FCA250N2A-2506EN

FFCAM 2025

Description of New Functions

MAKINO

Promise of Performance

Preface

This manual describes the functions added to MAKINO FFCAM 2025 and how to use them.

Created on

June 2025

List of Added Functions in FFCAM 2025

1. Installed Assist Function That Can Check for Tool Collision before Path Calculation

FFCAM 2025 has been equipped with the [Collision Check Assist] function, which can check and register tool settings to prevent interference before the path calculation.
By comparing the tool to be set with "Interference area" before displaying the results, this function allows you to confirm in advance whether the set tool will interfere with the machining geometry during machining.
By using this function to set a tool that will not interfere, you can eliminate the need to rework the settings through simulations after path calculation.

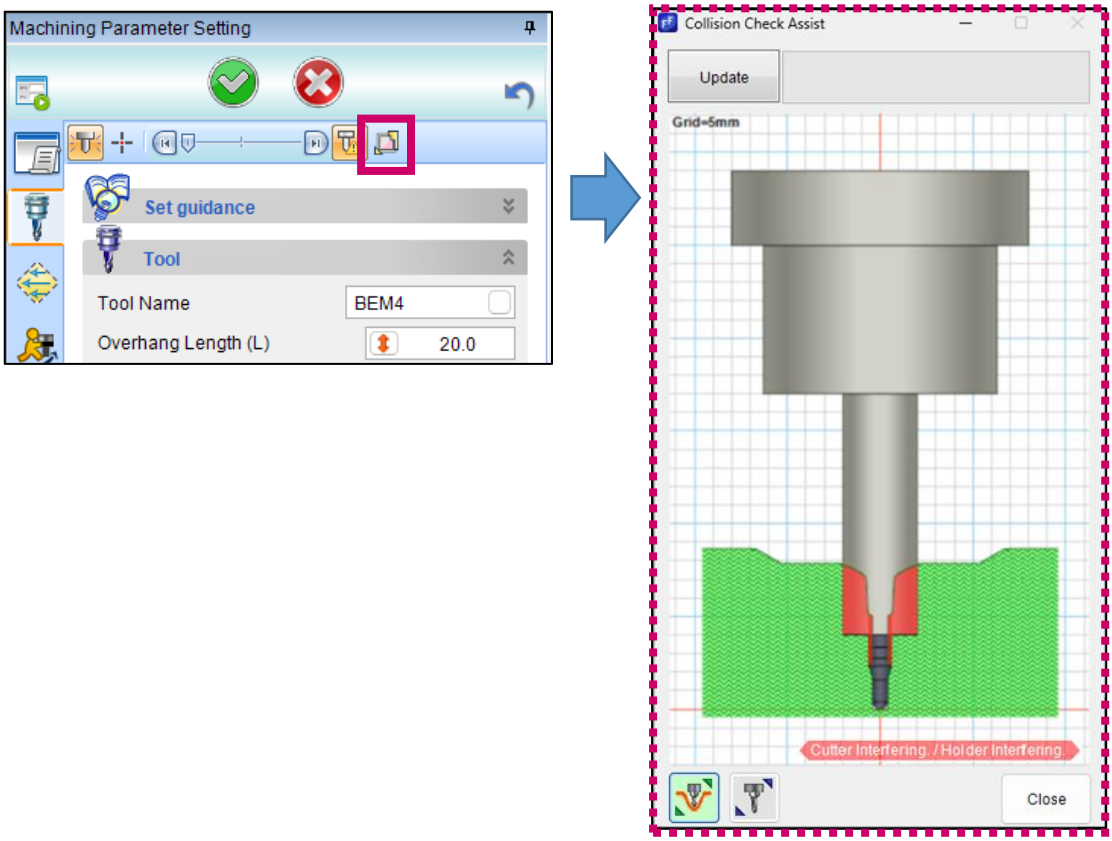
■ Setting Screen

From the mini toolbar in the upper part of the [Machining Parameter Setting]/[Tool Setting] screen, click the [Collision Check Assist] button to display the [Collision Check Assist] screen.

On the [Collision Check Assist] screen, you can visually check whether the tool to be set will interfere with the workpiece. On this screen, you can set and register tools while adjusting items such as the overhang length.

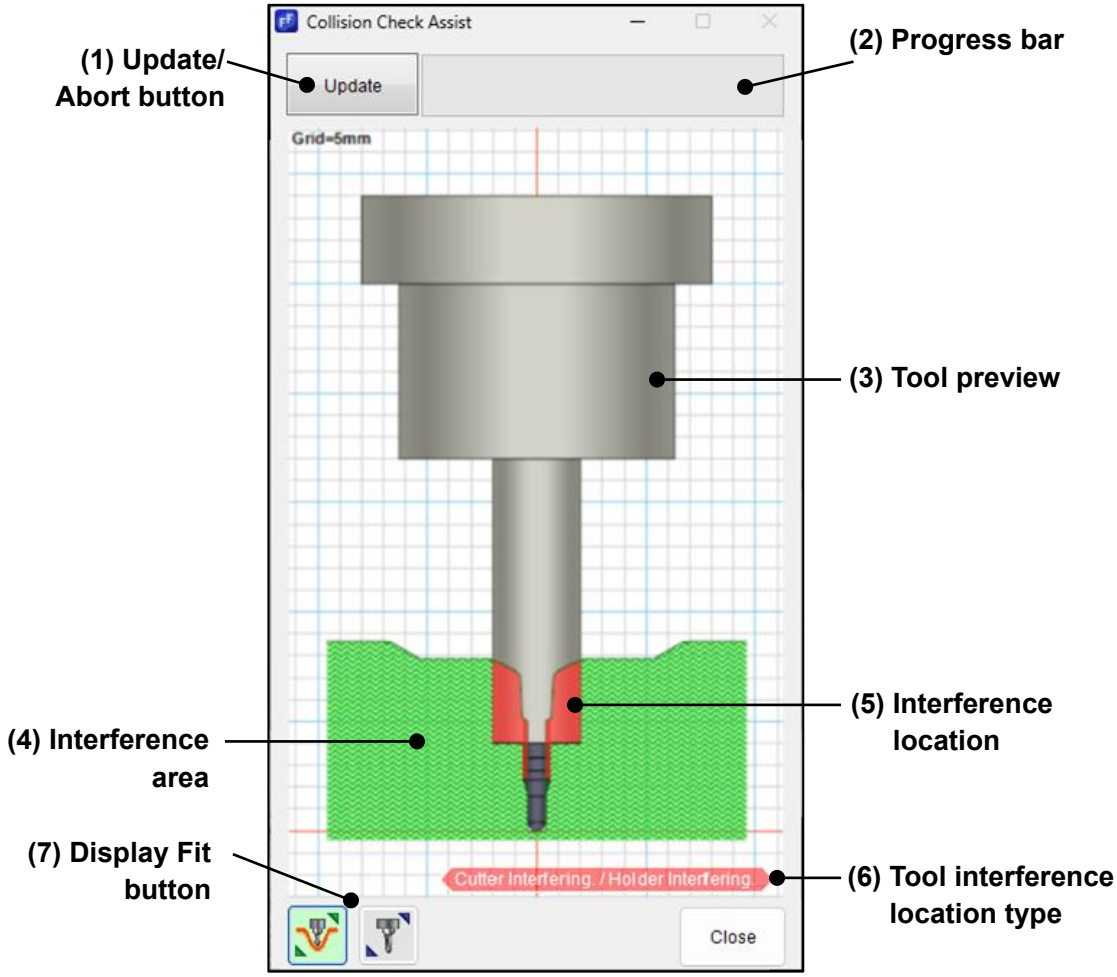
[Machining Parameter Setting]/[Tool Setting] screen

When you click the [Collision Check Assist] button, the [Collision Check Assist] screen is displayed, and calculations to find the interference area begin.
During calculations, the progress bar at the top of the screen is active. When calculations are complete, the [Collision Check Assist] screen becomes operational.



[Collision Check Assist] screen

The interference area (displayed in green) and the tool that is currently defined for the machining are displayed.
The interference area shows the maximum tool outline where no interference occurs. By setting tool dimensions that do not come into contact with the interference area, you can set a tool that will not interfere during machining.



(1) Update/Abort button

The displayed contents are updated.
If parameters that affect the interference area are changed while the [Collision Check Assist] screen is displayed, the [Update] button is highlighted in orange.

To update the displayed contents, click the [Update] button.

Also, while the displayed contents are being updated, the button switches from [Update] to [Abort].

To interrupt the update of the displayed contents, click the [Abort] button.

(2) Progress bar

The update progress of the displayed screen is displayed.

(3) Tool preview

Based on the value entered into the [Tool Setting] parameter, a preview of the tool shape is displayed.

(4) Interference area (green area)

The mapping of the interference area generated by FFCAM is displayed. The outline is generated from various information, such as the tool set in [Tool Setting], path assist information, the machining area set in [Machining Area], and the machining geometry registered to [Process Setting].
By editing the tool displayed in the preview to ensure that it does not come into contact with the interference area, you can prevent interference during machining in advance.
For example, when you want to change the overhang length, you can adjust the length while checking this area to set an appropriate length that avoids interference.

(5) Interference area (red area)

When the tool displayed in the preview is included in the interference area, that area will blink in red.
Change the tool settings to remove the red area.

(6) Tool interference area

If there is interference with the tool displayed in the preview, the name of the part of the tool that is interfering will be displayed.
When there is no interference, this label will not be displayed.

(7) Display fit button

Select whether the screen display will show the entire tool and interference area or zoom in on the tool.

Fit to tool and interference section

The view is adjusted to where the tool and interference area are entirely visible.

Fit to tool

The view is zoomed in on the tool.
Use this view when a wide interference area makes the tool appear small and checking the interference becomes difficult.

■ Machining that Can Use the Collision Check Assist Function

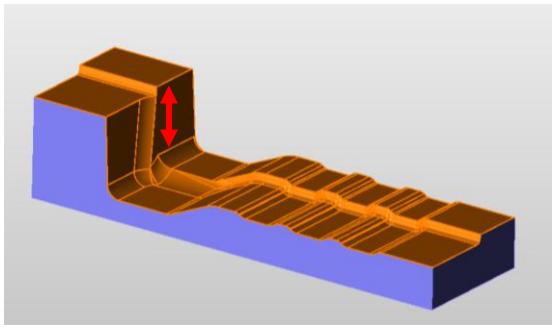
The Collision Check Assist function can be used with the following machining settings.

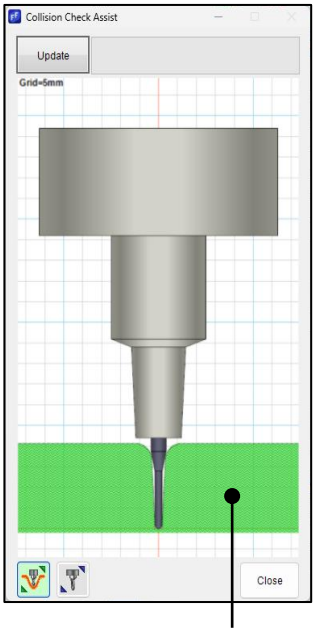
- Contour machining/Contour face cut machining
- Flat face machining
- Flat edge machining
- Route machining
- Projection machining
- Along-surface machining
- 3D equi-pitch closed machining
- 3D equi-pitch open machining
- Contour projection machining
- Corner R machining

■ Note

- In processes such as corner R machining or machining using stock, where tool paths are not created over the entire machining area, the overhang length required to avoid interference may be longer than what the interference area indicates.
  For example, in the machining shown in the image below, a wall where no tool path is created (indicated by the red arrow) is still included in the interference area. In this case, setting the overhang length based on the interference area may result in an overhang length that is longer than necessary.

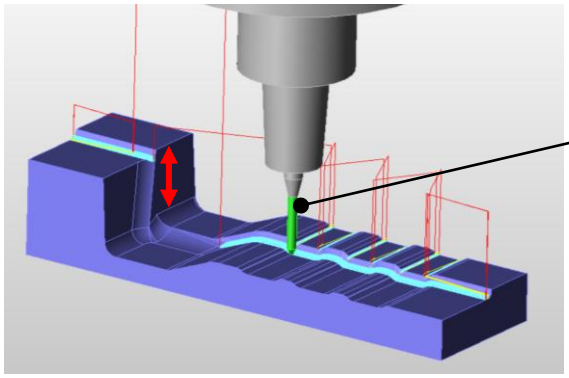
  For the machining area (orange section) of the image below, the interference area of the image to the right is displayed.





Interference area

Adjusting the overhang length based on the interference area results in an overhang length that is longer than necessary.



Longer overhang length

- Uncut areas from previous machining are not considered when generating the interference area. Always run a simulation after path calculation to verify that the CL data has no interference.
- Even machining that supports the Collision Check Assist function cannot use the function in the following cases.
  - When the angle is other than 0 for the Machining Direction and Reference Point function
  - When 5-axis simultaneous movement is enabled

2. Installed the Merge Tools Function to Integrate Tool Settings Across Machining Data

The [Merge Tools] function is installed on the FFCAM 2025.
The [Merge Tools] function allows you to manage the information of machining data for each tool setting in a list.

For example, this list makes it easier to check machining data that uses the same tool but has different overhang length settings. If multiple items of machining data have similar overhang length settings, they can be easily reset to have the same overhang length. By merging (integrating) the tool setting details between machining processes in this way, the number of tools to be prepared is reduced.

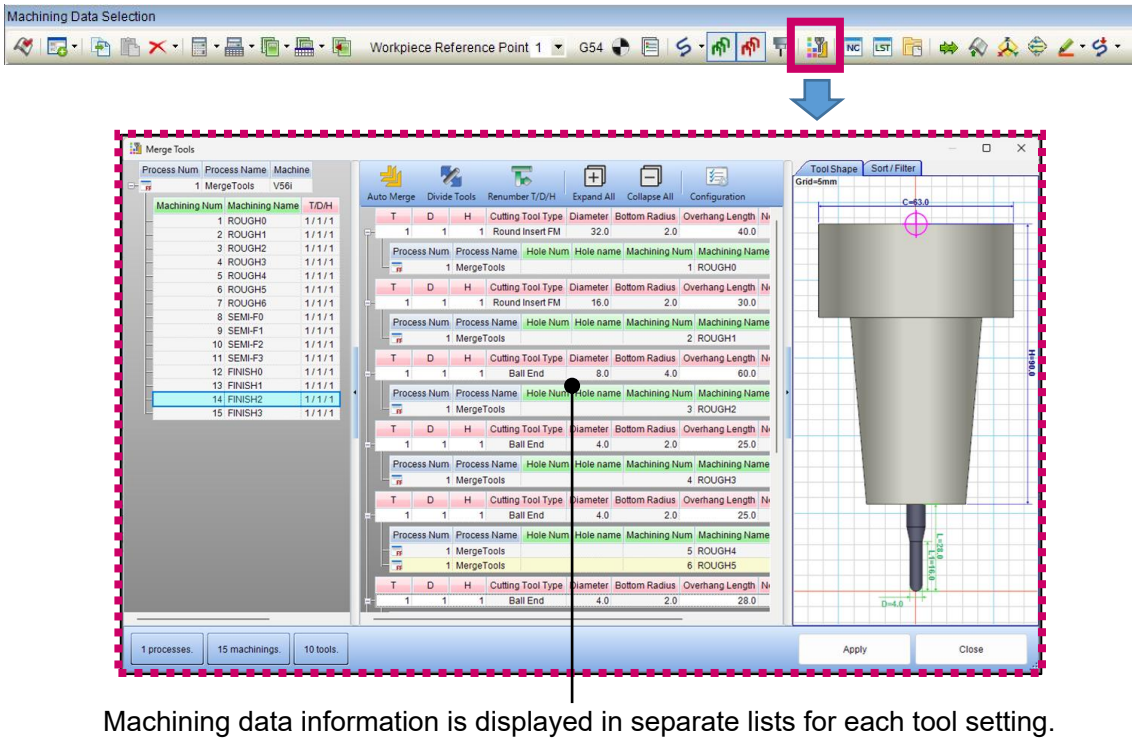
■ Setting Screen

From [Machining Data Selection] on the command bar, click the [Merge Tools] button to display the [Merge Tools] setting screen.

In the center of the [Merge Tools] setting screen, a tree is formed for each tool setting. Below it, the information of machining data with the same tool setting contents is displayed in a hanging list.

[Machining Data Selection]/Command bar

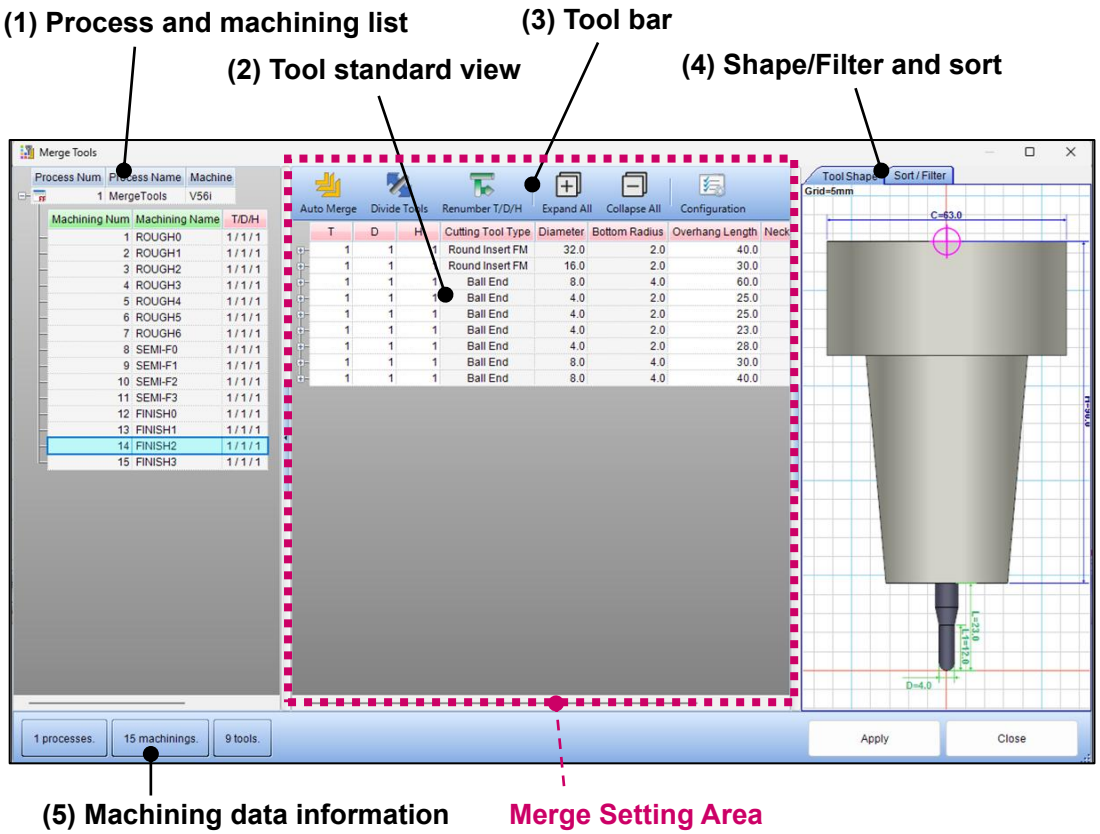
The [Merge Tools] setting screen is displayed when the [Merge Tools] button is clicked.



Machining data information is displayed in separate lists for each tool setting.

[Merge Tools] setting screen

Check the tool settings and overhang lengths and perform the merge operation using the "Merge Setting Area" at the center.



(1) Process and Machining List

Processing and machining information registered on the Machining Data Selection screen is displayed in a tree.

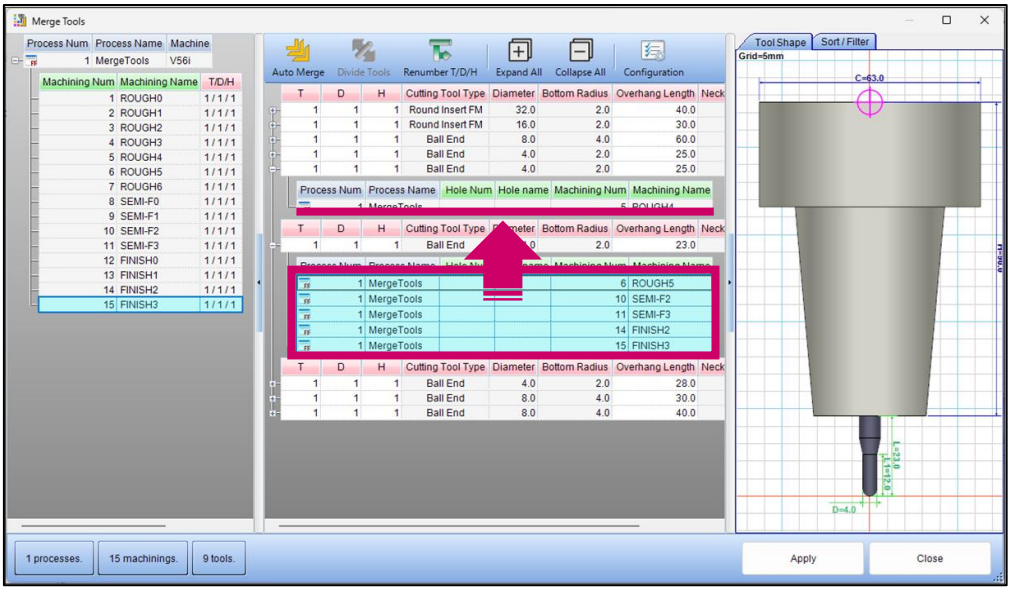
(2) Tool Standard View

In [Tool Standard View], machining data is listed in tree format as a hanging list below each tool setting.
You can change the tool settings of machining data by moving that data to the list of another tool setting.

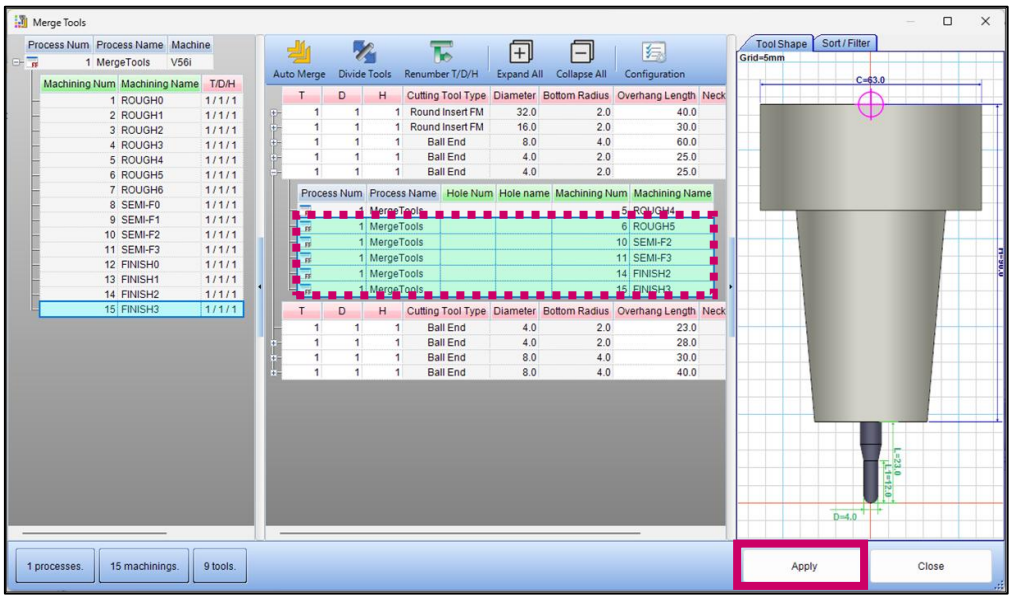
In [Tool Standard View], you can use the list to check all of the tools registered to the file being worked on. By using it in combination with the filter function, you can list only the tools with the same cutter and holder but different overhang lengths. Use this list to merge tools with similar overhang lengths and easily optimize the number of tools.

Merge operation example (Overhang length setting)

Drag and drop machining data with different overhang length settings into the list of a tool setting whose overhang length settings you want to match.



The machining data moves to the tool tree of the merge target.
By clicking the [Apply] button, the contents changed on the Merge Tools screen are reflected in the machining.



* When the path calculation for this machining data is already complete, its [CL] status in the [Machining Data Selection] list becomes yellow. Perform the calculation again.

(3) Tool Bar

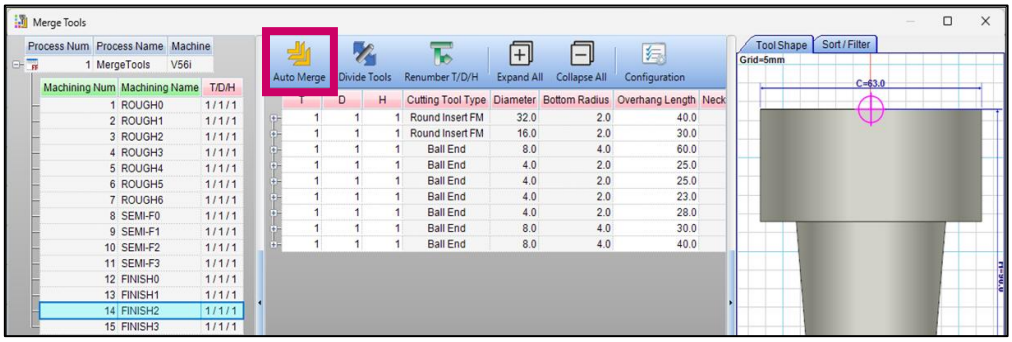
This is the menu for executing various functions in the "Merge Setting Area".

Auto Merge

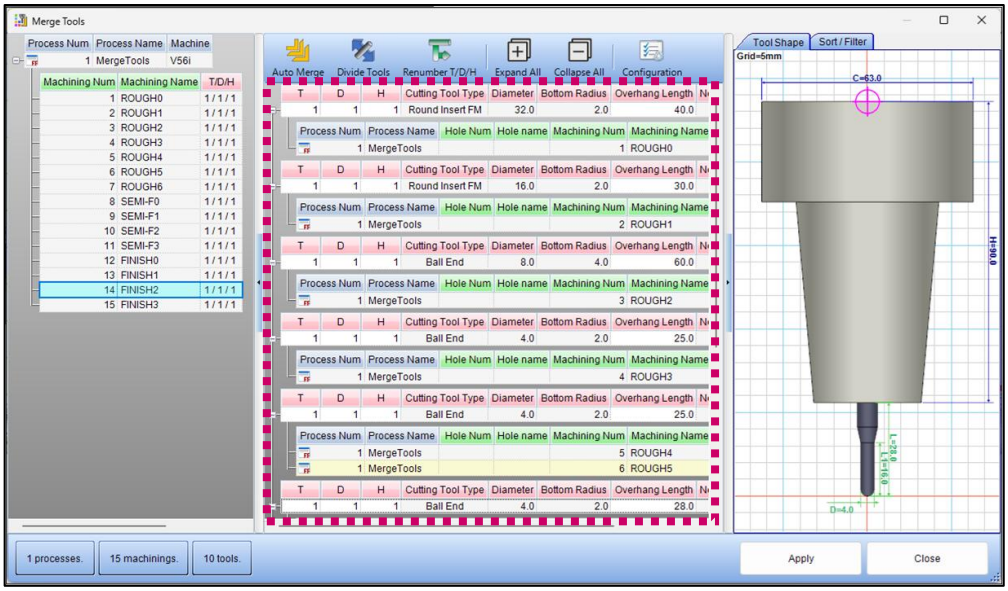
When multiple machining data has similar overhang length settings for the same tool setting, the FFCAM will automatically move machining data with similar overhang length settings between lists and merge them.
This function can automatically perform the merge operation described in the earlier section, "Merge operation example (Overhang length setting)", according to pre-determined rules.

Auto Merge operation example)

Click the [Auto Merge] button.



FFCAM automatically merges the overhang length settings of each machining data item.



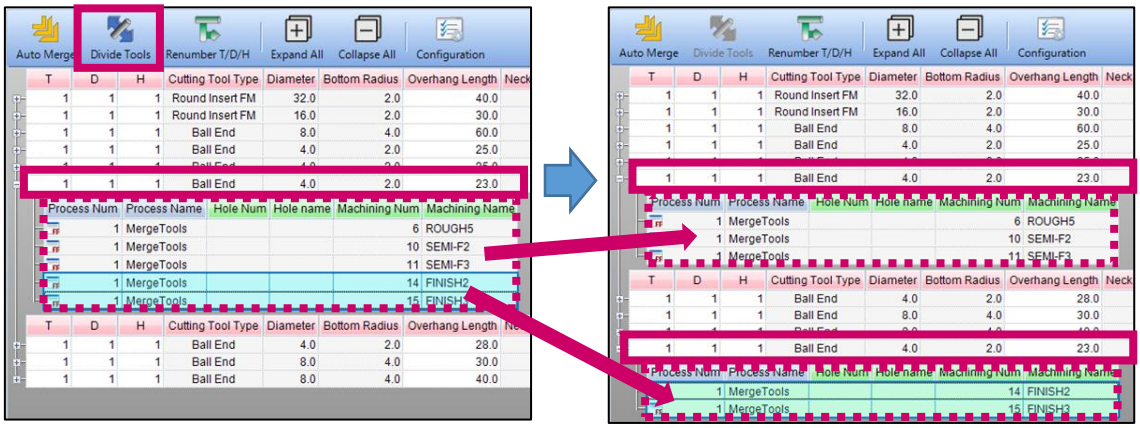
As described above, when merging the overhang length settings of machining data with the same tool setting on the [Merge Tools] setting screen, you can either use the aforementioned manual method or the [Auto Merge] function.

* [Auto Merge] merges shorter tools into longer tools within the rules set by FFCAM, to the extent that it does not significantly harm machining efficiency.

Divide Tools

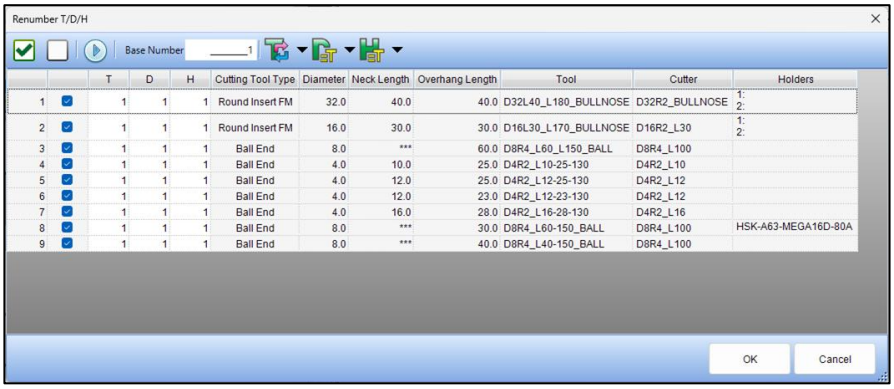
When multiple machining data items are in the same tool setting list, use this function to divide them into separate tool settings with the same tool setting values.
Perform tool division when operations require considerations such as tool life or switching to a new cutter due to a change in the machining process.

Operation example)
Select a machining operation, and click the [Divide Tools] icon to add new tools with the same settings.
The selected machining is assigned to the newly added tools.



Renumber T/D/H

The [Renumber T/D/H] screen is displayed.
In the [Renumber T/D/H] screen, you can renumber the T number, D number, and H number.



		T	D	H	Cutting Tool Type	Diameter	Neck Length	Overhang Length	Tool	Cutter	Holders
1	☑	1	1	1	Round Insert FM	32.0	40.0	40.0	D32L40_L180_BULLNOSE	D32R2_BULLNOSE	1: 2:
2	☑	1	1	1	Round Insert FM	16.0	30.0	30.0	D16L30_L170_BULLNOSE	D16R2_L30	1: 2:
3	☑	1	1	1	Ball End	8.0	***	60.0	D8R4_L60_L150_BALL	D8R4_L100	
4	☑	1	1	1	Ball End	4.0	10.0	25.0	D4R2_L10-25-130	D4R2_L10	
5	☑	1	1	1	Ball End	4.0	12.0	25.0	D4R2_L12-25-130	D4R2_L12	
6	☑	1	1	1	Ball End	4.0	12.0	23.0	D4R2_L12-23-130	D4R2_L12	
7	☑	1	1	1	Ball End	4.0	16.0	28.0	D4R2_L16-28-130	D4R2_L16	
8	☑	1	1	1	Ball End	8.0	***	30.0	D8R4_L60-150_BALL	D8R4_L100	HSK-A63-MEGA16D-80A
9	☑	1	1	1	Ball End	8.0	***	40.0	D8R4_L40-150_BALL	D8R4_L100	

* For details on the [Renumber T/D/H] screen, see the Help window of FFCAM.

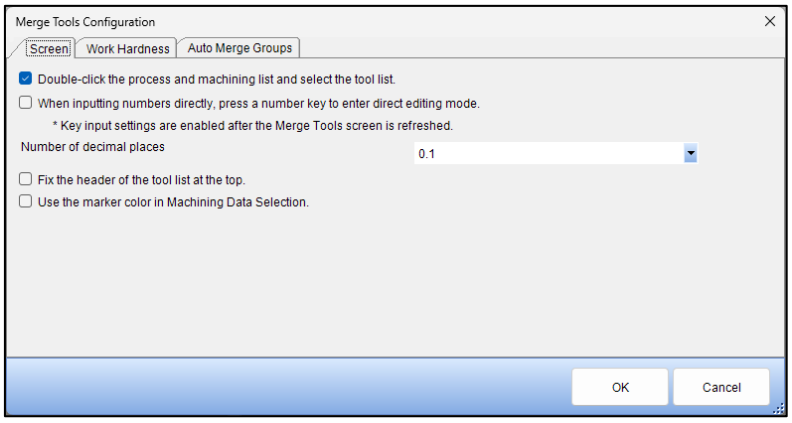
Expand all/Collapse all

Open (or close) the entire machining data list in the tree for each tool setting.

Settings

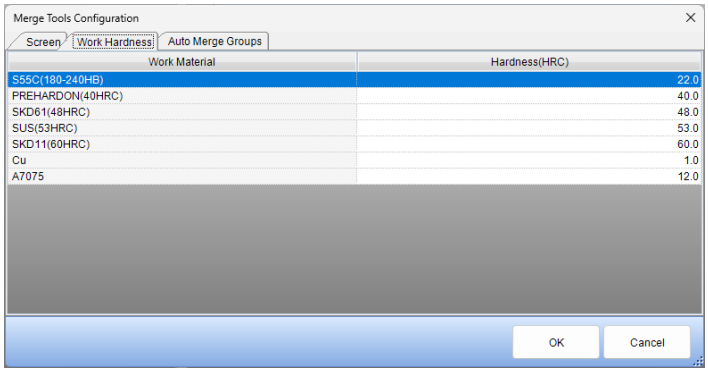
The [Merge Tools Setting] screen is displayed.
Set the following three items related to the [Merge Tools] function.

- **Screen**
  Configure the initial display settings for the [Merge Tools] screen.



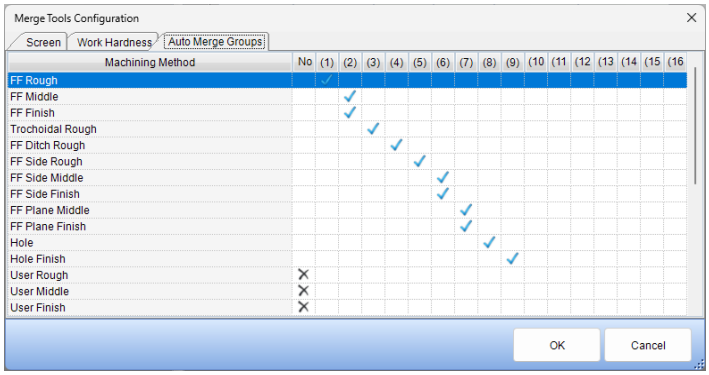
● **Work Hardness**

Set the initial hardness value for each registered workpiece as the HRC value.
The FFCAM uses the set hardness for the merge determination of auto merge.



● **Auto Merge Groups**

Set whether the [Auto Merge] function can target each tool machining method.



* For details on each setting, see the Help window of FFCAM.

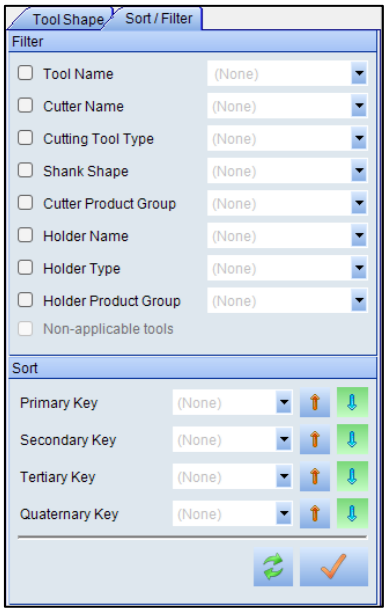
(4) Tool Shape, Sort/Filter

Tool Shape

A preview of tool shape is displayed together with its dimensions.

Sort/Filter

You can filter the tree displayed in [Tool Standard View] by item name or sort the items by type.



* For details on sorting and filtering, see the Help window of FFCAM.

(5) Machining Data Information

All data registered in [Machining Data Selection] (number of processes, number of machining, and number of tools) is displayed.

3. Added a Function to Easily Check the Machining Area of Machining Data

A function has been added that allows you to display machining areas on the model by simply selecting each machining data item in the [Machining Data Selection] list.
Now you can easily check the machining area of each machining data item without opening the [Machining Parameter Setting] screen.

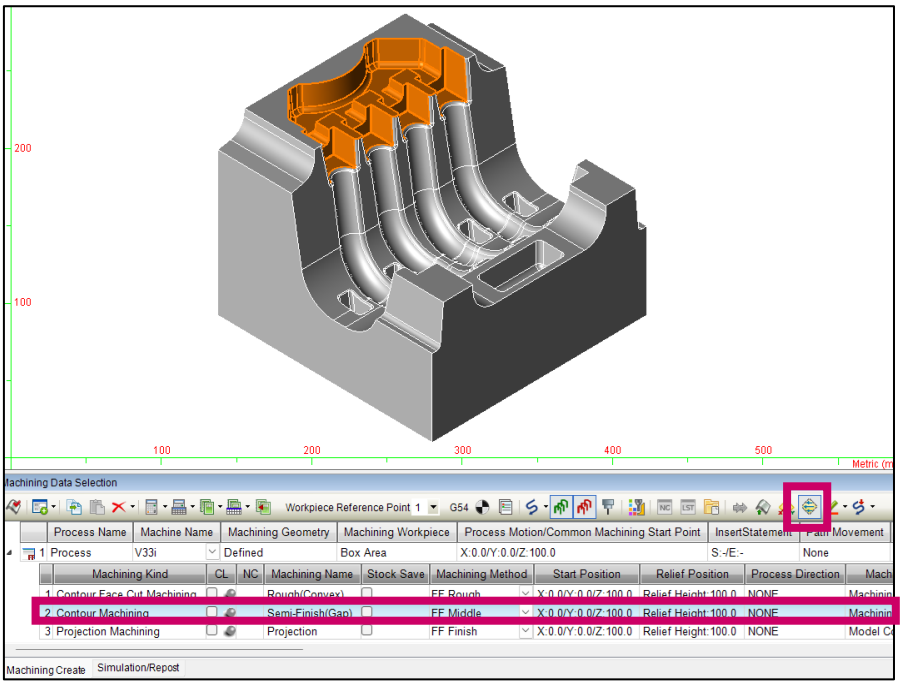
■ Setting Screen

Click the [Machining area display] button in the command bar of [Machining Data Selection] to select it.
When you select machining data in the list, the machining area set to that machining data is displayed on the model.

When a file has multiple machining data items, selecting each one also changes the displayed machining area. For example, by selecting machining data in order using the <↑> and <↓> keys on the computer, the machining area display will also switch accordingly.

Select [Machining area display] in the command bar of [Machining Data Selection].

When you select machining data in the [Machining Data Selection] list, the machining area set to that machining data is displayed on the model.



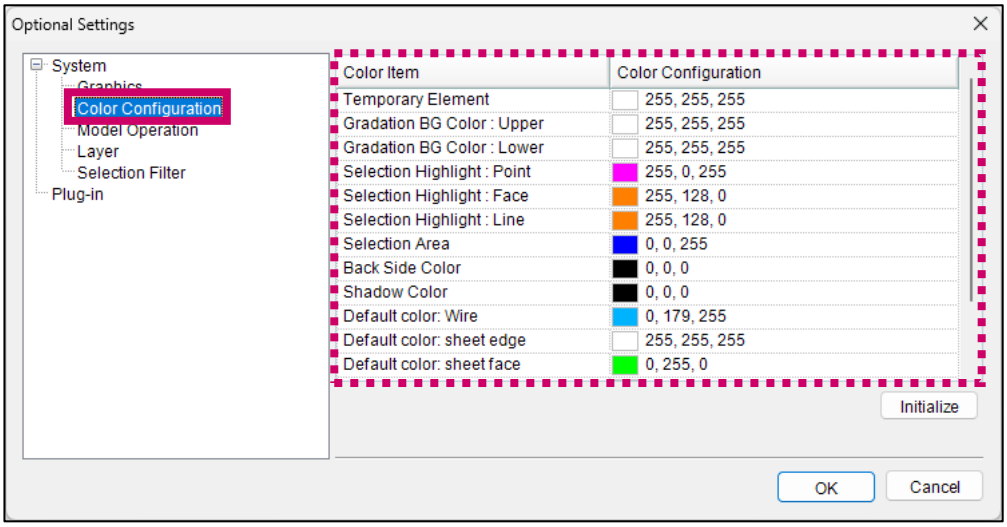
■ Settings of the Displayed Machining Area

The contents of the following machining area settings will be displayed on the model.

- Area Type
- Z Area Set
- Area Offset (Curve, Curved Surface)
- Area Set (XY Area, Circle Area, Curve, Curved Surface)

■ Color of the Displayed Machining Area

The color set in [Color Configuration] of [Optional Settings]/[System] is reflected on the model.



■ Supplementary Note

・ When the [Machining area display] button is selected, machining data without an area setting can be selected from the list. However, in such cases, the machining area will not be displayed on the model.

■ Note

・ When the model is selected in the graphics window, the machining area that is highlighted may be duplicated. In such cases, cancel the model selection.

4. Support for Auto Setting of Specify Contour Projection Machining in Contour Projection Machining Using Color Filters

On previous FFCAM, when a color filter was set to the machining area of machining data registered in the user database, copying and using the data would automatically apply the area settings to areas with the specified color.
In FFCAM 2025, this function can now also be used with [Specify Contour Projection Machining] for contour projection machining. When you specify the contour and projection surfaces by a color filter in [Specify Contour Projection Machining] of the contour projection machining data and register it to the user database, you can now automatically set the contour and projection when you copy and use the data.

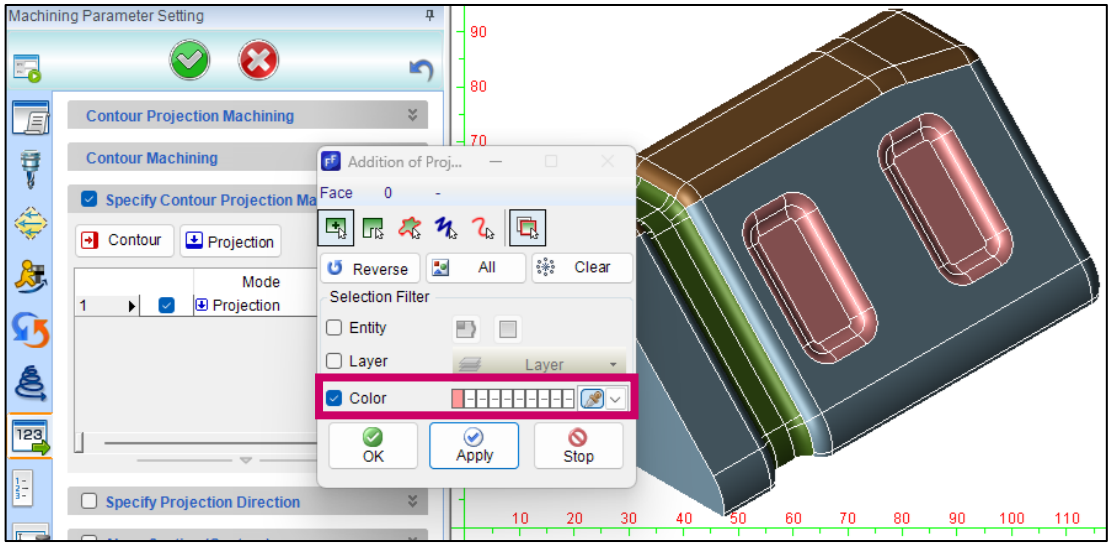
■ Setting Screen

Use the [Specify Contour Projection Machining] function with [Advance] mode for contour projection machining.
This function can be used to register the contour and projection surfaces. By specifying colors for each surface using a color filter, the function automatically assigns the specified color surfaces as the contour and projection surfaces when using the data copied from the user database.
Basically, the contour and projection surfaces are categorized by angle, but regardless of the angle, you can specify the surfaces to be set as the contour or projection surfaces using color.

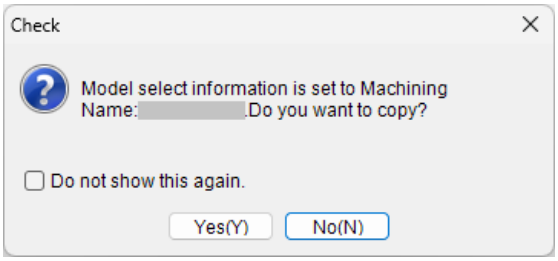
[Specify Contour Projection Machining] screen

Use a color filter to assign colors to the contour or projection surfaces.

When you register machining data with a specified color filter in the user database, and then copy and use the data, the surfaces with the applied colors are automatically set as contour or projection surfaces.



Register contour projection machining with a color filter set in the user database, and when you copy the data, the following message is displayed.
To retrieve the selection elements of the model based on the filter information and automatically apply them to the contour projection machining specification list, click [Yes].



■ Supplementary Note

・ This function is executed using the machining template saved in [Advance] mode.
In [Normal] mode, the specify contour projection machining function is not executed because it is not supported.

5. Improved List Display on the Machining Parameter Setting Screen

On some of the machining parameter setting screens and the approach vector setting screens, the list area used to display only a few lines, making the contents difficult to read.

In FFCAM 2025, the lists with a small number of displayed rows have been improved, and a function has been added to adjust the vertical width of the list display.

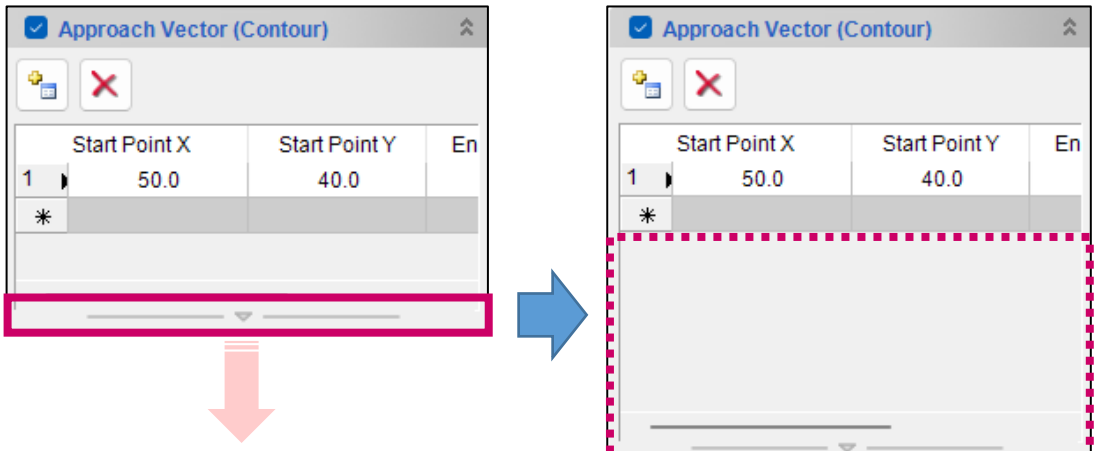
■ Setting Screen

Screen operation example)

[Approach Vector] Setting Panel

Drag the label part displayed in the lower section of the list up and down to vertically expand the display of the list portion.

If the number of rows in the list increases, you can expand the display range to check the entire list.



Drag the label.

Expand the display range.

■ Corresponding Areas

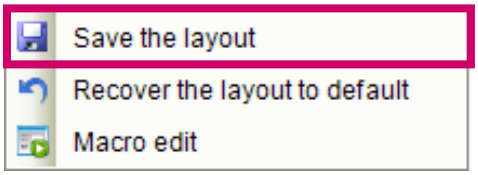
The list portions of the following machining parameter settings support display width adjustment.

Machining Name	Parameter Name
Along-Surface Machining	Drive Surface
3D Equi-Pitch Closed Machining	Guide Curve
3D Equi-Pitch Open Machining	Open Guide Curve
2D Route Machining	Guide Wire
3D Curve Machining	Reference Curve
Chamfering Route Machining	Guide Wire
Contour Projection Machining	Specify Contour Projection Machining, Specify Projection Direction, Approach Vector
Contour Machining/Contour Face Cut Machining	Approach Vector
Flat Face Machining	Approach Vector
Flat Edge Machining	Approach Vector
Contour Projection Machining	Approach Vector

■ Supplementary Note

- If you open the right-click menu on the machining setting screen and set [Save the layout], the display width of the changed list portion can be saved for subsequent times that the screen is displayed.

  * [Save the layout] is a function that saves the settings for each machining setting screen. Even if you save the machining setting screen in one place, the setting contents are not saved for other machining setting screens.



6. Improved Function that Checks for Duplicate Process and Machining Names When Registering to the User Database

In previous versions of FFCAM, when you opened [User Database] from the [Machining Data Selection] list and registered machining data or process data, if a machining name or process name was duplicated, only an error message was displayed, and you needed to cancel the operation temporarily.

In FFCAM 2025, a function that allows you to change the process or machining name has been added to the error message screen, allowing you to change the name without canceling the operation.

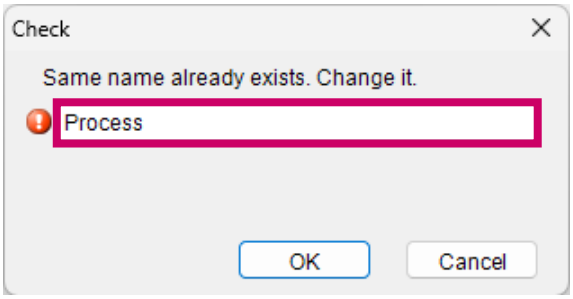
■ Setting Screen

When you open [User Database] from the [Machining Data Selection] list and register process data or machining data, if a machining name or process name is duplicated, the [Check] screen is displayed.

[Check] Screen

Change the name on the [Check] screen, and register it to the database.

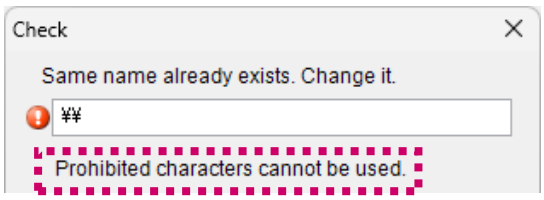
A name that is duplicated is displayed in the input field.
Change the name, and click [OK] to register the changed name to the user database.



If the changed name contains one of the following problems, a message is displayed below the input field.
Change it to a valid name.

- Name duplication
- Blank
- Prohibited characters (*, ?, ¥, ", <, >, |, /, :)
- Character limit (must be 64 half-width characters (32 full-width characters) or less)

Display example) Forbidden characters



7. Clarified Usage Conditions of the Tool Load Reduced Infeed Function

The usage conditions of the [Tool Load Reduced Infeed] function under [Area Detail Setting] of [Machining Area] have been clarified.

In previous versions of FFCAM, the [Tool Load Reduced Infeed] setting item was displayed even when the usage conditions were not met.
In FFCAM 2025, the setting item is grayed out to visually indicate that the usage conditions are not met.

■ Setting Screen

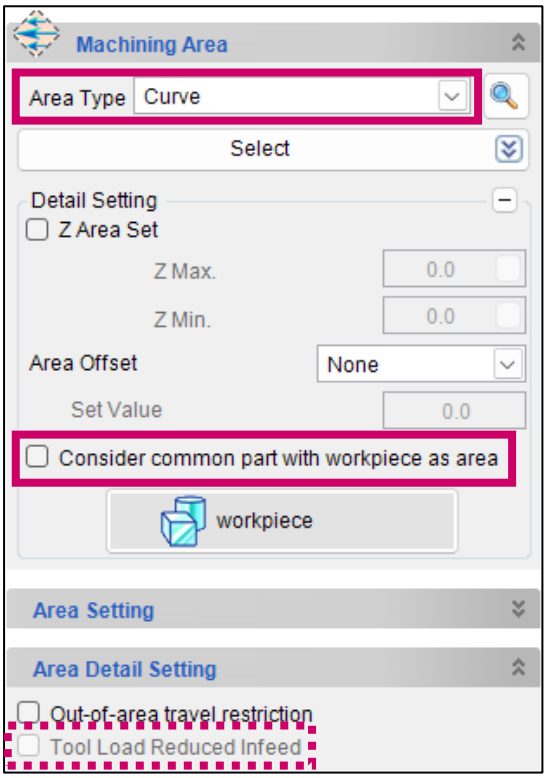
The usage conditions of the [Tool Load Reduced Infeed] function under [Area Detail Setting] of [Machining Area] are as follows.

- When [Area Type] is "MC work"
- When [Area Type] is other than "MC work" and "Consider common part with workpiece as area" is specified

[Machining Area] screen

When the settings do not meet the usage conditions, [Tool Load Reduced Infeed] under [Area Detail Setting] is grayed out and cannot be selected.

Display example) When [Area Type] is other than "MC work" and "Consider common part with workpiece as area" is not specified



The display is grayed out.

8. Added a Preview Function to the Common Relief Height Setting

The preview function that can be used when you input [Relief height] in the machining parameter settings can now also be used when you input the height of the [Common Relief Height] setting in the machining settings.

In previous versions of FFCAM, the [Common Relief Height] setting could only be executed by entering number values.
In FFCAM 2025, the preview function allows you to visually set and check the height.

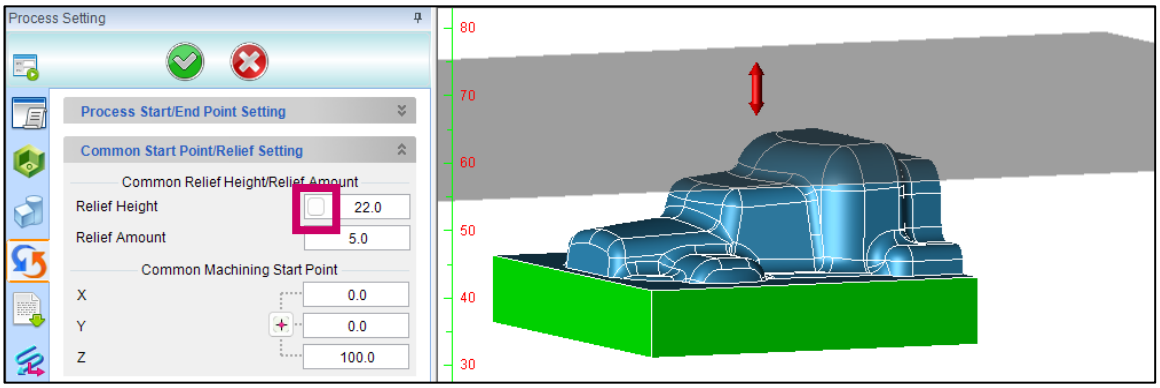
■ Setting Screen

In [Process Setting] -> [Process Start/End Point/Relief Setting] -> [Common Start Point Setting], a button for executing the preview setting has been added to the input field of [Relief Height].

* This function is the same as the preview setting button for the height input field in [Machining Parameter Setting] -> [Relief/Approach/Infeed Setting] -> [Relief Motion] -> [Relief height].

[Common Start Point Setting] Panel

When you click the preview setting button for the input field of [Relief height], a semi-transparent plane is displayed on the model. By moving this plane vertically, you can visually set and confirm the height.



9. Partial Improvements to Setting Screen for Machining Workpiece and Machining Area

The layout of input items have been changed in the [Box Area] setting screen of the machining workpiece and the [XY Area] setting screen of the machining area, making it easier to find the desired input fields.

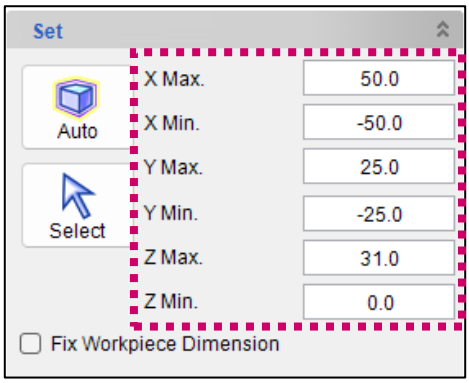
* The function has not changed.

■ Setting Screen

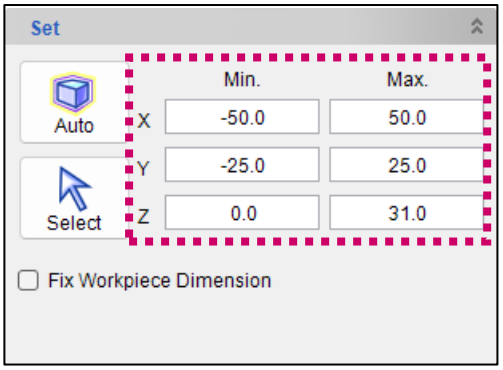
[Box Area] screen

The layout of the [Box Area] setting screen under [Process Setting] -> [Set of Machining Workpiece] has been changed.

Previous FFCAM



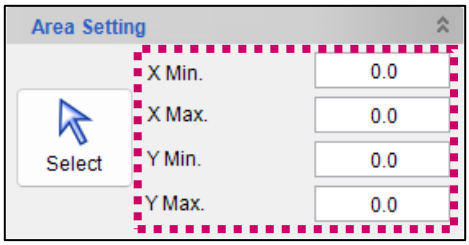
FFCAM 2025



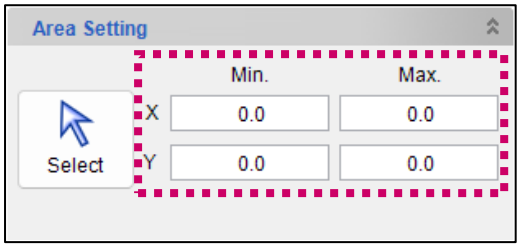
[Area Settings]/[XY Area] screen

The layout of the [XY Area] setting screen under [Machining Parameter Setting]/[Area Settings]/[Machining Area] has been changed.

Previous FFCAM



FFCAM 2025



10. Improved Angle Input Function When Setting Index Machining and Machining Direction

When performing [Machining Direction] for index machining, if the angle input value is smaller than the minimum set unit of the machine, FFCAM will automatically adjust the value.

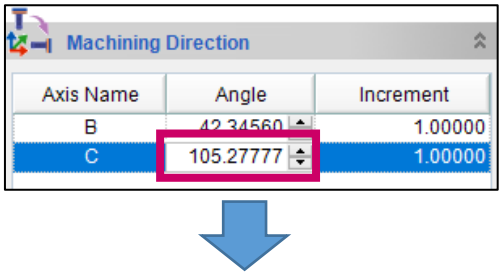
In previous versions of FFCAM, when the angle input value was smaller than the minimum set unit of the machine, the value had to be corrected either manually or using the [Index Adjustment] function.
In FFCAM 2025, because FFCAM automatically adjusts the value, manual correction of the value is no longer necessary.

■ Setting Screen

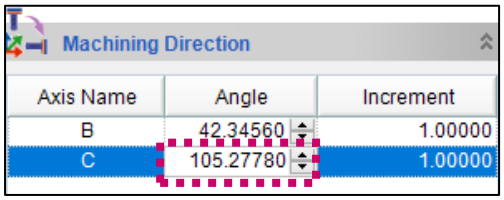
Input example) When a value smaller than the minimum set unit of "0.0001" is input in a machine setting

[Machining Direction] screen

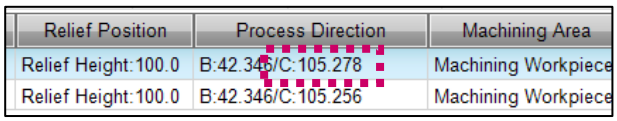
Enter "105.27777" for the C-axis angle.



The value of the C-axis is automatically adjusted to "105.27780".



At this time, the value of the machining direction in [Machining Data Selection] is displayed as "105.278".
(The output digits are rounded to three decimal places.)



* All values below the minimum set (display) unit are rounded to the nearest unit.

■ Settings in Which Values Are Automatically Adjusted to the Minimum Set Unit

When setting the following seven items, the input value of the minimum set unit is automatically adjusted.

1. Manual input
2. When picking a surface with [Surface Normal Direction]
3. When picking two points with [Two Points]
4. When specifying the machining direction with [Current View Direction]
5. When specifying the opposite direction with [Z Opposite Direction]
6. When calculating the index direction with [Machining Direction Calculation]
7. When specifying the rotation angle with [Direction Setting]

■ Supplementary Note

・ The [Index Adjustment] function under [Angle Adjustment] that was available in previous versions of FFCAM has been removed.

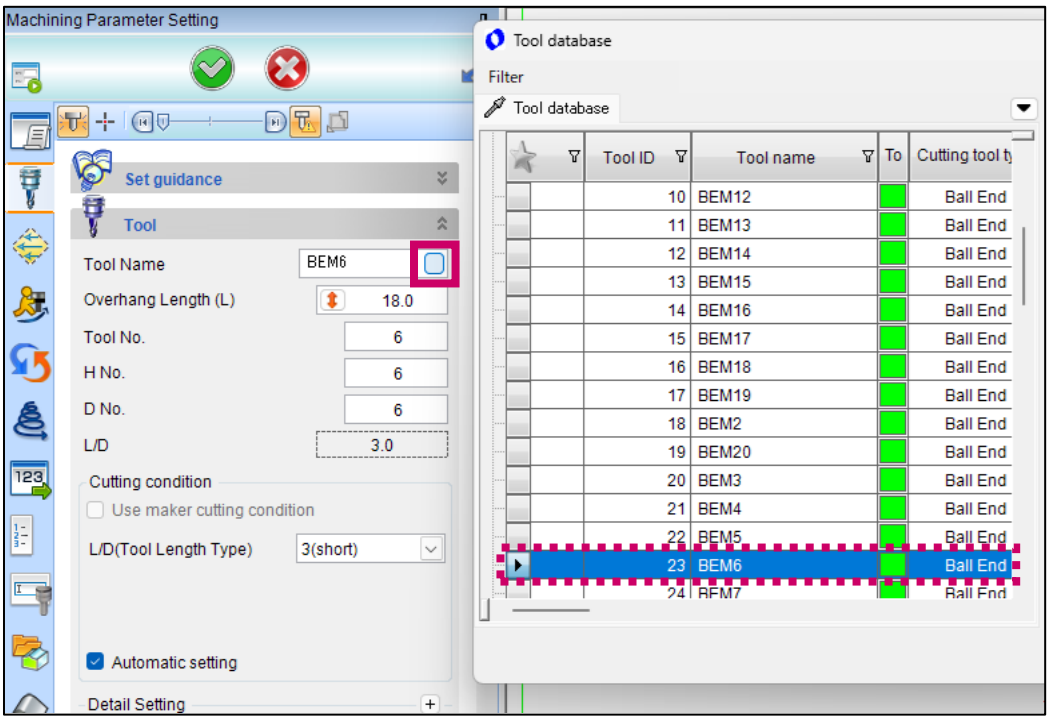
11. Improved Functionality for Selecting Tools from the Tool Database

In previous versions of FFCAM, when setting tools in the machining parameter settings, launching the tool database would open the screen with the first tool from the "Cutter Database" selected.

In FFCAM 2025, when the tool database is launched (if a tool is already set), the screen now opens at the position where the previously set tool is selected.
This makes it easier to reselect a tool of a different size from the same tool group.

■ Setting Screen

When you launch the tool database from [Tool Setting] in the machining parameter settings, if a tool has already been set, the screen is displayed at the position where that tool is selected.



■ Supplementary Note

- This function is available in FFCAM 2025 when a tool is selected.
  When using data created in an earlier version of FFCAM, the function becomes available after you reselect a tool in FFCAM 2025.

12. Added Machine Data to the Machine List

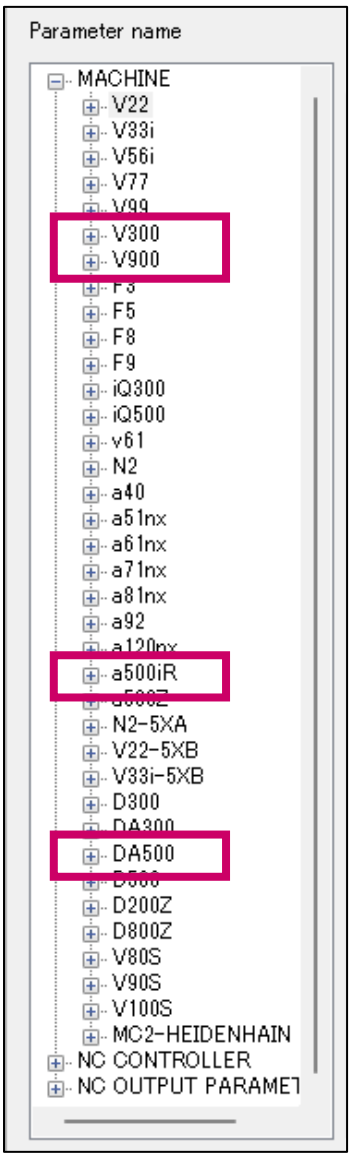
New machine data has been added to the FFCAM machine list.

■ Setting Screen

The following four machine data items have been added to the machine list.

- V300
- V900
- a500iR
- DA500

[Machine Parameters] screen



13. Enhanced the Curve Extension Function

The [Create Curve]/[Curve extension] functions have been enhanced.

In previous versions of FFCAM, a target curve could not be extended unless it directly intersected with a reference curve.
In FFCAM 2025, even if a target curve does not directly intersect another curve, it can still be extended to an expected intersection point if that point lies along the extension of a different curve.

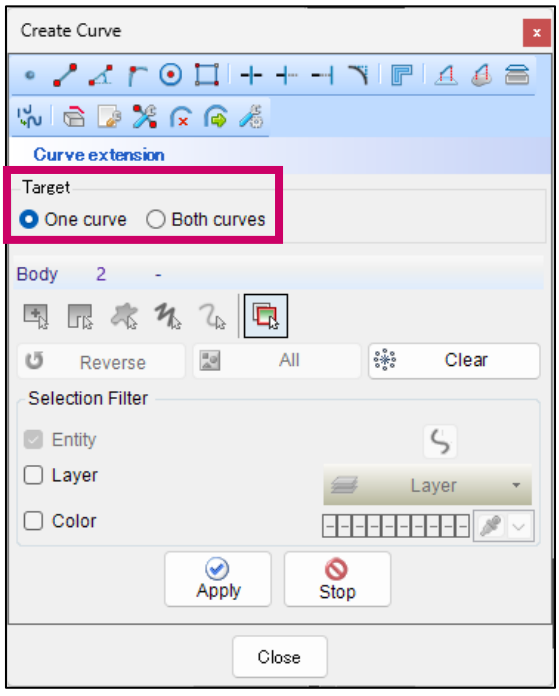
Also, in previous versions of FFCAM, the target curve and reference curve had to be set separately. However, in FFCAM 2025, specifying both curves now allows them to be extended simultaneously.

■ Setting Screen

Use [Curve extension] on the [Create Curve] screen to extend a curve.

The basic operation method remains the same as before, but you can now specify a curve that does not lie along the extension of the target curve as the reference curve.
When you specify a curve that is not an extension of the target curve, the target curve can be extended toward the intersection point with the extension of the specified curve.

[Create Curve]/[Curve extension] screen



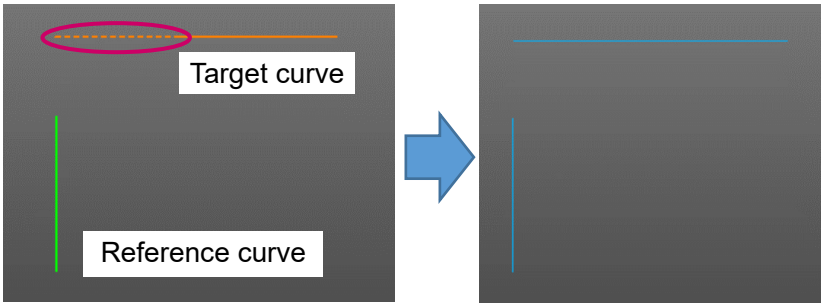
When you select [One curve] for [Target], only the target curve is extended toward the intersection point with the extension of the specified curve.
When you select [Both curves], both specified curves are extended toward the point where their extensions intersect.

■ Extension Example

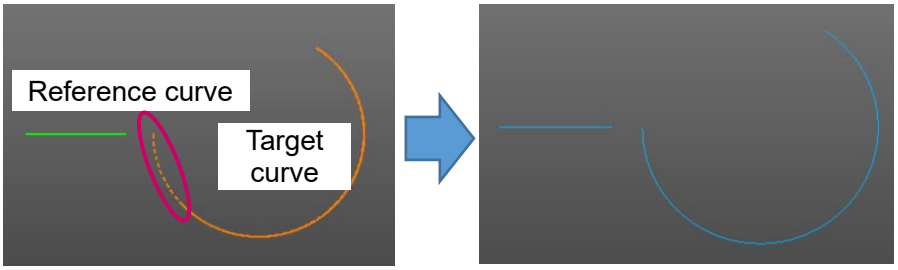
If [One curve] is specified

The target curve is extended toward the point where the extension of the first specified curve (reference curve) intersect with the extension of the second specified curve (target curve).

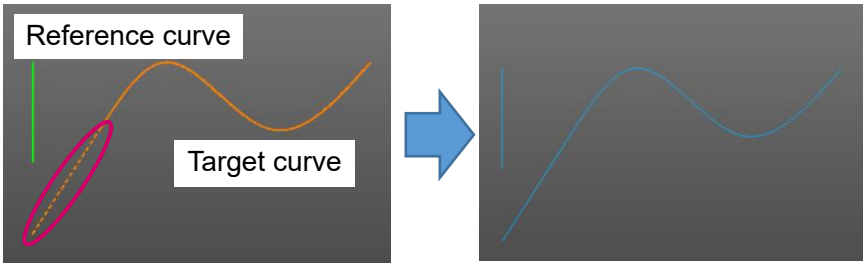
Straight line:



Arc:



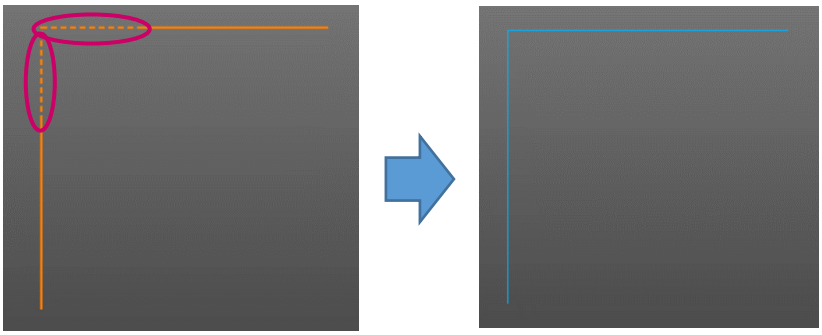
Spline:



If [Both curves] is specified

Both curves are extended toward the point where the extension of the first specified curve intersects with the extension of the second specified curve.

Straight line:



14. Enhanced Model Move/Copy Function

The "Move/Copy" function for models has been enhanced.

In FFCAM 2025, the number of copies can be specified when using [Parallel Movement] and [Rotate].

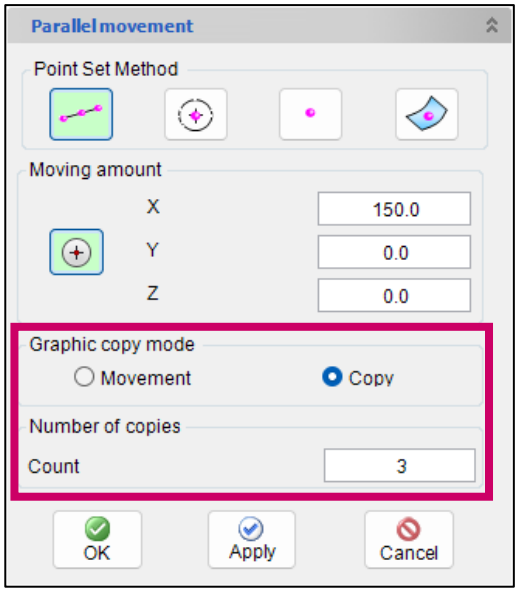
As a result, multiple copies of a model can now be made at once.

■ Setting Screen

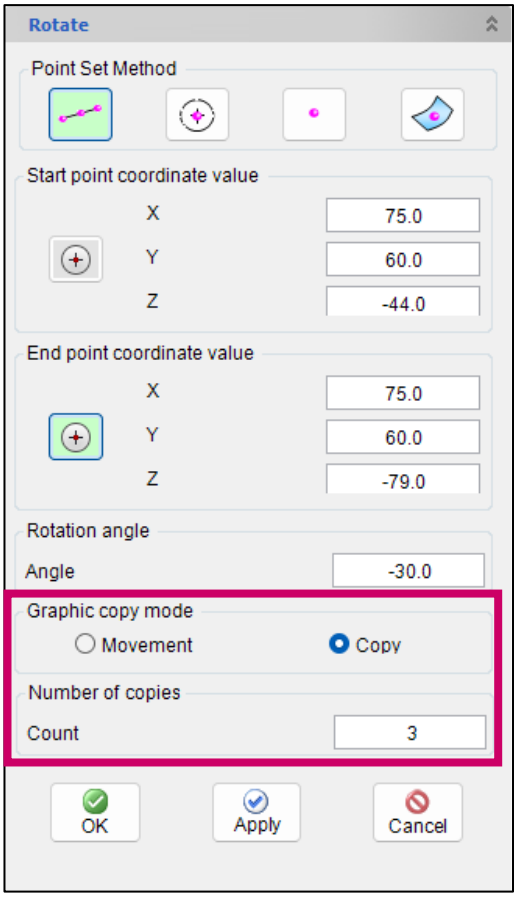
The copy function to make multiple copies of a model using [Parallel Movement] and [Rotation] is available on the [Edit]/[Move] screen.

When you select [Copy] in [Graphic copy mode], you can input [Number of copies]. A model will be copied according to the number of times input.

[Parallel Movement] setting screen



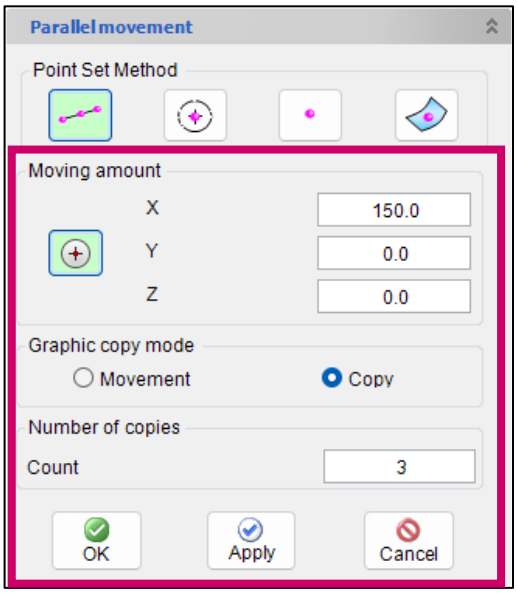
[Rotation] setting screen



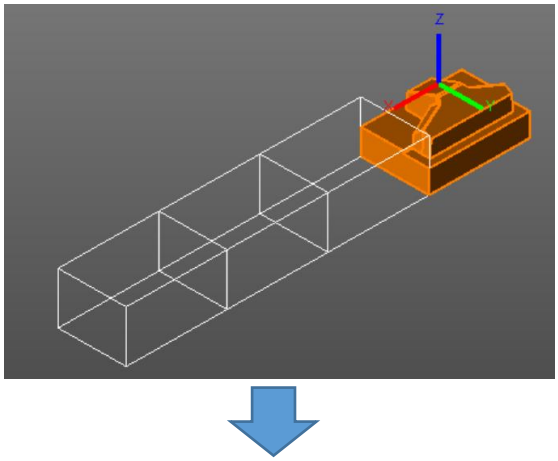
■ Setting Example

Parallel movement

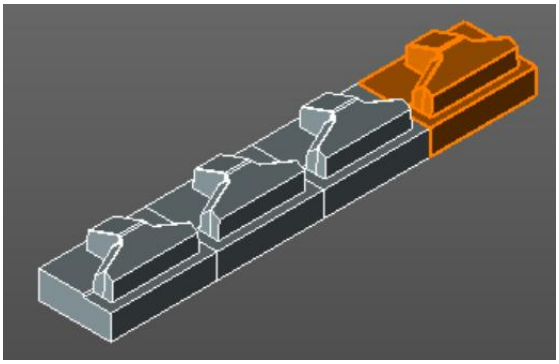
The model is copied the specified number of times, according to the set [Moving amount].



Set [Movement amount] and enter number of copies.
A preview box showing the number of copies is displayed.

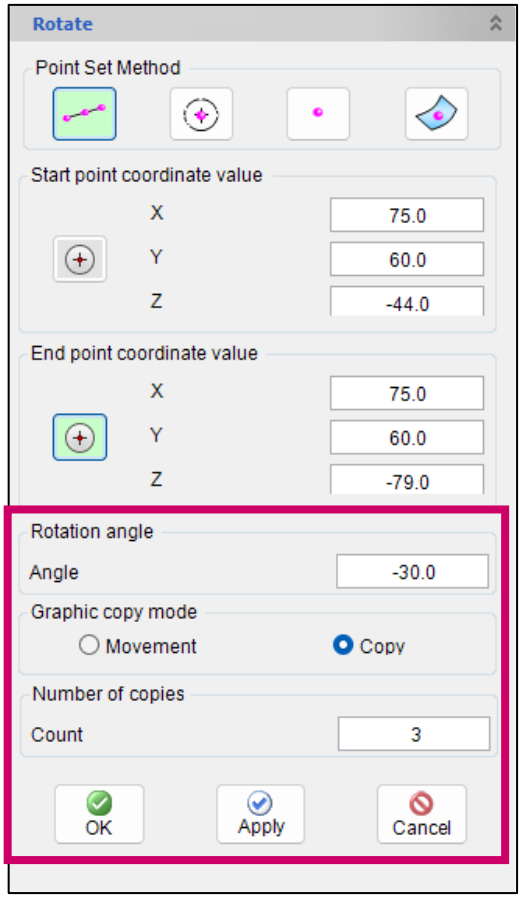


Click [OK] or [Apply] to copy the model.

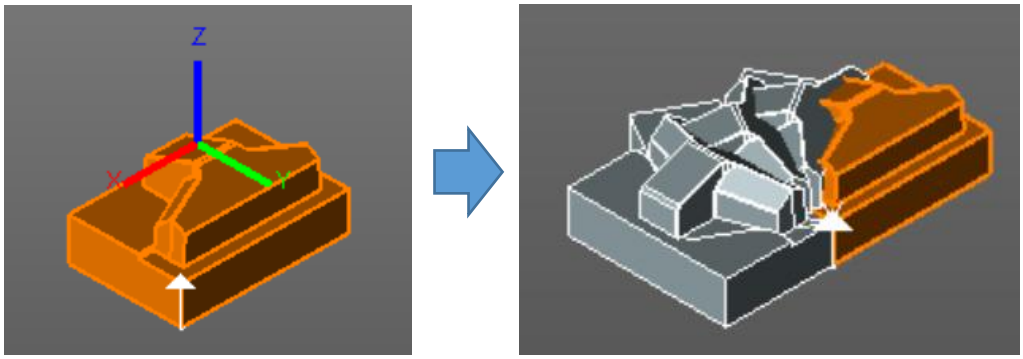


Rotation

The model is copied the specified number of times, according to the set [Rotation angle].



Set [Rotation angle] and enter number of copies.
Click [OK] or [Apply] to copy the model.



* A preview box is not displayed for [Rotation].

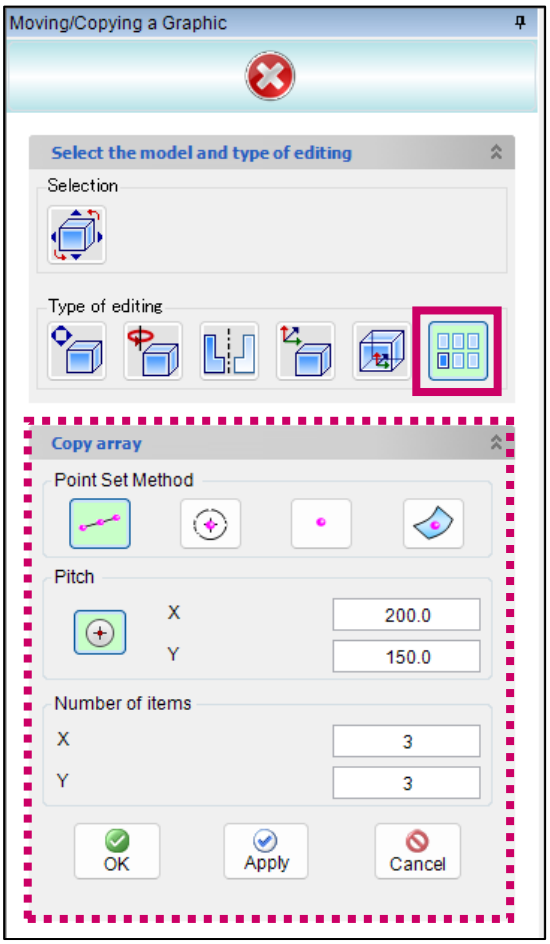
15. Added Copy Array Function to Model Move/Copy

The [Copy array] function has been added to the "Move/Copy" function for models.
The [Copy array] function is a function that makes multiple copies of a model aligned in a grid pattern, according to the number of items specified in the X and Y directions.

■ Setting Screen

You can use [Copy array] on the [Edit]/[Move] screens to copy the specified model multiple times in a grid pattern.

Click the [Copy array] icon in [Type of editing] to open the [Copy array] panel.
Specify the array method on the [Copy array] panel.



Copy array

Point set method

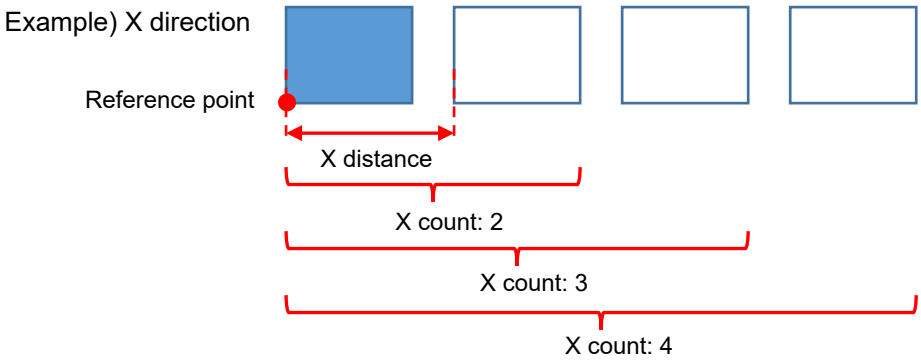
Select the point specification method when picking a point on the model with [Pitch].

Pitch

Specify the distance along the X and Y axes of the array.
Select the  icon and pick two points on the model to specify the distance.
Specify the reference point with point one, and then specify the destination with point two.
Here, the distance can be temporarily estimated, but afterward, adjust the X and Y values as needed to determine the distance that suits your purpose.
Input a value that is an absolute coordinate (XY) in either the + or - direction.

Number of items

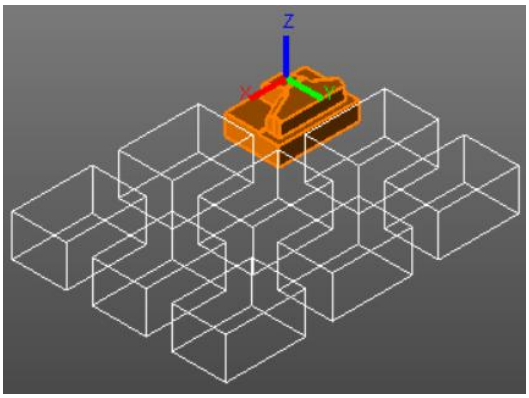
Specify the number of items in the X and Y directions for the array.
Input an integer value of 1 or greater.



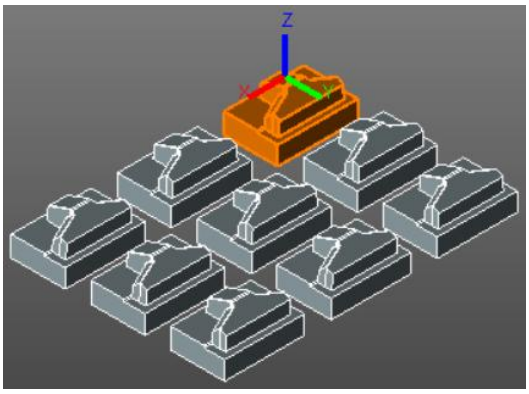
■ Setting Example

The model is copied for the specified number of items in X and Y, according to the "Movement distance" specified as the pitch.

Specify the "Movement distance" for the pitch and input the number of X and Y items in the array.



Click [OK] or [Apply] to copy the model to an array.



16. Enhanced Copy and Move Function of Curve Creation

The [Copy and Move] function for [Create Curve] has been enhanced.

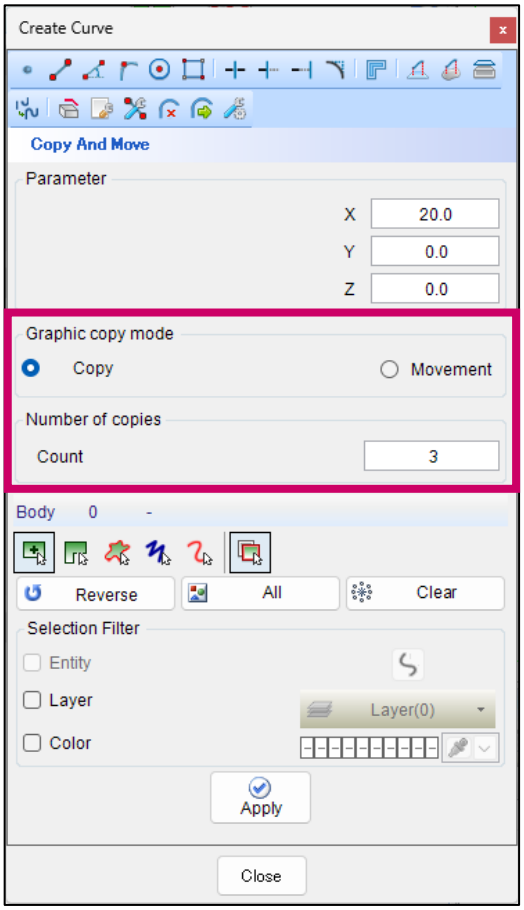
You can now specify the number of copies and copy multiple curves at once.

■ Setting Screen

You can specify the [Number of copies] in the settings for [Create Curve] and [Copy and Move].

A curve can be copied multiple times, similar to "Parallel movement" in "14. Enhanced Model Move/Copy Function".

[Create curve]/[Copy and Move] screen



17. Added Random Color Setting Function to the Angle Deviation Setting of Measurement

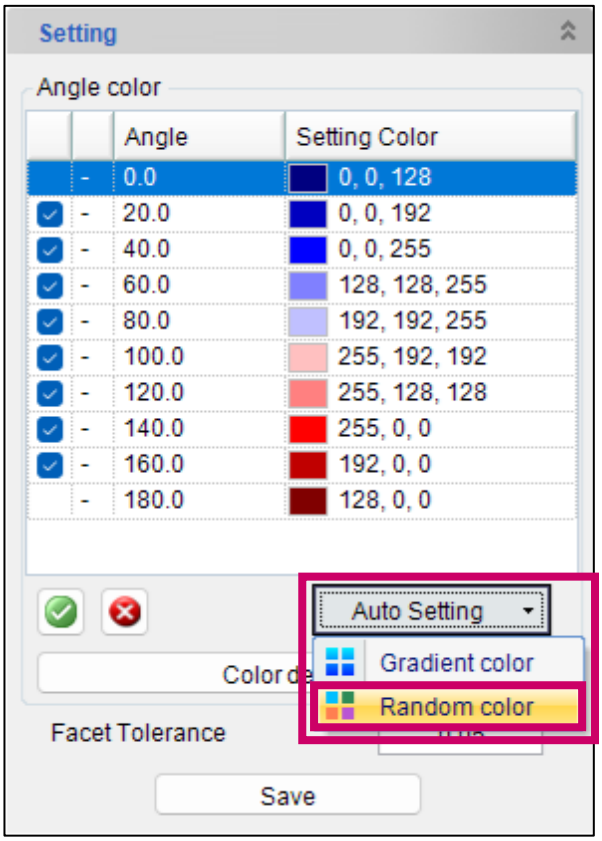
The random color setting function, which could previously be used with the [Corner R] setting in measurements, can now also be used with the [Angle deviation] setting under the same measurement.

■ Setting Screen

On the [Setting] screen of [Measurement]/[Angle deviation], select the [Auto Setting] method from the pull-down list. You can select between the previous function [Gradient color] or the newly added [Random color].

* This function is the same as [Auto Setting] on the [Setting] screen of [Measurement]/[Corner R].

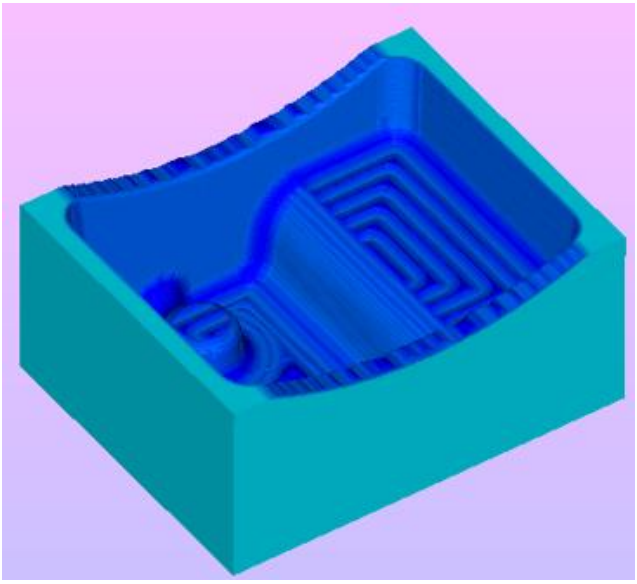
[Setting] screen of [Angle deviation]



You can select [Random color] from the pull-down list.

18. Improved Method for Displaying Uncut Workpiece During Simulation Create Data (Details)

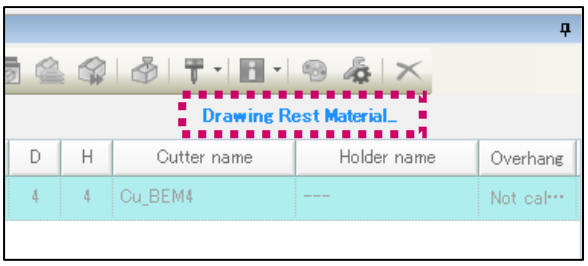
The display of the uncut workpiece during the "Create Data (Details)" simulation used to take time to update after performing view operations.
The display method has been improved to significantly reduce the time required for view operations on the uncut workpiece display.



The time it takes to display the uncut workpiece is slightly longer in FFCAM 2025 compared to FFCAM 2024. However, the time required for display after subsequent view operations has been significantly reduced.

■ Display During Processing

When the simulation engine is "Extend (3-5 Axis)":
While processing the uncut workpiece display, a message stating "Drawing Rest Material..." appears in the [Machining Data Selection] window.
* When the simulation engine is "Standard (3 Axis)", this message is not displayed.



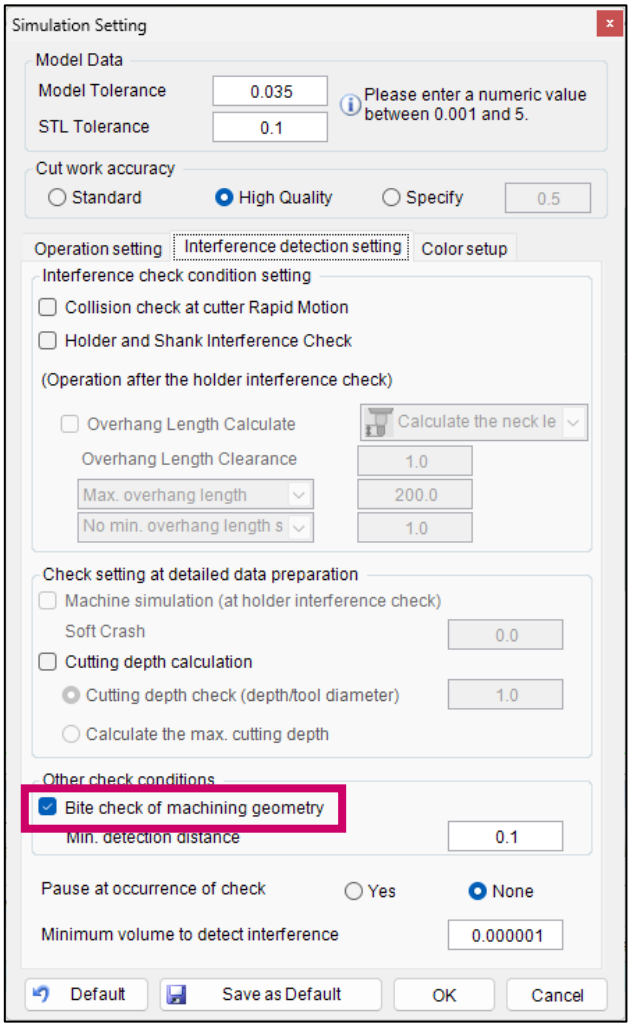
19. Changed the Initial Value of Bite Check of Machining Geometry in the Simulation Setting

In [Interference detection setting] on the [Simulation Setting] screen, the initial value set to [Bite check of machining geometry] has been changed.

■ Setting Screen

[Simulation Setting]/[Interference detection setting] screen
The checkbox for [Bite check of machining geometry] is now selected by default.

The initial value now has the checkbox selected.



■ Note

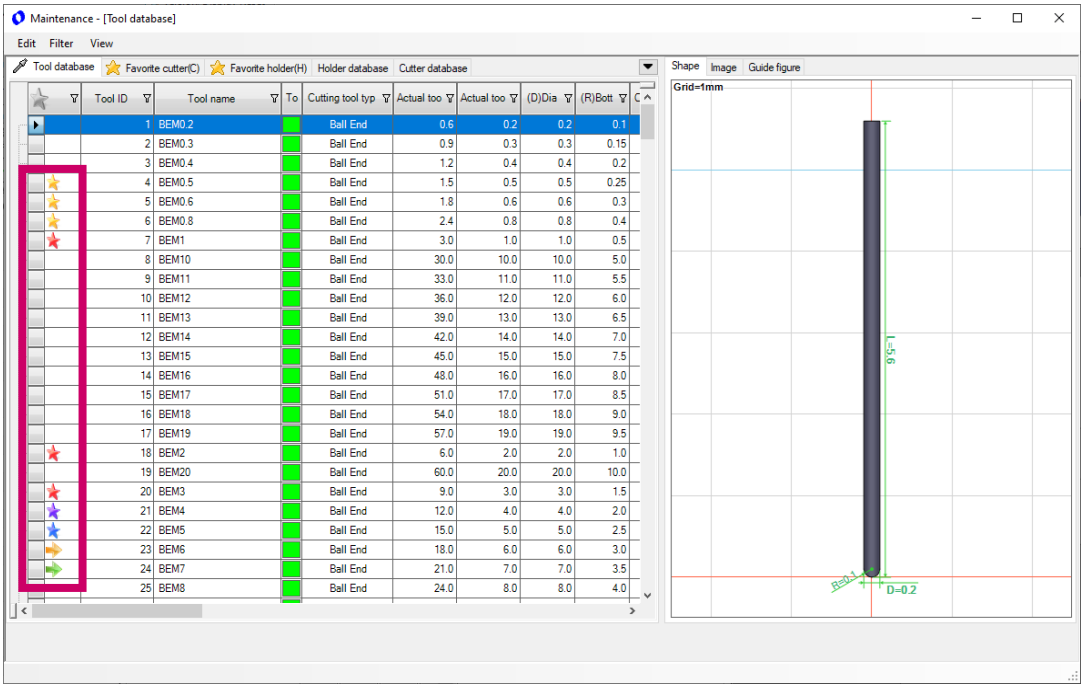
- Be aware that the initial value of data migrated through the user data conversion function will not be changed during a version update.

20. Added a Customization Function for Favorite Icons on Tool Database Maintenance Screen

Multiple types of the favorite icons can now be set for the Tool Database, Cutter Database, and Holder Database lists.
The picture, name, and attributes can be set for each favorite icon.

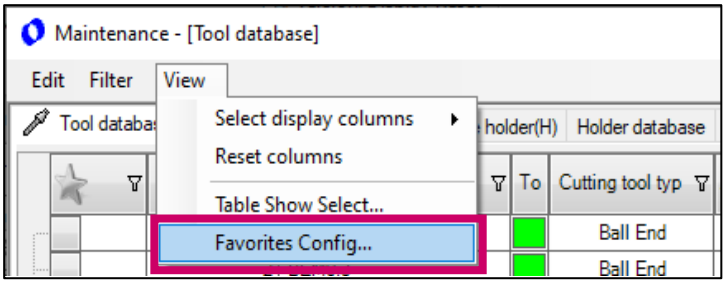
You can apply the set favorite icons to each data row in the lists and use them as markers for sorting, arbitrary grouping, and other forms of data management.

Example) Tool Database Screen



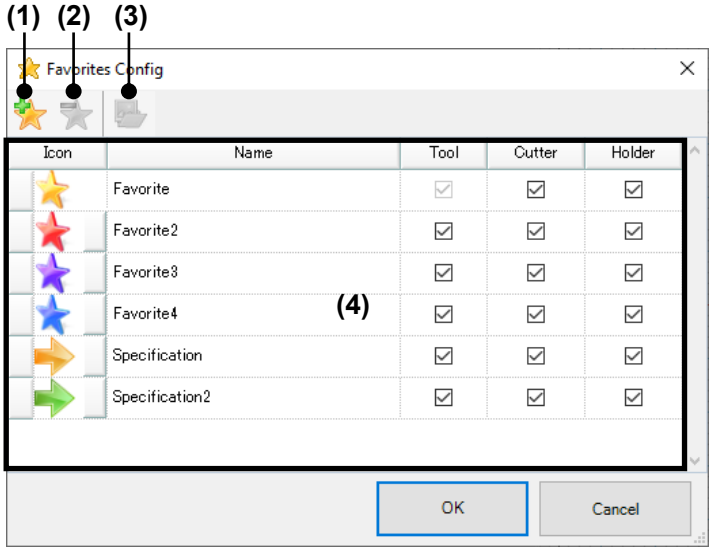
■ Editing Favorite Icons

On the Tool Database Maintenance screen, select "Favorites Config…" from the [View] menu to open the [Favorites Config] screen.



[Favorites Config] screen

Add, delete, and edit your favorite icons on the [Favorites Config] screen.



(1) Adds to favorite

Add a new favorite icon to the favorites list.

(2) Deletes favorite

Delete the selected favorite icon from the favorites list.

(3) Select the icon image

Launch the [Select icon] screen.
You can customize the icon images.
For more details, see the section on the [Select icon] screen described below.

(4) Favorites list

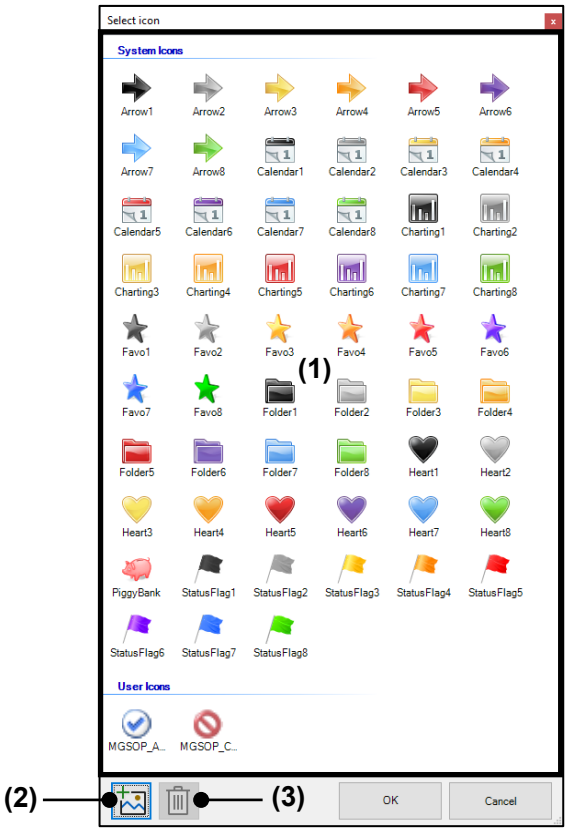
View and set favorite icons in a list.
The picture, name, and attribute (tool, cutter, holder) can be set for each favorite icon.

■ Registering Images Used in Icons

[Select icon] screen

Click the [Select the icon image] button on the [Favorites Config] screen to open the [Select icon] screen.

Select an image to be used for the favorite icon and register it.



(1) Icon list

A list of registered icons is displayed.

In addition to the icons registered to the system, icons arbitrarily added by the user are also displayed.

(2) Add user icon

Open the file selection screen.

The user can arbitrarily select icon images to add.

(3) Delete user icon

Delete the icon selected in the list.

* Icons registered to the system cannot be deleted.

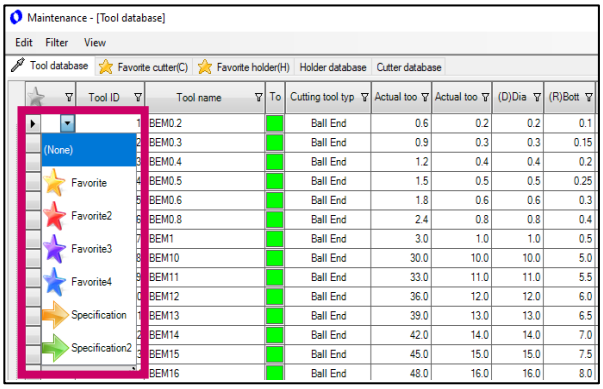
■ Applying Favorite Icons

This section explains how to apply favorite icons to data rows on the Tool Database, Cutter Database, and Holder Database lists.

Example) Tool Database Screen

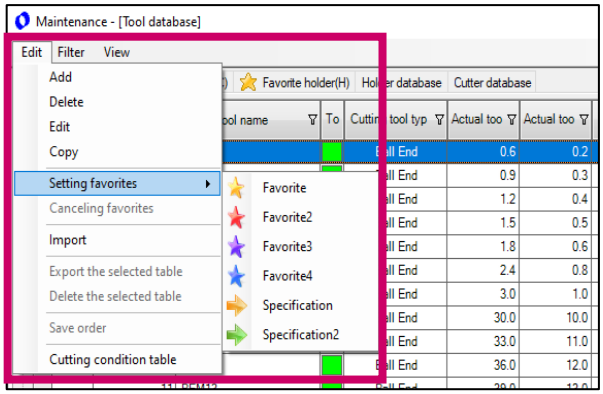
"Favorite" pull-down menu

Select an icon from the pull-down menu in the "Favorite" column of the data row.



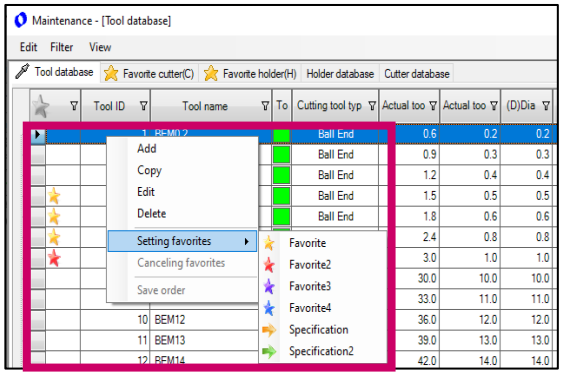
[Edit] menu

Select a data row and click the [Edit] menu -> "Setting favorites", and select an icon.



Right-click menu

Right-click a data row, click "Setting favorites", and select an icon.



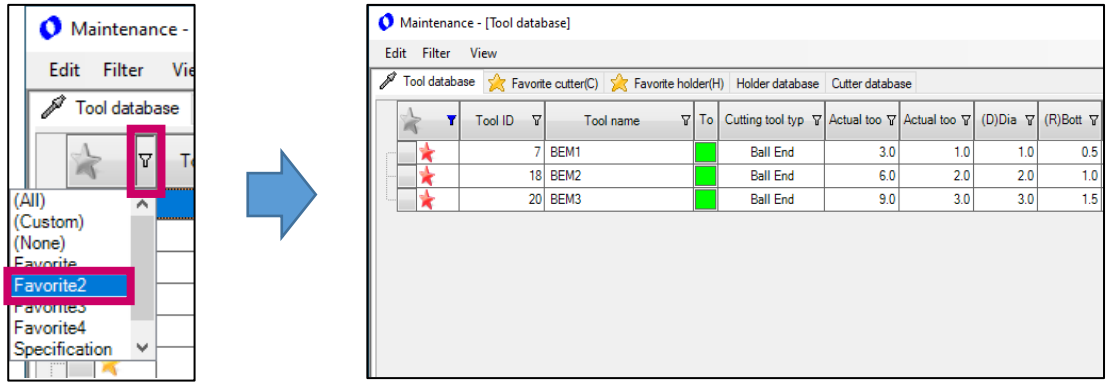
■ Sorting by Favorite Icon

Use the filter function in the "Favorite" column to sort data by icon.

Example) Tool database screen

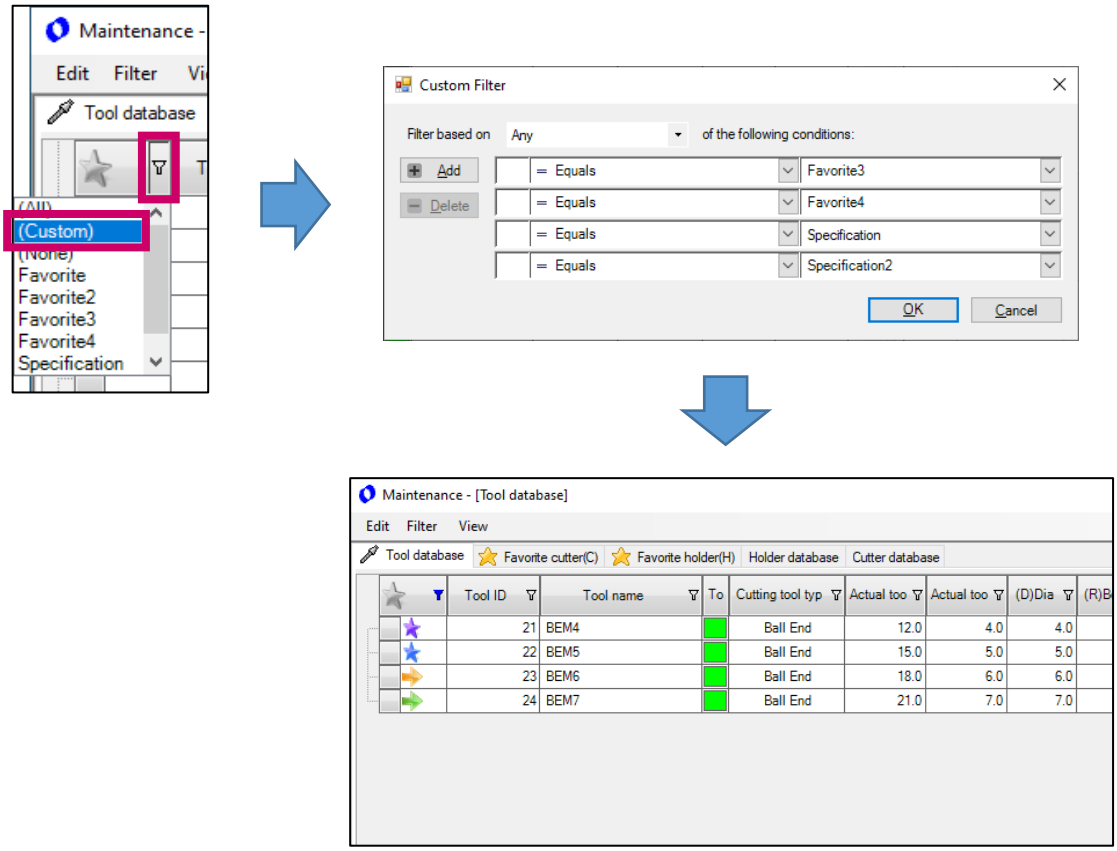
"Favorite" column pull-down menu

From the filter menu in the "Favorite" column, select an icon name to sort the data.



Custom filter

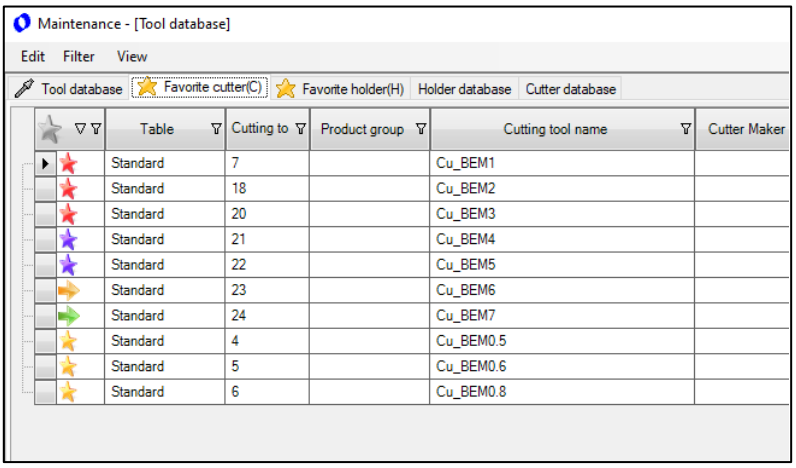
By selecting "(Custom)" from the filter menu, you can specify multiple icons to sort the data.



■ Displaying Favorites in a Table

In the [Cutter database] and [Holder database], when you apply a favorite icon to data, all data items with that icon will be displayed in the [Favorite cutter] and [Favorite holder] lists.

Example) Favorite cutter screen



21. Added a Reference Folder Setting Function When Importing Models

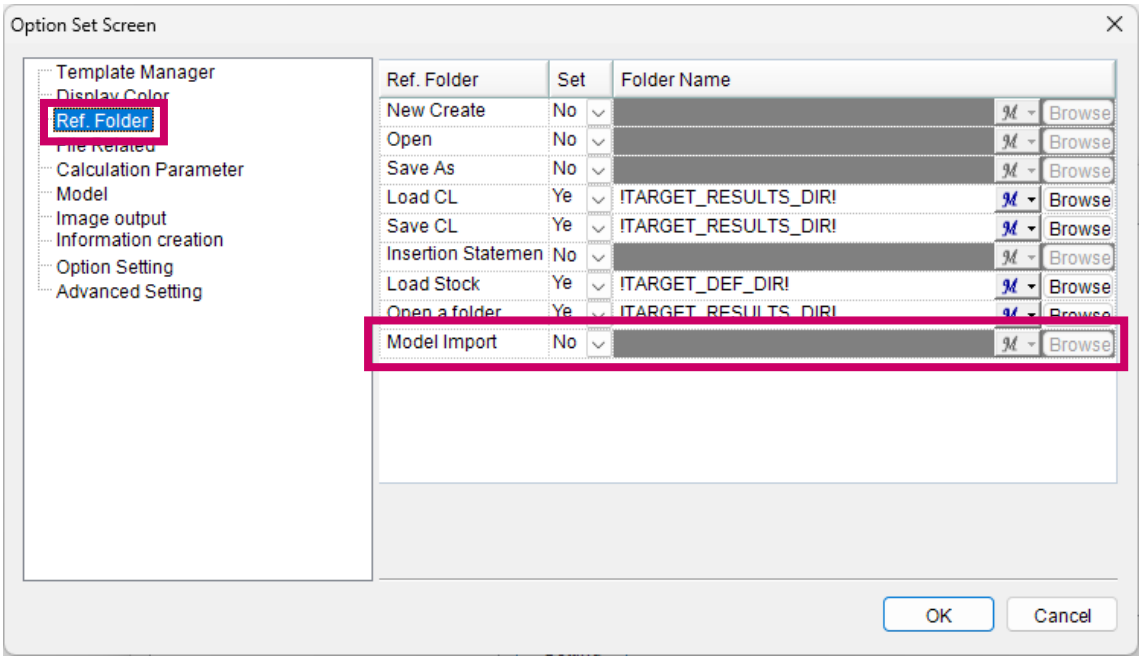
A function has been added to specify the data folder referenced during [Model Import] execution.

In previous versions of FFCAM, the folder used for the previous operation was referenced. However, in FFCAM 2025, the reference folder can be set in advance.

■ Setting Screen

Set the folder from [Ref. Folder] under [Option Set Screen].

[Option Set Screen]
In [Model Import] of [Ref. Folder], [set] specify to [Yes] to specify the folder to be referenced when [Model Import] is executed.
Specify the target folder either by using a macro or by specifying the path via the [Browse] button.



* If a reference folder is not specified, the folder used during the previous operation is referenced, as in earlier versions of FFCAM.

22. Added a Function to Set an Initial Value for View When Importing Models

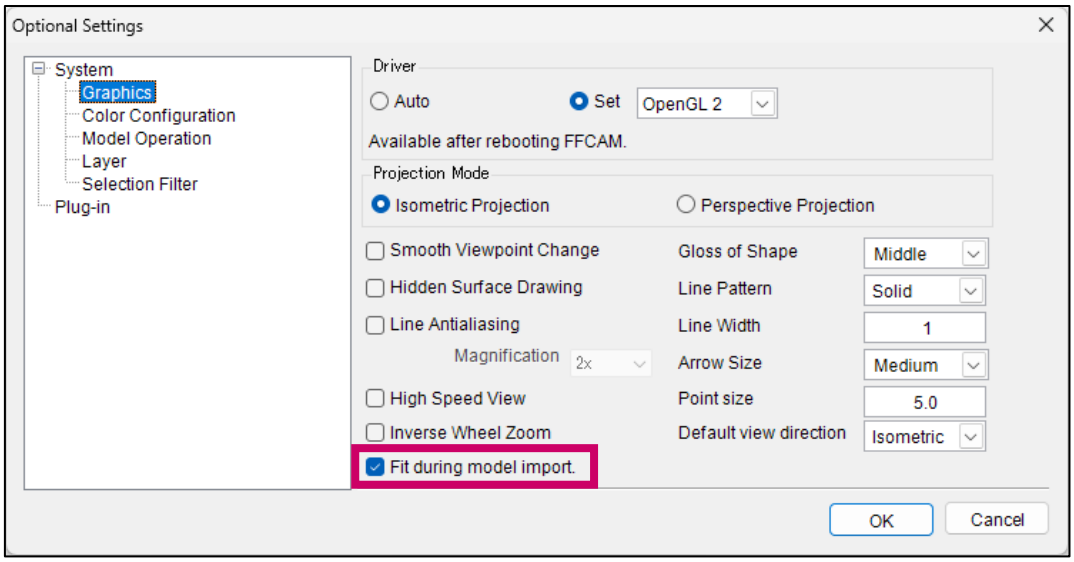
In previous versions of FFCAM, when a model was imported, the view was adjusted to fit the screen (the entire model was displayed within the screen).
If a model was imported while the screen was zoomed in or work was in progress, the view would reset to fit mode, which could interrupt the task.

In FFCAM 2025, a function has been added that allows you to set whether the view fits automatically during model import, providing the ability to select your preferred view method.

■ Setting Screen

Set the initial view when importing models from the [Optional Settings] screen.

[Optional Settings] screen
From [System]/[Graphics], specify the initial view using [Fit during model import.].



Checkbox: Selected (FFCAM initial value)

When a model is imported, the view is adjusted to fit the screen.

Checkbox: Not selected

When a model is imported, the view is not changed.

23. Enabled Changing the Initial View Direction

In previous versions of FFCAM, when a model was first displayed, the view direction was always "Isometric".

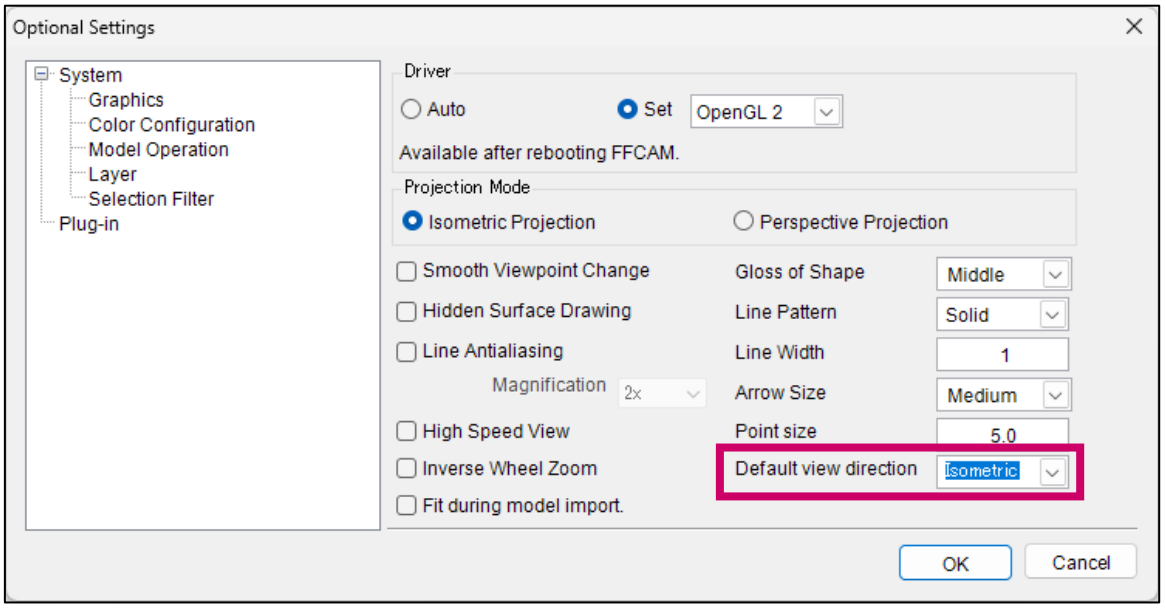
In FFCAM 2025, a function has been added to set the initial view direction, allowing you to select the view direction when models are displayed for the first time.

■ Setting Screen

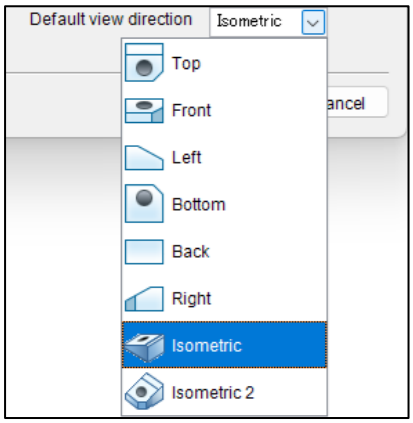
Set the initial view direction from the [Optional Settings] screen.

[Optional Settings] screen

From [System]/[Graphics], specify the initial view direction using [Default view direction].



Select from the following directions.



24. Added a Mode Icon Display to the Batch Calculation Schedule List

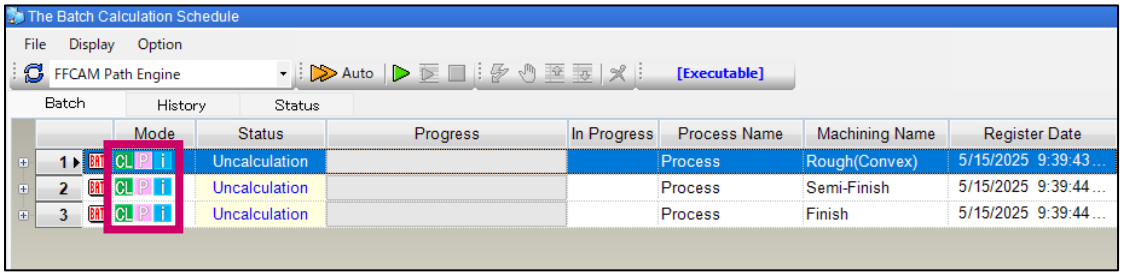
In [The Batch Calculation Schedule] list, "Mode icons" are now displayed to clearly indicate the type of process applied to each data item.

A mode icon indicating the processing details of each data item is displayed on the [Batch] and [History] lists of [The Batch Calculation Schedule].

■ Setting Screen

On the [Batch] and [History] lists of [The Batch Calculation Schedule], icons indicating the processing details of each data item are displayed in the [Mode] column.

Display example) [Batch] list



■ Mode Icons

See the following table for the processing details indicated by each icon.

Mode Icons	Processing Details
CL	CL calculation
3x	3-axis calculation (FFEngine)
5x	5-axis calculation (FF5Engine)
P	Post-processor output (FFpost)
i	Create machining information (FFpost - CreateInformation)
CSV	Output CSV
	Copy file (CL, NC, INF)
CA	Use Collision Check Assist

■ Supplementary Note

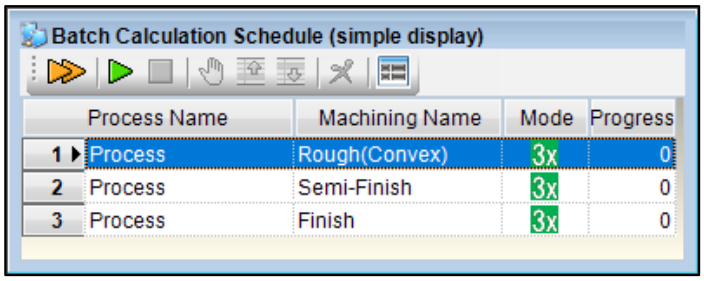
- The [Mode] column that appeared in previous lists has been renamed to the [In Progress] column.
  The displayed contents have not changed.

25. Added a Simple Display Function to the Batch Calculation Schedule

In previous versions of FFCAM, the display area of [The Batch Calculation Schedule] list was large, making it inconvenient to check.
In FFCAM 2025, a function has been added to switch [The Batch Calculation Schedule] list to a smaller, simplified display. The simple display list show only the most necessary information, such as batch calculation progress.

[Batch Calculation Schedule] list in simple display mode

You can check the progress of basic operations and each calculation on a small screen.

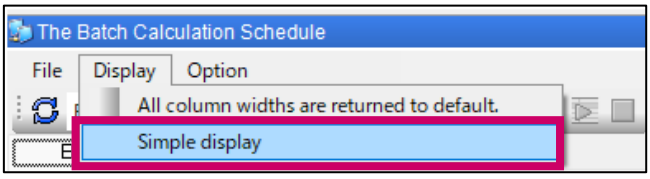


■ Setting Screen

[Batch Calculation Schedule] can be switched to a simple display in two ways.

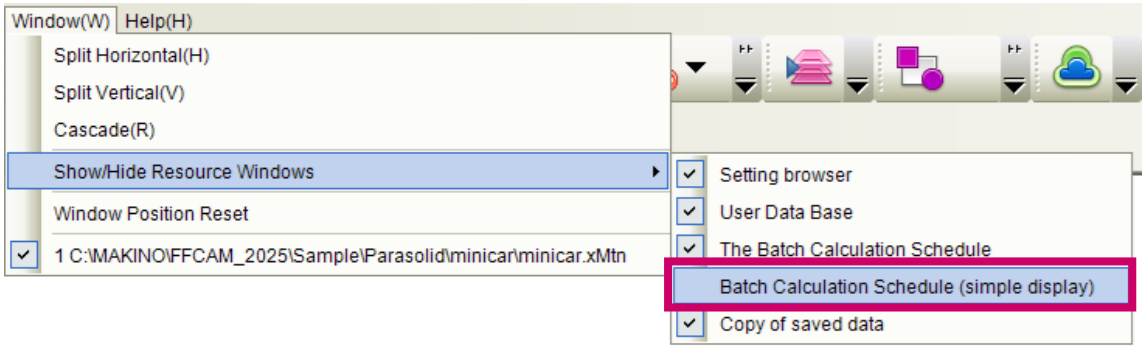
[The Batch Calculation Schedule] screen

In the [Display] menu, click [Simple display].



[Main] screen

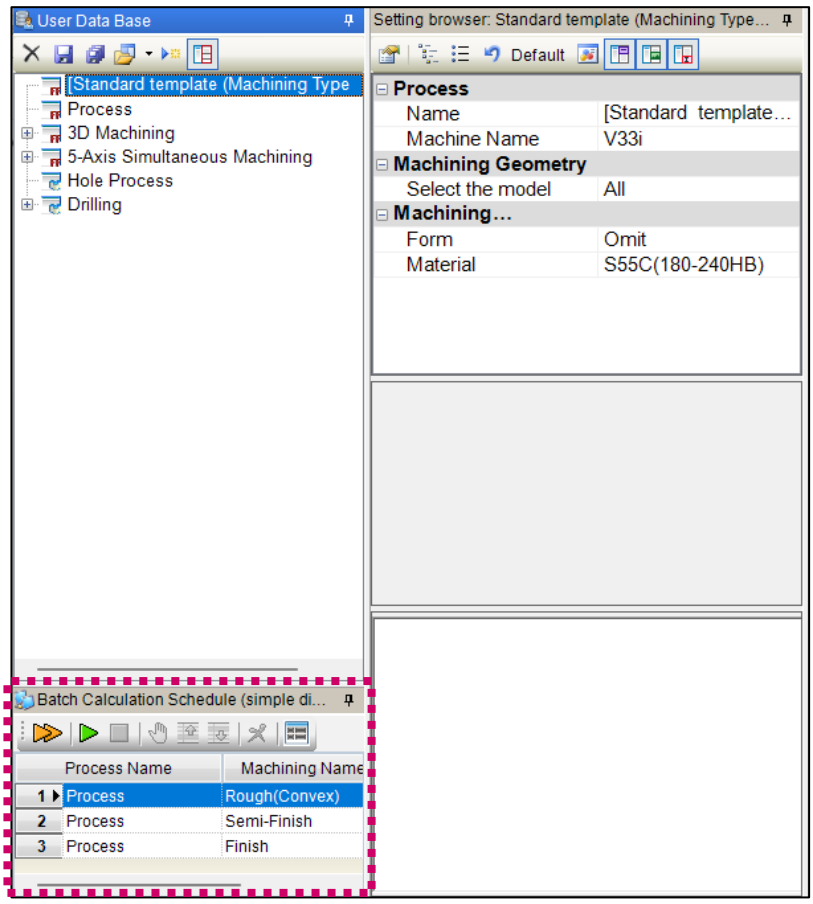
From the [Window] menu -> [Show/Hide Resource Windows], select [Batch Calculation Schedule (simple display)].



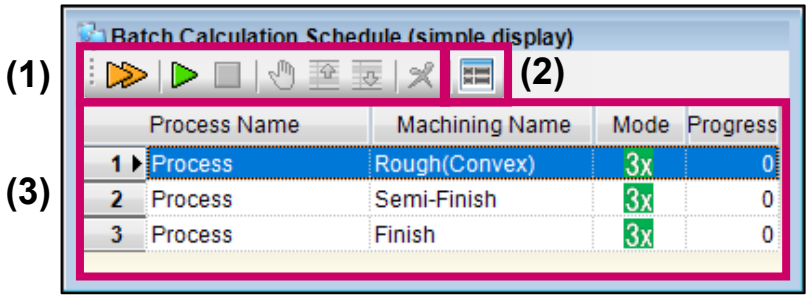
* To return to the normal display, select [The Batch Calculation Schedule].
You can alternate between [Batch Calculation Schedule (simple display)] and [The Batch Calculation Schedule]. When you select one, the checkmark is removed from the other.

■ Initial Display

The [Batch Calculation Schedule (simple display)] list is initially displayed in a tile layout below the left pane of the screen. (The example shows the user database.)



■ Screen Explanation



(1) Tool bar

The tool bar basically has the same functions as in the normal display, but the available tools are limited.

(2) Normal display button

Switch from simple display to normal display.

(3) List

The machining data registered for batch calculations is displayed line by line.
The lines cannot be expanded.
Here, you can check the machining name, process name, mode, and calculation progress for each machining data item.

* For information on the modes, see the previous chapter "Added a Mode Icon Display to the Batch Calculation Schedule List".

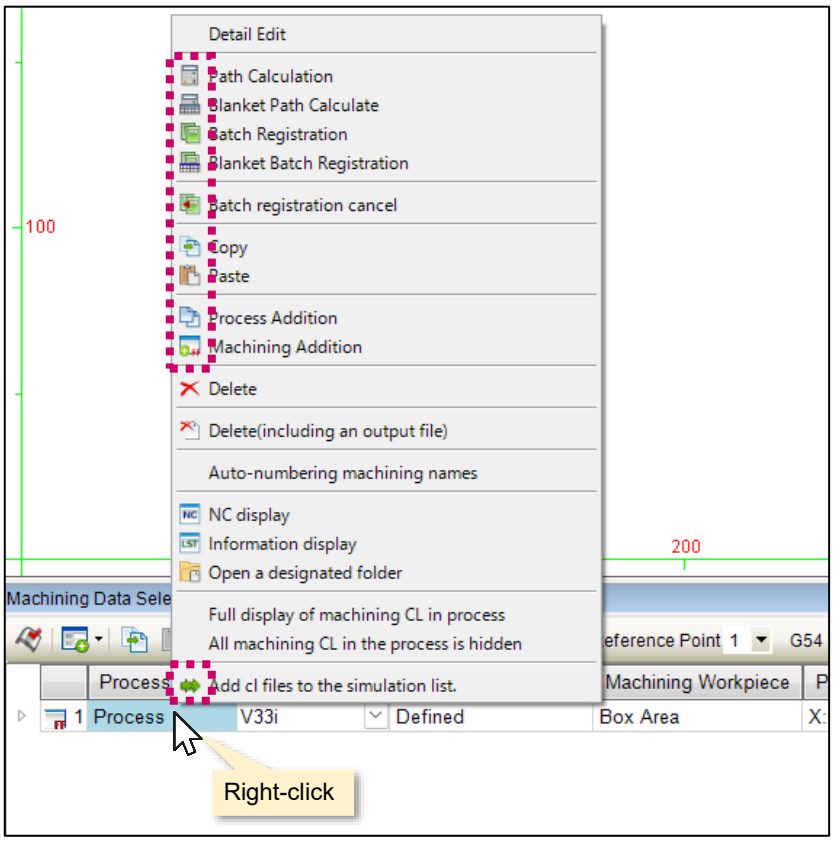
26. Improved Display of Machining Data Selection and Right-Click Menu

The menu displayed when you right-click on the [Machining Data Selection] list has been improved.
The right-click menu now shows the same icons that are displayed in the command bar, making it easier to find the desired function.

■ Setting Screen

[Machining Data Selection] screen

The right-click menu now shows the same icons that are displayed in the command bar.
Screen example) Right-click menu for the 3D machining process row



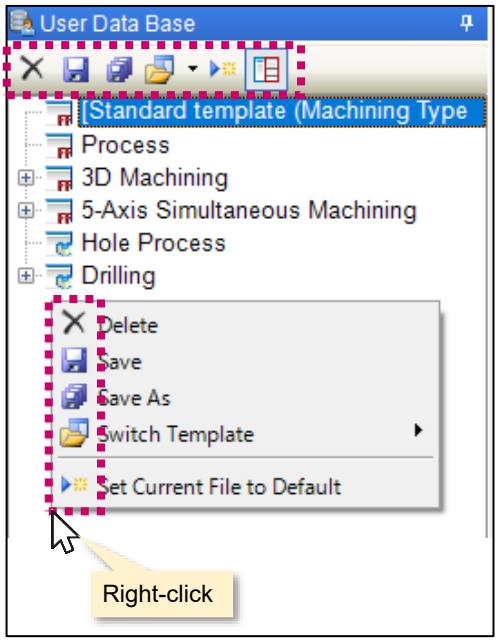
27. Improved Display of User Database and Right-Click Menu

The menu displayed when you right-click on the template row at the top of the [User Database] list has been improved.
The right-click menu now shows the same icons that are displayed in the user database tool bar.
Also, [Switch Template] can now be selected from a drop-down menu.

■ Setting Screen

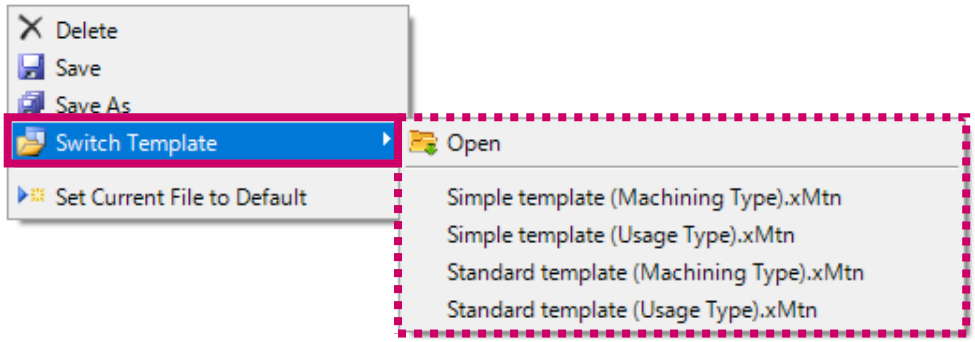
[User Database] screen

The right-click menu now shows the same icons that are displayed in the tool bar.



[Switch Template] can be selected from a drop-down menu.

* The pull-down menu is the same as the [Switch Template] menu on the tool bar.



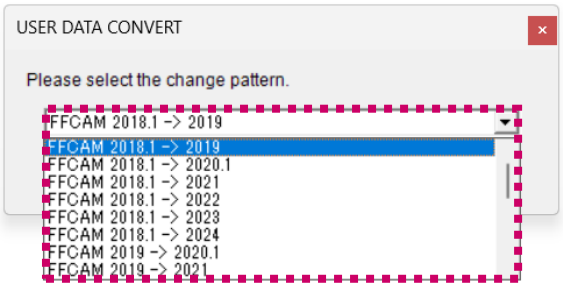
28. Improved the User Data Conversion Function

On a computer with multiple versions of FFCAM installed, the order in which conversion patterns are displayed when using the User Data Conversion function (FFcam.Userdata.Convert.exe) has been changed.
Conversion patterns are now displayed in order, starting with the latest version.

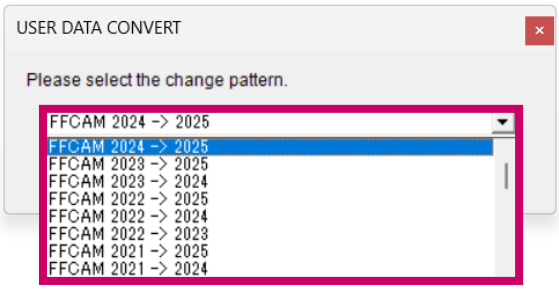
■ Setting Screen

[User Data Conversion] screen

Previous FFCAM

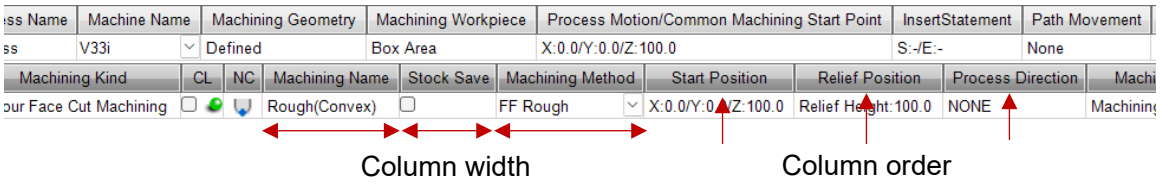


FFCAM 2025



29. Improved the Data Migration Tool

The data migration tool has been updated to support FFCAM2025.
With this update, a function has been added that allows the column widths and column orders in the [Machining Data Selection] list, which were manually modified by the user in a previous version, to be migrated and reproduced by FFCAM 2025.

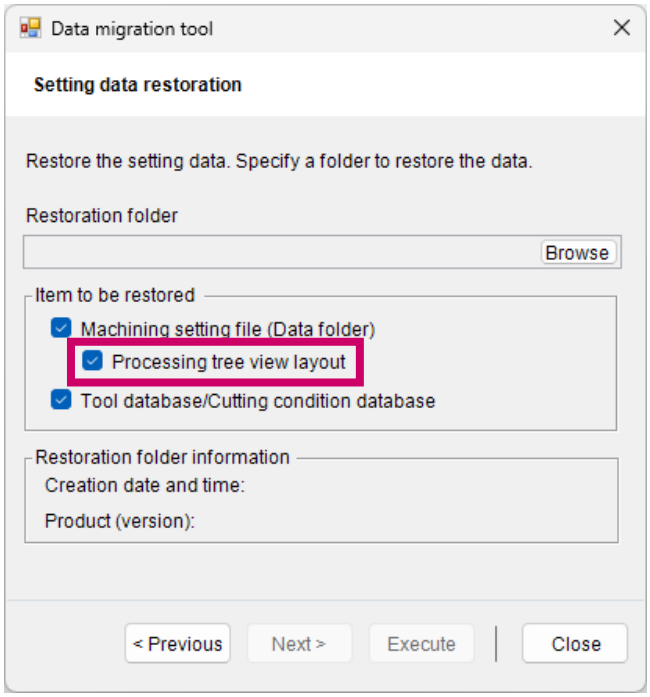


■ Setting Screen

In [Data migration tool], under [Item to be restored]/[Machining setting file (Data folder)], a checkbox for [Processing tree view layout] has been added.

[Data migration tool] screen

When the [Processing tree view layout] item is selected and data is restored, the column widths and column orders in the [Machining Data Selection] list, which were modified in a previous version, will be reproduced on the FFCAM 2025 screen.



■ Supplementary Note

- If there are differences in the [Machining Data Selection] list items between the recovery source version and FFCAM 2025 (such as functions added through a version upgrade), the columns for newly added items are displayed at the end (right edge) of the listed columns.

30. Enhanced Machining Parameter Macro Function

Three new functions have been added to the machining parameter macros.

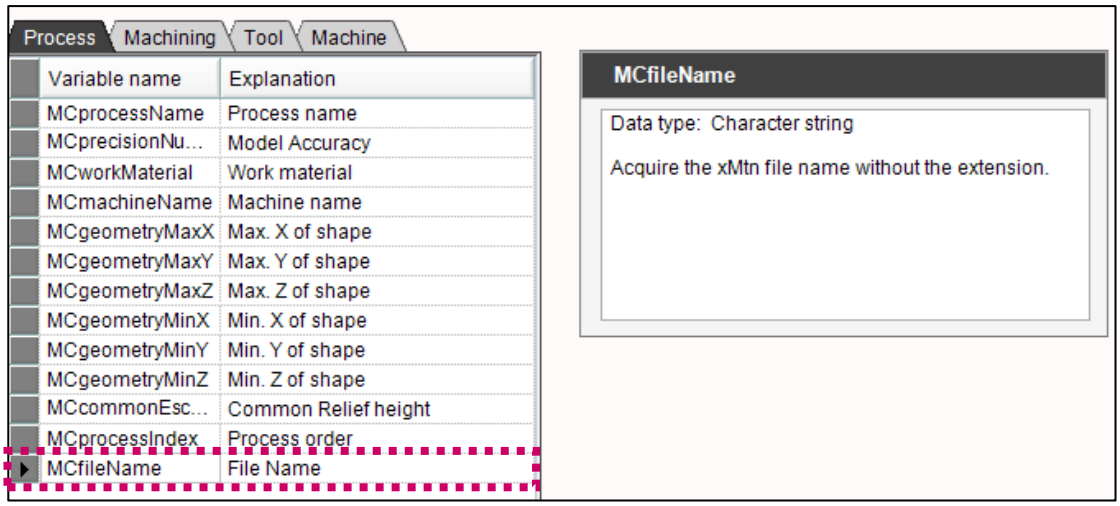
- Added a variable for outputting file names to custom macros (encrypted macro).
- Added support for the custom macro message output macro in drilling operations.
- Added a macro for setting the process start and end points to the process parameter macros.

■ Custom Macros

Added a variable for outputting file names to custom macros.

Added a variable for defining file names to the [Process] tab of the [Custom macro edit] and [Encrypted macro edit] screens.

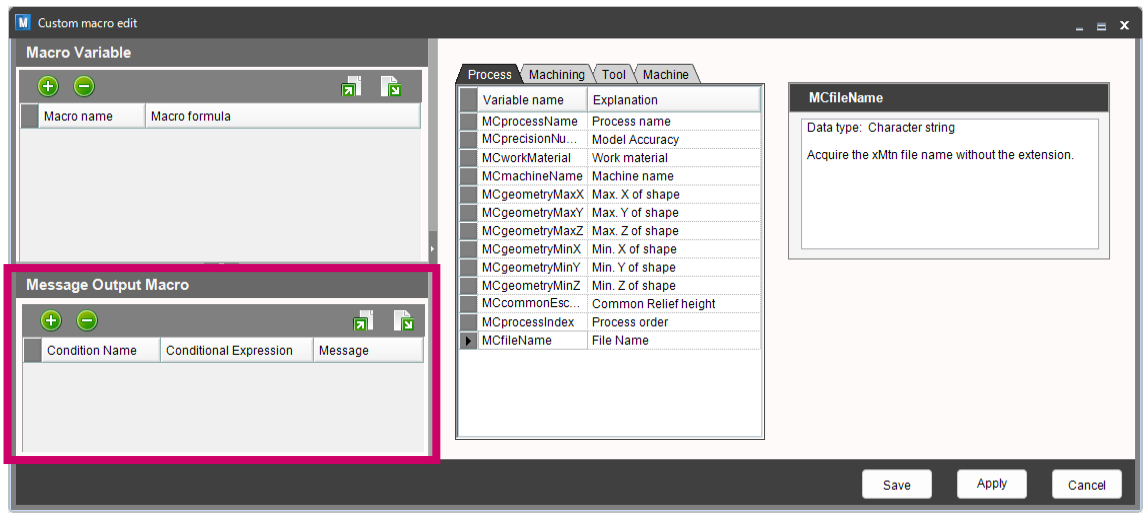
[Process] tab



Variable Name	Explanation	Remarks
MCfileName	File name	Data type: Character string Acquire the xMtn file name without the extension.

The macro check and message output functions can now be used with drilling processes and drilling.
Message output macros for drilling processes and drilling can be set from the [Message Output Macro] panel.

[Message Output Macro] panel



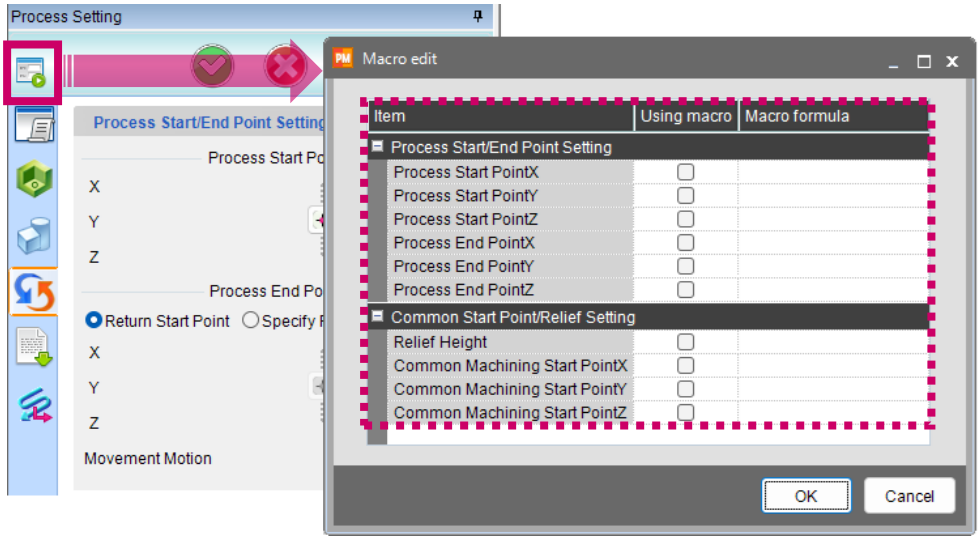
* The contents displayed on the screen have not changed.

■ Process Parameter Macros

On the [Process Start/End Point Setting] screen of [Process Setting], a parameter macro for the process start/end point setting can now be set.

[Process Start/End Point Setting] screen -> [Macro edit] screen

Open the [Macro edit] screen on the [Process Start/End Point Setting] screen, and set the macro.



See to the following table for the parameters that correspond to the macro settings.

Process Setting (Milling)		
Settings Tab	Target	Parameter Name
Process Start/End Point/ Relief Setting	Process Start/End Point Setting	Process start point X
		Process start point Y
		Process start point Z
		Process end point X
		Process end point Y
		Process end point Z
	Common Start Point Setting	Relief height
		Common machining start point X
		Common machining start point Y
		Common machining start point Z

Process Setting (Drilling)		
Settings Tab	Target	Parameter Name
Process Start/End Point/ Relief Setting	Process Start/End Point Setting	Process start point X
		Process start point Y
		Process start point Z
		Process end point X
		Process end point Y
		Process end point Z
	Common Start Point Setting	Common machining start point X
		Common machining start point Y
		Common machining start point Z

31. Improved Output to Include Collision Safeguard Data Before Machining Data Calculation

Collision safeguard data can now be output even before machining data is calculated.

In previous versions of FFCAM, collision safeguard data could not be output until the machining data had been calculated.
In FFCAM 2025, collision safeguard data can now be output even when the machining data has not been calculated or has been re-edited after calculation.

■ Setting Screen

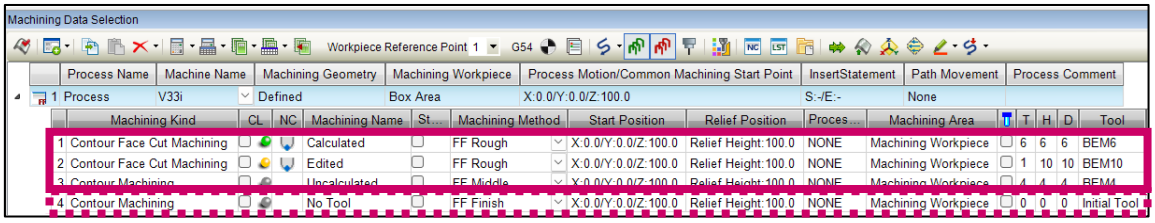
Output collision safeguard data from [Professional data output].

In previous versions of FFCAM, data output was possible only when the path calculation for the machining data was complete (CL icon is green). In FFCAM 2025, however, data can be output even when the machining data is uncalculated (CL icon is gray) or has been edited after calculation (CL icon is yellow).

However, data cannot be output if no tool is set or if the tool set for the machining data is invalid.

[Machining Data Selection] screen

Setting example)



Collision safeguard data can be output for machining data 1 to 3.
Collision safeguard data cannot be output for machining data 4 because a tool is not set.

■ Note

- After collision safeguard data is output, if the machining definitions are edited, be sure to check for any discrepancies between the output data and the current machining definitions. If there are any discrepancies, an inaccurate assessment of the situation based on the collision safeguard may lead to interference.
- Data can be output from the professional data output function to the collision safeguard and work plan. This function only supports output to the collision safeguard. As in previous versions of FFCAM, output to the work plan is limited to machining data with completed path calculations (CL icon is green).

32. [Vericut Interface] Changed Specifications of Tool Information for TLS Files

The specifications of TLS files (tool setting files) that are output from the Vericut Interface (optional software) on FFCAM have been changed.

Due to specification changes in Vericut 9.3, only one cutter component can be registered to each tool. Therefore, even a cutter used in combination with a shank must be registered as one component.

Consequently, the cutter shape of TLS files output from FFCAM are all output as an outline of a rotating body.

■ Supplementary Note

- The output function of the collision safeguard interface (optional software) uses the same specifications.

33. [Machine Simulator] Improved Marker Display

The size of the marker displayed during interference display or position changes in the machine simulator has been changed to a fixed size on the screen.

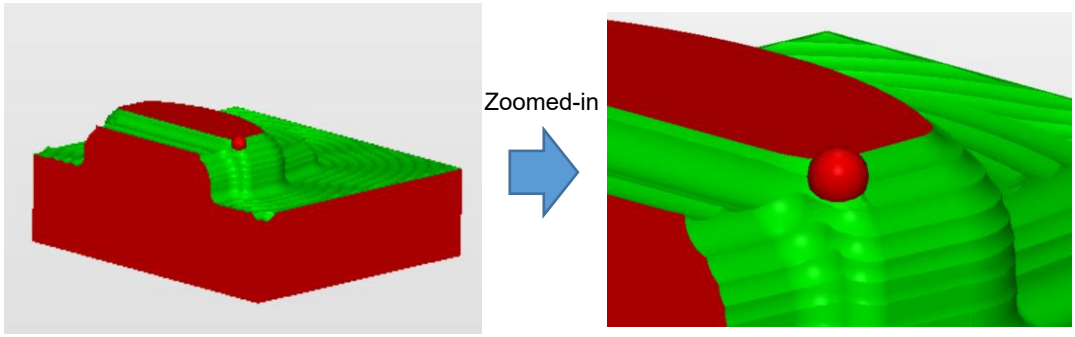
In previous versions of FFCAM, the size of the marker was fixed relative to the model. As a result, zooming in on the screen would enlarge the marker as well, sometimes obstructing precise position checks.
In FFCAM 2025, the size of the marker is fixed relative to the screen, so even when zoomed in, the marker itself does not enlarge.

■ Display Example

Example) Interference point marker during simulation

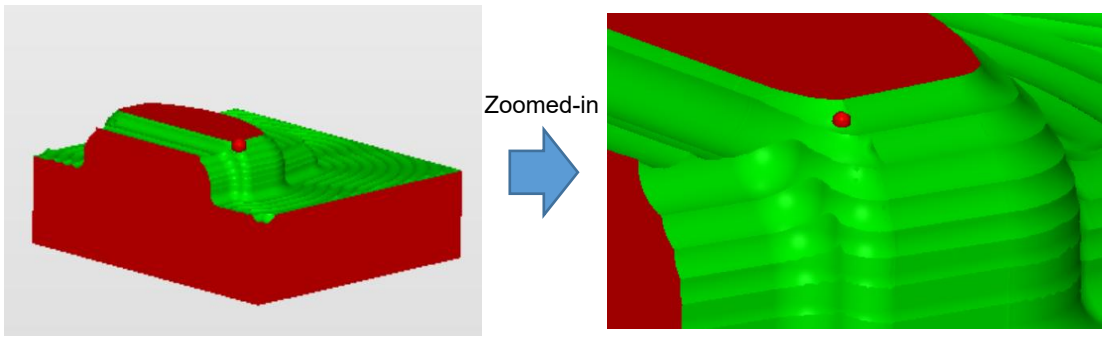
Previous FFCAM

When the screen is zoomed in, the marker also enlarges.



FFCAM 2025

When the screen is zoomed in, the size of the marker does not change.

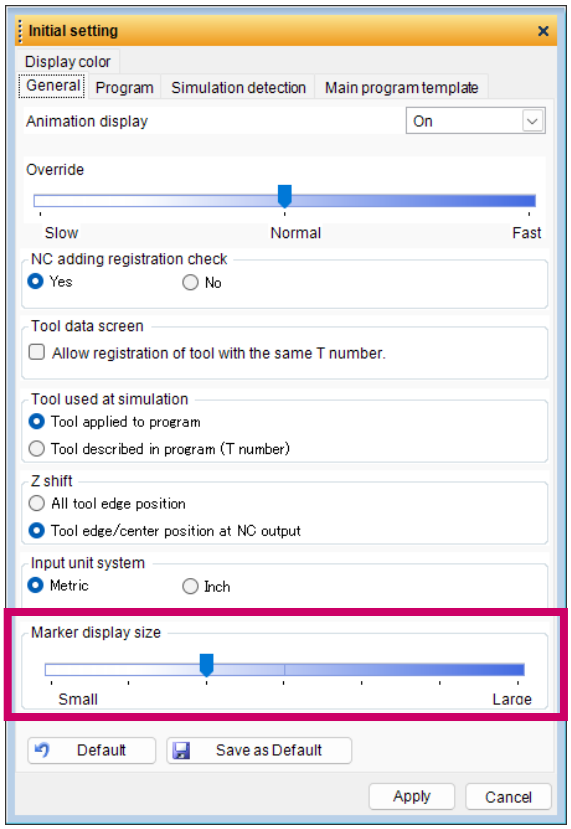


■ Setting Screen

Set the marker display size from the [Initial setting] screen/[General] tab of the machine simulator.

[Initial setting] screen/[General] tab

Set the marker display size using the [Marker display size] slider bar.



■ Functions Where Display Size of Marker Is Applied

The marker display size setting is applied to the following eight functions.

1. Interference point marker when simulation stops due to interference
2. Marker, selection marker, and interference marker when specifying points in [Manual operation]
3. Marker in [Reference point]
4. Marker and selection marker when specifying points in [Work offset]
5. Marker and selection marker during [Parallel move] with [Positioning]
6. Marker and selection marker during [Rotation] with [Positioning]
7. Marker and selection marker during [Axis alignment] with [Positioning]
8. Marker and selection marker when specifying points in [Face alignment] in [Positioning]

34. [Machine Simulator] Same Tool Setting Recognized for Both 3D Machining and Drilling

In previous versions of FFCAM, if data using the same tool for both 3D machining and drilling was registered, a "tool number duplication error" would occur in the machine simulator, preventing the simulation from running.

In FFCAM 2025, the machine simulator function has been improved to allow simulation even when data using the same tool for both 3D machining and drilling is registered.

■ Compatible Tools

For the following tool types, even when data using the same tool for both 3D machining and drilling is registered, a tool number duplication error no longer occurs, allowing the simulation to be executed.

- Ball
- Flat
- Bull Nose
- TA Ball
- Round Insert FM