102-NF-19000

FFCAM 2019

Description of New Functions

MAKINO

Preface
This manual describes the functions added to MAKINO FFCAM 2019 and how to use them.

Created on
May 2019

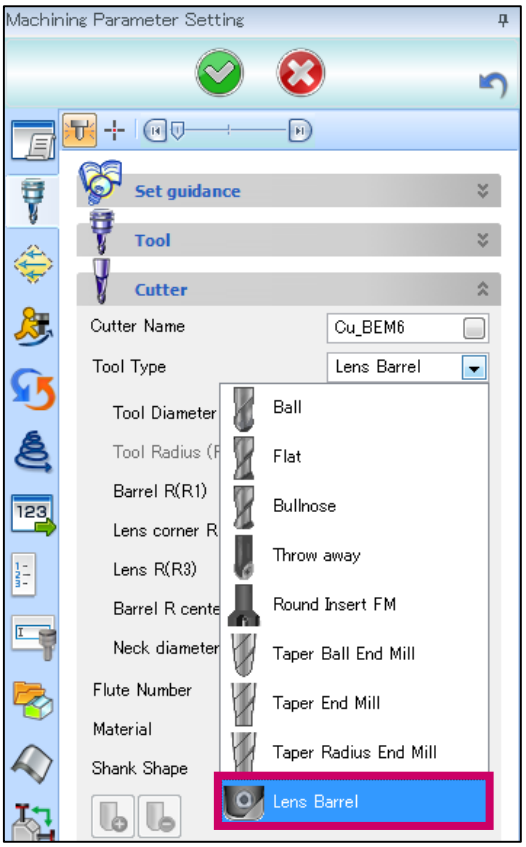
List of Added Functions in FFCAM 2019

1. Removed Restriction on Lens Barrel Tool (Contour Face Cut)

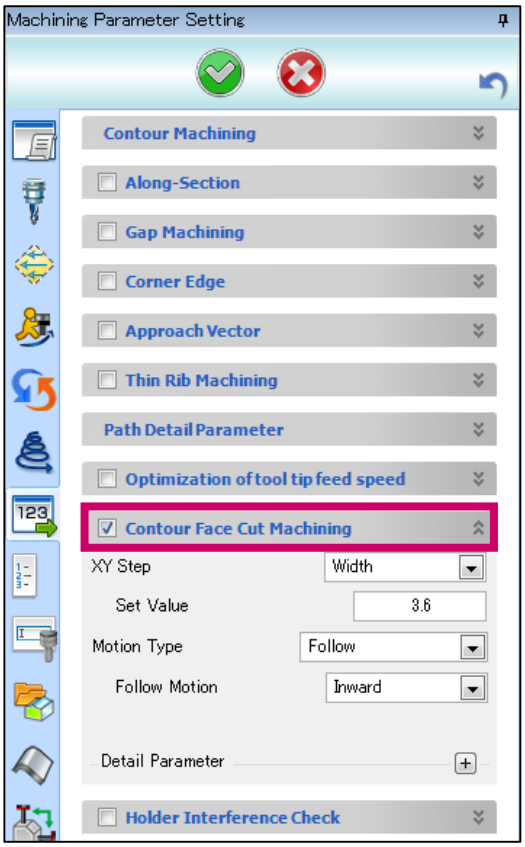
It is now possible to select contour face cut machining when using lens barrel tools. This function allows you to perform rough and finish machining of contour using lens barrel tools.

■ Setting Screen

- **Tool Setting**



- **Machining Parameter Setting**

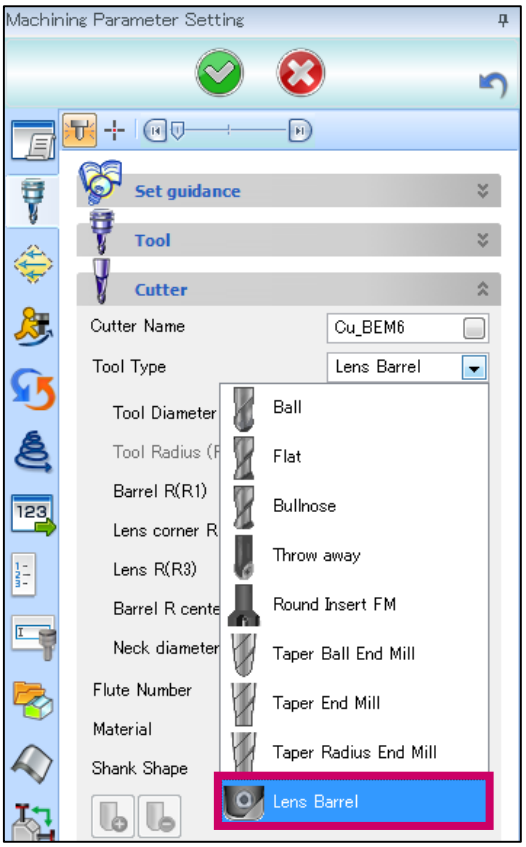


2. Removed Restriction on Lens Barrel Tool (Stock Support)

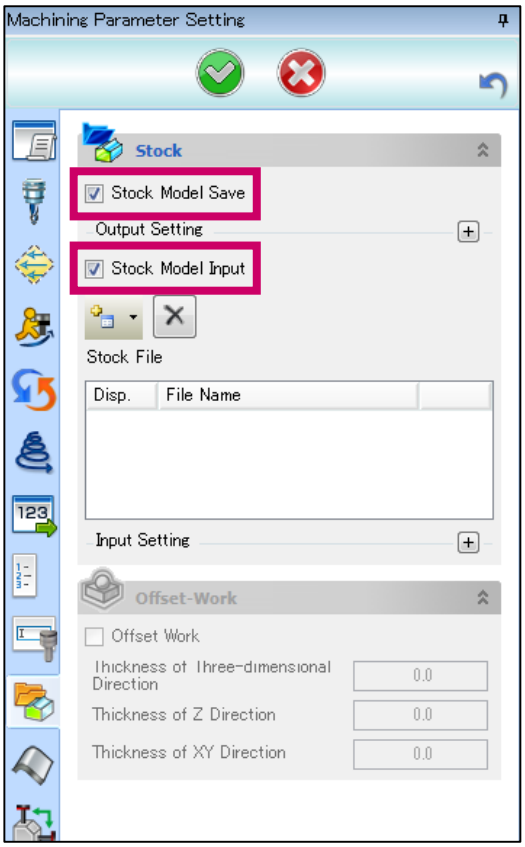
It is now possible to read and save stock files when using the lens barrel tool. This function allows you to accurately identify the uncut portions after using lens barrel tools and create machining without waste in the next process.

■ Setting Screen

- **Tool Setting**



- **Uncut Geometry Setting**



3. Addition of Function to Specify the Progress Direction for Tool Path

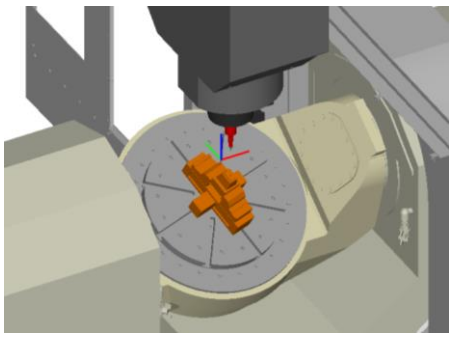
A function has been added to obtain the progress direction for the tool path from the geometry when "One-way" or "Zigzag" is selected for scan motion or contour face cut motion.
For index machining function, even in the case of machining where the angle of the scan motion is unclear, you can now set the progress direction without measuring or by trial and error.

The following machining types can be set.

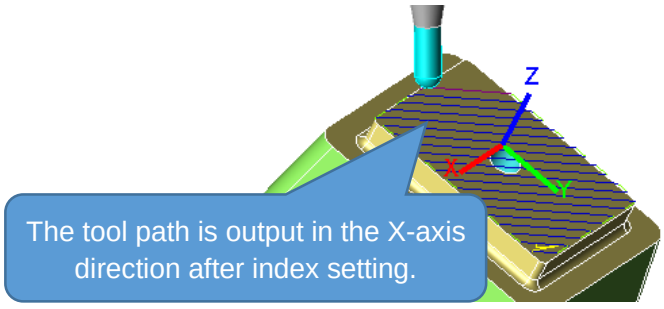
- Projection Machining
- Specify Projection Direction of Contour Projection Machining
- Flat Face Machining
- Contour Face Cut Machining

* Any one of the machining types can be set when the Path Operation is "One-way" or "Zigzag".

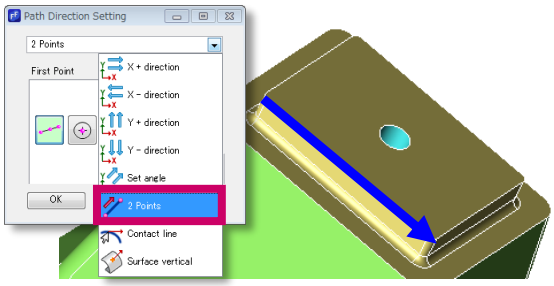
■ Examples for the Creation of Index Machining Data



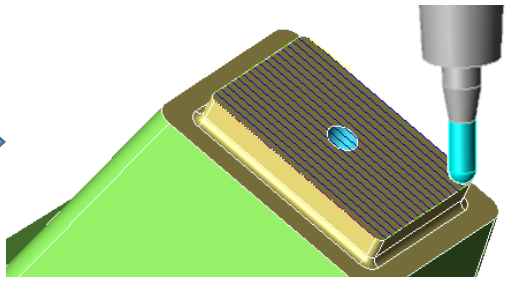
(1) Results of setting zigzag motion in the X-axis direction using the previous function



(2) When using "Function to Specify the Progress Direction for Tool Path" with FFCAM 2019



In [Path Direction Setting] (Example: 2 Points), specify the edge of the blue arrow.



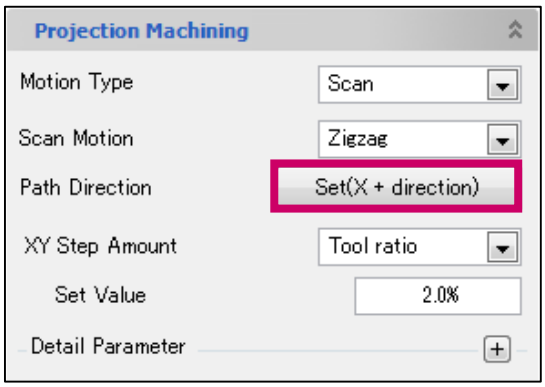
The tool path is output in the X-axis direction before the index setting.

■ Setting Screen

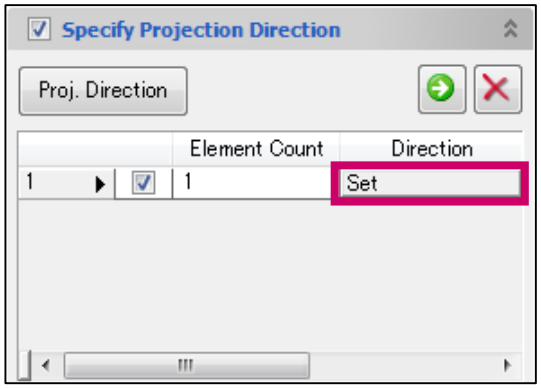
Click the [Set] button in [Path Direction] of Projection Machining, Specify Projection Direction of Contour Projection Machining, Flat Face Machining, or Contour Face Cut Machining.

Operation examples:

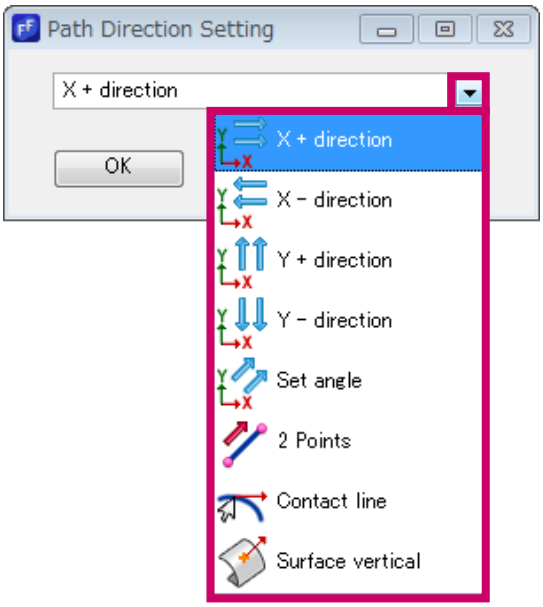
- **Projection Machining**



- **Specify Projection Direction (Contour Projection Machining)**

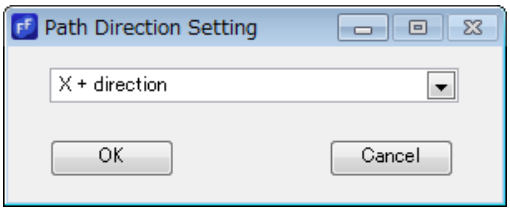


Set the path direction in the displayed [Path Direction Setting] screen.



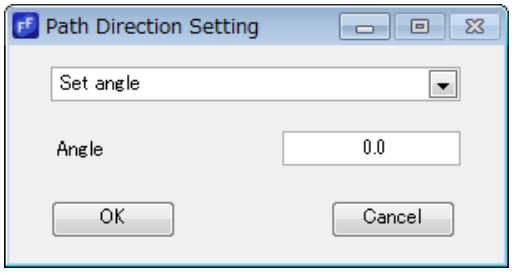
(1) X + Direction, X - Direction, Y + Direction, Y - Direction

Set the progress direction for the tool path to the X axis and Y axis (+/-) direction of the machining coordinate system.



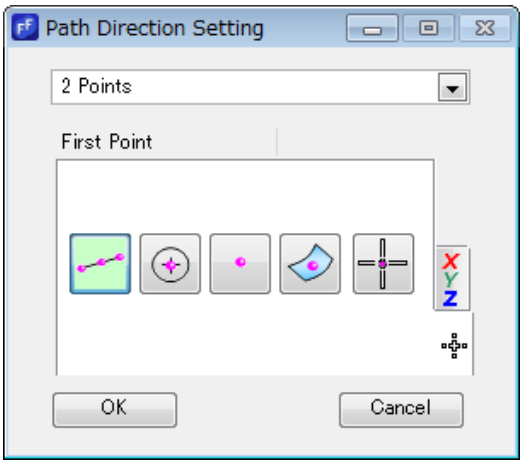
(2) Set Angle

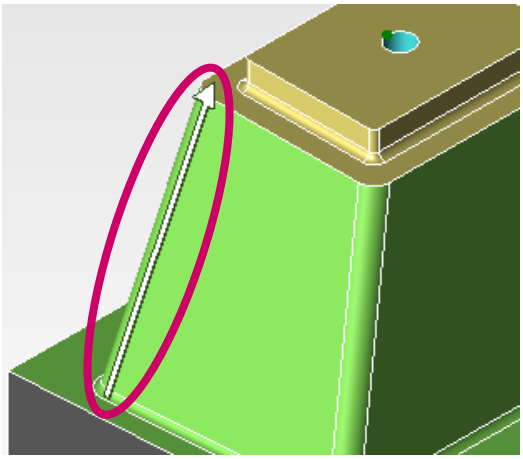
Specify an angle for the +X axis of the machining coordinate system, and set the path direction.



(3) 2 Points

Select two points on the geometry.
The path direction is set along the straight line that connects the two points.
The first point is the start point of the vector, and the second point is the end point of the vector.

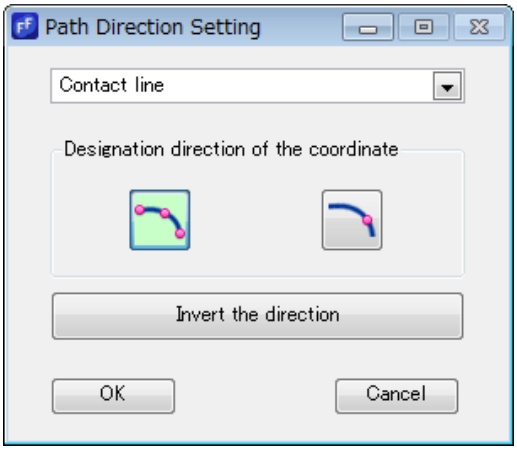


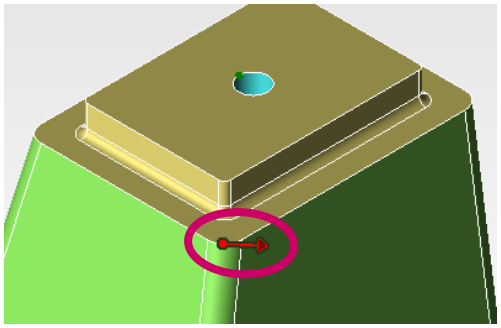


A white arrow showing the path direction is displayed when the second point is selected.

(4) Contact Line

Select a point on the curve or edge of the geometry.
The contact line direction of the selected point is set as the path direction.





A red arrow showing the path direction is displayed when you select a point on the curve or edge.

 **(Endpoint, Middle point)**

Select the endpoint or midpoint of the curve or edge.

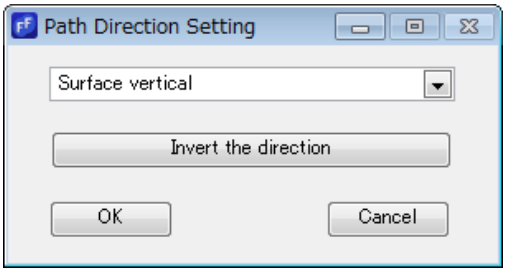
 **(Point on Curve)**

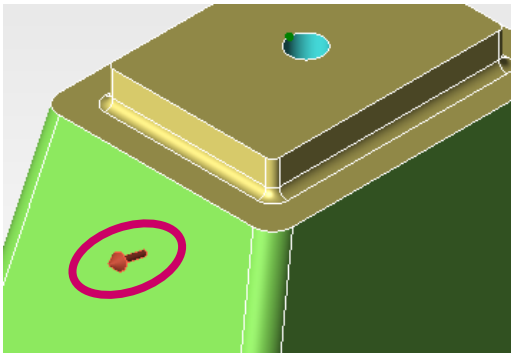
Select an arbitrary point on the curve or edge.

Invert the Direction
Reverses the path direction that is set in the point selection.

(5) Surface Vertical

Select a face on the geometry.
The normal direction of the selected face is set as the path direction.
Note that when the face is a curved surface, the normal direction of the selected position is the path direction.





On selecting a face, a red arrow is displayed to indicate the path direction.

Reverse Direction
Reverses the path direction that is set in the face selection.

■ Note

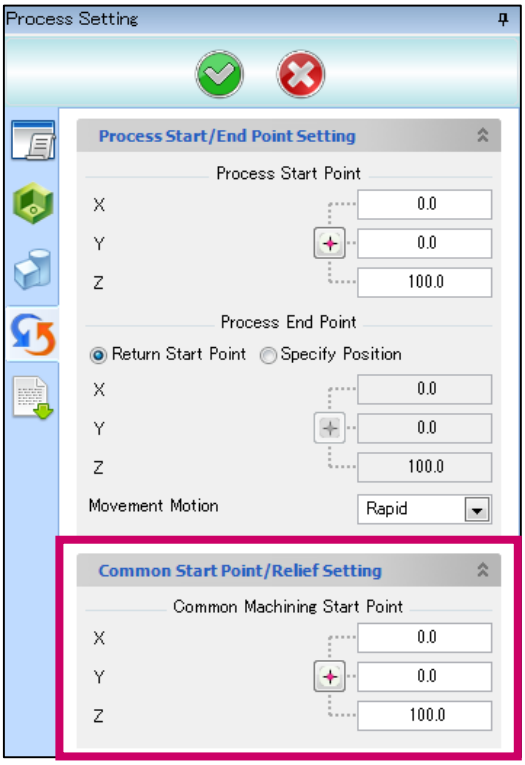
- An error message is displayed when the path direction and the tool axis direction are set to the same direction.
- If the index angle is set and "2 Points" or "Contact Line" is specified, and the setting is done away from the machining surface, the tool path may deviate from the specified details.

4. Addition of Common Machining Start Point Setting for Drilling

A common machining start point can now be set at the start/end point of drilling.
Since the start/end point of multiple drilling points can be defined together, the work time can be reduced, and rework due to missing the settings can be eliminated.

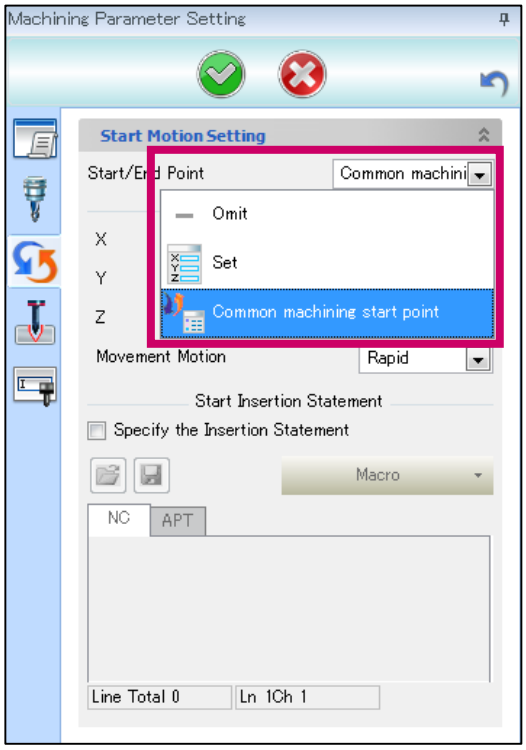
■ Setting Screen

- **Hole Process (Process Setting)**



[Common Start Point Setting] field has been added.

- **Drilling (Machining Parameter Setting)**



From the [Start/End Point] drop-down list, you can select "Common Machining Start Point".

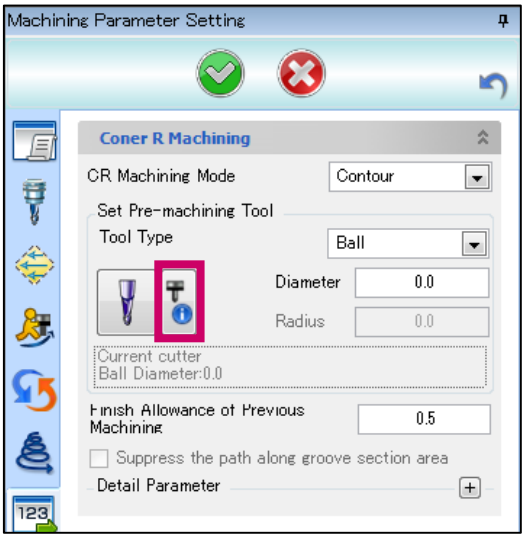
5. Addition of Function to Obtain Pre-Machining Information

The setting values of the pre-machining tool and finishing allowance of pre-machining for Corner R Machining and Contour Machining (when [Tool Load Priority] is specified in the machining order) can now be obtained from the pre-machining.
FFCAM 2018.2 or earlier required users to input a value for pre-machining.
In FFCAM 2019, the value set in the pre-machining can be obtained by just clicking the [Acquisition of Pre-machining tool/Finishing allowance] button.
Even if you forget the pre-machining information, you can set the pre-machining information without returning to pre-machining screen for confirmation.

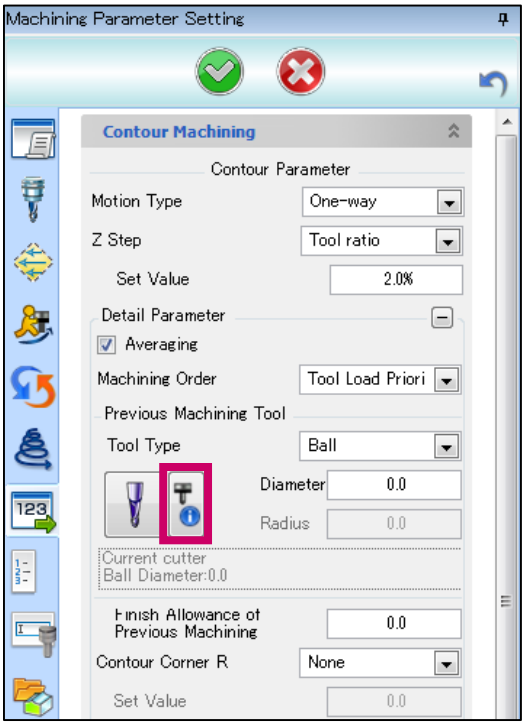
■ Setting Screen

[Acquisition of Pre-machining tool/Finishing allowance] button has been added.

● Corner R Machining



● Contour Machining



■ Note

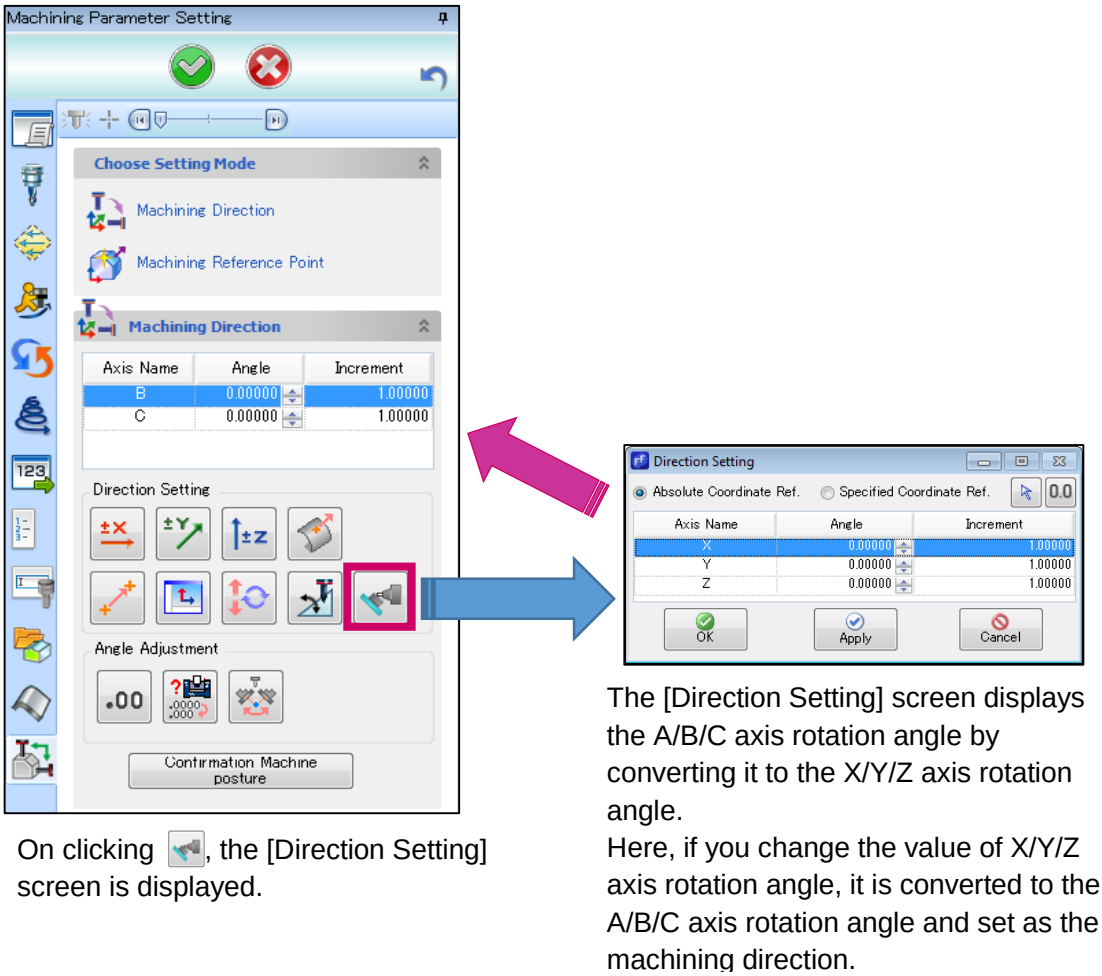
- If pre-processing information is not present, the [Acquisition of Pre-machining tool/Finishing allowance] button cannot be used.
- When a pre-machining tool is registered to Corner R Machining, the setting value cannot be acquired if the pre-machining tool is Lens Barrel tool.

6. Addition of Machining Direction Setting Function Independent of Machine Configuration

In index machining of FFCAM, the machining direction is set by directly entering the angle for the machine rotation axis. However, in the case of machines where the rotation axis is not parallel to the X/Y/Z axis, setting the required angle was sometimes difficult.

In FFCAM 2019, a function to set an angle based on X/Y/Z axis has been added.
You can set an angle regardless of the machine axis configuration.

■ Setting Screen



On clicking , the [Direction Setting] screen is displayed.

The [Direction Setting] screen displays the A/B/C axis rotation angle by converting it to the X/Y/Z axis rotation angle.
Here, if you change the value of X/Y/Z axis rotation angle, it is converted to the A/B/C axis rotation angle and set as the machining direction.

Direction Setting

Absolute Coordinate Ref.

The machining reference of the angle set in the [Direction Setting] screen is the coordinate system of the model.

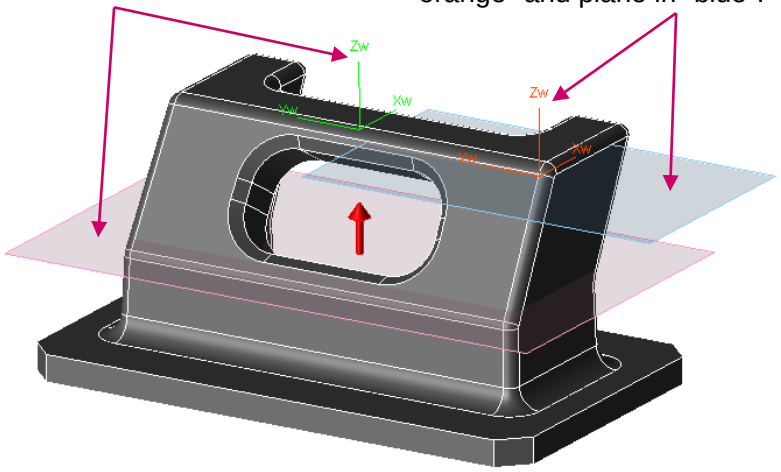
Specified Coordinate Ref.
The machining reference of the angle set in the [Direction Setting] screen is an arbitrary coordinate system.
When the [Specified Coordinate Ref.] radio button is selected, the value of the X/Y/Z axis becomes "0", and the rotation angle of each axis is input.

Sets the machining reference point of the angle set in [Direction Setting] screen to an arbitrary position on the model.

An "orange" coordinate axis and "blue" plane are displayed at the machining reference point.
* The default machining reference point is the same position as the model origin.

The geometry's coordinate axes are displayed in "green" and plane in "red".

The geometry coordinate axes set in the [Path Direction] screen are displayed in "orange" and plane in "blue".



When the  button is selected, the value of the X/Y/Z axis becomes "0", and the rotation angle of each axis is input.

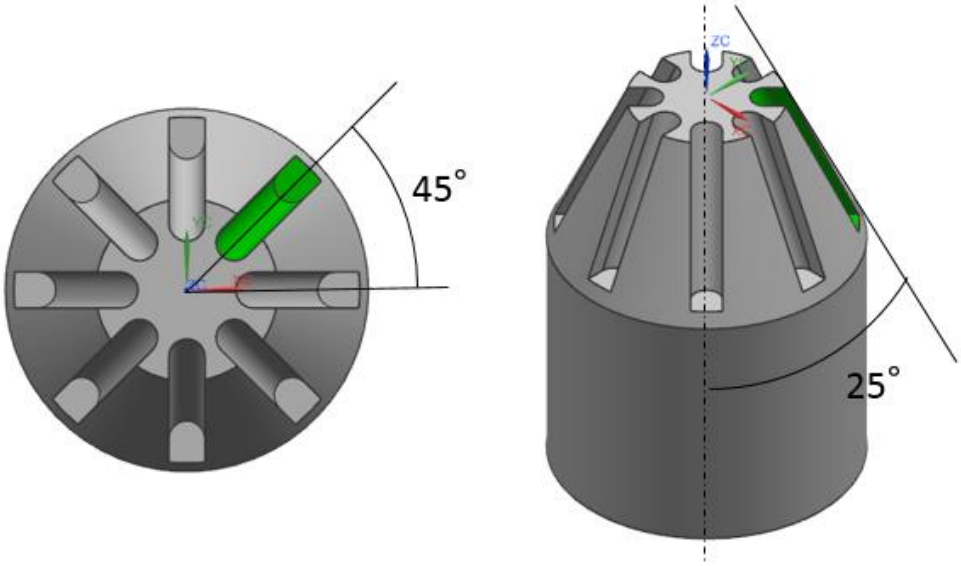
OK
The X/Y/Z axis rotation angles changed in the [Direction Setting] screen is converted to the A/B/C axis rotation angles and the machining direction is set.
(The [Direction Setting] screen closes.)

Apply
The X/Y/Z axis rotation angles changed in the [Direction Setting] screen is converted to the A/B/C axis rotation angles and the machining direction is set.
(The [Direction Setting] screen does not close.)

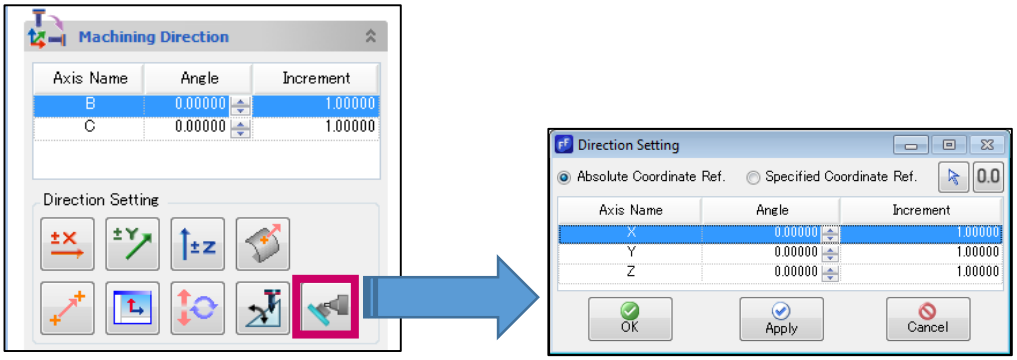
Cancel
Discards the changes and closes the screen.

■ Setting Example

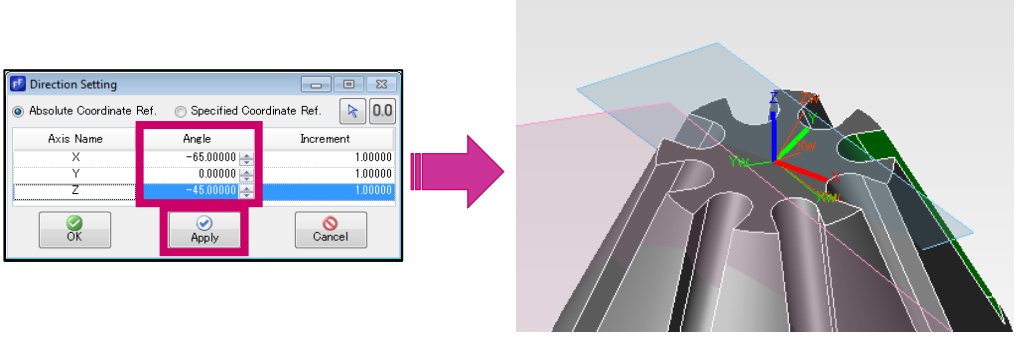
● Angle Setting When Machining the Groove in the Green Part of the Geometry Below



1. Click  .
   [Direction Setting] screen is displayed.



2. Enter "-65 degrees" for the X-axis rotation and "-45 degrees" for the Z-axis rotation and click [Apply].
   The "orange" coordinate axes and "blue" plane are oriented in the specified direction.



3. Click [OK] to close the [Direction Setting] screen.
   The values calculated for the machine rotation axis will be set in [Machining Direction].
   Also, in the graphics window, the coordinate system of the geometry (green coordinate axes and red plane) is changed to the set direction.

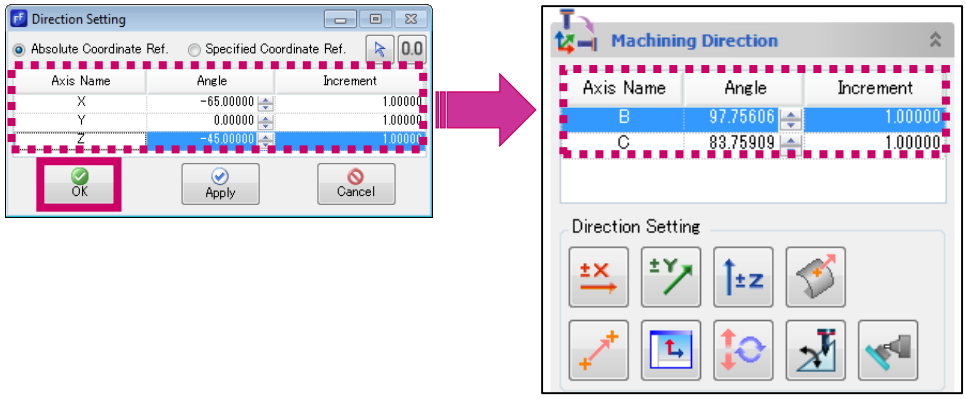
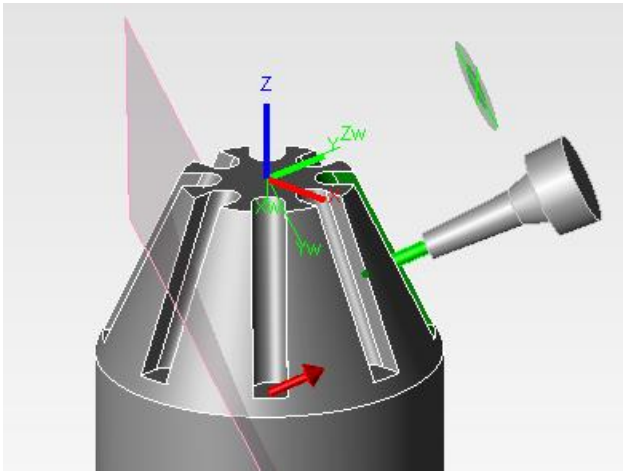


Image after setting



■ Note

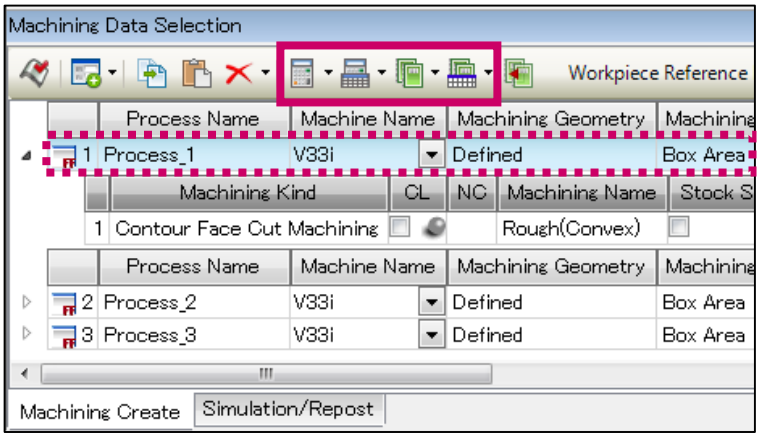
When you start the [Direction Setting] screen again after setting the indexed angle, the set indexed angle is converted to the X/Y/Z axis rotation angles again and displayed.
Since the input values of the [Direction Setting] screen are not stored, the X/Y/Z axis rotation angles may be different from the previous input values if you open the screen again.

7. Enhancement of Path Calculation Type When Process is Selected

In FFCAM 2018.2 or earlier, calculation methods such as batch calculation could not be executed during path calculation when a process was selected.
In FFCAM 2019, calculation methods such as batch calculation can be executed even when a process has been selected.

■ Setting Screen

[Batch Path Calculation] and [Blanket Batch Registration] can be executed when a process has been selected.



■ Note

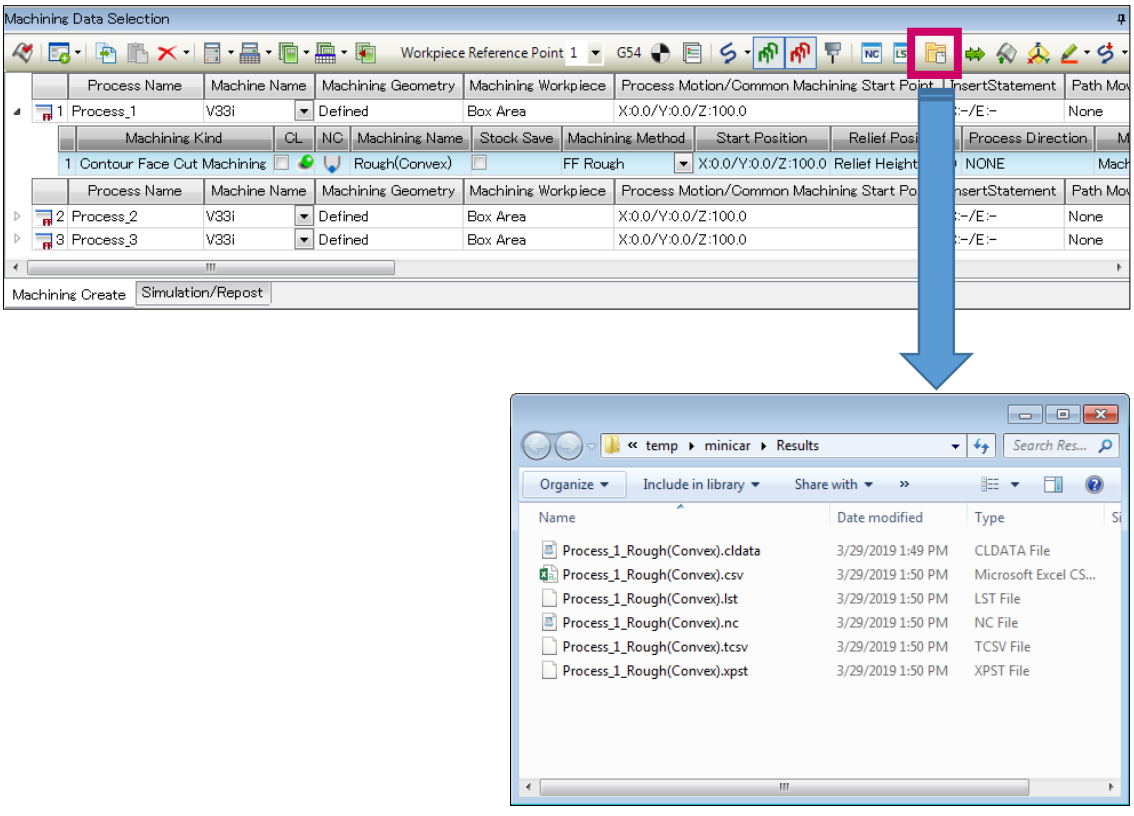
If multiple processes are selected, [Path Calculation], [Blanket Path Registration], [Batch Calculate], and [Blanket Batch Registration] cannot be executed.

8. Addition of Function to Open the Location for Saving FFCAM Data

To check files (such as NC, CSV, and INF) created with FFCAM, the folder where the files are stored may be opened in Windows Explorer.
If the files are saved in folders buried deep in the folder hierarchy, many folders have to be opened before you can reach the target folder.
In FFCAM 2019, you can now open the folder where the file used at present is saved with one click. You can easily access the required folder.

■ Setting Screen

From the toolbar of the Machining Data Selection screen, click on  (Open a designated folder) to open the folder where the file is saved.



* The [Open a designated folder] can also be selected from the menu that is displayed by right-clicking on the list of "Process" or "Machining" in the Machining Data Selection screen.

■ Explanation

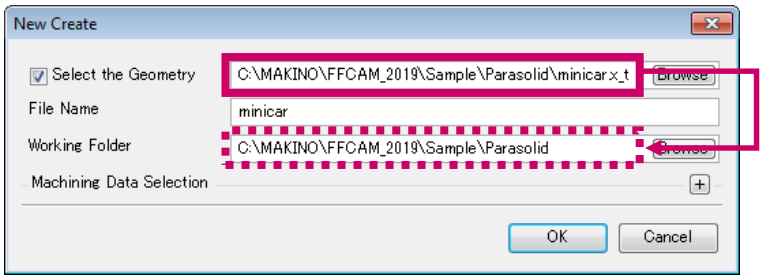
This function refers to the "Results" folder of the work data by default.
The reference location can be changed in the [Option Set Screen] -> [Ref. folder] -> [Open a folder] field.

9. Improved the Method of Specifying Working Folder When a New File is Created

When a new file is created with FFCAM, it is necessary to specify the location where the geometry data and FFCAM work data has to be created.
In the file operation method of FFCAM 2018.2 or earlier, when matching the designated folder of geometry data with that of FFCAM work data, you had to specify the same folder twice to select the geometry and working folder.
In FFCAM 2019, a function to automatically set the working folder to the folder where geometry data exists has been added. You can start the creation of a new file smoothly.

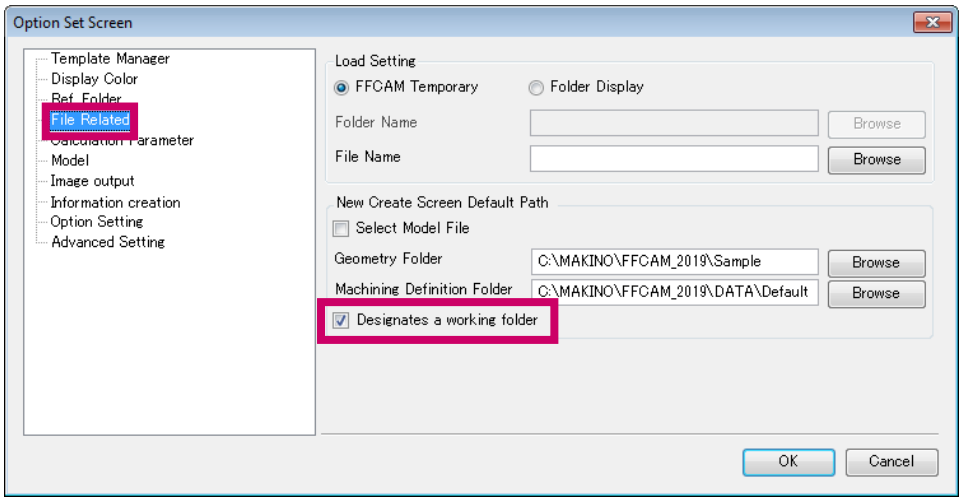
■ Setting Screen

When using the function to create a working folder automatically



By selecting a geometry, the working folder is set to the folder where geometry data exits.

When using the function to create a working folder automatically, set the working folder from [Option Set Screen] -> [File Related].



Designates a working folder

Check ON

Specify a working folder when creating a new file. (Old function)

Check OFF (Default)

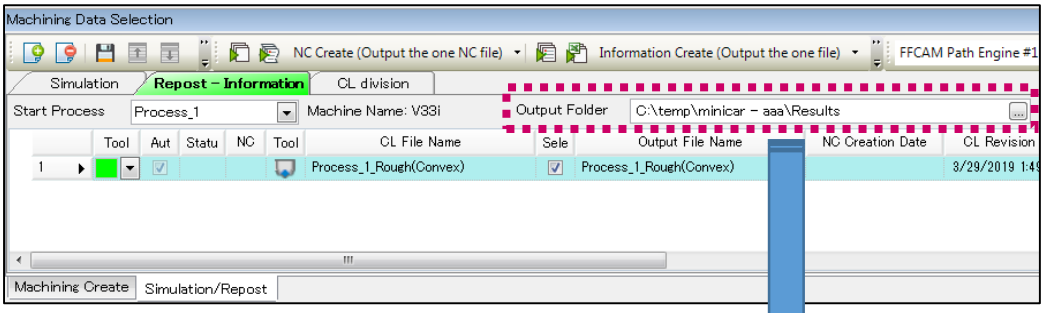
If geometry data is selected when creating a new file, the working folder is set to the folder where geometry data exits. (New function)

10. Change in NC File Output Destination During Reposting

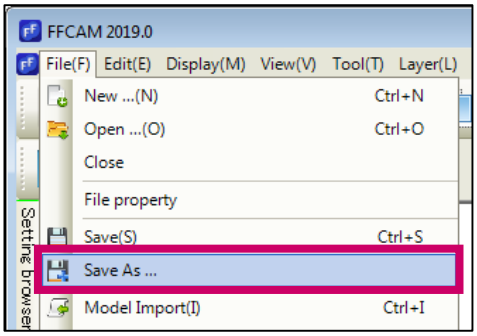
In FFCAM 2018.2 or earlier, when working data was saved as new, the location of the file before saving by Save As was specified as the NC file output destination at repost.
For this reason, it was not possible to identify the output NC file, and the output destination of the NC file had to be changed.
In FFCAM 2019, the NC file output destination at repost is now automatically changed to the Results folder of the file that is saved by Save As.

■ Setting Screen

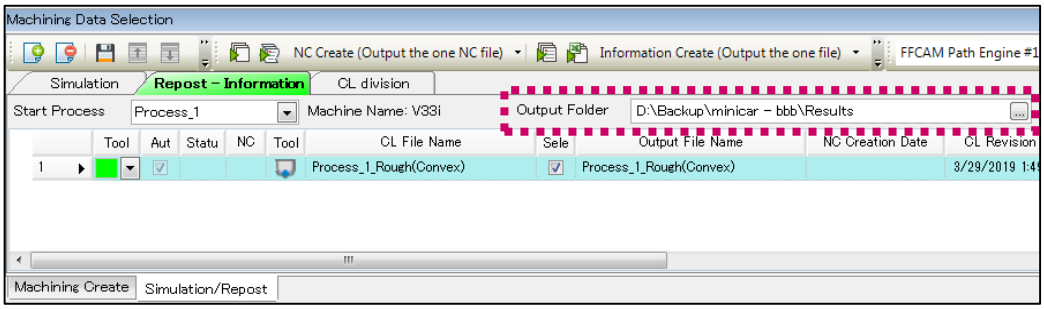
NC file output destination before saving working data by Save As



Save working data by Save As



The NC file output destination is changed to the Result folder of the working data after saving by Save As.

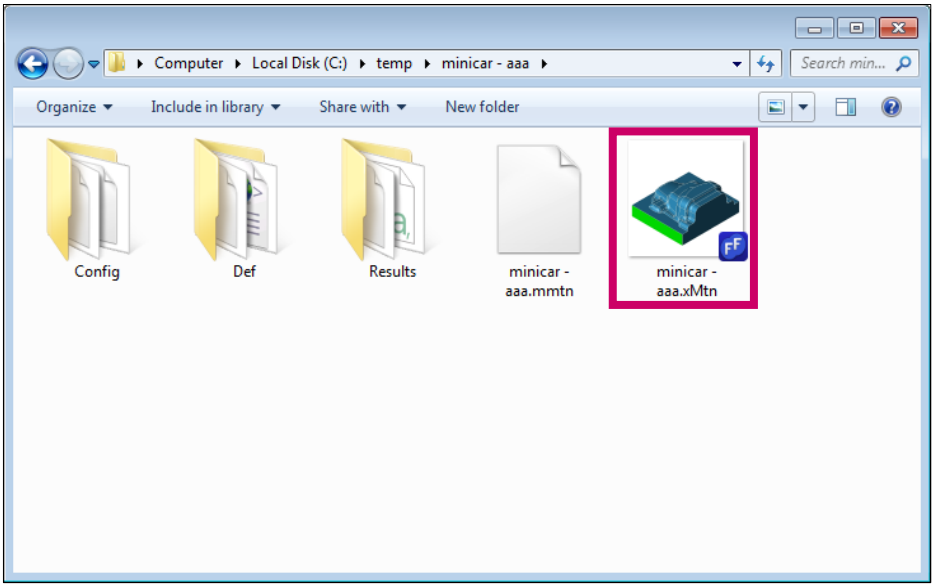


11. Addition of Function to View Files as Thumbnails

The saved FFCAM file (.xMtn) is now displayed with a thumbnail of the geometry image in Windows Explorer. Since the model geometry can be confirmed without opening the file, making file selection easier.

■ Explanation screen

xMtn files are displayed with a thumbnail.



■ Note

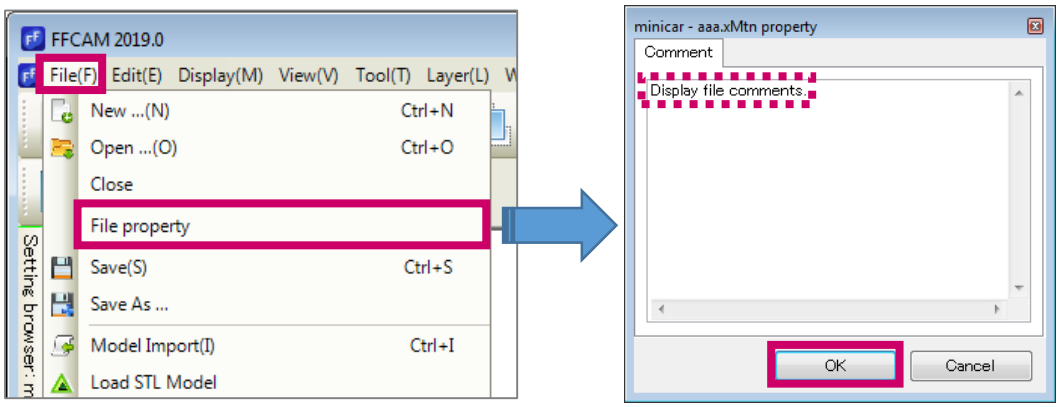
Thumbnails are not displayed for xMtn files that have not been saved in FFCAM 2019.

12. Addition of Comment Display Function for Files

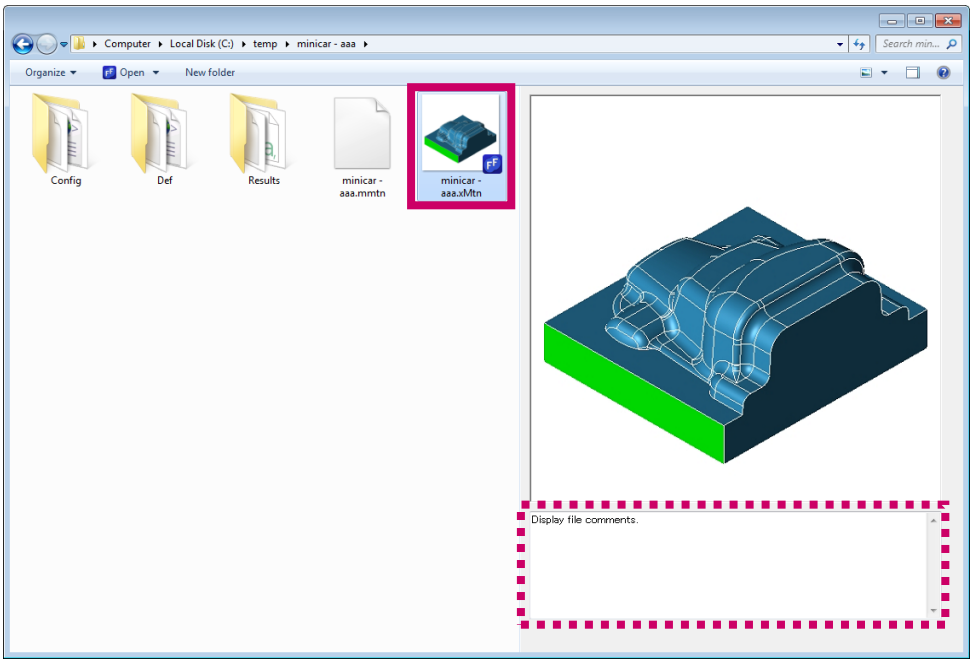
Comments can now be added to FFCAM files (.xMtn).
The comments that are added can be viewed and checked in Windows Explorer.
Using it together with the *Function to View Files as Thumbnails* of the previous chapter makes file selection in FFCAM easier.

■ Setting Screen

Go to [File] -> [File property] to open the window for entering comments.
Enter the comment and click [OK].



When you select the xMtn file in Windows Explorer, the entered comment is displayed in the preview window.



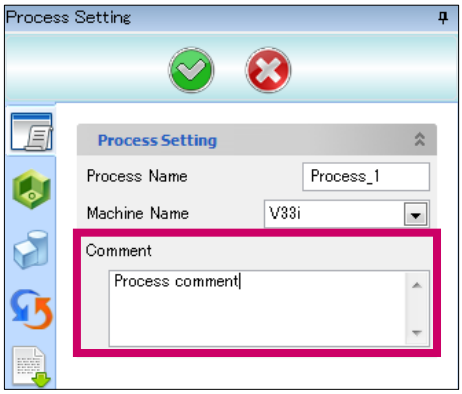
13. Addition of Comment Display Function for Process

Comments can now be added to processes.
You can record and convey the details and production intention of the machining that has been created by entering the explanation for the machining, and matters to be handed over.

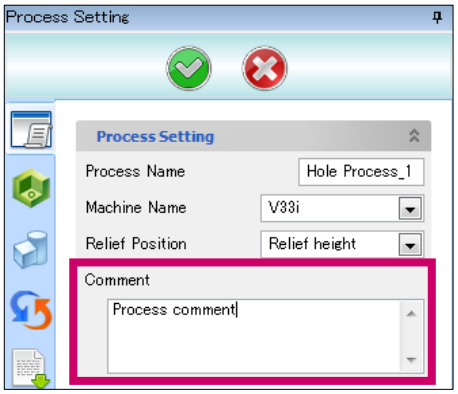
■ Setting Screen

Comments can be entered and confirmed in the [Process Setting] screen.
Up 40 double-byte characters or 80 single-byte characters can be entered for comments.

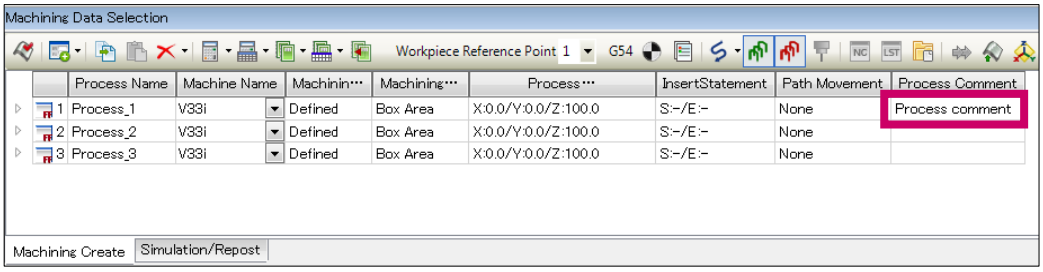
- **Milling Machining**



- **Drilling**



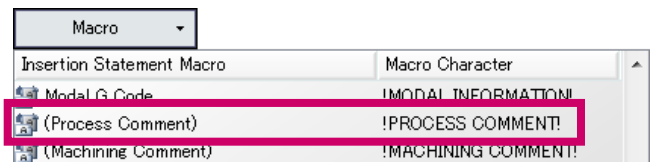
Comments can be entered and confirmed in the Machining Data Selection screen.



■ Supplementary Note

A command to output "Process Comment" has been added to the insert statement macros and output parameter lists on the FFPOST Edit Form.

- **Insert Statement Macros**

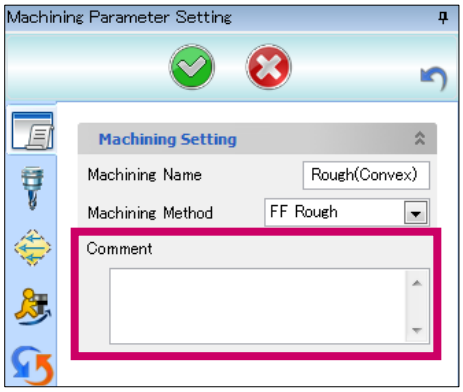


14. Enhancement of Comment Display Function for Machining

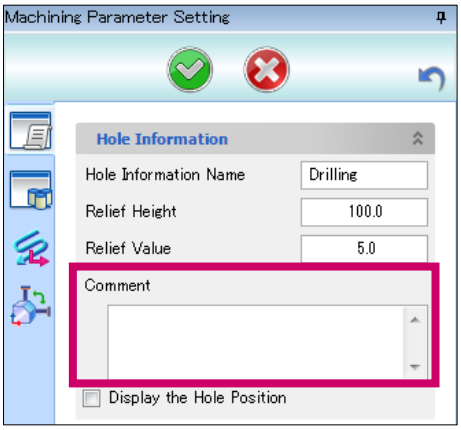
The size of the entry field for machining comments has been increased.
Even if a long comment is entered, the whole comment can be viewed.
Up 40 double-byte characters or 80 single-byte characters can be entered for comments.

■ Setting Screen

- **Milling Machining**



- **Drilling**

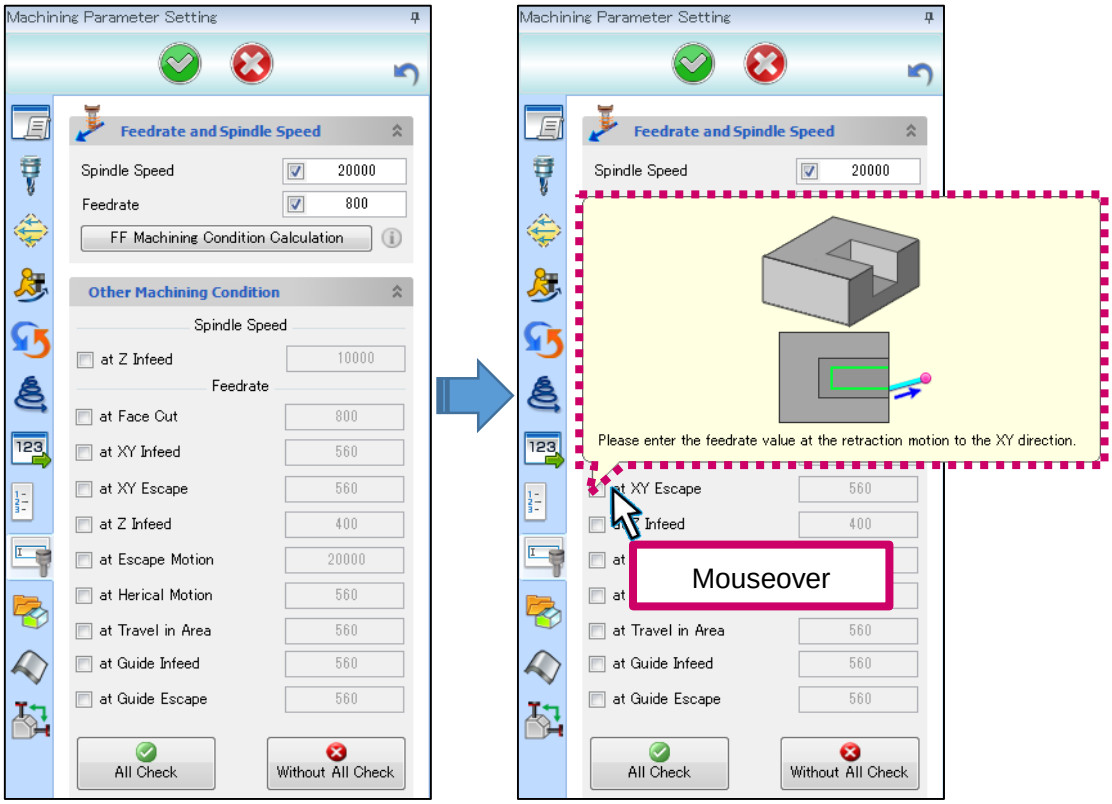


15. Enhancement of Tip Comments for Parameters

The number of parameters displaying tip comments for machining definition has been increased.
The summary of parameters whose usage is difficult to understand only by their name can be confirmed without opening HELP.
In FFCAM 2019, tip comments of the machining condition settings have been added.
In future versions, additional tip comments will be added.

■ Operation Screen

Examples of tip comment display:

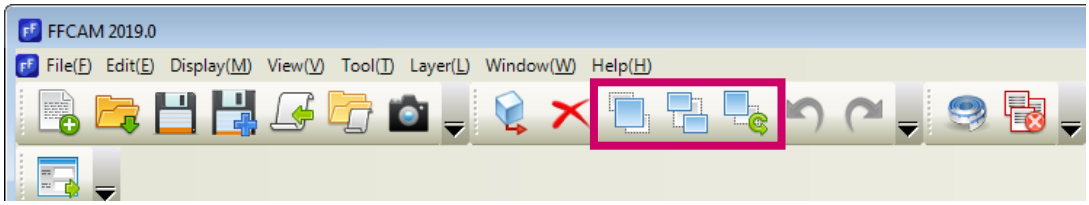


16. Addition of Blank Operation Icon to Toolbar

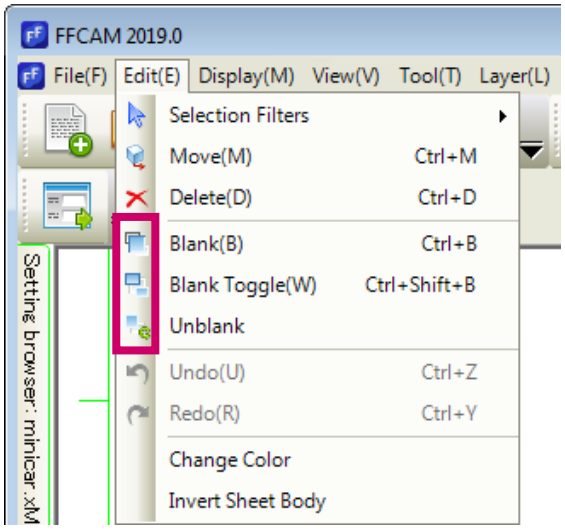
A blank operation icon has been added to the toolbar.
In FFCAM 2018.2 or earlier, blank operations had to be done from the [Edit] menu.
In FFCAM 2019, since the blank function can be executed from the toolbar, the time required to execute the blanking operations can be reduced.
The functions displayed as icons are easy to find and the operability is improved.

■ Setting Screen

Icons for blank operations are provided on the toolbar.
(Functions of each icon are the same as that of the Edit menu. See the figure below.)



The icons are also displayed in the menu.

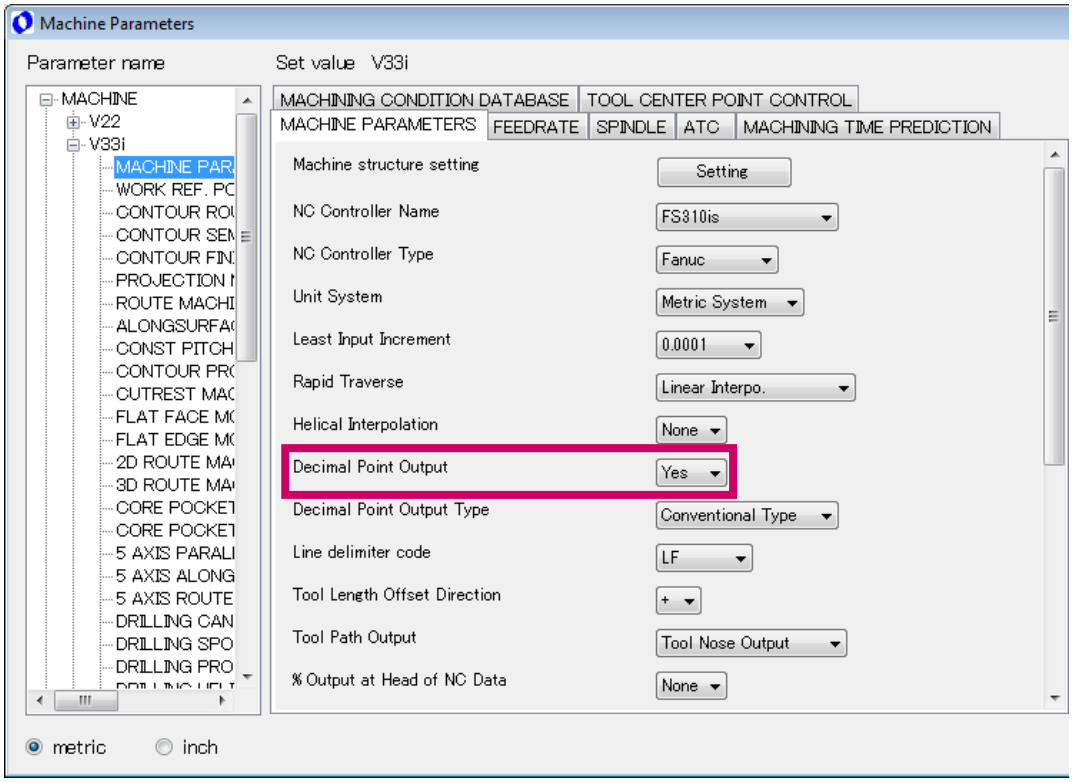


17. Change the Default Machine Parameters

The default of [Decimal Point Output] of the machining parameter has been changed to "Yes".
* Applies to all machines registered to the machining parameter by default.

■ Setting Screen

Example: Machining parameter of machine "V33i"

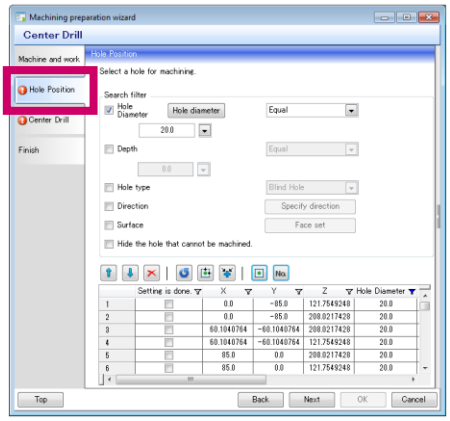


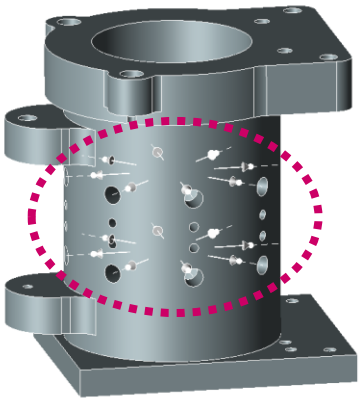
18. Addition of Function to Set the Hole to the Center of Geometry Direction in the Machining Preparation Wizard

A function to set the hole direction to the center of the geometry has been added to the setting list of [Hole Position] of the Hole Machining Data Creation of Machining Preparation Wizard. You can align the direction of the selected hole to the center direction of the geometry.

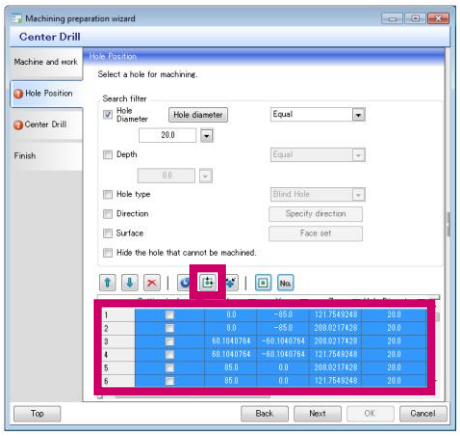
■ Operation Screen

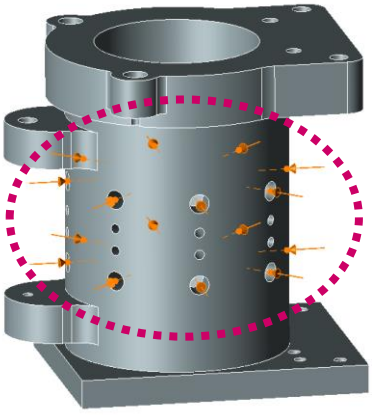
1. When setting with [Hole Position] of Machining Creation Wizard, the hole direction of each hole is displayed by an arrow.
   * In the example below, the hole direction is from the inside to the outside of the geometry.





2. Select a hole to modify the hole direction from the selection list.
   Click the (Hole direction is towards the center of Box) button.
   The direction of the selected hole will be towards the center of the geometry (from the outside to the inside).





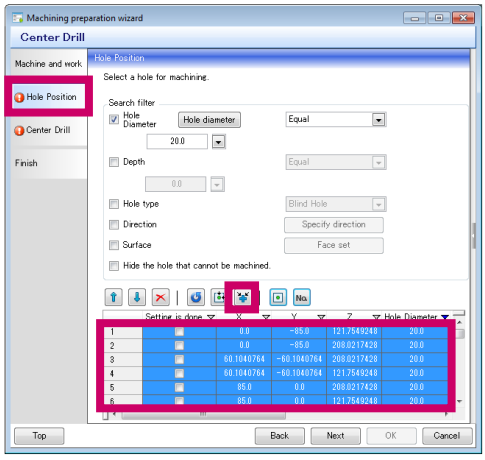
* A marker can be displayed at the center of the geometry.
  For details, refer to *20. Addition of Function to Display the Center of Geometry in the Machining Preparation Wizard*.

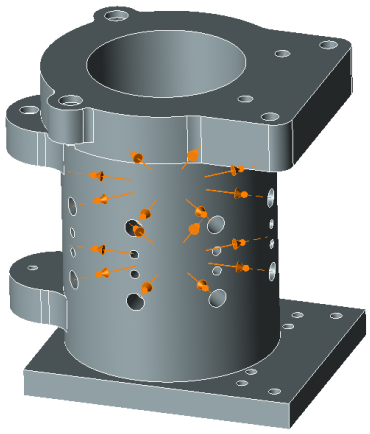
19. Addition of Function to Set the Direction of Hole to the Specified Point in the Machining Preparation Wizard

A function to set the hole direction to the specified point has been added to the setting list of [Hole Position] of the Hole Machining Data Creation in Machining Preparation Wizard. You can align the direction of the selected hole to the specified direction.

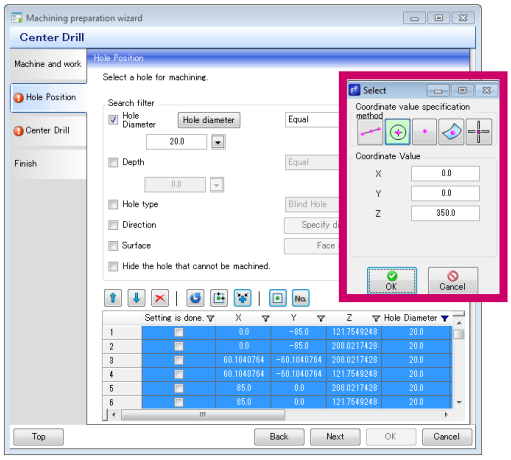
■ Operation Screen

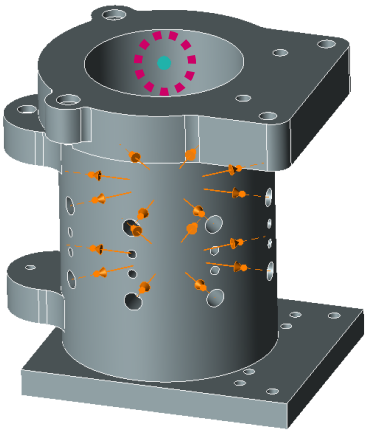
1. Select a hole to modify the hole direction from the selection list of [Hole Position] settings.
   Click the  (Hole direction is towards the designated point).
   The Select screen appears.



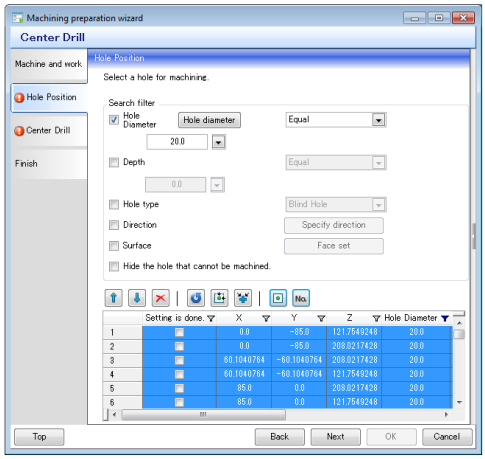


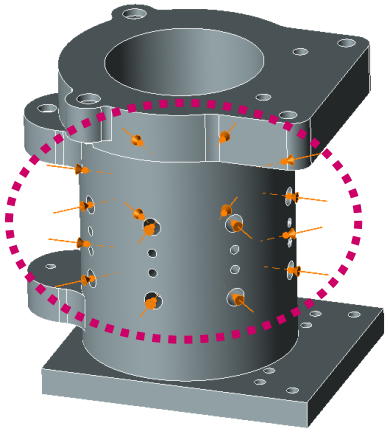
2. Select [Coordinate value specification method] in the Select screen.
   Pick and specify a point on the geometry that will serve as the reference for the hole direction.
   A marker appears on the specified position.





3. Click the [OK] button in the Element Selection screen.
   The direction of each hole will be towards the direction of the specified position (Example: From the outside to the inside).



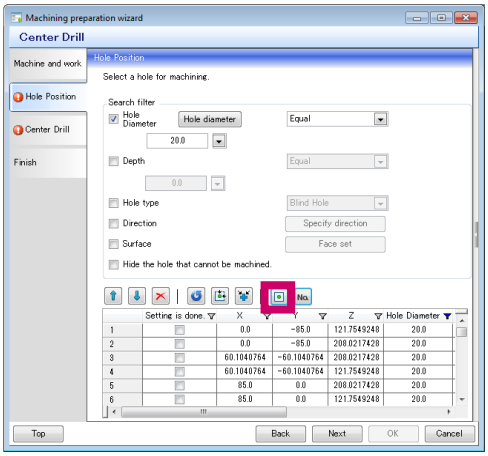


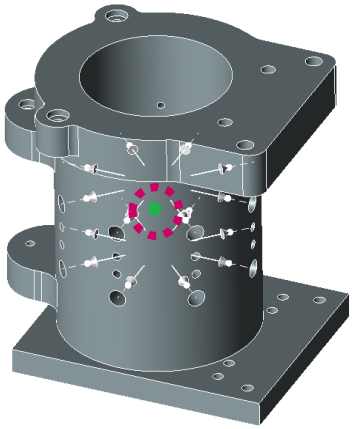
20. Addition of Function to Display the Center of Geometry in the Machining Preparation Wizard

A function to indicate the geometry center using a marker has been added to the setting list of [Hole Position] of the Hole Machining Data Creation of Machining Preparation Wizard. The geometry center can be confirmed in the graphics window.

■ Operation Screen

Click the (Indicates the center of BOX) button in the selection list of [Hole Position] settings.
A marker appears at the geometry center.



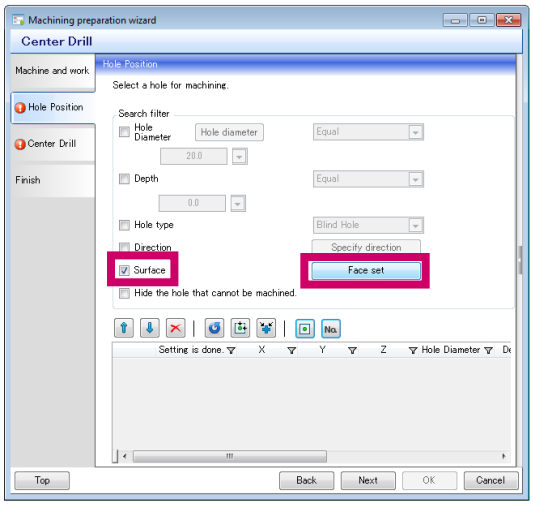


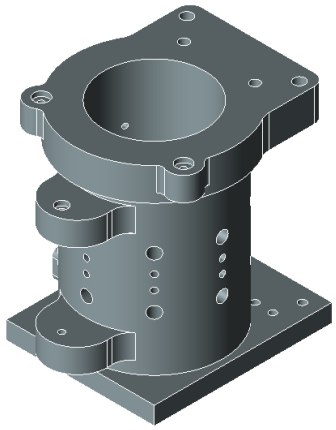
21. Addition of Surface Filter Function in the Machining Preparation Wizard

The [Surface] (Surface Normal Direction) item has been added to the search filters of [Hole Position] of the Hole Machining Data Creation in Machining Preparation Wizard.
When selecting a hole to be machined, only holes on the selected surface can be registered.

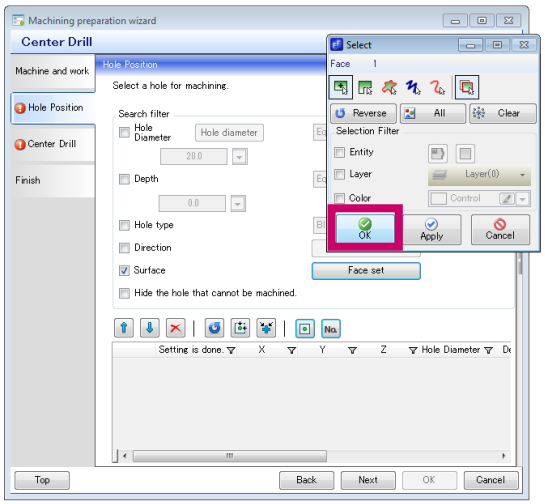
■ Operation Screen

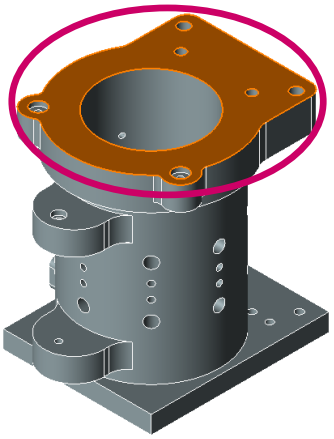
1. Select the [Surface] checkbox in the search filter of [Hole Position] setting, and click the [Face set] button.
   The Element Selection screen appears.



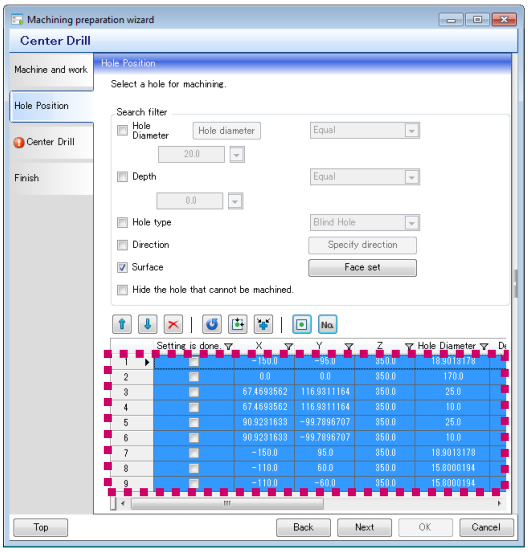


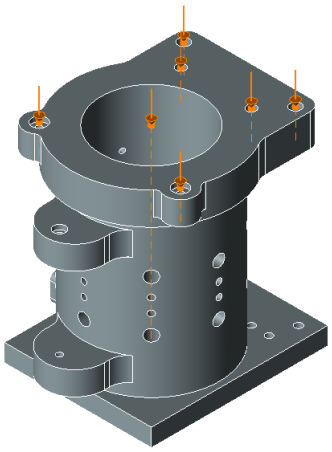
2. Select a surface of the geometry, and click the [OK] button in the Element Selection screen.





3. Only holes on the selected surface are registered in the setting list.



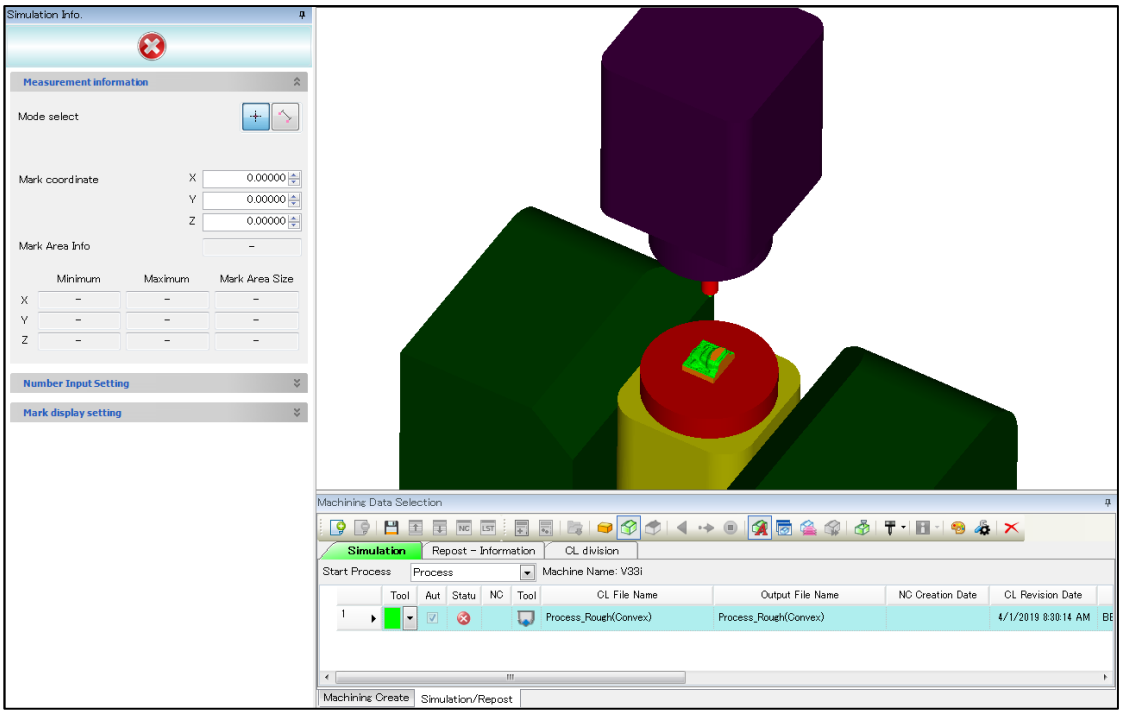


22. Improved Machine Display Function During Measurement in Simulation

Machine geometries are no longer hidden even when measurement information is used during simulations in machine display.
Measurement can be carried out while the geometry is setup has been completed.

■ Setting Screen

Machine geometries are not hidden even if the measurement data is used.

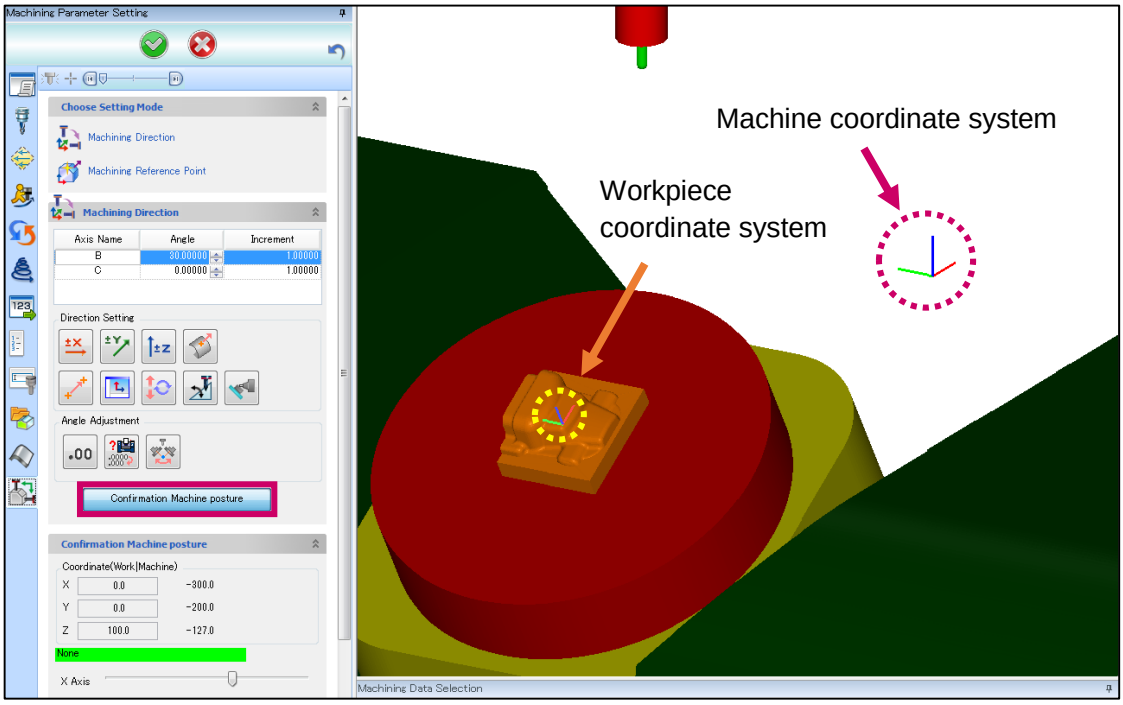


23. Addition of Work Coordinate System Display Function in the Confirmation Machine Posture

The coordinate system image after the rotation axis operation is now displayed during the confirmation machine posture.
The workpiece geometry and direction of the machining reference point after the rotation axis operation can be confirmed.

■ Setting Screen

[Confirmation Machine posture] screen displays images of the workpiece coordinate system and machine coordinate system, respectively.

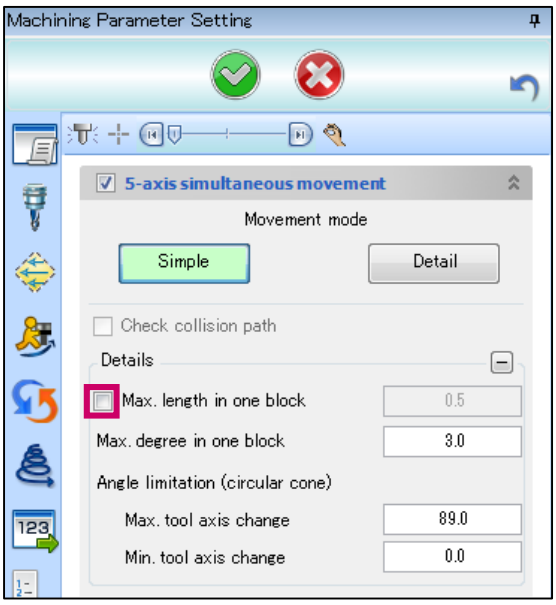


24. Change in the [Max. Length in One Block] Parameter in 5-Axis Simultaneous Movement

For 3-5 axis conversion machining, it is now possible to select whether to use [Max. length in one block] parameter of 5-axis simultaneous movement.
"OFF (do not use)" is selected by default. This parameter is no longer mandatory, so set this parameter as needed.

■ Setting Screen

A checkbox has been added to the [Max. length in one block] parameter of 3-5 axis conversion machining.



Max. length in one block

Check OFF (Default)
The parameter of Max. length in one block is not used.

Check ON
The parameter of Max. length in one block is used and set.

■ Note

- When importing data of FFCAM 2018.2 or earlier, the checkbox is selected ("ON"), and the setting value is also carried over.
- For 5-axis dedicated simultaneous machining, the checkbox is not added.

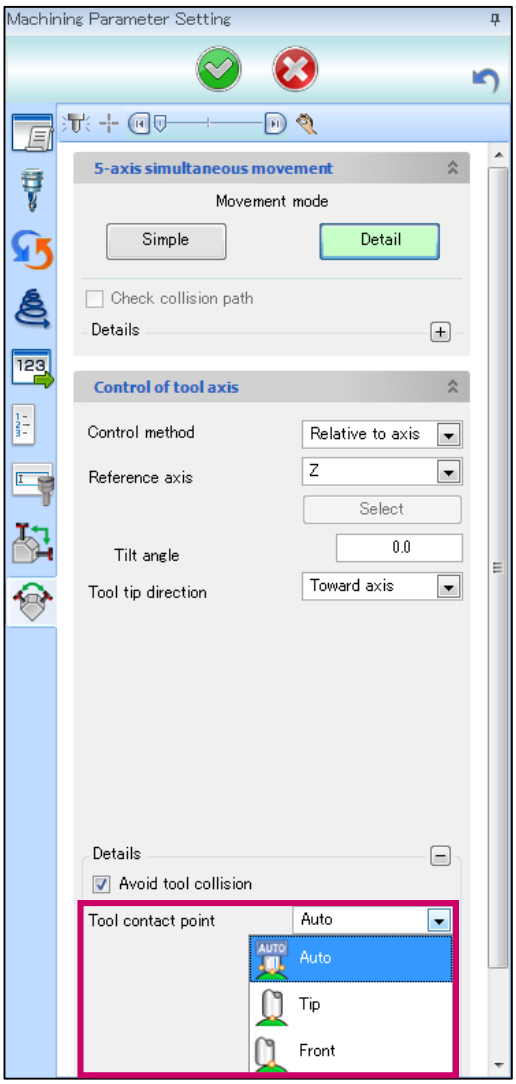
25. Addition of Function to Specify Tool Contact Points in 5-Axis Dedicated Simultaneous Machining

In 5-axis dedicated simultaneous (Parallel, Along Surface, Route) machining, the portion where the tool touches the machining work can be now specified.
When using flat end mill and bullnose end mill tools, you can specify the part of the tool to be used such as "Corner" and "Bottom face".

■ Setting Screen

● 5-Axis dedicated simultaneous machining

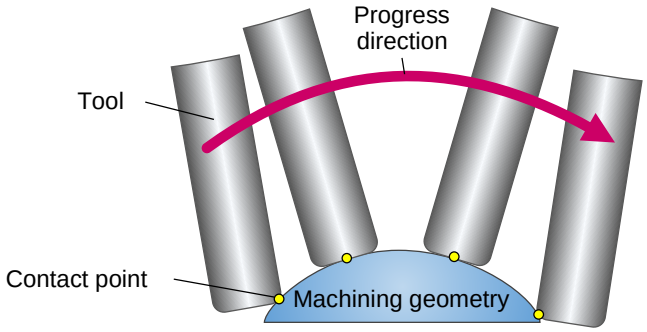
[Tool contact point] parameter has been added to [5-axis simultaneous movement] -> [Detail] -> [Control of tool axis]. Select the tool contact point from the drop-down list.



Tool Contact Point

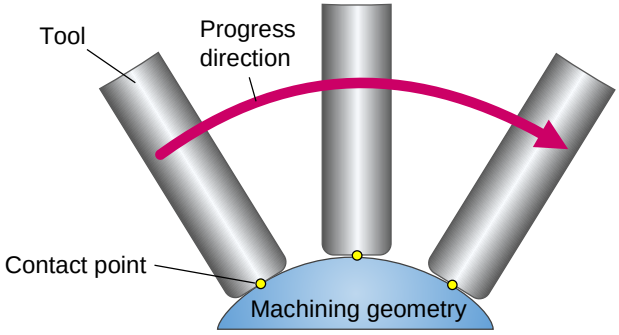
Auto (Default)

The contact position between the tool and drive surface is determined automatically according to the tilt of the tool.



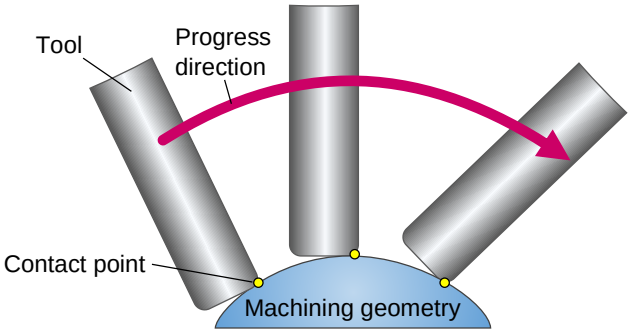
Tip

The tool position is set so that the tool tip center touches the drive surface.
A collision may occur between the tool and workpiece depending on the geometries of the tool and drive surface.
Therefore, use this parameter with [Avoid tool collision] enabled.



Front

The tool posture is set so that the front blade touches the drive surface to the progress direction of the tool.
A collision may occur between the tool and workpiece depending on the geometries of the tool and drive surface.
Therefore, use this parameter with [Avoid tool collision] enabled.



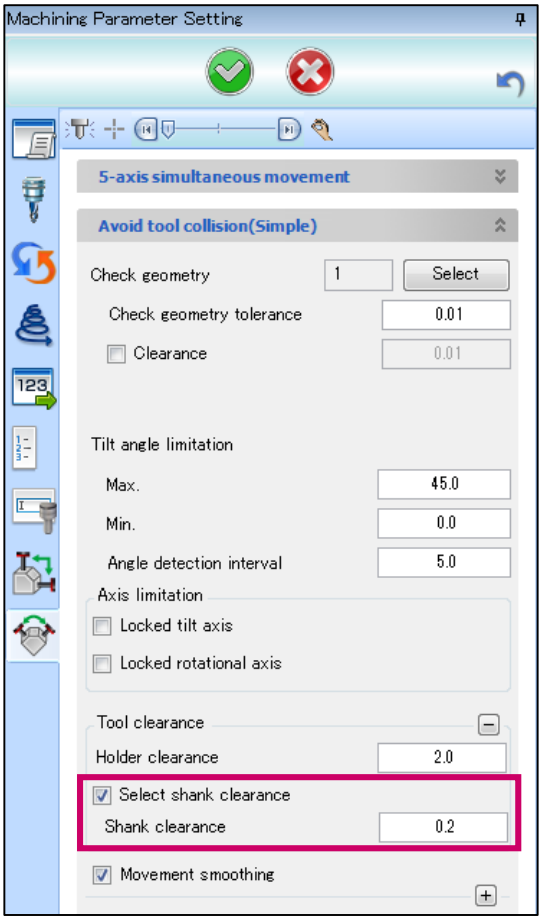
26. Change in Default Tool Clearance of Avoid Tool Collision in 5-Axis Simultaneous Movement

The default tool clearance of "avoid tool collision" in 5-axis simultaneous movement has been changed.
If a small-diameter pencil neck tool was set and calculated with the default tool clearance, the holder clearance setting value is applied to the tool tip, and an error might occur.
By specifying the shank clearance for the tool tip in advance, calculations can now be performed with the default value for a tool with small-diameter and a short neck length.

■ Setting Screen

- **Avoid Tool Collision in 5-Axis Simultaneous Movement**

  In FFCAM 2019, the tool clearance [Select shank clearance] is set to "ON" by default.
  Shank clearance is applied to a tool tip.

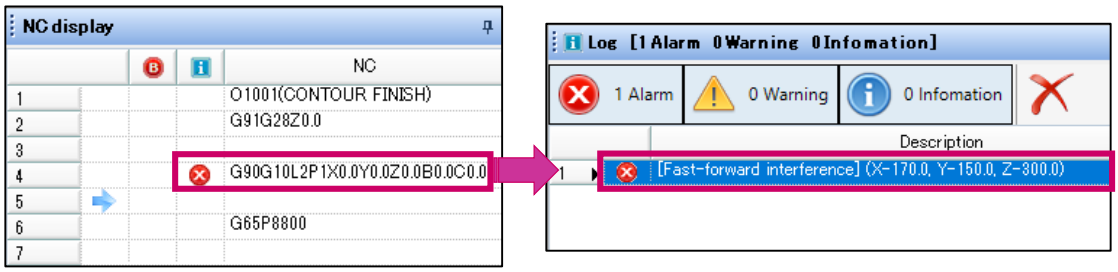


27. Addition of Detail Parameter for "Ignored Code" In Machine Simulator

Due to the G and M codes that are not supported by Machine Simulator, unexpected operations may be performed during simulation.
In Machine Simulator V.2.1, you can specify code that is not executed for simulation (Ignored Code) to avoid such unexpected operations.
Codes registered as ignored code are read as NC program but do not operate during the simulation.

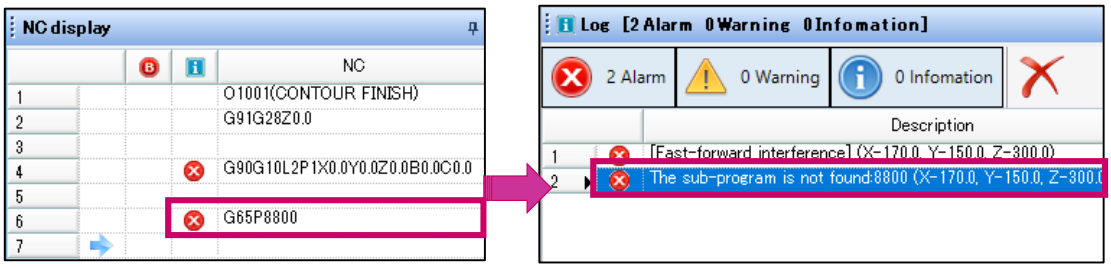
■ Unexpected Operations Due to Unsupported Codes

Example 1: Malfunction of simulation by argument X/Y/Z of G10 macro (offset value setting of workpiece coordinate system)



Since G10 is a macro to set values in the machine, there is no actual movement of the axis.

Example 2: Error output by macro programs not registered in NC Data Center



Sub programs were not registered in NC Data Center because these macros do not affect the simulation.

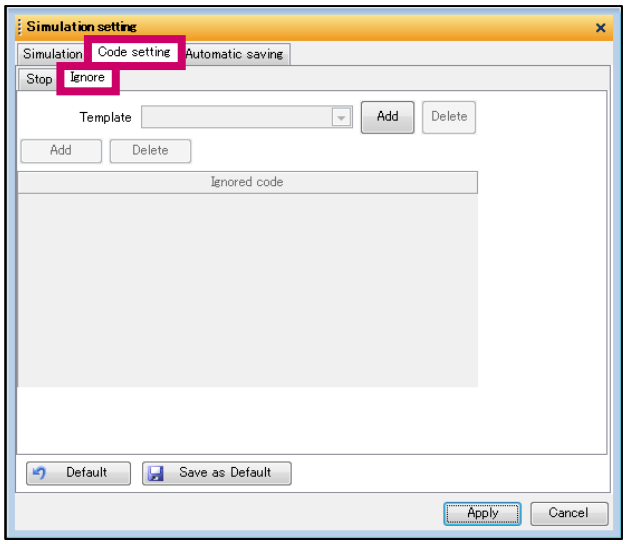
"Ignored Code" has been set to avoid the above operations during simulation.

■ Setting Screen

● Registration of Ignored Code

Set the ignored codes on the [Ignore] tab in [Code setting] tab of the [Simulation setting] screen.

* The conventional [Stop setting] is set from the [Stop] tab of the [Code Setting] tab.

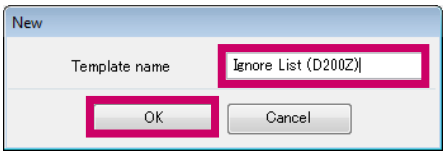


Template

A template is required to set ignored codes.
Create a new template, or select an existing template from the drop-down list.

Add

Creates a new template.
On clicking [Add], the [New] screen is displayed.
Enter a template name in the [New] screen and click [OK].



Delete

Deletes the selected template.

[Ignored code] setting field

When a new template is created, or an existing template is selected, the [Ignored Code] setting field becomes available.

Add

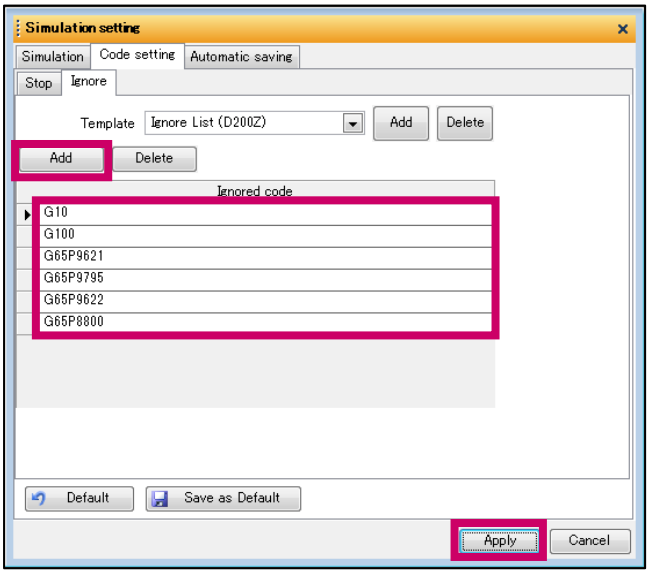
Adds a row to enter the ignored code.

Delete

Deletes the selected row.

(a) Setting image

1. Click [Add] to add a row.
2. Enter the code to be ignored on the row which is added.
3. Repeat steps 1 to 2 depending on the number of codes to be entered.
4. After entering all the codes to ignore, click on [Apply].
   The template is created and registered in the Machine Simulator.



(b) Examples for entry of ignored codes

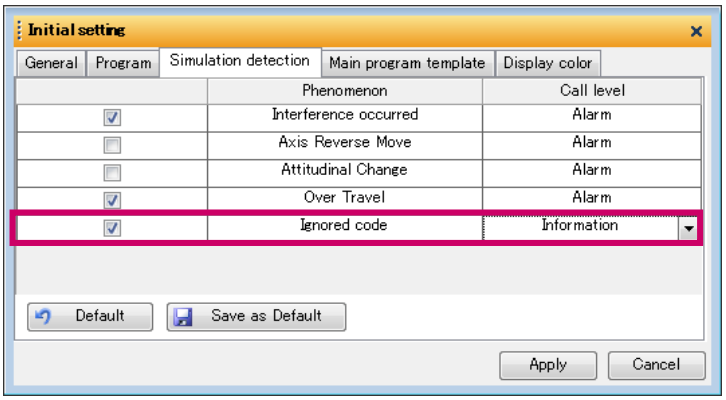
Blocks containing strings entered as ignored code are excluded from the simulation operation.
* Up to 198 single byte characters can be entered for ignored codes.

Ignored code (example)	Target to ignore (example)
G100	G100 X20. Y30. H6
G65	G65 P9621 H1. D1. Z3.0 block G65 P8800
P8800	G65 P8800
G68 X	G68 X20.0 Y15.0 block * If the order of argument specification is different (Example: G68 Y20.0 X15.0), the block is not subject to the ignored code.

● Enabling/Disabling ignored codes

Enable/disable an ignored code on the initial settings screen.
Also, set the method to output to the log when an ignore code is read.

Perform these setting from [Initial setting] -> [Simulation detection] tab -> [Phenomenon] -> [Ignored code].



Checkbox

Toggle to enable/disable ignored code.

Check ON (Default)
Enables ignored code.
Simulation of blocks containing ignored code is not executed.

Check OFF
Disables ignored code.
Simulation of blocks containing ignored code is executed.

Call level

Set the call level for log output from the drop-down list.
The default value is "Information".

* For details on the call level, see *28. Enhancement of Call Level of Logs in Machine Simulator*.

■ Note

Combination of ignored code and stop code or breakpoint

If stop codes or breakpoints have been set to a block with ignored codes, the simulation is paused at that block.

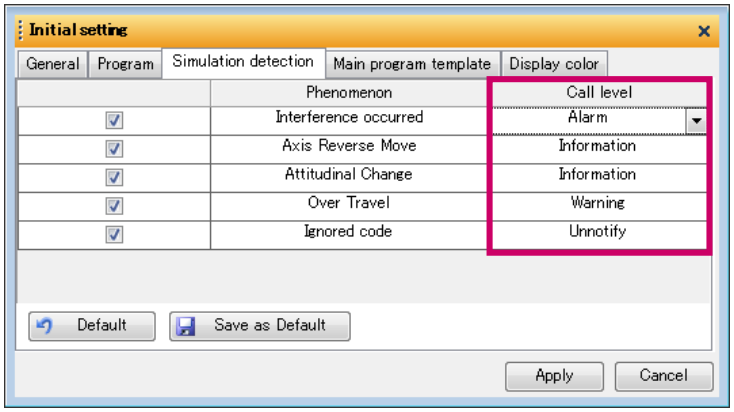
28. Enhancement of Call Level of Logs in Machine Simulator

"Information" has been added to the call level of logs of the Machine Simulator.
Previous "Warning" and "Alarm" of call level were linked with the function to pause the simulation.
"Information" logs only information without pausing the simulation.
You can record the phenomenon of stopping and confirming the simulation and the phenomenon of logging only information without stopping the simulation.

■ Setting Screen

- **Initial Settings -> [Simulation detection] tab**

  Set the call level for each phenomenon from the drop-down list.



Call level

Alarm
Outputs an alarm if a phenomenon occurs.

Warning
Outputs a warning if a phenomenon occurs.

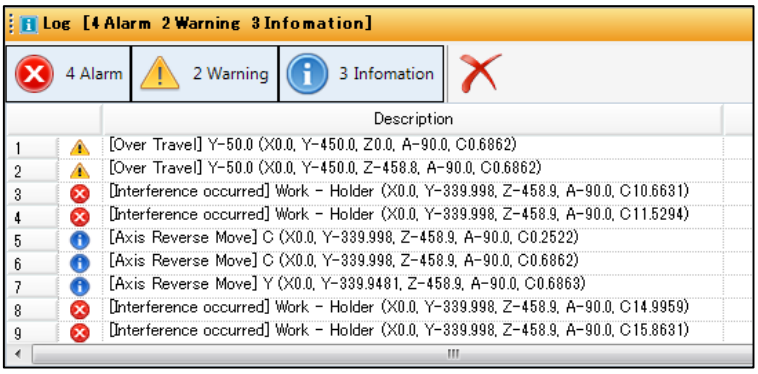
Information
Outputs only information if a phenomenon occurs.

Unnotify
Outputs nothing even if a phenomenon occurs.
* "Unnotify" can be selected only for [Ignored code].
  For details on [Ignored Code], see the previous chapter *Addition of Detail Parameter for "Ignored Code" In Machine Simulator*.

■ Example of Log Output

- **Log screen**

Each phenomenon is displayed at the call level set in [Initial Settings].

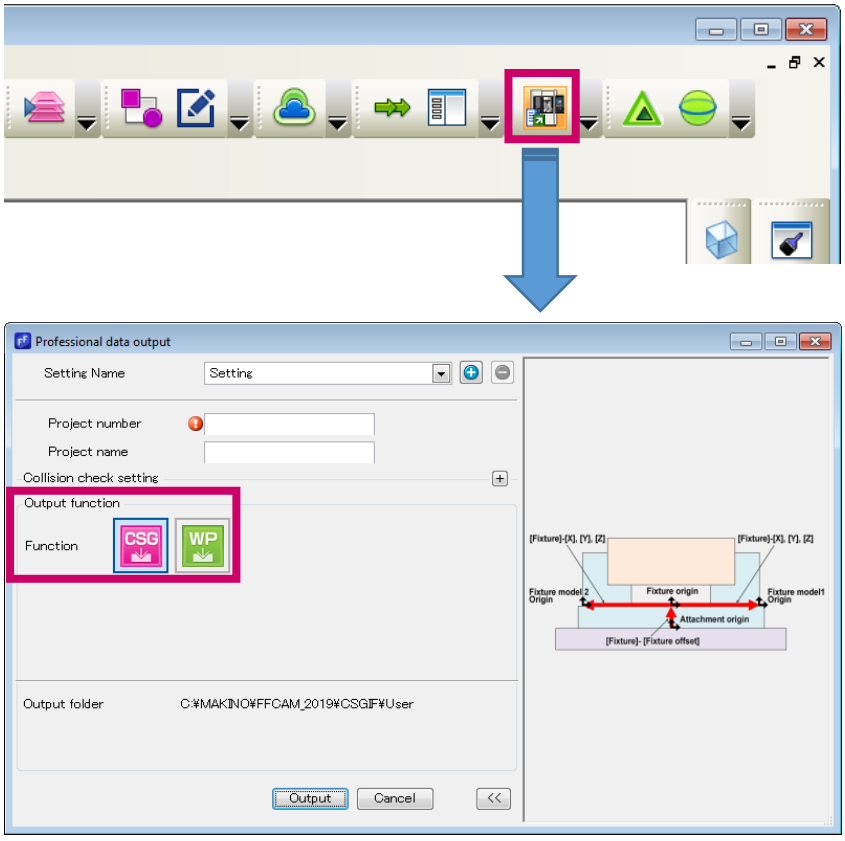


29. Addition of Professional Data Output Function

An interface that outputs data for "Collision Safeguard" and "Work Plan" has been integrated as [Professional Data Output] function.

■ Setting Screen

On clicking [Professional data output] icon, the [Professional data output] screen is displayed.



■ Output Function

- Function

Click [CSG] (Collision Safeguard) or [WP] (Work Plan) icon
and select the output function.

What is Collision Safeguard?

It is a software which prevents interference and damage from occurring in the machining chamber of the machining center.
While the machine is running, the software "prefetches the machine operation" and "simulates it" to stop the machine before any interference occurs.

What is the Work Plan?

It is a function that can collectively manage data, such as NC programs used as main/sub programs and data on Collision Safeguard (CSG), required to operate the machine.

---

30. Update of Parasolid Supported Version

Parasolid V.30 is now supported.