Shopfloor Programmer for EdingCNC (V0.3)

Shopfloor Programmer for EdingCNC is a tool for easy and fast programming of basic shapes at the machine. It is also possible to create a file with a series of shapes in a row. This enables the processing of multiple workpieces without having to enter each of the shapes again every time.

Disclaimer:
Shopfloor Programmer for EdingCNC V0.3 is Beta software and may contain errors. Keep this in mind when using the software. Always use it in a way that unexpected movements or automatic starts can not cause any damage or injury! The use of "Shopfloor Programmer for EdingCNC" is at own risk! The creators shall not be liable for any special, incidental or consequential damage.

Shopfloor Programmer for EdingCNC V0.3 has been developed and tested on EdingCNC version v4.03.xx. The software may be freely distributed but , but may not be sold or otherwise commercially exploited.

1.0 Installing the software (summary for Experts):

1.1 Add the Shopfloor code as in the file "Shopfloor-code.cnc" to the file macro.cnc

1.2 Enable the extended user buttons in the cnc.ini file and replace the EUB19 to EUB32 calls in cnc.ini by the text given in the file "Shopfloor EUB19-32.txt"

1.3 Add the pictures from the directory "Shopfloor Dialogpictures" to the directory dialogpictures in the EdingCNC installation directory.

1.4 Copy the icons in the directory "Shopfloor Icons19-32" into the user icons directory of the EdingCNC software (e.g.: C:\CNC4.03\icons\op_f_key\user\).

1.5 Start EdingCNC and go to the tab "Setup", page 2. Enter in the field "file name to load" _shopfloor_teach.cnc and enable "watchFileChanged and "load automatically". Restart EdingCNC to activate these changes.

2.0. More extended installation description:

2.1. Add Shopfloor code to the file macro.cnc

First; make a copy of your current macro.cnc and cnc.ini files! This may be in the same directory but with just a slightly different name. It will allow you to restore the original situation if things do not work out as expected.

De file Shopfloor-code.cnc contains the code required to run the Shopfloor Programmer. All text in this file should be copied into the file macro.cnc. It is not important where you put this code but it makes sense to add it at the end, behind the existing code.

You can find the macro.cnc file in the EdingCNC installation directory. Open as well Shopfloor-code.cnc as macro.cnc in e.g. Notepad and copy the content of Shopfloor-code.cnc into macro.cnc

2.2. Enabling the Extended User Buttons

The code assumes the use of the "Extended User Buttons" (EUBs). Under function key F12 comes then a sub menu with 40 extra user buttons. This is desired because Shopfloor Programmer needs 14 User Keys. If you prefer to have the functions (or a part of them) under the standard used buttons, you need to do a some re-programming yourself.

The Shopfloor functions are assigned to EUB19 to EUB32. Therefore EUB11 to EUB18 remain available. This is of importance if User Buttons below the drive enable buttons are enabled (e.g.: showLeftUser11Button=1 in the cnc.ini file).

Enabling and assigning the Extended User Buttons is done in the cnc.ini file. This file is in the EdingCNC install directory. Take care: the EdingCNC software needs to be closed in order to make changes to the cnc.ini file.

Open cnc.ini in e.g. Notepad and search for "useExtendedUserButtons". Set this to "1". Scroll down to EUB19 and delete the lines for EUB19 to EUB32. Open de file "Shopfloor EUB19-32.txt" and copy the content of this file to where the EUB19-32 calls were. Now save cnc.ini.

2.3. Add the Shopfloor Dialogpictures

Go to the directory dialogpictures in the EdingCNC installation directory. Copy the figures from the directory "Shopfloor Dialogpictures" to this directory. In case of a new installation there should be no files to be overwritten.

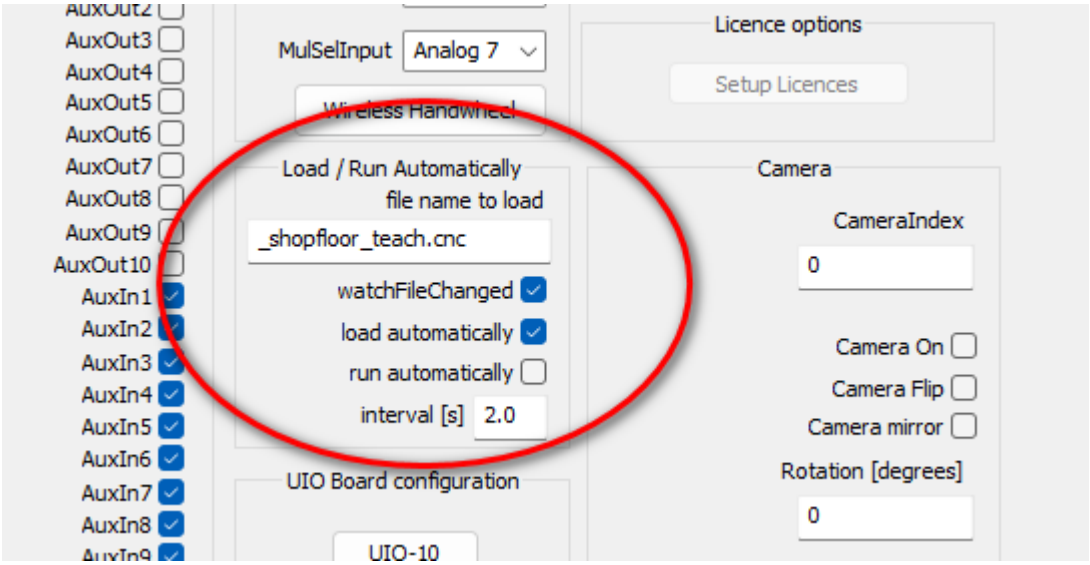
2.4. Adding the Shopfloor user icons

Go to the directory where the "op_f_key user icons" are. This is a subdirectory directory of the EdingCNC installation directory, e.g. : C:\CNC4.03\icons\op_f_key\user\
Copy the .bmp files from the directory "Shopfloor Icons19-32" to the EdingCNC user icons directory. Existing icons should be overwritten. If you like you can first make a back-up of the user icons directory.

2.5. Changing the configuration

Some adjustments in EdingCNC are needed to make Shopfloor working properly. It is necessary to adjust the Load/Run automatically settings. The file that is created by Shopfloor is: _shopfloor_teach.cnc To load this file automatically you should check the watchFileChanged and load automatically flags. This makes that every time the G-code is generated, the code is immediately loaded and ready to run.

Example of the configuration changes:

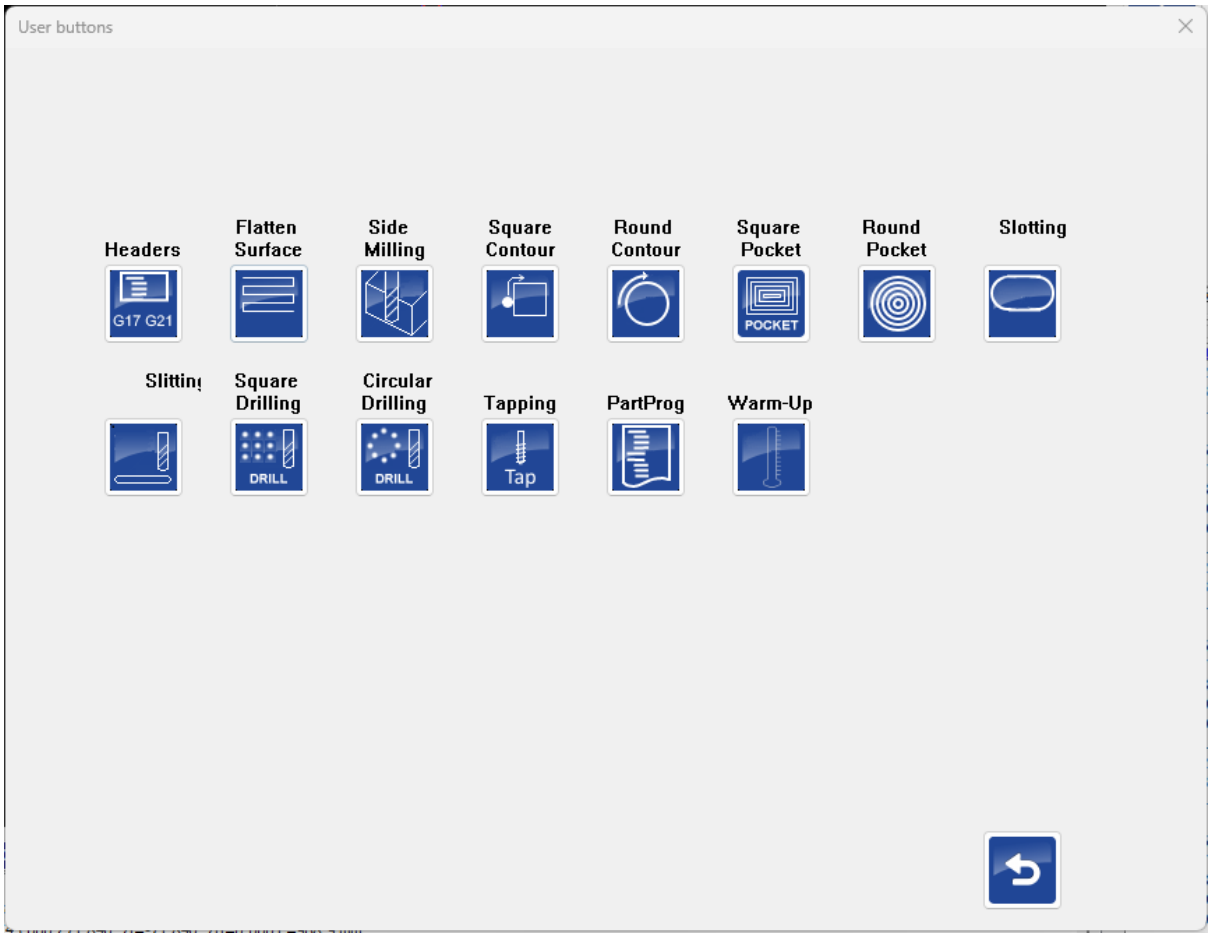


Now Shopfloor Programmer is ready to be used. Re-start Eding CNC to activate the new configuration. Preferably start in simulation mode the first time. (safety first! Disconnect the EdingCNC interface to force simulation mode).

Check if there are any error messages. Solve this first before continuing.

3.0. How to use Shopfloor Programmer

By pressing the user-menu F12 button you will get the extended user button menu as shown below. Here you can select the macro you want to use. The Extended User Button menu will look like this:



In the empty spaces you may see unused buttons with a number. This doesn't matter but if you like to remove them, you can do this by disabling them in the cnc.ini file.

Each of the macros can be used independently from others. The parameters entered for each of the macros are saved in the cnc.ini file, so they will appear a next time you run the same macro, also after power off.

General rules valid for each macro:

- Fields without indication of units like (rpm) or (0..1) are in millimeters.
- Entering a valid tool number is for most macros mandatory because of the radius compensation. The radius compensation is calculated in the macro, G41 or G42 should not be active.
- A stepover is entered as a number between 0 and 1 (must be >0). This is a factor of the tool diameter. For example if the tool diameter is 6mm and the stepover value is 0.5 the actual stepover distance will be 3mm.
- The kind of coolant is entered in the field "Coolant" as: 0=none, 1=Mist(M7) and 2=Flood (M8)
- If, at the start of a macro the actual Z-position is below the safe-Z level of the macro, the Z-axis will move to the save-Z level first before moving to the X/Y start position. If the actual Z-position is higher than safe-Z the machine goes to the X/Y start position first and then lowers to the safe-Z level.
- The minimum safe-Z value that can be entered is "Z start" + 1mm
- Each of the macro dialog-panels have a picture with additional information about the parameters to be entered and an indication of how the milling path could look.

4.0 Available macros

Header (and footers)

This allows to put a standard set of G- and M-code commands in front, and/or behind a series of macros stored in the file _shopfloor_part.cnc
These headers and footers are fixed in the Shopfloor code. Changes or additions have to be made in the Shopfloor code in the file macro.cnc file.

Flatten

To flatten the topside of a workpiece. In the field "direction" you can select to flatten in the X direction, in the Y direction or leave it at "0" and the direction will automatically be in the direction of the longest side of the workpiece. The X start and Y start fields are the lower left corner of the area to flatten. X size and Y size need to be positive numbers.

SideMilling

This is to mill the sides of a workpiece. It is useful to make a side flat, but can also be used to mill a step or an open slot at one of the sides. In the field "side" you should enter the side to be milled where 1 is left, 2 is top, 3 is right and 4 is lower side. The milling direction is always climb milling, this determines the X & Y starting position (see figure at the SideMilling dialog picture). X size and Y size need to be positive numbers and always go from the starting point along the workpiece and towards the center of the workpiece.

Square Contour
A square or rectangular contour with the choice of inside or outside radius compensation. Useful to make holes in sheet material or relative thin workpieces (inside compensation). Or to free up stock material from a sheet (outside compensation). It also allows to finish the outside of a square or rectangular workpiece.

Round Contour
A round contour with the choice of inside or outside radius compensation. Useful to make round holes in sheet material or thin workpieces (inside compensation). Or to free up stock material from a sheet (outside compensation). Can also be used to make the outside of a cylinder at size.

Square Pocket
Creates a square or rectangular pocket using a helix lead-in in the center.
Take care, the diameter of the helix lead-in is: [tooldiameter + tooldiameter * stepover]. If the X size or Y size is smaller than the helix, the outline of the helix will be visible in the pocket walls.

Round Pocket
Creates a round pocket using a helix lead-in in the center.
The diameter of the helix lead-in is: [tooldiameter + tooldiameter * stepover]. This means the diameter of the pocket cannot be smaller than this.

Slotting
Milling a slot in the X or Y direction depending on which is the longest size. The start position is in the center of the slot.

Slitting
The slitting function makes a straight cut from position X-start, Y-start to position X-end , Y-end. This is useful to cut sheet material in pieces, make an angled corner at sheet material or make slots with the diameter of the tool.
Note; there is no radius compensation. The entered path is the centerline of the tool and also the X/Y start and X/Y end position is at the center of the tool

Square Drilling
Drills holes using the G83 canned cycle (peck drilling). X start and Y start assign the position of the first hole. From this position it is possible to drill just a single hole, a line of holes in the X or Y direction or a matrix of holes.
Example 1: X increment = 10, Y increment is random, but bigger than the tool diameter! X number of holes = 6 and Y number of holes = 1. This results in a single row with 6 holes in the X direction with 10mm spacing between the holes.
Example 2: X increment = 20, Y increment = 20, X number of holes = 6 and Y number of holes = 6. This results in a matrix of 36 holes with a spacing of 20mm between the holes.
X increment and Y increment need to positive values and at least the tool diameter.
X nr. of holes and Y nr. of holes must be 1 or higher.

Circular Drilling
This allows to drill holes in a circle or on an arc. The field "Start Angle" is the angle of the first hole to the X-axis (counter clockwise). The field "Arc Angle" is the angle of the arc the holes should be drilled on. For a full circle enter 360 degrees.
When drilling an arc the first and last hole are at the arc ends. When drilling a full circle (arc angle 360 deg) the holes are evenly distributed over the circle.
The field "diameter" cannot be less than the tool diameter. The minimum number of holes is 2

Tapping
Making a thread using G84 for "Ridgid Tapping". This allows to tap holes with the milling spindle. A spindle sensor / encoder needs to be available to use this.

PartProg
After a macro function has been programmed you may select "PartProg" to add this macro to the file _shopfloor_part.cnc. Enter a "1" in the field PartProg to start a new file or "0" to append this macro to the file _shopfloor_part.cnc. After pressing OK in PartProg the last used macro shows up again (you still can make changes) and when you press OK for this macro it will be added to the file: _shopfloor_part.cnc.
To run the file _shopfloor_part.cnc go to the "Auto" menu in EdingCNC and load it just as any other CNC job. All macros will be rendered and you can check the result on correctness.
Note: after executing a G-code file in the "Auto" menu it might be necessary to re-load the file _shopfloor_teach.cnc in order to make Shopfloor Programmer as before.

Warm Up
This warms up the Spindle an distributes the oil over the rails and ball screw spindles
The machine will first distribute the oil by moving all axis over there full travel range.
Then the Spindle is warmed up by running it in steps where each step the rpm's are increased. As last the homing sequence is repeated.

Take care: make sure the machine bed is empty when running warm up. Otherwise something may be hit when moving all axis over their full travel.

Shopfloor Programmer for EdingCNC was initiated in 2014 by Niels Saarloos and upgraded in the years after until V0.2.2 in 2016.
In 2022 it has been upgraded to V0.3 by Kars Schaapman. See the notes at the beginning of the Shopfloor Programmer code for more details about the extensions and improvements.

Date of last edit: 25 November 2022