102-NF-20000

FFCAM 2020

Description of New Functions

MAKINO

Preface

This manual describes the functions added to MAKINO FFCAM 2020 and how to use them.

Created on

February 2020

List of Functions Added to FFCAM 2020

1. Addition of Function to Define the Macro for Machining Parameter Setting

Frequently used data for the machining parameter settings was provided through a template. However, since the values set by the template are fixed values, each time, it was necessary to re-enter the values of parameters whose values changed with other settings.

The function to define macro provided in FFCAM2020 automatically changes the value of the parameter whose value changes depending on the situation. This function eliminates the need to re-enter values and prevents missed settings for related parameters.

■ Setting Screen

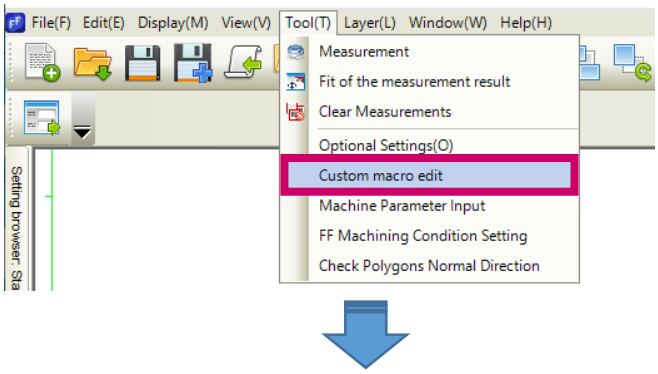
The basic flow to set the function to define a macro is as follows.

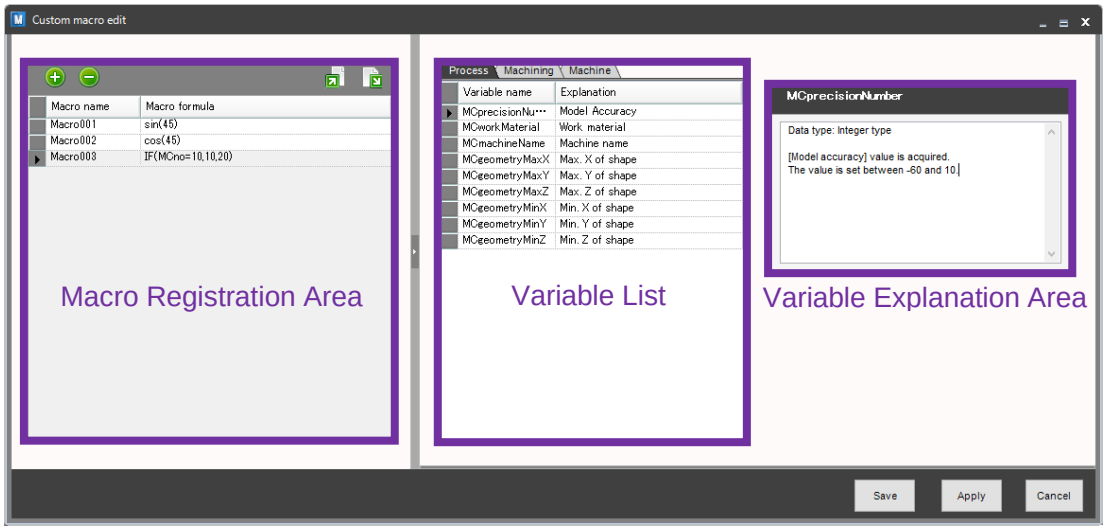
1. On the [Custom macro edit] screen, create a macro and register it in FFCAM.
2. On the [Macro edit] screen, apply the macro to each parameter of machining.

Custom Macro Edit Screen

This screen enables you to create a macro by combining mathematical and conditional formulas, and register it in FFCAM.

To open the edit screen, select [Custom macro edit] from the Tool menu.





(1) Macro Registration Area

Enter mathematical or conditional formulas of the macro to be registered in FFCAM in the list format.

Add Line icon

Adds a line for entering macro formulas to the list.
Clicking the icon adds a line to the last line.

Delete Line icon

Deletes the selected line of macro formulas.

Macro Formula CSV Import icon

Imports macro formulas from a CSV file into the list.
The contents of the list before import are deleted and overwritten by the imported contents.

Macro Formula CSV Export icon

Exports the macro formulas in the list to a CSV file.

Macro name

Enter and set a name for the macro.

Macro formula

Enter a mathematical formula, conditional formula, or variables.

(2) Variable List

This is a variable list of FFCAM parameters that can be used for creating mathematical and conditional formulas in the [Macro Registration Area].
Variables with variable names that begin with "MC" can be used in mathematical and conditional formulas.
You can copy the variable name and paste it into [Macro formula] of the [Macro Registration Area].

Process, Machining and Machine tabs

Lists each variable for process, machining, and machine parameters.

Variable name

Displays the variable names of FFCAM parameter variables.

Explanation

Displays a description of each variable and the name of the corresponding FFCAM parameter.

(3) Variable Explanation Area

Displays a detailed description of the variable selected in [Variable List].

(4) Save

Registers the set macro formula in FFCAM and closes the edit screen.

(5) Apply

Registers the set macro formula in FFCAM and keeps the edit screen open.

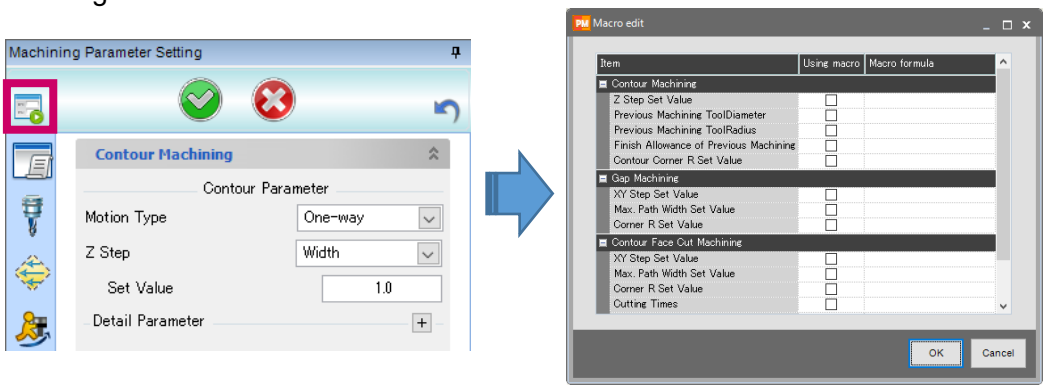
(6) Cancel

Discards the settings and closes the edit screen.

Macro Edit Screen

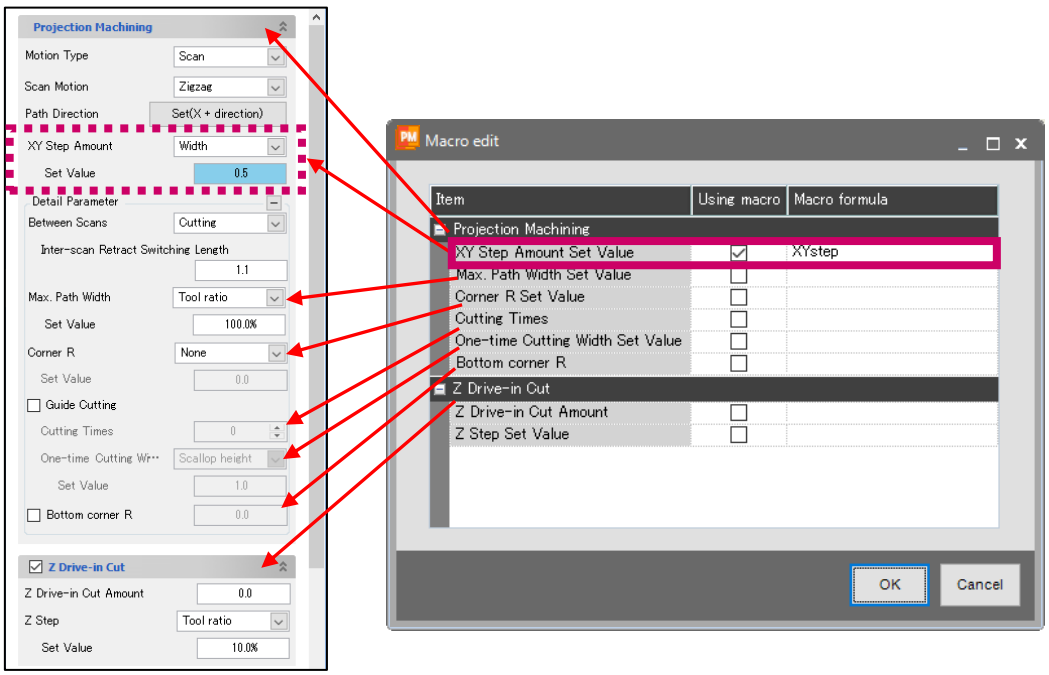
In the [Macro edit] screen, define the macro registered in the [Custom macro edit] screen and apply it to the machining parameters. (Simple mathematical and conditional formulas can be created only in the [Macro edit] screen.) The definition set here is maintained for each working file.

To open the edit screen, click the [Macro edit] icon on Machining Parameters Setting.



Explanation of Screen

Lists the parameters to which the macro can be applied.
In the list, you can select whether to apply the macro and define the macro formula.



(1) Using macro

Set whether a macro is applied to each parameter.

Check: ON

Macro is applied to the corresponding parameter.
In the [Machining Parameter Setting] screen, the background of the parameter entry field to which the macro is applied becomes blue. (Example: [XY Step Amount]/ [Set Value] in the figure on the previous page)
Macros are executed during calculation. When the [Machining Parameter Setting] screen is opened after the calculation, the values after execution of the macro is displayed.

Check: OFF

Macro is not applied to the corresponding parameter.
If a macro or mathematical formula is defined in [Macro formula], it is not applied during the calculation.

(2) Macro formula

Define a macro by entering the macro name, mathematical formula, or conditional formula registered on the [Custom Macro Edit] screen. (There is no limit on the number of characters)

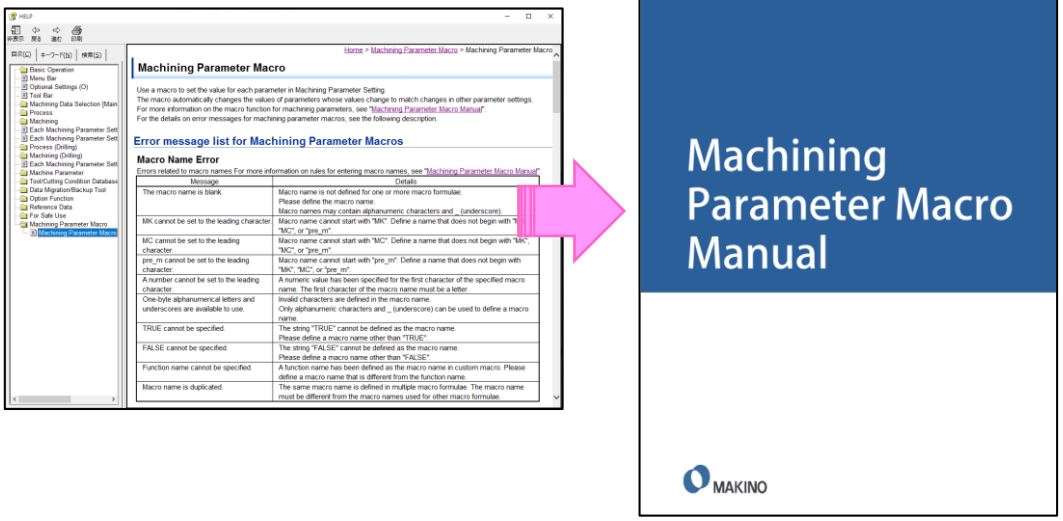
(3) OK

Saves the settings in a working file (xMtn) and closes the edit screen.
Macros applied to each parameter are executed during calculation.

(4) Cancel

Discards the settings and closes the edit screen.

For details of the macro setting function, see "Machining Parameter Macro Manual" in Help of FFCAM.



2. <CR Machining> Addition of Tool Path Output Control Function by Tilt Angle

In the [Contour], [Angle-area] and [Pencil] modes of CR machining, a function to specify the output range of the tool path by the tilt angle of corners has been added.

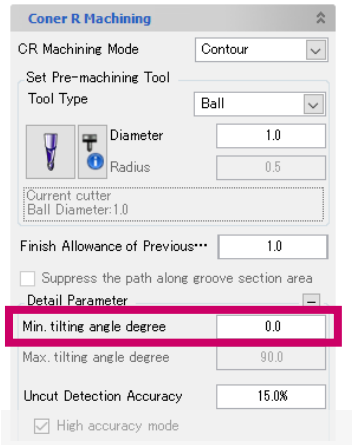
In the previous CR machining, the tool path was output for the entire range specified by [Contour] and [Angle-area], and the tool path was created even for areas where each machining method was not suitable (gentle slope for Contour, steep slope for Along-Area). Therefore, parameters and machining conditions had to be adjusted.

In FFCAM2020, the function allows you to create the tool path only in appropriate areas for each machining mode.

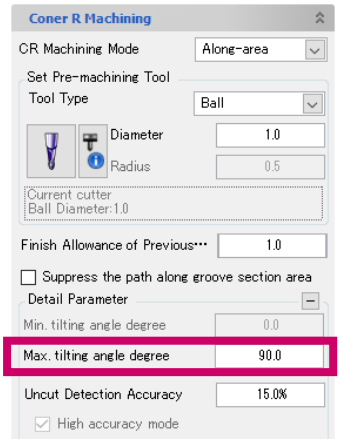
■ Setting Screen

Specifies the area to create a tool path with corner and tilt angle.

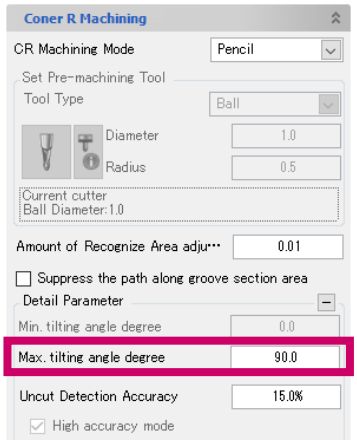
- Contour



- Along-Area



- Pencil



(1) Min. tilting angle degree

Can be set when the [CR Machining Mode] is [Contour].

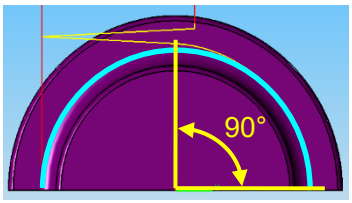
A tool path is created at corners with tilt angle smaller than the specified angle.

Angle can be specified in the range of 0° to 75°.

(2) Max. tilting angle degree

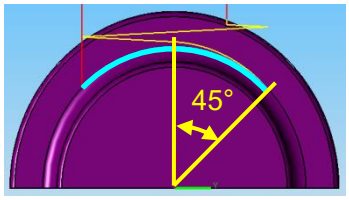
Can be set when [CR Machining Mode] is [Along-area] or [Pencil].
A tool path is created at corners with a tilt angle greater than the specified angle.
Angle can be specified in the range of 15° to 90°.

When Max. tilting angle is set to 90°



0° to 90°

When Max. tilting angle is set to 45°



0° to 45°

3. <Tool Setting> Addition of Function to Set Overhang Length and Neck Length Using Mouse

The [Overhang Length] and [Neck Length] values of the tool settings can be set by dragging the tool shape drawn on the graphics window.

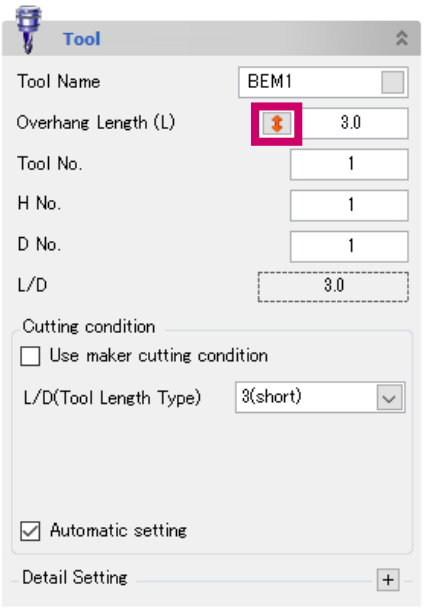
In the past, the tool lengths were changed by numerical input only, and it was necessary to check the length while repeatedly entering numerical values to set the desired tool length.

This function allows you to visually set the tool length by using the tool shape drawn on the graphics window. This visual setting eliminates trial and error in numeric input, allowing quick determination of tool length.

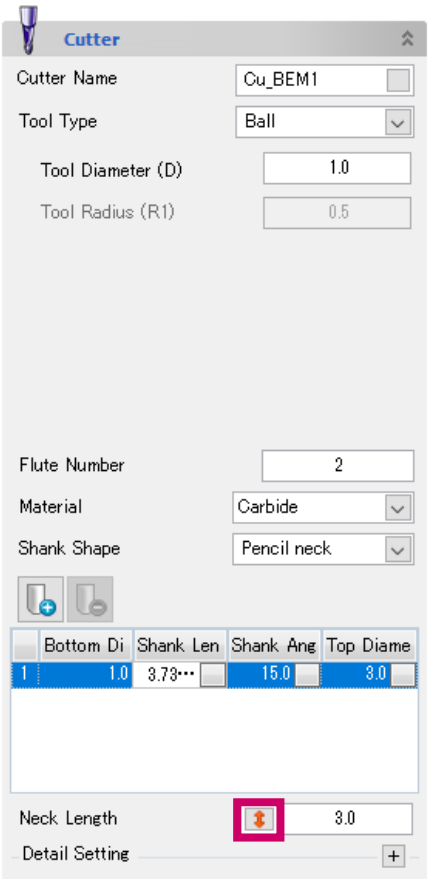
■ Setting Screen

Buttons for activating the function to "set the length by dragging" have been added to the [Overhang Length] of tool parameters and [Neck Length] of cutter parameters.

- Tool parameters



- Cutter parameters



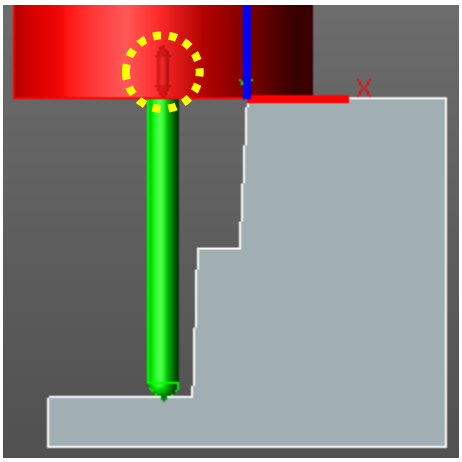
● Operation Procedure

Click the  (length settings) button of the [Overhang Length] parameter or [Neck Length] parameter.

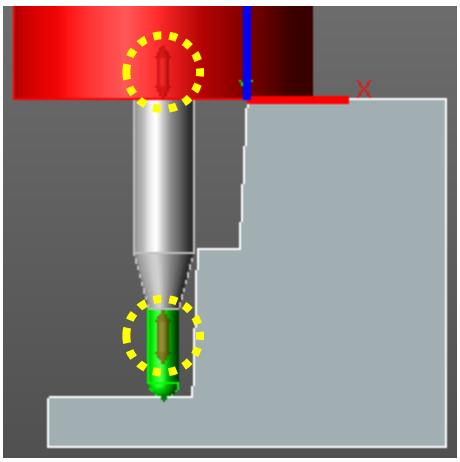
A red arrow to adjust the length appears on the tool shape in the graphics window.
An Overhang Length Confirm dialog box is also displayed.

- Arrow to adjust the length

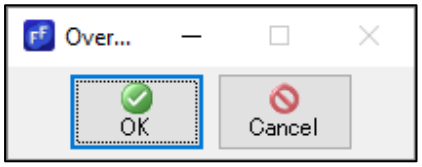
When the shank shape is "Straight", an arrow is displayed at the connection between the cutter and the holder.



When the shank shape is "Pencil neck", arrows are displayed at the connection between the cutter and shank, and the connection between the cutter and holder.

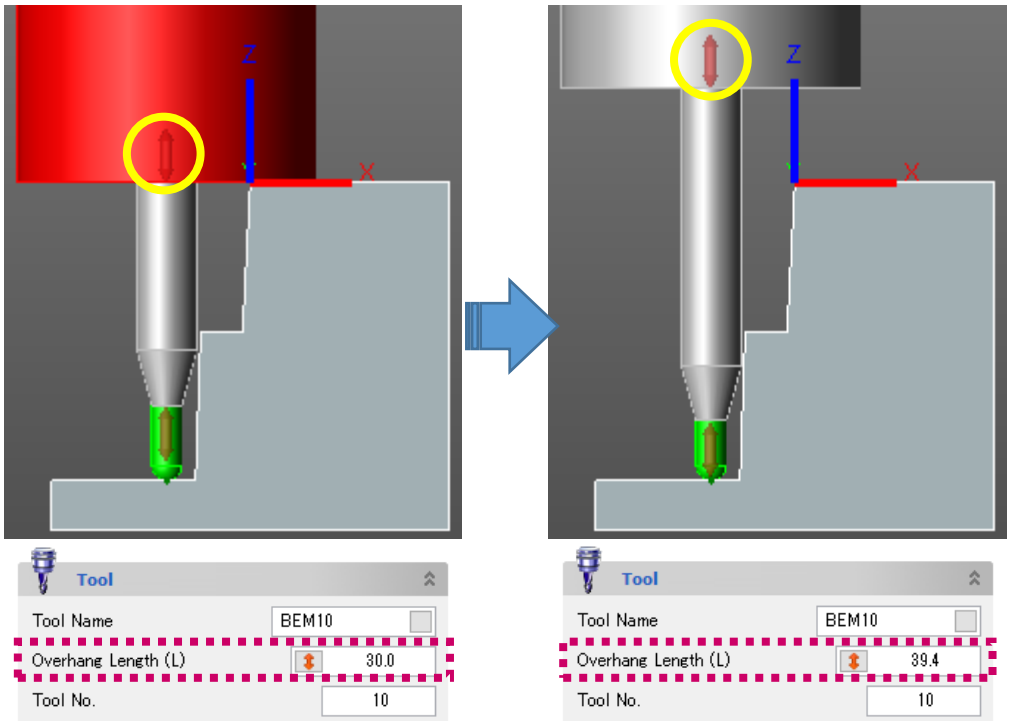


- [Overhang Length Confirm] dialog box

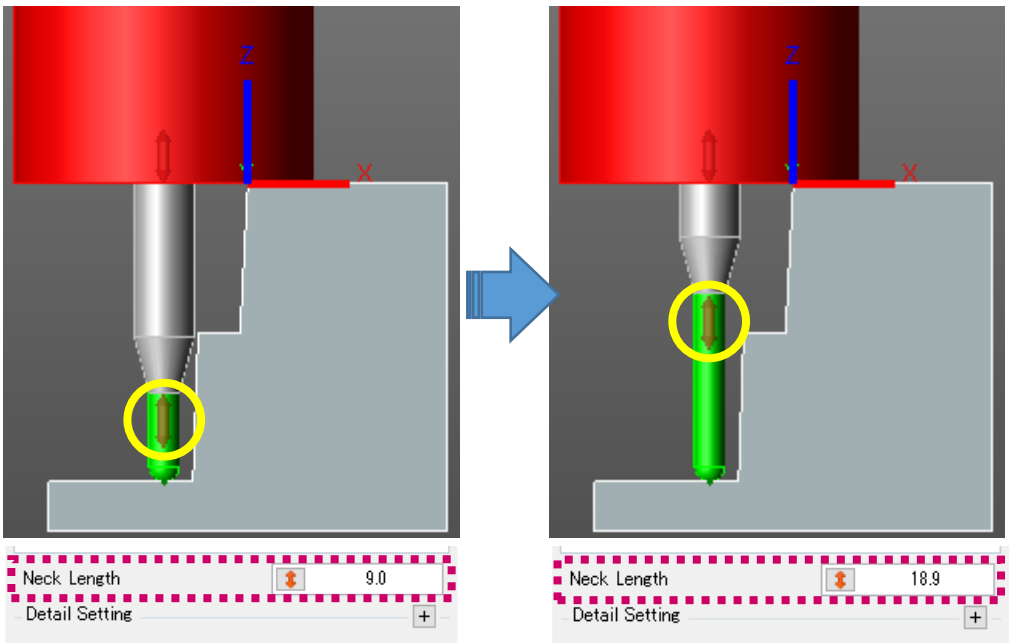


Drag the displayed arrow up and down.
The length of the tool drawn on the screen changes according to the drag operation.
At the same time, the values in [Overhang Length] and [Neck Length] of tool parameters are also changed.

The value of [Overhang Length] is changed when you drag the arrow up and down, at the connection between the tool and holder.



The value of [Neck Length] is changed when you drag the arrow up and down, at the connection between the cutter and shank.



In the [Overhang Length Confirm] dialog box, click [OK] or [Cancel].

OK

The value set by dragging is fixed and saved.

The red arrow disappears.

Cancel

The set value is canceled and reverts to the value before the length was changed by dragging.

The red arrow disappears.

■ Note

- For the numerical values set by this function, the decimals are displayed as follows.
  - For metric display: Rounded off to two decimal places.
  - For inch display: Rounded off to four decimal places.

- When setting the length by dragging, the following restrictions apply:

Item		Overhang Length	Neck length
Upper limit	Upper limit value	No upper limit	Overhang Length
	Operation when the upper limit value is exceeded	None	The value above the upper limit value is not set.
Lower limit	Lower limit value	Straight tool: Cutting Edge Length * "Tool radius" for drilling tools Pencil Neck Tool: Up to the upper edge of the shank taper	Cutting Edge Length * "Tool radius" for drilling tools
	Operation when the lower limit value is exceeded	The value below the upper limit value is not set.	

4. <Tool Drawing> Addition of a Function to Simplify the Interference Check Between the Tool and Machining Geometry

A function to draw the tool in red when there is interference between the tool and machining geometry during tool drawing has been added.

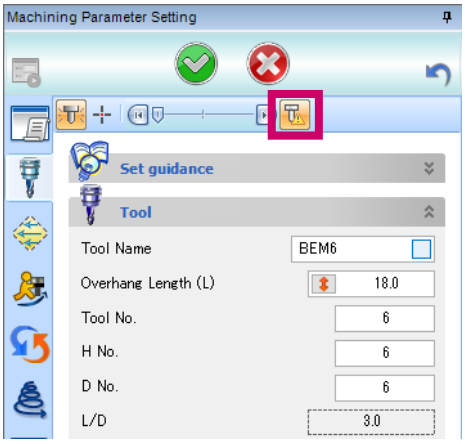
Previously, tool was drawn mainly to examine the tool length, but only changes in the overhang length and neck length were drawn, and there was no way to visually check for interference.

By using this new function when drawing the tool, it is easy to check if there is any interference between the tool and machining geometry and prevents oversight of interference.

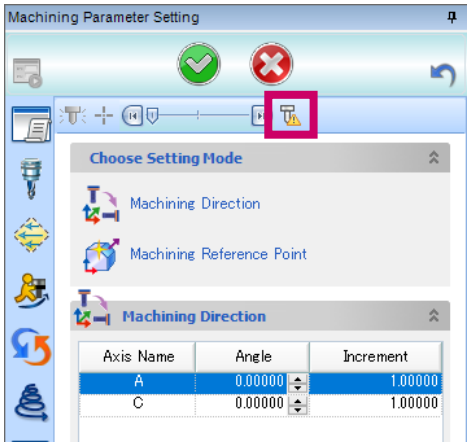
■ Setting Screen

A tool interference check button has been added on the following setting screens.

- Tool Setting



- Machining Direction/Machining Reference Point Setting



* A tool interference check button has also been added to the setting screen for Simultaneous 5-axis machining (option).

When you click the tool interference check button and it is highlighted, the function to check the tool and machining geometry interference is enabled on the tool drawing screen.

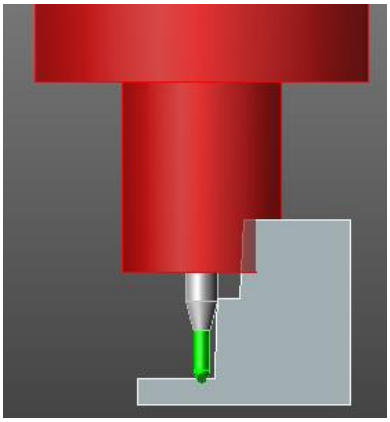
When the interference check function is enabled and interference is detected between the tool and machining geometry, the tool interference location is displayed in semitransparent red.

The interference check function can be enabled when each "Tool display location", "Drag operating setting for overhang length and neck length", or "Semitransparent display" function is used.

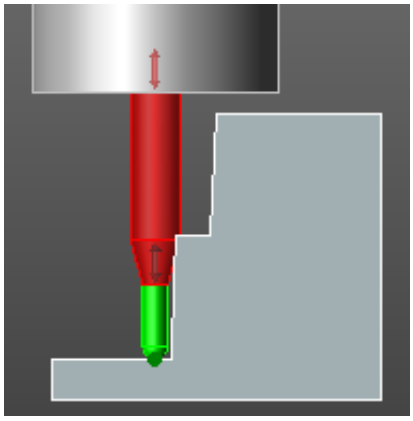
■ Examples of Interference Detection Display

If interference is detected, the tool locations shown in red are holders, shanks, and cutters.

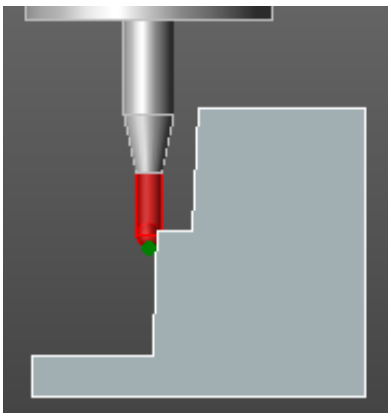
Detected holder interference



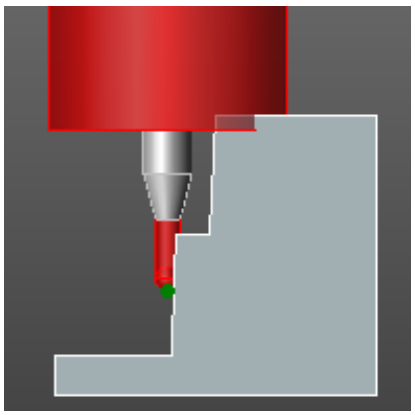
Detected shank interference



Detected cutter interference



Simultaneously detected holder and cutter interferences



■ Note

- Reference check is performed for the models displayed in the graphic window.

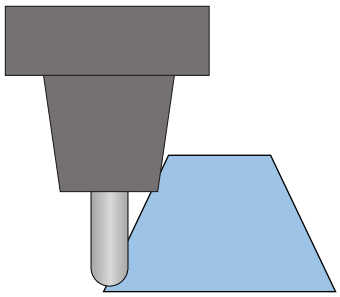
5. <Tool Drawing> Addition of a Function to Draw Tool Connection

The previous tool drawing function was used to draw the tool so that the center of tool tip is located at the specified location.

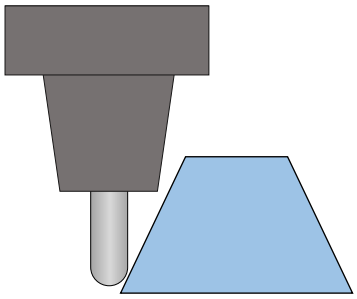
Therefore, the tool and holder display did not match the tool position during machining. When checking for interference, it was necessary to consider the difference between the displayed tool position and actual tool position during machining.

In FFCAM2020, a function is added to draw the tool so that it touches the specified point. Interference of tools and holders can be confirmed more accurately using this function.

- Draw a tool at the center of the tool tip position (Conventional method)

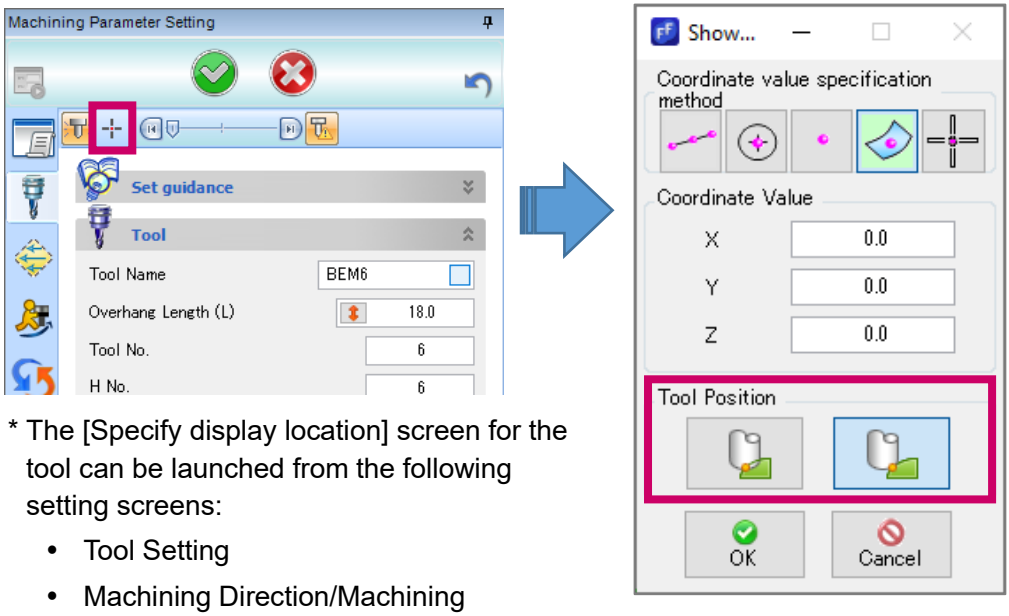


- Draw a tool so that it contacts the workpiece (New function)



■ Setting Screen

In the [Specify display location] screen, a [Tool Position] parameter to specify how to position the tool to be drawn has been added.



* The [Specify display location] screen for the tool can be launched from the following setting screens:
  - Tool Setting
  - Machining Direction/Machining Reference Point
  - Simultaneous 5-axis machining (option)

Tool Position

Select the position to draw the tool. You can select the following two types of positioning methods.

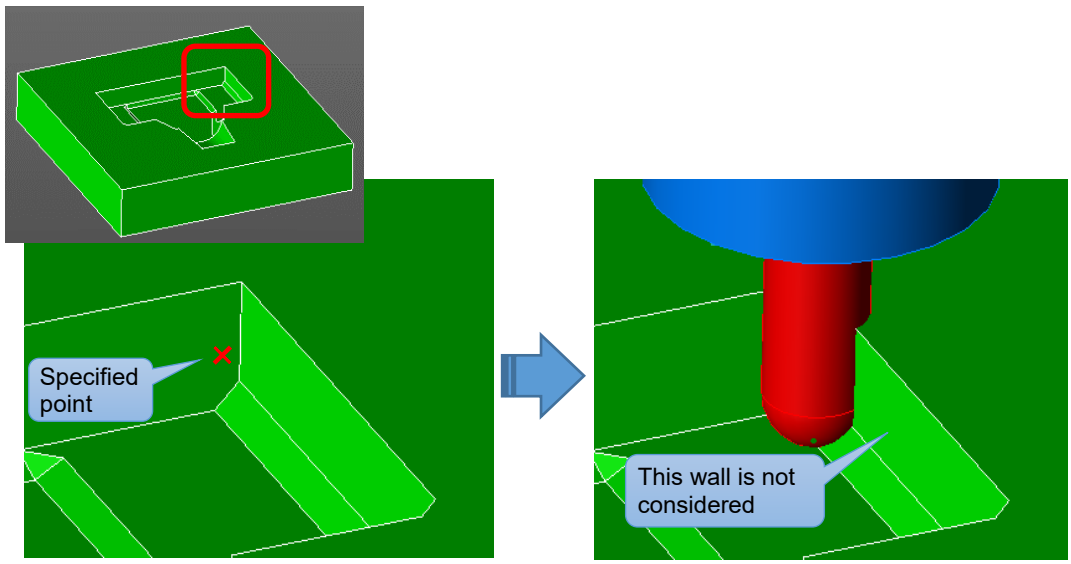
- **Tip button**
  Displays the tool at the position where the specified point matches with the center of the tool tip.
- **Contact Point button**
  Displays the tool at the position where it touches the specified point.

■ Note

- The [Tool Position] parameter can be used when [Coordinate on the face] is selected in the [Coordinate value specification method].
- The table below shows the default values for the [Specify display location] screen of the tool.

	FFCAM 2019.0	FFCAM 2020.0
Coordinate value specification method	Anchor Point/Middle Point	Face
Tool Position	- (Tip)	Contact point

- For "contact point", the tool is positioned so that the tool touches the specified point. Geometries other than the specified point are not considered. (See the figure below)



6. <Simulation/Repost Screen> Addition of Machine Name Display Function to the Start Process

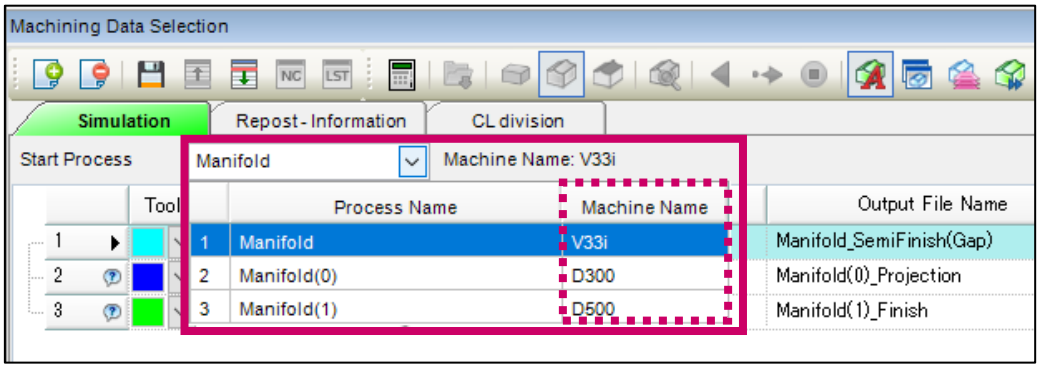
Perform repost processing after selecting the machine to execute the post process in [Start Process] on the simulation repost screen.

Previously, only the process name was displayed in the [Start Process] list, and it was not possible to select a process by the machine name. Therefore, it was necessary to know in advance the machine registered to the process before the setting.

FFCAM2020 makes it easier to select a machine in [Start Process] by displaying the machine name in the [Start Process] list.

■ Setting Screen

The machine names are listed in the [Start Process] drop-down list.



7. <Report Information Screen> Addition of Main Program Creation Function

NC programs created by FFCAM may be used as sub programs.
In this case, the main program had to be created on the screen of the machine or in a text editor.

In FFCAM2020, the function to easily create a main program on the [Report Information] screen has been added.

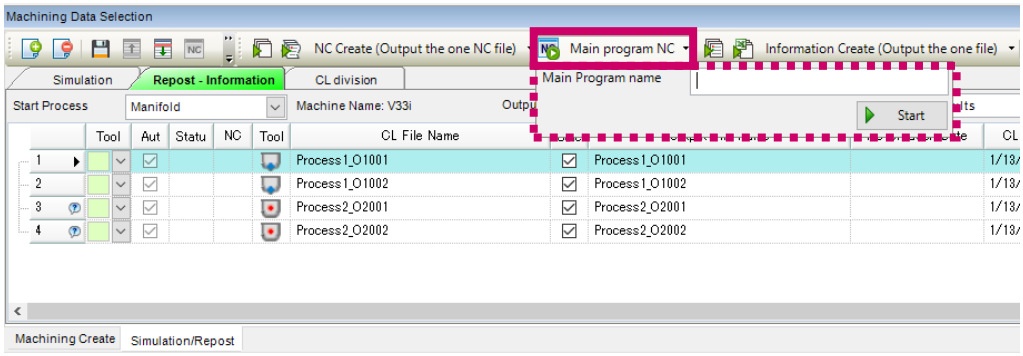
To customize the main program, contact us or your local sales representative. If you place an order for customization of the main program, the following [Main program NC] menu can be displayed.

■ Setting Screen

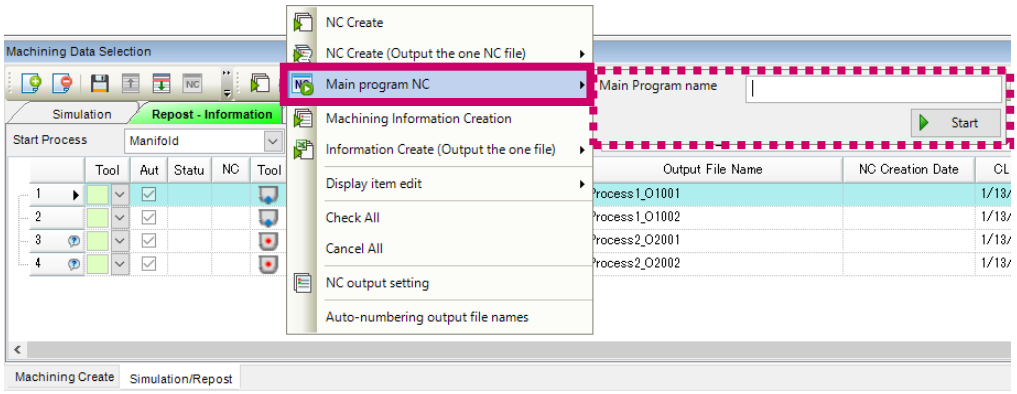
The main program creation program starts on the [Report Information] screen.

You can use the following two types of start methods:

① Click the [Main program NC] button on the tool bar.

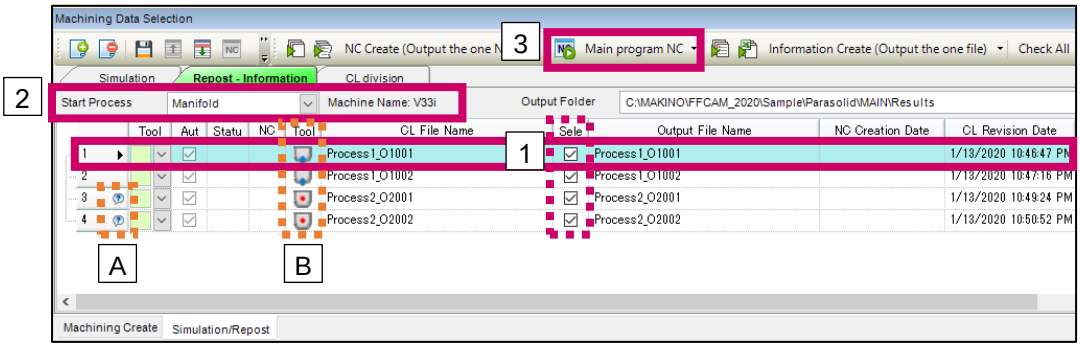


② Right-click on the screen and select [Main program NC] from the pop-up menu.

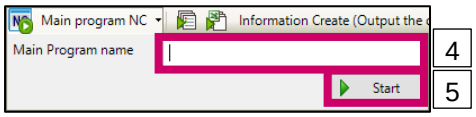


Operation Procedure

Repost Information Screen



Main Program Creation Function - Startup Screen



1. Select the CL file.
   You can create a main program for CL data for which the check box to select CL is checked.
2. Select the start process.
   The main program is created in the "Conversion Table" of the machine of the process set in [Start Process].
   See "Conversion Table Setting" below for more information on the "Conversion Table".
3. Start the main program creation function.
4. Enter the main program name (file name).
   Enter the name to be used in the machine and the program name to be set at the beginning of the NC program. There are no restrictions on the input values.
5. Click [Start] to output the main program file.
   The output file name is "*main program name*.extension".
   The file is output to the folder specified in [Output Folder].

<Supplementary Notes>

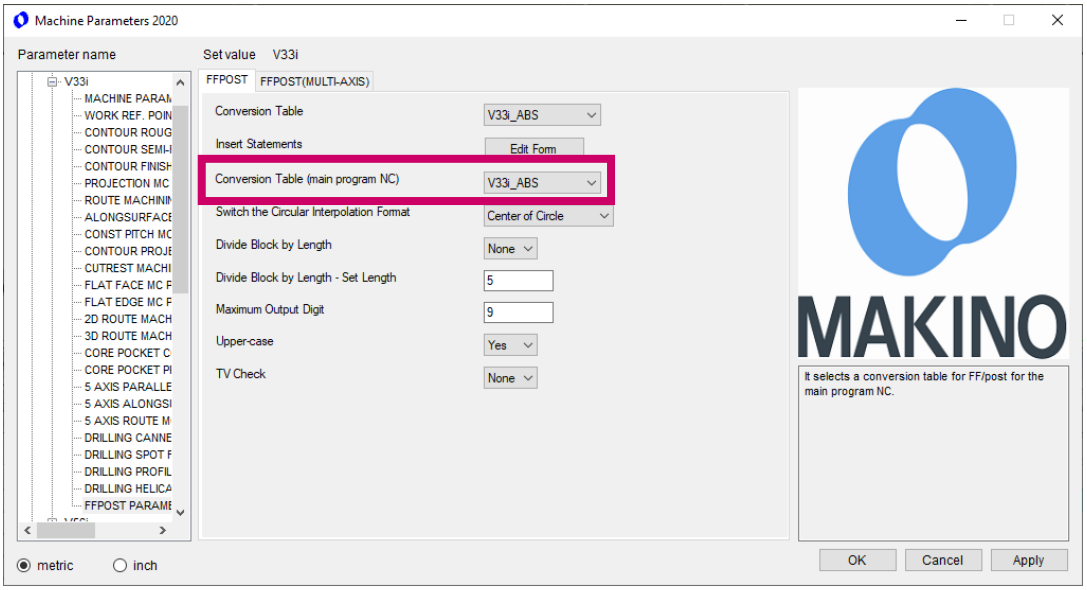
A) The main program can be created even if the machine in [Start Process] and the machine during CL creation are different.

B) The main program can be created even if "Tool Nose" and "Tool Center" are mixed at the NC output position.

■ Settings for Conversion Table

A conversion table must be set up before creating the main program.

Select the post file used to create the main program from [Machine parameters] -> [FFPOST Parameters] -> [FFPOST] tab -> [Conversion Table (main program NC)].



■ Note

Please contact us or your local sales representative to create the post file.

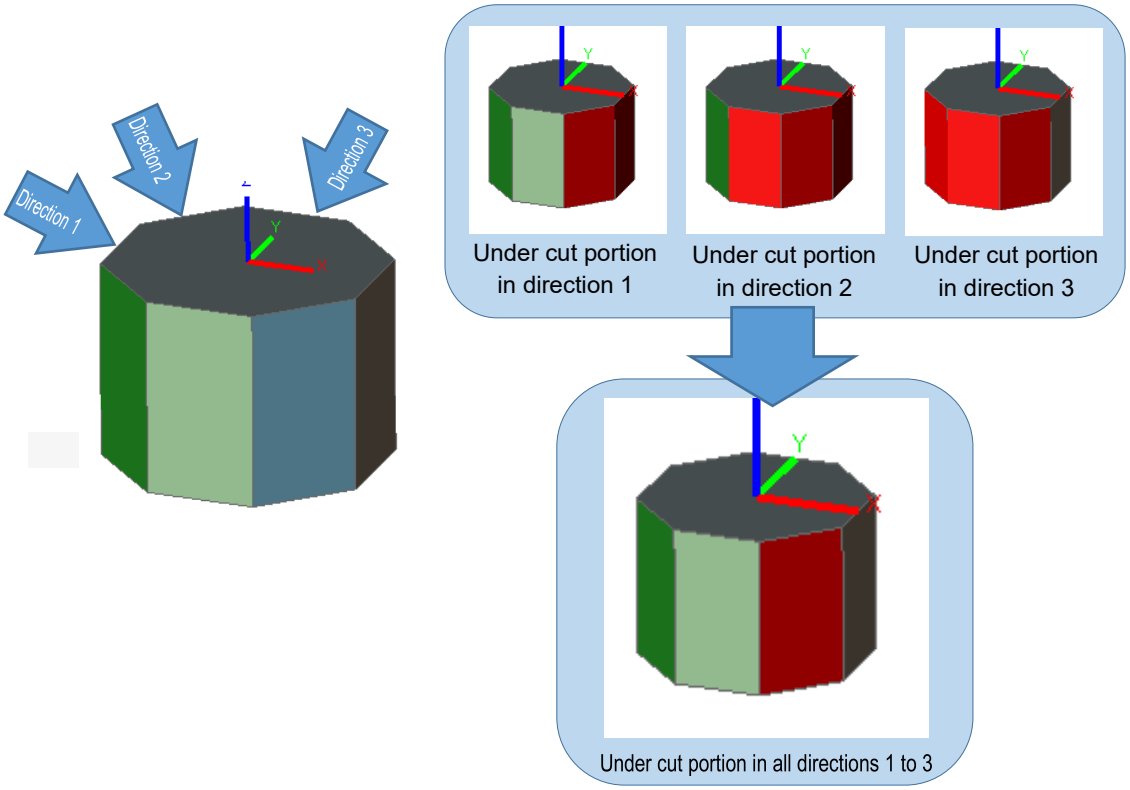
8. Addition of the Under Cut Function for Multiple Machining

A function to display under cut portions of multiple machining and process (multiple processes) together has been added.
Previously, the under cut portions could only be displayed one for each machining at a time. Even for a geometry that requires machining from multiple directions, it was necessary to check under cut portions by displaying them for each machining and examine them comprehensively.
If the number of machining operations is large or the geometry is complex, some areas which cannot be machined (under cut portions) may be missed.

With FFCAM2020, it is now possible to check under cut positions for multiple machining operations at once, making it easier to find areas that cannot be machined.

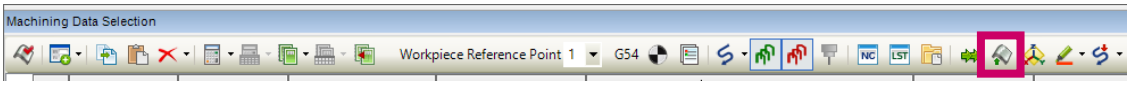
■ Function Explanation

When executing display of under cut portions, if you select multiple machining or process (multiple processes), the under cut portions are displayed for the entire selected machining.



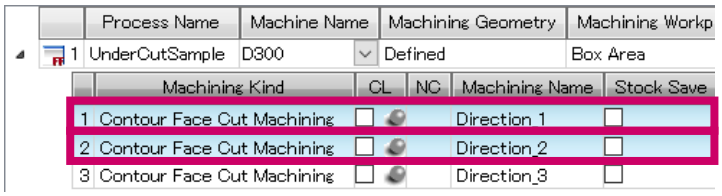
■ Setting Screen

Click the [Under Cut] icon to execute display of under cut portions. Select multiple machining or processes (multiple processes).

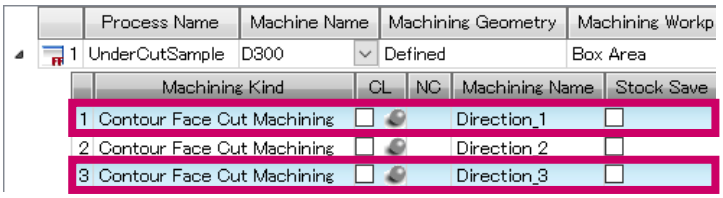


- **Multiple Machining Selection**
  Displays the under cut portions for all the machining that has been selected.
  The following selection methods are supported.

  Select multiple continuous machining



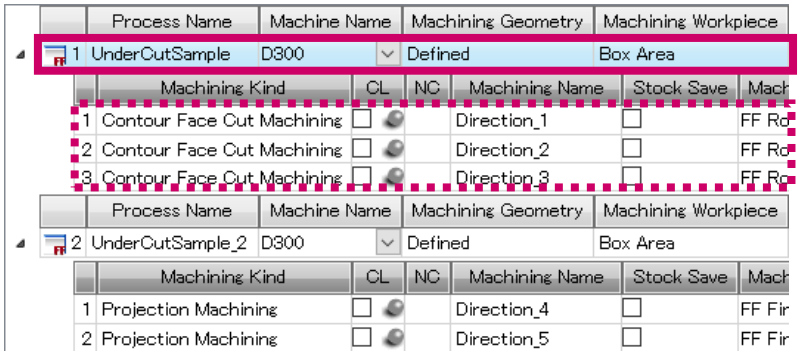
  Select multiple discontinuous machining



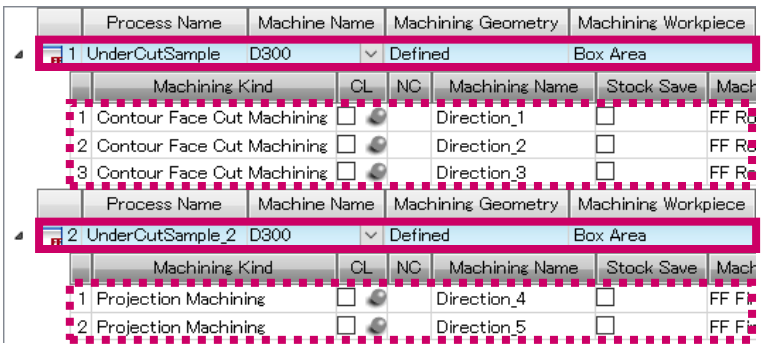
  * You cannot select machining from different processes.
  * For drilling, the "Hole Information" field is available for selection.

- **Process Selection**
  Displays the under cut portions for all the machining in the selected process.
  The following selection methods are supported.

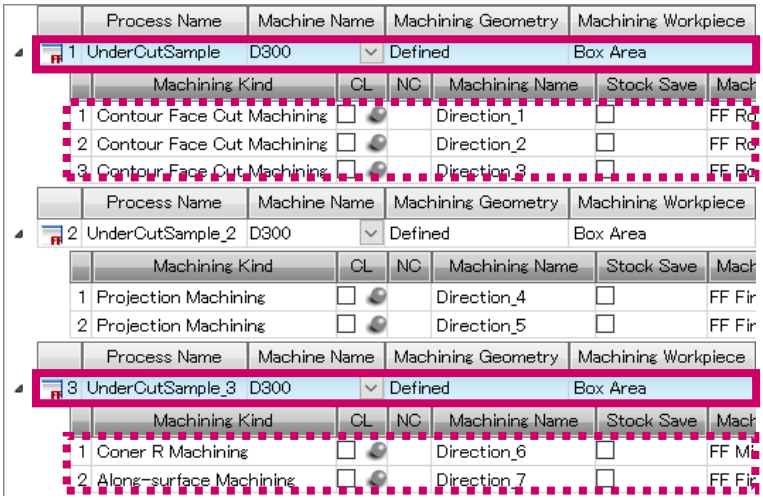
  Select single process



Select multiple continuous processes



Select multiple discontinuous processes



■ Note

Under cut is applied to the models displayed in the graphics window.

9. <Along-Section Machining> Improved/Enhanced Scallop Height Constant Motion Function

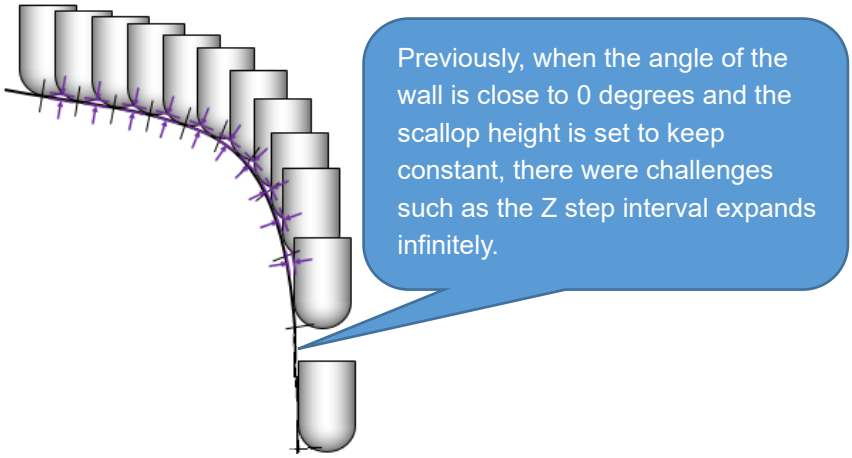
The [Scallop height constant motion] function in Along-Section setting for contour machining has been improved.

Previously, [Max. Z Step] parameter of [Scallop height constant motion] did not allow detailed setting, and it was sometimes difficult to output the tool path suitable for the walls of the geometry.

In FFCAM2020, the setting method of the [Max. Z Step] parameter has been reviewed and improved so that the tool path suitable for the walls of the geometry is output.

In addition, the [Scallop height constant motion] function has been added to the Along-Section setting of corner R machining and contour projection machining.

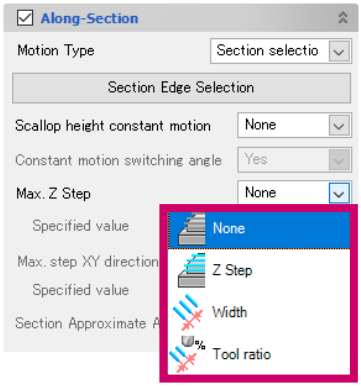
- **Scallop height constant motion**
  This function outputs a tool path with a constant scallop height for Along-Section.
  The gaps between tool paths are not uniform, depending on the tilt angle of Along-Section.



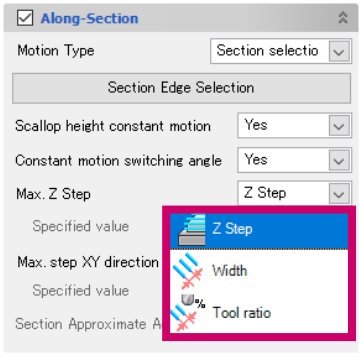
■ Setting Screen

Along-Section Setting

When [Scallop height constant motion] is "None"



When [Scallop height constant motion] is "Yes"



● Improved [Max. Z Step] Parameter

The following items have been modified in [Max. Z Step] parameter in [Scallop height constant motion].

- "Width" and "Tool ratio" are added to the [Max. Z Step] set value for the Along-Section setting.
- "Yes" of the set value of [Maximum Z Step] is changed to "Z Step".
- When [Scallop height constant motion] is set to "Yes", the set value of "None" for [Max. Z Step] is hidden.

● Improved Operation When [Contour Face Cut Machining] Is Set

The [Scallop height constant motion] function is now available when [Contour Face Cut Machining] is "On".

Table for [Max. Z Step] Setting Menu

Set values for each parameter			**Setting menu for Max. Z Step** (O: Show -: Hide)	
Contour Face Cut Machining	Motion Type	Scallop height constant motion	[None]	[Z Step] [Width] [Tool ratio]
OFF	Section selection	None	○	○
		Yes	-	○
	Auto	None	○	○
		Yes	-	○
ON	Section selection	None	○	○
		Yes	-	○
	Auto	None	○	○
		Yes	-	○

(1) Max. Z Step

Specifies the maximum infeed in the Z direction for the tool path created by the Along-Section function.

- None

  Max. Z Step is not specified.

- Z Step

  Makes the maximum infeed amount in the Z direction equal to the Z Step of contour.

- Width

  Enter the maximum infeed amount in the Z direction directly.
  Enter a value in [Specified value]. (Units: mm/inch)
  Enter a value greater than or equal to the Z Step of contour. (Z step of contour ≦ Max. Z Step)

- Tool ratio

  Enter the maximum infeed amount in the Z direction as a percentage of the tool diameter.
  Enter a value in [Specified value]. (Units: %)
  Enter a value greater than or equal to the Z Step of contour. (Z step of contour ≦ Max. Z Step)

(2) Specified Value

Specify the value for the maximum Z Step.
Enter a value when [Width] or [Tool ratio] is specified in [Max. Z Step]. (Specified value ≧ Z Step of contour)

Data in the Old Version

If an old version of the xMtn file is read, the parameter settings are changed as shown in the following table. (■: Setting value to be changed)

		Setting value for FFCAM 2019		**Setting value for FFCAM 2020**	
Contour Face Cut Machining	Motion Type	Scallop height constant motion	Max. Z Step	Scallop height constant motion	Max. Z Step
None	Section selection	None	None	None	None
			Yes		Z Step
		Yes	None	Yes	Z Step
			Yes		Z Step
	Auto	-	-	None	None
					None
		Yes	None	Yes	Z Step
			Yes		Z Step
Yes	Section selection	-	None	None	None
			Yes		Z Step
	Auto	-	-	None	None

■ Addition of Function

● Extended [Scallop Height Constant Motion] Function for Other Machining

The [Scallop Height Constant Motion] function has been added to the Along-Section setting in the following machining.

- Corner R Contour Machining
- Corner R contour + Along-area machining (contour motion part)
- Contour projection machining (contour motion part)

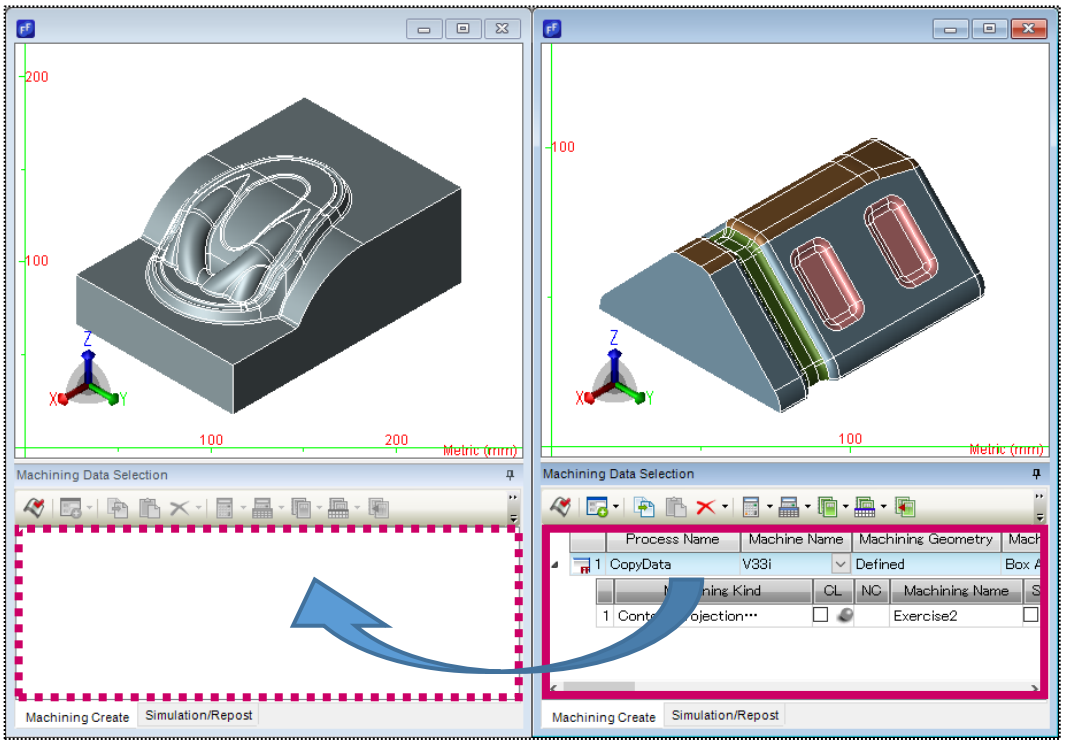
10. Improved Function to Copy Process and Machining Data Between Working Windows

When process data or machining data is copied from one working window to another, the destination window of [Machining Data Selection] had to have at least one process data.

In FFCAM2020, you can copy process and machining data even when the [Machining Data Selection] in the destination window is empty.

■ Operation Screen

Process and machining data can be copied regardless of the presence process data in [Machining Data Selection] of the destination window.



11. Addition of Multiple Setting Value Copy Function on the Machining Data Selection Screen

The set values on the machining data setting screen can be copied simultaneously for multiple machining data.

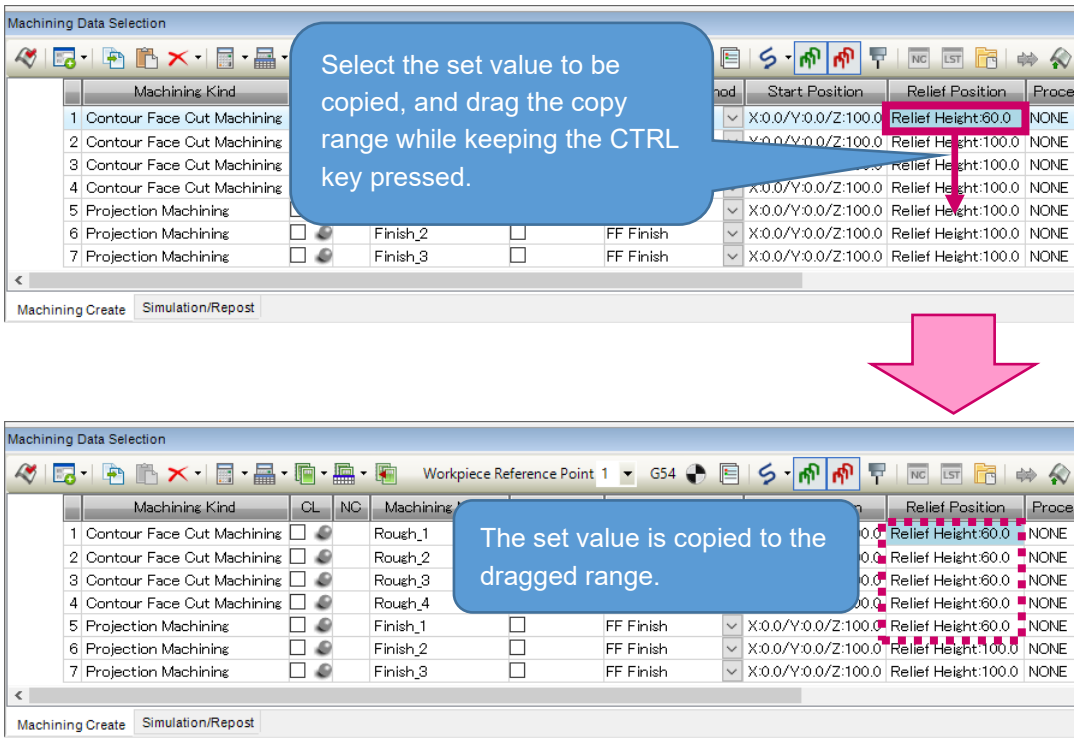
The set values of each machining on the machining data setting screen had to be copied one by one.

For machining in the process (rough, semi-finish and finish), the relief position, finishing, tolerance, and others are often set to the same value.

To streamline the setting task in this situation, FFCAM2020 now supports multiple copies of set values.

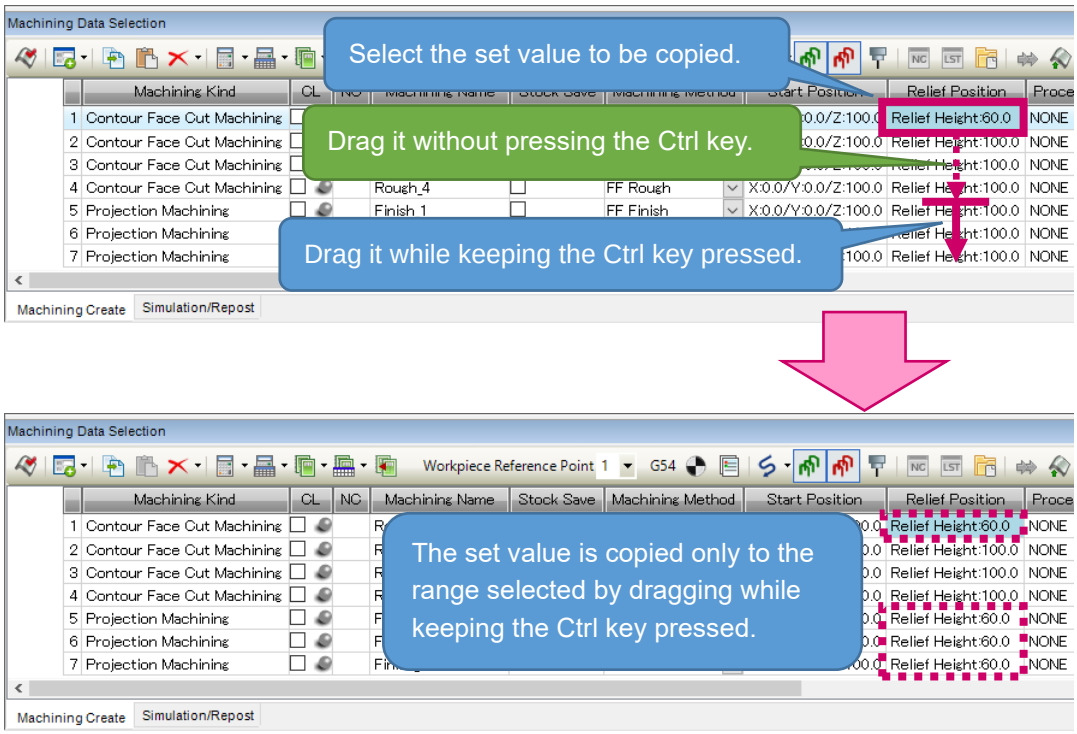
■ Operation Screen

Select a cell for the set value, and drag it while keeping the Ctrl key pressed to copy it to other cells at the same time.



Discontinuous copying is also possible. The value is copied to the cell selected by dragging while keeping the Ctrl key pressed.

Example of discontinuous copy operation



■ Supplementary Note

Copying is possible in both upward or downward directions.

12. Improved Drawing Size of Point Objects

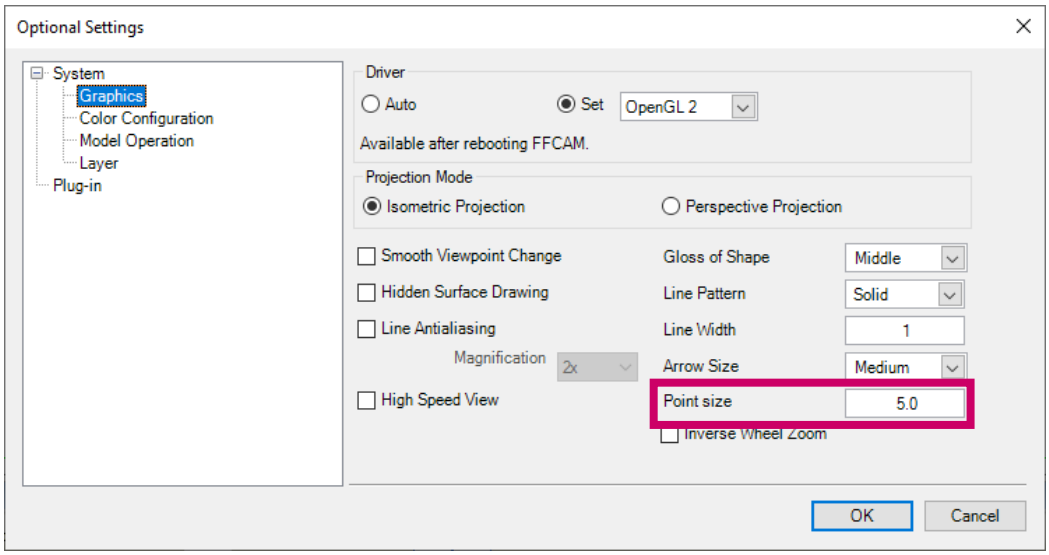
Previously, point objects in FFCAM were drawn at a fixed size on the screen.

Depending on the resolution of the PC monitor and setting of the graphics area, the drawing size of point objects becomes small and sometimes cannot be seen on the screen.

In FFCAM2020, you can adjust the drawing size of point objects to make it easier for the user to see the point objects.

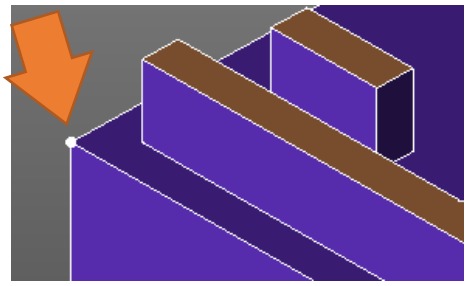
■ Setting Screen

The drawing size of point objects can be set from the [Optional Settings] screen -> [System] -> [Graphics] -> [Point size].

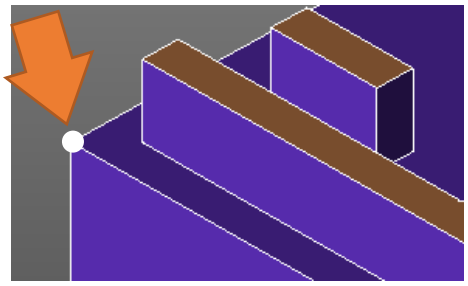


Point size

Sets the drawing size of the point object in the range between 0.1 to 20.0. (Default value: 5.0)



Default size of point object



Larger point object

■ Note

You cannot specify the drawing size when selecting or creating a point. (Will be drawn with default values)

13. Improved STL Geometry Import Method

STL geometry files now can be imported from the [Model Import] menu.

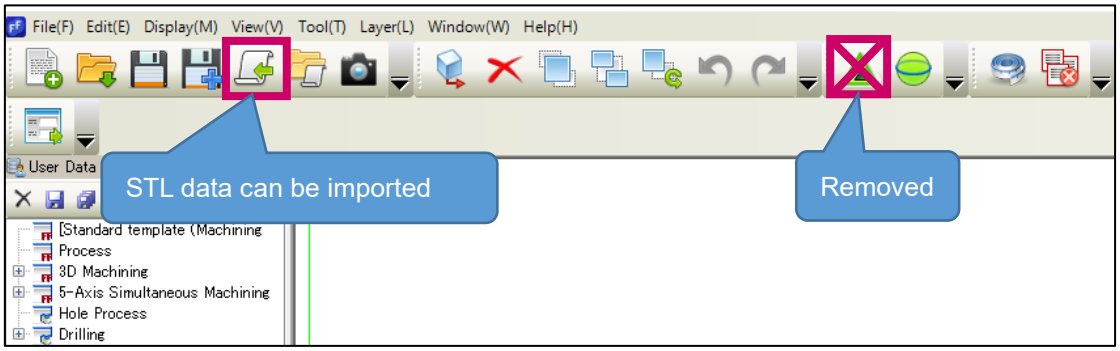
Previously, to import STL geometries into FFCAM, you had to import from a menu that was different from the menu for other geometries in other formats.

In FFCAM2020, an improvement has been made so that even the STL geometries can be imported from the same menu as other geometries.

■ Operation Screen

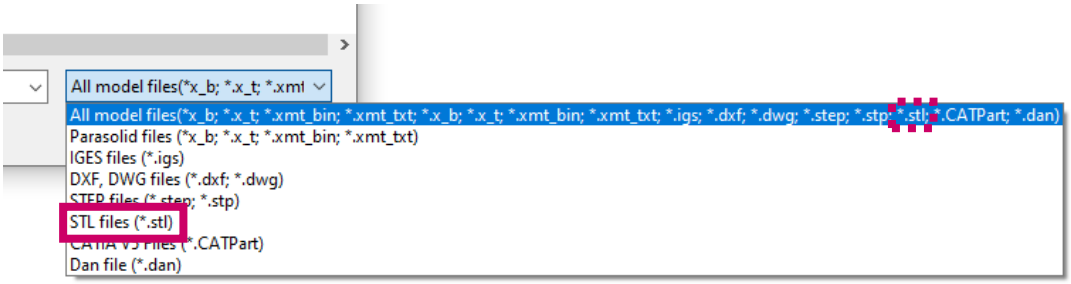
The [Load STL Model] menu has been removed, and you can select and import "*.stl" files from the [Model Import] menu.

(Screen of FFCAM2019 is used for explanation)



* You can also import "*.stl" files from the menu bar by selecting [Files] -> [Model Import].

- File format selection screen of [Model Import] (FFCAM 2020)



14. <Machine Parameter> Addition of a Local Variable Search Function to the FFPOST Edit Screen

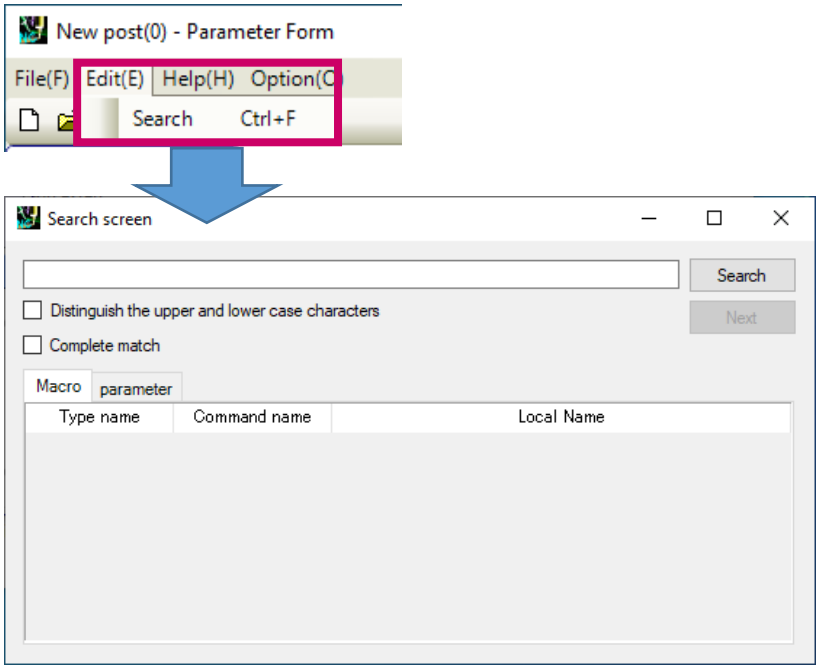
The FFPOST edit screen provides a number of local variables for post editing.

The wide variety of local variables allows additional customization, but at the same time makes it challenging to find the required local variables.

In FFCAM2020, a function to search for local variables in the FFPOST edit screen has been added.

■ Setting Screen

Select [Edit] -> [Search] from the FFFPOST edit screen menu bar to launch the local variable search screen.
(Alternatively, press Ctrl + F key to launch the FFPOST edit screen.)



(1) Field to enter text

Enter the character string you want to find. Only one word can be searched at a time.

(2) Search

Runs the search. (Alternatively, press the Enter key to execute the search)

(3) Next

Moves the selection from the selected row to the next row in the search results list.

(4) Distinguish the upper and lower case characters

Searches for words that match the character case.

(5) Complete match

Searches for exact phrases. Partial matches are not searched.

(6) Search result list

A list of local variable names is displayed in the search results.
Select the [Macro] tab to search for local variables in [Output NC Information] on the FFPOST edit screen, and select the [Parameter] tab to search for local variables in [Output Parameter List].

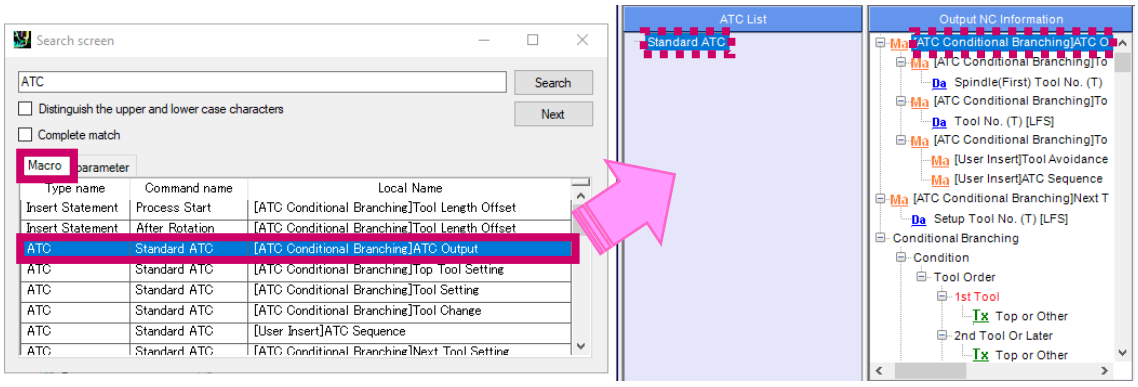
- **Macro tab**

  When you select a row in the result list, the corresponding local variable is selected in [Output NC Information] in the FFPOST edit screen. The POST edit screen switches to the appropriate mode (NC Output, Insert Statement, ATC, Macro).

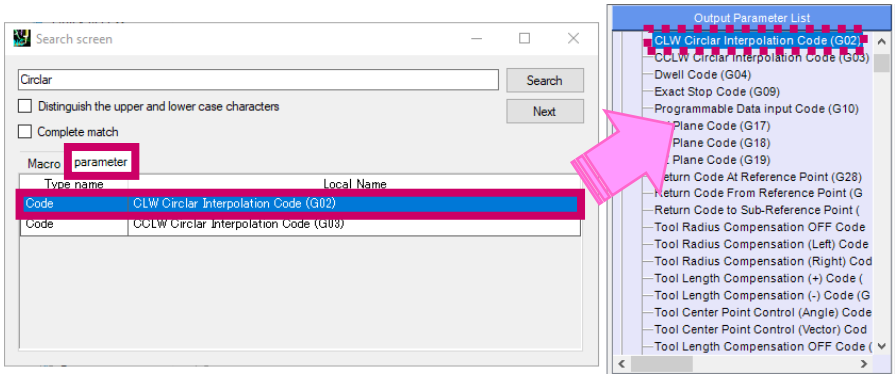
- **Parameter tab**

  When you select a row in the result list, the corresponding local variable is selected in [Output Parameter List] in the FFPOST edit screen.

- When a row from the result list is selected in the [Macro] tab screen



- When a row from the result list is selected in the [Parameter] tab screen



15. <Machine Simulator (Option)> Added the Function to Color-Code the Cut Rest of Cut Workpieces

In the machine simulator, a function has been added to display the uncut workpiece after simulation with color-code based on the cut rest amount.

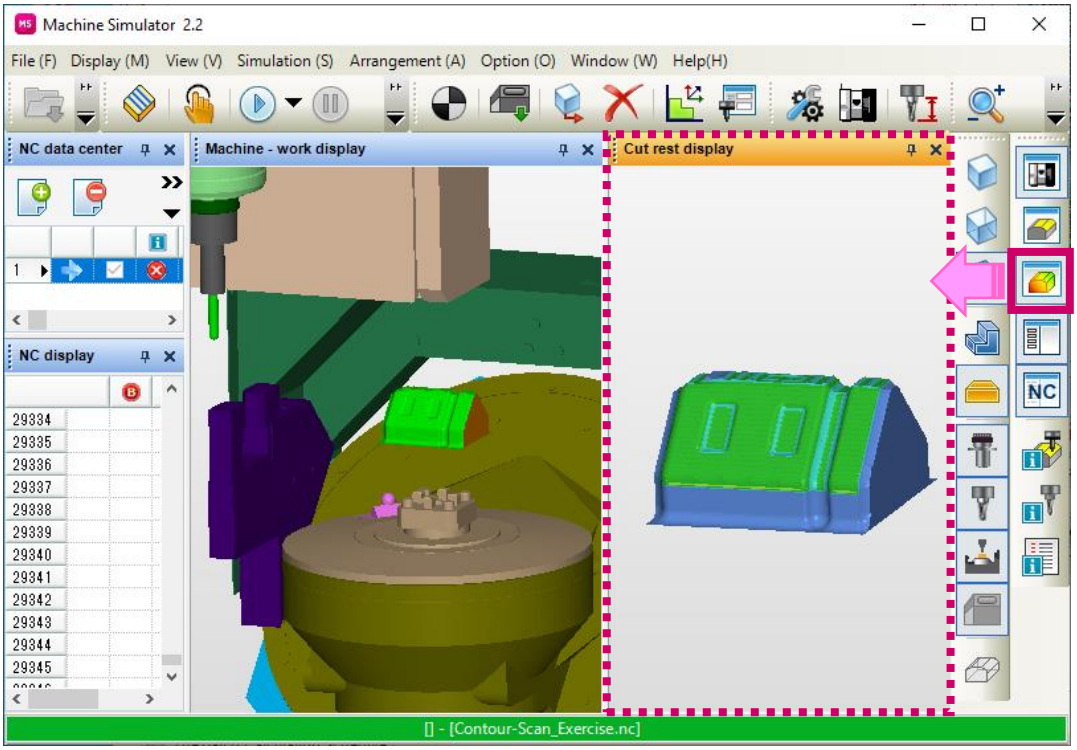
Previously, it was difficult to check cut rests and bites on the cut workpiece displayed in the machine simulator.

In FFCAM2020, the machine simulator provides a color-coded display of the cut rest amount after simulation, making it easier to check the machining results.

■ Setting Screen

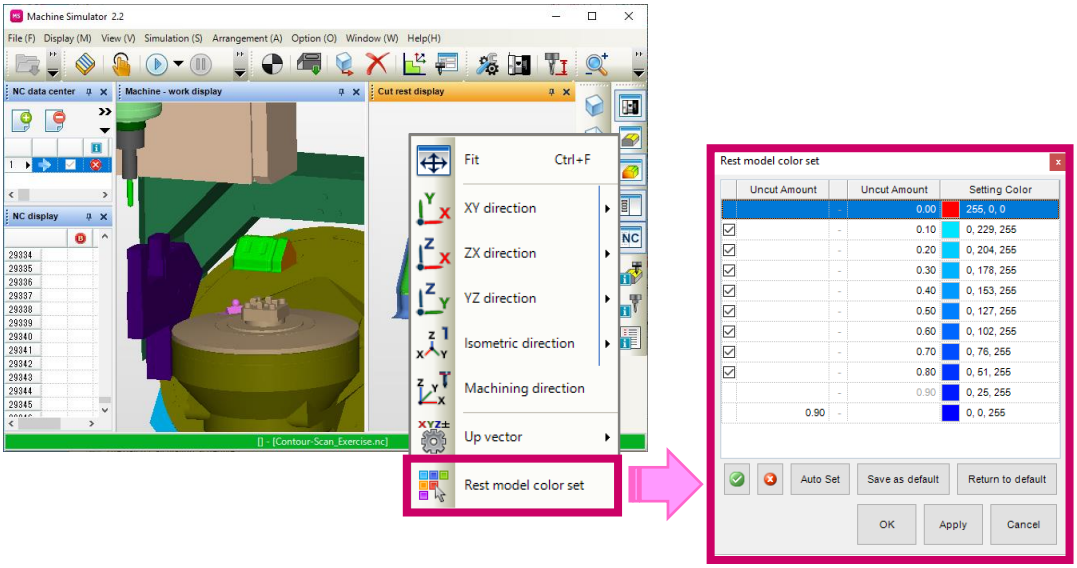
● How to Start Cut Rest Display

After the simulation, click the [Cut rest display] icon on the toolbar.
The [Cut rest display] screen is displayed with the cut workpiece, color-coded based on the rest amount.



How to Set the Cut Rest Color

Right click on [Cut rest display] screen, and select [Rest model color set] from the menu that is displayed. The [Rest model color set] screen starts.
Set the display color in [Rest model color set].



* The setting method of the [Rest model color set] screen is the same as that of the [Rest Model Color Setting] in FFCAM machining data selection and simulation.

- **Save as default**
  The value set in [Rest model color set] is registered as the default value.
- **Return to default**
  Returns the setting in [Rest model color set] to the default value.

How to Confirm the Initial Setting Value

Click the [Initial Setting] icon on the toolbar to display [Initial Setting] screen.
You can check the initial setting value in [Cut Rest Display Color] on the [Display Color] tab.

16. Maintaining User Database Templates When Upgrading FFCAM (1)

When FFCAM is upgraded, "Converting the Setting data" is performed, and the settings of the previous version are transferred to the new version.

* For information on "Converting the Setting data", see "Installation/Uninstallation Manual" in the FFCAM installer.

After converting the setting data, the user database template is saved so that the new version of FFCAM continues to refer to the template destination folder of the previous version of FFCAM.

FFCAM templates are stored in the "Template folder" by default.

If the setting data is converted during the version upgrade, the new version of FFCAM will refer to the "Template folder" of the previous version.

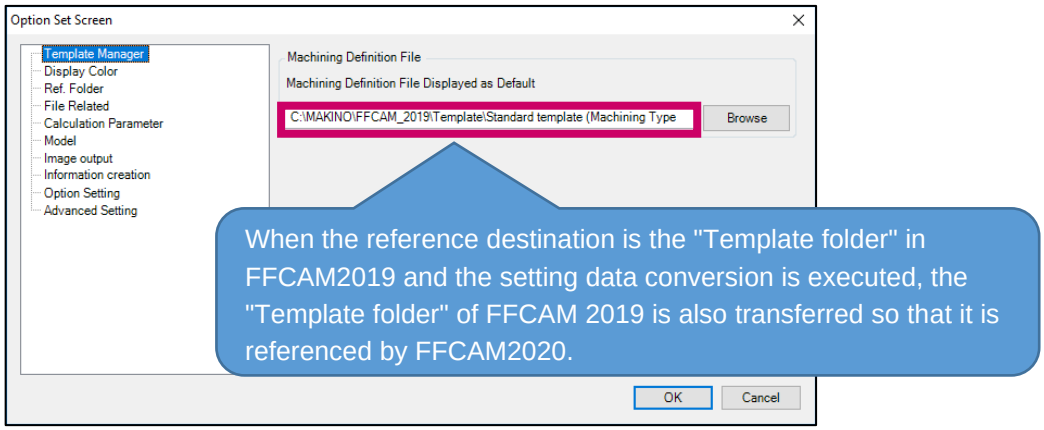
If you uninstall an older version of FFCAM, the old version of "Template folder" referenced by the new version of FFCAM will be removed inadvertently. In this case, the customized template files saved in the folder are also deleted.

In FFCAM2020, to prevent this issue, the specification has been changed so that the "Template folder" is not removed even after installation.

■ Explanation Screen

When FFCAM is upgraded and the setting data is converted, the settings of Option Set Screen -> [Template Manager] -> [Machining Definition File] are converted without any changes.

Example: Option Set Screen of FFCAM2019



FFCAM2020 and newer versions will not remove "Template folder" during uninstall to avoid losing the referenced folders.

In versions prior to FFCAM2019, the "Template folder" is removed during uninstallation. We recommend that you use the functions of FFCAM2020 described in the next chapter to manage this issue.

17. Maintaining User Database Templates When Upgrading FFCAM (2)

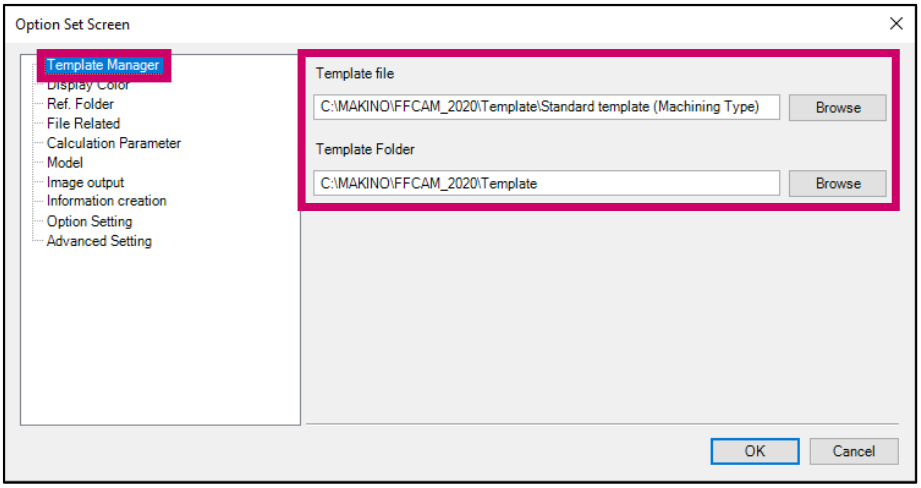
In FFCAM2020, any folder can be specified as the destination to save the template file of the user database.

As described in the previous chapter "16. Maintaining User Database Templates When Upgrading FFCAM (1)", in addition to the measures to prevent loss of the "Template folder", a parameter has been added to allow more flexible management of user database template data.

By managing the destination to save template files, users can efficiently migrate template files when upgrading.

■ Setting Screen

A parameter has been added to the [Template Manager] on the Option Set Screen to specify the location to save the template file of the user database.



(1) Template file

Specifies the user database template file to be displayed at startup.
(This is the same parameter as the old [Machining Definition File Displayed as Default].)
When the setting data is converted from FFCAM2019 to FFCAM2020, the setting of [Machining Definition File Displayed as Default] of FFCAM2019 is transferred to the [Template file] of FFCAM2020.

(2) Template folder (new function)

Specify a folder to save the user database template.
The default is "Template folder" in FFCAM, but it can be changed.

If the setting data is migrated during FFCAM version upgrade, the settings made in [Template file] and [Template Folder] are also transferred.

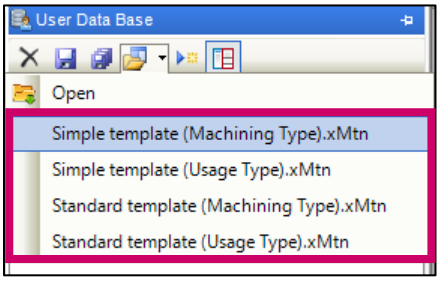
It is recommended to change the location for saving the user database template from "Template folder" to an arbitrary folder that can be managed using the above functions.
When FFCAM is upgraded, the arbitrary folder to be managed is transferred as a reference, so it is not deleted accidentally when FFCAM is uninstalled.

■ Template File

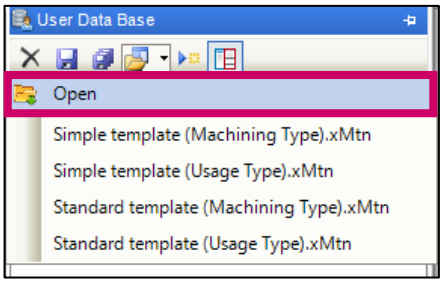
The folder specified as a template folder is subject to the following operations.

User Data Base Screen

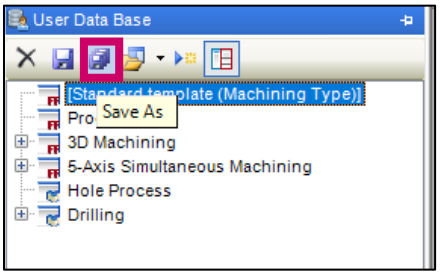
- Destination for saving the files to be displayed in the candidate list



- Initial reference folder for [Open]



- Initial reference folder for [Save As]



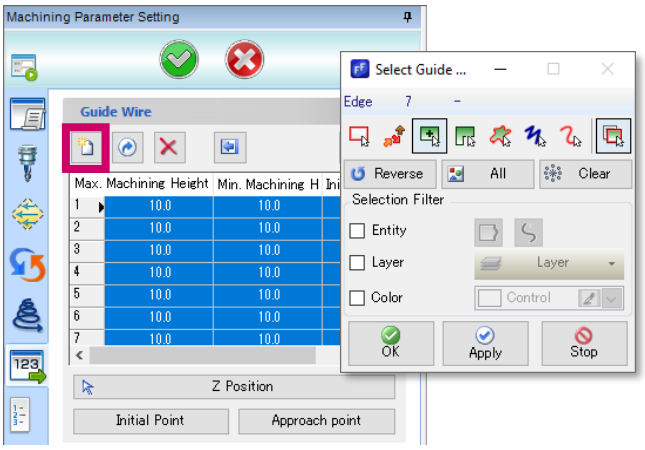
18. <2D Route Machining> Added a Batch Registration Function for Multiple Guide Wires

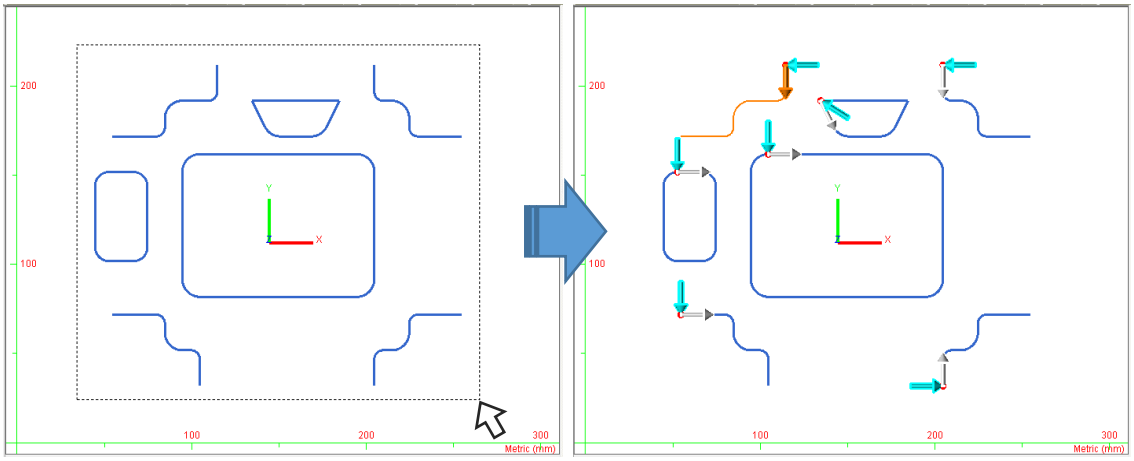
Previously, when registering a guide in 2D route machining, it was necessary to select wires of each contour one by one even if there were multiple guide wires.

In FFCAM2020, you can now select multiple guide wires at once by selecting the range. When the number of wires are more, the number of steps and time required to select them can be reduced.

■ Setting Screen

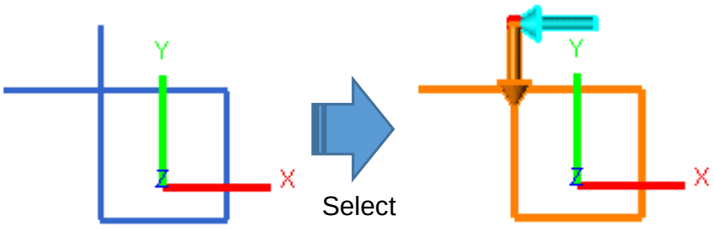
When selecting wires with the [Guide Wire] parameter, you can register several noncontinuous wires at once.



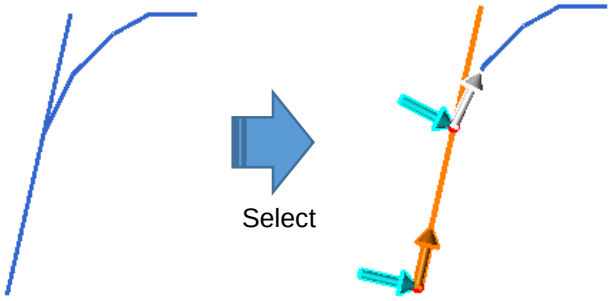


■ Example of Selecting a Complex Wire

- Crossed wire



- Branching wire



■ Note

If the distance between ends of the wires is less than or equal to 0.1 mm, the wires are considered to be continuous. * The wires may not be recognized due to calculation error.

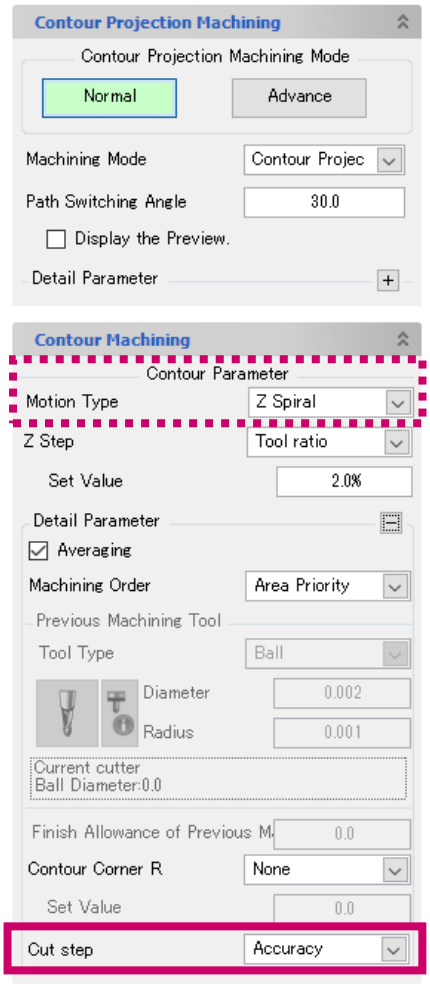
19. <Contour Projection Machining> Addition of Cut Step Parameter to Spiral Motion

A parameter for [Cut step] has been added to the contour spiral motion in the contour projection machining.

With the addition of the [Cut step] parameter, you can adjust the accuracy of the spiral motion and calculation speed.

■ Setting Screen

The [Cut step] can be set when [Motion Type] is set to [Z Spiral] in [Contour Parameter] of contour projection machining.



* The [Cut step] parameter is set in the same way as the [Cut step] parameter for contour machining.

■ Note

If [Motion Type] is not [Z Spiral], [Cut step] cannot be set.

20. Improved Display Function of the Measurement Result Balloon

In the measurement function, the function to fit the annotation balloon display within the screen has been added.

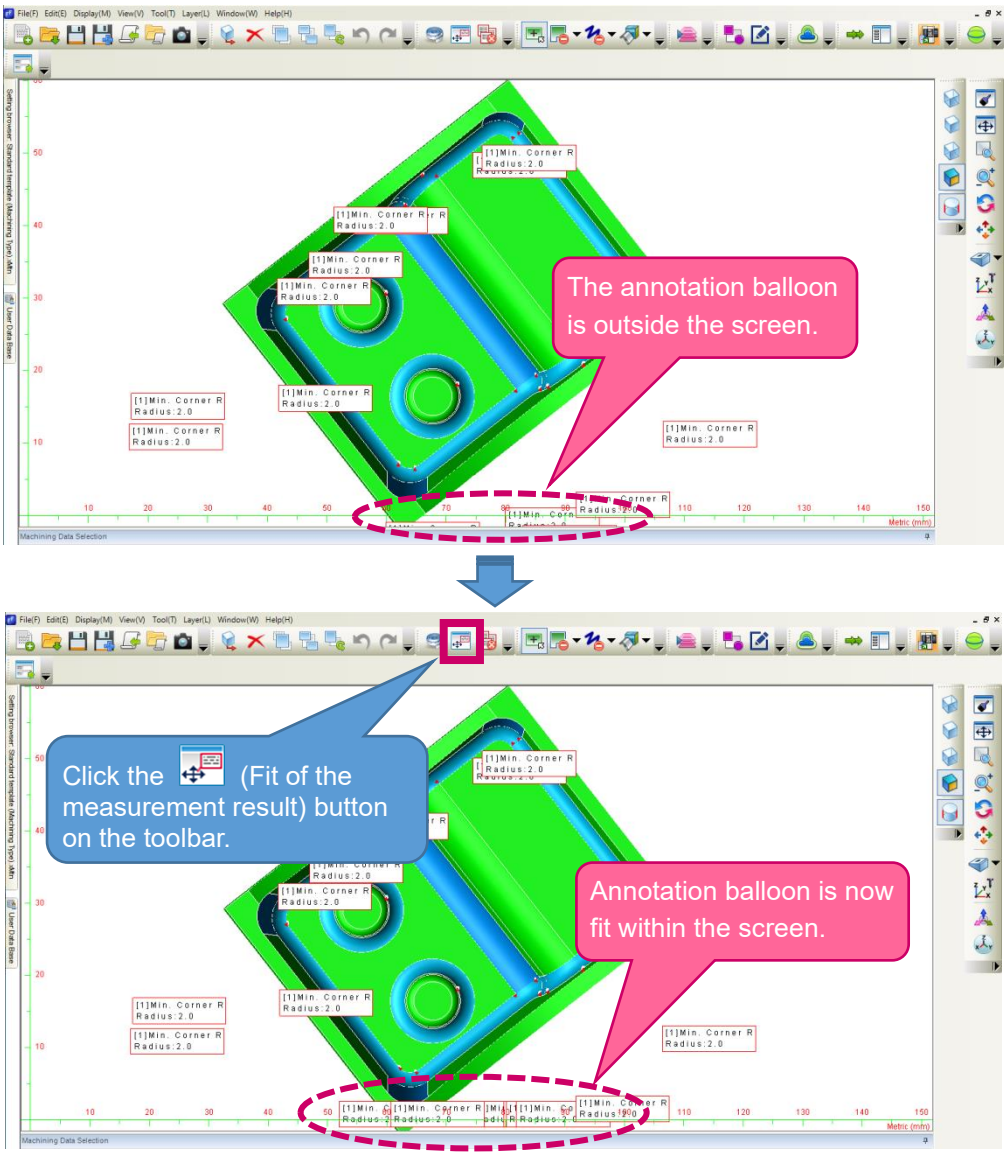
The measurement function displays the measurement results in an annotation balloon in the graphics window.
In some cases, annotation balloons may move out of the screen and disappear.

The [Fit of the measurement result] function allows you to move annotation balloons to fit within the screen.

■ Operation Screen

After measuring with the measurement function, click the [Fit of the measurement result] button on the toolbar.

- Screen after measurement by measurement function



* Selecting [Tool] -> [Fit of the measurement result] from the menu bar also has the same result.

■ Conditions to Fit

When [Fit of the measurement result] is executed, the annotation balloons that satisfy the following two conditions are moved to fit within the screen.

- Annotation balloons that are outside the screen or have been cut off.
- Annotation balloons whose measurement point of the corresponding annotation is within the display range of the screen.

If the measurement point and its annotation balloon are both outside the screen (or have been cut off), the annotation balloon will not be moved to fit inside the screen.

■ Note

- Annotation balloons may overlap after moving.
- The moved annotation balloon may overlap with the annotation wire and measurement point.

21. <2D Route Machining> Addition of Auto Setting Function fir Max./Min. Machining Height of Index Setting

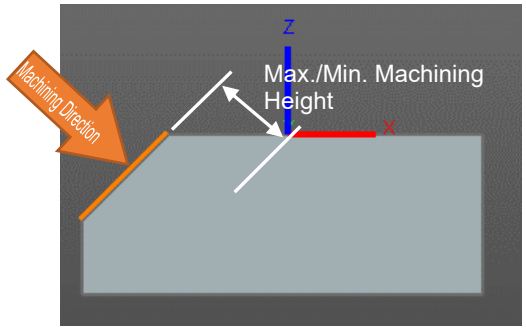
When registering a contour in 2D route machining, it is necessary to acquire and define the maximum and minimum machining height of the contour.

Previously, the Z height could not be automatically defined if the machining direction was set to a direction other than + Z in the index settings for 2D route machining.
Therefore, every time a contour is registered, it is necessary to enter the values for [Max. Machining Height] and [Min. Machining Height], which required time and effort.

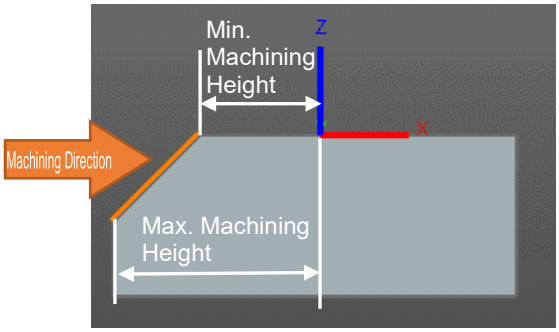
In FFCAM2020, a function has been added to automatically define [Max. Machining Height] and [Min. Machining Height] even when the machining direction is other than + Z in 2D route machining.

Automatic setting of max./min. machining height in FFCAM2020

- When the specified contour is on a plane perpendicular to the machining direction:
  The same value is set for the maximum and minimum machining heights.



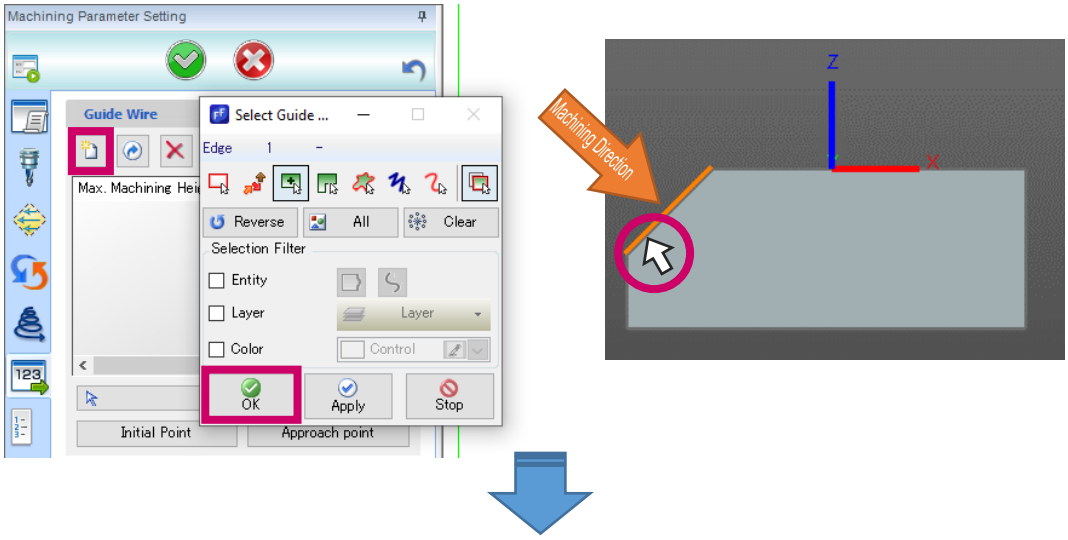
- If the specified contour is not on a plane perpendicular to the machining direction:
  Based on the machining direction, the highest point is set to the maximum machining height and the lowest point is set to the minimum machining height.



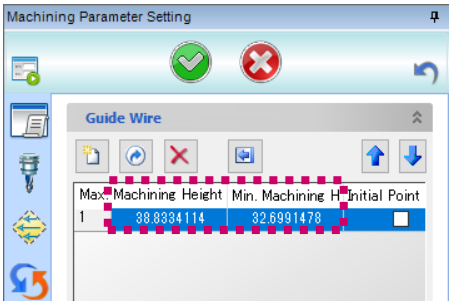
■ Setting Screen

When a new contour is registered in 2D route machining, the maximum and minimum machining height is automatically defined according to the position of the curve/edge even if the machining direction is other than the +Z axis direction.

Register a new contour.



The contour is registered with the maximum and minimum machining heights set.



■ Note

The maximum and minimum machining heights are set when a new contour is registered. When redefining the contour or changing the machining direction after defining the contour, the maximum and minimum machining heights are not updated automatically.

22. Addition of a Function to Measure the Cylinder Area of Geometry

The function to measure the cylinder area of geometries has been added to Measurement.

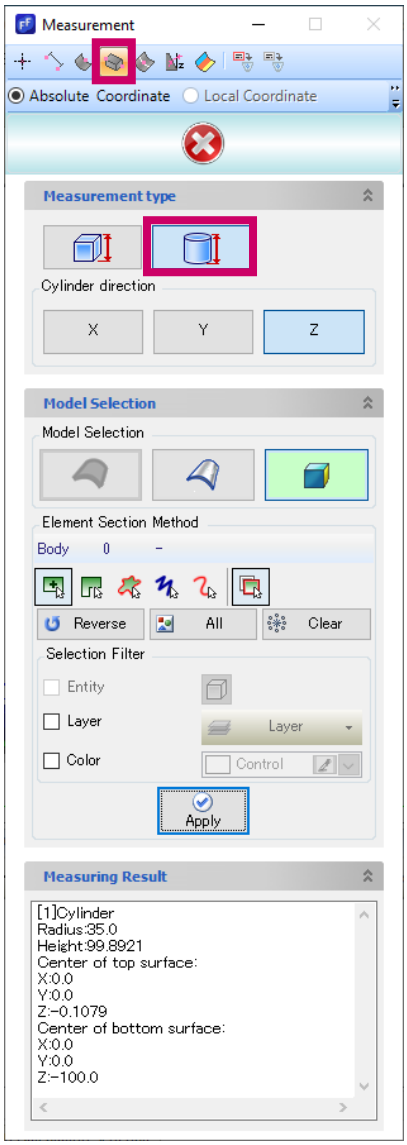
Previously, the only function to measure the size of a geometry was to measure each maximum and minimum value of the XYZ coordinates of the geometry in the box area.

When you want to check the size of the geometry with cylinder area, the required numeric values (radius and height of the cylinder area) cannot be obtained from the measurement of the box area, and radius and height were measured respectively.

In FFCAM2020, the function to measure the geometry size in the cylinder area has been added, and the required numeric values can be obtained efficiently.

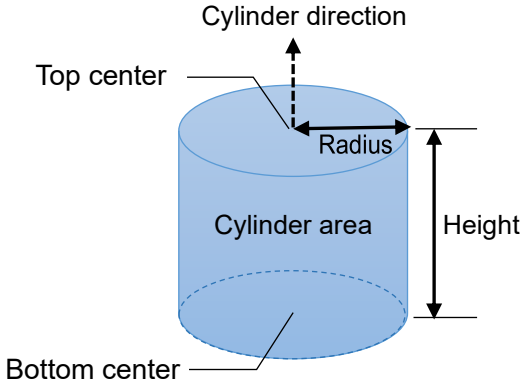
■ Setting Screen

Click the [Measurement] icon on the [Maximum and Minimum on Geometry] screen. Select the [Cylinder] button of [Measurement type] and measure the cylinder area (radius and height) of the geometry.

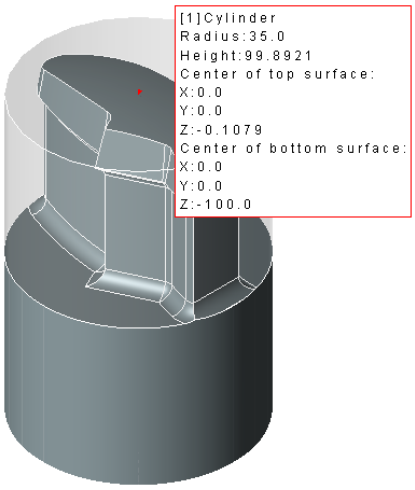


* If you select the [Box] button in [Measurement type], you can perform measurements with the box area in the same way as the conventional [Maximum and Minimum on Geometry] function.

- **Measurement points in the cylinder area**



- **Illustration of measurement result**



Maximum and Minimum of Geometry

(1) Absolute Coordinates

Acquires the center coordinates of the top and bottom center of the cylinder area as absolute coordinates.

* When the [Measurement type] is "Cylinder", [Local Coordinate] cannot be specified.

(2) Boundary Hide

- Selected
  The bounding box is cleared after the measurement menu is closed.
- Not selected
  The bounding box continues to be displayed even after the measurement menu is closed.
  * You can delete the bounding box with [Clear Measurements].

(3) Measurement type

- Box (default value)
  Measures the maximum and minimum values of the box area of the geometry. (Old function)
- Cylinder
  Measures the radius and height of the cylinder area of the geometry. (New function)

(4) Cylinder direction

Select the direction of the center axis of the cylinder area to be measured with the "X (axis)", "Y (axis)", and "Z (axis)" buttons.* Select this option when [Measurement type] is "Cylinder". The default value is "Z (axis)".

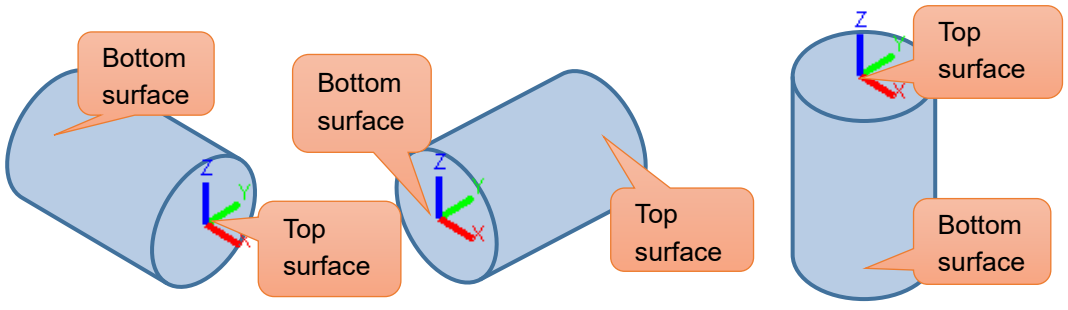
(5) Model Selection

Set the geometry selection method.
After selecting the geometry, click [Apply] to display the measurement results.

(6) Measurement Result

When a cylinder area is measured, the values of "Radius", "Height", "Center of top surface", and "Center of bottom surface" are displayed as the measurement results.

* Each coordinate value of the cylinder area is displayed as an absolute coordinate value based on the coordinate origin on the screen.



Cylinder direction: X axis　Cylinder direction: Y axis　Cylinder direction: Z axis

* The + side of the axis direction specified in [Cylinder direction] is the top surface, and - side is the bottom surface.

23. Updated Motion Conditions of Contour Corner R Function

The motion of the [Contour Corner R] function was improved in FFCAM 2018. (See below)

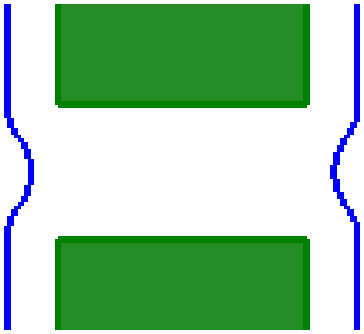
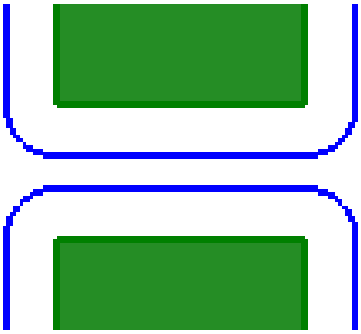
The improved motion was not applied when [Contour Face Cut Machining] was enabled.
* When [Contour Face Cut Machining] is enabled, the motion before the improvement was applied.

With the [Contour Corner R] function of FFCAM 2020, the improved motion is now applied even when [Contour Face Cut Machining] is enabled.

■ Explanation of Operation of [Contour Corner R] Function

In the operation before improvement, the tool path was not output for gaps that are less than or equal to twice the radius specified by [Contour Corner R].

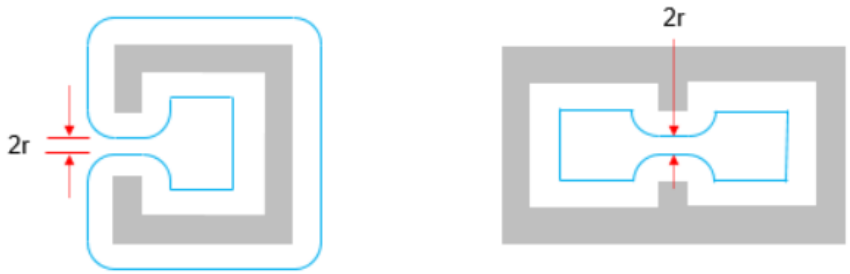
In the operation after improvement, a tool path can be output for gaps that are less than or equal to twice the radius of specified by [Contour Corner R].

● **Operation before improvement** Tool path is not created for grooves. 	● **Operation after improvement** Tool path is created for grooves. 

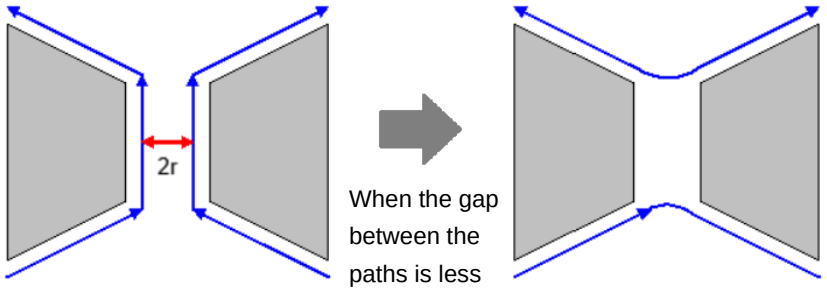
■ Notes for Operation After Improvement

The tool path may not be output for gaps that are less than or equal to twice the radius specified by [Contour Corner R] when the machining area or machining workpiece is not set.

When the tool path is a single looped path and the machining area or machining workpiece is not set, corner R such as (a) or (b) shown below is inserted for the gaps having less than or equal to twice the radius specified by [Contour Corner R].

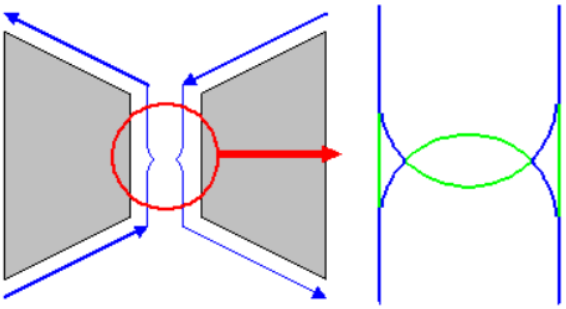


(a) In the figure below, corner R is inserted to fill the gap.



When the gap between the paths is less than or equal to 2r:

(b) The corner R shown below is inserted depending on the gap shape.



Green portion is omitted.

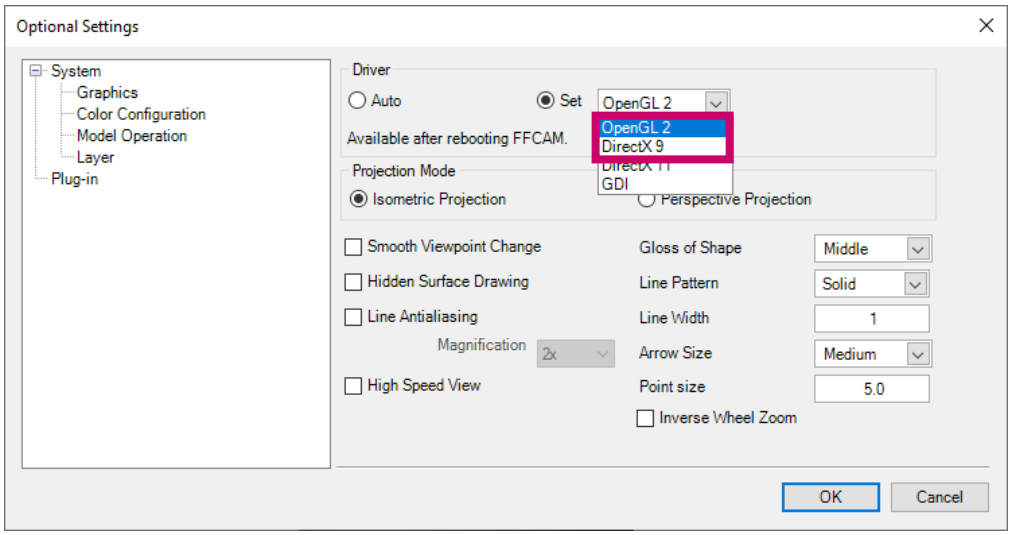
Tool path when corner R overlap

24. Change in Supported Graphics Drivers

In FFCAM2020, the software rendering system was updated.
As a result, use of graphics drivers "OpenGL" and "DirectX 9" is not recommended.

■ Setting Screen

Optional Settings (System/Graphics) screen



Driver (Set)

- **OpenGL 2**

  Supports OpenGL 2.0 or higher driver.
  The previous specification item of "OpenGL" has been abolished.
  "OpenGL" is not recommended, but can be used as "OpenGL 2".
  * If FFCAM2019 with "OpenGL" is upgraded to FFCAM2020, "OpenGL 2" will be specified.

- **DirectX 9**

  "DirectX 9" is not recommended but can be used.

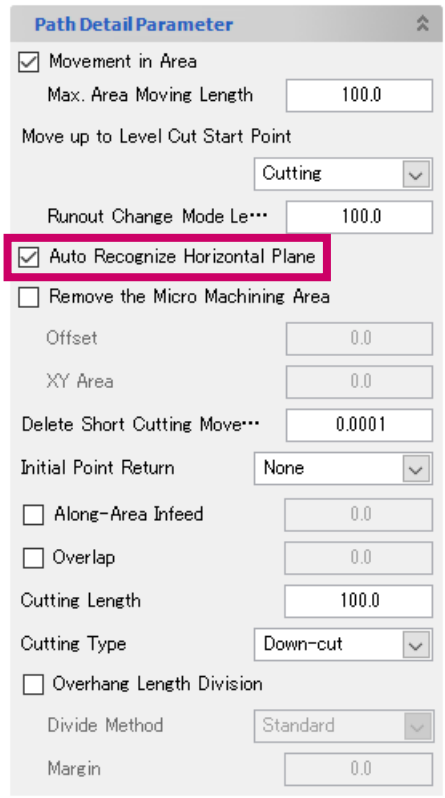
25. Improved the Function to Recognize Horizontal Plane Automatically

Previously, the [Auto Recognize Horizontal Plane] function also detected the horizontal surface outside the machining area and the tool path was output even at an unnecessary contour level.

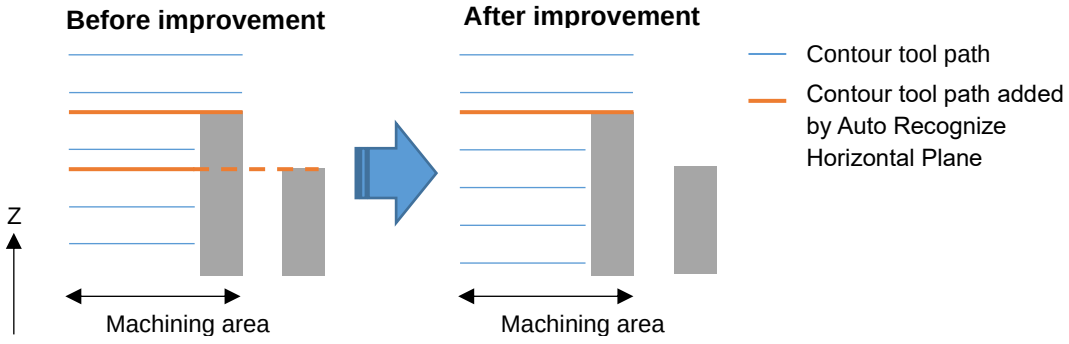
In FFCAM2020, the function has been improved so that it recognizes only the horizontal plane in the machining area and outputs a streamlined tool path.

■ Setting Screen

Machining Parameter Setting/Path Detail Parameter



■ Illustration of Tool Path Output



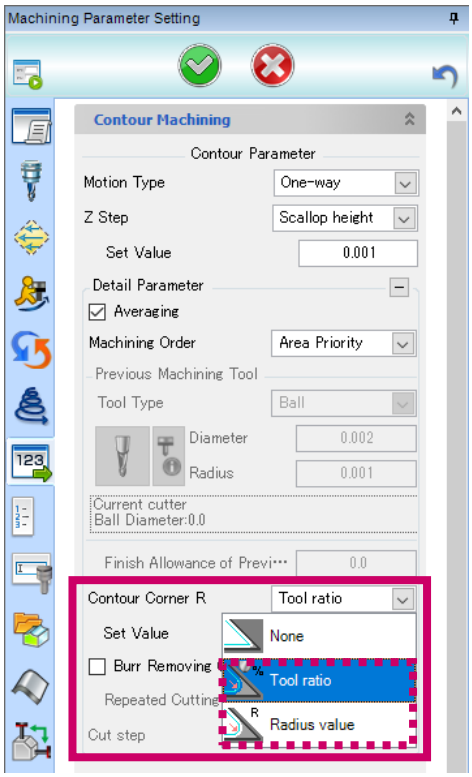
26. Improved Tool Path Calculation Speed for Contour Machining

In FFCAM2020, the function has been improved for faster calculation of tool path when "Contour Corner R" and "Outside area infeed of contour infeed" are set in contour machining.

■ Setting Screen

Path calculation time is reduced when the following parameters are set.

- Contour Corner R setting



- Infeed outside area setting