

Mobile CAM CNC

Important informations

Read before using!

A. Get started:

- If your machine does not support coolant through, put a value of "0" in the coolant input in the settings section. However, if your machine does handle coolant through, select the appropriate value for your machine (usually "17" (M17) or "50" (M50) for FANUC, "88" (M88) for HAAS).
- Set a tool approach Z distance and the tool retract distance.
- In the "prepare" section, select the appropriate material if you want to use any "Smart Cycle"!
- Set a work offset.
- Set a safety distance (this is the value of the horizontal approach distance between the tool and the workpiece).
- **Warning!** Only the information about the thickness of the workpiece is really important, other information such as the shape of the workpiece and the beginning and end of the workpiece is only to represent the shape of the workpiece.

B. Inserting tool data:

The meaning of the color of the tool item:

- green:tool occupied
- yellow:tool inaccessible
- red:warning (from ver:1.3.1)
- purple:auto-generated tool (from ver:1.3.1)

- Tool storage is limited to 30 tools by default. The number of tools can be increased or decreased in the "Settings" section. The maximum tool storage capacity is 50 tools.
- The tools quantity in the storage should match the machine you want to generate the program on.
- The correct drill angle must be entered for drills as it is required to calculate the correct allowance! If you do not want the ATSA algorithm to calculate an allowance, enter "180".
- The operational length of the tool must be less than the tool length!
- **Warning!** The operational length of a chamfering tool, unlike other tools, is the vertical length between the smallest cutting diameter and the largest cutting diameter! This value is important to the ATSA algorithm for calculating the correct tool position and selecting the correct milling strategy!
- **Warning!** The operational length of tools such as: Drills, taps must be greater than the depth value of the selected operation!
- If you do not want the tool to be taken into account by the ATSA algorithm, disable it by selecting "Inaccessible".
- If you want to keep a tool as a tool that is permanently placed in your machine magazine, select "Permanent". In this case, after deleting all operations, the tool will remain in the tool storage.

C. Program:

Interesting information:

1. If you have already used a cycle in your program and you want to use another cycle of the same category, the new cycle will remember the settings from the previous one, all to increase the speed of program creation.
2. Did you know you can set a math expression in the "start angle" field? Thanks to this function, if you will use more cycles from the same category, you do not need to calculate or remember the distance between them, you just need to add a new value to the old one (for example: $90 + 45$, when you open a new cycle it will be: 135, so you can add new value and so on).

Warning!

The order of operations will be changed by the algorithm! For example, if you add Smart Tapping first and face milling next, Mobile CAM CNC will perform face milling first and Smart Tapping next. **To know more, read the sections “Code and code formating”.**

Some of the most interesting cycles:

1. Using “Smart Tapping”:

How the Smart Drilling and Smart Tapping cycle works:

- adds a central drill, if HSS drill is used
- adds a chamfer operation by default (the user can disable it by setting the value to “0”)
- generates an intermediate drill for deep drilling (if the drilling is deeper than 20 x the diameter of the drill bit)
- selects a right canned cycle depending on the working depth and the type of drill bit
- selects the coolant flow according to the drill bit used
- selects the best drill bit among the tools in the tool storage
- calculates speed rate and feed rate depending on material, depth and drill bit used
- adjusts the feed rate and feed rate depending on the workpiece materials, working depth, etc.
- assigns the required depth to the operation preceded by the main operation, e.g. adding a chip allowance and / or an allowance for a drill angle
- reacts to any change in the tool storage, to a change in workpiece thickness or to change in the “settings” panel
- if any tool is automatically generated or deleted by the cycle, a new notification will be sent

- Select the desired name of your tap, to make it easier you can use "Tap standard filter" to select the appropriate tool category.
- If you want to get the thread going through the workpiece, enter the same depth as the thickness of the workpiece (for example: the thickness of the workpiece is 20mm (in imperial 0.7874 inch), enter -20 (in imperial -0.7874 inch)), the ATSA algorithm will add the appropriate allowance needed to achieve it.
- The chamfer operation is set to -1 mm by default (in imperial -0.0393 inch). If you do not want to perform a chamfer operation, set the value to "0". You can also set a deeper value for the chamfer operation, but remember that the absolute value should not be greater than the operational length of the chamfer tool (for example: if the operational length of the chamfer tool is 15 mm (in imperial 0.5905 inch), the chamfer operation cannot be deeper than -15 mm (in imperial -0.5905 inch)).
- If you want to start the operation below the workpiece zero, set a negative value for "Origin Z shift", this will also recalculate the retraction (R) value.
- If you want to start the operation above the workpiece zero, set a positive value for "Origin Z shift", this will also recalculate the retraction (R) value.
- **Warning! "Smart Tapping" automatically generates all the necessary tools if it does not find the right tools in the tool storage!** Obviously, if you want to use a different tool than the auto-generated one, you can do so by: First of all, add a new tool that you want to use instead of the auto-generated tool and then remove or make inaccessible auto-generated tool.

2. Using "Smart Drilling":

- **Warning!** To use this cycle, you must have the drill or bits already in your tool storage!
- Select the desired drill bit from the tools offered by the tool recommendation system by pressing the tool number button. Information about the selected tool will appear above.
- The rest of the steps are the same as for "Smart Tapping"

3. Using face milling cycle:

- **Warning!** To use this cycle, you must have the right tools already in your tool storage!
- If you have added an allowance to a workpiece ("Z" value greater than "0"), it will be displayed in the new open face milling cycle.
- The coolant will be turned off automatically for the following materials
 - carbon steel
 - low carbon steel

- medium carbon steel
- cast iron
- ductil cast iron
- plastic

D. Graphics:

"MENU" - open the sidebar

"COOR" - display the coordinates of the selected point

"MEAS" - measure the distance between two points

"SIML" - start the simulator

"PATH" - display the tool path generated by the cycle

"MOVE" - view more additional features

"OPLC" - go to the cycle settings of the selected point

E. Code and code formating:

The way the code is formatted by the algorithm:

1. Face milling:

- Roughing
- Finishing

2. Shoulder milling:

- Roughing
- Finishing

3. End milling:

- Roughing
- Finishing

4. Center drilling:

- Roughing
- Finishing

5. Drilling

6. Boring

- Roughing
- Finishing

7. Reaming

8. Chamfer milling

9. Tapping

- **The Z value is red to get attention immediately due to the highest cause of collision with the wrong value**
- The default settings for the Code view:
 - highlighter - on
 - next tool - on
 - line number - on
 - decimal point - on
 - comments - on
 - header comments - on
 - line number increment - 1
 - generate code mode - subprogram
- **Warning!** The code will be saved as it was in Code view, but the code won't be colored!
- **To see the content of a subroutine, touch the name of the subroutine in Code view**
- the code is formatted by the NC (natural code) algorithm to reduce the amount of code repetition