

64bit version

FFCAM 2022

Release Notes



Promise of Performance

Copyright © 2022 MAKINO MILLING MACHINE CO., LTD. All rights reserved.

Introduction

This manual is described about the release note of Makino Milling Machine product FFCAM.

Please read it through before using FFCAM.

Caution

- (1) No part of this manual may be reprinted without the written permission of Makino Milling Machine Co., Ltd.
- (2) No part of this manual may be reproduced/re-edited without the authority of the copyright holder. Distributing reproductions is an infringement of the copyright holder's rights.
- (3) Every effort has been made to ensure the accuracy of all the information in this manual. However, if you notice that any is part unclear, incorrect, or omitted in the manual, please contact your Makino sales representative.
- (4) Makino Milling Machine Co., Ltd. assumes no liability to any party for any damage caused by using this product or manual.
- (5) Information confirmed upon creation of this manual, such as machine models and software versions, is shown in the manual. Ask your Makino sales representative for the latest information after the creation of this manual.
- (6) The specifications of this product and the contents of this manual may be revised for improvement of the product without prior notice.
- (7) Screens used in this manual:
 - Screens shown in this manual may vary depending on the model.
 - Screens shown in this manual may slightly differ from the actual display.

<<Contents>>

1. FFCAM Release Notes 1

 1.1 Restrictions on Functions..... 1

 1.1.1 CAM function..... 1

 1.2 Function Revision (Addition/Change) History..... 3

 1.3 Correction of Problems 5

1. FFCAM Release Notes

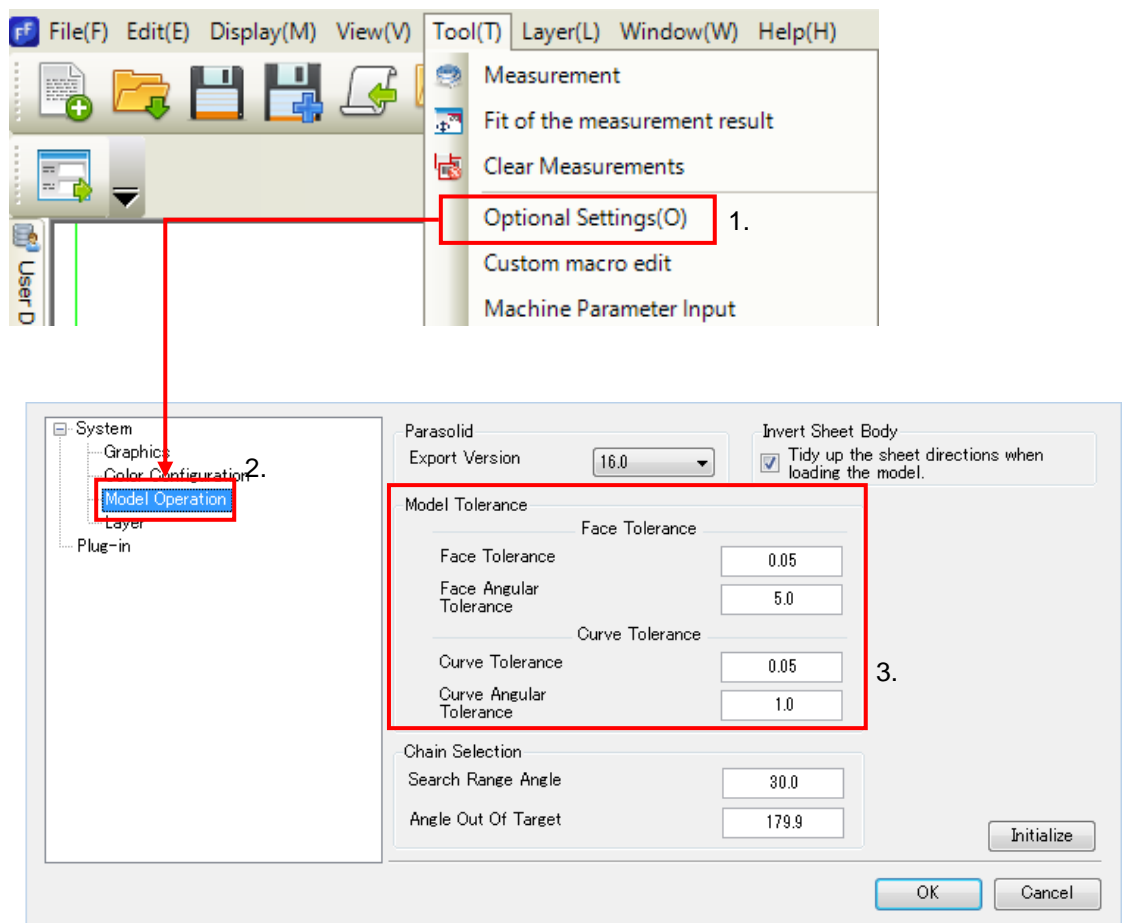
1.1 Restrictions on Functions

1.1.1 CAM function

(1) An error may occur during importing the model data by the graphic driver, and then the FFCAM operation is disabled. In such a case, increase the “Model Tolerance” value of the system parameters.

- The recommended tolerance value can be set by pressing the “Initialize” button.
- The recommended tolerance value for a Face and a Curve is 0.05 respectively. Depending on the model size and personal computer performance, and a bigger value may be input.

Setting Procedure:



- (2) When "Helical" is specified for Tilt Infeed Motion in machining with a round insert face mill, a toolpath may be generated with collision between the bottom of the face mill and the workpiece even if the function to "Avoid cutting by the bottom of Round Insert cutter" is selected.

For this reason, set "Along-route" at Tilt Infeed Motion when using a round insert face mill.

When "Helical" needs to be set for Tilt Infeed Motion, confirm that the following conditions are satisfied:

- Helical Radius > bottom diameter of the round insert face mill
- Helical Infeed Tilt < Max cut angle of the round insert face mill

- (3) When toolpath is calculated after moving or copying the machining model, the wrong toolpath will be calculated. The reason is that the previous intermediate data is used. After moving or copying the machining model, be sure to re-register the machining model before calculating toolpath.

- (4) The output file name, generated with batch and having many machining, is limited to 240 single byte characters.

If it exceeds the limit, the following message is displayed.



Please enter up to 240 one-byte characters for path of the output file.
(Currently specified path : 396 one-byte characters)

- (5) Over four machining definition files cannot be opened at the same time.

1.2 Function Revision (Addition/Change) History

FFCAM 2022.0.0

Machining functions

DUG No.	Description
2015-2032 2015-2495 2018-0469 2018-0913	In the high function mode of the contour projection machining, the function which does not output the projection path on the horizontal surface was added.

Operation functions

DUG No.	Description
2018-1017	When a curve or a curved surface is specified in the area setting, the function which can use the machining area specified in other machining was added.
2017-0949	When the process and machining data in the tree view is changed, adjusting the name of the loading stock file became possible.
2017-1568 2017-1587 2017-1981 2021-0629	It was changed to display the estimated machining time on the machining line and the hole machining line on the tree view.
2021-0334	The function to draw the tool path in a batch operation was added for the process and machining on the tree view.
2021-0260 2021-0930	When the Calculate tool path button is clicked and when machining is copied from the user database, the message box opens. It was changed so the message box can be hidden by the screen operation.
2021-0632 2021-0633	The min. detect width of the machining remain processing function was changed so it corresponds to the variable of the macro function.
2021-0565	In the tool axis control function of the 5-axis simultaneous movement, when the reference axis is specified in a discretionary direction, displaying the direction on the Machining parameter screen became possible.
2021-0624	The animation speed of the CL simulation was changed from seven levels to nine levels.
2021-1257	It was changed that the simulation is not executed when the machining work data is invalid in the CL simulation. In a such case, the condition is notified in the message box.

2020-0095	It was changed so the work ref point value in the repost function can be kept. The function to set the color freely was added for the Color setting screen in the measuring function and the simulation function.
2019-0363	In the "Geometry Maximum & Minimum" function of measurements, it was changed so the geometry size of the measurement result is displayed.
2021-0666	The function to specify the color RGB with real numbers from 0 to 1 was added.
-	The access menu to the FFCAM product information web page was added to the FFCAM menu bar "Help."
-	For the case which FFCAM ends due to an unexpected PC condition, a function to avoid damaging the FFCAM operation data was added. It backs up the currently opened operation data in FFCAM in a specified interval.
-	A new machine was added to the machine parameter.
2018-0387 2020-1711 2021-0596 2021-1028	It was changed so the parameter "NC file extender" of the machine parameter can be specified for each machine.
2018-1184	It was changed so the parameter "Safety mode at hole ATC" of the machine parameter can be specified for each machine.
2017-2456 2021-0552 2021-1511	A function was added so discretionary files saved in the FFCAM Data folder can be restored by using the Data migration tool.

Post Processor

DUG No.	Description
-	In the command list in the "NC output" tab on the Conversion table edit screen and the list of macros in the "Macro" tab, the specification of the pasting operation was changed. The pasting position was changed based on the function to be pasted.
-	In the command list of "NC output" and the list of macros in the "Macro" tab on the Conversion table edit screen, "Move up" and "Move down" were added to the right-click menu.
-	In the "NC output" and "Macro" tabs on the Conversion table edit screen, the shortcut keys for copying and pasting were added.
-	In the output parameter list on the Conversion table edit screen, a method to replace the string was added.
-	In the list of commands on the "NC output" tab on the Conversion table edit screen, the display color for the pasting operation was changed.

2020-0273	In the list of commands in the "NC output" tab on the Conversion table edit screen, it was changed so the name of added command can be changed.
-----------	---

1.3 Correction of Problems

FFCAM 2022.0.0

Machining function

DUG No.	Description
2020-0842	Tool collisions occur in contour machining. This problem was corrected.
2020-1628	In the model with many bosses, the calculation time for the Contour face cut machining was long. This problem was corrected.
2021-0494	In the Machining Remain processing function of the Flat Face Machining, if "Tilt infeed out of area" is validated while tilt infeed is "Omit", invalid path is output. This problem was corrected.
-	When the Profile approach operation type is "Tangent" in the Flat face machining, if the Cut start point is specified to "Approach position", invalid path is output. This problem was corrected.

Operation functions

DUG No.	Description
-	When the tool data is loaded from the tool DB, the same value is set to the Tip angle and the Gradient angle. This problem was corrected.
-	When the parameter "Previous Machining Tool" of the corner R machining is specified by a macro, the parameter check is not conducted for the macro in the Calculate tool path. This problem was corrected.
2021-0533	When the macro is specified to the parameter "Spindle speed in Z step" in the machining condition setting, an exception occurs when the machining is tried to be opened on the tree view. This problem was corrected.
2021-0725	When the machining work format of the machining data in the user database is specified to cylinder, and when such machining data is copied to the tree view, the machining workpiece format of the data is changed to the box. This problem was corrected.
2021-1588	When the template file in the user database is damaged, the FFCAM cannot be started. This problem was corrected.

-	After copying the hole information line on the tree view, the color of the source line is not reflected to the target of copying. This problem was corrected.
2021-1804	In the simple interference check function in the FFCAM tool drawing, the drawing without interference is output even when the tool is interfering the product geometry. This problem was corrected.
-	In the chain select function on the Select screen, adjacent curves cannot be acquired if minute curves are continuing. This problem was corrected.
-	When the HASP key for the network is used, displaying the right-click menu on the FFCAM screen tree view is slow. This problem was corrected.
-	"Online help" in "Help (H)" on the FFCAM menu bar was changed to "Help".
2021-1457	The XY Direction Cut Width tool chip message of the contour face cut machining guide cutting was invalid. This problem was corrected.
2020-0845	For the R point of the drilling machining, the message displayed after the calculate tool path is in English. This problem was corrected.
2021-0163	In the message displayed after Calculate tool path, inappropriate message regarding the path approach was corrected.
2021-0405 2021-0683	During the screen operation of the CL display, unexceptional condition occurs. This problem was corrected.
2019-1745	When the simulation is conducted with the machine drawn in the CL simulation, the simulation speed becomes slow. This problem was corrected.
2020-0658 2020-1407 2021-0113 2021-0140	When the CL simulation is executed in the multi thread, the post process of the CL simulation is not executed correctly. FFCAM abnormally ended by this problem. This problem was corrected.
2021-0068 2021-0108	In the CL simulation, the pencil neck type cutter is used. When the cutter and the shank diameters are the same, interference of the shank cannot be detected correctly. This problem was corrected.
2020-0073	In the CL simulation, a bite to the product geometry occurs due to the tool tolerance value. This problem was corrected.
2017-1536	Due to the operational procedure of FFCAM, the setting file for the CL simulation is not output. This problem was corrected.
2020-1422 2021-0301 2021-0309 2021-0502	When "Output to one NC" is conducted in the repost, the machining is output in redundancy in the machining instruction. This problem was corrected.
2021-0220	While starting FFCAM, unexceptional condition occurs in reading of the layout file on the

	simulation repost screen. This problem was corrected.
-	The cutter blade length was not output in the FFCAM collision safe guard interface function; therefore, the tool interference could not be detected in the collision safe guard. This problem was corrected.
2016-0322 2021-1204	In the VERICUT interface function in FFCAM, the cutter blade length was not output; therefore, tool interference could not be detected in VERICUT. This problem was corrected.
-	In the Excel machining instruction output function, "Taper blade length" and "Taper angle" in the list of this setting screen are displayed redundantly. This problem was corrected.
-	In the output function of the Excel machining instruction sheet, "Head of NC-DATA" and "Tail of NC-DATA" are not displayed on the setting screen list.
-	If the file path of the Excel file specified on the FFCAM option screen is invalid, an unexceptional condition occurs when the Excel machining instruction sheet output function setting screen is started up. This problem was corrected.
2021-0283	While the parameter "Tool list" of the setting screen is validated in the Excel machining instruction sheet function, when the FFCAM repost function is executed, unexceptional condition occurs. This problem was corrected.
-	When specific screen operation is conducted in the calculate tool path parameter on the Option setting screen, the parameter "output file name" value becomes empty. This problem was corrected.
-	When FFCAMDB file path is different in the tool DB backup PC and the tool DB restoration PC, restoring the tool DB fails. This problem was corrected.
2021-0169	In the data migration tool, the display information of the main program NC button on the Repost screen was not restored. This problem was corrected.
-	FFCAM Chinese (simplified Chinese characters) version. In the specified character of the screen message, the characters were Japanese. This problem was corrected.
-	FFCAM German version. The labels on the Machining condition setting screen were not displayed properly. This problem was corrected.