FFCAM 2017

New Function Manual



Copyright © 2017 MAKINO MILLING MACHINE CO.,LTD All rights reserved.

Preface

This manual describes the functions added to MAKINO FFCAM 2017 and how to use them.

Created on

April 2017

List of Added Functions in FFCAM 2017

| 1. Addition of the Contour Face Cut Machining/High Feed Machining Function | 1 |
|--|----|
| 2. Addition of the Corner R Machining/ Groove Path Restriction Function | 6 |
| 3. Addition of Contour Projection Machining / Bottom Corner R Function | |
| 4. Addition of Flat Face Machining/Round Insert cutter Interference Check Function | 13 |
| 5. Addition of Route Machining / Zig-zag Drive-In Function | |
| 6. Addition of 2D Route Machining/ Zig-zag Drive-In Function | 16 |
| 7. Heidenhain Canned Cycle Support | |
| 7.1 Drilling | |
| 7.2 Thread Cutting | |
| 7.3 Chamfer Machining | |
| 7.4 Spot Face Machining | 24 |
| 8. Addition of Machining Name Consecutive Number Renaming Function | 25 |
| 8.1 Machining Name Consecutive Number Renaming | 25 |
| 8.2 Output File Name Consecutive Number Renaming | |
| 9. STEP Import Layer Compatibility | |
| 10. Improvement of CL Edit and Operability | |
| 11. Addition of CL Edit / Deletion of Block Unit Function | |
| 12. Batch Calculation Schedule Improvement | |
| 13. Indexing Direction Calculation Function | |
| 14. CATIA V5 Import Layer Compatibility (Optional) | |
| 15. Added the Function for Using Multi-Edged Tool | |
| 16. New addition of Machine Simulator function (Optional) | |

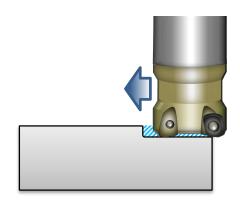
1. Addition of the Contour Face Cut Machining/High Feed Machining Function

"High feed machining" function has been added to reduce cutting time.

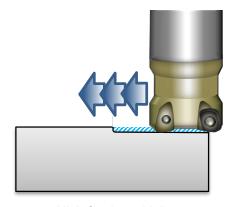
Path operations limited for high feed radius tools are carried out with utilizing the characteristics of the tools.

What is "High Feed Machining"

"High feed cutting" is a high speed cutting process using high feed cutting tools that is faster than cutting processes using general tools. By setting the cutting depth in the Z direction to a shallow depth, and widening the cut in the XY direction (same or slightly smaller than the tool base diameter), cutting chips become thicker and reduces the machining time. Generally, high feed cutting is used for the rough machining of work with large cutting area and removal stock.



General rough machining

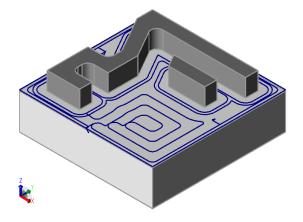


High feed machining

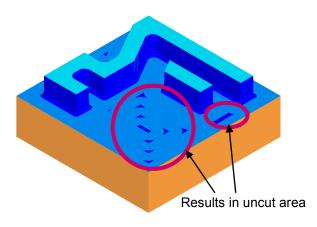
Precautions for "High Feed Machining"

In "high feed machining", as "the infeed in the XY direction is set wide", uncut areas will remain at corners, etc. when applied to conventional roughing machining.

In the following example, [Follow] at Motion Type are set in contour machining and the "pick width in the XY direction" is set the same as the tool bottom diameter. The interval among picks at the corners gets wider, it results in many uncut area remained.



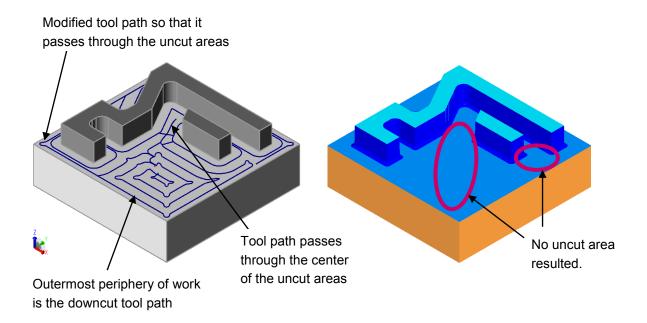
Tool path (When showing only one Z level)



Simulation results

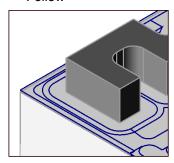
Characteristics of Tool Path of [Follow (high-feed machining)]

When "pick width in the XY direction", the characteristic of "high feed machining", is set to wide in [Follow (high-feed machining)], unique tool paths are output to prevent uncut areas. In addition, regardless of the geometry, tool paths for cutting the outermost periphery of the work are output by down-cut.

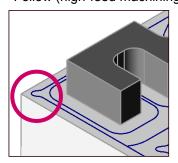


Comparison

Follow



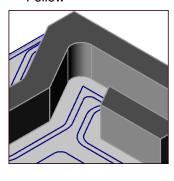
• Follow (high-feed machining)



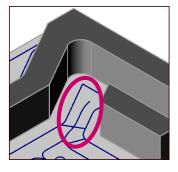
Outputs tool paths are modified so that they pass through the uncut areas.

The pick width in the XY direction is also adjusted.

Follow

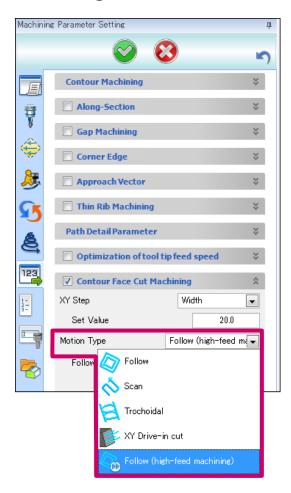


• Follow (high-feed machining)



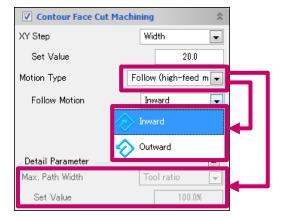
Outputs forward and return tool paths passing through the center of uncut areas.

■ Setting Screen

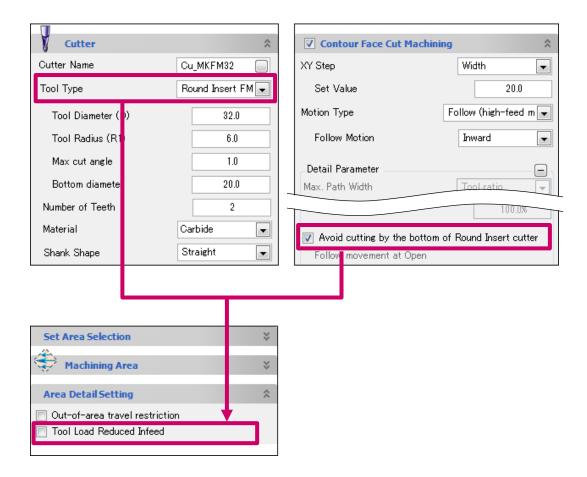


■ Note

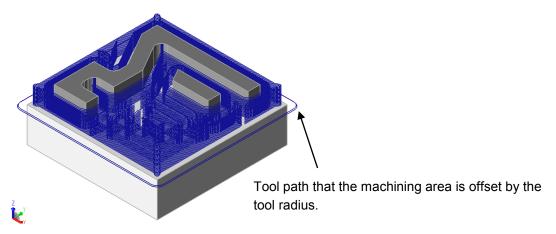
- (1) When [Follow (high-feed machining)] is set, [Auto Set] cannot be set at [Follow Motion].
- (2) When [Follow (high-feed machining)] is set, [Max. Path Width] cannot be set.



(3) When [Round Insert FM] is set in [Tool Type] and the [Avoid cutting by the bottom of Round Insert cutter] checkbox is checked, even if the [Tool Load Reduced Infeed] checkbox of [Machining Area] is unchecked, the tool path offset to the outside the tool radius will be output considering safety.



As shown in the following figure, unnecessary paths might be output in some cases. Adjust and use the machining area.



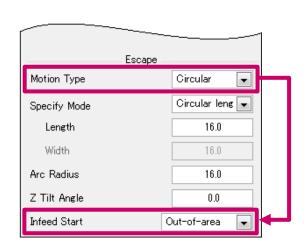
Path Approach Motion

When [Follow (high-feed machining)] is set, [Omit], [Circular], and [Contact line] can be used at [Motion Type] in [Path Approach Motion]. [Vertical] cannot be used.

When [Circular] is selected, select [Out-of-area] at [Infeed Start]. [Approach Position] at [Infeed Start] cannot be used.

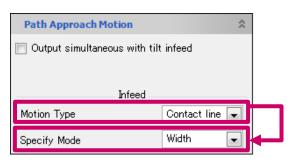
• Infeed/Circular **Path Approach Motion** Output simultaneous with tilt infeed Infeed Motion Type Circular • Circular leng 🕶 Specify Mode Length 16.0 Width 16.0 Arc Radius 16.0 Z Tilt Angle Infeed Start Out-of-area

• Escape/Circular

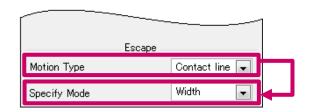


When [Contact line] is selected, select [Width] at [Specify Mode]. [Length] at [Specify Mode] cannot be used.

• Infeed/Contact Line



• Escape/Contact Line

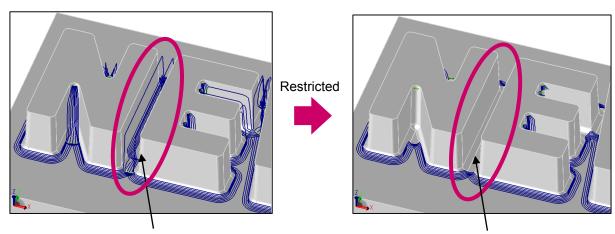


2. Addition of the Corner R Machining/ Groove Path Restriction Function

In [Corner R Machining], the tool path of [Along-area] may be irregular at the grooves. This function was added to solve this problem.

The uncut areas are detected whether they are grooves or not and [Along-area] paths are restricted.

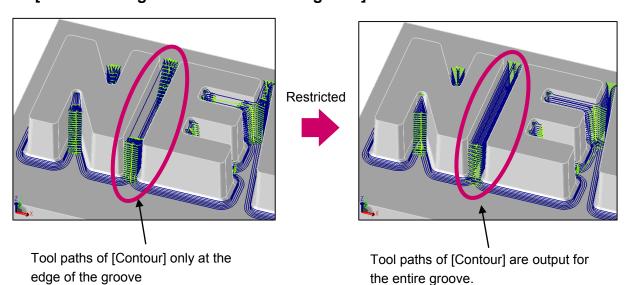
• [CR Machining Mode/Along-area]



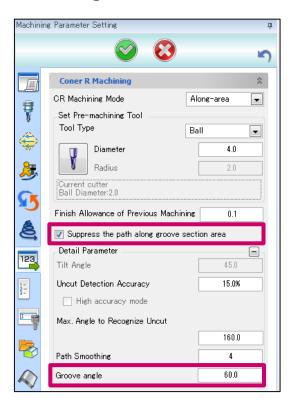
Tool paths to get irregular in groove areas

Tool paths to be removed in the grooves

• [CR Machining Mode/Contour + Along-area]



Setting Screen



(1) Suppress the path along groove selection area

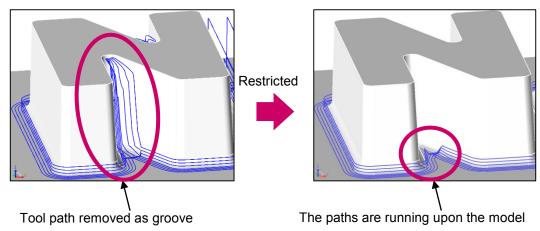
When [CR Machining Mode/Along-area] is set

Based on the cross-section of the uncut area of models, determines if uncut areas have groove shapes.

The tool paths in the areas to be detected as the groove shape are removed.

The areas to be removed paths by this function are added the virtual model by the system in order to prevent from interfering by other tool paths and then resulting in overload.

The shape of tool paths near the removed are becomes of riding upon the model added by the system.

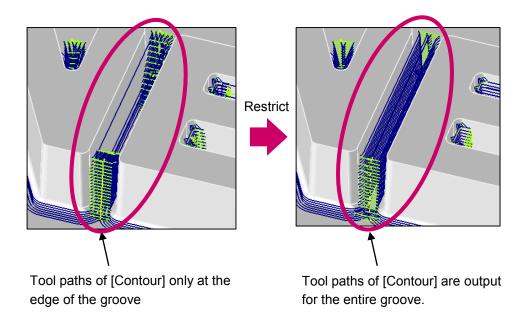


7

• When [CR Machining Mode/Contour + Along-area] is set

Based on the cross-section of the uncut area of models, determines if uncut areas have groove shapes.

The areas to be detected as the groove shape are output with tool paths for contour machining regardless of the switch tilt angle of [Contour] and [Along-area].



(2) Groove angle

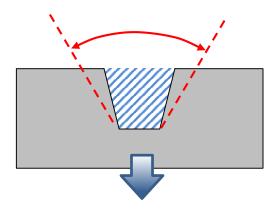
Specifies the Max. tilt angle for the groove.

This is used to determine if uncut areas have grooves.

When set [Groove angle] = 60 degree (red arc and dot line)

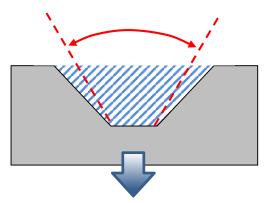
When the groove equal or less than 60 degree.

When the groove more than 60 degree.



The area is detected as a groove.

The tool paths of [Along-area] get restricted.



The area is detected as a non-groove.

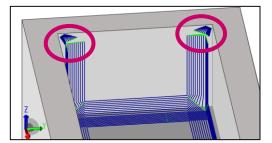
The tool paths is not restricted.

■ Note

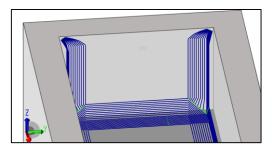
- (1) The function might not work rightly, when the target uncut area has branch tool paths.
- (2) In some cases, a groove angle is detected smaller than the actual angle, and path would not be output.

For example, when the function is run for the groove angle of 89 degree to the rightangled corner, some parts of are detected as a groove and the generated paths might get disconnected.

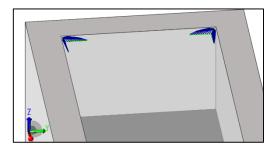
Example) No path is output
 When setting the groove angle 89 degree



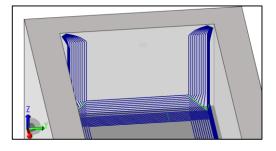
Example) Paths are output
 When setting the groove angle 80 degree



- (3) When the difference between the previous tool radius and machining tool radius is large, the corner is detected to be a groove, and a path may not be output in some cases.
 - Example) No path is output
 When setting the pre-machining Tool Radius = 3.0
 Machining Tool Radius = 0.5



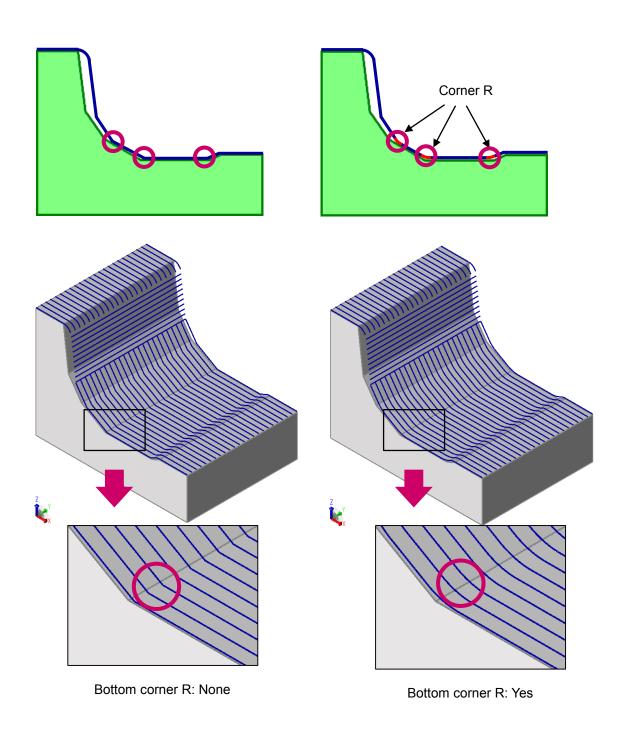
Example) Paths are output
 When setting the pre-machining Tool Radius = 3.0
 Machining Tool Radius = 1.0



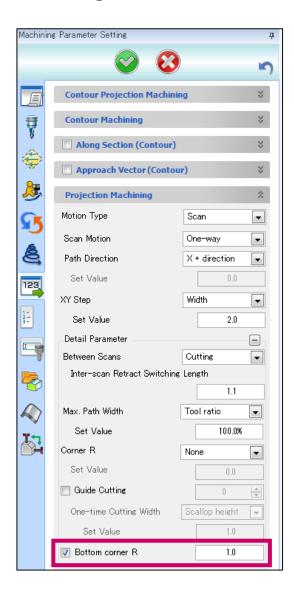
- (4) When the values of the groove width in the model shape and of the machining tool radius are close, the distance of output paths might get wider.
- (5) When using this function tends to increase the calculation time by approximately 20%. Moreover, when the Pre-machining Tool or Machining Tool Type is not a Ball, the calculation time gets further increased.
- (6) When [Contour+Along-area] with the switching tilt angle being large used, the tool load may becomes higher because the "contour path" may be output earlier than the "alongarea path" depending on the shape. As a result, the load might get higher.

3. Addition of Contour Projection Machining / Bottom Corner R Function

The corner R function can be added to bottom in [Contour Projection Machining]. This restricts tool load at corners and prevents tool chatter during machining.



■ Setting Screen



(1) Bottom corner R

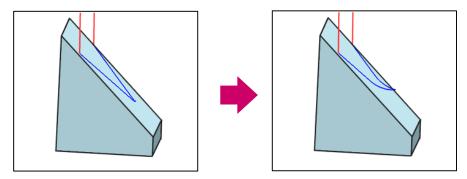
The corner R is inserted in the area where tool paths get right angled at the concave corner of the shape.

The radius of the corner R can entered into the text box.

Specify a smaller value than the corner R of the Tool for the corner R radius.

■ Note

(1) [Bottom corner R] inserts corner Rs in the plane direction passing through the Z axis. For this reason, corner R paths might be added to the area paths without concave angle.



(2) The function cannot be used with a flat end mill.

4. Addition of Flat Face Machining/Round Insert cutter Interference Check Function

The interference between the round insert tool base and the work can be checked using [Flat Face Machining].

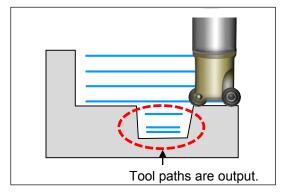
The function can be used when setting at [Tool Type] as [Round Insert FM], and at [Motion Type] as [Follow], [Scan], or [Trochoidal].

What is Round Insert cutter Interference Check Function

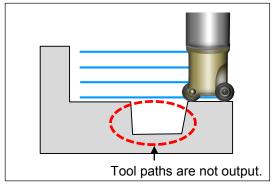
The interference check function of round insert base prevents interference between the round insert cutter base and the work.

Carries out interference check, and if interference is found to occur, prevents interference as follows.

• When there are narrow pockets with which the tool base interferes.

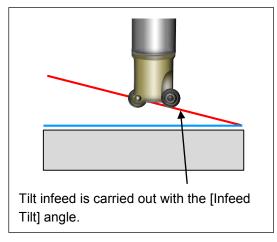


Avoid cutting by the bottom of Round insert cutter: OFF

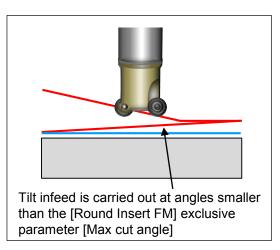


Avoid cutting by the bottom of Round insert cutter: ON

• Tilt Angle at Infeed



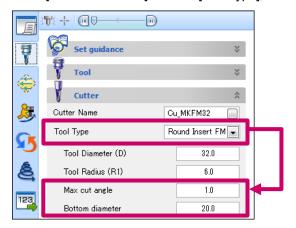
Avoid cutting by the bottom of Round insert cutter: OFF



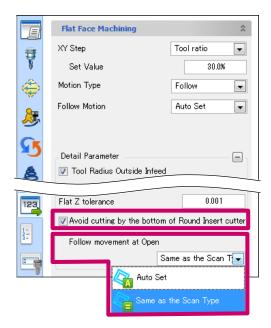
Avoid cutting by the bottom of Round insert cutter : ON

Tool Setting

When [Round Insert FM] is set at [Tool Type], enter into [Max cut angle] and [Bottom diameter].



■ Machining Setting



(1) Avoid cutting by the bottom of Round Insert cutter

Set whether or not to perform interference check between the Round Insert cutter bottom face and work.

(2) Follow movement at Open

Set the follow motion mode when the machining area is an open pocket. This can be set when the [Motion Type] is set [Follow].

Same as the Scan Type

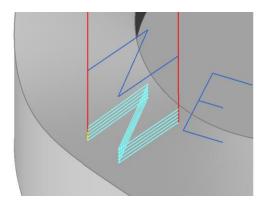
The same motion as setting at [Follow Motion] is carried out.

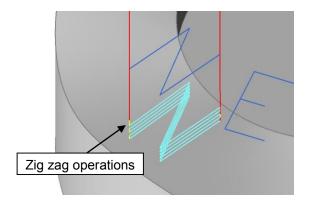
Auto Set

When the pockets in the machining area can be cut into from outside, the auto set mode of the Follow is run.

5. Addition of Route Machining / Zig-zag Drive-In Function

The [Zig zag] at [Motion type] in [Z Drive-in Cut] of [Route Machining] was added. The open curves can be driven-in efficiently and the machining time gets reduced.

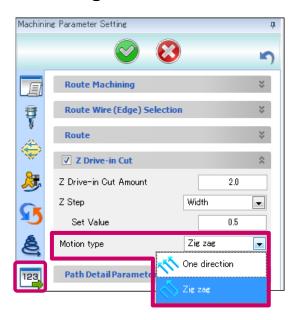




Motion type: One direction

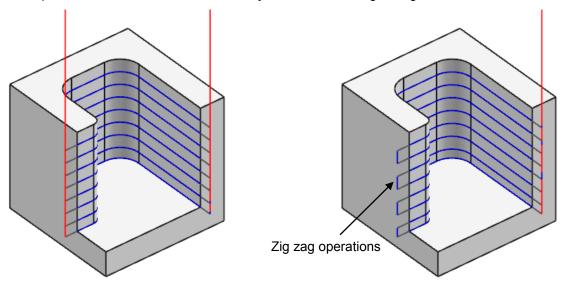
Motion type: Zig zag

■ Setting Screen



6. Addition of 2D Route Machining/ Zig-zag Drive-In Function

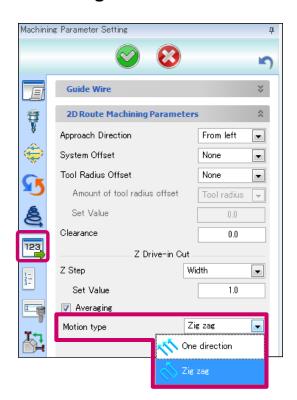
The [Zig zag] at [Motion type] in [Z Drive-in Cut] of [2D Route Machining] was added. The open curves can be driven-in efficiently and the machining time gets reduced.



Motion type: Zig zag

■ Setting Screen

Motion type: One direction

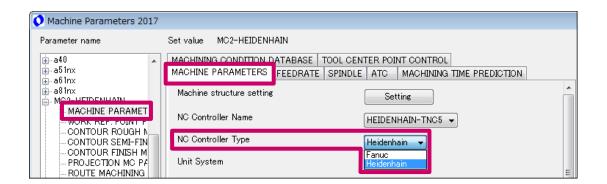


7. Heidenhain Canned Cycle Support

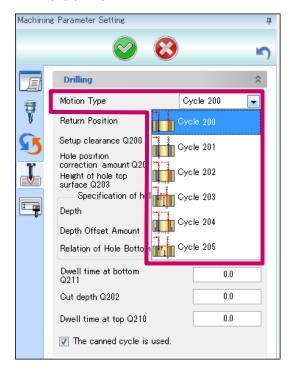
Heidenhain canned cycle can be used in [Drilling], [Thread Cutting], [Spot Face Machining], and [Chamber Machining].

When machines set with [Heidenhain] are selected at [NC Controller Type] in [MACHINE PARAMETERS], the canned cycles of "Heidenhain" are shown at [Motion Type]. When a machine set with [Fanuc] at [Controller Type] is used, the canned cycles of [Fanuc] in are shown.

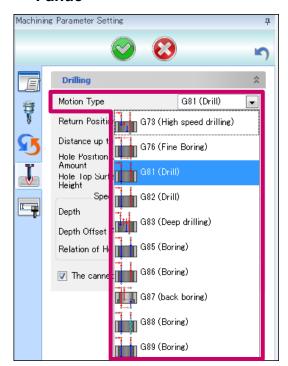
■ Machine Parameters



Heidenhain

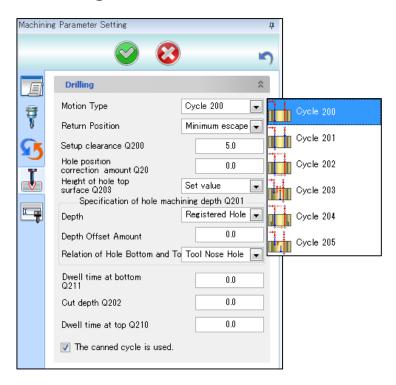


Fanuc



7.1 Drilling

Setting Screen



(1) Motion Type

The following option of Heidenhain Canned Cycle can be chosen at [Motion Type] in [Drilling].

- Cycle 200 (Drill)
- Cycle 201 (Reamer)
- Cycle 202 (Boring)
- Cycle 203 (Universal drill)
- Cycle 204 (Back boring)
- Cycle 205 (Universal drill)

(2) Return Position

The item sets the returned position of the tool after drilling.

The position is chosen from [Minimum escape height] or [Second setup clearance Q204].

- Minimum escape height

 The height of the reference pair
 - The height of the reference point is [Height of hole top surface+ "Setup clearance Q200"]. And the tool moves among holes with the height of [Position without interfere with model or work + "Setup clearance Q200"].
- Second setup clearance Q204
 The distance entered at [Second setup clearance Q204] is referenced and the height is set.

(3) Setup clearance Q200

This sets the position of R point.

The distance up to the R point based on the hole position to be set at every hole with the hole information.

Note:

- Negative values cannot be entered.
- When the higher values than 0 are set at [Slope Height] in the holes in the hole information, the standard position to calculate the R point is changed.
- Only the [Slope Height] from the hole position shifted in the Z plus direction is used as reference. The position to which [Setup clearance Q200] added to this will be the R point.

(4) Hole position correction amount Q203

R point and the hole position based on the machining depth can be shifted, when hole position correction amount set.

This parameter is used to lower the R point after the top surface of the hole has been machined.

Negative values cannot be set.

(5) Height of hole top surface Q203

The Hole Top Surface Height of the set hole can be chosen at [Set value] or [Work top surface].

(6) Specification of hole machining depth Q201

Set [Depth], [Depth Offset Amount], and [Relation of Hole Bottom and Tool]. (Is the same as the setting parameters of Fanuc canned cycle.)

(7) Shift amount Q251

Enter the shift amount of escape motion.

(8) Dwell time at bottom Q211

Enter the pause time at the bottom.

(9) Cut depth Q202

Enter the cutting depth for one time.

(10) Second setup clearance Q204

[Return Position] is an item displayed for [Second setup clearance Q204]. Enter any distance to be set as the tolerance.

(11) Dwell time at top Q210

Enter the pause time at the position of [Setup clearance] when pulling out the tool from the drill hole.

(12) Feed rate at retracting Q208

Enter the feed rate when pulling out the tool from the drill hole.

(13) Angle for spindle orientation Q336

Enter the angle for positioning the tool before motion.

Note:

If setting the [Angle for spindle orientation Q336], check the position of the tool tip. Set the angle so that the tool tip is positioned parallel to the coordinate axis.

(14) Free travel orientation Q214

Enter the tool escape direction after hole drilling. The input value is as follows.

- 0: The tool is not allowed to move freely.
- 1: The tool is allowed to move freely in the negative direction of the spindle.
- 2: The tool is allowed to move freely in the negative direction of the counter shaft.
- 3: The tool is allowed to move freely in the positive direction of the spindle.
- 4: The tool is allowed to move freely in the positive direction of the counter shaft.

Note:

Set the direction for the tool to move freely such that the tool is away from the drill bottom.

Depending on the motion type, setting "0: The tool is not allowed to move freely." might not be set.

(15) Minimum cut depth Q205

Enter the minimum value for depth of cut at [Reduced value].

(16) Decreased value Q212

Enter the value for decreasing the depth of cut after cutting motions.

(17) Retraction at chip breaking Q256

Enter the distance for pulling back the tool at chip fracture.

(18) Chip breaking count until retraction Q213

Enter the times number of chip fractures when pulling back the tool.

(19) Advanced stop distance at top Q258

Enter the setup tolerance after the first chip fracture.

(20) Advanced stop distance at bottom Q259

Enter the setup tolerance after the last chip fracture.

(21) Deeper starting point Q379

Enter the start depth of drilling.

(22) Pre-positioning feed rate Q253

Enter the feed rate up to the start depth of drilling.

(23) Back boring width Q249

Enter the back boring width.

(24) Tip length Q252

Enter the length of the tool tip.

(25) The canned cycle is used.

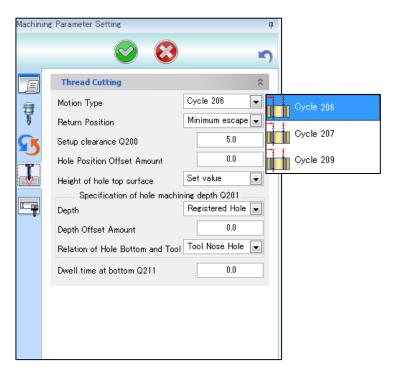
Select whether to output with canned cycle or cutting movement commands.

- ON: Output canned cycle
- OFF: Output cutting movement commands

The check box is always On, when [Motion Type] is set [Cycle 202] or [Cycle 204].

7.2 Thread Cutting

■ Setting Screen



(1) Motion Type

Select the [Motion Type] of [Thread Cutting] from Heidenhain Canned Cycle.

- Cycle 206 (Tapping)
- Cycle 207 (Rigid tap)
- Cycle 209 (Rigid tap)

(2) Machining depth until chip breaking Q257

Enter the cut depth at running chip fracture.

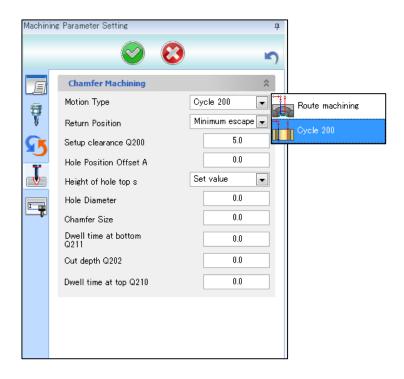
There is no chip fracture when 0 entered.

(3) Rotating orientation

Select the rotation direction at "Right-hand thread" or "Reverse thread".

7.3 Chamfer Machining

■ Setting Screen



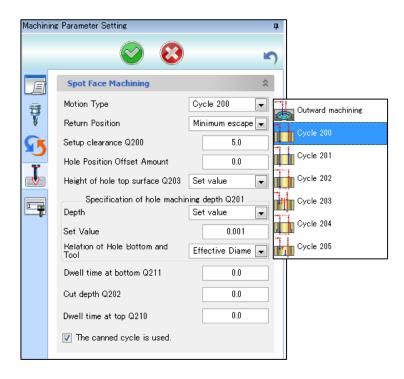
(1) Motion Type

The [Motion Type] in [Chamfer Machining] can be chosen as [Route Machining] or the Canned Cycle of Heidenhain.

- Route machining
- Cycle 200 (Drill)

7.4 Spot Face Machining

■ Setting Screen



(1) Motion Type

The [Motion Type] in [Spot Face Machining] can be chosen as [Outward Machining] or an option in the Canned Cycle of Heidenhain.

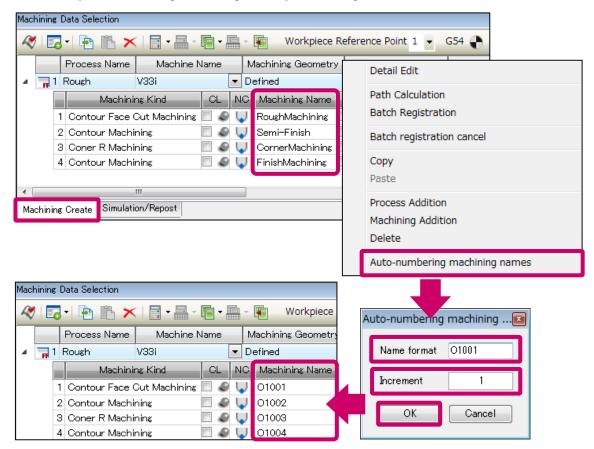
- Outward machining
- Cycle 200 (Drill)
- Cycle 201 (Reamer)
- Cycle 202 (Boring)
- Cycle 203 (Universal drill)
- Cycle 204 (Back boring)
- Cycle 205 (Universal drill)

8. Addition of Machining Name Consecutive Number Renaming Function

[Machining Name] and [Output File Name] can be renamed with using consecutive numbers. The machining name copied from the User Database can be changed collectively to names managed easily.

8.1 Machining Name Consecutive Number Renaming

Select one or several target processes or machining from the [Machining Create] process list, and select [Auto-numbering machining names] from the right-click menu.



(1) Name format

Enter the fixed character strings added before the consecutive number. (Can be omitted)

(2) Increment

Enter a positive integer. The upper limit value of use is "100000000".

If selected in the "Process" list

All processing names in the process can be renamed using consecutive numbers.

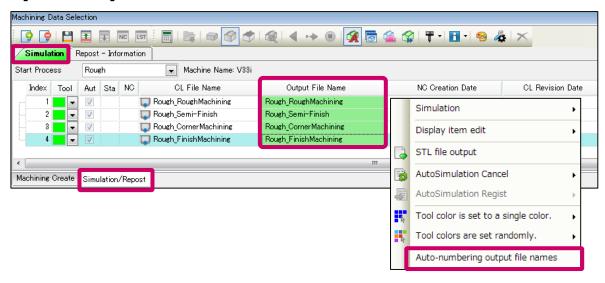
• If selected in the "Machining" list

All machining names of selected files can be renamed using consecutive numbers. If the [Machining Name] of the list is hidden, [Auto-numbering machining names] cannot be used.

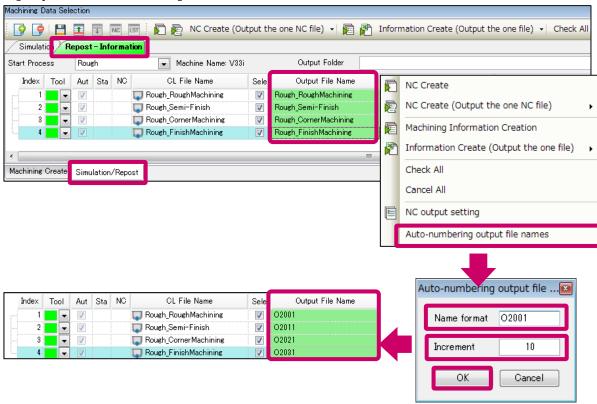
8.2 Output File Name Consecutive Number Renaming

Select one or several target files from the [Simulation/Repost] process list, and then select [Auto-numbering output file names] from the right-click menu.

• [Simulation] tab



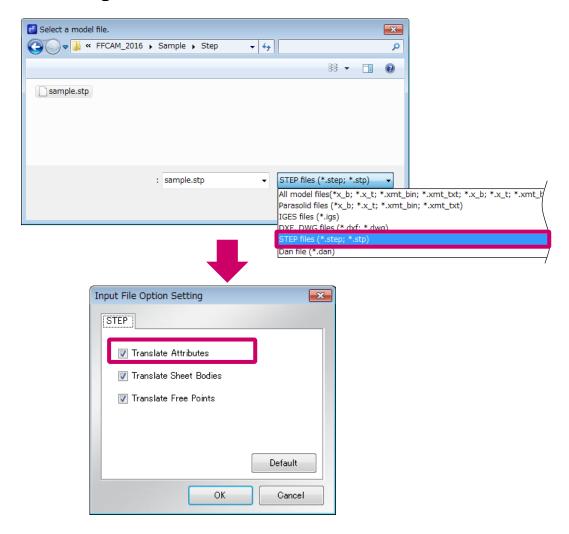
• [Repost - Information] tab



9. STEP Import Layer Compatibility

Layer attribute conversion has been supported when importing "STEP" data. When the [Translate Attributes] checkbox in [Input File Option Setting] is On, "layer attributes" is converted in the same way as "color attributes". No changes have been made to the setting screen.

■ Setting Screen



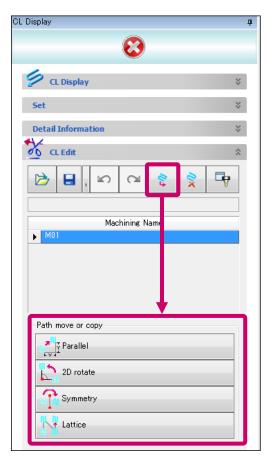
- When the character string in layer Number is single byte made up of only "0 to 9", the attributes is converted as in the following (1), (2), and (3).
 - (1) Converted to the layer number when the layer number is between "0 to 256".
 - (2) Converted to "0" when the layer number is "less than 0".
 - (3) Converted to layer number "256" when the layer number is "257 or more".
- When the layer number is other the above (1), (2), or (3), enters the layer elements to empty layer numbers according to the STEP conversion processing order.
 When a layer numbers is more than "256", it is converted to "256".

10. Improvement of CL Edit and Operability

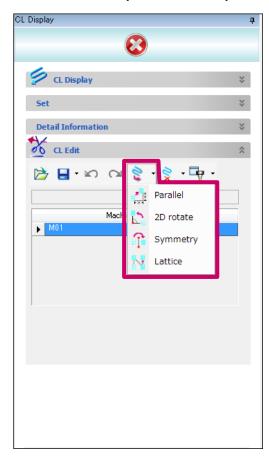
The selection method of functions such as "Move/Copy", "Delete CL", and "Change Setting" of [CL Edit] has been changed to the toolbar dropdown list. Operability has improved as screen operations are simpler and more efficient.

■ Setting Screen (Move/Copy)

Old Function

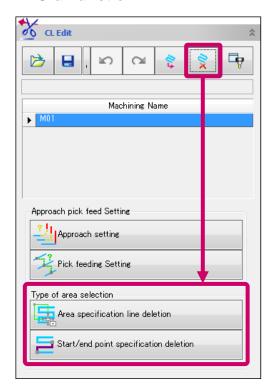


• New Function (FFCAM 2017)

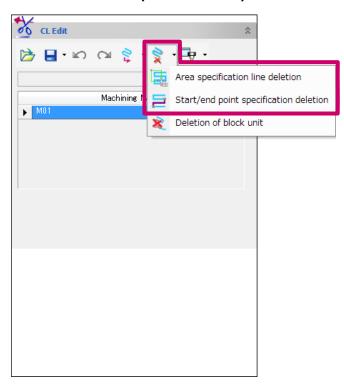


■ Setting Screen (Delete CL)

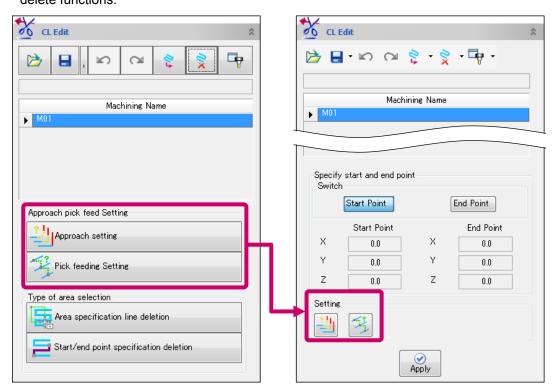
Old Function



• New Function (FFCAM 2017)

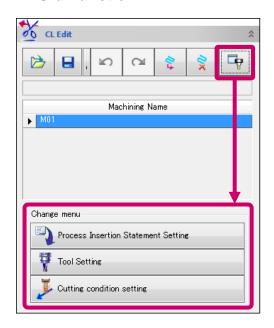


* [Approach setting] and [Pick feed Setting] have been moved inside the setting screens of delete functions.

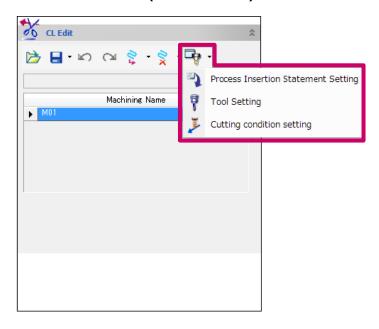


■ Setting Screen (Change Setting)

Old Function



• New Function (FFCAM 2017)

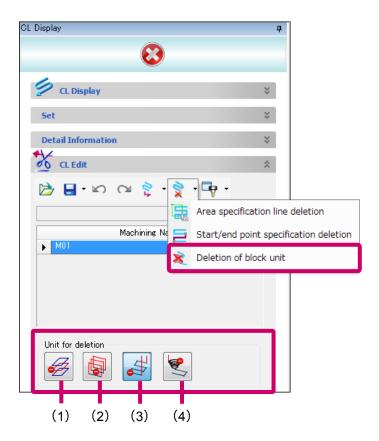


11. Addition of CL Edit / Deletion of Block Unit Function

The [Deletion of block unit] function has been newly added to the "Delete CL" function. CL can now be selected in the units of "Z level", "cutting", "segment", and "approach". This enables the objectives of the operators to be reflected more easily. Operation mistakes can also be prevented by limiting the CL entities to be deleted.

This function that deletes in "approach" units can not only delete existing cutting operation CL, but can also substitute to any "approach".

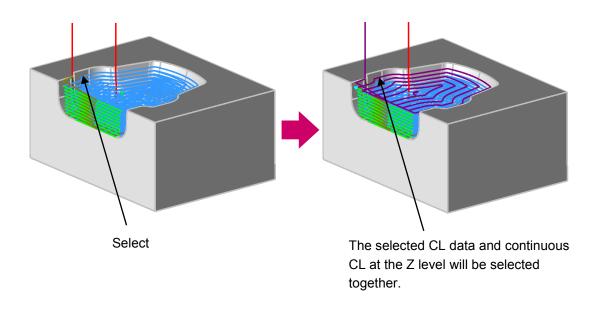
■ Setting Screen



(1) (Z Level) button

Click the (Z level) button, and select the CL to be deleted.

The selected CL data and continuous CL at the Z level will be selected together. Clicking the [Apply] button deletes the selected CL.

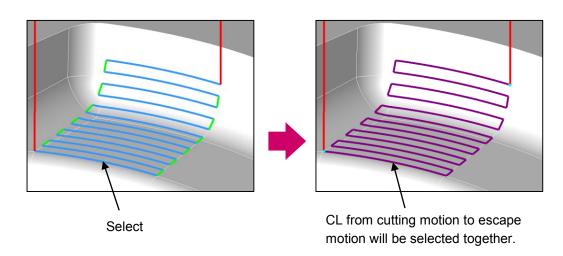


(2) (Cutting) button

Select the CL to be deleted by clicking the (cutting) button.

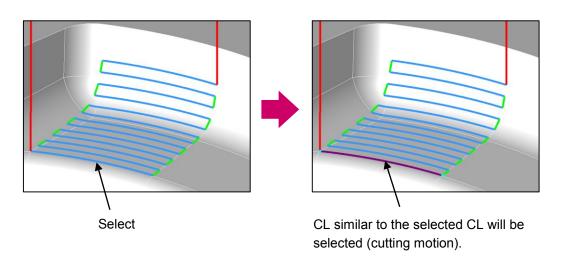
CL in the cutting range (from cutting motion to escape motion) of the specified CL data will be selected together.

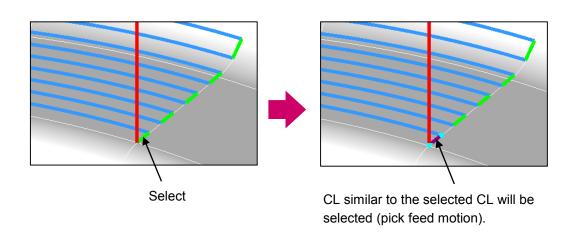
Clicking the [Apply] button deletes the selected CL.



(3) (Segment) button

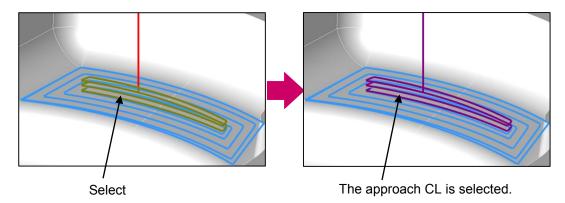
Click the (segment) button and select the CL to be deleted. Similar CL data continuing from the selected CL will be selected. Clicking the [Apply] button deletes the selected CL.





(4) (Approach) button

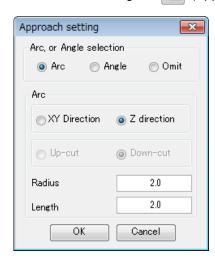
Click the (Approach) button and then the approach CL to be deleted or changed.

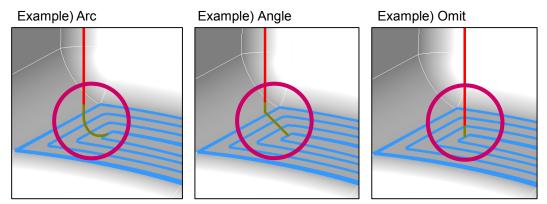


• | (Approach setting)

After selecting the approach CL with the (Approach) button, next click the (Approach setting) button to set the approach.

When the [Apply] button is clicked, the selected approach CL will be regenerated based on the settings of [24] (Approach setting).



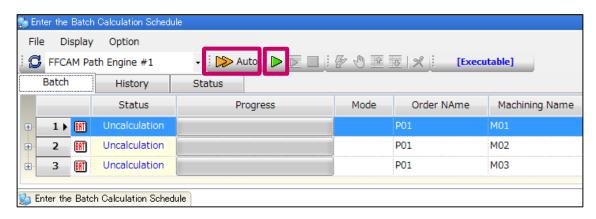


12. Batch Calculation Schedule Improvement

The automatic execution function has been added to the [Batch Calculation Schedule] screen. This eliminates the need to open the [Batch Calculation Schedule] screen to start batch calculation, thereby improving work efficiency.

■ Auto (Auto) button: OFF

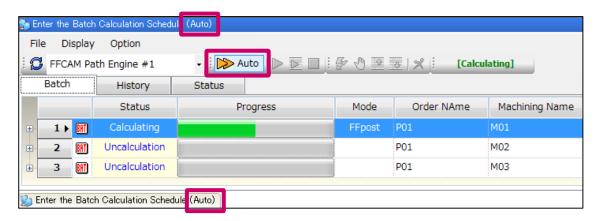
When the (Start Schedule) button is clicked, batch calculation starts. This is the old function.



■ [▷ Auto] (Auto) button: ON

To use the function, click the [Auto] button to highlight the button.

This function automatically starts batch calculation when performing [Batch Registration] or [Blanket Batch Registration] of process list of FFCAM. If [Batch Registration] is carried out during batch calculation, calculation will be carried out including additionally registered processes.



When the Auto Execute function is enabled, "Enter the Batch Calculation Schedule (Auto)" will be displayed on the title bar and tab of the Batch Calculation Schedule screen.

13. Indexing Direction Calculation Function

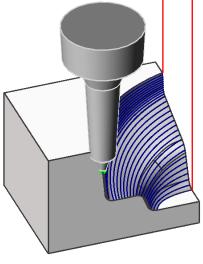
Until now, to shorten the "overhang length" in indexing machining and enhance the conditions, path creation and simulation had to be repeated to determine the "machining direction" enabling machining without interference or undercuts.

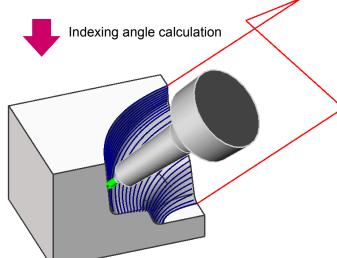
This function is able to select the surfaces to be machined, and calculate "machining directions" in which interference between the tool and machining geometry will not occur. Furthermore, the "overhang length" required for that direction is also calculated. Calculating the "indexing direction" in which machining can be carried out without interference and undercut beforehand reduces the adjustment time required by operators.

This function can be used in the following conditions:

- The tool and machining geometry must be set.
- The "Overhang Length" of the tool must be set.
- When using a pencil neck tool, the "Neck Length" must be set.

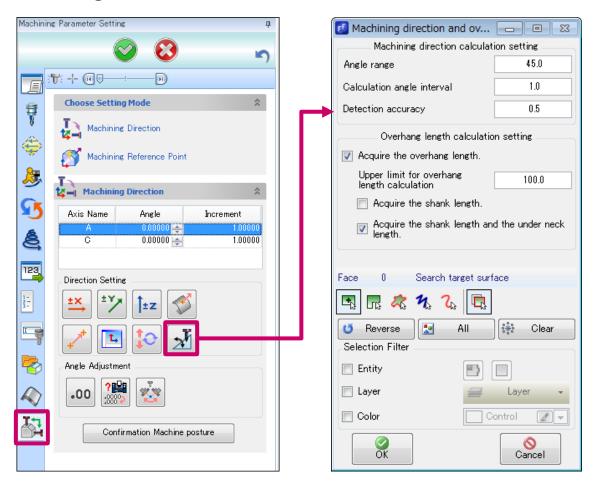
- [Machining Geometry] Settings
- [Tool (Overhang Length)] Settings





- Machinable [Machining Direction] Calculation
- Calculation of required [Overhang Length]

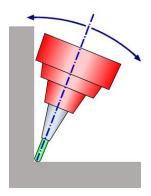
Setting Screen



Machining direction calculation setting

(1) Angle range

Sets the range calculating machining direction with based on the tool axis direction. Setting range: 0° to 89° (Default value: 45°)



(2) Calculation angle interval

Sets the angle interval to be calculated.

(3) Detection accuracy

Sets the detection accuracy with mesh width.

Overhang length calculation settings

(4) Acquire the overhang length

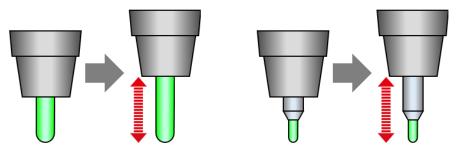
When the check box checked, the "Overhang Length" is also calculated when calculating the machining direction.

(5) Upper limit for overhang length calculation

When calculating the "overhang length", set the upper limit value of "overhang length".

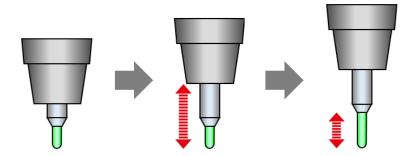
(6) Acquire the shank length

When calculating the "overhang length", only the "tool overhang length" is calculated. The "neck length" is not changed.



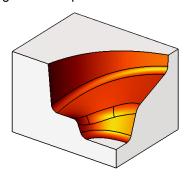
(7) Acquire the shank length and the under neck length

When calculating the "overhang length", the "tool overhang length" and "neck length" is calculated. First, the calculation is done with the "Tool Overhang Length" that can be avoided interference. If interference cannot be avoided using "tool overhang length", "neck length" is added and calculated.



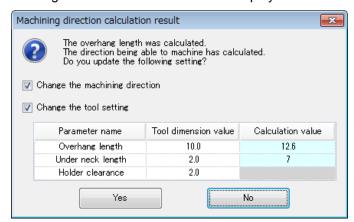
(8) Search target surface

Select the surface for calculating the machining direction. The selected surfaces get highlighted. Multiple surfaces have to be selected.



(9) Machining direction calculation result

The machining direction calculation result is displayed.



Yes

[Machining Direction] and [Tool Setting (Overhang Length/Neck Length)] are changed and the [Machining Direction Calculation Results] screen is closed.

No

[Machining Direction] and [Tool Setting (Overhang Length/Neck Length)] are not changed and the [Machining Direction Calculation Results] screen is closed.

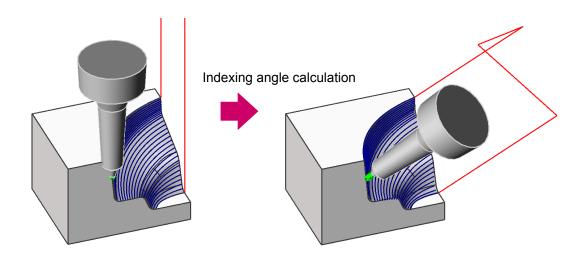
■ Note

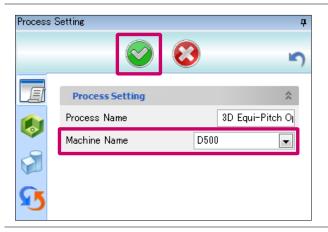
- (1) Interference between tool and material shape, or tool and machine structure (table and tool) cannot be detected.
- (2) The interference check cannot be carried out on the following specific setting areas: the area of setting Z area, the area of not machining, clamp area, check surface, offset plane, edging, tilt area, uncut area and stock area, because interference check is performed only above the machined area.
- (3) As interference check is carried out only on the machining area, interference check of movements outside the machining area cannot be performed such as escape, pitch motion, approach, and retract.
- (4) Interference check cannot be performed normally in places where the outer periphery of the area is a wall with respect to each axial direction.
- (5) When geometries without thickness such as seat bodies are selected, interference check can not carry out normally.
- (6) When the machining enable angle is minute range, the calculation result might not be given depending on lattice and angle accuracy.

■ Operation Procedure

As an example, the following introduces the procedure to automatically calculate and change "machining direction", "overhang length", and "neck length" that can be machined by "indexing direction calculation function" for tool paths created with three axes.

In this data, the "overhang length" and "neck length" have been set short.

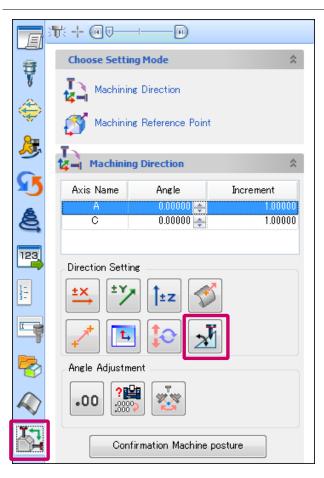




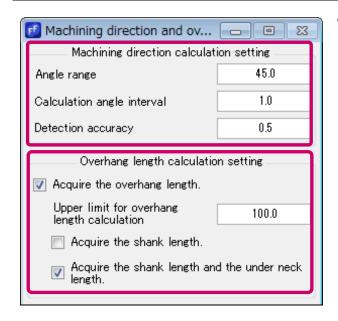
- Open "3Dpitch_open_auto.xMtn."
 Save to: FFCAM installation folder
 ¥Sample¥Parasolid¥3Dpitch_open_auto
- Double-click the leftmost cell of the process column.
- 3. Select "D500" from [Machine Name] and click .



- Double click the left most cell of the machining column.
- 5. Click (Tool Setting) and check the tool settings.



- 6. Click the (Machining Direction and Reference Point) button.
- 7. Click (Machining Direction Calculation Function).



 Set [Machining direction and overhang length calculation].
 Configure settings according to the

following:

Machining direction calculation setting:

Angle range : 45
Calculation angle interval : 1
Detection accuracy : 0.5

Overhang length calculation setting:

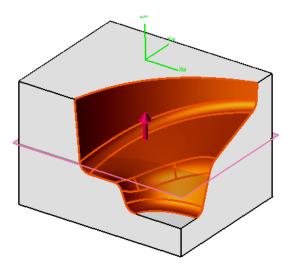
Acquire the overhang length: ON

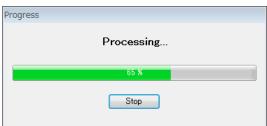
Upper limit for overhang

length calculation : 100

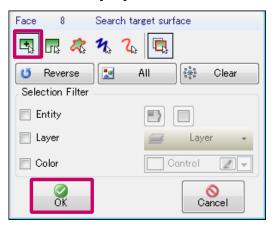
: ON

 Acquire the shank length and the under neck length.

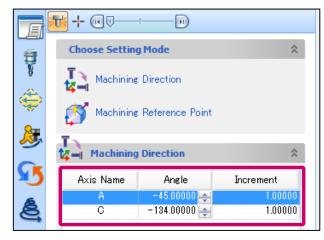




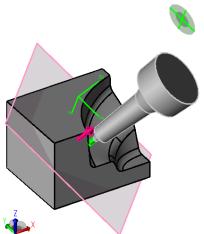
9. Select the face shown in the figure on the left, and click [OK].

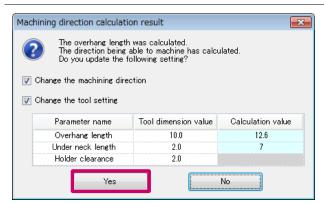


The "calculation of the machining direction" starts.

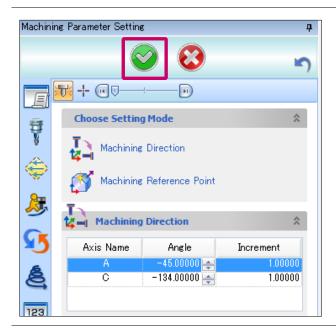


10. When the calculation ends, the results are displayed.

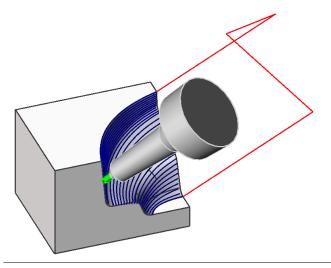




11. Check [Machining direction calculation result] and click [Yes].



- 12. Click .
- 13. Click (Path Calculation) to execute [Path Calculation].

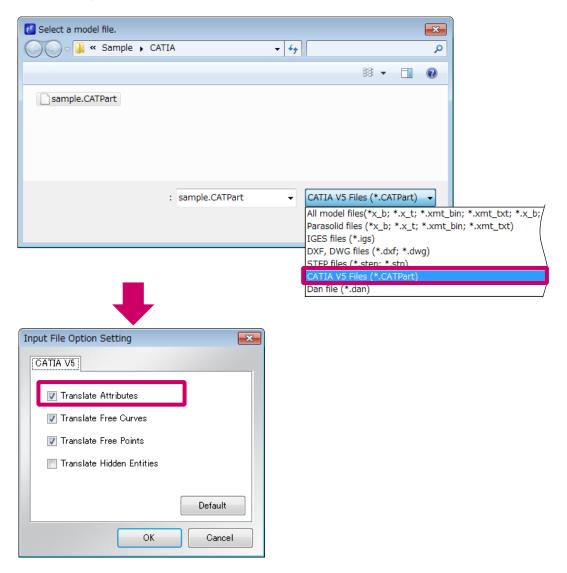


The result shown in the figure on the left will be obtained.

14. CATIA V5 Import Layer Compatibility (Optional)

Layer attribute conversion is now supported when importing "CATIA V5" data. If the [Translate Attributes] checkbox in [Input File Option Setting] is selected, "layer attributes" will also be converted in the same way as "color attributes". No changes have been made to the setting screen.

Setting Screen



- (1) Converted to the layer number if the layer number is between "0 to 256".
- (2) Converted to layer number "256" if the layer number is "above 257".
- (3) Converted to layer number "0" if the layer number is not set.

15. Added the Function for Using Multi-Edged Tool

Function for effectively using multi-edged tools ("4-blade" or "6-blade" tools) have been added. By specifying the [Feed speed] according to the tool and work contact surface, machining can be carried out in making use of the advantages of multi-edged tools.

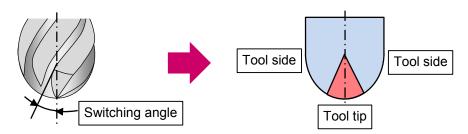
With multi-edged tools, the feed speed can be increased when machining with the tool side which consists of many edges. However, in many cases, as the "number of edges at the tool tip" of multi-edged tools will be half the "number of edges at the tool side", there is a need to decide the [Feed speed] in taking into account the area around the base where there are only a few edges.





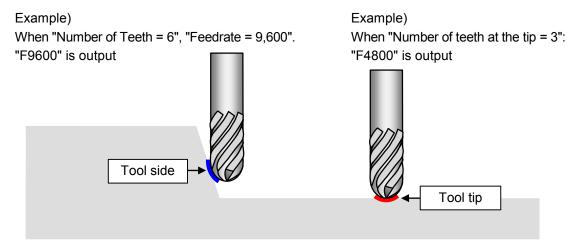
Set this function with the [Feed speed] parameter set to "Maximum number of tool edges". Set the "No. of Edges" separately for the tool tip with little edges to decelerate [Feed speed] of the tool tip. Set tool tip at [Switching angle].

In addition, set the [Number of teeth at the tip] as the number of edges of the tool tip.

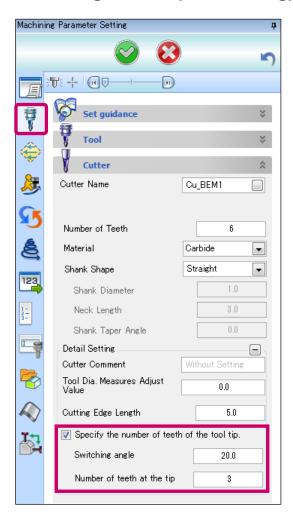


Calculate the feed speed of the tool tip using the following equation.

[Tool tip feed speed] = [Feed speed] x [Number of teeth at the tip] ÷ [Tool Setting No. of edges]



Setting Screen (Tool Setting)



(1) Specify the number of teeth of the tool tip

Set whether to perform [Specify the number of teeth of the tool tip].

When [Optimization of tool tip feed speed] is set to ON, the [Feed speed] of the tool tip will be adjusted according to [Specify the number of teeth of the tool tip].

(2) Switching angle

"Tool side" and "tool tip" are recognized based on [Switching angle].

The [Feed speed] parameter value is output for tool side.

The [Feed speed] is decelerated and output for tool tip. ($0 \le [Switching angle] < 90$)

(3) Number of teeth at the tip

Set the Number of teeth of the tool tip. ($0 \le [Number of teeth at the tip])$

Setting Screen (Machining Parameter Setting)



(4) Optimization of tool tip feed speed

Set whether to use the [Optimization of tool tip feed speed] function.

When [Optimization of tool tip feed speed] is set to ON, the [Feed speed] for machining with the tool tip will be adjusted according to the value set at [Specify the number of teeth of the tool tip] of [Tool Setting]/[Tool].

(5) Optimization accuracy

Set the accuracy when performing [Optimization of tool tip feed speed]. $(0.001 \le [Optimization accuracy])$

(6) Margin Amount

The area recognized as "tool tip" increases. $(0 \le [Margin Amount])$

(7) Optimization Minimum Distance

Minimize changes in [Feed speed] for short distances.

[Feed speed] will not change if the cutting distance at the tool tip is shorter than the [Optimization Minimum Distance]. (0 \leq [Optimization Minimum Distance])

■ Note

- The only multi-edged tool which can be used for this function is "ball end mill". (Taper ball end mill is not supported.)
- This function can be used for the following "machining types".

[Contour Machining]

[Projection Machining]

[Contour Projection Machining]

[3D Equi-Pitch Machining (Close/Open)]

[Along-Surface Machining]

- This function cannot be used together with the following functions.

[Contour Face Cut Machining] (Cannot be set)

[Thin Rib Machining] (Cannot be set)

[Z Drive-in Cut] (Cannot be set)

[5-Axis Simultaneous Machining] (Error during calculation)

[Index Machining] (Error during calculation)

[Taper Tool] (Error during calculation)

[Overhang Length Division] (Error during calculation)

[Finishing Allowance/Specify Offset face] (Error during calculation)

[CL Edit]

16. New addition of Machine Simulator function (Optional)

"FFCAM 2017" has a new simulation function using "NC data".

This function enables simulations up to 5-Axis simultaneous machining using "NC data", as well as "detection of interference with machines" and "verifications of insert statements" not possible with simulations using "CL data".

This function is an optional function for users of "5axis Machining".

