102-NF-18000

FFCAM 2018

Description of New Functions

MAKINO

Preface
This manual describes the functions added to MAKINO FFCAM 2018 and how to use them.

Created on
May 2018

List of Added Functions in FFCAM 2018

1. Addition of the Model top Automatic Recognition Function 1
2. Improvement of Contour Corner R Function 6
3. Improvement of Follow Motion [Inward Cutting] 8
4. Addition of Function for Creating Angle Line 9
5. Addition of Function for Editing Curves 10
6. Addition of Guide Display Function at the Time of Curve Creation 12
7. Circle Preview 15
8. Changing the Plane Selection Screen During Curve Creation 16
9. Lens barrel tool 17
10. Supports Multiple Taper Neck Stages 19
11. Support for Two Piece Holder 24
12. Improvement of the User Data Base Function 26
    12.1. Change in Switching Method in User Data Base 27
    12.2. How to Start the Setting Browser 28
    12.3. Explanations for Process and Machining of User Data Base 29
    12.4. Changes in User Data Base 31
13. Batch copy (move) of multiple machining 34
14. Information Added to Machining Instructions 35
15. Output of Calculated Neck Length 39
16. Addition of Insertion Statement Macros 42
17. Improvement of Collision Safeguard Data Output Function 43
    17.1. Support for Professional 6 44
    17.2. Addition of Tool Library Output Target 45
    17.3. Addition of Function for Setting Numbers and Names to Materials 46
    17.4. Addition of Function for Adding Numbers and Names to Fixtures 47
18. Changes in Behavior of Tool Drawing Icon 48
19. Adjustment of Screen Size of Tool DB Maintenance 49
20. Standardized STL Model Machining Function 50
21. 5-axis Simultaneous Support for Hairline Finish Machining Function 51

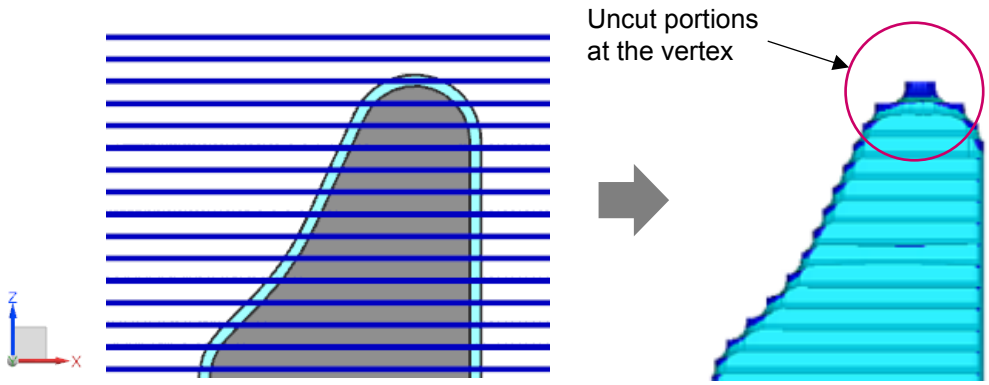
1. Addition of the Model top Automatic Recognition Function

[Model top automatic recognition] is another new function that has been added. This function automatically recognizes the machining geometry vertices and adds the tool path of rough contour machining to its Z level.

This function reduces the uncut portions between contour steps at the machining geometry vertices to prevent holder interference.

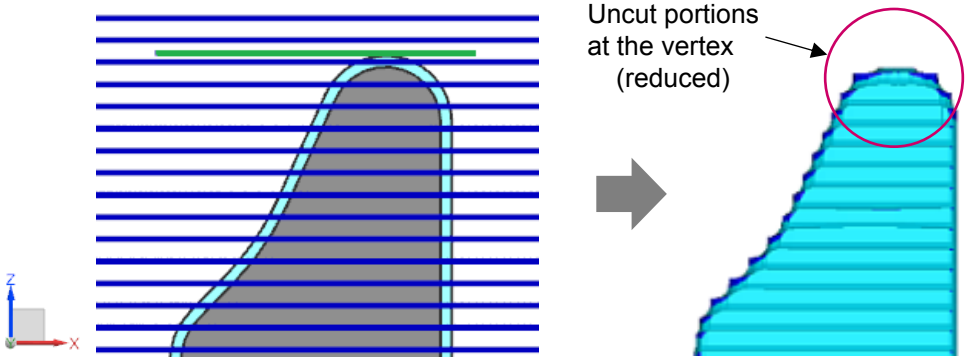
This function can be used for "Rough Contour Machining", "Contour Semi-Finish Machining", and "Contour Finish Machining".

- **Normal Contour Machining**

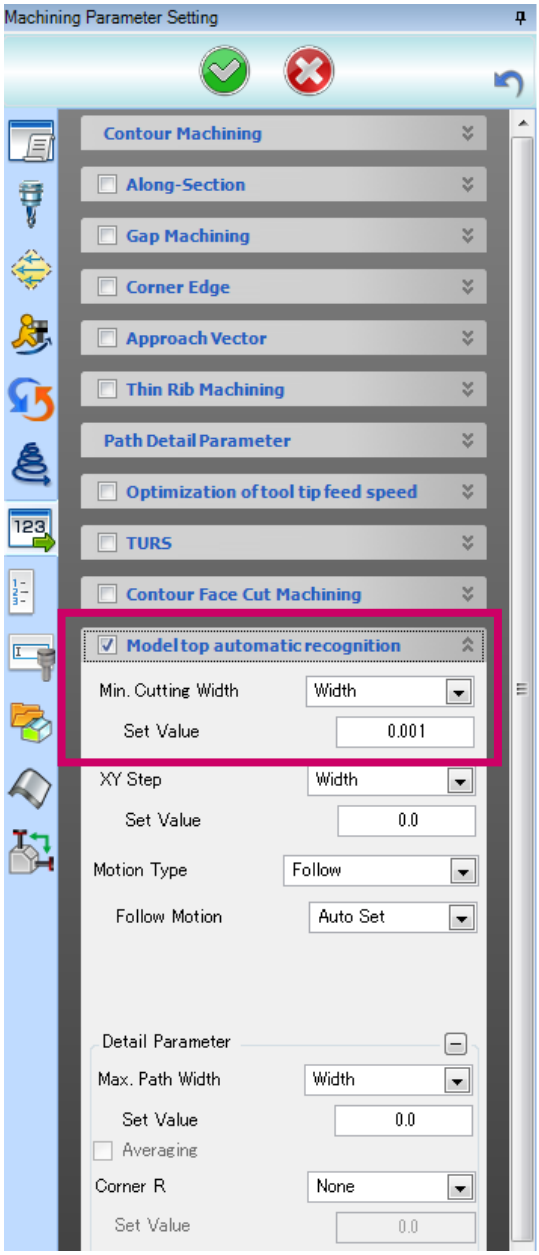


- **Contour Machining Using [Model top automatic recognition] Function**

Tool path added by [Model top automatic recognition] function



■ Setting Screen

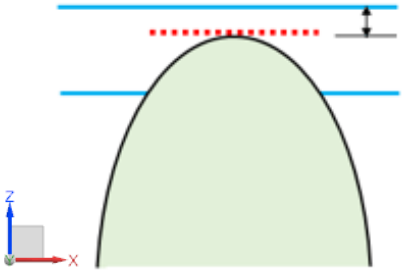


(1) Model top automatic recognition

Selecting the checkbox automatically recognizes the vertices of machining geometry and adds the tool path of rough contour machining to its Z level.

(2) Min. Cutting Width

This is the value to be referenced when determining whether or not to add the tool path to the vertex Z level.
If the gap between the tool path to be added to vertex Z level and the tool path that is one place above is less than or equal to the value set in [Min. Cutting Width], the tool path will not be added to the vertex Z level.
Set either [Scallop Height], [Width], or [Tool Ratio].



(3) XY Step

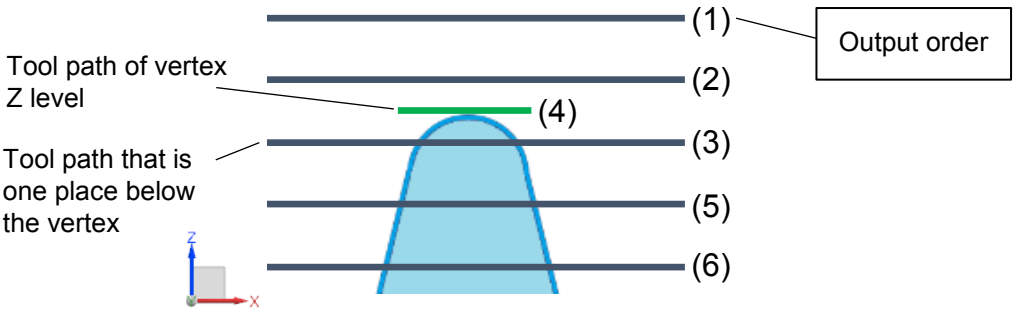
Specify [XY Step] of the tool path added by [Model top automatic recognition] function.

(4) Motion Type

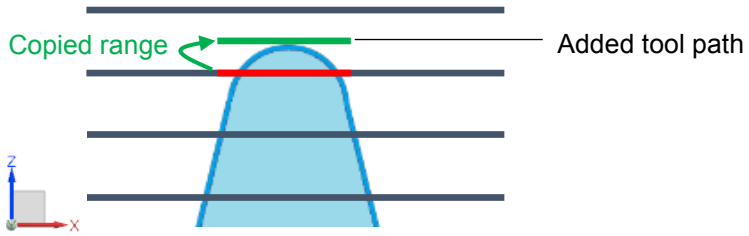
Specify [Motion Type] of the tool path added by [Model top automatic recognition] function.

■ Features of [Model top automatic recognition] Function

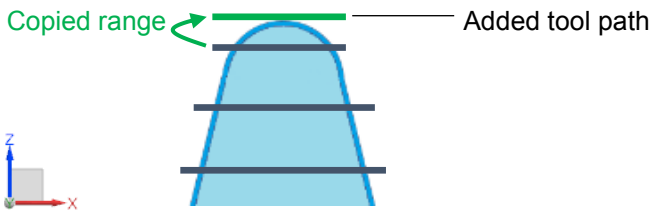
(1) The tool path of the vertex Z level is output right after the tool path that is one place below the vertex is output.



(2) When "Rough Contour Machining" is selected, the range in which the tool touches the shape in the tool path that is one place below the vertex is copied to the vertex Z level. The tool path of "Rough Contour Machining" is added for the copied range.



(3) When "Contour Semi-Finish Machining" and "Contour Finish Machining" are selected, the output range of the topmost tool path is copied to the vertex Z level.
The tool path of "Rough Contour Machining" is added for the copied range.



■ Note

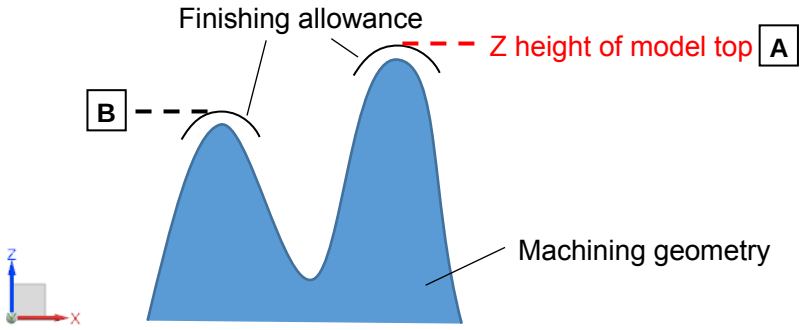
(1) If the following parameters are specified, [Model top automatic recognition] function cannot be used.

(a) When [Motion Type] of [Contour Machining] parameter is [Z Spiral]

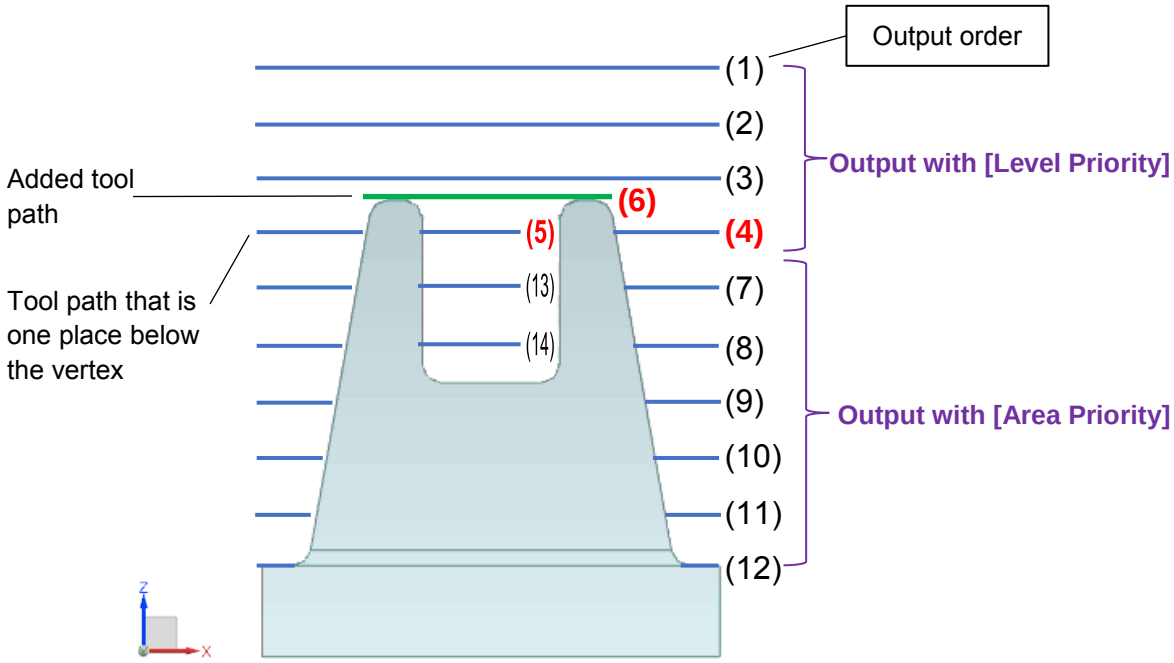
(b) When [Contour Face Cut Machining] parameter of [Motion Type] is [Trochoidal]

(c) When [Thin Rib Machining] is used

(2) The highest point at Z level of machining geometry is recognized as the "Peak". (See A below)
The top of the projection that is lower than the highest point of machining geometry is not recognized as the "Peak". (See B below)
Finishing allowance and offset plane are included in the height.



(3) If [Area Priority] is set for the shape shown below, up to the tool path that is one place below the vertex is output with [Level Priority]. The tool paths after the tool path at the vertex are output with [Area Priority].



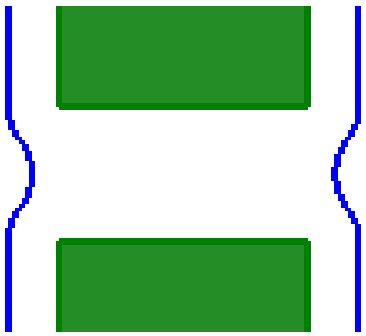
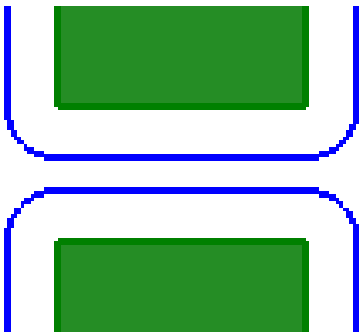
2. Improvement of Contour Corner R Function

In the previous FFCAM [Contour Corner R] function, the tool path was not output for gaps that were less than or equal to twice the radius specified by [Contour Corner R].

In FFCAM 2018, this function has been improved so that the tool path can be output for gaps that are less than or equal to twice the radius specified by [Contour Corner R].

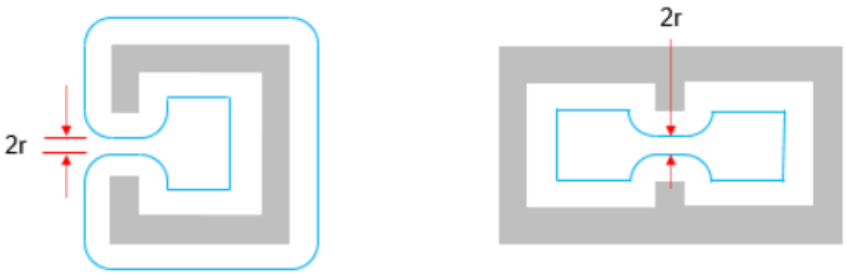
However, the tool path may not be output for gaps that are less than or equal to twice the radius specified by [Contour Corner R] when the machining area or machining workpiece is not set. For details, see "■ Note".

■ Tool path changes

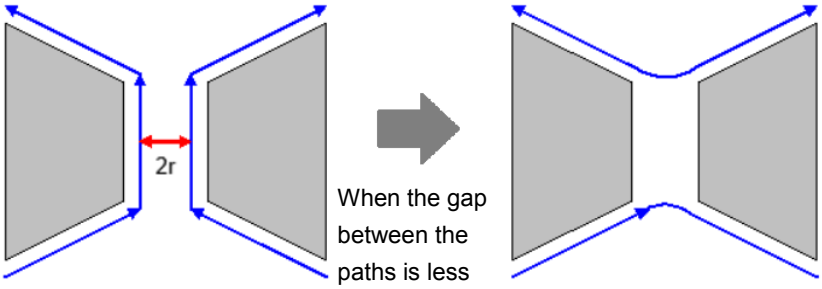
● **FFCAM 2017 or earlier** A path is not created for the groove. 	● **FFCAM 2018** A path is created for the groove. 

■ Note

When the tool path is a single looped path with the machining area or machining workpiece 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].

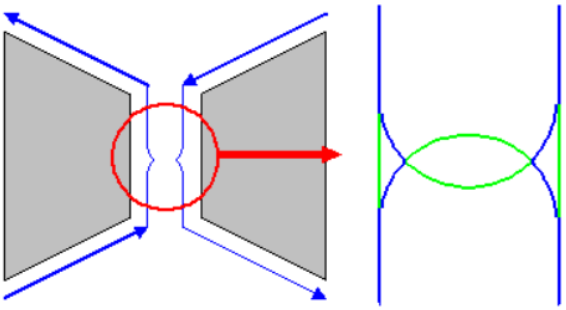


(a) In the figure below, corner R is inserted to fill the gap.



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

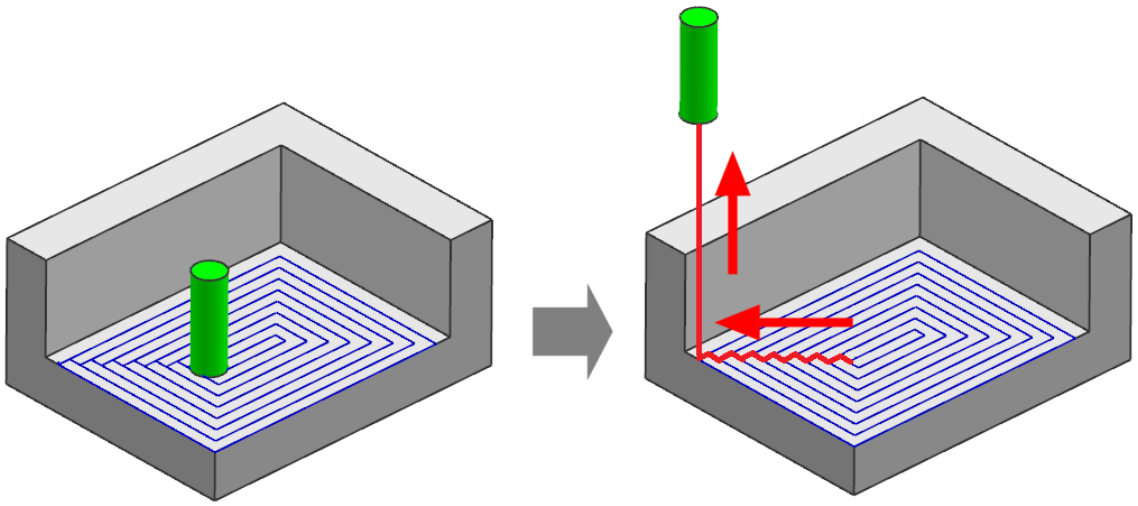
3. Improvement of Follow Motion [Inward]

In the previous FFCAM follow motion [Inward], a tool path that returns to the outside was output after the tool path that moved inward was output.

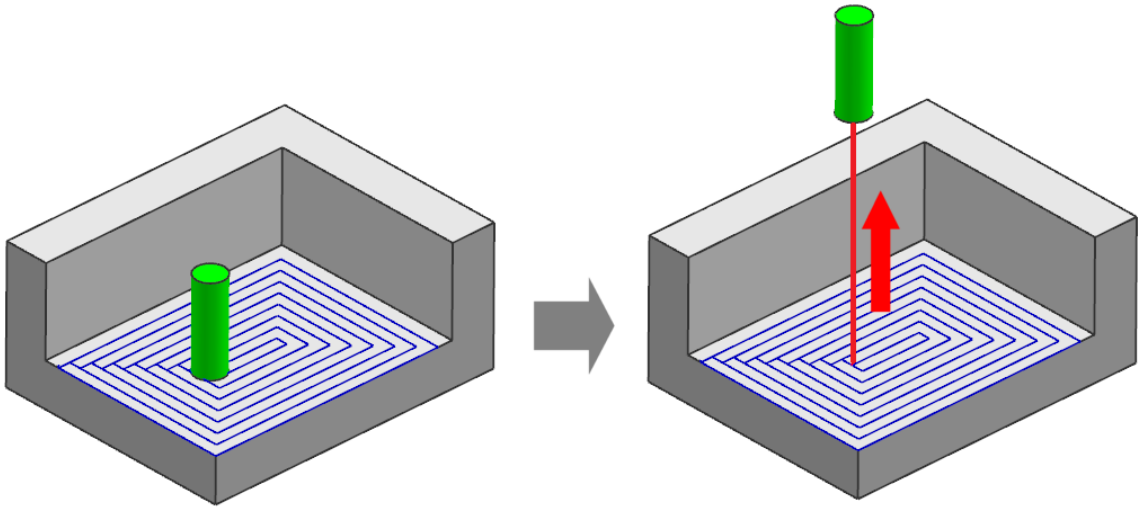
In FFCAM 2018, this function has been improved to stop output of the motion from going back to the outside after "Inward".

This improvement eliminates unnecessary tool paths and enhances the machined surface quality.

- **Operations of FFCAM 2017 or earlier**

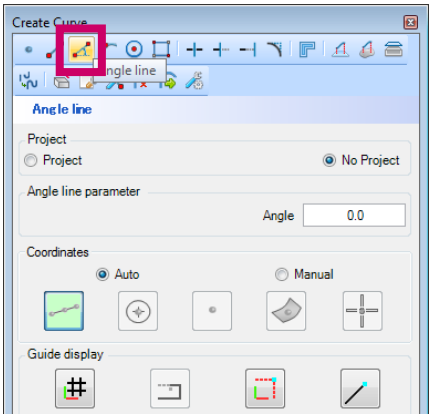


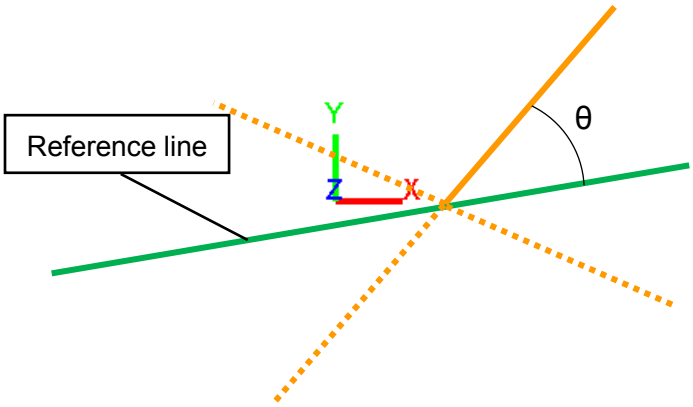
- **Operations of FFCAM 2018**



4. Addition of Function for Creating Angle Line

A straight line can be created from any straight line at a certain angle.
The angle can be set with respect to a reference line or X-axis. If you specify the angle (θ), you can create curves in four directions as an orange straight line shown in the following figure. The angle line can be set without limitations on the direction or positive and negative angles to be input for reference line.
For the method to create the angle line, refer to Chapter 22 "FFCAM Training Manual B".



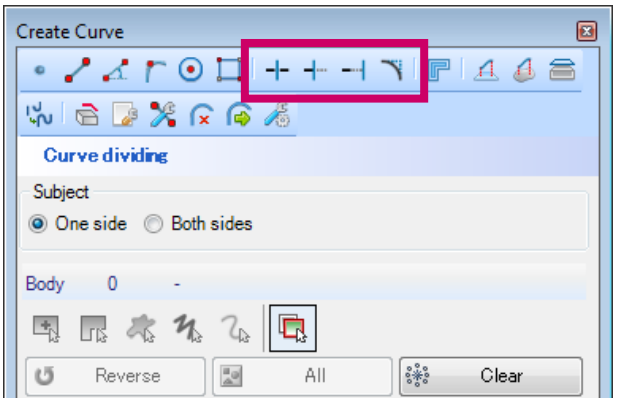


5. Addition of Function for Editing Curves

The following functions have been added to the Create Curve function.
You can edit the existing curves.

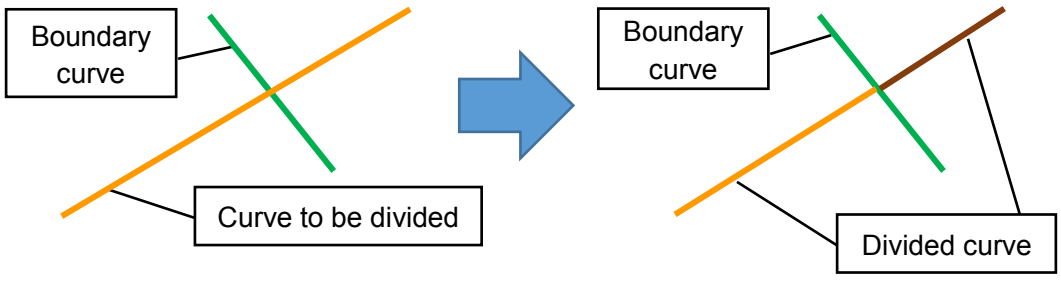
- Curve dividing
- Curve trimming
- Curve extension
- Curve fillet

For the detailed description of each function and the operating instructions, refer to Chapter 22 "FFCAM Training Manual B".



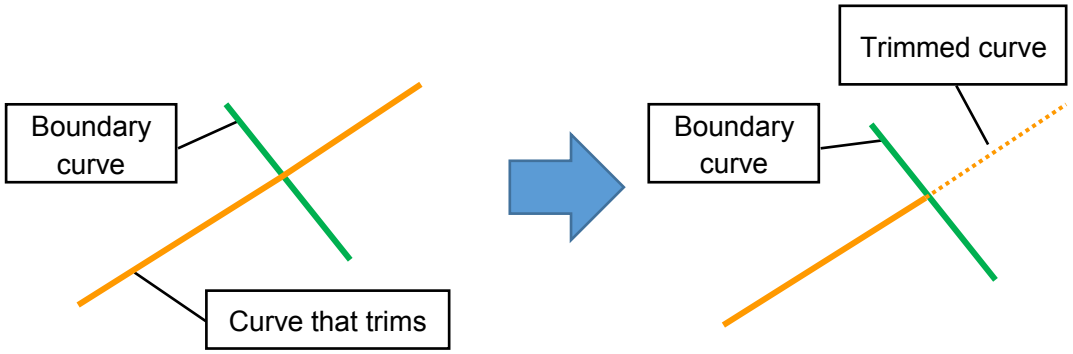
(1) Curve Dividing

Divides a curve into two curves at the boundary curve.



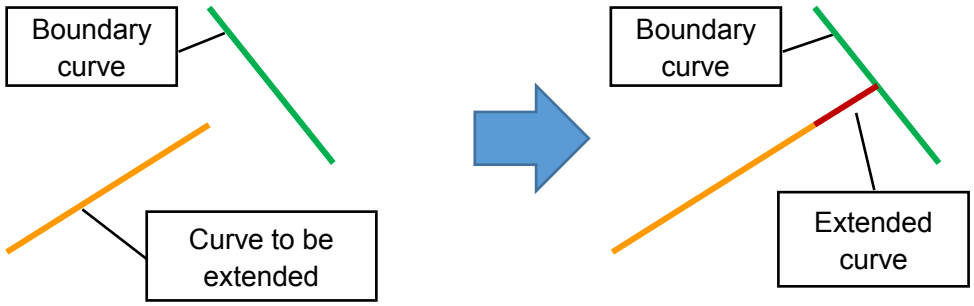
(2) Curve Trimming

Trims a curve at the boundary curve.



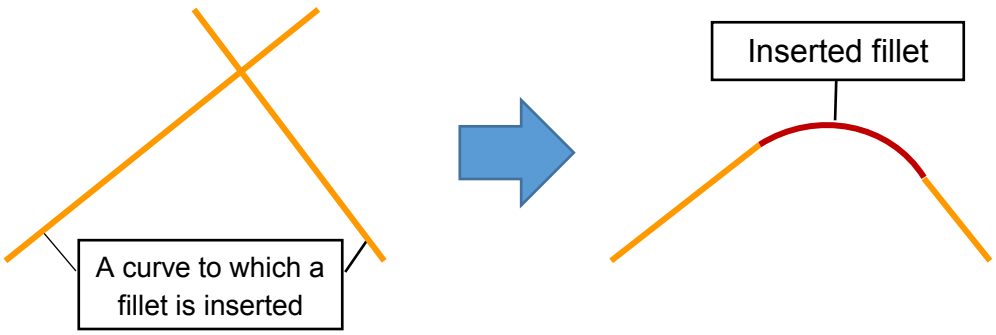
(3) Curve Extension

Extends a curve until it touches the boundary curve.



(4) Curve Fillet

Inserts a fillet of the specified radius between two curves.

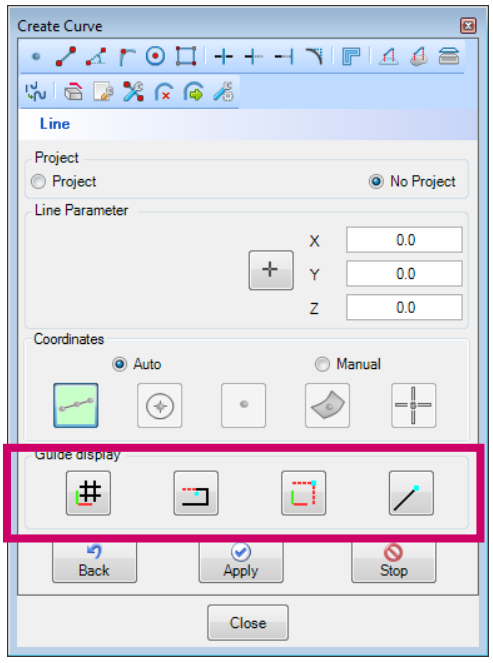


6. Addition of Guide Display Function at the Time of Curve Creation

The following functions have been added to facilitate creation of curves.

- Display a grid
- Display a restriction
- Display a distance
- Display a length

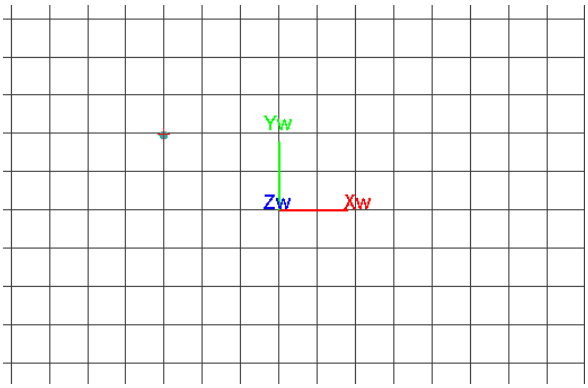
For the detailed description of each function and the operating instructions, refer to Chapter 22 "FFCAM Training Manual B".



(1) Display a Grid

Displays grids (lattice) that acts as reference for positioning end points when curves or vertex are created.

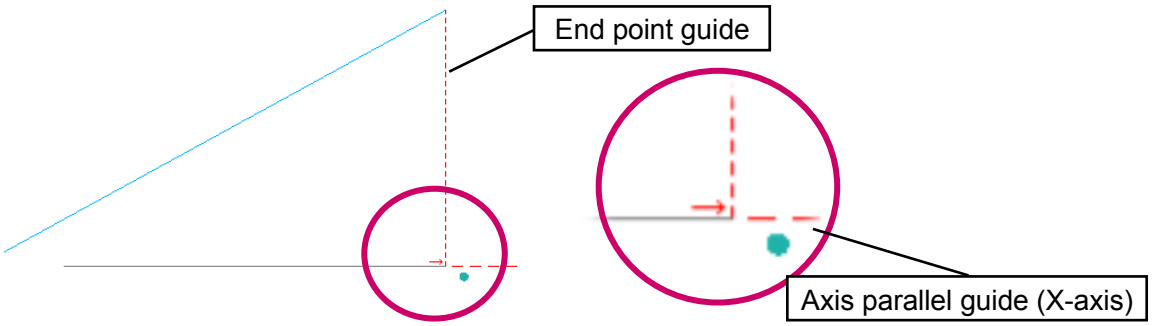
If you click near the intersection of grids, the end point (point) of the curve is created on the intersection of the grids.



(2) Display a Restriction

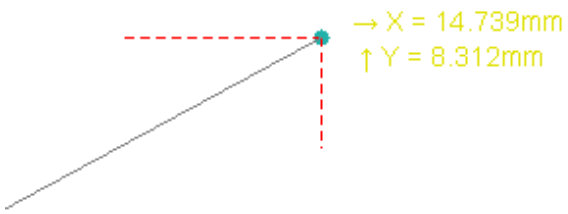
When you create a straight line, the following binding guide indicating the reference for drawing is displayed.

- Axis parallel guide
  Displays a guide parallel to the X-axis or Y-axis.
- Right angle guide
- Parallel guide
- End point guide



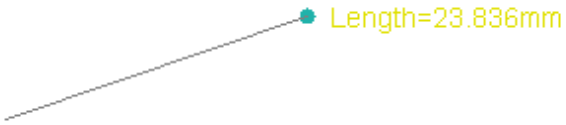
(3) Display a Distance

Displays the distance from the origin to verify the curve creation position.



(4) Display a Length

Displays the length of the straight line being created.



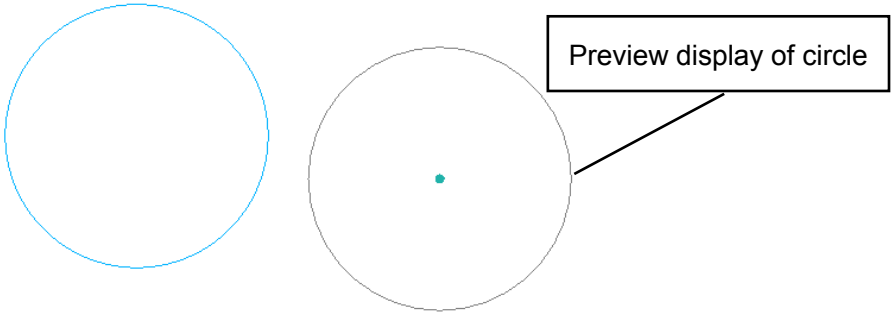
■ Available Commands

The Create Curve functions where the guide display can be used and the available guide types are as follows.

Function	Guide Display Type			
	Grid	Restriction	Distance	Length
Vertex	○			
Line	○	○	○	○
Angle line	○		○	○
Spline	○			
Circle	○		○	
Rectangle	○			

7. Circle Preview

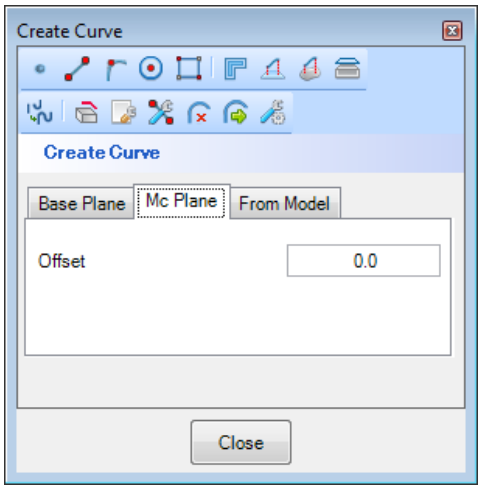
When you create a circle, if you enter a radius value before positioning circles, the preview of the circle that will be created is displayed.
The function allows you to easily imagine the location where the circle is to be arranged by viewing the size of the circle before positioning it.



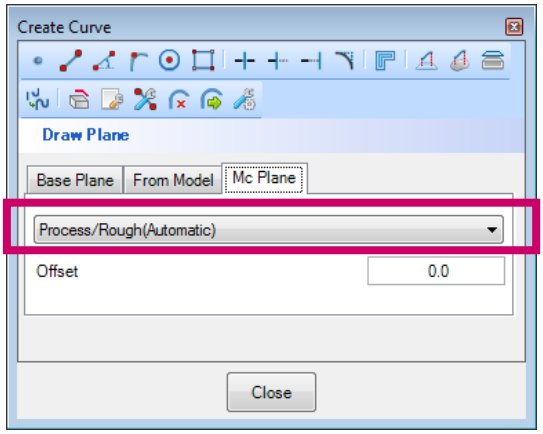
8. Changing the Plane Selection Screen During Curve Creation

The machining to determine the machining direction can now be selected from the [MC Plane] menu when selecting a plane to work for creating curves. In FFCAM 2017 or earlier, the machining direction is determined by the machining selected in the main window. In FFCAM 2018, you can select the machining that is to be used as the reference from the plane selection screen.

■ Setting Screen



FFCAM 2017



FFCAM 2018

(1) Machining Name List

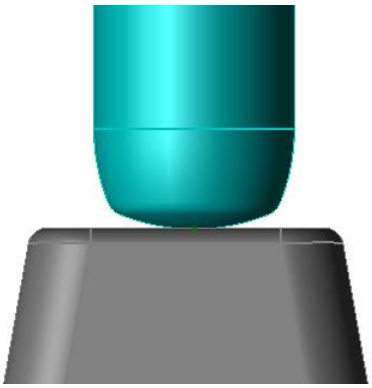
Select the machining direction to create a curve from the list.
The process name/machining name is displayed.

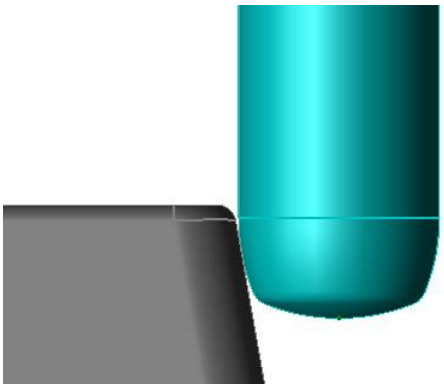
(2) Offset

Specify the distance from the machining origin of the selected machining. If you enter "0", a plane that passes the origin of the selected machining will be the reference plane. If you enter a non-zero value, the reference plane will move in the Z-axis direction by the specified value.

9. Lens barrel tool

Lens barrel tool is now supported.
The lens barrel tool has gentle R at the bottom and side surfaces. Therefore, in machining, the step can be set to a larger value than when a ball end mill with the same tool diameter is used, resulting in shorter machining time.

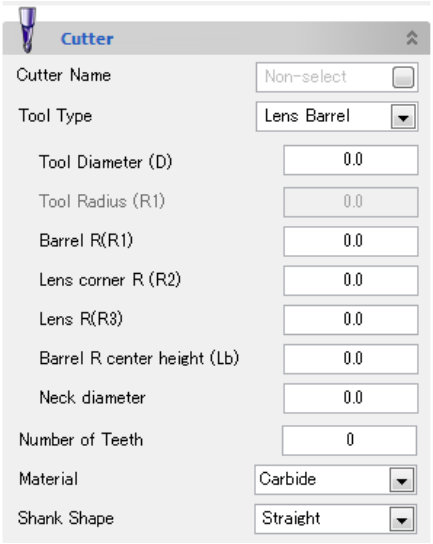


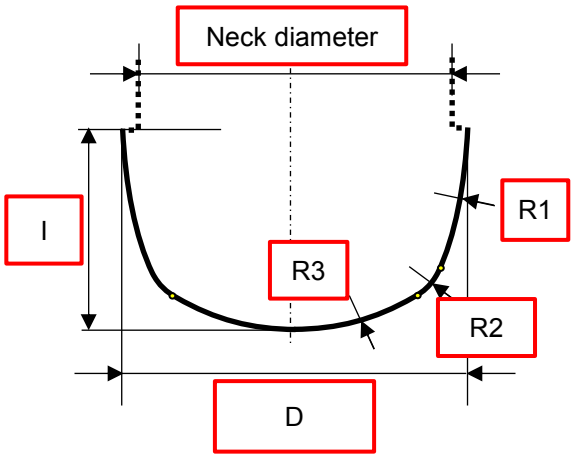


■ Setting Screen

(1) Tool Setting Screen

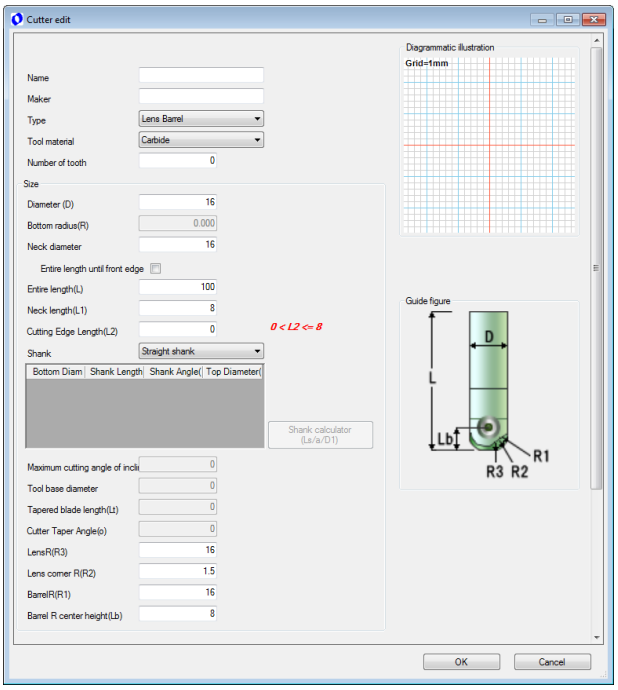
Selecting [Lens Barrel] in [Tool Type] displays the setting parameters for the lens barrel tool.





(2) Tool Database

Selecting [Lens Barrel] in [Type] displays the setting parameters for the lens barrel tool.



■ Note

(1) The available machining is as follows.

- Contour Machining
- Projection Machining

However, when "Core/Pocket Machining" or "5-axis simultaneous movement" is specified, you cannot use these machining.

(2) The parameters that are not available for Contour Machining are as follows.

- Z Spiral
- Along-Section
- Contour Face Cut Machining
- Thin Rib Machining

(3) The parameters that are not available for Contour Machining and Projection Machining are as follows.

- Specify Scallop Step
- Specify Tool Contact Point for Machining Area
- Stock Model Save/Input
- Overhang Length CL Division
- Optimization of Tool Blade Feed Rate
- Tool Usage Range Setting
- Indexed Angle Automatic Calculation Function

(4) NC data output is not possible for "Tool Center". NC data will be output with Tool Tip, even when [Tool Path Output] is set to [Tool Center Output] in [Machine Parameter].

10. Supports Multiple Taper Neck Stages

You can now set a tool shank up to 5 stages.
The tool shape that is closest to the actual tool can be registered, enabling a more accurate interference check of the shank.

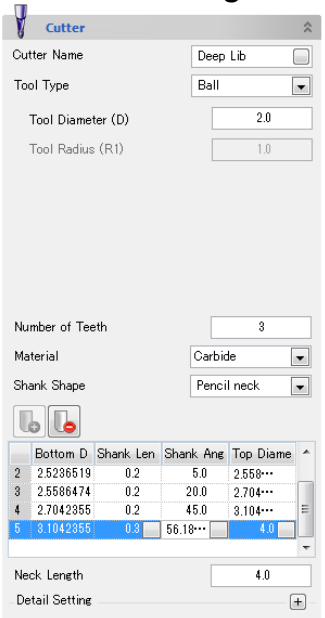
- **One-stage shank (conventional)**

- **Two-stage shank**

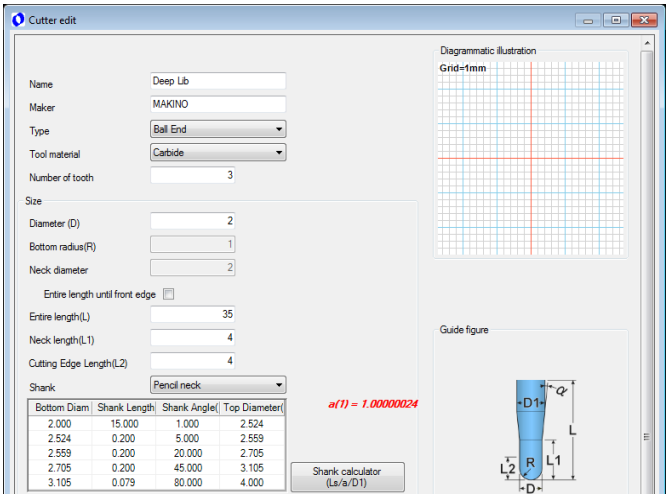
- **Five-stage shank**

■ Setting Screen

- **Tool Setting**



- **Tool Database**



(1)  (Adds a shank to the selected position.)

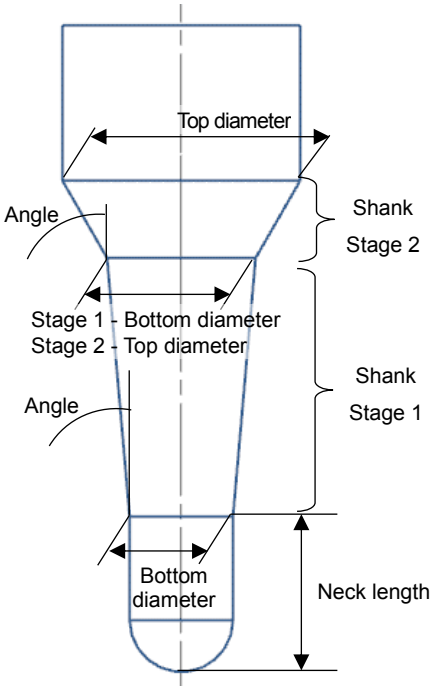
Adds a stage to the shank shape.
The stage is added below the line selected.

(2)  (Deletes a selected shank.)

Deletes a stage from the shank shape.
The line selected is deleted.
The line cannot be deleted if the shank has only one stage (one line).

(3) Shank Parameter

Set the dimensions of shank. One stage of the shank is set by a line of the setting parameter. Up to five stages of shank can be registered. □ button is displayed on the right side of each parameter. Enter any three of the [Bottom Diameter], [Shank Length], [Shank Angle], and [Top Diameter], and click the □ button of the remaining parameter, and the value for the remaining parameter is automatically calculated.

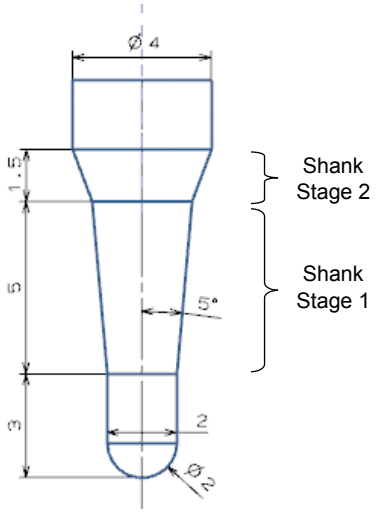


- Bottom diameter
  Set the diameter of the bottom side (tool tip side) of the shank shape.
  For the first stage of the shank, the tool diameter is automatically set.
  For the second and subsequent stages of the shank, the top diameter of the previous stage is automatically set.
  Set a value of the tool diameter, or a value larger than the top diameter of the previous stage.
- Length
  Set the length of the shank.
- Angle
  Set the angle for the shank shape.
- Top diameter
  Set the diameter of the top side (spindle) of the shank shape.

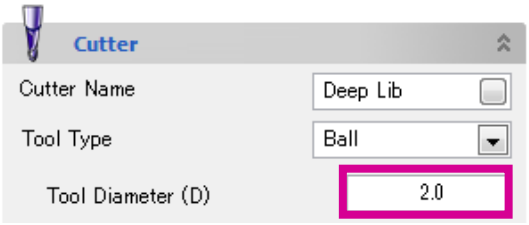
■ Setting Procedure

E.g. Registering the pencil neck tool for shank of multiple stages

The registration procedure for shanks is explained with the example of the following tool.



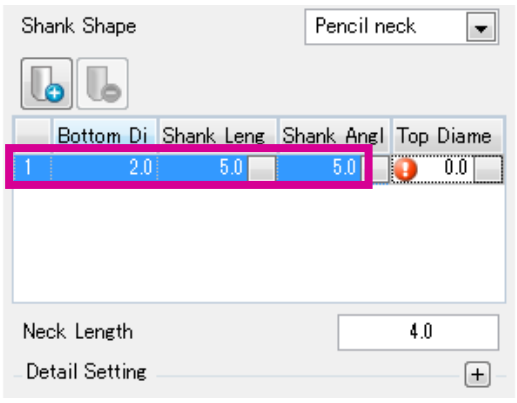
Operation Procedure:



Register the first stage.

1. Set the tool edge parameters as follows.

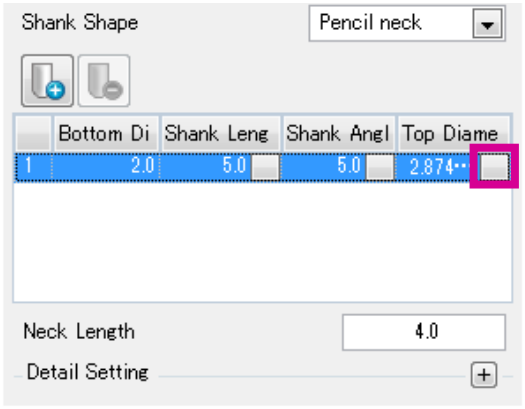
Tool diameter : 2.0
Number of edges : 2
Shank shape : Pencil neck
Neck length : 3.0



2. Configure the first stage holder as follows:

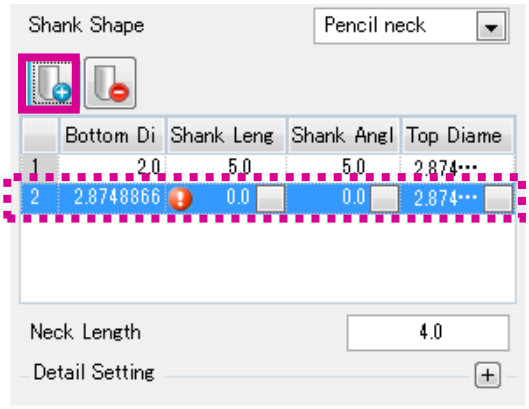
Bottom diameter : 2.0
Shank length : 5.0
Shank angle : 5.0

If there is any inconsistency in the input value, a mark is displayed.



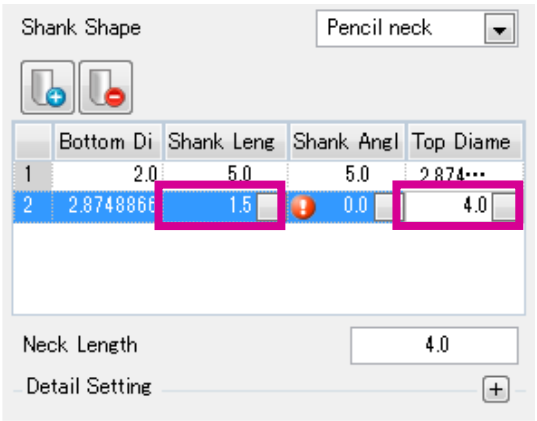
3. Click the □ button on the right side of the [Top Diameter] parameter.

The value in [Top Diameter] is automatically calculated.



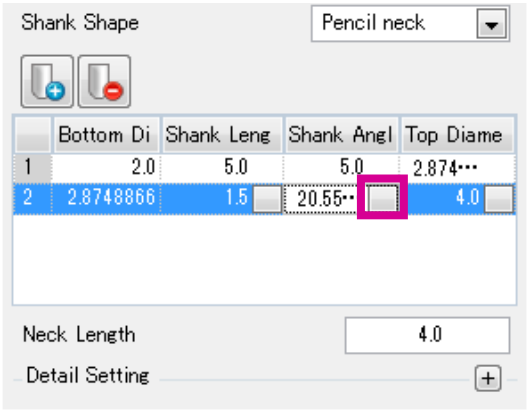
4. Click  (Adds a shank to the selected position.).

A stage will be added. The value of [Bottom Diameter] for the second stage is set to the same value as that of [Top Diameter] for the first stage.



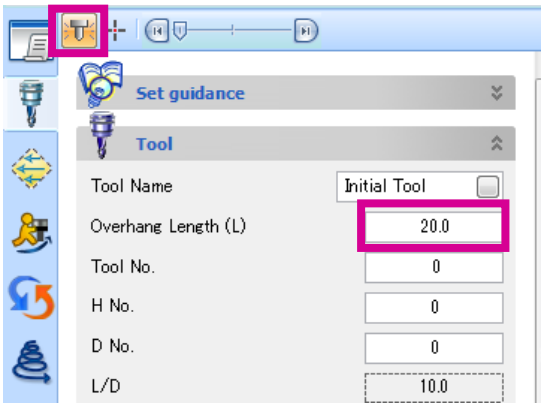
5. Configure the second stage holder as follows:

Shank length : 1.5
Top diameter : 4.0



6. Click the □ button on the right side of the [Shank Angle] parameter.

The value in [Shank Angle] is automatically calculated.



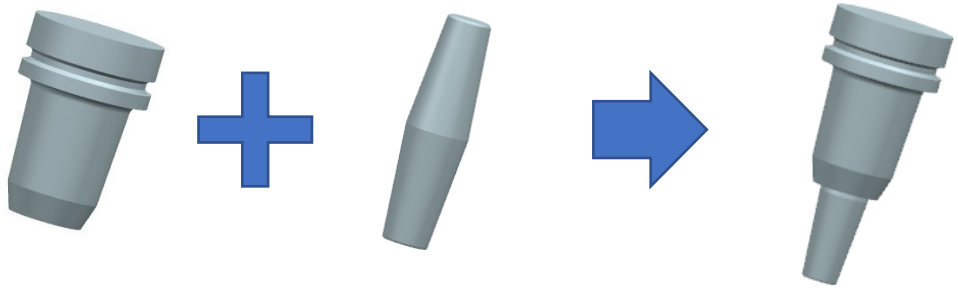
7. Enter "20.0" in [Overhang Length (L)].

8. Click  (Displays the tool shape on the screen) to check the tool shape.

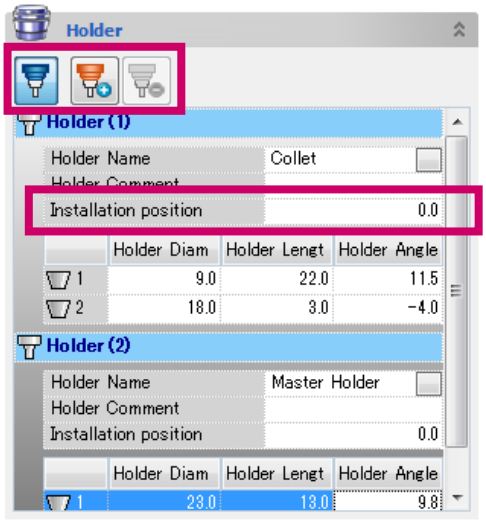
11. Support for Two Piece Holder

For the holder registration, the holder body and the collet can be used in combination, after separately registering them.
In FFCAM 2017 or earlier, the body and collet had to be registered together, and each combination of the body and collet had to be registered to the holder database. In FFCAM 2018, you can register the body and collet as separate holders, and you just have to register the bodies and collets owned by your company.

- **Body**
- **Collet**
- **Usage Image**



■ Setting Screen



(1)  (Switch enabled/disabled of the second holder.)

Toggles to enable or disable the second holder.

 Adds the second holder to allow the registration.

 Disables the second holder.

(2) (Add a step to the selected position.)

Adds a stage for the holder. A new stage will be added after the selected stage.

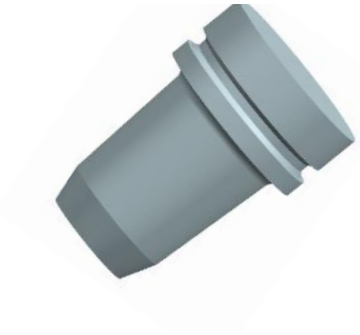
(3) (Delete a selected step.)

Deletes the selected stage.

(4) Installation Position

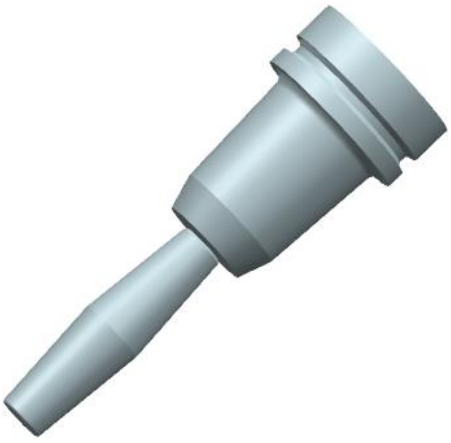
Set the position for combining the holder (body) and collet.
If you set a collet or holder up to the position which enters the part to be connected, specify the part to be connected and the connection position in [Installation position].

- **Body**

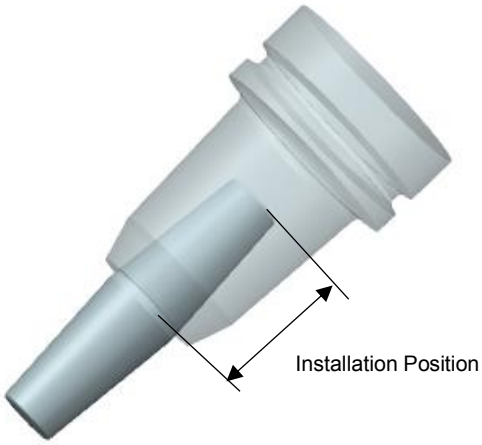


- **Collet**

- **When the installation position is 0**



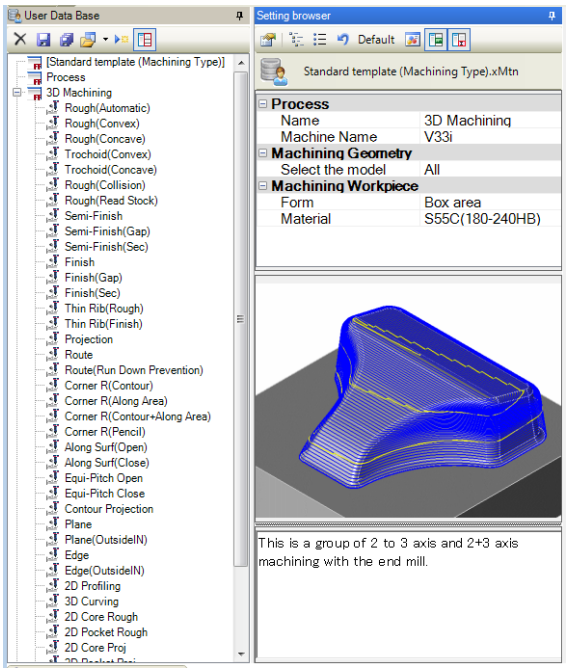
- **When a value is entered in the installation position.**



12. Improvement of the User Data Base Function

The display and function of User Data Base have been changed.
Switching between multiple user databases has now become faster. The explanations added to the machining in User Data Base makes it easier to understand the machining details of User Data Base and find an optimal machining method.

- Change in switching method in User Data Base
- Change in how to start the setting browser
- Explanations and pictures are added to the setting browser
- Changes in User Data Base

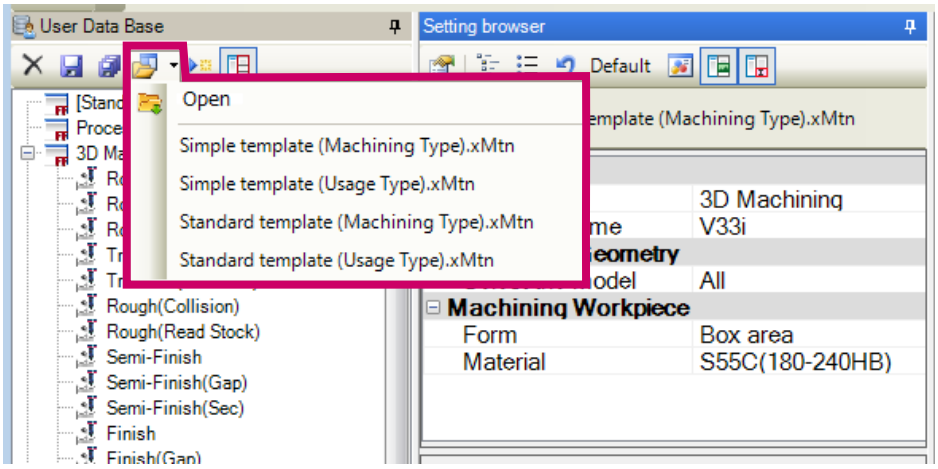


12.1. Change in Switching Method in User Data Base

You can now switch User Data Base from the drop down list, in addition to the previous file selection method.

The number of steps for required for switching is less than that of file selection, and you can use multiple User Data Bases by smoothly switching between them.

■ Setting Screen



■ How to Register to the List

The User Data Base can be displayed in the list by storing the User Data Base file into the following folder.

(C:¥MAKINO)¥FFCAM 2018 ¥ Template¥

* (C:¥MAKINO) is the installation folder of FFCAM 2018.

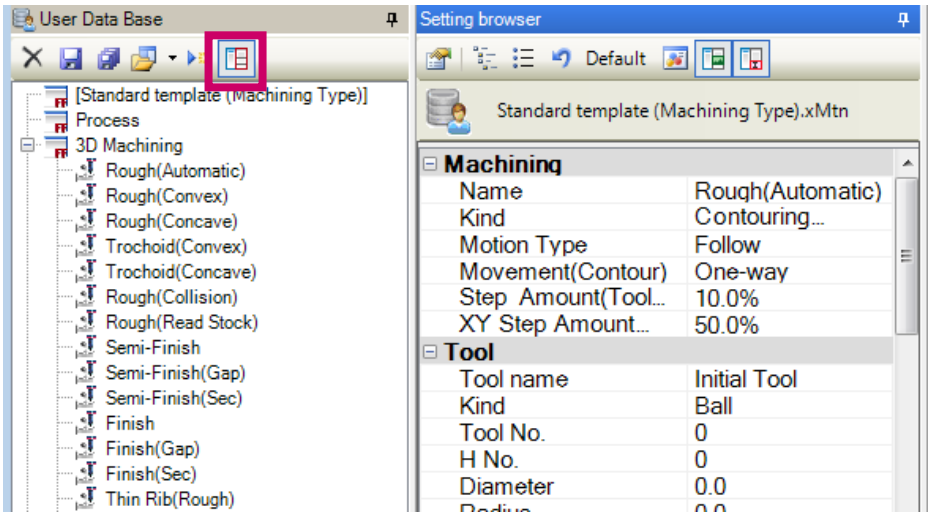
■ Note

The files that are saved in folders other than the Template folder are not listed. Select from [Open].

12.2. How to Start the Setting Browser

From FFCAM 2018, the setting browser display ON and OFF can be toggled by the setting browser icon. The setting browser is now displayed on the right side of the User Data Base. You can also move the setting browser to the previous position.

■ Setting Screen



(1)  (Setting Browser)

Displays the setting browser while the icon is in the pressed state.
Click it again to close the setting browser.

12.3. Explanations for Process and Machining of User Data Base

You can now register text descriptions and images to the process and machining registered in User Data Base. This enables the details of the process and machining in User Data Base to be understood more easily.

■ Setting Screen

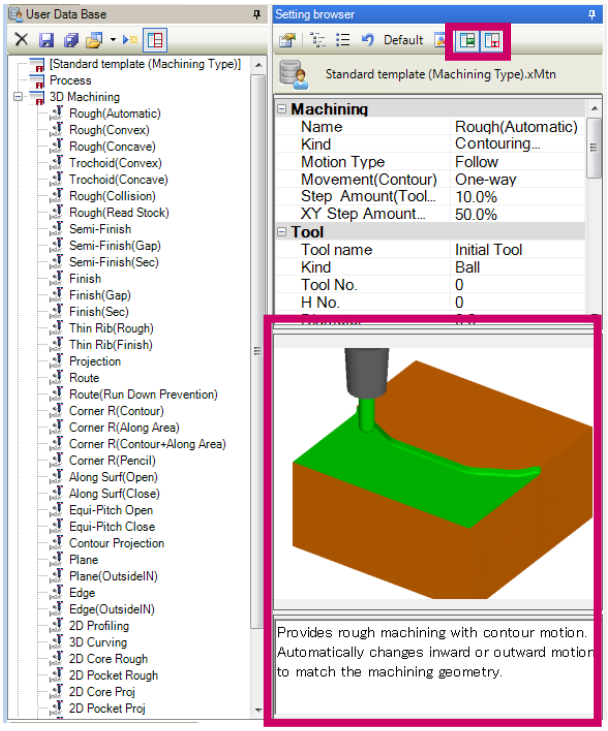


Image area

Comment area

(1)  Image

Displays or hides the image area.
When you click the icon, the icon is pressed and the image area is displayed. To hide the image area, click the icon again to release it.

- How to register images
  The following three methods can be used for registering images.
  (a) Select "Open..." displayed by right-clicking on the image area.
  Select an image file.
  (b) Select "Screenshot" displayed by right-clicking on the image area.
  Register the FFCAM graphic window as the image.
  (c) Drag and drop in Windows Explorer
  Drag and drop an image file to the image area.
- The file types that you can register
  The following file types can be registered in the image area.
  ・BMP
  ・JPG
  ・PNG
  ・GIF (including GIF animation files)

(2)  Comment

Displays or hides a comment area.
When you click the icon, the icon is pressed and the comment area is displayed. To hide the comment area, click the icon again to release it.

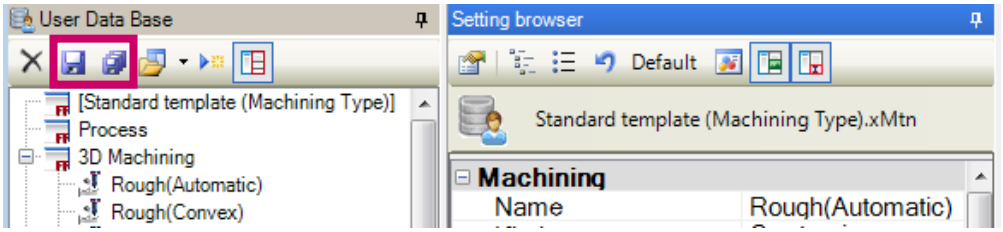
- How to register comments
  Directly enter the text in the comment area.

■ How to Set

Refer to "18.4 Setting Browser" in "FFCAM 2018 Training Manual B".

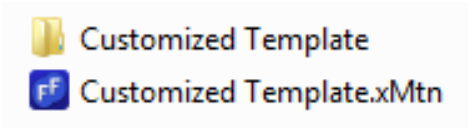
■ Note

(1) The settings for the image and comment areas are not automatically saved. Save the settings using the Save icon in User Data Base. Otherwise, the changed settings are discarded.



(2) Maximum file size for an image that can be registered in the image area is 3MB. Registering files of large sizes slows down the display speed of the setting browser.

(3) The contents of the image and comment areas are saved in the same folder as that of the User Data Base files. Create a folder with the same name as User Data Base, and save the images and comments in the folder.
When copying User Data Base to other FFCAM, the folder where the images and comments are saved should also be copied together.



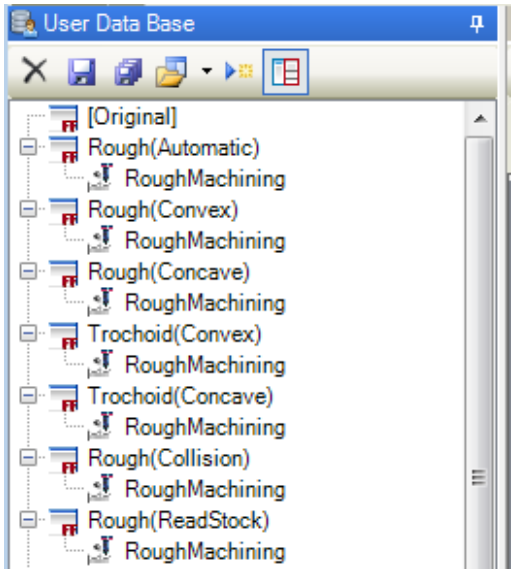
The folder in which images and comments are saved

User Data Base file

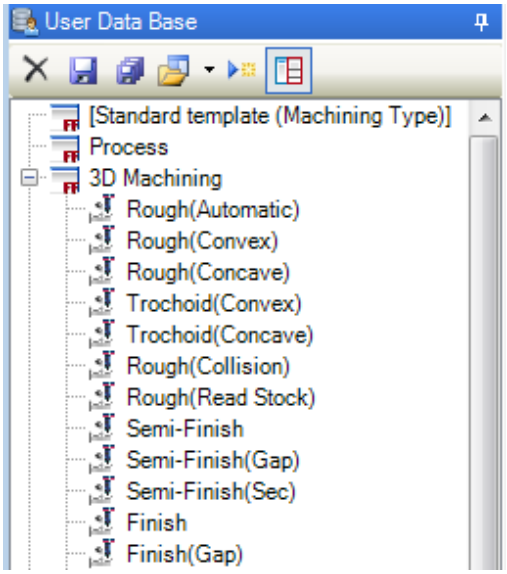
12.4. Changes in User Data Base

The layout of the process and machining in User Data Base has been changed.
In FFCAM 2017 and earlier, the process and machining are registered together in User Data Base. Therefore, to add machining, you were required to open the process tree before selecting the machining.
In FFCAM 2018, since the process and machining are registered separately, you can select and add machining directly without opening the tree.

■ User Data Base



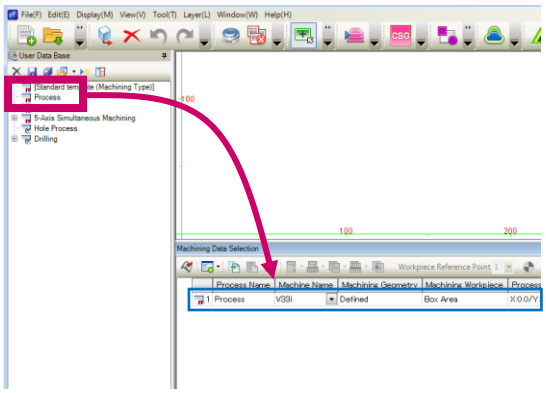
FFCAM 2017



FFCAM 2018

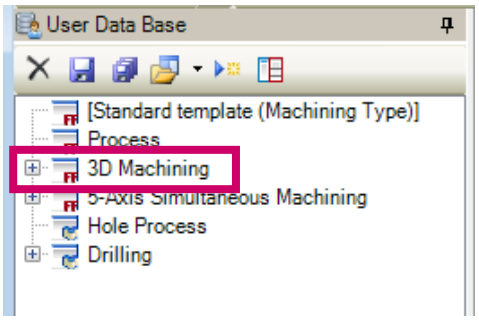
■ Operation of New User Data Base

Operation Procedure:



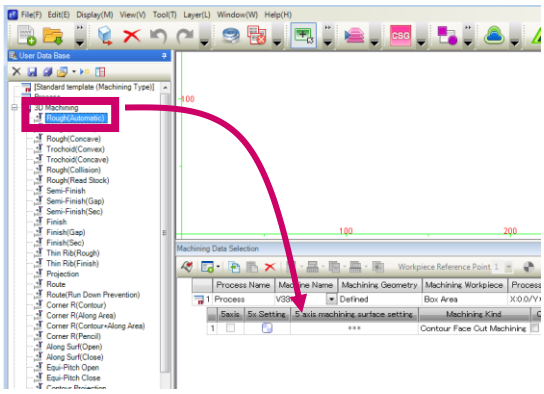
1. Select [Process] from [User Data Base], and drag and drop it to the main window.

The process is copied.



2. Click ⊞ on the left side of [3D Machining].

This will expand the tree of [3D Machining].



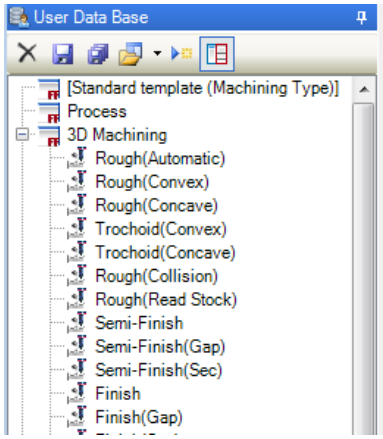
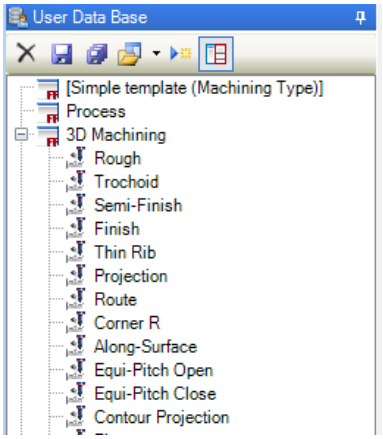
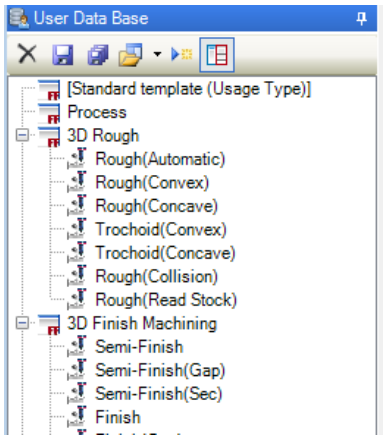
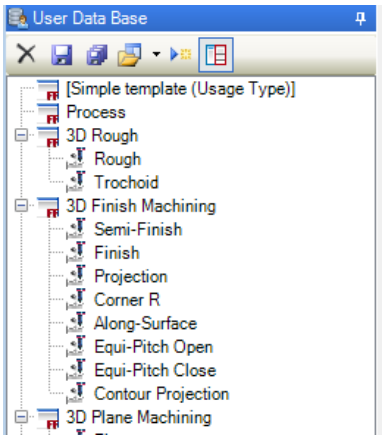
3. Select the machining that you want to register to the process in Step 1, and drag and drop it onto [Process] in the main window.

The process will be added.

4. To add more machining, repeat Step 3.

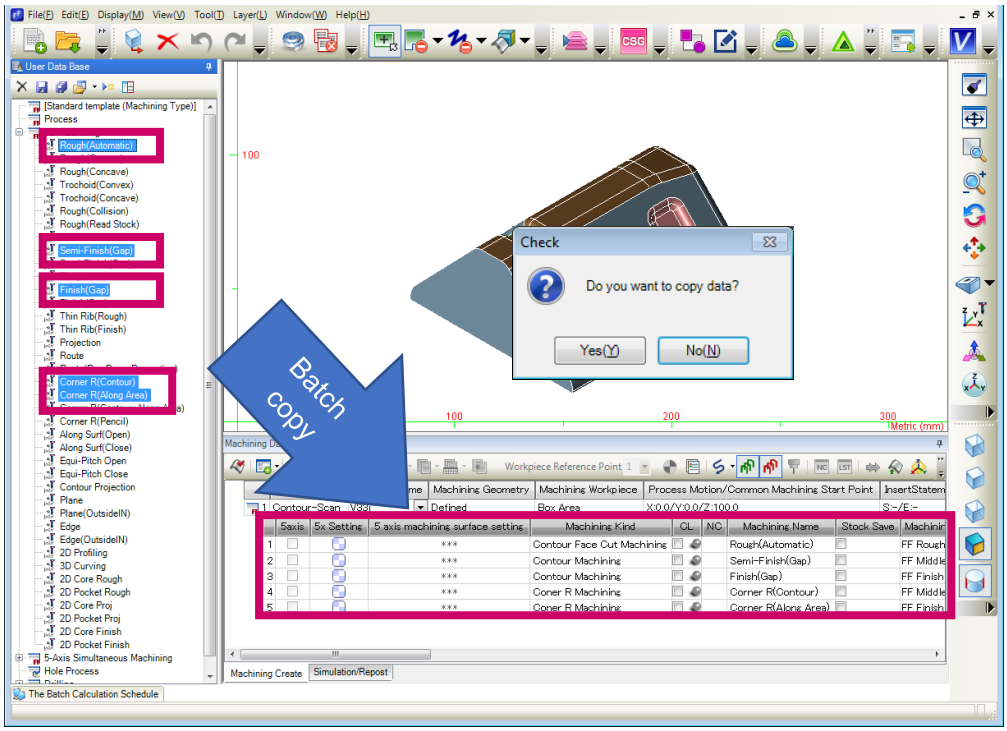
■ Types of User Data Base

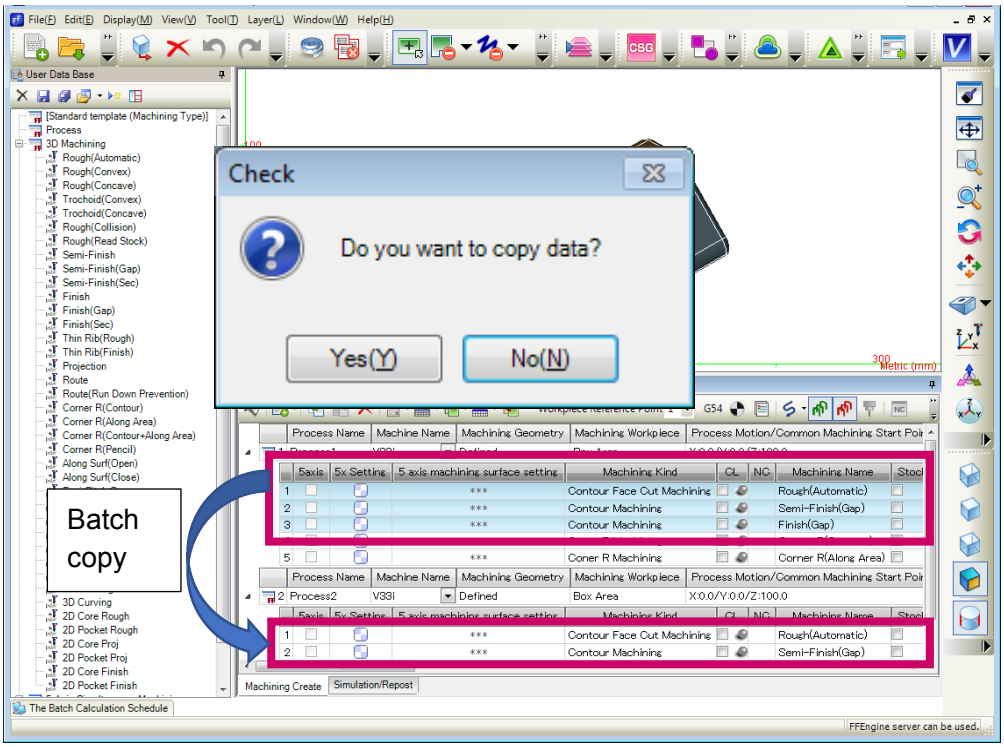
There are four types of User Data Base due to variation in machining grouping in User Data Base.

	User Data Base where machining is divided by changing the machining parameters	User Data Base where machining is not divided by the machining parameter
Grouped by the machining types		
Grouped by the tool movements		

13. Batch copy (move) of multiple machining

The batch copy (move) of multiple machining and processes is now available by operations in User Data Base and in the main window.



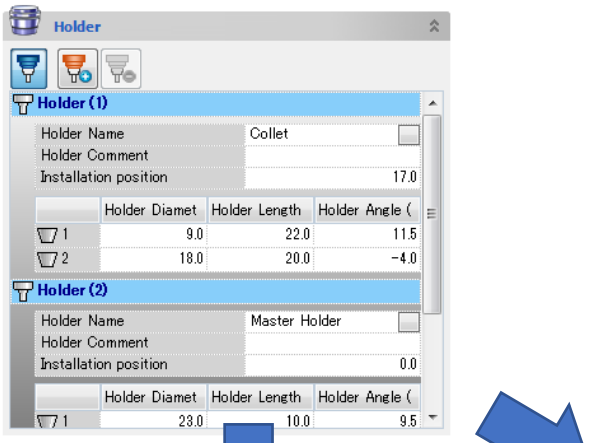


14. Information Added to Machining Instructions

With the addition of the tool and holder functions, information on these functions is now output to the machining instructions (lst, tcsv, and html).
The information on the following parameters are output.

- Information on taper tools
- Information on lens barrel tools
- Information on shank of multiple stages
- Information on two piece holders

● Example of Outputs: Two Piece Holders




```
*** TOOL LIST ***
Machining Name      T-No TOOL NAME          TOOL COMMENT
Rough(Automatic)    1    Initial Tool
T-No TOOL KIND              DIA   NOSE-R LENGTH   OUTPUT   EDGE D   H   MAT.
1    BALL EM                2.00  1.00  20.00 Head     2  1  1  Carbide
T    SHANK SHAPE    NECK LENG   SHANK DIA   SHANK TAPER ANG
1    STRAIGHT SHANK 20.00       2.00        0.00
HOLDER
T 1  HOLDER1 Name : Collet
                DIA    LENG   ANG     Installation Position : 17.0
          1   9.00   22.00  11.50
          2  18.00   20.00  -4.00
       HOLDER2 Name : Master Holder
                DIA    LENG   ANG     Installation Position : 0.0
          1  23.00   13.00   9.80
          2  27.50   28.00   6.50
          3  40.00   25.00   0.00
          4  30.00   35.00  -3.00
```

lst file

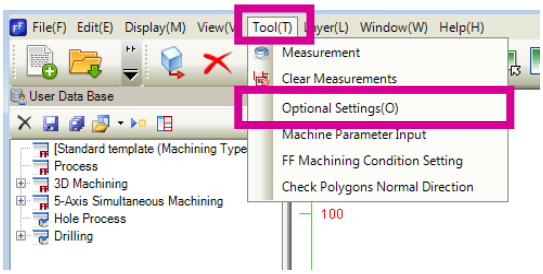
Shank1 Top Diameter	2.00 mm
Holder Name	Collet
Holder Comment	
Holder Installation Position	17.00 mm
Holder Dia1	9.00 mm
Holder Leng1	22.00 mm
Holder Ang1	11.50 degree
Holder Dia2	18.00 mm
Holder Leng2	20.00 mm
Holder Ang2	-4.00 degree
Holder2 Name	Master Holder
Holder2 Comment	
Holder2 Installation Position	0.00 mm
Holder2 Dia1	23.00 mm
Holder2 Leng1	13.00 mm
Holder2 Ang1	9.80 degree
Holder2 Dia2	27.50 mm
Holder2 Leng2	28.00 mm
Holder2 Ang2	6.50 degree
Holder2 Dia3	40.00 mm
Holder2 Leng3	25.00 mm
Holder2 Ang3	0.00 degree
Holder2 Dia4	30.00 mm
Holder2 Leng4	35.00 mm
Holder2 Ang4	-3.00 degree
Cutting Dist	951.4224 m

tcsv file

■ How to Output in CSV and HTML Files

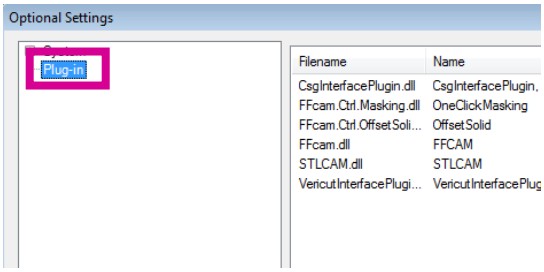
The parameters added by FFCAM 2018 are all output to a tcsv file, but not to a csv file by default. To output the parameters in a csv file, set the following setting items.

Operation Procedure:

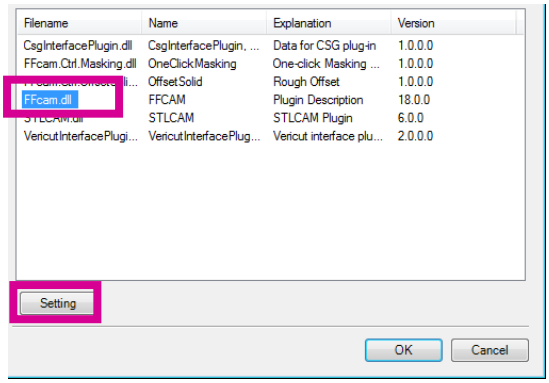


1. Select [Tool] → [Optional Settings] on the menu bar.

[Optional Settings] screen is displayed.

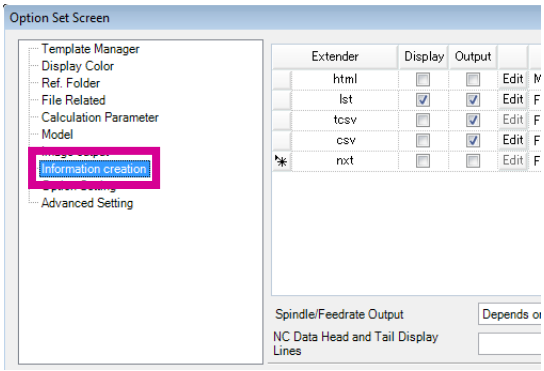


2. Select [Plug-in] from the setting item tree.



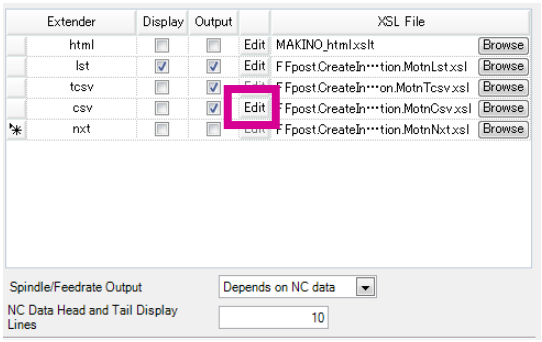
3. Select [FFcam.dll] from the plug-in list, and click the [Setting] button.

[Option Set Screen] is opened.



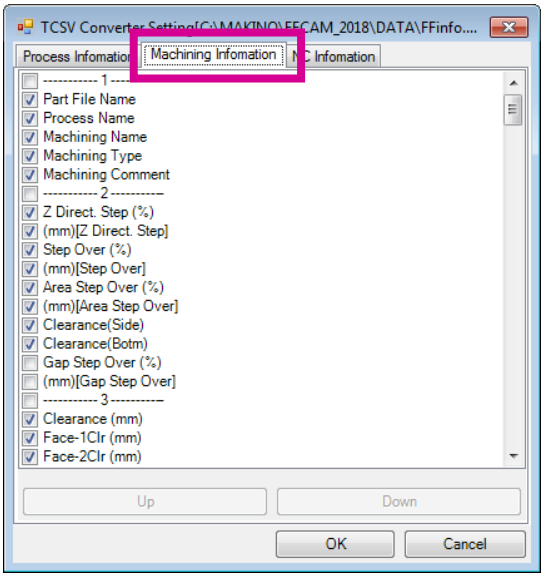
4. Select [Information creation] from the setting item tree.

● How to Output in CSV file

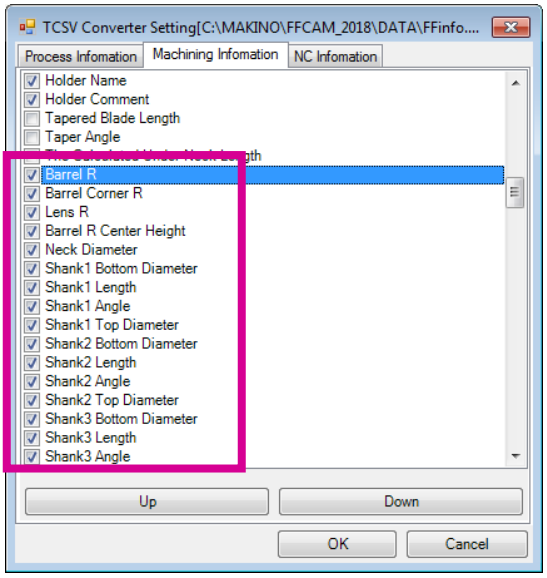


5. Click the [Edit] button on the csv line.

The [TCSV Converter Setting] screen opens.



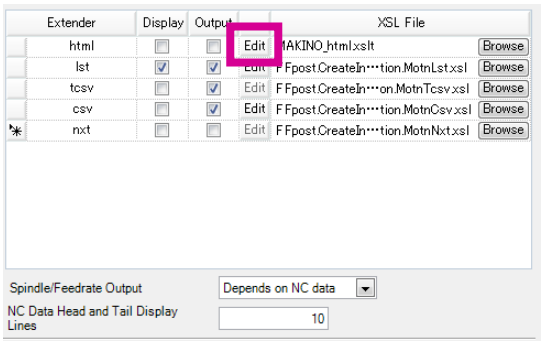
6. Select the [Machining Information] tab.



7. From the list, select the item that you want to output in a csv file, and tick the box ☑.

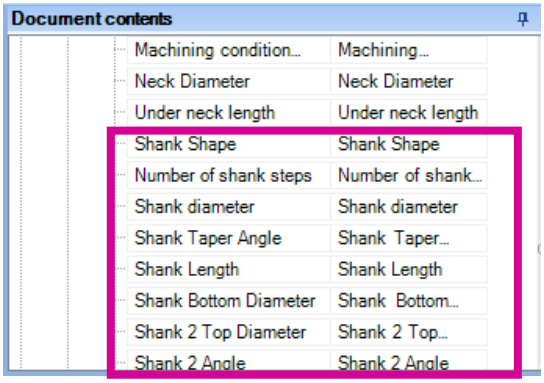
8. Click [OK] to close all the screens.

● How to Output in HTML file



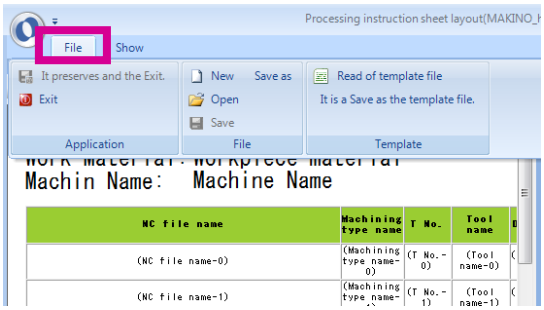
9. Click the [Edit] button on the HTML line.

The [Layout of Machining Instructions] screen opens.



10. From [Document contents], select [Process] → [APT-CL] → [Machining].

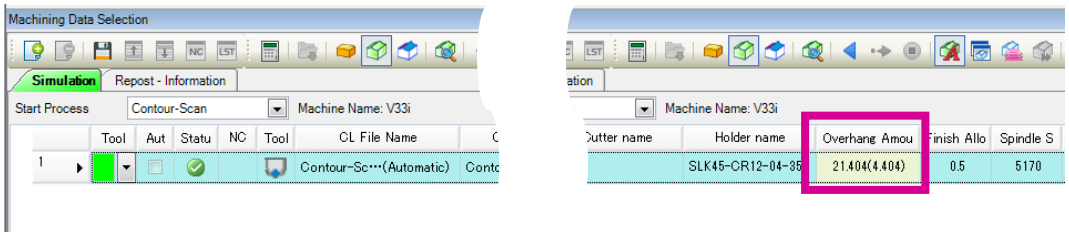
11. Arrange the items that you want to output in a HTML file on the [Layout] window.



12. Open the [File] tab to save the setting.

15. Output of Calculated Neck Length

The values of the neck length calculated by Overhang Length Calculate in Simulation can now be output in the machining instructions (tcsv and HTML).



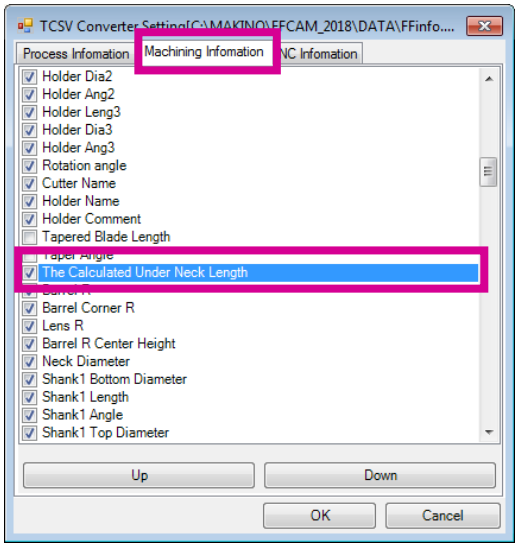
*** PROCESS LIST ***		
Machining Infomation (1)		
Part File Name	ContourAndScan	
Process Name	Contour-Scan	
Machining Name	Rough(Automatic)	
Machining Type	CONTOUR(FOLLOW)	
Machining Comment		
Clearance	0.50000 mm	
Face-1(Side)Clr	0.00000 mm	
Face-2(Bottom)Clr	0.00000 mm	
Upper Clr	0.00000 mm	
Z Direct. Step	Tool Ratio 10.00000 (0.20000 mm)	
Step Over	Tool Ratio 50.00000 (1.00000 mm)	
Tolerance(In/Out)	0.01000 mm/0.01000 mm	
Tool Type	BALL ENDMILL	
Diameter	2.00000 mm	
Nose-R	1.00000 mm	
Tool Len	20.00000 mm	
Overhang	21.40350 mm	
The Calculated Under Neck Length	4.40350 mm	
T no	1	
D no	1	

■ How to Output

After executing Overhang Length Calculate in Simulation, create CSV and HTML files using the post process or Create Machining Information Function.

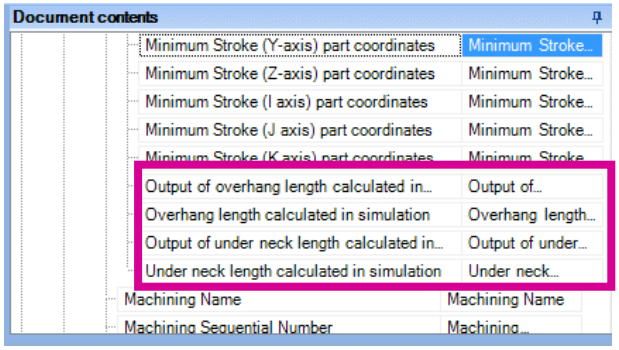
● How to Output in CSV file

Open [TCSV Converter Setting] screen and add the file from [Machining Information] tab.

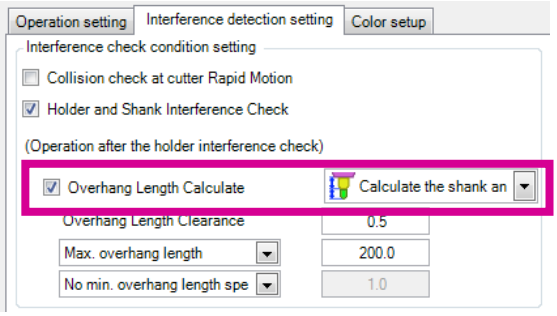


● How to Output in HTML file

Open the [Layout of Machining Instructions] screen and from [Document Contents], select [Process] → [APT-CL] → [Machining] → [Machining/Tool Information].
Select the following four items.



- Output of the overhang length calculated in Simulation
  Outputs information on whether the overhang length is calculated.
  Outputs "true" if the overhang length is calculated, and outputs "false" if the overhang length is not calculated.



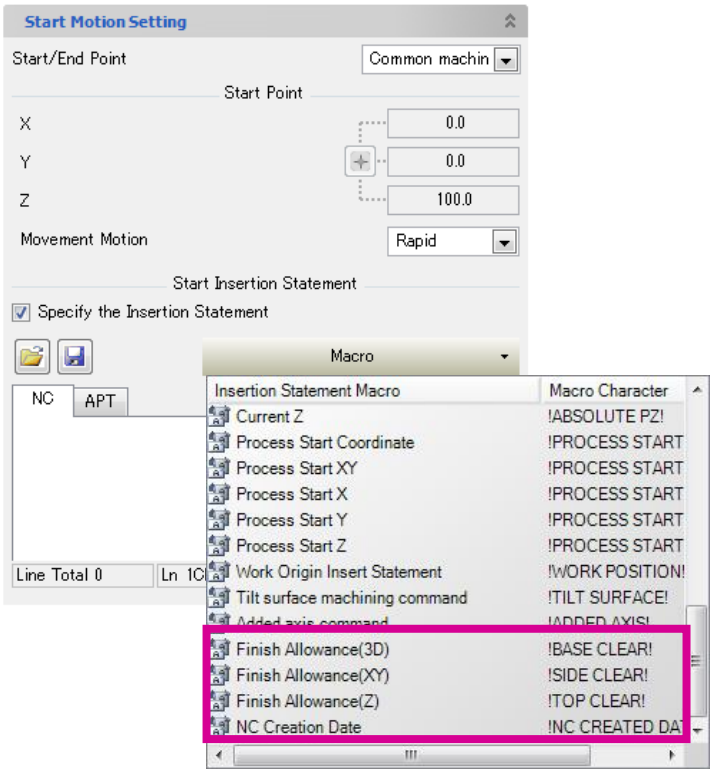
- Overhang length calculated in Simulation
  Outputs the overhang length calculated in the simulation.
  If the overhang length is not calculated, the overhang length registered for the tool parameter is output.
- Output of the neck length calculated in Simulation
  Outputs the information on whether the neck length is calculated.
  Outputs "true" if the neck length is calculated, and outputs "false" if the neck length is not calculated.
- Neck length calculated in Simulation
  Outputs the neck length calculated in the simulation.
  Outputs "-1" if the neck length is not calculated.

16. Addition of Insertion Statement Macros

The following information is added to the macro insert statements.

- NC creation date and time
- Finishing allowance 3D direction
- Finishing allowance XY direction
- Finishing allowance Z direction

■ Setting Screen



You can set from Process Start/End Insert Statement and Machining Start/End Insert Statement.

■ Macro Setting Position and Output Results

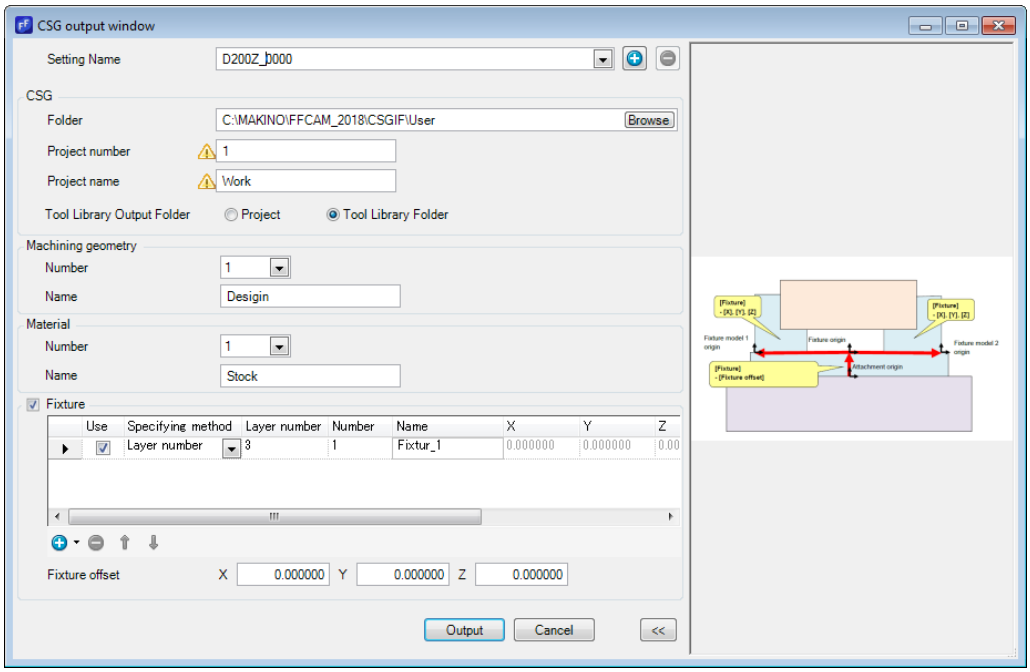
Macro name	Macro character	Position where macros can be set	Output result (example)
NC Creation Date	!NC CREATED DATE!	Process Start/End	2018/01/31 08:43:17
Finishing Allowance(3D)	!BASE CLEAR!	Process Start/End Machining Start/End	0.5
Finishing Allowance(XY)	!SIDE CLEAR!		0.02
Finishing Allowance(Z)	!TOP CLEAR!		0.02

17. Improvement of Collision Safeguard Data Output Function

The following functions have been added for the collision safeguard data output function (herein after referred to as CSG). In FFCAM 2017 and earlier, some operations were needed on the machine side when setting the CSG data output function or to output data in CSG. FFCAM 2018 can carry out the setup of CSG data output function and data output without operations on the machine side.

- Support for Professional 6
- Addition of tool library output target
- Addition of function for setting numbers and names of materials
- Addition of function for adding numbers and names to fixtures

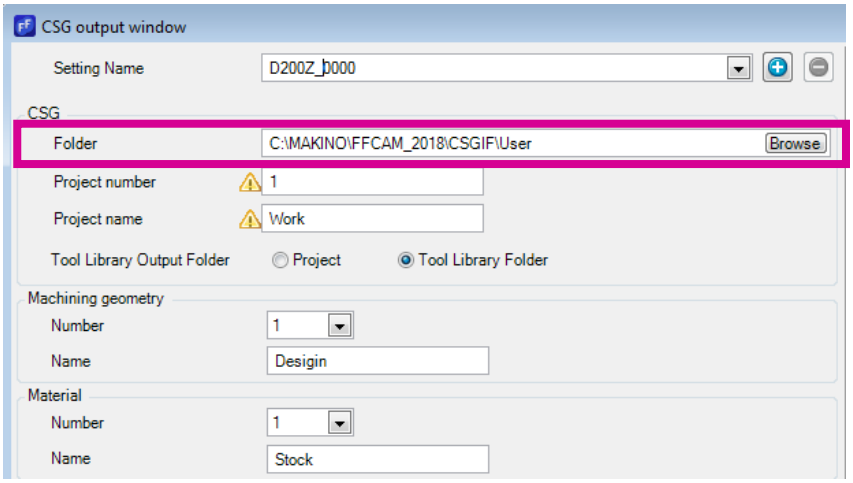
■ Setting Screen



17.1. Support for Professional 6

When the controller of machining is Professional 6, the CSG data output function can be set up only by operation of CSG output screen.

■ Setting Screen



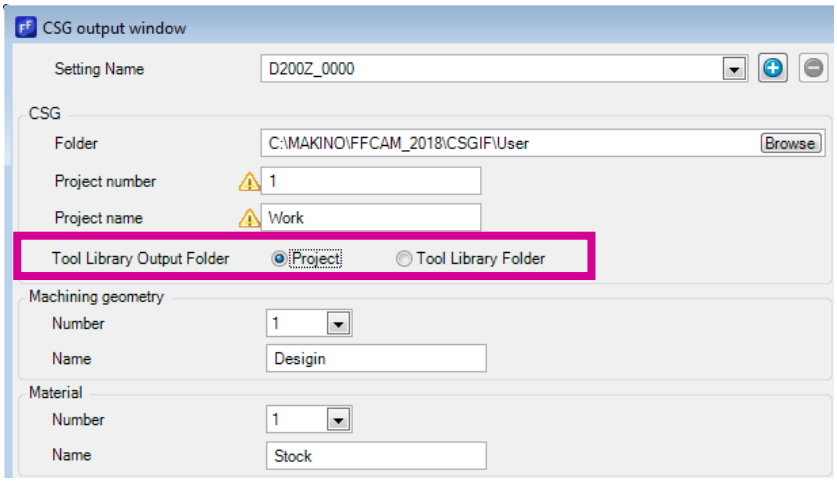
(1) Folder

- For Professional 6
  Specify the following folder regardless of the type of machine.
  (C:¥MAKINO)¥FFCAM_2018¥CSGIF¥User¥
  * (C:¥MAKINO) is the installation folder of FFCAM 2018.
- For Professional 5
  Specify the folder in which "Import.vbs" is saved after installing CSG in your FFCAM-installed PC.
  E.g.: C:¥MAKINO_CSG¥MachineName_MachineNumber¥User¥

17.2. Addition of Tool Library Output Target

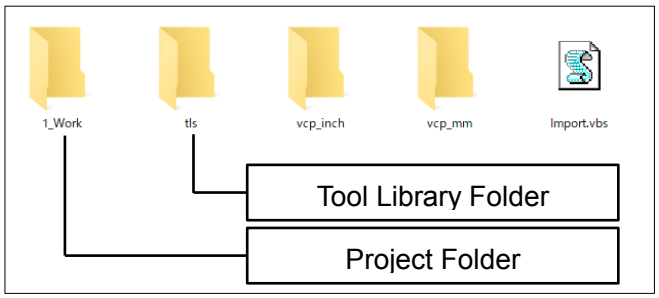
Output folder of CSG tool library can be specified now.
Output folder can be selected from the tool library folder or project folder.

■ Setting Screen



(1) Tool Library Output Folder

Specify the folder to which tool files (~.tls) are output.



- Project
  Tool library files are output to the same place as that of the project files.
  The project folder is created in the following place.
- Tool Library Folder
  Tool library files are output to the "tls" folder.

■ Note

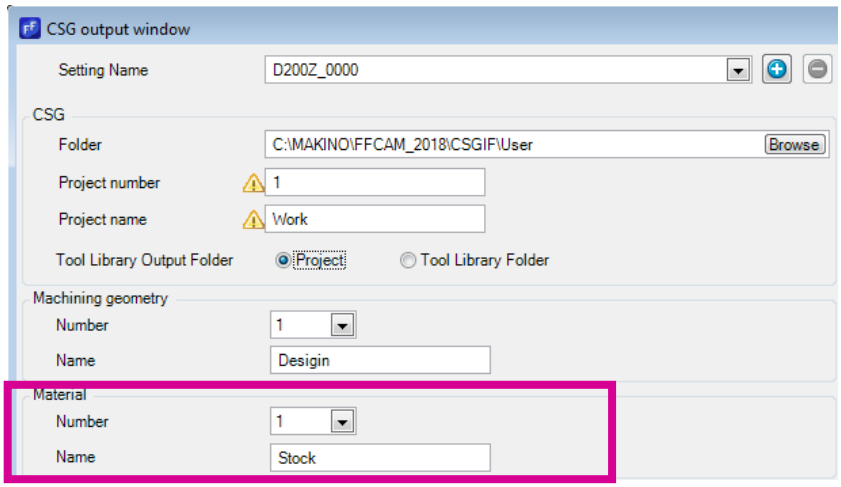
The output destination of tool library files can be selected only if the CSG version is 15.12.24 or greater. For the versions prior to 15.12.24, specify [Tool Library Folder] for [Tool Library Output Folder].

17.3. Addition of Function for Setting Numbers and Names to Materials

Machining workpiece (material) can be output to CSG after allocating the number and name to it.

Allocating numbers eliminates the need for editing on the machine side after being output to CSG.

■ Setting Screen



(1) Material

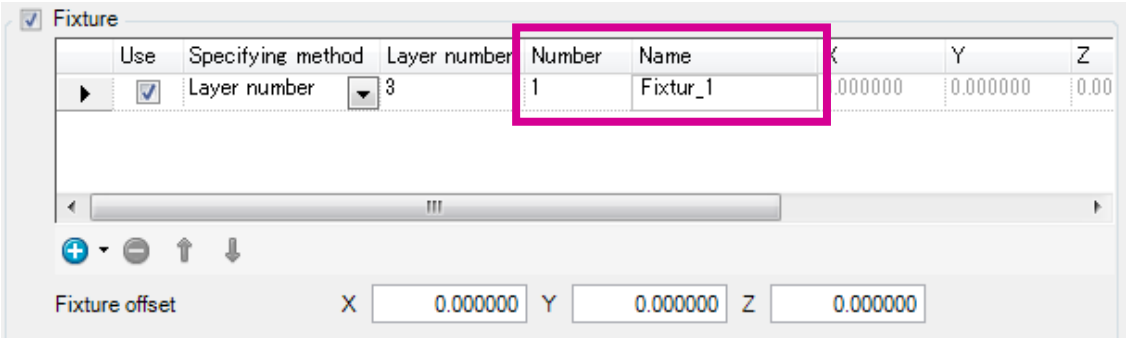
Used to output the machining workpiece in CSG. The output machining workpiece is registered as [Stock] in CSG.

- Number
  The number to be used for the position alignment function of models. Specify any number from 1 to 9.
  Specify "- (hyphen)" if you don't want to move the model for the position alignment function of models.
- Name
  The name to be displayed in CSG screen. You can omit the name.

17.4. Addition of Function for Adding Numbers and Names to Fixtures

In FFCAM 2017, the [Comment] area was used for registering the numbers and names of fixtures. In FFCAM 2018, parameters has been added to allow [Number] and [Name] to be entered respectively.

■ Setting Screen



(1) Fixture

Used to output the fixture model in CSG. The output fixture model is registered as [Fixture] in CSG.

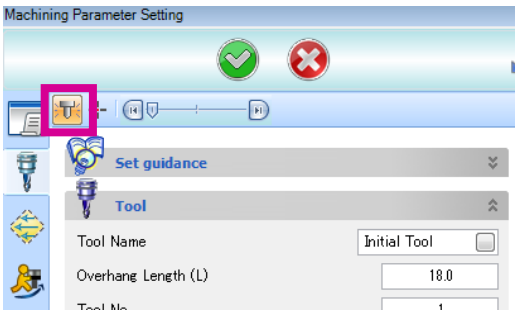
- Number
  The number to be used for the position alignment function of models. Specify any number from 1 to 9.
  Specify "- (hyphen)" if you don't want to move the model for the position alignment function of models.
- Name
  The name to be displayed in CSG screen. You can omit the name.

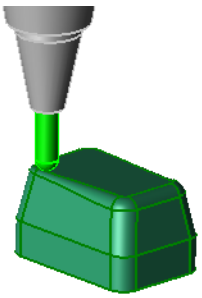
18. Changes in Behavior of Tool Drawing Icon

On the [Machining Parameter Setting] screen, the state for drawing tool shapes on the screen is now the default option.
The screen with tool drawing is started by default even if the tool drawing icon is not clicked on the tool selection screen or machining direction setting screen.
In addition, since the ON/OFF state of the tool drawing is saved when the screen is closed, frequent switching of the tool drawing state is no longer necessary.

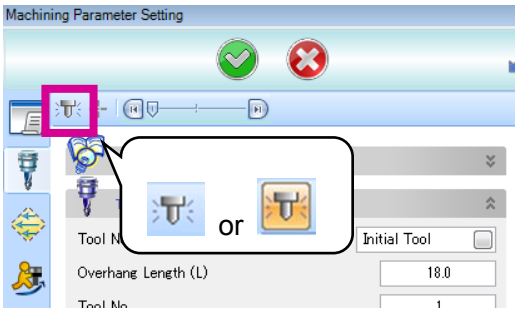
■ Behavior of Tool Drawing Icon

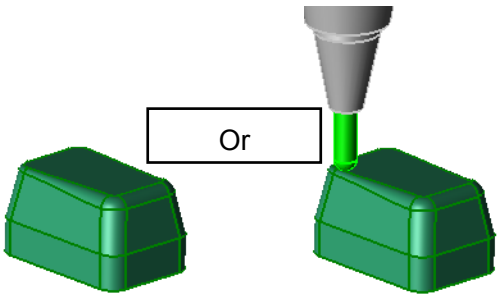
- After starting FFCAM, when [Machining Parameter Setting] screen is opened for the first time, the tool drawing icon is pressed and tool drawing turns ON.





- When the [Machining Parameter Setting] screen is opened the next time, the [Machining Parameter Setting] screen opens with the same state as that of the previous tool drawing ON/OFF.

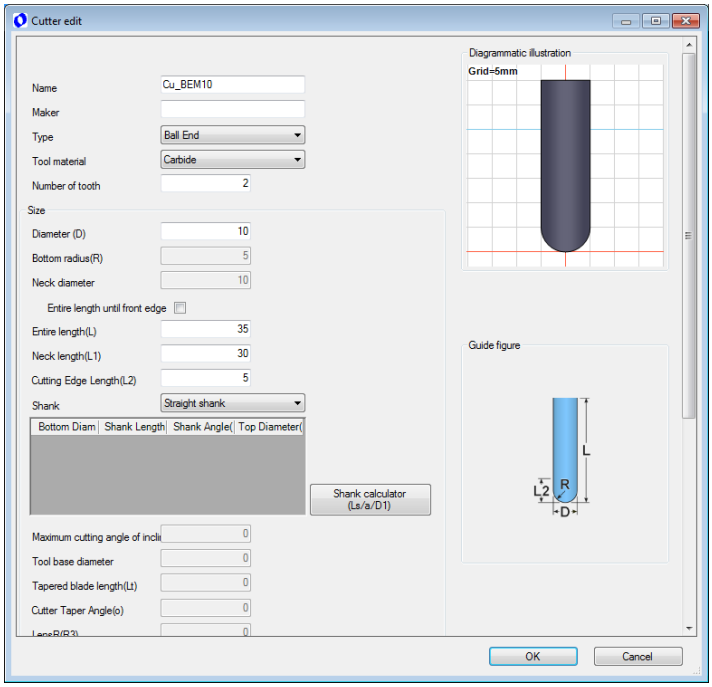




19. Adjustment of Screen Size of Tool DB Maintenance

The height of the [Maintenance - [Tool database]] screen can be changed.
The scroll bar is displayed depending on the changes made to the height.
The size can be changed from the following screens.

- [Tool edit] screen
- [Cutter edit] screen
- [Holder edit] screen



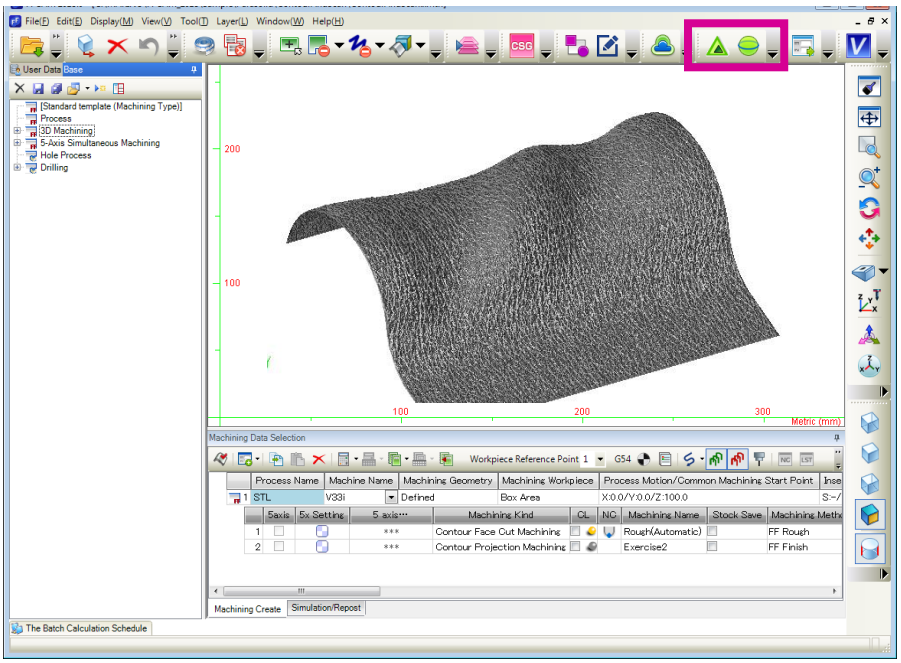
20. Standardized STL Model Machining Function

The "STL Model Machining function", which was sold as option in FFCAM 2018 and earlier, is now available as a standard function.

■ STL Model Machining Function

This is a function that allows you to output NC data from the polygon (STL data) model created from computer graphics, image data (photograph), and digitizing (actual object). Since the function can load STL data without converting to CAM data, the design shapes can be reproduced accurately. In addition, the function enables high-speed processing of large data and minute shape data processing, which cannot be processed by other STL data capable of handling CAM.

■ Function Specific to STL Model Machining Function



(1) Load STL Model

Loads STL data into FFCAM.
The STL data in both the text and binary format can be loaded.

(2) STL Area Line Creating

Extracts borders of the under parts, standing walls, and free edges of STL solids in the view direction, and creates the area curve.

21. 5-axis Simultaneous Support for Hairline Finish Machining Function

The "Hairline Finish" machining function supports 5-Axis Simultaneous Machining.
The "Hairline Finish" machining function is an optional software.

■ Hairline Finish

Hairline machining is machining to add hair-like fine stripe patterns on the surface of stainless steel or aluminum. To add metallic texture to plastic products, the application of hairline machining for molds has been increasing.
The hairline finish machining function is used to edit the tool path output from FFCAM and create the NC program for hairline machining.