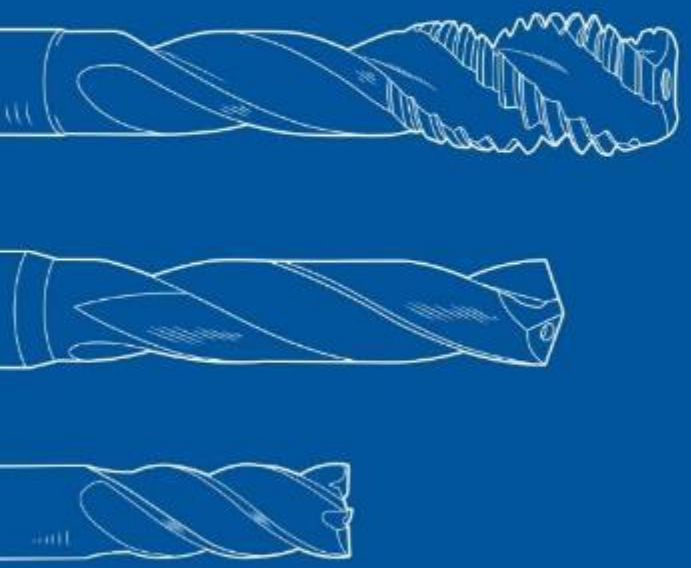
shaping your dreams

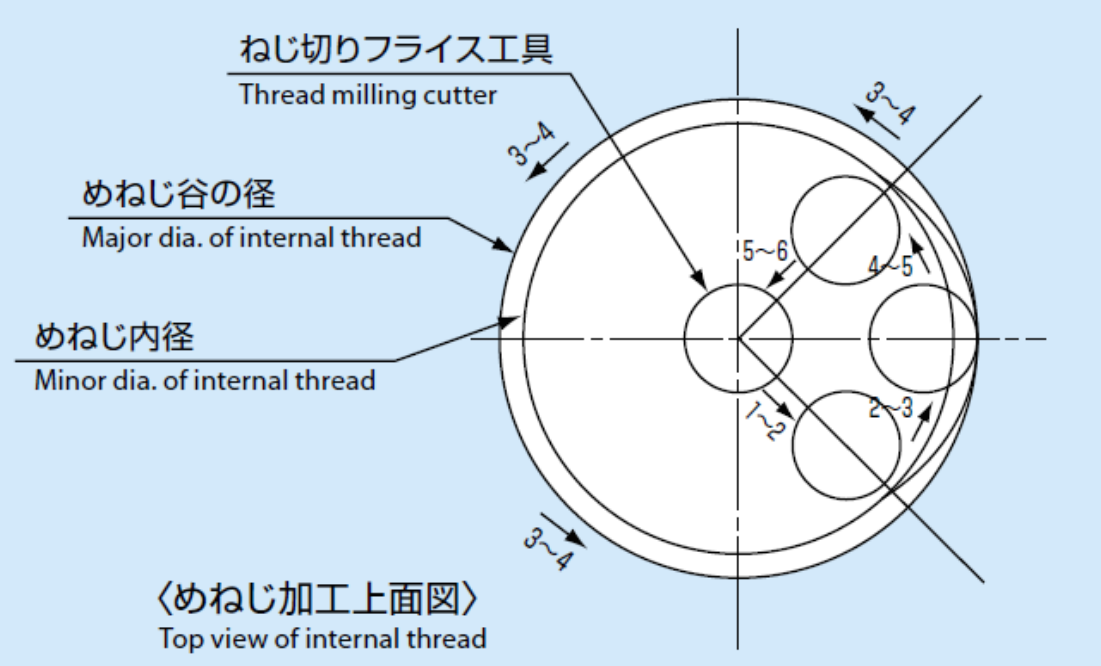


ThreadPro

User’s manual

How to use Thread mill

Thread mills are used on numerical controlled machining centers equipped with 3-axis control and helical milling function.



動作順序 Step	動作内容 Function
1～2	クリアランスを残しながら早送りで接近 Move to edge (maintain clearance).
2～3	アプローチ(ヘリカル切削しながら切り込む) Cut with helical milling.
3～4	本切削(360° ヘリカル切削) Mill the circumference of the circle.
4～5	リリース(ヘリカル運動しながら切り離す) Pull away from edge.
5～6	中心まで早送りで戻る Remove tool.

When approach and release, helical milling should be used as shown above for machining precision and efficiency.

WEB Application : ThreadPro

Features

・You can use it on PC, smart phones or tablets without downloading software.

・RPRG is produced. (RPRG : Reference value for tool radius offset)

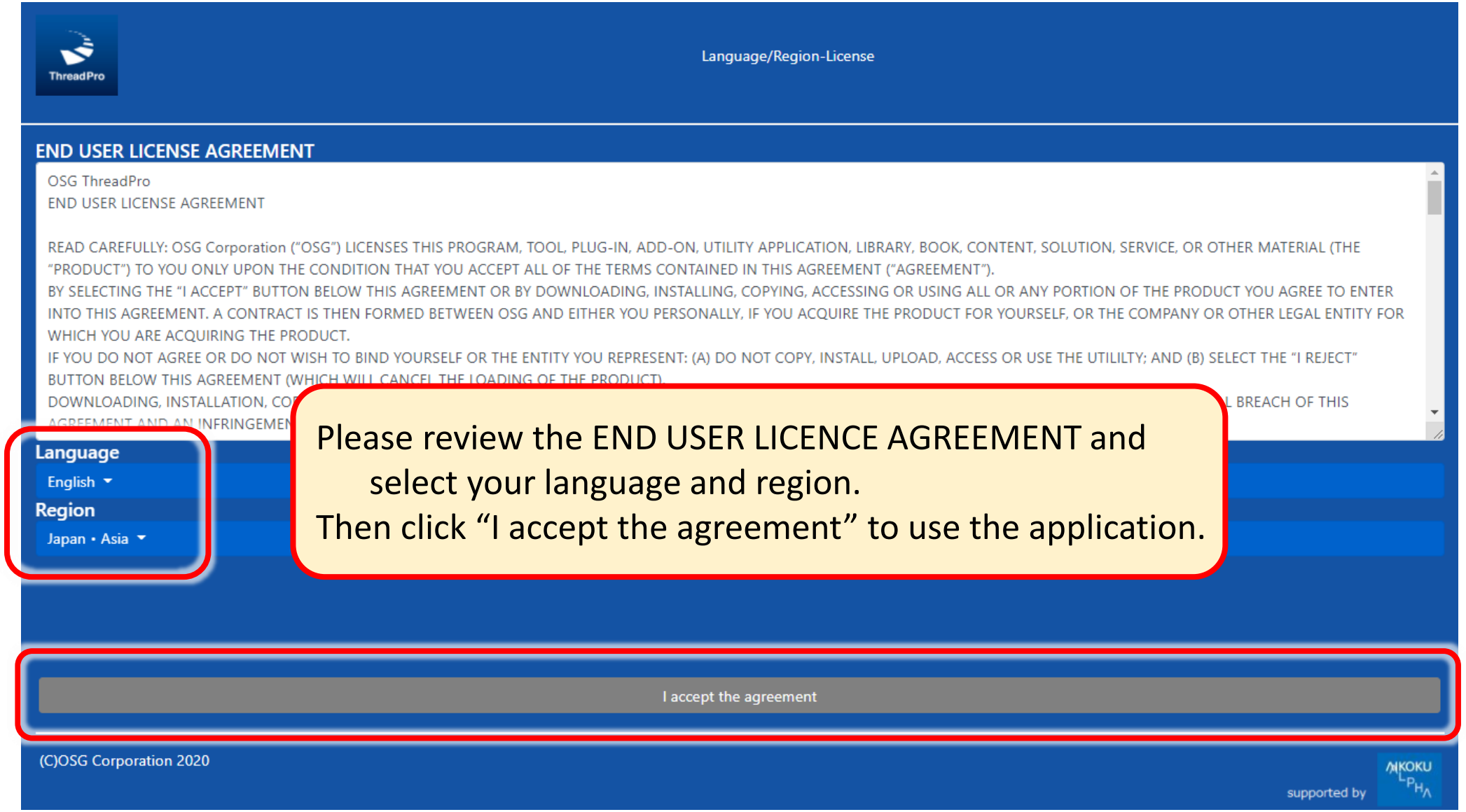
・Insert screw threads are available as well.

・New thread mill products are automatically updated.

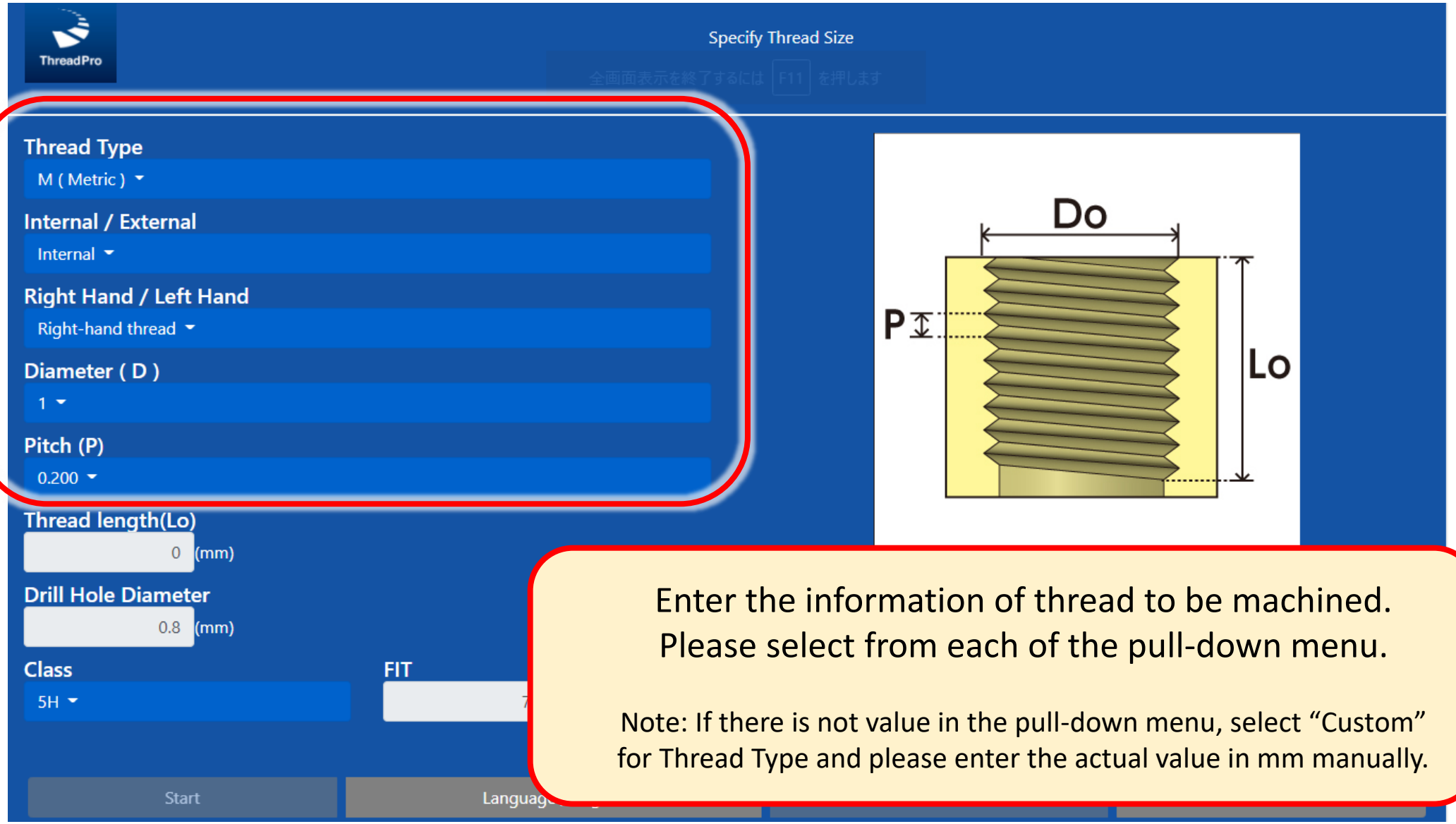
・Multiple languages（12 languages）

・Multiple NC languages（8 NC languages）

Selection of Language and Region/Terms of use



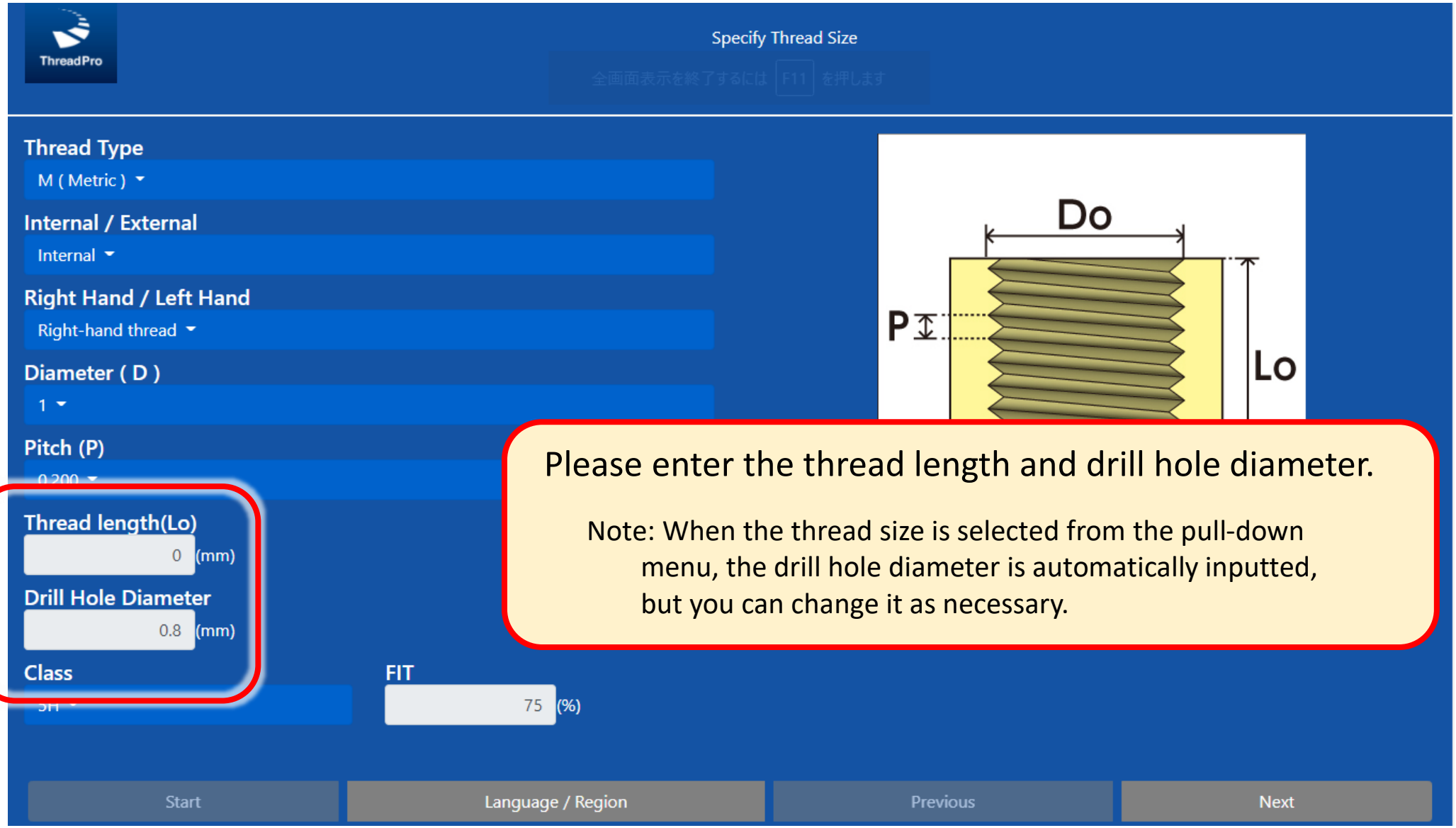
Thread information-1



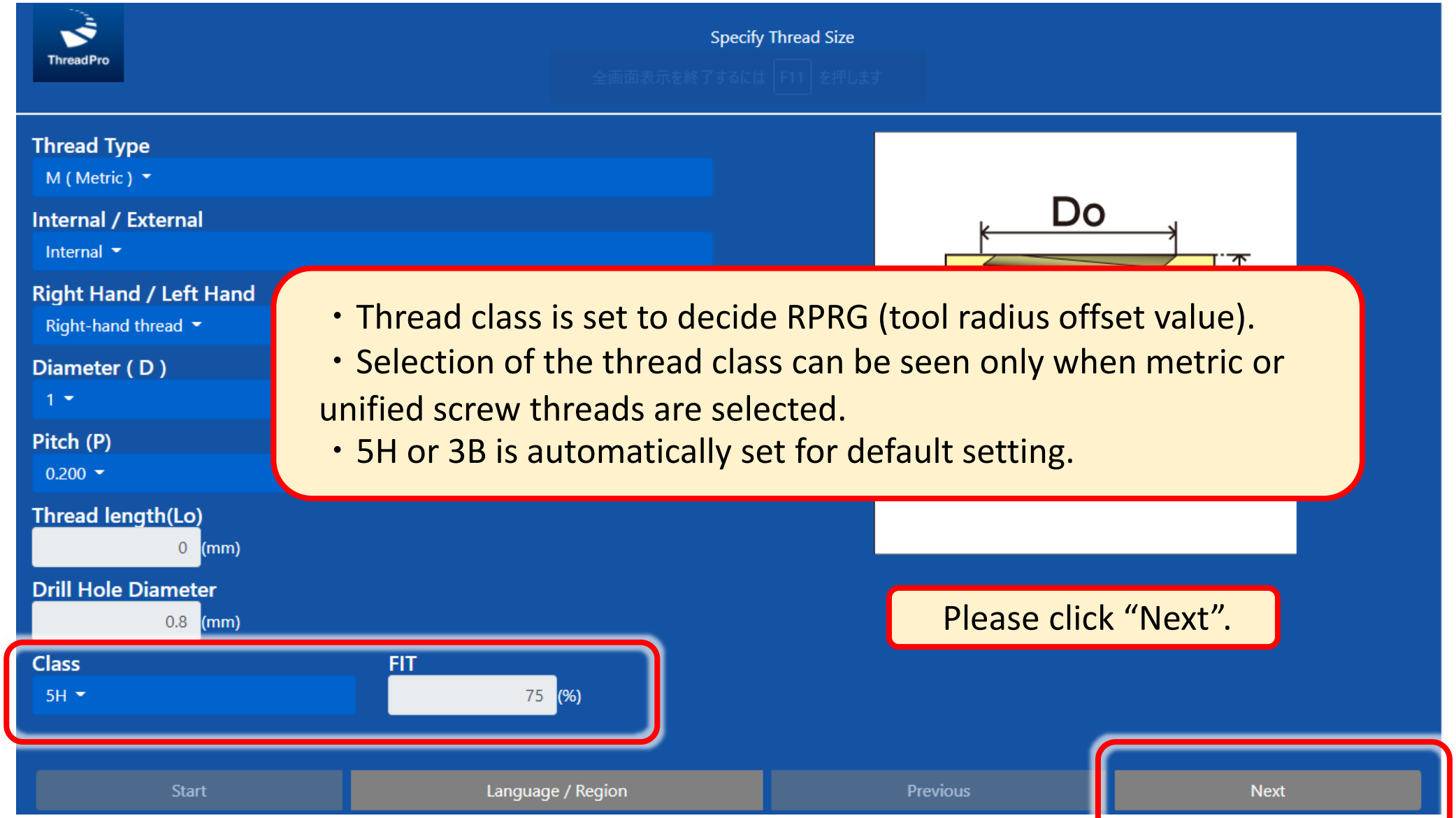
Enter the information of thread to be machined.
Please select from each of the pull-down menu.

Note: If there is not value in the pull-down menu, select “Custom” for Thread Type and please enter the actual value in mm manually.

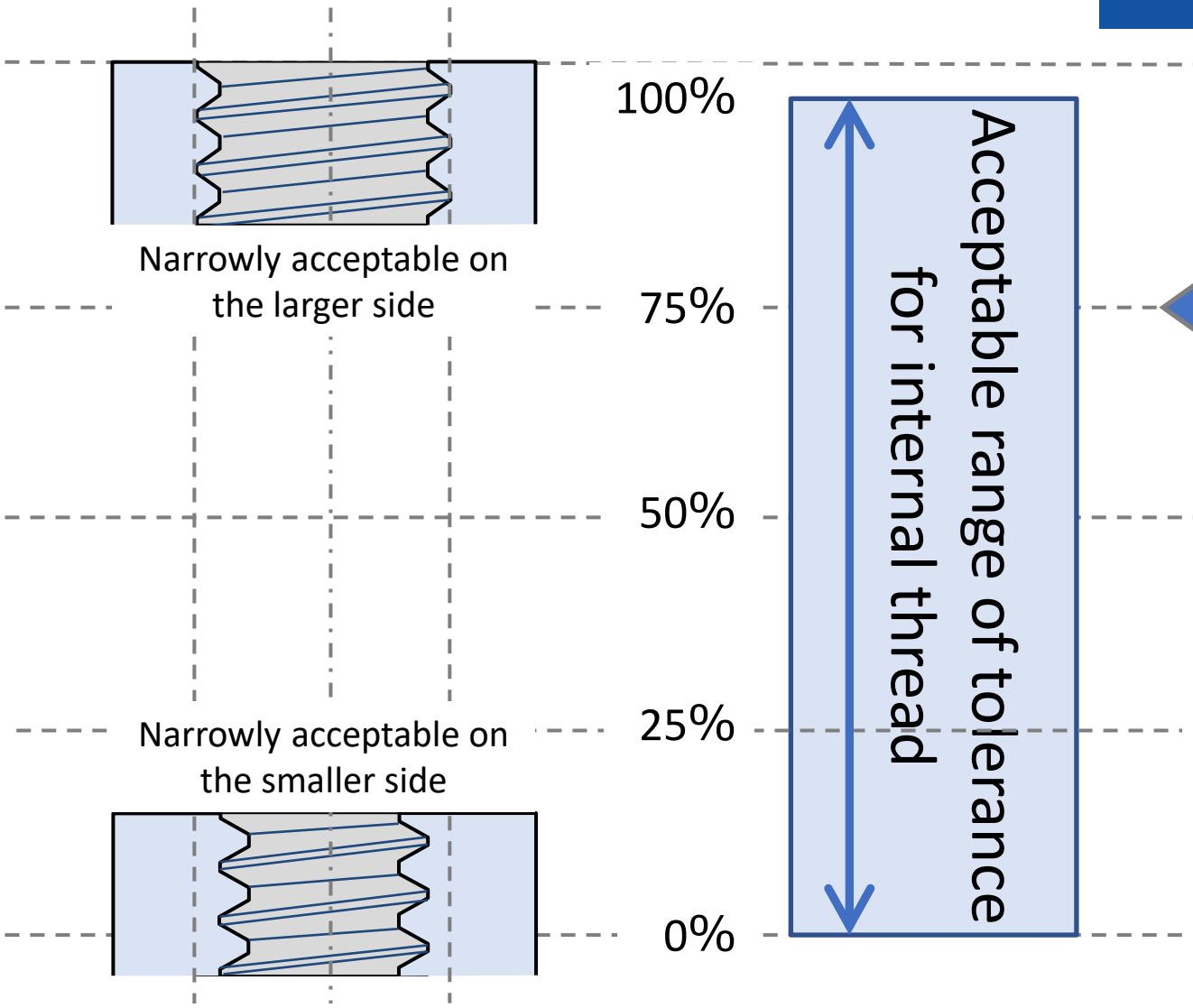
Thread information-2



Thread information-3



What is “FIT 75%” ?

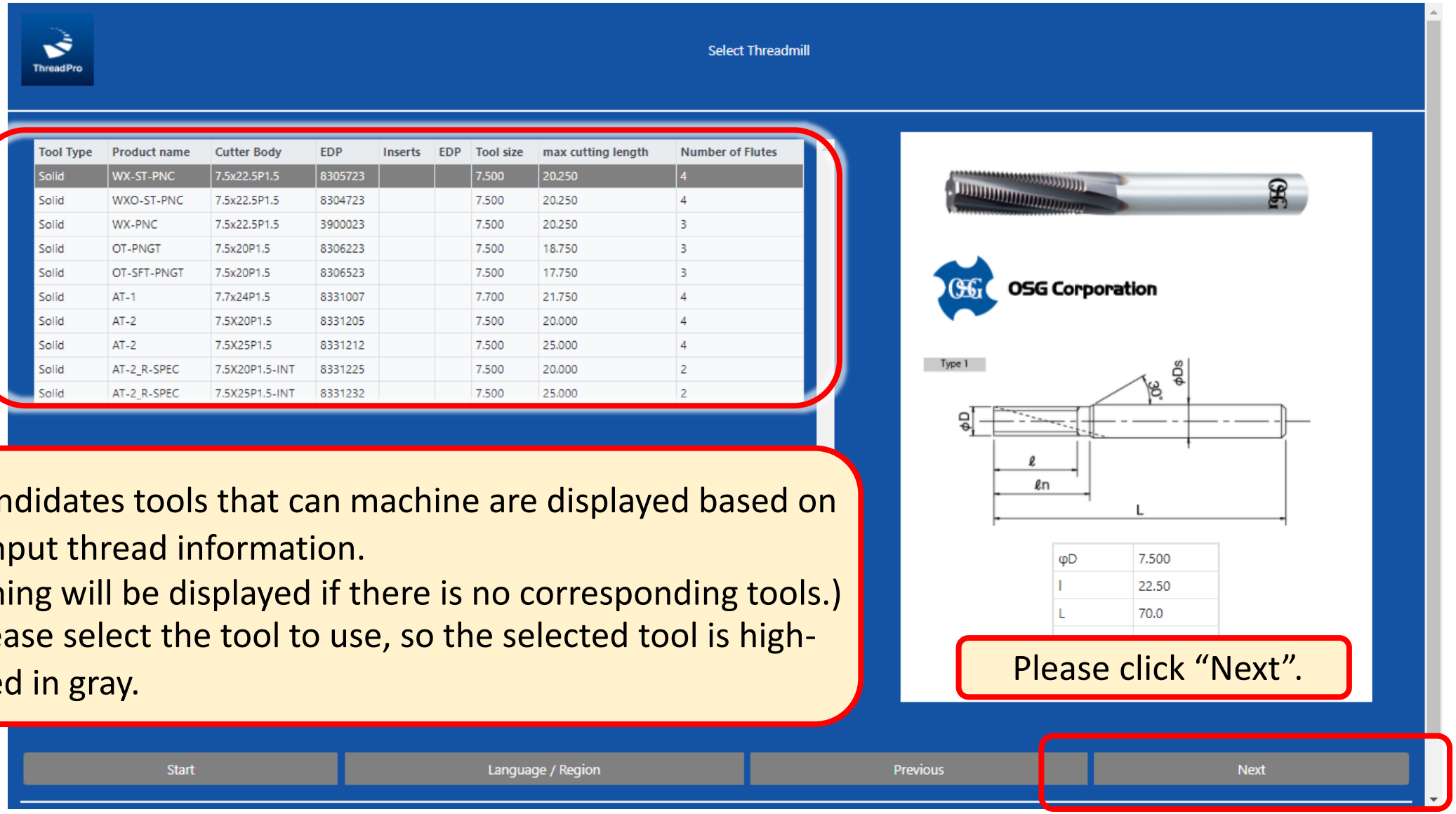


Aim Here!

75% of larger limit for internal thread

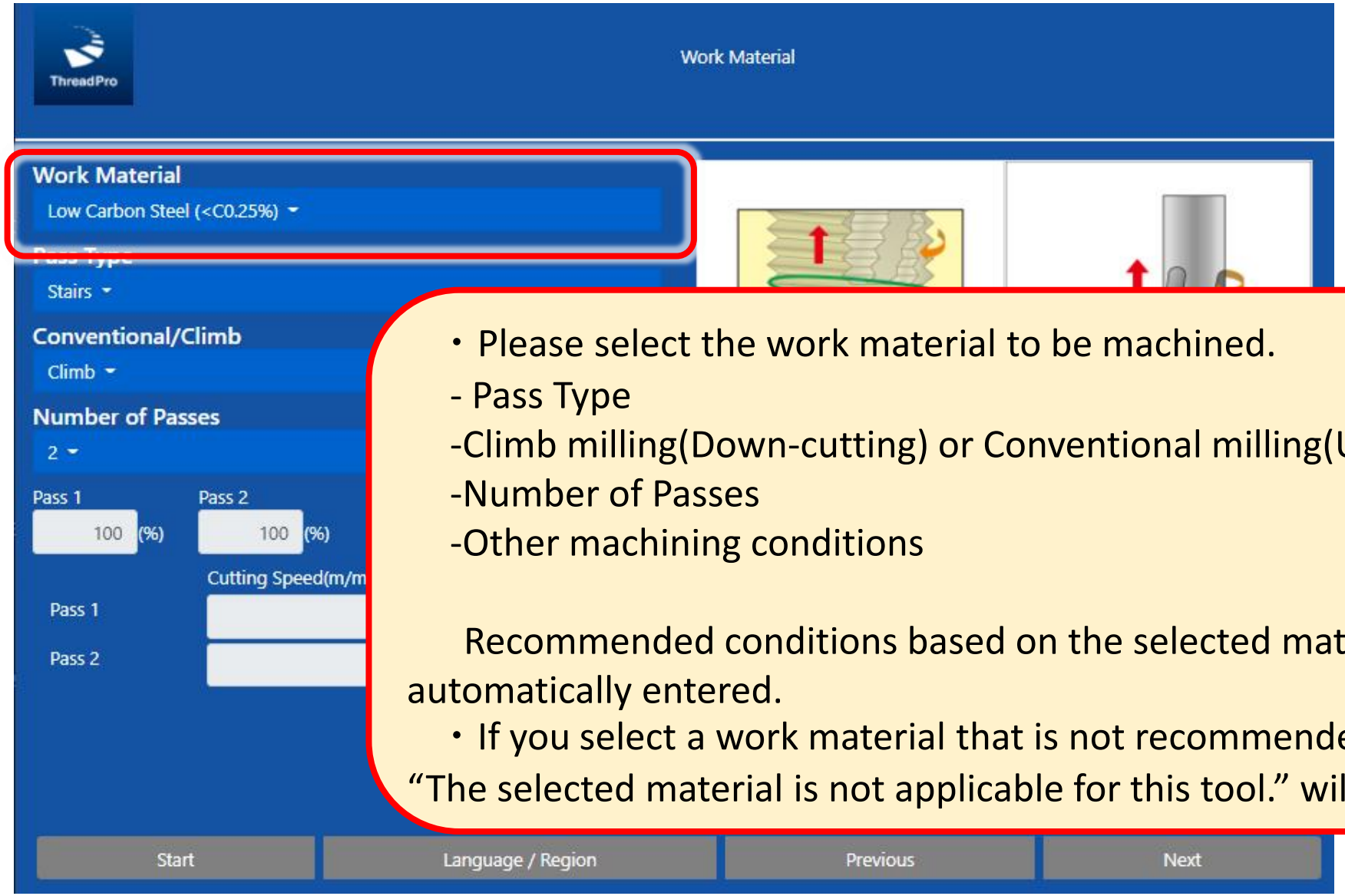
It means to aim at the acceptable range of threads. Default values are 75% (larger side) for internal threads, and 25% (smaller side) for external threads in light of their engagement.
You can change these values as necessary.

Tool selection



・Candidates tools that can machine are displayed based on the input thread information.
(Nothing will be displayed if there is no corresponding tools.)
・Please select the tool to use, so the selected tool is high-lighted in gray.

Machining conditions



・Please select the work material to be machined.

- Pass Type

-Climb milling(Down-cutting) or Conventional milling(Up-cutting)

-Number of Passes

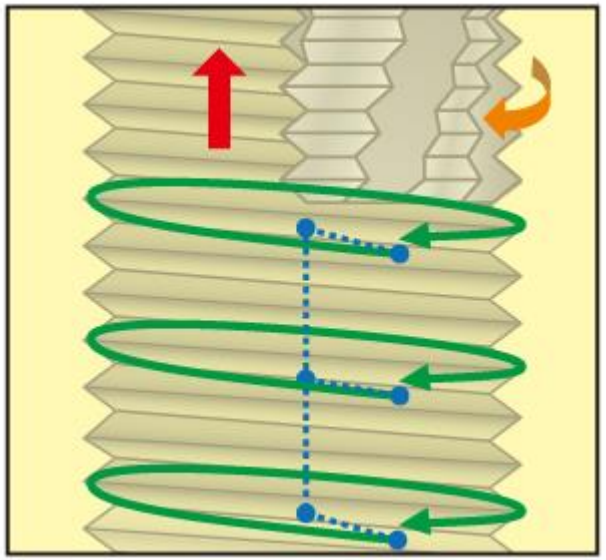
-Other machining conditions

Recommended conditions based on the selected material are automatically entered.

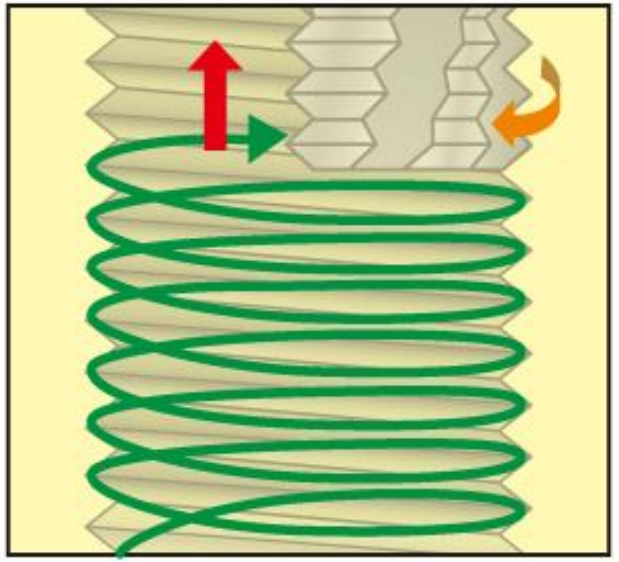
・If you select a work material that is not recommended, a message “The selected material is not applicable for this tool.” will appear.

Stairs type (Multi pass)/Continuous type (Single pass)

The stairs type (multi pass) for more efficiency, the continuous type (single pass) for more quality can be selected according to the application.
The stairs type is basically recommended but select the type that suits your machining environment and purpose.



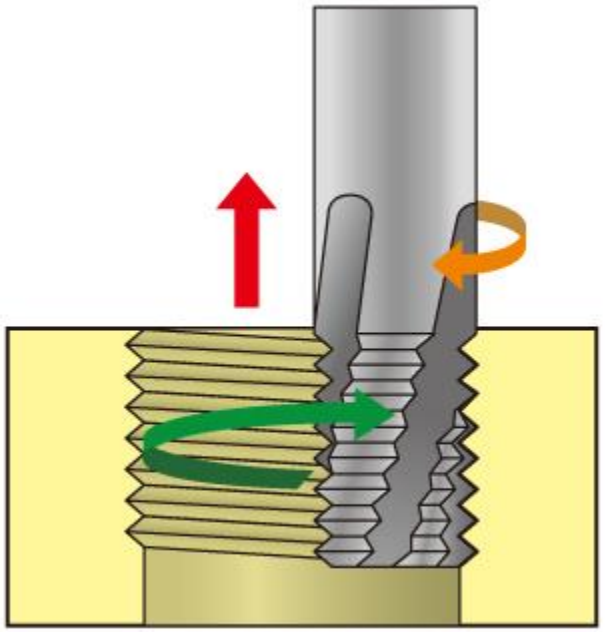
Stairs type



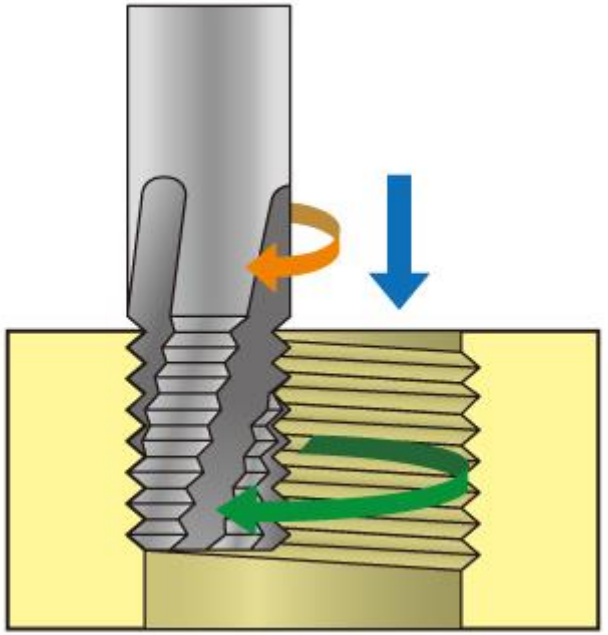
Continuous type

Climb milling (Down-cut) / Conventional milling (Up-cut)

For an internal right-hand thread, climb milling (down-cut) refers to cutting upward from the bottom. While conventional milling (up-cut) refers to cutting downward from the hole entrance to bottom.
It depends on whether right or left-hand thread and milling direction of the tool.

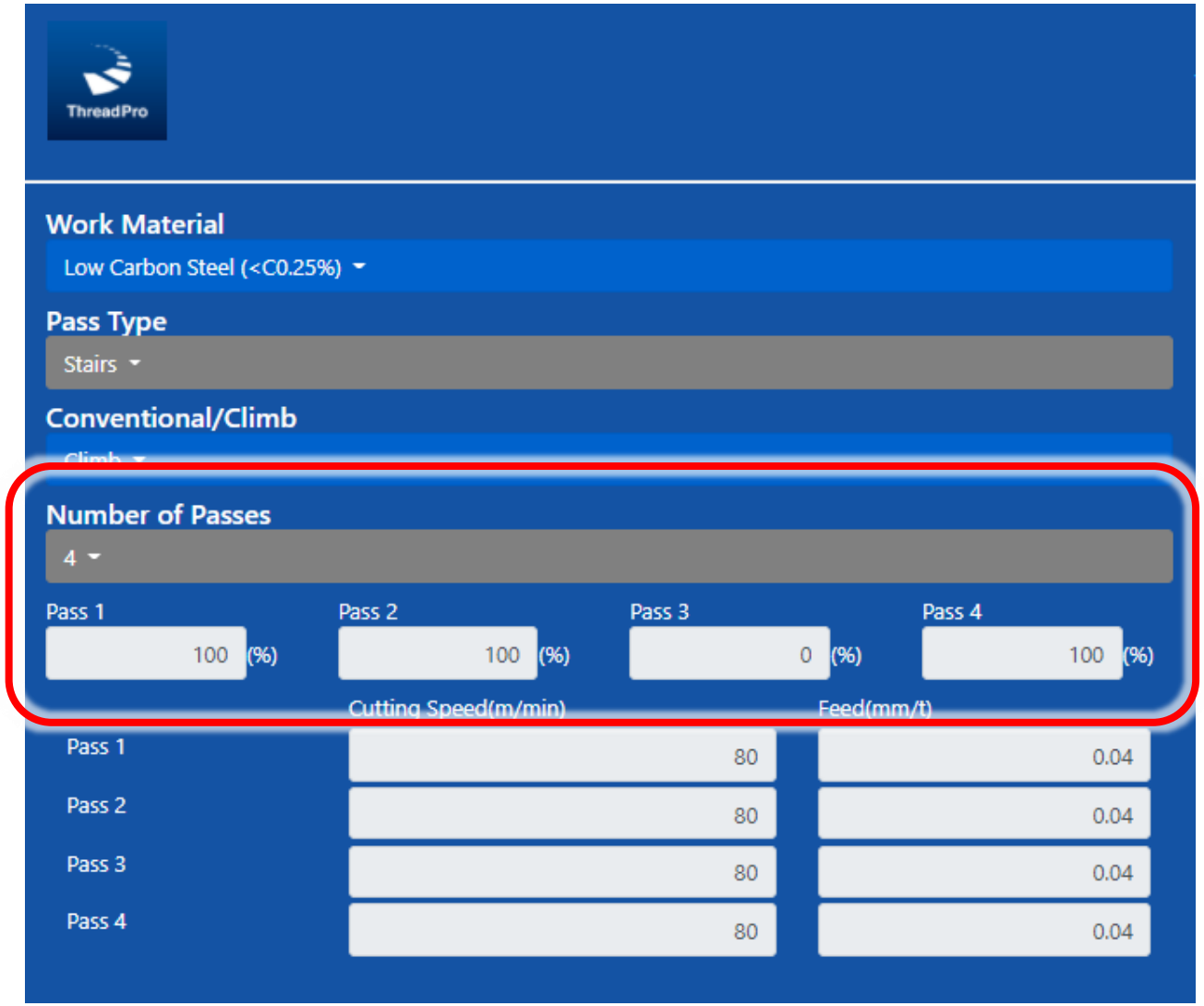


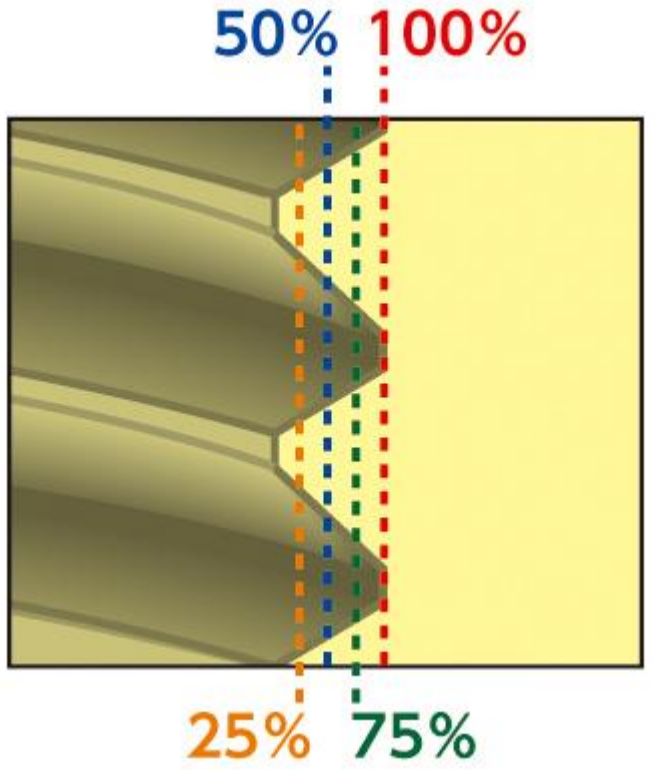
Climb milling
For tool durability



Conventional milling
For finishing quality

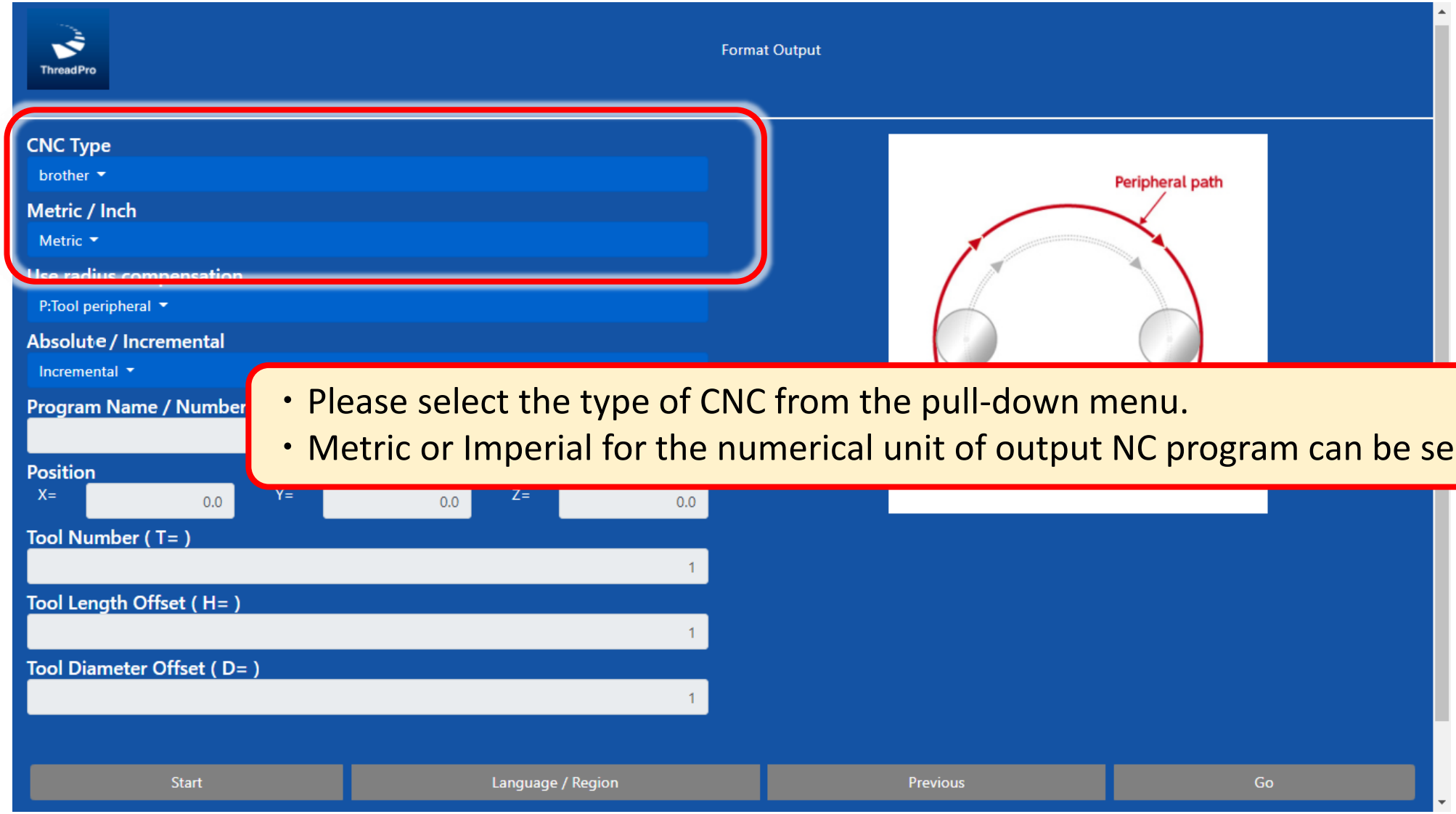
Number of Passes





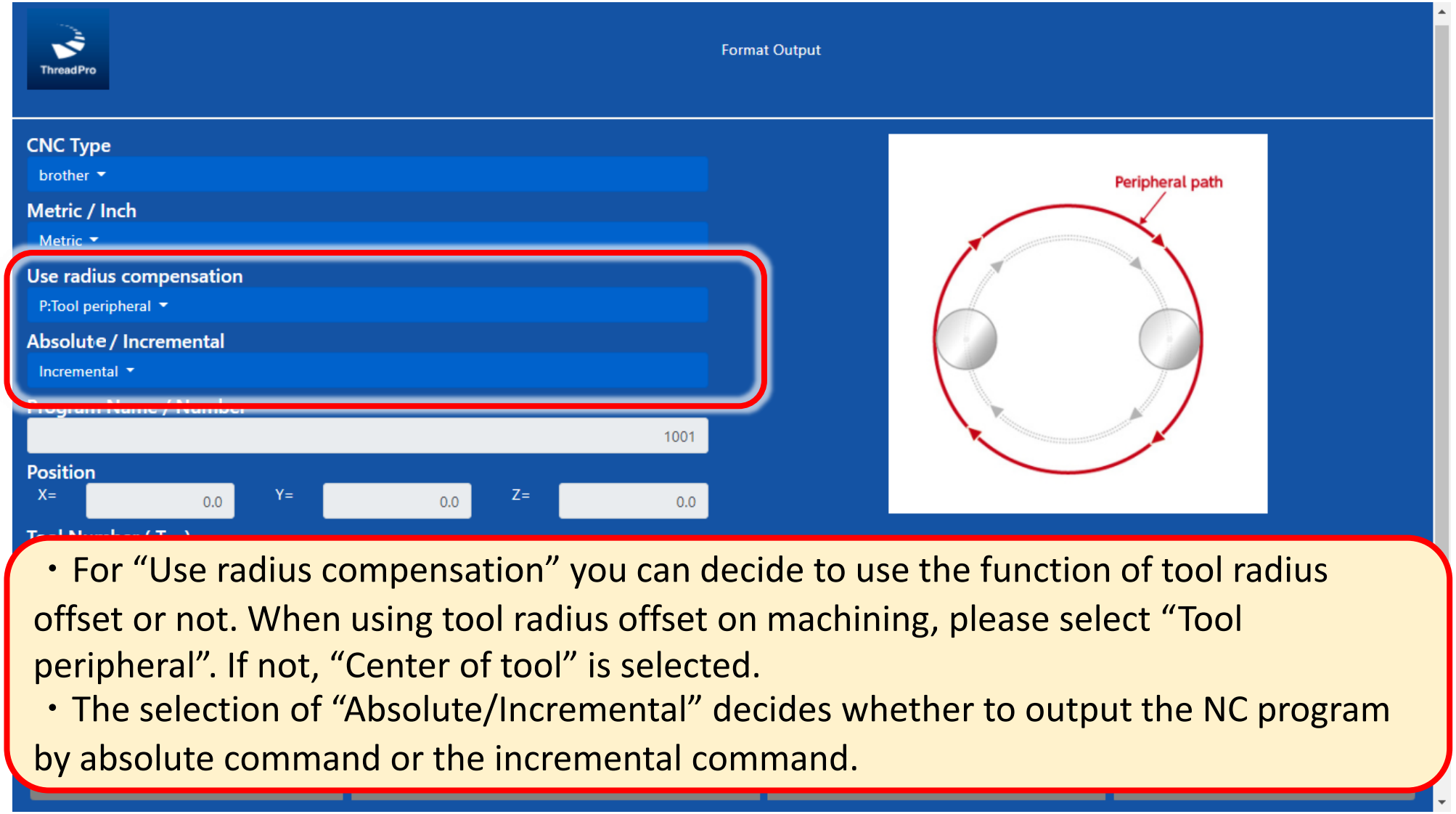
The amount of cut can be adjusted on each pass. Also, by entering 100% in “Pass-1” you can easily create a zero-cut path program.

Type of CNC



- Please select the type of CNC from the pull-down menu.
- Metric or Imperial for the numerical unit of output NC program can be selected.

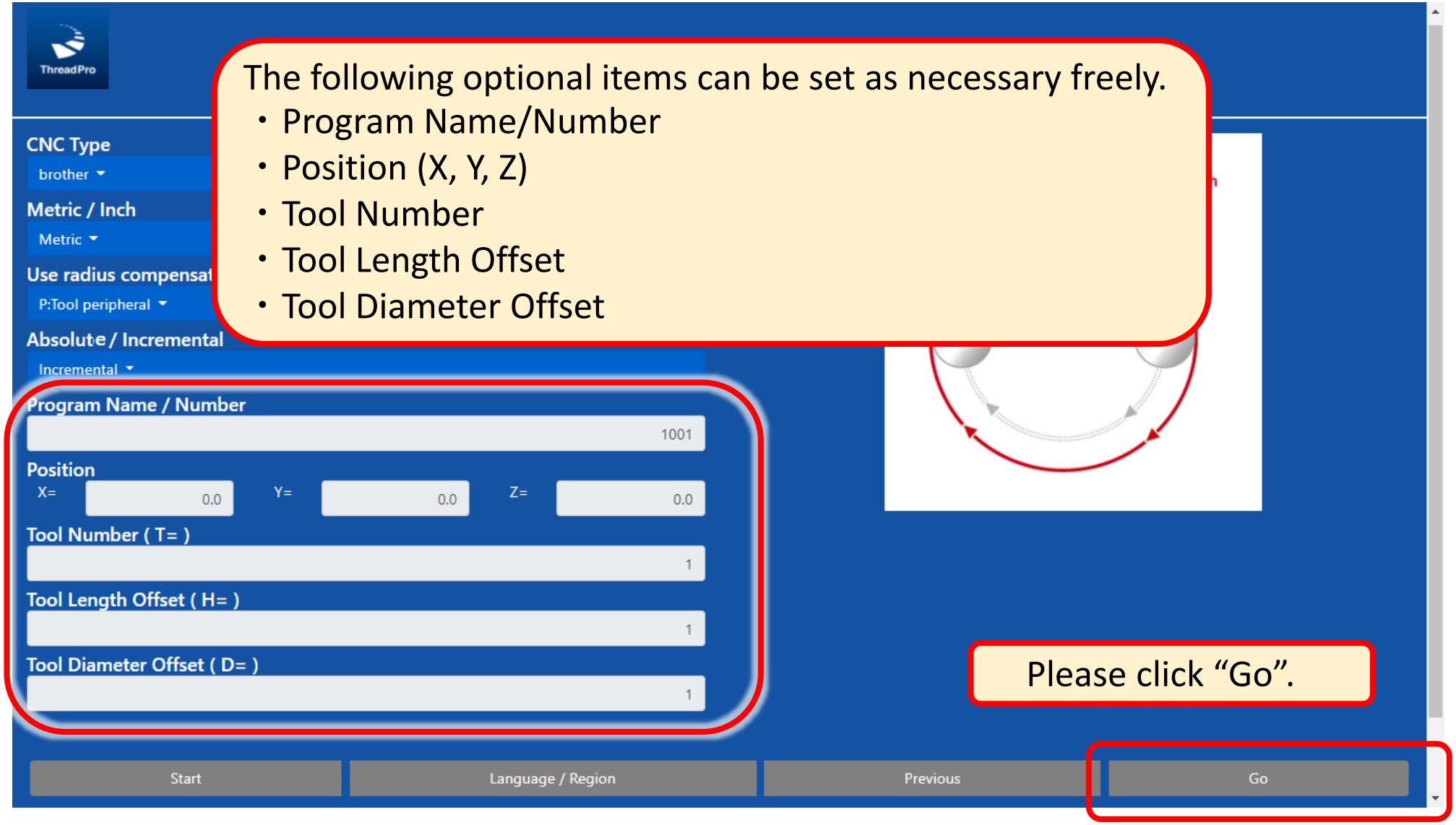
Tool radius offset



・For “Use radius compensation” you can decide to use the function of tool radius offset or not. When using tool radius offset on machining, please select “Tool peripheral”. If not, “Center of tool” is selected.

・The selection of “Absolute/Incremental” decides whether to output the NC program by absolute command or the incremental command.

Optional items

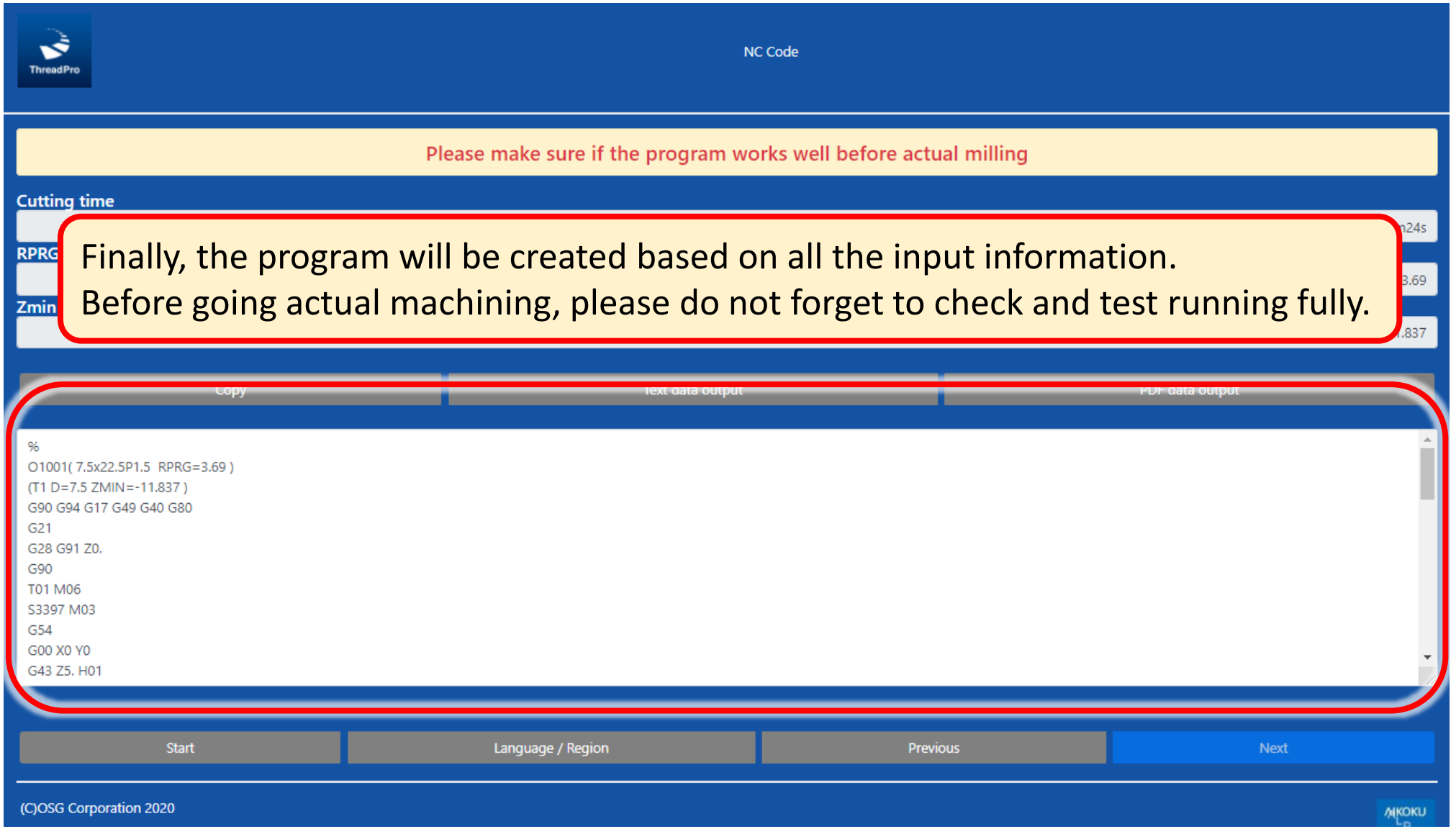


The following optional items can be set as necessary freely.

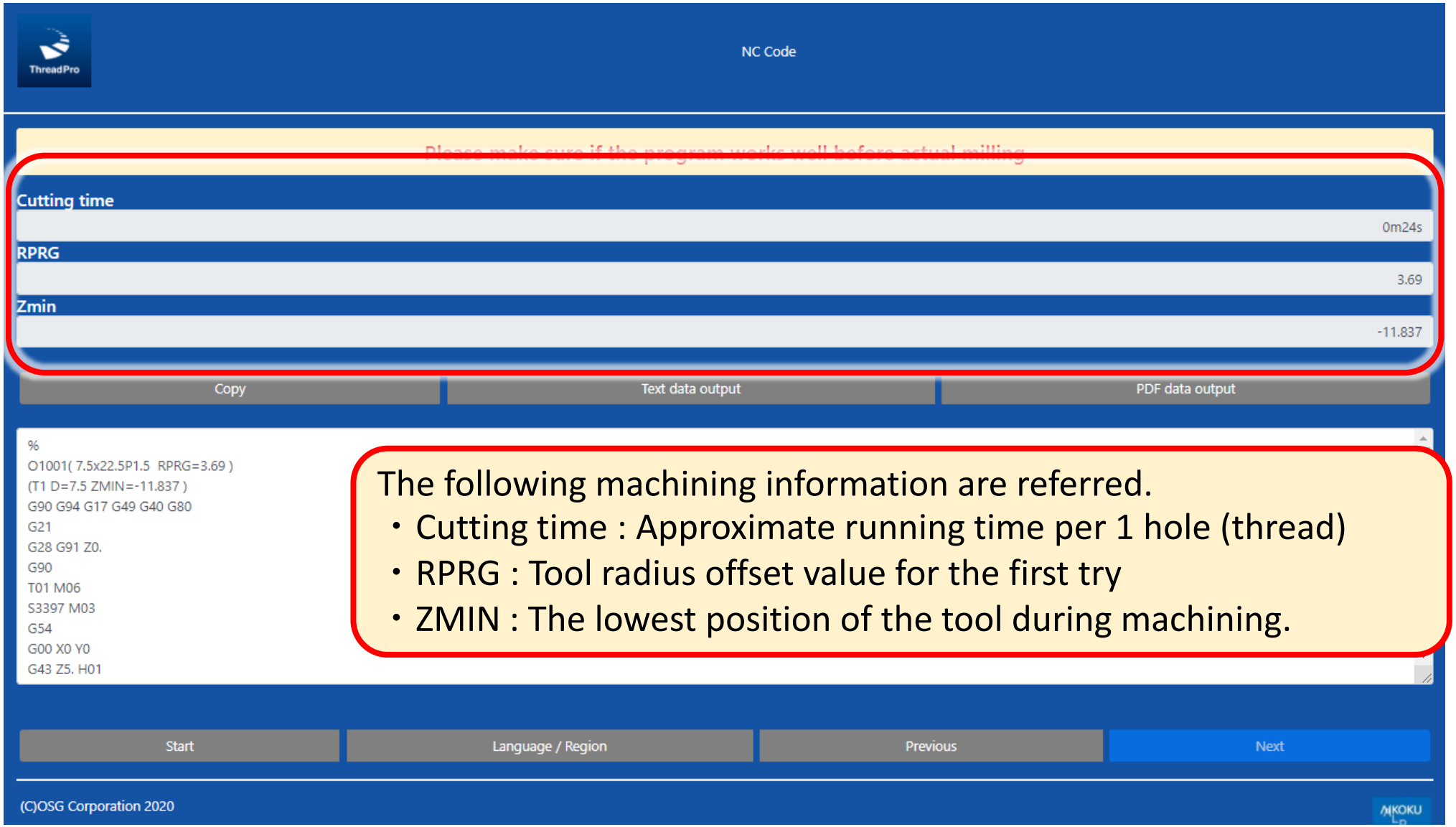
- Program Name/Number
- Position (X, Y, Z)
- Tool Number
- Tool Length Offset
- Tool Diameter Offset

Please click “Go”.

Creation of NC program



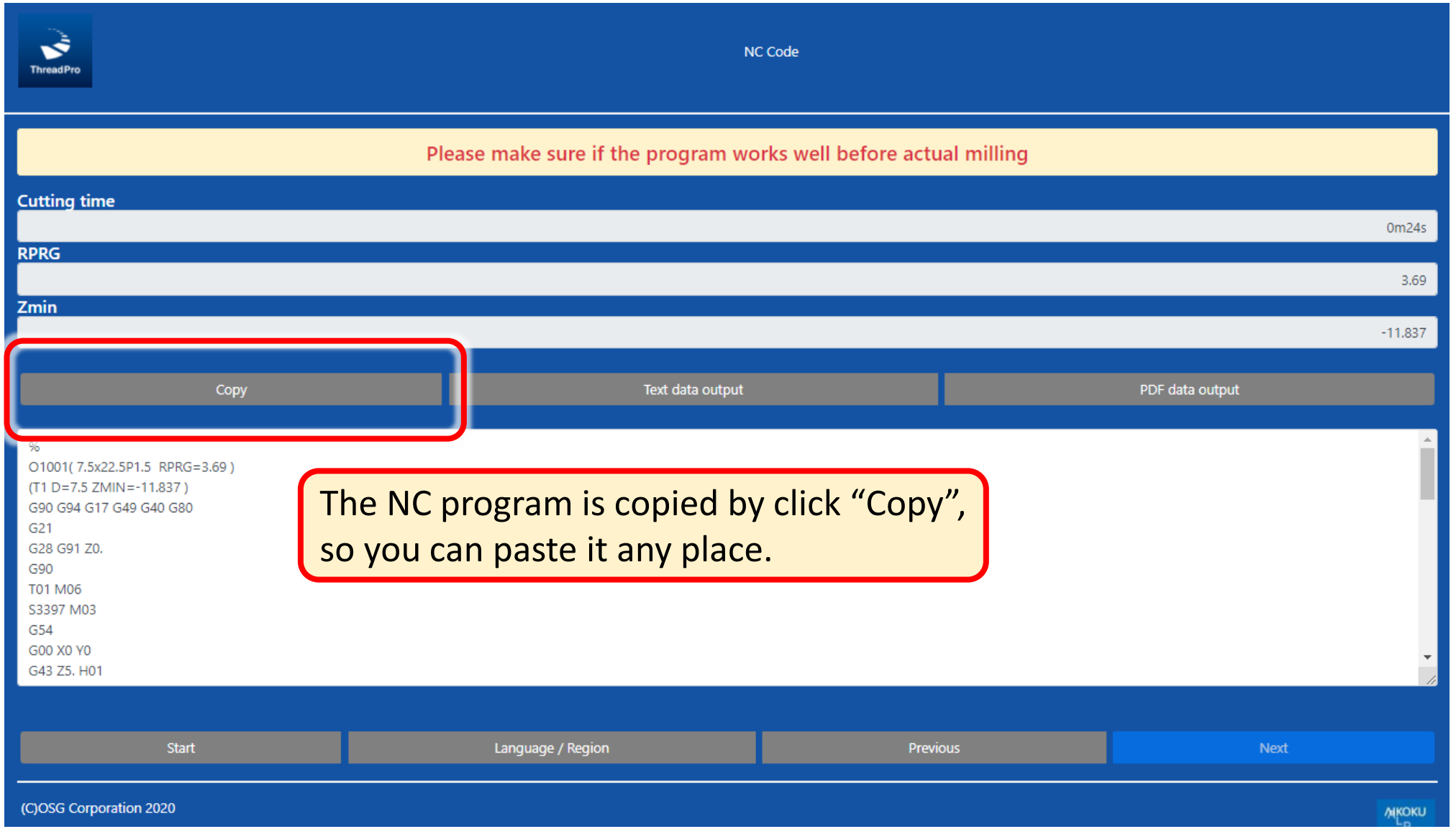
Reference information



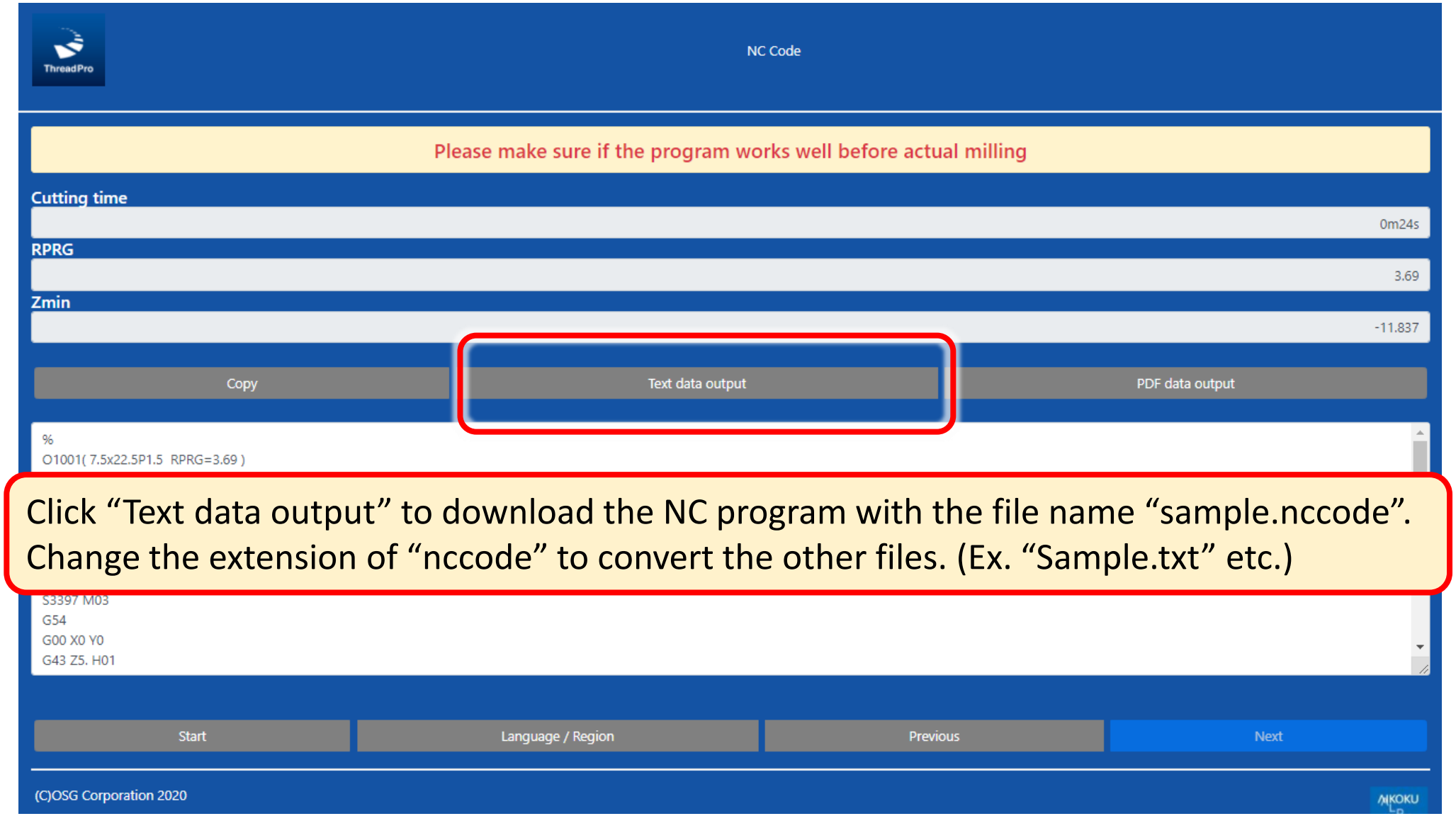
The following machining information are referred.

- Cutting time : Approximate running time per 1 hole (thread)
- RPRG : Tool radius offset value for the first try
- ZMIN : The lowest position of the tool during machining.

Function 1 (Copying NC program)

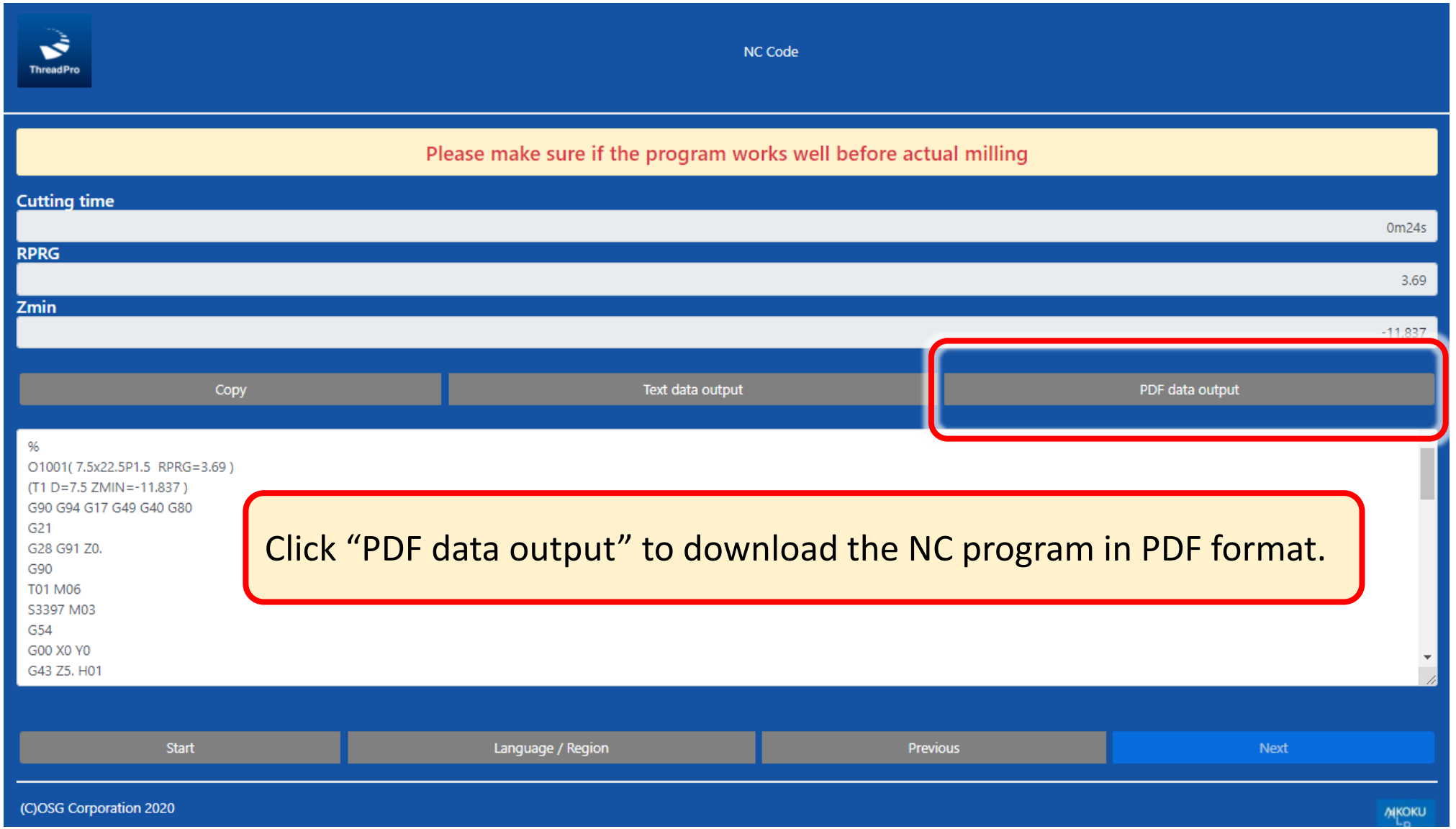


Function 2 (Download NC program)



Click “Text data output” to download the NC program with the file name “sample.nccode”.
Change the extension of “nccode” to convert the other files. (Ex. “Sample.txt” etc.)

Function 3 (Download in PDF)



shaping your dreams