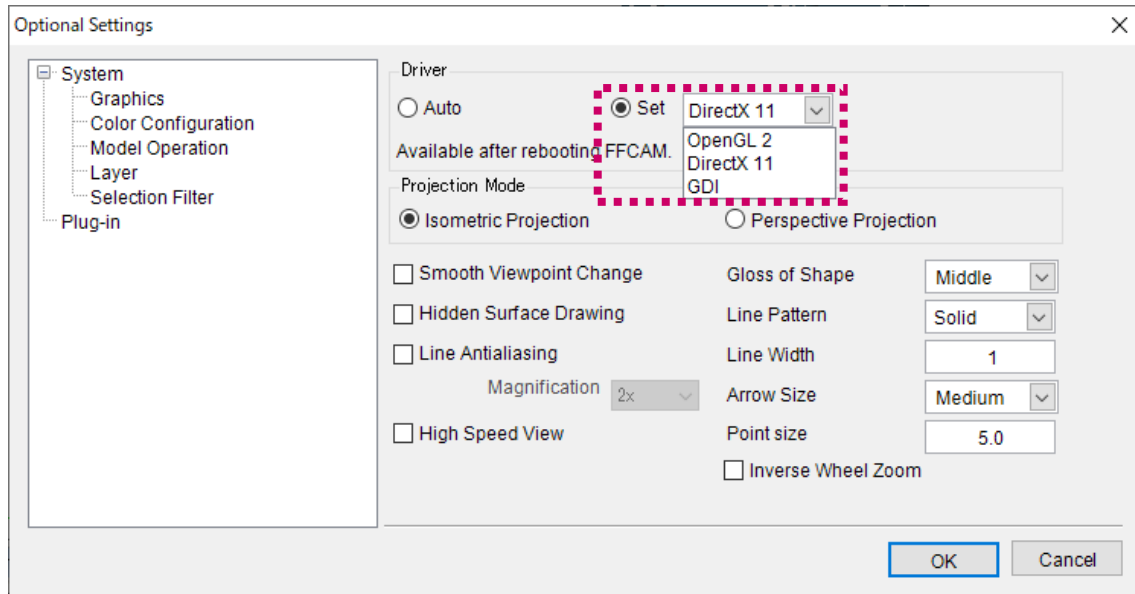
17.Addition of Parasolid Version

The following Parasolid output versions have been added to FFCAM2023.

- 32.0
- 33.0
- 33.1
- 34.0
- 34.1

18. Support for Software and Licenses

In addition to the existing hardware (HASP key) licensing provision, FFCAM2023 adds a software license provision method.

Once a software license is installed, FFCAM can be activated without the possession of a HASP key.

Please consult with us at the time of FFCAM2023 installation to determine which license to use.

The license is set in the [License Setting] application in the [FFCAM2023] installation folder.
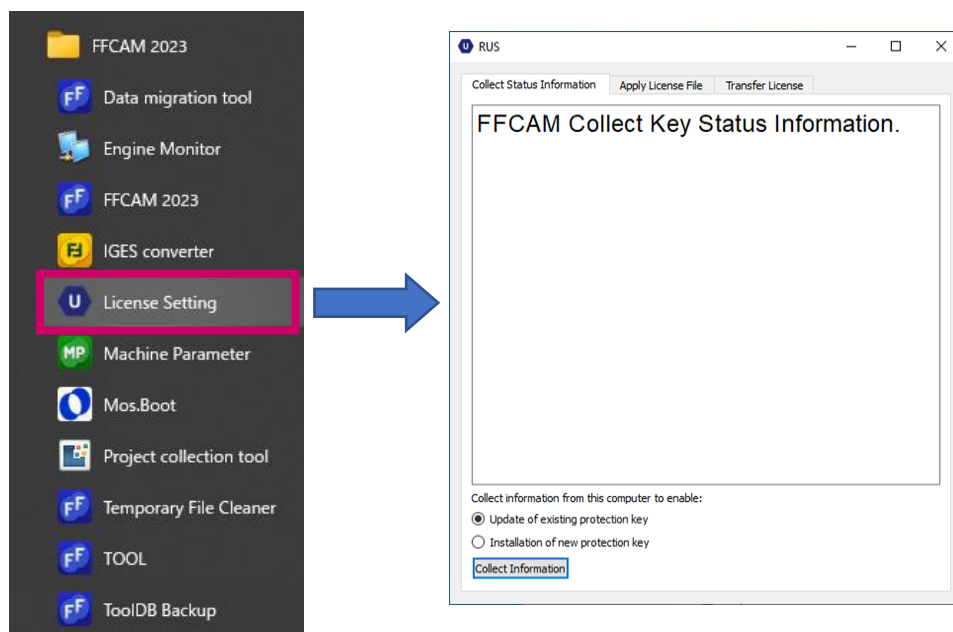
Supplement

- FFCAM will run even if there are multiple licenses for any of the licenses on the network.

- Software licenses are available for FFCAM2023 and later versions.
  Versions from FFCAM2022 are earlier are not supported.