MMTS / MMTM / MMTU
NC Code Generator For Thread Mill

NS TOOL CO.,LTD.

12 Jan. 2022

Ver. 2.1

Introduction

- This program is used to output sub-programs for female thread machining with the threading tools "MMTS", "MMTM" and "MMTU" manufactured by NS TOOL.

- Before using this program, please make sure to agree to the "NS Tool Software Terms of Use". You can check this agreement when you download this program, when you start the program for the first time, and during using the program.

- If you do not agree to the "NS TOOL Software Terms of Use", please stop using this program and delete all downloaded files.

Notes

- The operation of this program has been checked with a NC control unit manufactured by FANUC. Please check the G-code when using other control units.

- When using this program, please be sure to check the operation on the machine before actual machining.

- Use a machine that supports helical interpolation (simultaneous circular interpolation of X, Y, and Z axes) to process the program with arc output.

- Please note that we cannot be held responsible for any disadvantages or damages caused by the use of this program.

Notes on operation tests and operating environment

- Operation has been confirmed on Windows®10 (64bit version).
- "NC Code Generator For Thread Mill V2.1.exe" and "DAT" folder should be stored in the same directory hierarchy. Storing them in different locations will interfere with startup and operation.

How to use 1 of 3

- Double-click the "NC Code Generator For Thread Mill V2.1.exe"executable file to start the software.

- Please read the terms of use that will be displayed after startup, and click "I Agree" only if you agree to the terms. After that, the main screen will be displayed.
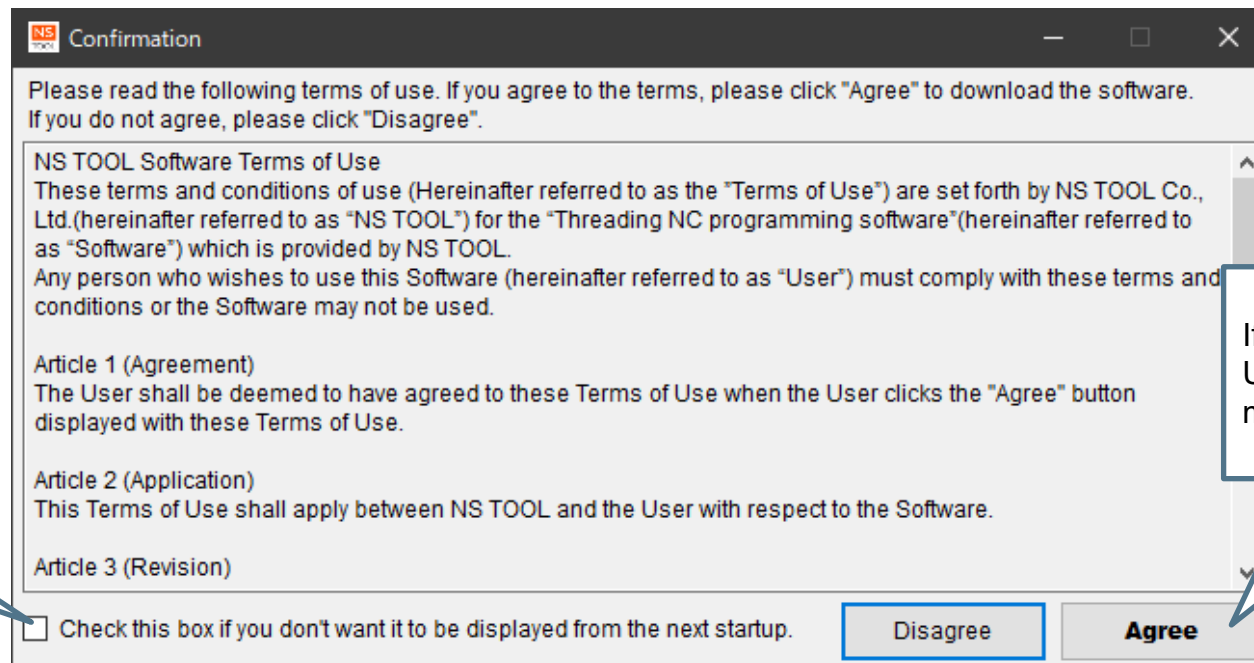
If you check here and click "Agree", this screen will be omitted from the next startup.

If you agree to the Terms of Use, please click here. The main screen will be displayed.

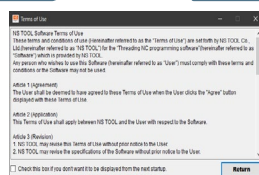1 Select "Tool Type" and "Thread Size" in the green area "Tool Shape and Work material".

2. After input the "Machining Thread Depth" in the same frame, click "Check Depth". Then, select "Work Material".
  *If you do not click "Check Depth", you will not be able to select "Work Material".
  *If a value greater than the effective length of each tool size is input in the "Machining Thread Depth" field, an error will occur.
  *The pitch cannot be changed.

3. In the orange area, it will be displayed that the reference cutting conditions (rotation speed, feed rate, number of times to cut, feed per tooth, and depth of cut).

4. ★ mark parameters can be changed depend on the preference. Change the necessary parameters and then click [Confirm]. If the machine does not support helical interpolation, uncheck "Arc output" in the lower left corner, and then click [Confirm].

  ※ Please note that the numerical values will not be reflected in the program unless you click "Confirm" after setting the cutting conditions.

  ※ The "Feed per tooth" and "Depth of Cut" will be automatically applied when you click [Confirm] after changing the necessary parts such as "Feed Rate" and "Number of times to Cut".

  ※ The "Helical R Compensation" is the correction value of the final helical R. If the screws are loose or tight after completion, adjust them here.

Click here to view the Terms of Use.

©2021 NS TOOL CO., LTD.

5. In the gray "NC output settings" frame, the output format, Feed and Output Program can be changed .

※ Please refer to page 8 of this manual for details on the initial position when NC output is performed with G91 (INC).

※ If you choose a sub-program, please prepare the main program by yourself.

6. Click [Calculation] in the blue frame to display the processing program.

This is where you can edit the displayed machining program. The results of editing here will be output without any error checking.
Please pay close attention to the editing process.

7. Click [Output] to output the NC data to the specified saving storage folder. Click [Exit] to exit this program.

©2021 NS TOOL CO., LTD.

Set the program number, tool number, and screw position.

The standard program number to be output is [O0001].

The standard tool number and tool length compensation number are [No. 1 (T1, H1)].

The position of the screw to be machined uses the XYZ coordinates registered in
[G54 (workpiece coordinate system)] as the origin as standard.

X / Y --- Screw hole center position

Z --- Top surface position of screw holes

If you want to change these, please edit the program directly that appears after clicking [Calculate].

The results of editing here will be output without any error checking.
Please pay close attention to the editing process.

Initial position when outputting G91 (INC)

The initial position when NC is output by G91 (INC) is set to (X, Y, Z) = (0, 0, 50).

When NC output is performed by G91 (INC) as a subprogram and continuous machining is performed, Please be sure to add a code to the main program to return to the initial position after each hole is machined.

The tool tip position at the end of the subprogram is (X,Y,Z) = (0,0,2.5).

Initial position when NC is output by G91 (INC)
※ X0.Y0.Z50 for the G54 coordinate system.

G54 origin position

50mm
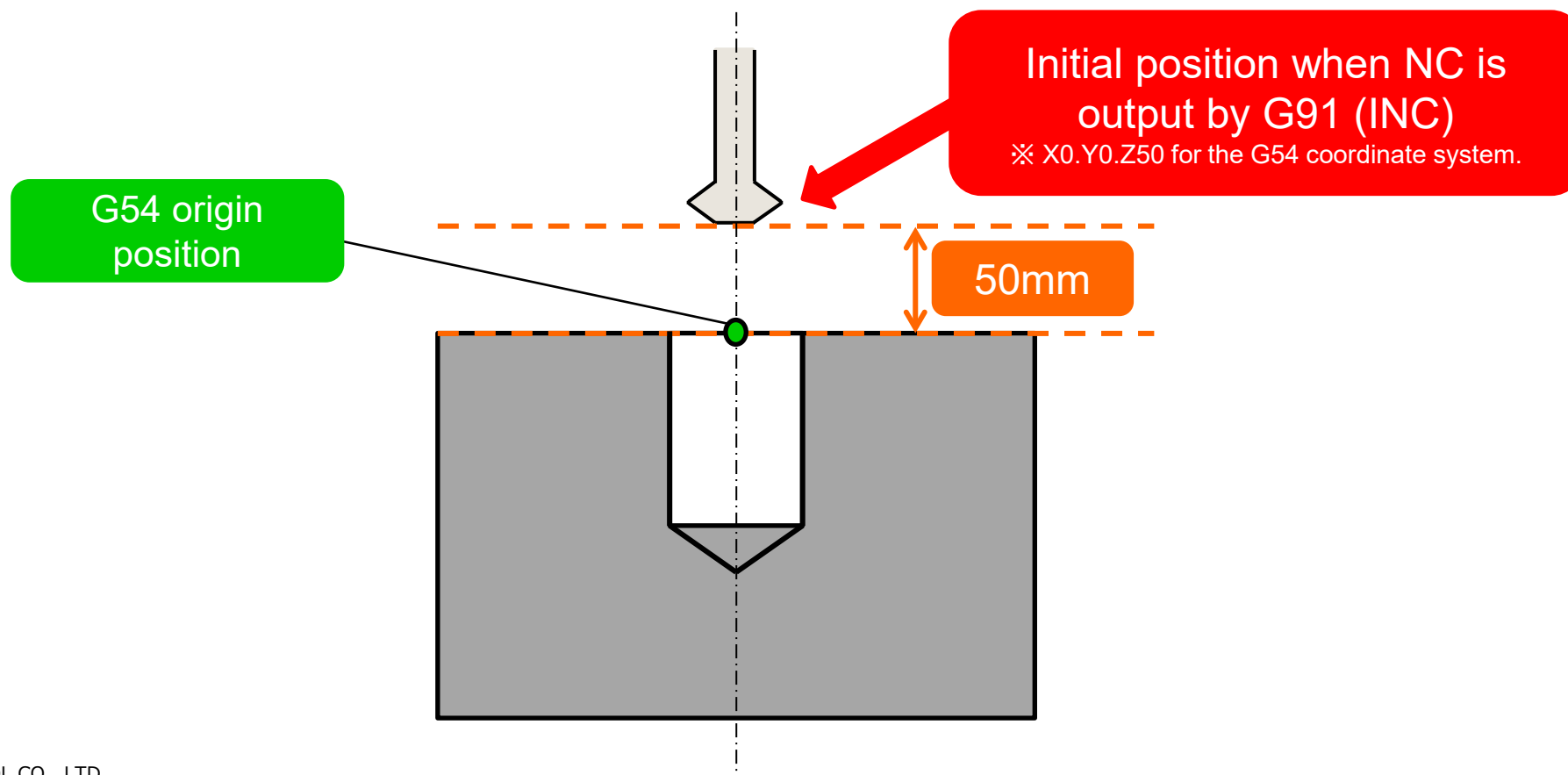
Arc output

If NC output is performed using G91 (INC) without arc output, the machining position may be shifted due to relative coordinate position error, and the screw may not be inserted.

If you want to create a program without arc output, we recommend that using G90 (ABS) for NC output.

If you are using a machine that supports helical interpolation, to check the box of the "Arc output" and output NC is recommended.

When the error message window appears

If you enter a value that exceeds the range we expect for the parameters that need to be entered, or if you click the button without entering a value, the following message may be displayed.
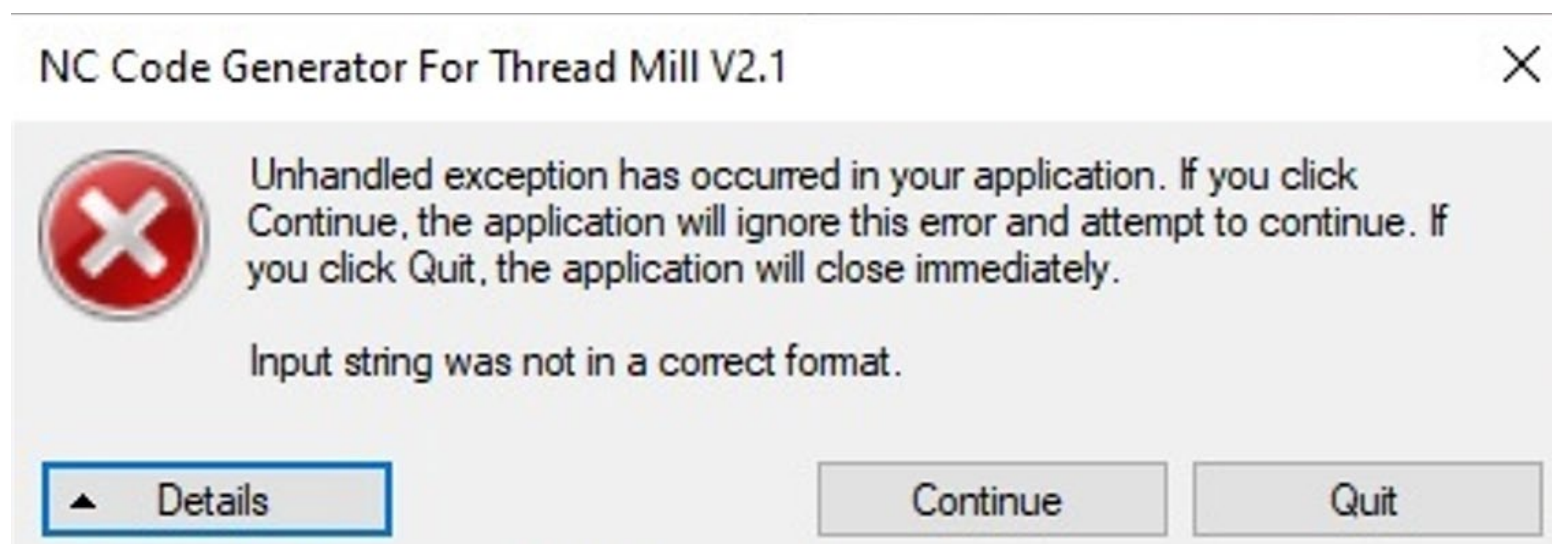
If the message appears, please click [Exit (Q)] and restart the threading program.