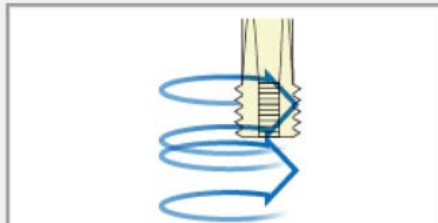# Thread Milling Programmer Manual

Choose a path type

○ Continuous

◉ Axial divide (recommended)

If the threading length is long, move it to the tool 1 revolution (axial direction) for additional cutting.

Internal thread specifications

M14 L= 10.0 P= 1

Program number   O   0100

Tool length compensation number   H   30

Tool radius compensation function

◉ Use (G41 / G42)   Tool compensation function

○ Don't use   D   30

Reference process time   4' 49" 92

Ref. large end hole size   --

Ref. hole size   13.09mm

Adjusting the finished diameter   +0.000 mm

NC program

```
%
O0100
(SIZE=M14 P=1.0 L=10.000)
(TOOL=MC-HLC 10204N100M)
G54 G00 X0 Y0 M08
G43 Z0.000    H30 S1110 M03
G90
G00 Z-10.963
G41 G01 X-0.000   Y5.808    D30 F86
G03 X0.000    Y-6.808   Z-10.500 J-6.308   F41
Z-9.500   J6.808
X3.154    Y-5.963   Z-9.423   J6.308
G40 G00 X0.000    Y0.000
Z-10.965
G41 G01 X-0.000   Y6.044    D30 F88
G03 X0.000    Y-7.044   Z-10.500 J-6.544   F45
Z-9.500   J7.044
X3.272    Y-6.167   Z-9.423   J6.544
```

☐ Edit code

How to use Thread Milling Programmer

Index

Introduction

・The NC program generated by this software does not always guarantee the accuracy of internal threads.

・Before using the NC program for actual machining, check the operation on a sample piece.

・Please note that we are not responsible for any issues caused by using this software or the generated NC program.

・This software works on Windows 10 and newer versions.

・Nominal size and pitches that can be selected in this program are in accordance with JIS standards. If you need a special diameter or pitch, please contact us from the URL or QR code below.

YAMAWA Customer Consultation

https://www.yamawa.com/en/support/consult.html

How to install on Windows 10

1. Download the installer from YAMAWA Website (https://www.yamawa.com/en/download/programmer.html)

2. Start the installer.

Thread Millig Programmer x.xx exe

3. In some cases, the folloing window shows up. If it doesn't, please go on to step No.5.
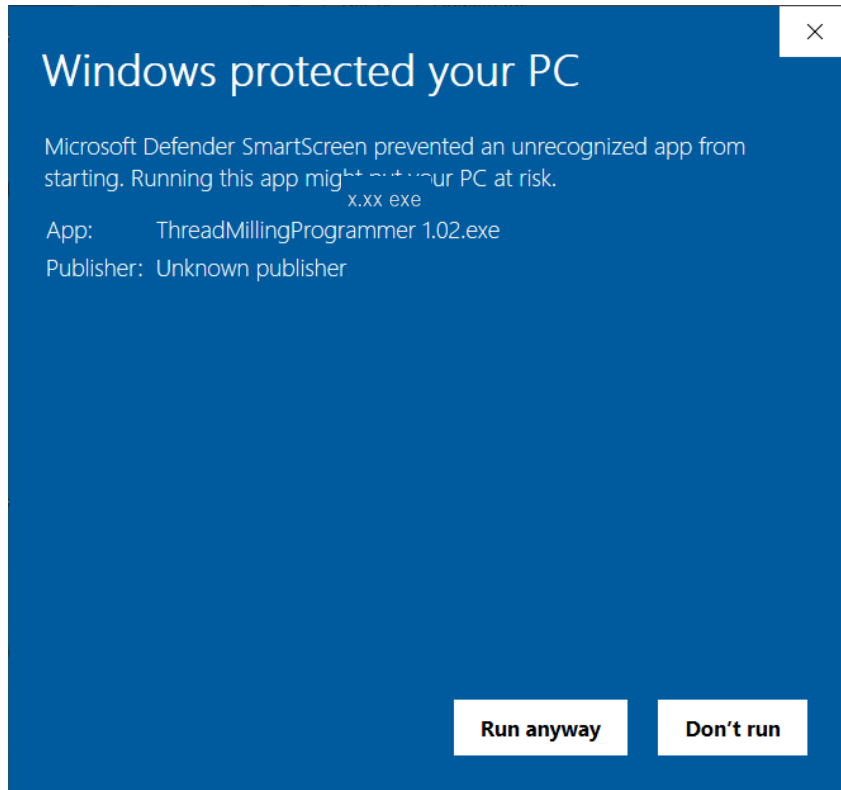   Click the "More info" on the window.

Windows protected your PC

Microsoft Defender SmartScreen prevented an unrecognized app from starting. Running this app might put your PC at risk.
More info

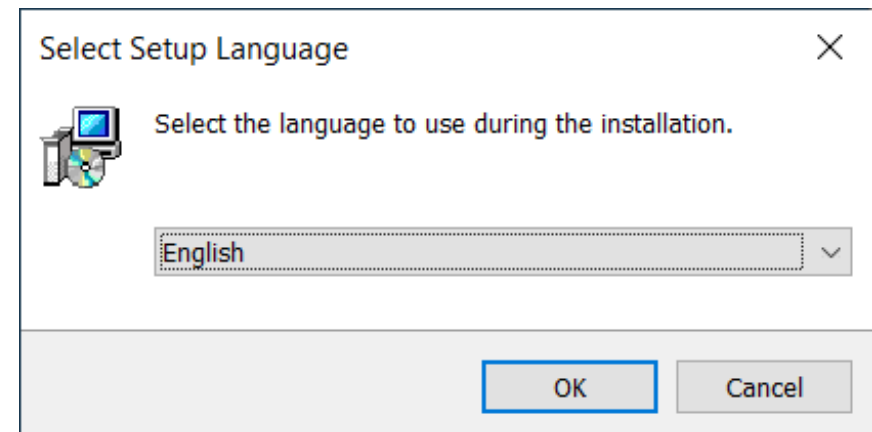Don't run

· The screen on the left is for Windows10

<Remark>

· The screen is different except for Windows 10.
It may be operated on other than Windows 10, but please install at your own risk.

· If a message such as "Security software blocked" appears, click "Trust" or "Accept" to continue downloading this software.
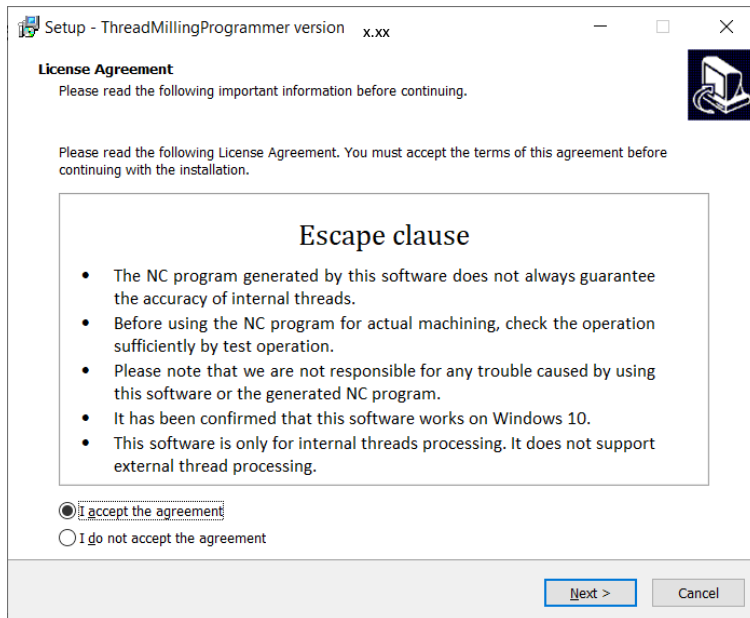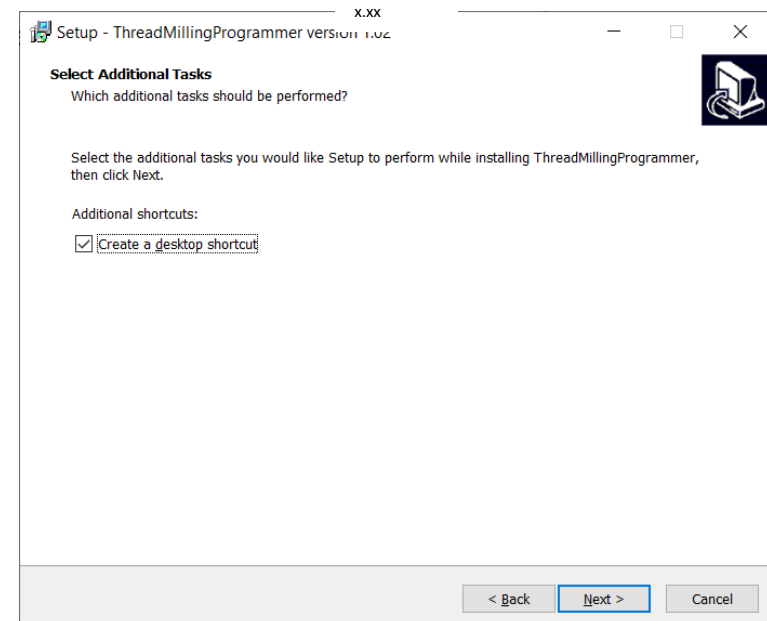
4. Click the "Run anyway" button.

Windows protected your PC

Microsoft Defender SmartScreen prevented an unrecognized app from
starting. Running this app might put your PC at risk.
x.xx exe

App: ThreadMillingProgrammer 1.02.exe
Publisher: Unknown publisher

Run anyway	Don't run

5. Select the language and click the "OK" button.

Select Setup Language   ✕

Select the language to use during the installation.

English   ⌄

OK   Cancel

6. Read the License Agreement and Escape clause before continuing.
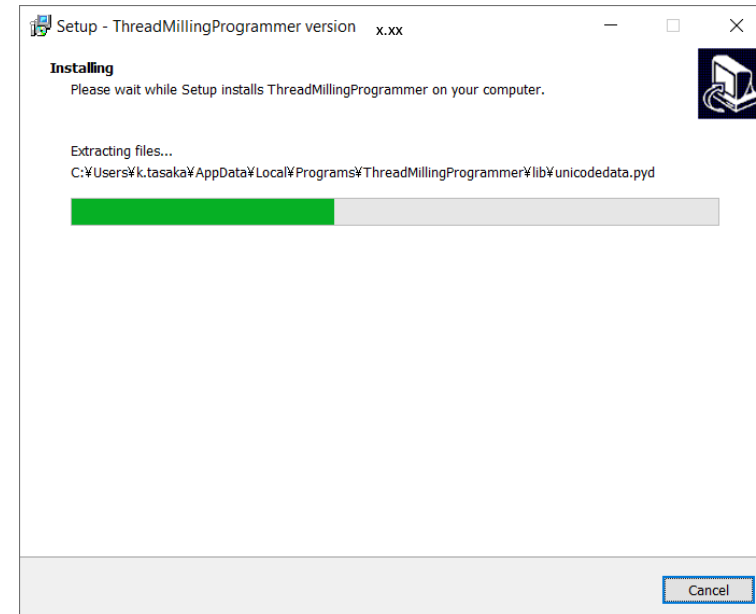   If you agree with the terms, click the circle of"I accept the agreement"and click the "Next"button.

Setup - ThreadMillingProgrammer version x.xx — □ ✕

License Agreement
Please read the following important information before continuing.

Please read the following License Agreement. You must accept the terms of this agreement before continuing with the installation.

Escape clause

- The NC program generated by this software does not always guarantee the accuracy of internal threads.
- Before using the NC program for actual machining, check the operation sufficiently by test operation.
- Please note that we are not responsible for any trouble caused by using this software or the generated NC program.
- It has been confirmed that this software works on Windows 10.
- This software is only for internal threads processing. It does not support external thread processing.

◉ I accept the agreement
○ I do not accept the agreement

Next >   Cancel

7. Click the circle of "Create a desktop shortcut" according to your preferenceand and click the "Next" button.

Setup - ThreadMillingProgrammer version 1.02 x.xx — □ ✕

Select Additional Tasks
Which additional tasks should be performed?

Select the additional tasks you would like Setup to perform while installing ThreadMillingProgrammer, then click Next.

Additional shortcuts:

☑ Create a desktop shortcut

< Back   Next >   Cancel

8. Click the "Next" botton if the setup option is ok as shown below.

Setup - ThreadMillingProgrammer version x.xx

Ready to Install
Setup is now ready to begin installing ThreadMillingProgrammer on your computer.

Click Install to continue with the installation, or click Back if you want to review or change any settings.

Additional tasks:
    Additional shortcuts:
       Create a desktop shortcut

< Back   Install   Cancel
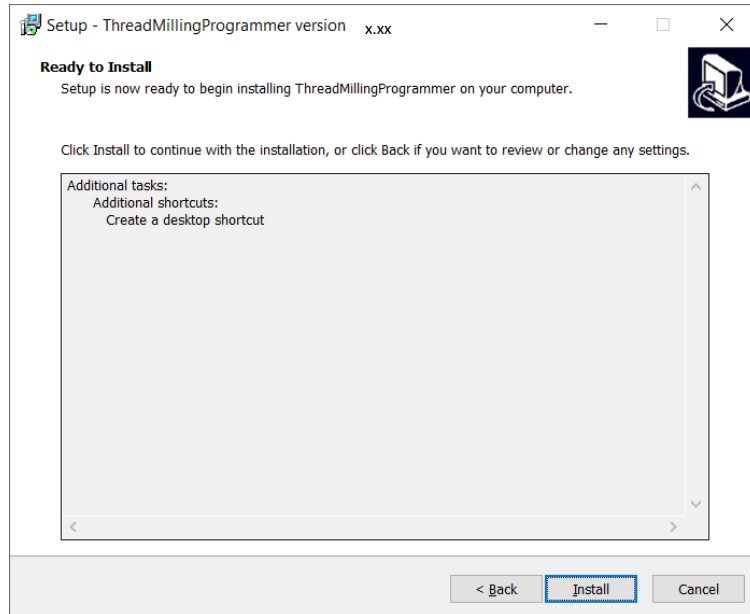
9. Installing the software

Setup - ThreadMillingProgrammer version x.xx

Installing
Please wait while Setup installs ThreadMillingProgrammer on your computer.

Extracting files...
C:¥Users¥k.tasaka¥AppData¥Local¥Programs¥ThreadMillingProgrammer¥lib¥unicodedata.pyd

Cancel

10. Installation has been completed.
    Check the square of "ThreadMillingProgrammer"to start the software automatically.
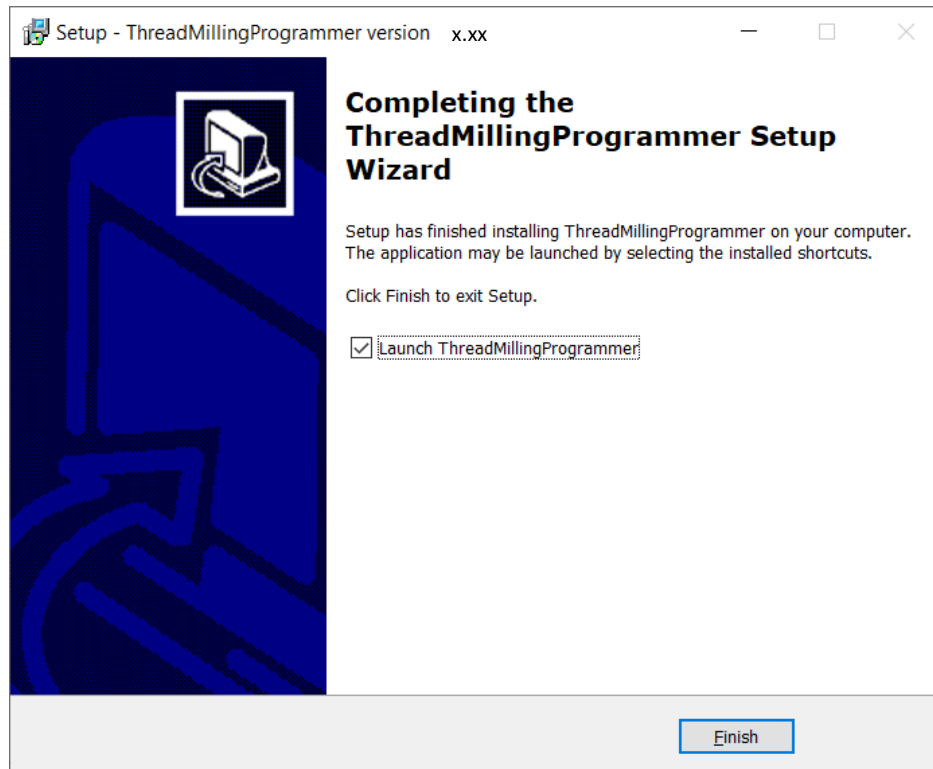
Initial setting

1. Read the Escape clause carefully and click the "Agree and start using" if you agree with the clauses.
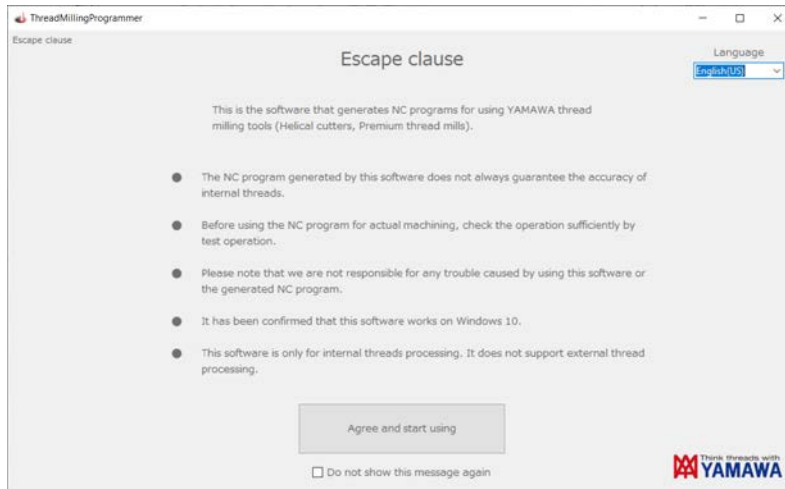
2. "Initial setting" menu shows up for the first use. Change the setting according to your application. If you're not sure about the setting or don't have any preference for now, you can also change the initial setting later.
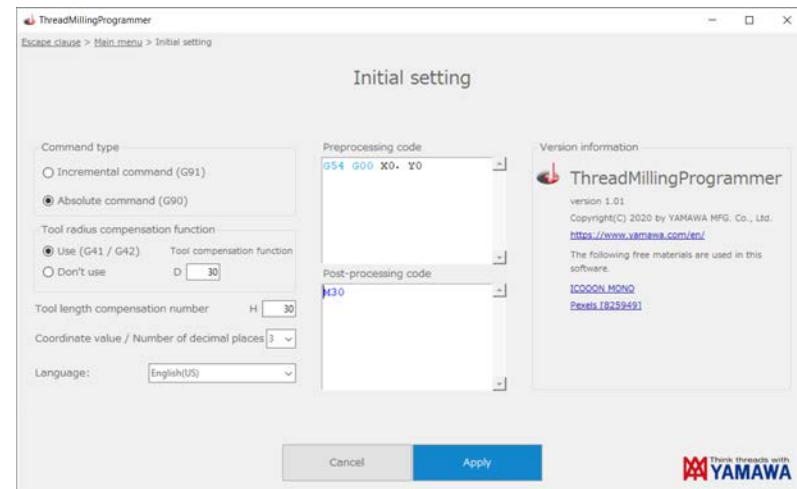
Preprocessing code / Post-processing code

You can add Preprocessing codes before the NC program and Post-processing code after NC Program by putting each code in the following window. By setting the codes you want to add before and after the internal thread processing, such as selecting the coordinate system, turning the coolant on/off, and the end code (M30/M99) of the main/subprogram, you can save the trouble of manually editing each time and avoid forgeting to input it.

Example

The following program is generated when you input the Preprocessing code and Post-processing code in each window on the right.

1. O number, Tool info etc.

2. <**Preprocessing code**>

   Coordinate system setting, Coolant No.

3. Threading program

4. <**Post-processing code**>

   Coolant off, End of the program

5. Finish mark

Preprocessing code

`G54 G00 X0 Y0 M08`

Post-processing code

`M09`
`M30`

☐ Edit code

```
%
O0100                              Automatically
(SIZE=M14 P=1.0 L=3.000)            generated
(TOOL=MC-HLC 10204N100M)
G54 G00 X0 Y0 M08          Preprocessing code
G90
G43 H30 S1110 M03
G00 Z-3.875
G41 G01 X-6.064 Y-1.064 D30 F85   Automatically
G03 X0.000 Y-7.128 Z-3.750 I6.064
Z-2.750 F46 J7.128                  generated
X6.064 Y-1.064 Z-2.625 J6.064 (Threading program)
G40 G00 X0.000 Y0.000
Z0.000 M05
M09
M30                        Post-processing code
%
                          Automatically generated
```
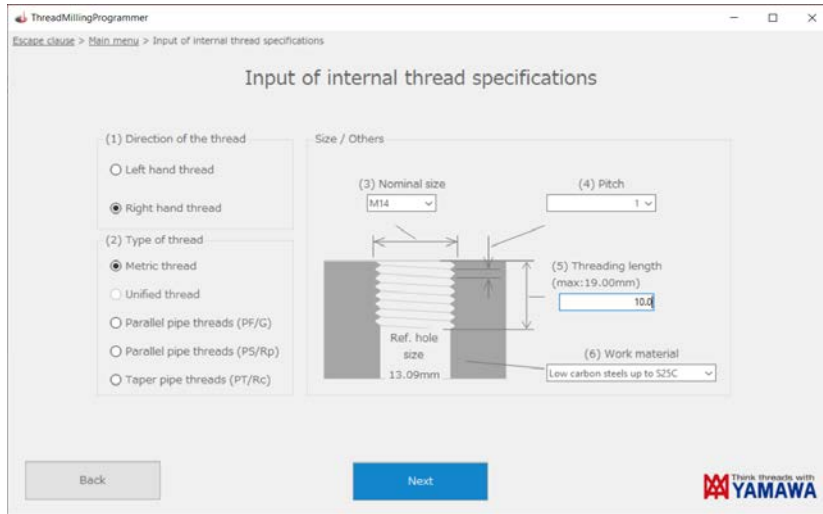
Basic usage

You can easily output a NC program simply by inputting conditions and dimensions according to the menu displayed in ThreadMillingProgrammer.
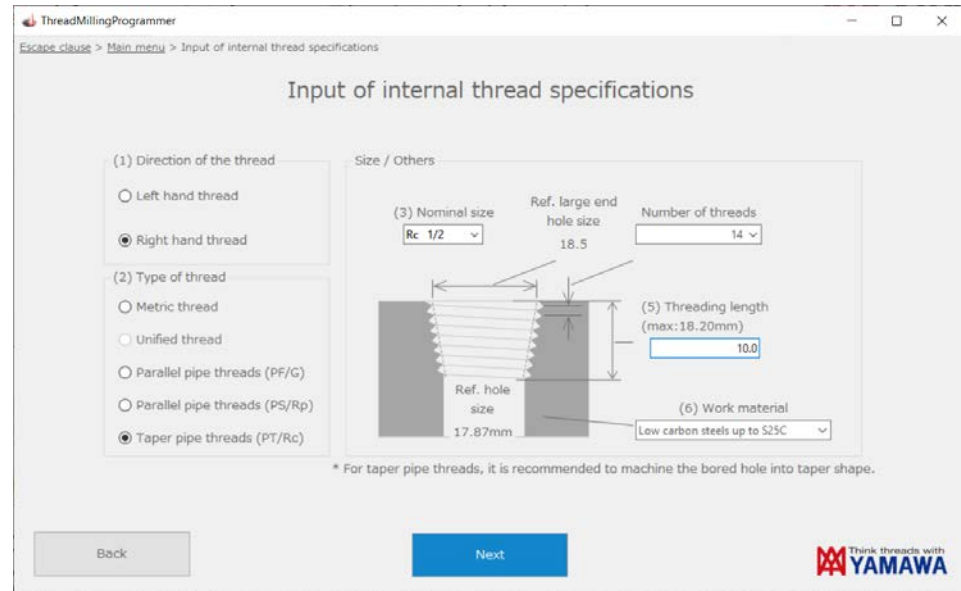
1. Select the type of thread mill you have decided to use.
   If you click the "Not decided", you can check the features of each thread mill so you can find the best one for your needs.
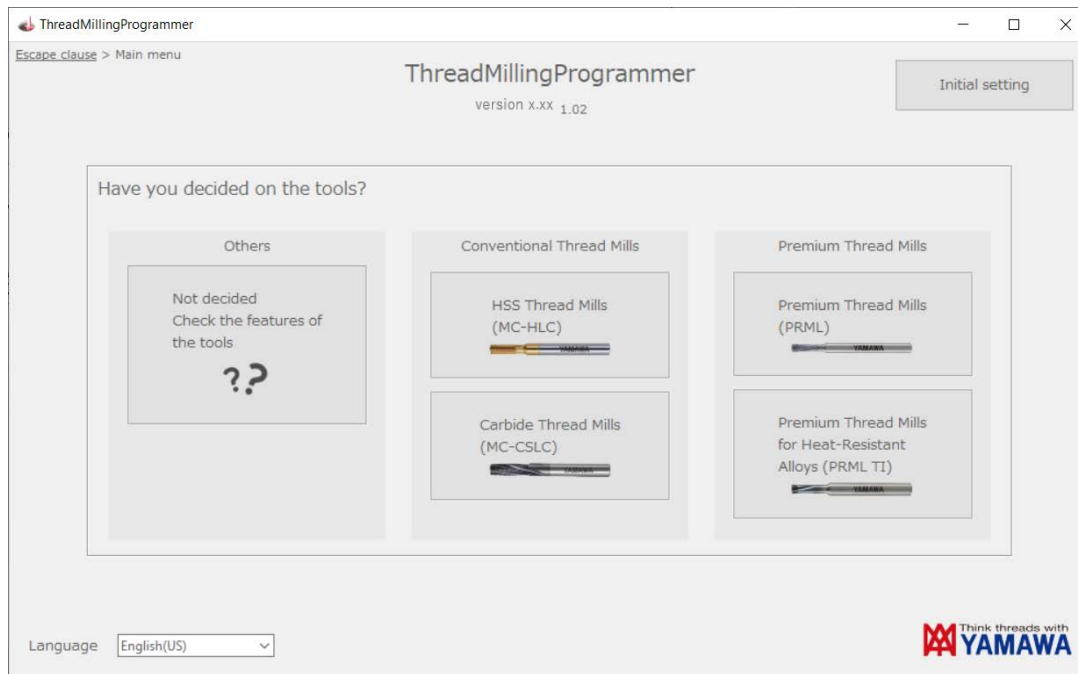
2. Select or enter the details of the internal thread to process in order from (1) to (6) on the menu screen.

When "Tapered thread for pipe (PT/Rc)" is selected as the thread type, the recommended large end diameter is displayed. It is recommended to cut the bored hole in a tapered shape to reduce the machining load on the tap.
Please cut the bored hole using the dimension displayed as the recommended large end diameter as a guideline.
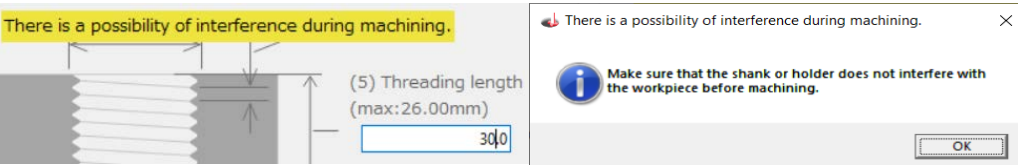
If the threading length in (5) is set beyond the max. value, as shown above, a caution will be displayed highlighted in yellow. However, you can still create the NC program. Please proceed with the trial process carefully, making sure that the neck part of the threadmill does not interfere with the entrance of the internal thread.

3. Select the nominal size of the tool, feed rate and cutting speed.

The feed rate and cutting speed can be changed by inputting a value, moving the bar to the left or right, or clicking in the colored area of the graph. You can find the best suitable condition by referring to the tool rotation speed and reference process time.

4. Set the processing method. If you don't have any preferences, select the one with "recommended".
   In most cases, there should be no problem with one pass threading.

If you enter 2 or more as the number of passes, you can change the ratio to cut off by dividing the total height of the thread.
The height of each cut changes when you drag the border.

5. The NC program is output.

Set the program number, tool compensation, etc. according to your application.
You can also make fine adjustments to the finished dimensions after machining a test piece.
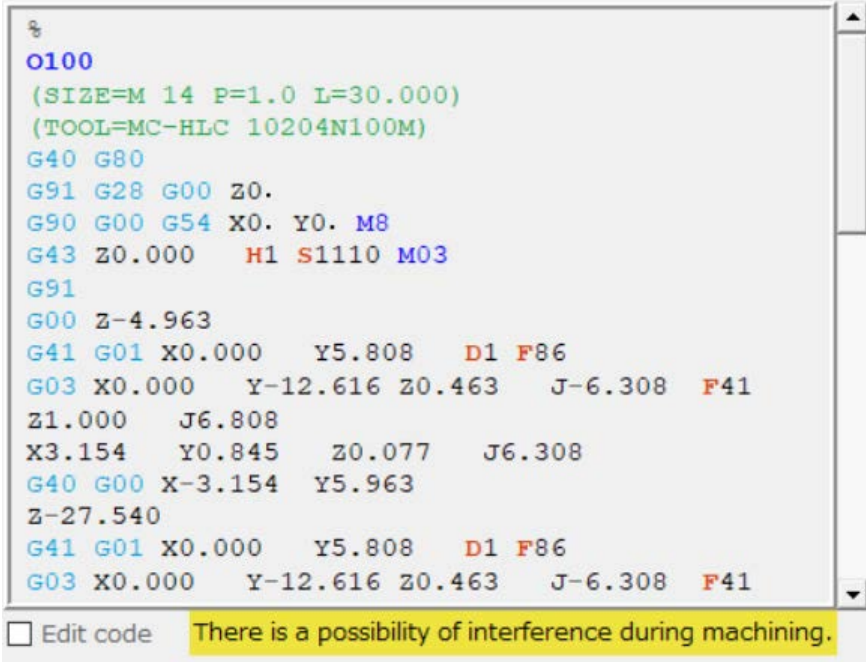If you check "Edit code", you can make corrections on the screen based on this program.

*Program number (O), tool length compensation number (H), and tool diameter compensation number (D) can be assigned arbitrary numbers.

*The tool diameter compensation value to be input to the machine tool is generally the radius of the tool.

If the threading length is set beyond the max. value, a caution will be displayed highlighted in yellow on the program window as well. Please proceed with the trial machining carefully.

```
%
O100
(SIZE=M 14 P=1.0 L=30.000)
(TOOL=MC-HLC 10204N100M)
G40 G80
G91 G28 G00 Z0.
G90 G00 G54 X0. Y0. M8
G43 Z0.000    H1 S1110 M03
G91
G00 Z-4.963
G41 G01 X0.000    Y5.808    D1 F86
G03 X0.000    Y-12.616 Z0.463    J-6.308    F41
Z1.000    J6.808
X3.154    Y0.845    Z0.077    J6.308
G40 G00 X-3.154    Y5.963
Z-27.540
G41 G01 X0.000    Y5.808    D1 F86
G03 X0.000    Y-12.616 Z0.463    J-6.308    F41
```

☐ Edit code   `There is a possibility of interference during machining.`

6. Click the "Save File" button and select the place to save in the dialog.

Save as					✕
← → ∨ ↑ 🖳 › This PC › Desktop			∨	↻	🔍 Search Desktop

File name: ThreadMillProgram001

Save as type: Text file

∨ Browse Folders ⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩⬩ Save | Cancel