FCA240N2A-2403EN

FFCAM 2024

Description of New Functions

MAKINO

Promise of Performance

Preface

This manual describes the functions added to MAKINO FFCAM 2024 and how to use them.

Created on

March 2024

List of Functions Added to FFCAM 2024

1. Enhancement to Allow Along-Surface Machining to be Performed on Multiple Areas with a Single Machining Data

The along-surface machining function has been enhanced so that a single machining data can be used for multiple areas.

The along-surface machining in the previous FFCAM requires the creation of one machining data for each area.

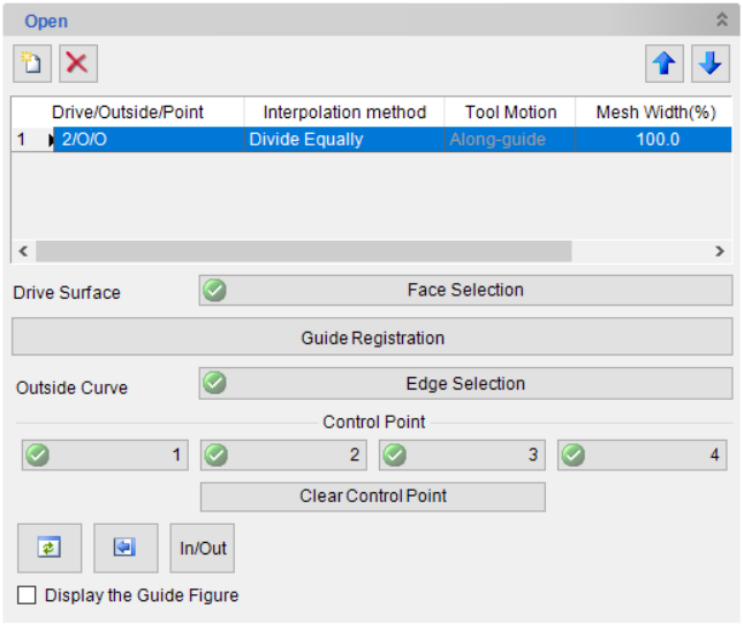
Along-surface machining in FFCAM 2024 allows machining setting for multiple areas using a single machining data. This function reduces the man-hours required to create machining data for each area and facilitates data management.

■ Change of Setting Screen

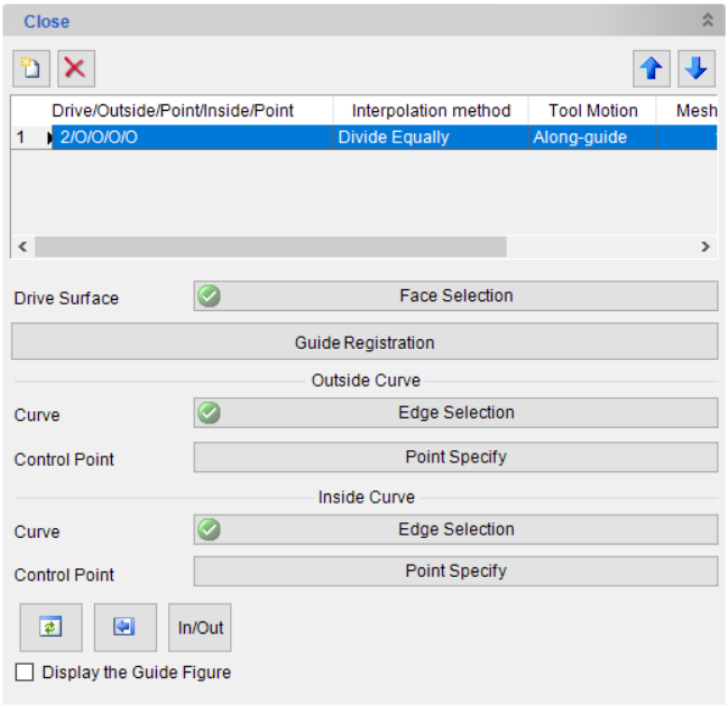
In FFCAM 2024, the [Along-Surface Machining] setting screen has been changed with the enhancement of the function.
The corresponding Along-Surface Machining Area List and setting items are displayed, depending on the along-surface geometry type (Open or Close).
Multiple along-surface machining areas can be registered in a list format, and each registered machining area can be individually set. This enables multiple area settings to be managed with a single machining data.

* Machining data with different along-surface geometry types (Open or Close) must be created separately.

Setting screen for Open



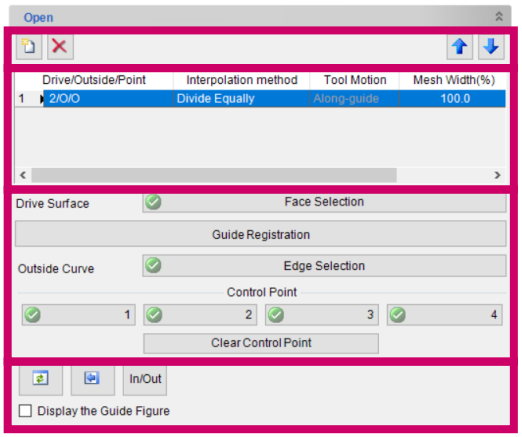
Setting screen for Close



* With the changes to the setting screens, the [Step] parameter has been moved to the [Set] panel.

■ Setting Screen

Open



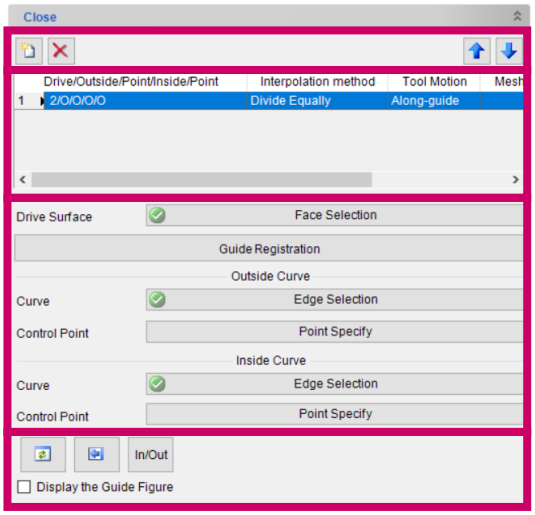
(1)

(2)

(3)

(4)

Close

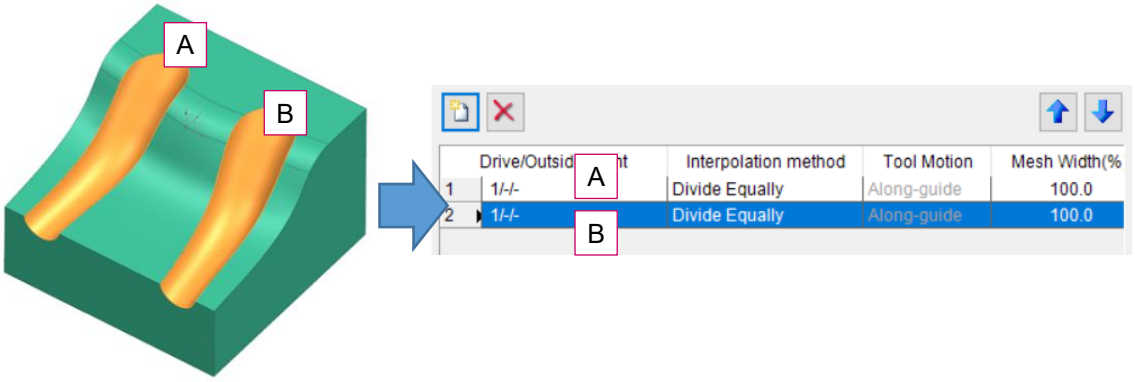


(1) Along-Surface Machining Area List, Control Buttons

Add Button

Press the Add button to display the [Select Surface] screen.
Specify the drive surface of the model on the [Select Surface] screen.
The specified drive surface is registered to the [Along-Surface Machining Area List].

If multiple separate surfaces are specified, a drive surface is registered to the list for each separate surface.



Delete Button

Deletes the selected row.

Move Up/Down Button

Moves the selected rows up or down.

(2) Along-Surface Machining Area List

Drive/Outside/Point (Open)

Displays the information as follows.
Number of elements in the drive surface/Whether outside curves are registered/Whether control points are registered
A "○" mark is displayed when curves or control points are registered.

Drive/Outside/Point/Inside/Point (Close)

Displays the information as follows.
Number of elements in the drive surface/Whether outside curves are registered/Whether outside control points are registered/Whether inside curves are registered/Whether inside control points are registered
A "○" mark is displayed when curves or control points are registered.

Interpolation Method

Displays the interpolation method registered. The interpolation method can be changed directly in this column.

Tool Motion

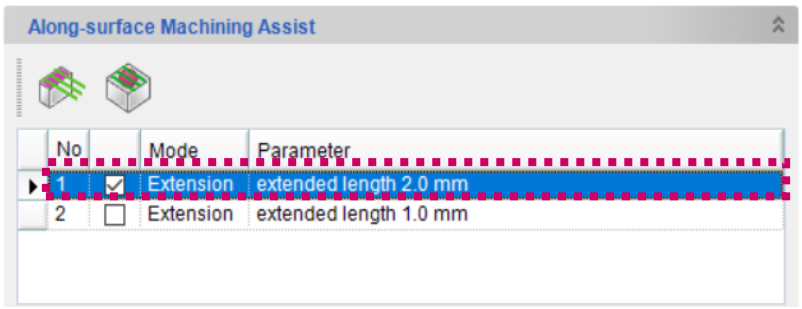
Displays the type of the registered tool motion. The type of tool motion can be changed directly in this column.

Mesh Width

Displays the registered grid width value as a percentage. The value of the grid width can be changed directly in this column.

Toolpath Assist

Specify whether to use [Along-Surface Machining Assist].
Add a check mark in the check box to use it.
In the [Along-Surface Machining Assist] screen, Toolpath Assist is applied to surfaces with a check mark.



(3) Registration of Drive Surface, Curve, and Control Points

Open

Set the following parameters for each machining area registered in [Along-Surface Machining Area List].
The setting method and setting details are the same as those of the previous FFCAM.

- Drive Surface/Face Selection
- Guide Registration
- Outside Curve/Edge Selection
- Control Points/1, 2, 3, 4
- Clear Control Points

Close

Set the following parameters for each machining area registered in [Along-Surface Machining Area List].
The setting method and setting details are the same as those of the previous FFCAM.

- Drive Surface/Face Selection
- Guide Registration
- Outside Curve/Edge Selection
- Outside Control Points/Point Specify
- Inside Curve/Edge Selection
- Inside Control Points/Point Specify

(4) Curve Editing

Common to Open/Close

Set the following parameters for each machining area registered in [Along-Surface Machining Area List].
The setting method and setting details are the same as those of the previous FFCAM.

- Change Start Point
- Change Path Direction
- Change In/Out
- Display the Guide Figure

■ Notes

- Open and Close cannot be consolidated into a single machining data. The machining data must be created separately.
- The tool path is output in order from the top row of the [Along-Surface Machining Area List]. The tool path of movement between setting data of the rows is output in "Relief Height".
- When along-surface machining data of previous versions of FFCAM is loaded into FFCAM 2024, the data is registered to [Along-Surface Machining Area List] as one machining area data. The along-surface machining data created by FFCAM 2024 cannot be read by previous versions of FFCAM.

2. Enhancement of Registration Function for Cylindrical Geometry in Machining Workpieces

In previous FFCAM, only cylinders parallel to the Z-axis direction (part coordinate system) could be registered as cylindrical geometry in the machining workpiece.
FFCAM 2024 can now register cylindrical geometries parallel to the X-axis and Y-axis directions as machining workpieces.

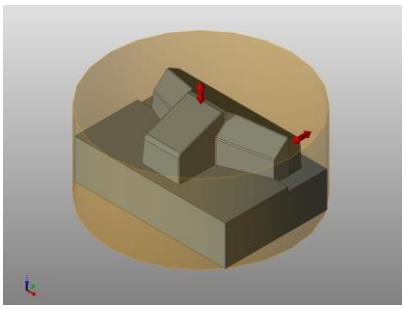
■ Setting Example

[Set of Machining Workpiece] / [Set] of [Cylinder] form

Set the placement direction of machining workpiece with cylindrical geometry in [Cylinder Axis Direction] on the [Set] screen.
The placement direction is specified by the axis direction (part coordinate system).

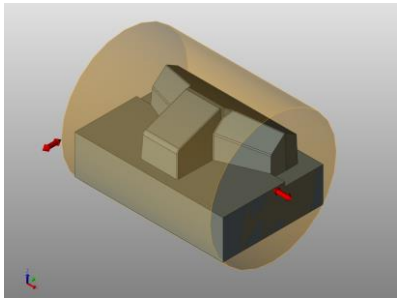
Cylinder Axis Direction: Z axis (default value)

The setting is the same as for previous parameters. Cylindrical geometry in the Z-axis direction is placed.



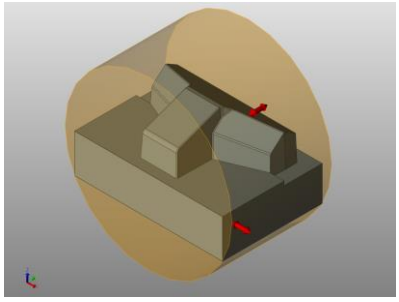
Cylinder Axis Direction: X axis

This is a new setting. Cylindrical geometry in the X-axis direction is placed.



Cylinder Axis Direction: Y axis

This is a new setting. Cylindrical geometry in the Y-axis direction is placed.



■ Set Screen

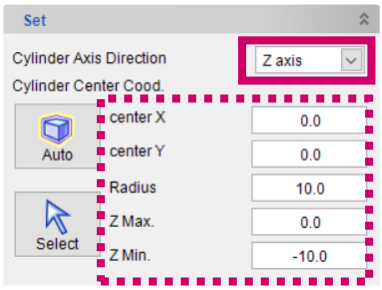
How to Specify Cylindrical Geometry

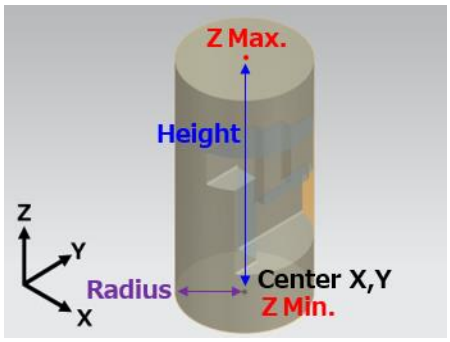
This section explains how to specify the direction and size of cylindrical geometry in the [Set] screen.

The coordinate points for setting the [Center Point], [Maximum], and [Minimum] change depending on the axis direction selected in [Cylinder Axis Direction]. The differences are shown in the figures below.

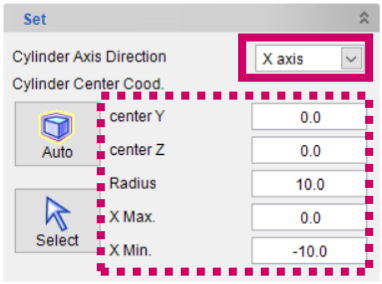
Cylinder axis direction: Z axis

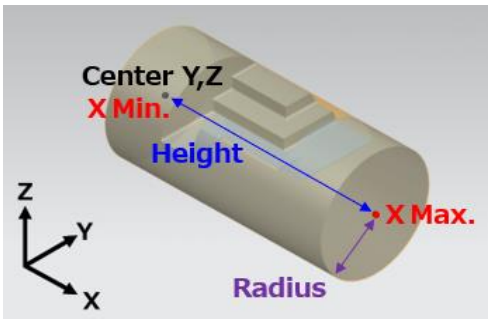
* The setting is the same as for previous parameters



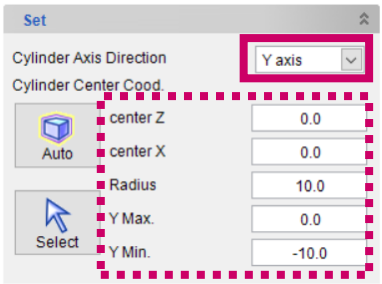


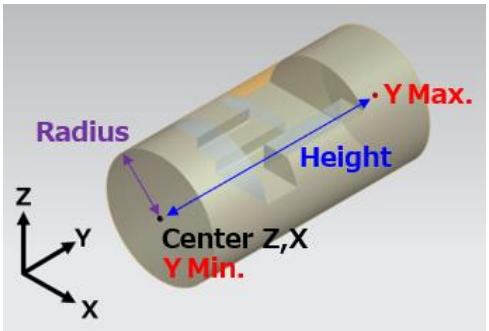
Cylinder axis direction: X axis





Cylinder axis direction: Y axis





3. Rearrangement of Hole Positions Sort Function

In FFCAM 2023, the [Rearrangement of Hole Positions] function was added.

In FFCAM 2024, a function to search for hole positions within a range has been added to the [Rearrangement of Hole Positions] function.

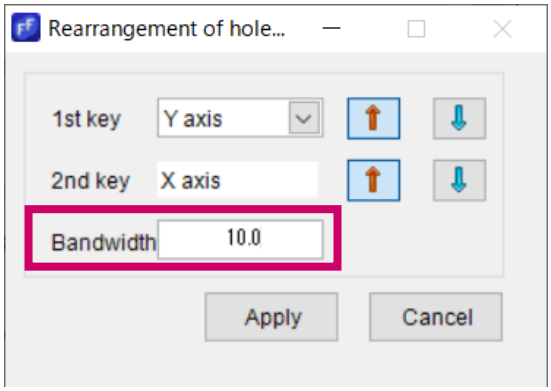
This search function allows holes to be included in the hole machining sequence, even if they are a slight distance from the specified straight line in the progress direction. This function can reduce the distance traveled by the tool when drilling is performed.

■ Setting Screen

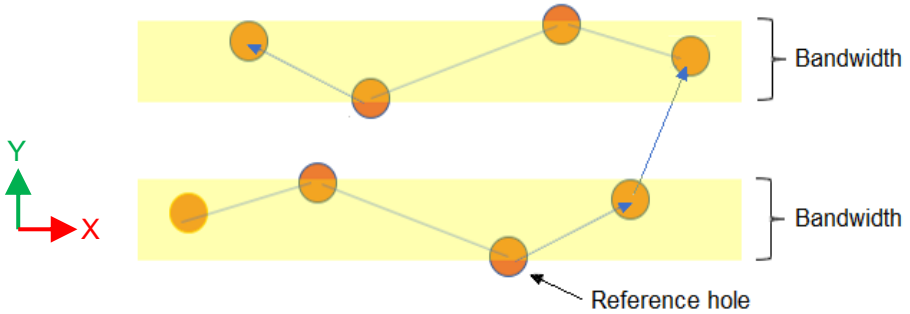
[Rearrangement of Hole Positions] screen

In FFCAM 2024, the [Bandwidth] parameter has been added.
The function searches for hole positions in the progress direction within the range specified in [Bandwidth] and includes them in the machining sequence.

Setting example)
1st key: Y axis ascending / 2nd key: X axis ascending

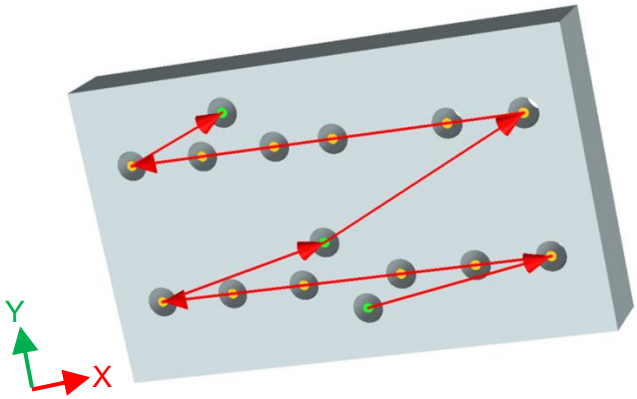


When [Bandwidth] is entered, the hole with the smallest Y coordinate value (center point) becomes the reference hole, and holes contained within the specified range (Y axis + direction) from the reference hole are registered to the machining sequence in order in the X axis direction.

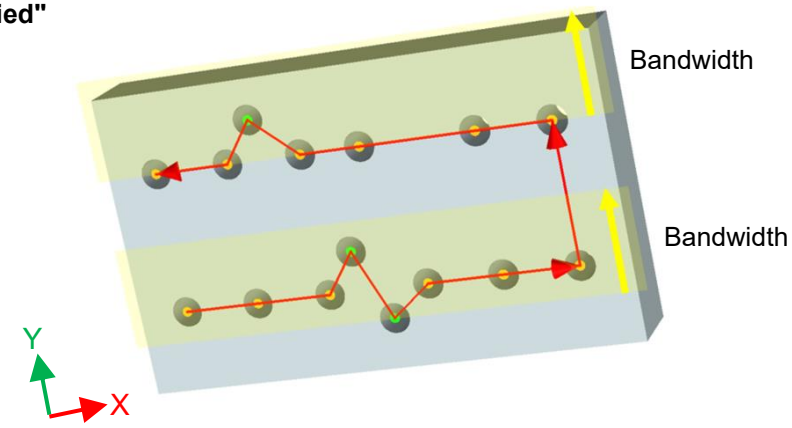


Machining order illustration)
1st key: Y axis ascending / 2nd key: X axis ascending

Bandwidth "0"



Bandwidth "specified"



When [Bandwidth] is set, the distance traveled by the tool during drilling can be reduced.

4. Addition of Comment Input Function to Machining Settings for Drilling

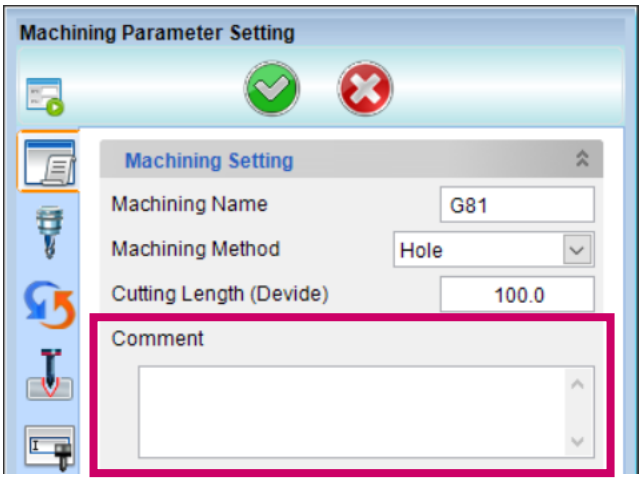
Comments can now be entered on the [Machining Settings] screen for drilling.

The entered comments can be viewed and edited in the process list [Machining Data Selection].

■ Setting Screen

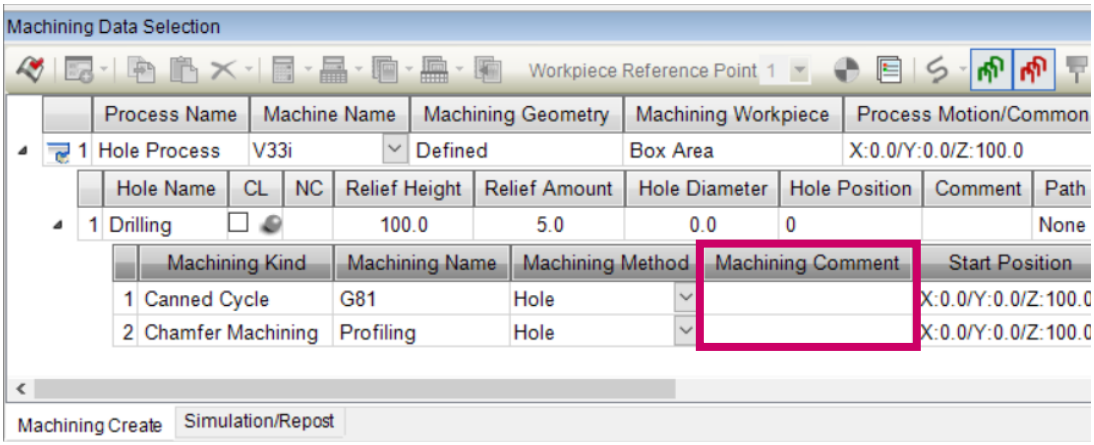
[Machining Settings] screen (Drilling)

Comments can be entered in the [Comment] field.



[Machining Data Selection] screen

Comments can be viewed and edited in the [Machining Comment] field of the process list.



5. Addition of Check Geometry and Offset Plane Settings to the Automatic Element Selection Function

In FFCAM 2023, a function was added to automatically select a model for some settings and to automatically select setting elements when a machining data template is copied from [User Database] by dragging and dropping it to the [Machining Data Selection] list.

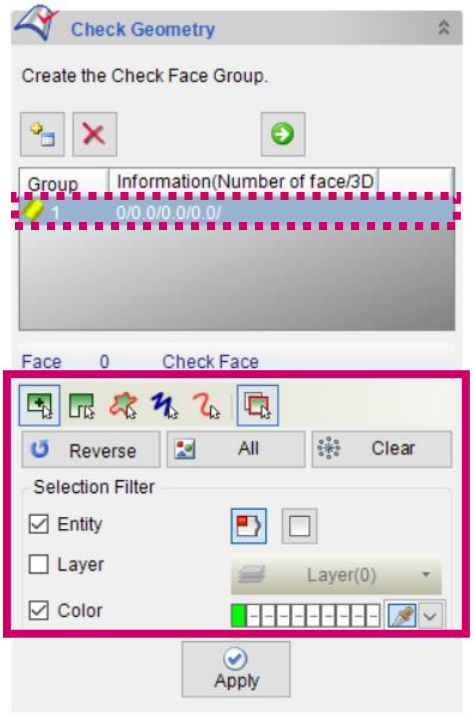
In FFCAM 2024, the *Automatically Select Elements* function has also been applied to the [Check Geometry] and [Offset Plane] settings. When copying machining data templates, elements for [Check Geometry] and [Offset Plane] can be selected and acquired automatically.

This function is applicable to all machining data for which [Check Geometry] and [Offset Plane] can be set.

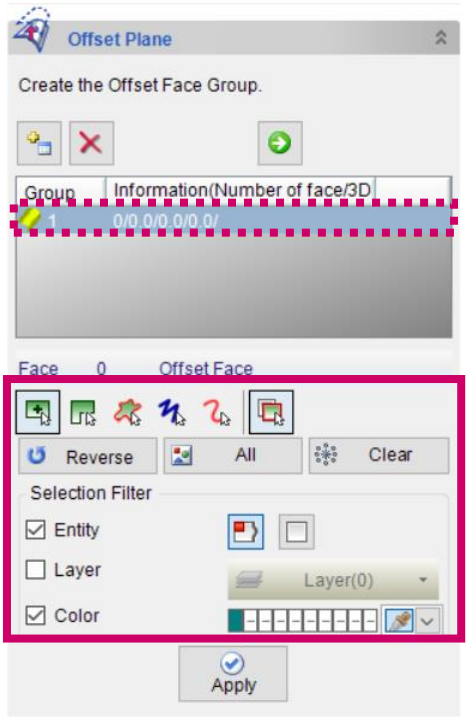
■ Setting Screen

When the machining data that was registered in [User Database] is copied to the [Machining Data Selection] list, the selection and acquisition of [Check Geometry] and [Offset Plane] that were set in the template are automatically performed. The element acquisition is done based on the filter information for element selection in the template.

[Check Geometry] setting screen



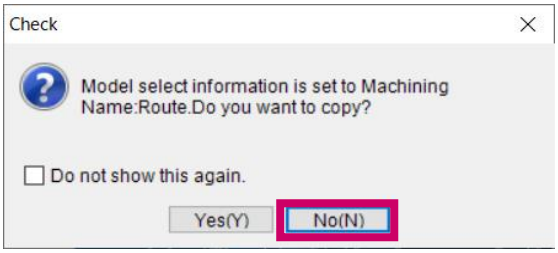
[Offset Plane] setting screen



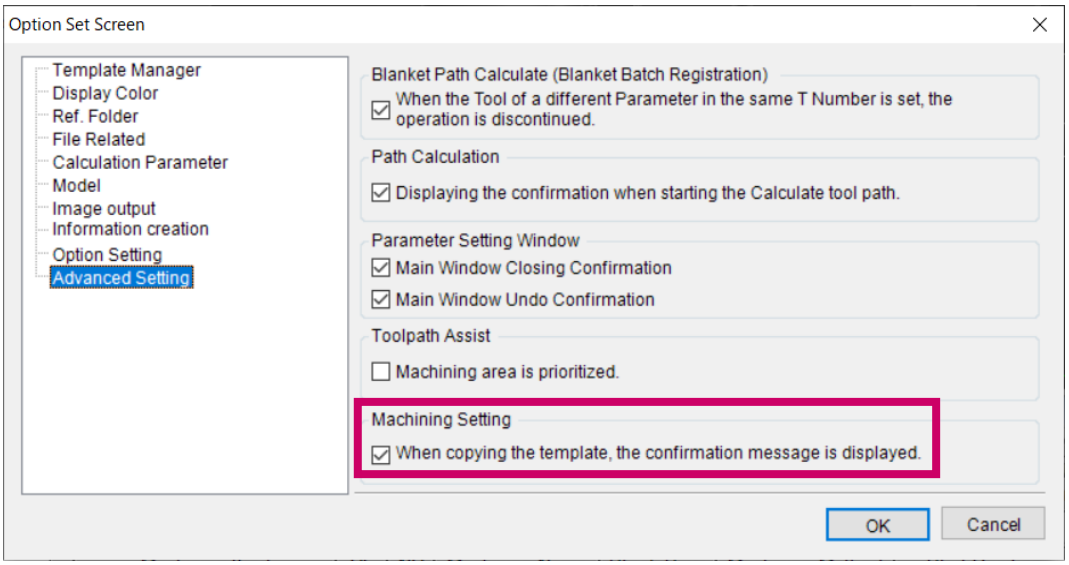
Surfaces are automatically registered based on the filter information for element selection.

■ Supplementary Note

- When a template is copied, a confirmation screen is displayed asking whether or not to auto-select the model and element. Select [No] if you do not want the model and element to be automatically selected.



The confirmation screen can be set to be displayed or not in the [Option Set Screen].



- Models that are not displayed on the screen (layer hidden or blank) are also automatically selected.
- If a model does not have any parts that meet the selection conditions, the model and element are not automatically selected.

■ Note

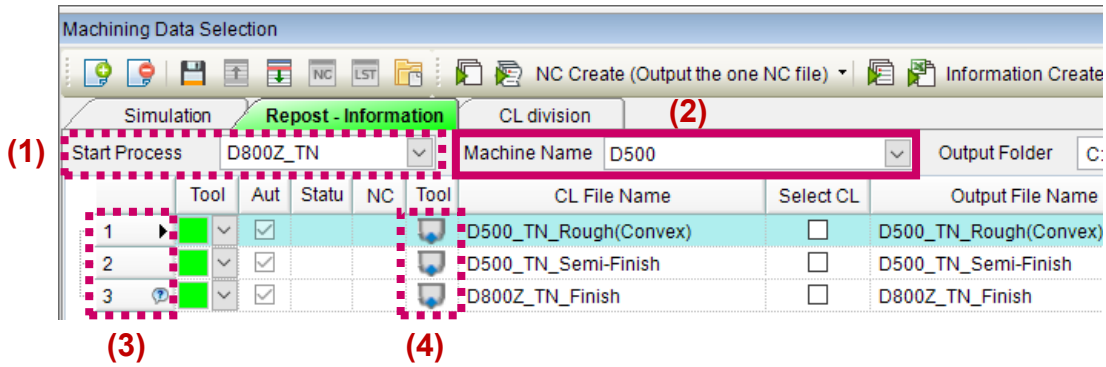
- If the first and second lines of the list contain the same filter information for element selection, the results of element acquisition for those first and second lines may be the same. (Duplicate)

6. Enhancement to Enable Changing of Machine Name in the [Repost - Information] Screen

For reposting with a previous FFCAM, in order to change the machine name and create NC data or output machining information, you had to return to machining data creation and execute the path calculation again.
In FFCAM 2024, machine names can now be changed in the [Repost - Information] screen, and through the repost process, NC data can be created or machining information can be output with the changed names. Unlike previous FFCAM, there is no need to return to machining data creation.

■ Setting Screen

[Repost - Information] screen



(1) Start Process

This parameter is the same as before.
The process name set in the [Machining Create] screen is displayed.
Previously, reposting (NC data creation and machining information output) was performed by the machine set for the process specified here.

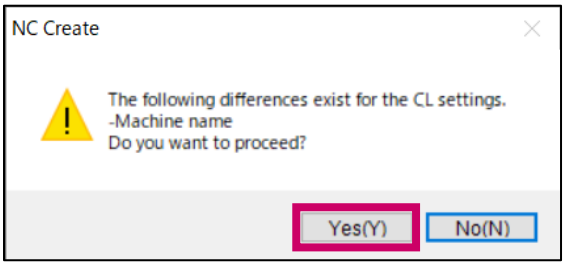
(2) Machine Name

This is a new parameter provided in FFCAM 2024.
Any machine other than the machine set for the [Start Process] process can be specified.
In FFCAM 2024, reposting (NC data creation and machining information output) is performed by the machine specified here.

(3) CL List Number Field

If the machine specified in [Machine Name] is different from the machine that was set when each CL data was created, a notification mark appears to the right of the CL List Number.

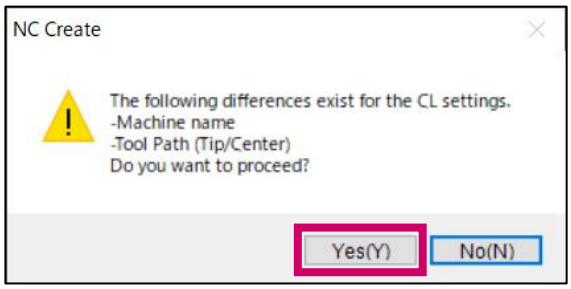
When you select CL data with the notification mark and execute the reposting process, a warning message is displayed.
When you select [Yes (Y)], reposting is executed, and NC data creation (machining information output) is performed.



(4) Tool Path (Tip/Center)

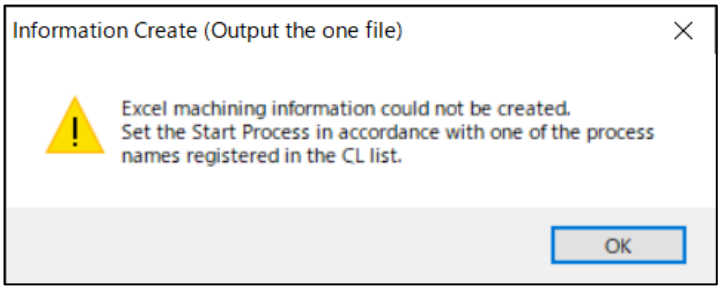
If the type of Tool Path (tip or center) that was set when each CL data was created is different, a warning message appears when the reposting process is executed.
When you select [Yes (Y)], reposting is executed, and NC data creation (machining information output) is performed.

* If both the machine type shown in (3) above and the tool path type shown in (4) above are different, a warning message appears when the reposting process is executed.
When you select [Yes (Y)], reposting is executed, and NC data creation (machining information output) is performed.



■ Notes

- If you change the machine name and perform reposting, be sure to run the simulation and check the results.
- Excel machining information can be output if at least one of the process names in the CL list matches the process names specified in the start process.
  However, if none of the CL list matches, an error message appears, and the Excel machining information is not output.
  * In this case, Excel machining information is not output, but the existing machining information is output.



7. Addition of Function to Output a Check Message for Parameter Macros

In FFCAM 2023, the "CUSTOMMSG" function that outputs a check message was added to the parameter macro functions.
This function can be assigned to a single parameter to easily set the check message output.

In FFCAM 2024, a function dedicated to check messages, which can be set even more flexibly, has been added.
With this dedicated function, check messages can be output even when multiple checks are assigned to a parameter or when a check that is not tied to a specific parameter is desired, so it is suitable for a wider range of applications.

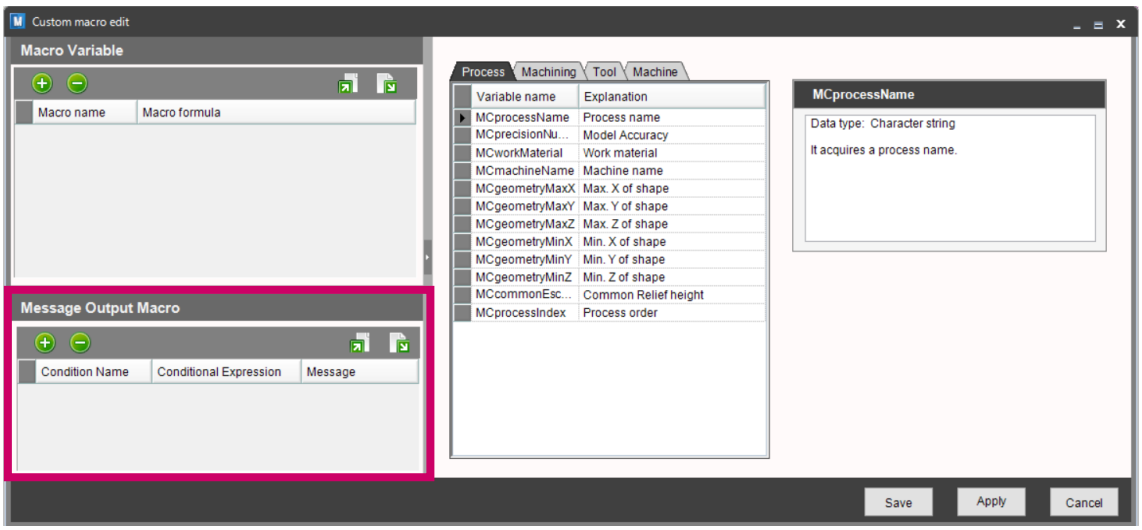
■ Setting Screen

[Custom Macro Edit] screen

The [Message Output Macro] item has been added.

This item allows you to set conditional expressions for check message output without having to assign them to specific parameters.

When a conditional expression is executed during path calculation and the result is "False", the message that you set is output. * If the result of the calculation is "False", the calculation is canceled.



Message Output Macro

Add condition icon

Adds a line for entering a conditional expression to the list.
Clicking this icon adds one line to the last line.

Delete condition icon

Deletes the selected conditional expression line.

CSV import icon

Imports the conditional expressions described in a CSV file into the list.
The contents of the list before importing are deleted and overwritten with the imported contents.

Export to CSV icon

Writes the listed conditional expressions to a CSV file.

Condition Name

Enter the name of the conditional expression.
The entered text is displayed in the "Item" field when a message is output by the message output function.

Conditional Expression

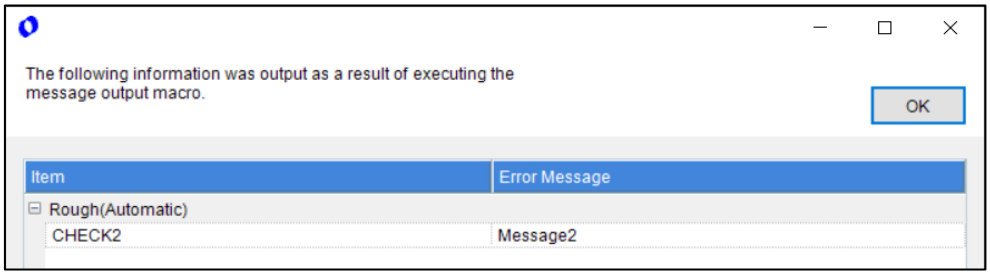
Enter a conditional expression to output a message.
A message is output when the result of the entered conditional expression is "False".
Parameter variables, macro variables, operators, and functions can be used in conditional expressions.
* Conditional expressions should be entered so that the calculation result is the True value.

Message

Enter a message to be displayed when the determined result of the conditional expression is "False" during path calculation.

■ Example of Message Screen Output

If the conditional expression set in [Message Output Macro] is "False" during path calculation, a message screen similar to the following is displayed, and the calculation is canceled.



For more information on how to set a message output macro, see "Machining Parameter Macro Manual" viewed from "Machining Parameter Macros" in FFCAM Help.

8. Addition of Parameter Macro Functions to Drilling

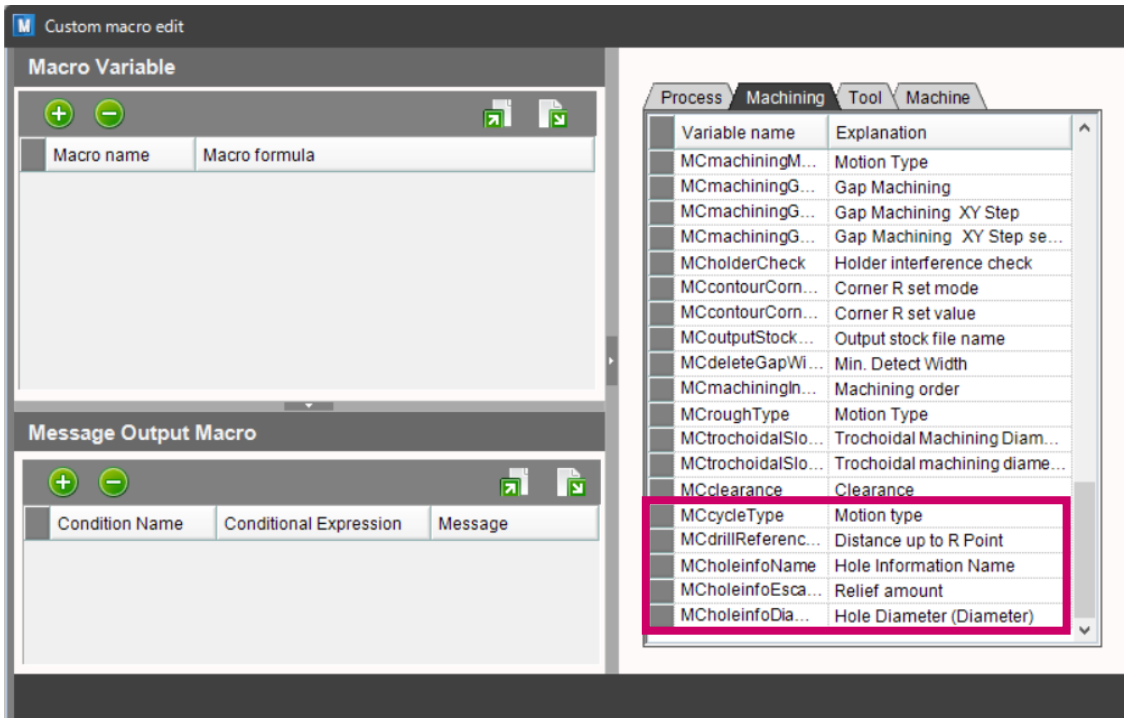
Parameter macros can now be used for drilling.

Macro variables related to drilling can be defined in the [Custom Macro Edit] screen.

Macros can also be set by opening the [Macro Edit] screen from the setting screens of [Hole Process], [Hole Information], and [Drilling].

■ Setting Screen (Custom Macro Edit)

Define macro variables related to drilling in the [Custom Macro Edit] screen.



In addition to the previous macro variables (some of which can also be used for drilling), the following macro variables dedicated to drilling have also been added.

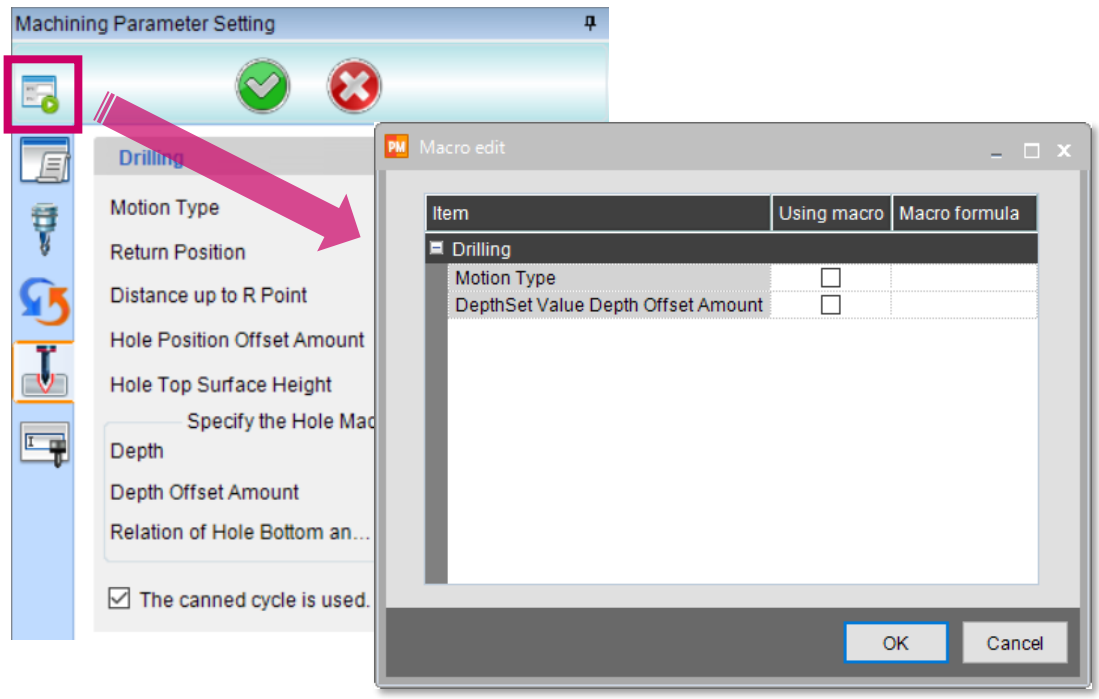
Machining		
Variable name	Parameter name	Explanation
MCcycleType	Motion Type	Acquires the motion type of drilling.
MCdrillReferenceQuantity	Distance up to R Point	Acquires the distance up to the R point.
MCholeinfoName	Hole Information Name	Acquires the hole information name.
MCholeinfoEscapeValue	Relief Value	Acquires the relief value in the hole information.
MCholeinfoDiameter	Hole Diameter (Diameter)	Acquires the hole diameter at the hole location.

Tool		
Variable name	Parameter name	Explanation
MCcutterChamferLength	Bevel Length	Acquires the bevel length of the "reamer" or "tap" cutter.
MCcutterPitch	Pitch	Acquires the pitch of the "tap" cutter.

■ Setting Screen (Macro Edit)

An icon to open the [Edit Macro] screen has been placed on the [Hole Process], [Hole Information], and [Drilling] setting screens.

Example) [Drilling] setting screen → [Macro Edit] screen
Open the [Macro Edit] screen to set macros for several parameters.



See the tables below for lists of parameters that correspond to the macro settings in the [Hole Process], [Hole Information], and [Drilling] settings.

Hole Process setting		
Setting tab	Target item	Parameter name
Process Setting	Process Setting	Process Name

Hole Information setting		
Setting tab	Target item	Parameter name
Hole Information	Hole Information	Relief Height
		Relief Value

Drilling settings		
Setting tab	Target item	Parameter name
Machining Setting	Machining Setting	Machining Name
		Comment
Tool Setting	Cutter	Diameter
	Holder	Holder clearance
		Shank clearance
Machining Condition Setting	Feed Rate and Spindle Speed	Spindle Speed
		Feed Rate
	Other Machining Condition	at XY Infeed
		at XY Escape
		at Z Infeed
		at Escape Motion
Machining Parameter Setting	Drilling	Motion Type
		Depth Offset Amount/ Depth Set Value
	Tapping	Motion Type
		Depth Offset Amount/ Depth Set Value
	Spot face machining	Motion Type
		Depth Offset Amount/ Depth Set Value
		Specify Method of Spot Face Range Set Value/ Depth Offset Amount
		Prepared Hole Diameter
		Hole Diameter
		XY drive-in cut amount
		Z drive-in cut amount
	Chamfer (Profile)	Motion Type
		Hole Diameter
		Chamfer Size
	Helical Machining	Depth Offset Amount/ Depth Set Value
		Hole Diameter
		Z drive-in cut amount

9. Addition of Function to Acquire Pre-machining Information to Parameter Macro Functions

The function "PREM" to acquire specified pre-machining information has been added to parameter macro functions.

This function can automatically acquire the values of specified parameters from specified pre-machining.

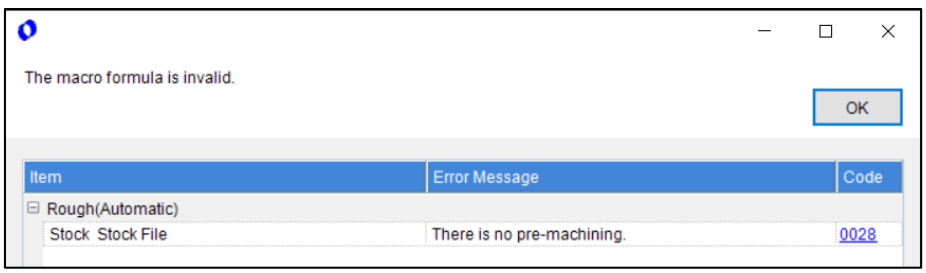
■ Specifications of Function

Function name	PREM
How to use	PREM(MC variable name,integer value)
First argument	Specify MC variable (CAM parameter variable). This cannot be specified in a macro expression.
Second argument	Specify the preceding machining target by the number prior, as an integer value of 1 or more. This can be specified in a macro expression.
Return value	The function acquires the parameter value of the first argument of PREM for machining that precedes the specified machining by the numerical value of the second argument.
Restrictions	・ This function is applied during path calculation. ・ Pre-process or pre-hole information cannot be acquired.
Examples	1) PREM(MCoutputStockName,3) Acquires the stock file name of the machining three executions prior. 2) PREM(MCoutputStockName,Uservar) Acquires the stock file name of previous machining by the integer value obtained by the "UserVar" macro variable.

In the following cases, an error occurs, and an error message is displayed.

- If only one argument is specified or three or more arguments are specified
- If no arguments are specified
- If a variable other than an MC variable is specified as the first argument
- If a value other than an integer is specified as the second argument
- If the specified pre-machining does not exist

Example of error screen)



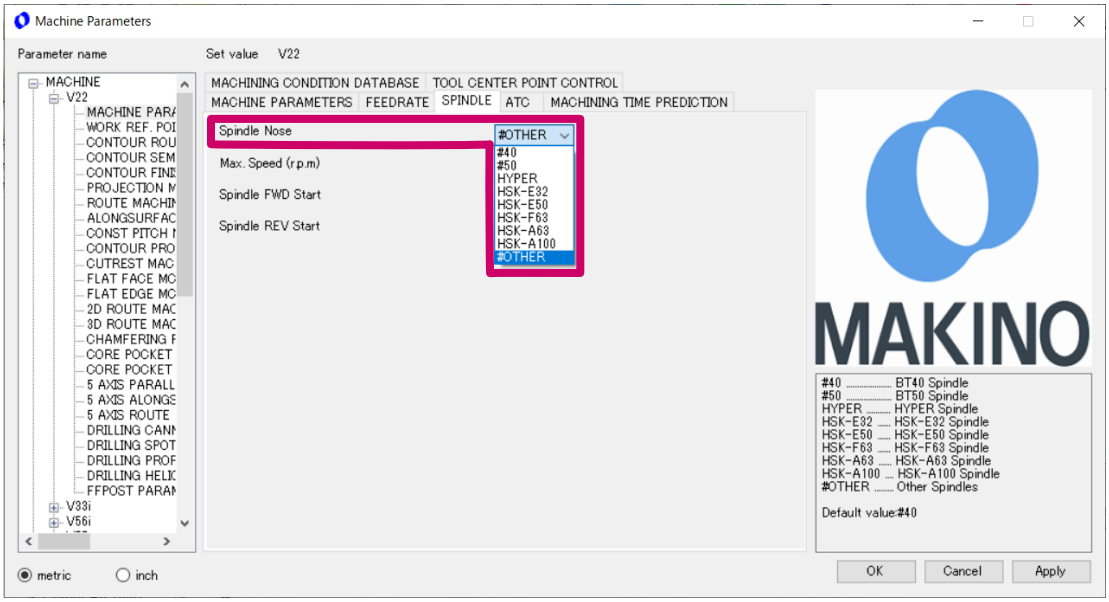
10. Updates to Spindle Nose

Items have been added and renamed for [Spindle Nose] in the machine parameters and tool database.

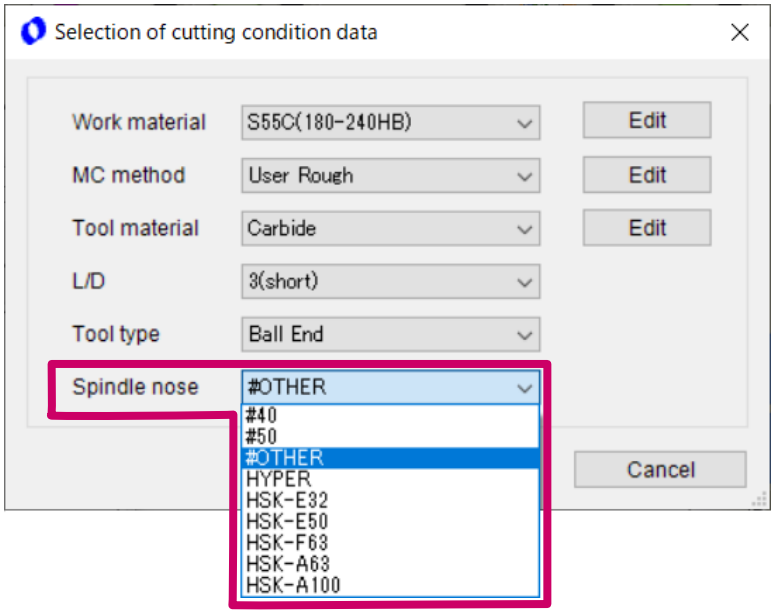
■ Setting Screen

The selections in the [Spindle Nose] list have been updated.

Machine Parameters



Tool database / [Selection of machining condition data] screen



■ Changes to the List

See the tables below for the changes from previous FFCAM.

Previous FFCAM versions		FFCAM 2024 and later
#40		#40
#50		#50
HYPER		HYPER
HSKE32		HSK-E32
HSKE50		HSK-E50
HSKF63		HSK-F63
- (* Newly added from FFCAM 2024)		HSK-A63
- (* Newly added from FFCAM 2024)		HSK-A100
#CM		#OTHER

11. Addition of Function to Delete Redundant Data when Importing Models

A function that deletes redundant data when models are imported has been added.
Automatically deletes redundant data when executing [New Create] or [Model Import].

* Redundant data refers to elements (vertices, edges, faces) that are set more than necessary in the model data.

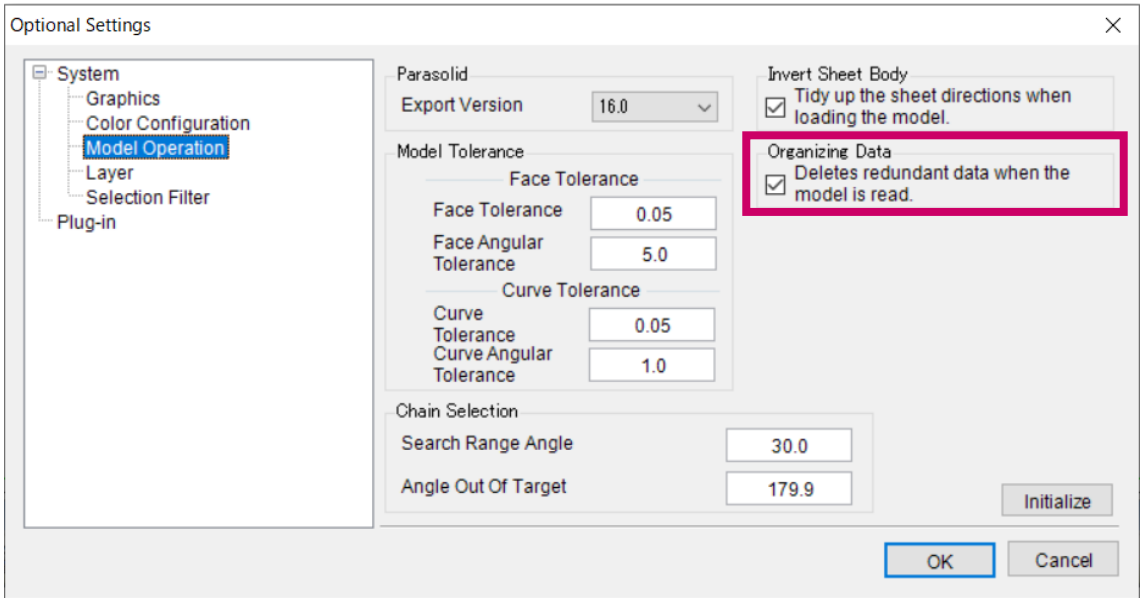
■ Setting Screen

[Optional Settings] / [Model Operation] screen

In the [Model Operation] screen of [Optional Settings], set whether to delete redundant data when importing a model.

To delete redundant data when importing models, add a check mark to the [Deletes redundant data when the model is read.] check box.

* The box is not checked in the initial settings after installation.

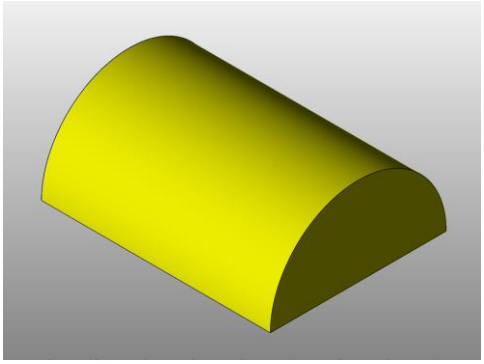


■ Example of Model Import

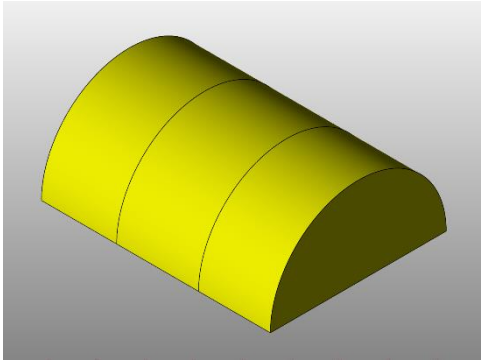
The following figures show the difference in model import status depending on the setting of the [Deletes redundant data when the model is read.] check box.

Example) If a model with the same face divided is imported

Check mark: Selected



Check mark: Not selected



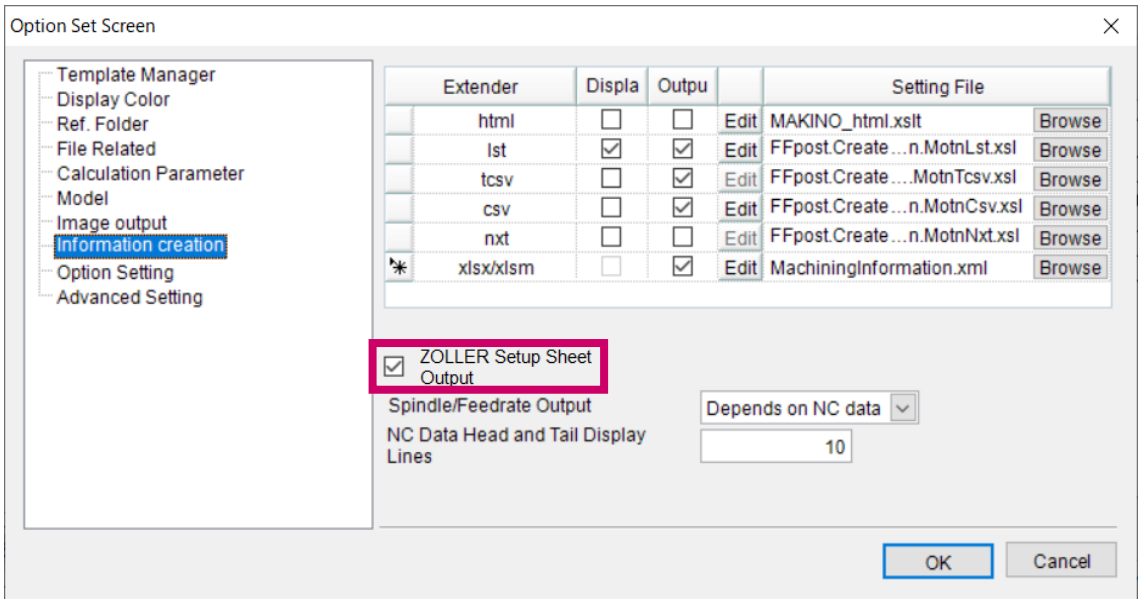
12. Addition of Function to Output Setup Sheets Compatible with ZOLLER Tool Measuring Machines

A function has been added to output setup sheet (.xml) files that can be read by a ZOLLER tool measuring machine when Repost is executed.
The ZOLLER setup sheet is output when [Information Create (Output the one file)] is executed on the [Repost - Information] screen.

The ZOLLER setup sheet is a list of External Tool ID (Identification number) and T number output from FFCAM. This sheet can be imported and used by the ZOLLER tool measuring machine.
For more information on the ZOLLER setup sheet, see the FFCAM Help.

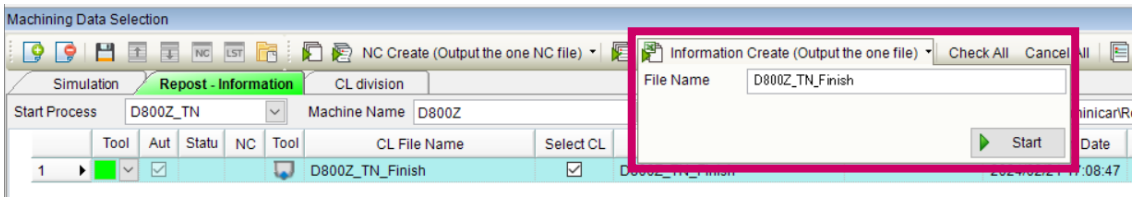
■ Setting Screen

[Option Set Screen] / [Information Creation] screen
The [ZOLLER Setup Sheet Output] check box has been placed on the [Information Creation] screen of [Option Set Screen].
When outputting the ZOLLER setup sheet, add a check mark to this check box.



[Repost - Information] screen

When you execute [Information Create (Output the one file)] on the [Repost - Information] screen, the ZOLLER setup sheet is output together with the other machining information.



Output destination for the ZOLLER setup sheet

The sheet is saved to the folder set in [Output Folder] of the [Repost - Information] screen. The file name is "Output File Name_SetupSheet.xml".

13. Improvement of Tool Database Maintenance to Allow Modification of Registered Holders

In previous FFCAM, holders registered by [Tool Edit] of Tool Database Maintenance could not be changed.

In FFCAM 2024, a function to delete holders registered by [Tool Edit] has been added, and the holders of registered tools can now be changed.

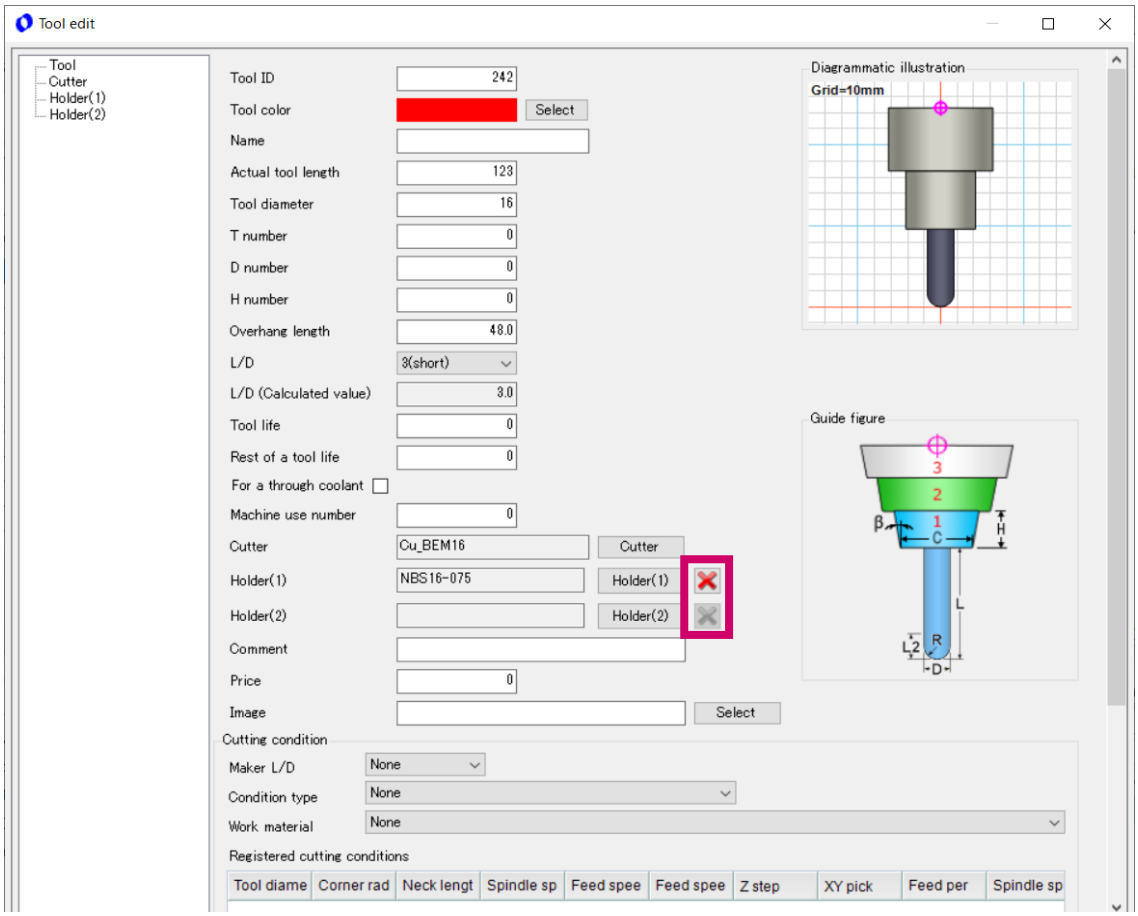
■ Setting Screen

Tool Database Maintenance / [Tool Edit] screen

A [Delete] button has been added to the [Holder (1)] and [Holder (2)] items.

Clicking the [Delete] button deletes the registered holder.

After deletion, the holder can be re-registered.



14.Improvement of [Machine Simulator] Path Display of Canned Cycles

In previous Machine Simulator (option), the machining operations of canned cycle paths were not displayed.

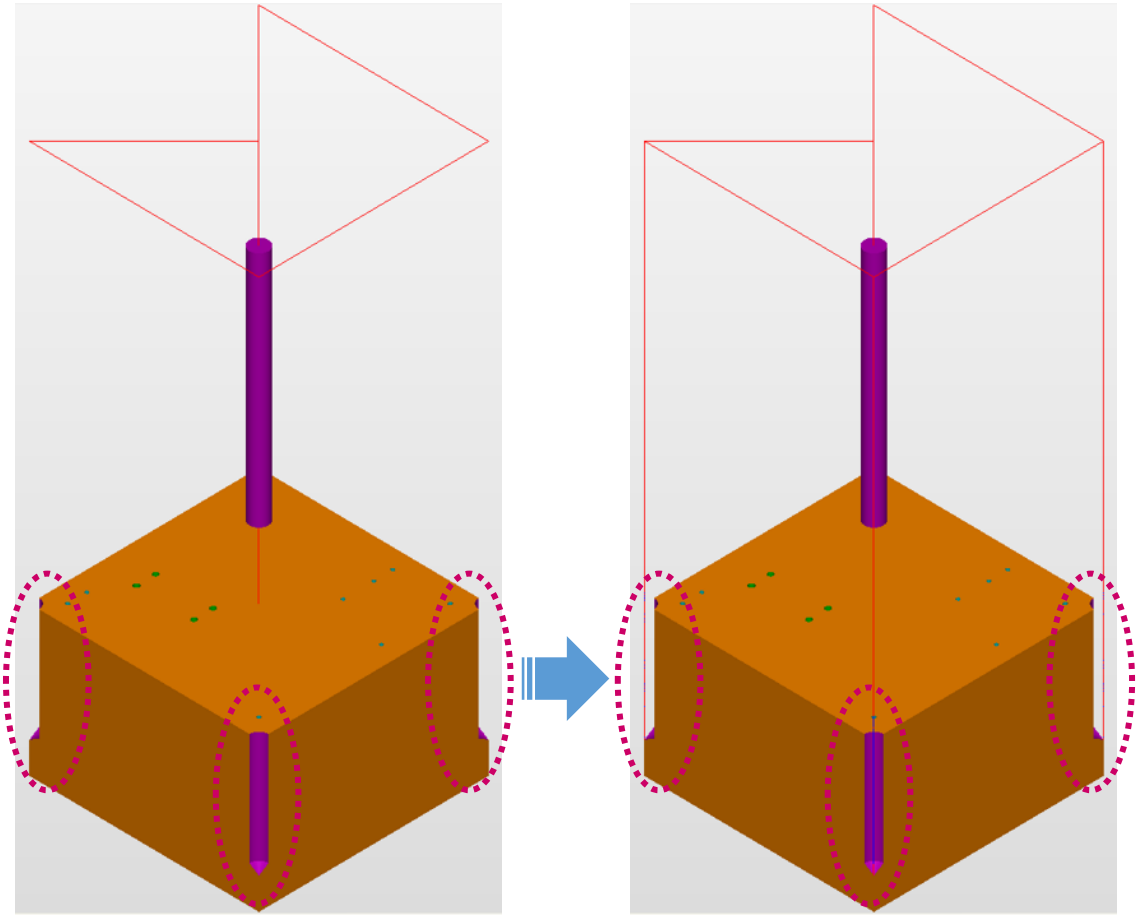
In Machine Simulator V.2.4, the machining operations of canned cycle paths are now displayed.

The path display of Z-feed machining, etc., makes it easier to check the motions.

■ Example of a Simulation Screen Display

Previous Machine Simulator
For canned cycles, only the approach is shown

Machine Simulator V.2.4
For canned cycles, the machining is also shown



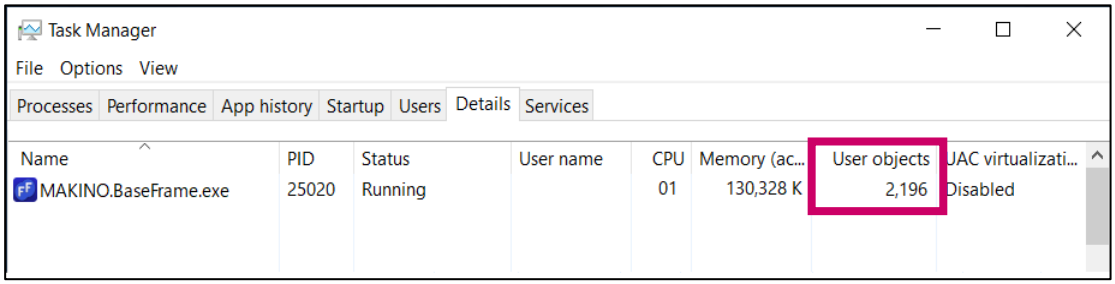
15. Improvements to Stabilize the FFCAM System

In previous FFCAM, when working while a file was open for a long period of time or while multiple files were open, the increase in user objects (temporary files created and used by FFCAM during operation) could lead to unstable operation or abnormal termination.

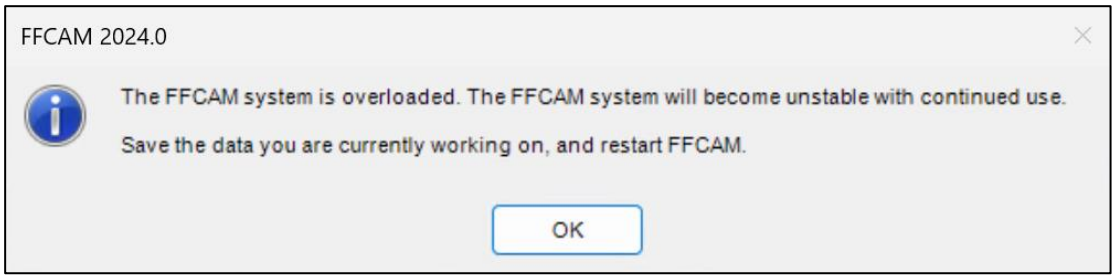
In FFCAM 2024, the program was revised to mitigate the rise in the number of user objects, and a function that monitors the number of user objects was also added, to improve system stability.

■ How to Check the Number of User Objects

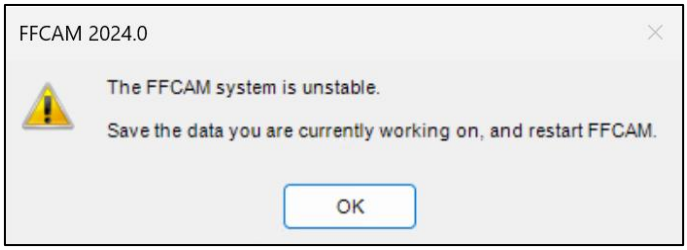
The number of user objects can be checked in "Details" of "Task Manager" on Windows.



When the number of user objects reaches 8000, the following message appears, prompting the user to restart.



When the number of user objects reaches 9000, the following message appears, prompting the user to restart.



16. Supported Parasolid Versions

The following Parasolid versions are now supported in FFCAM 2024.

- 35.0
- 35.1