



Эта страница пока что не переведена.

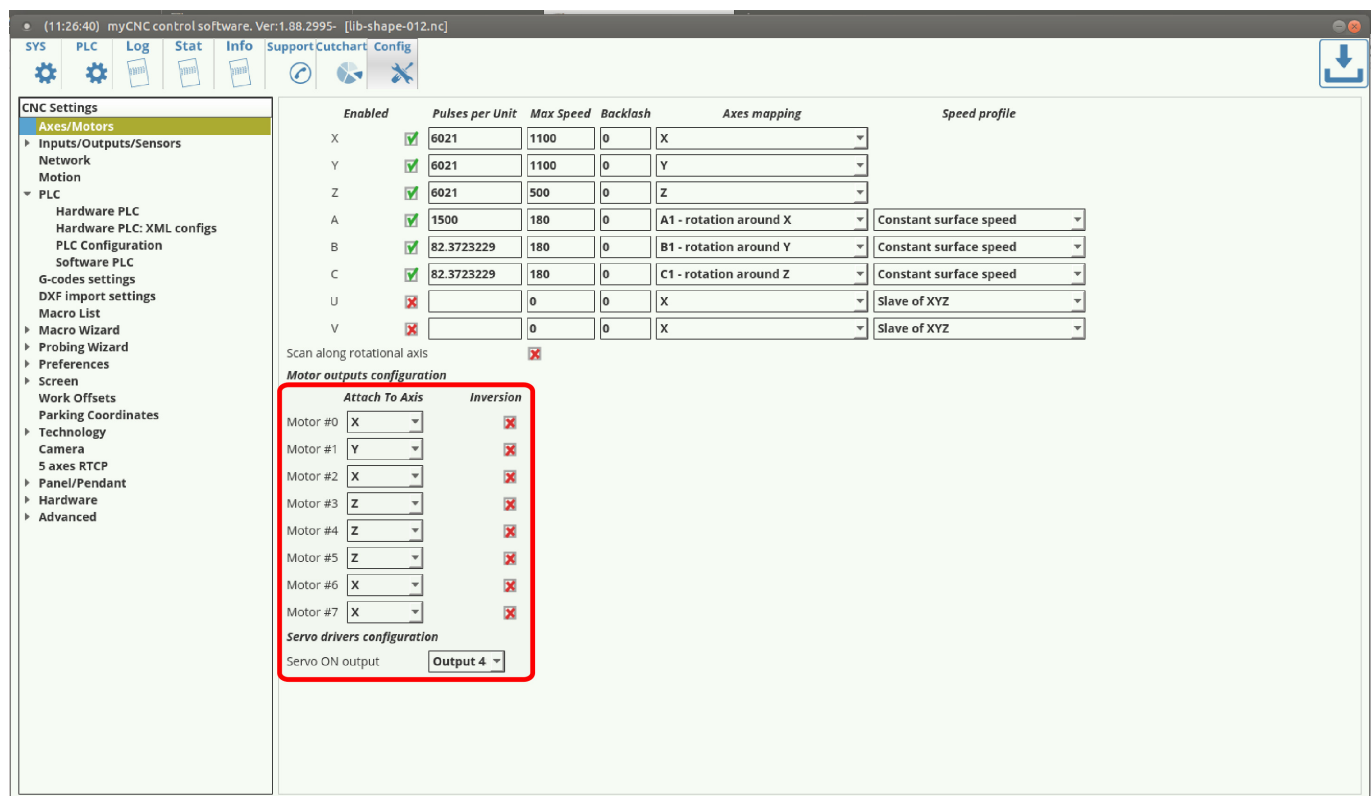
Примеры настройки myCNC

NOTE: The myCNC team recommends utilizing the examples provided in this manual (as well as other manuals in this documentation) as a starting point for your machine setup. When possible (and applicable), it is recommended to keep changes to a minimum. In general, using these examples as the basis for your PLCs/macro commands allows for an easier setup process.

How to set up Axes and Pulses per Unit

During the initial setup, the axes on your machine (X, Y, Z, A, etc) may be plugged differently from the desired software configuration. In order to assign the correct motor to the axis in the myCNC software:

1. Go to **CNC Settings > Axes/Motors**
2. Navigate to **Motor Outputs Configuration**
3. Assign the connected motors to the needed axes.

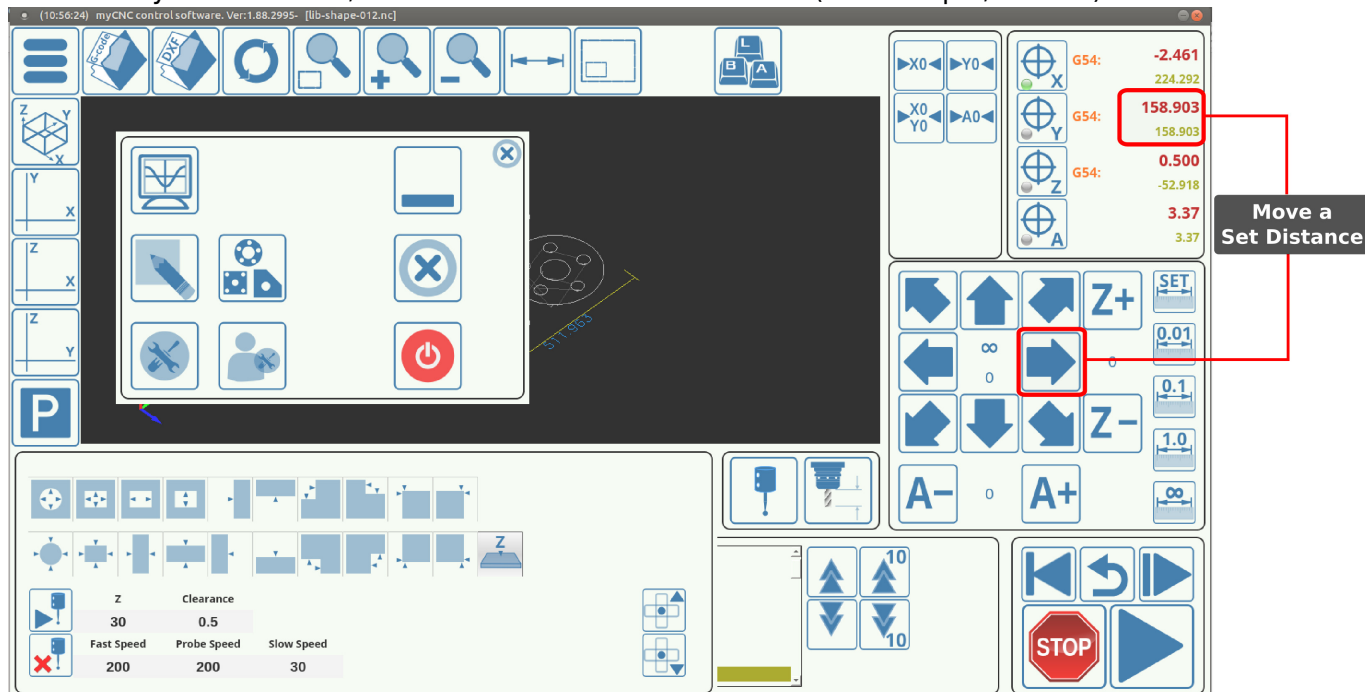


In the same window, the general calibration for the pulses per unit can be performed. This is done in order to calibrate the software readings of the machine (how much the machine “thinks” it has moved) versus the actual physical movements of the machine (how much it has actually moved). To do that:

1. Physically record the position of the spindle/knife/sensor/etc on the machine with a ruler, caliper,

camera attached to the machine arm, etc.

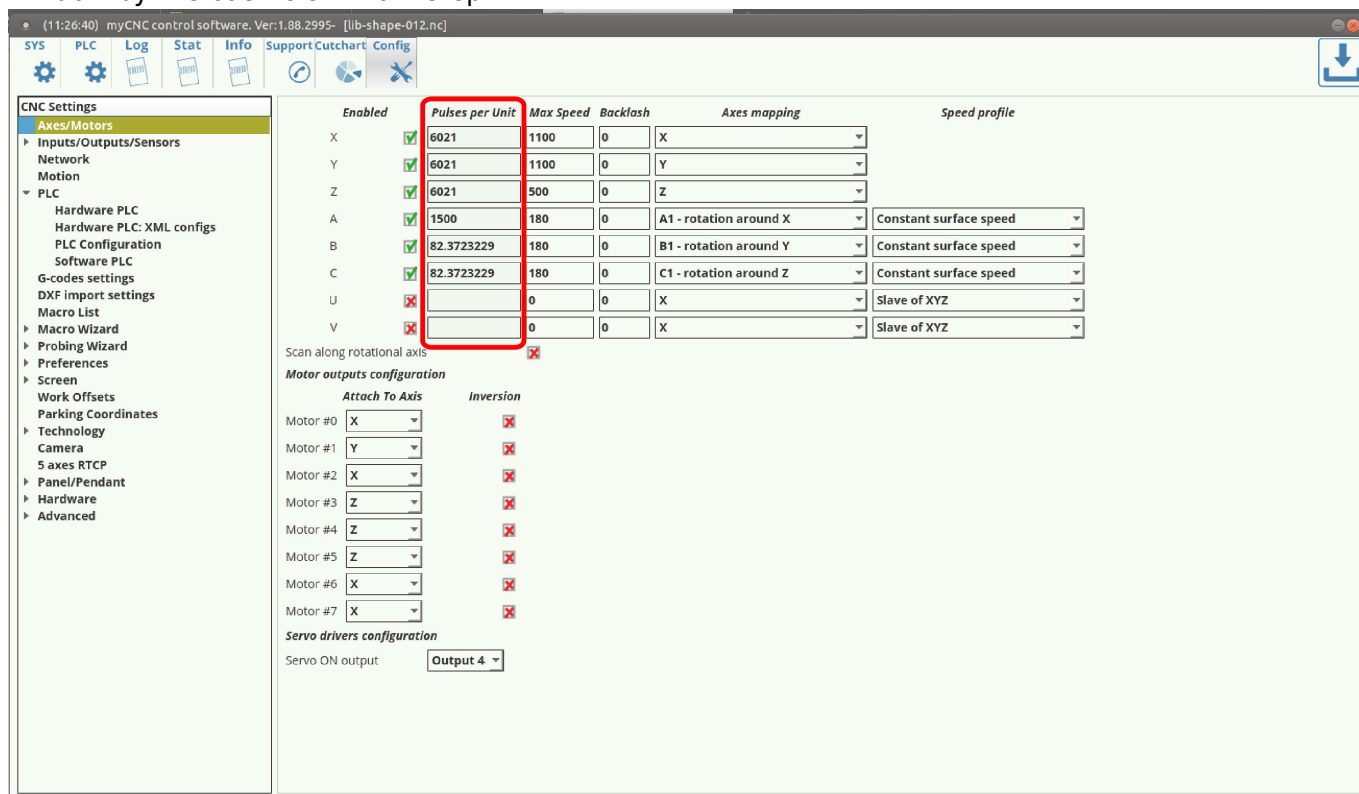
2. In the myCNC software, move the machine a set distance (for example, 50 mm).



3. Measure the new physical position of the machine and record the distance actually physically traveled.

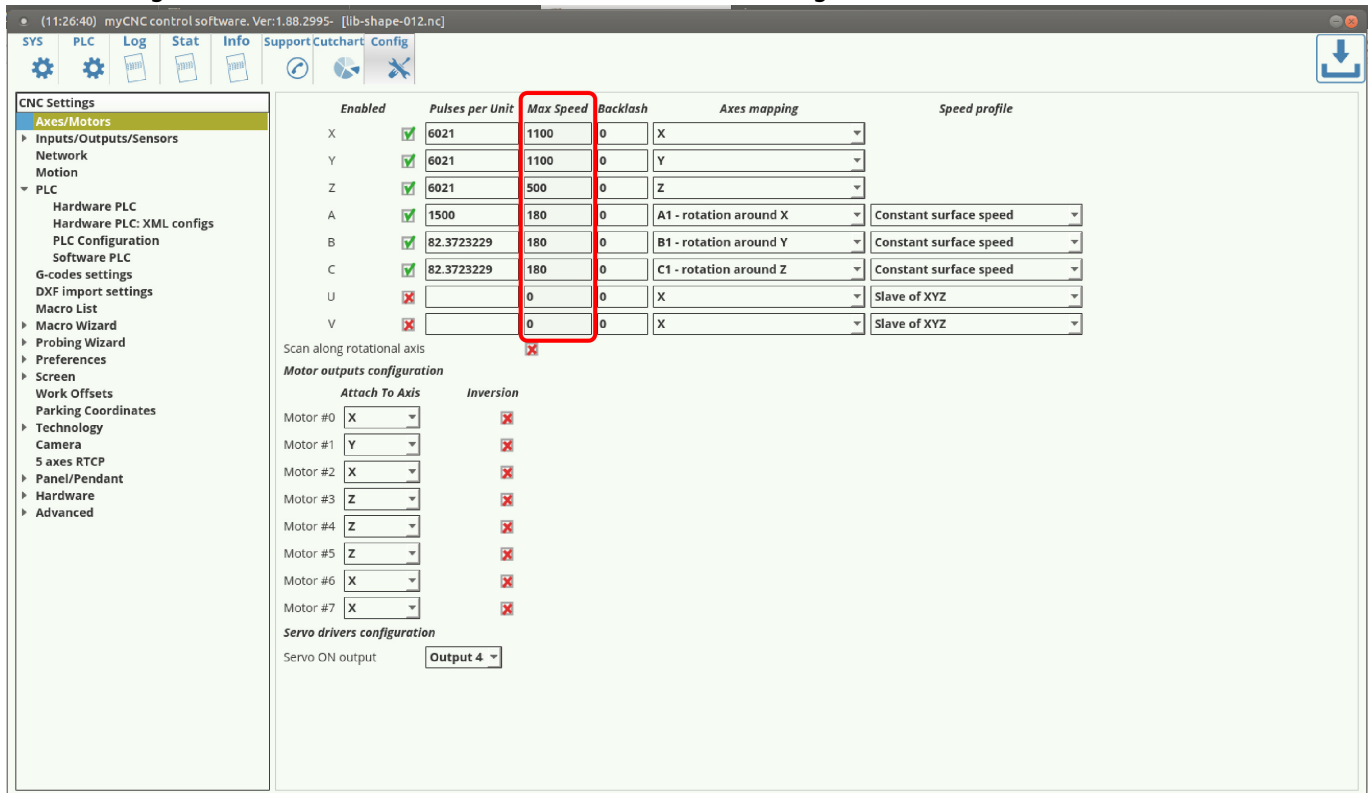
4. Divide the actual distance by the software distance (for example, 25 mm actually traveled divided by 50 mm that the software recorded)

5. Divide the number of **Pulses per Unit** from the myCNC software CNC Settings > Axes/Motors window by the coefficient from step #4.



6. Input the new number of pulses per unit into the software.

If the new number of pulses per unit is brought up too high without changing the maximum motor speed, the motors will start making jerking movements when moved. This is due to keeping the **Max Speed** the same. In order to eliminate the jerking movements, bring down the **Max Speed** value in CNC Settings > Axes/Motors until the movement is smooth again.



Max Speed is set in units/second (mm/s or inch/s), depending on what units you have chosen in your G-codes settings.

7. Set the Backlash in units (mm/inch) for each axis. Backlash is the value that allows for some unexpected play in the machine due to clearance or looseness of mechanical parts. When the controller sends a movement command to the motor, the motor may turn briefly before any actual axis movement begins. That turn is defined as backlash, and can be accounted for in this setting.

How to set up Tangential Knife Cutting

MyCNC control has built-in tangential knife control

Tangential knife control is activated by writing "1" to Global Variable #7005 (GVAR_TANGENTKNIFE_ENABLE)

If Tangential control is activated, 2D tool path programming is enough to make tangential knife cutting. myCNC control software automatically

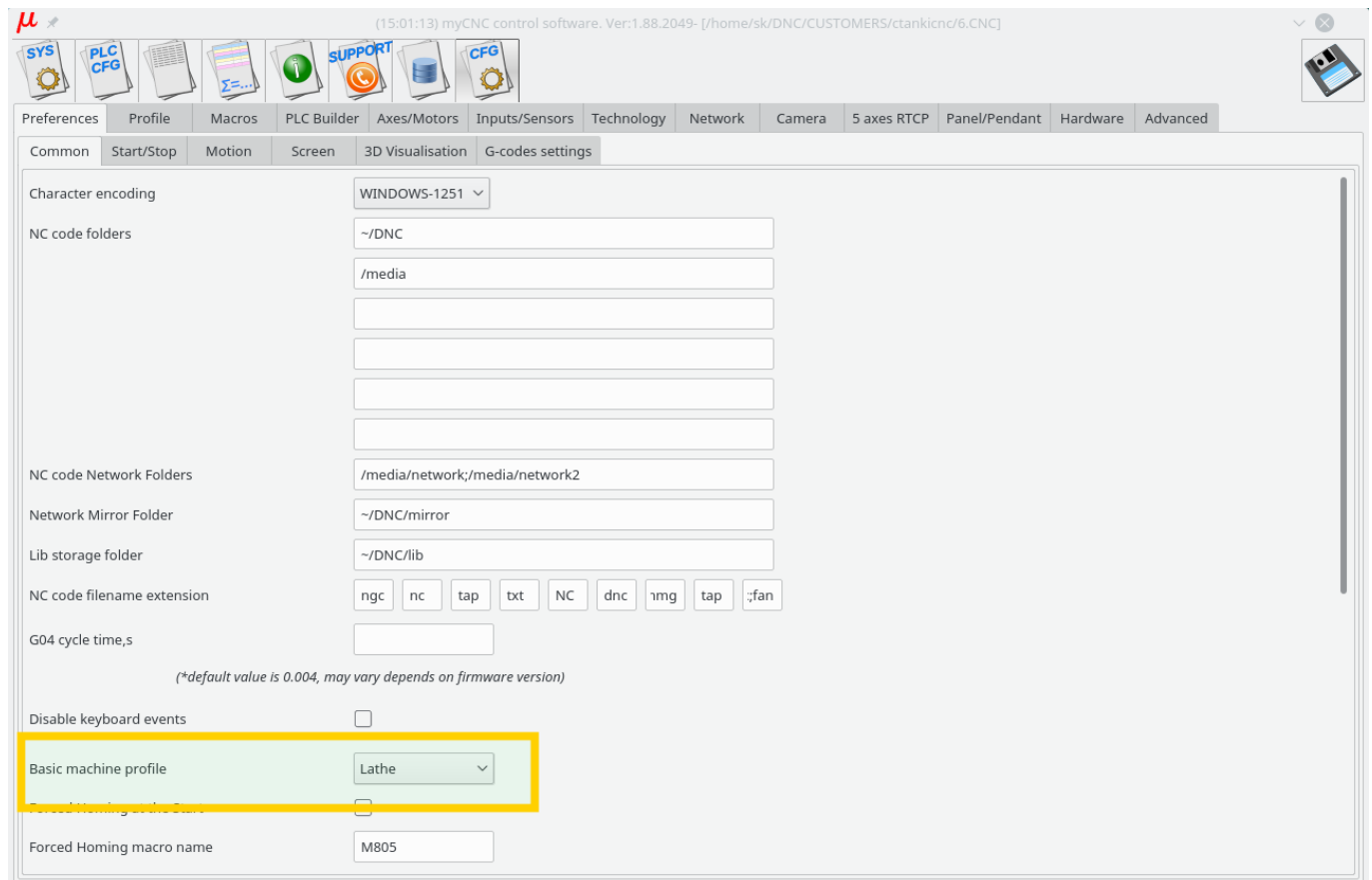
- Calculates the angle of next motion;
- Lifts up the knife to safe height;
- Rotate knife accordingly next XY motion direction;
- Moves down knife on working height.

How to set up a Probing Wizard in Mill profile

In order to properly set up a probe system on a mill profile from scratch on your machine (especially if the software has not, for any reason, been updated to its latest version) please consult the [QuickStart manual for the probe sensor setup](#):

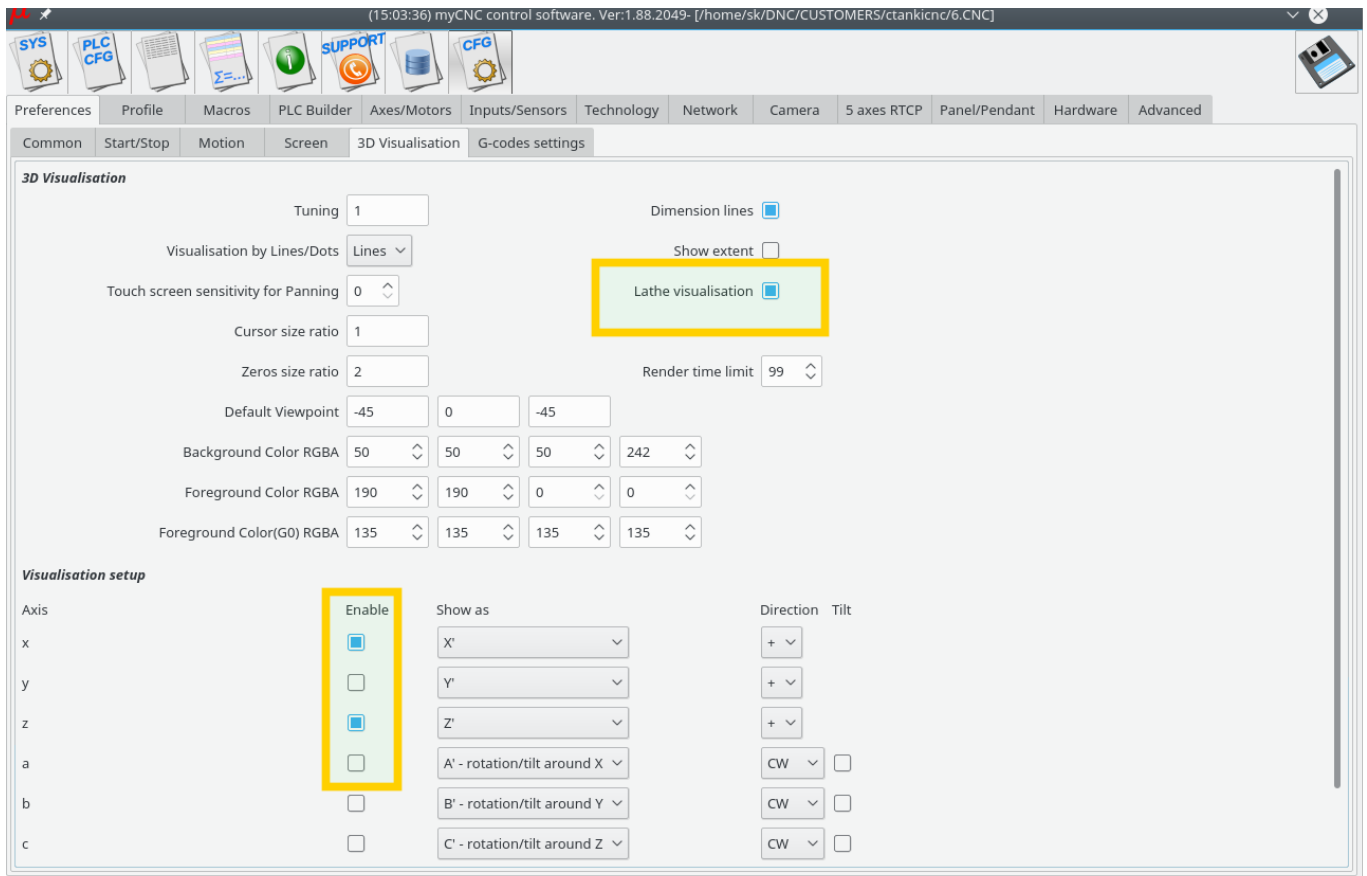
How to set up a Lathe/Turning machine

1. Select Basic profile as “Lathe” in **Cfg - Preferences - Common** dialog

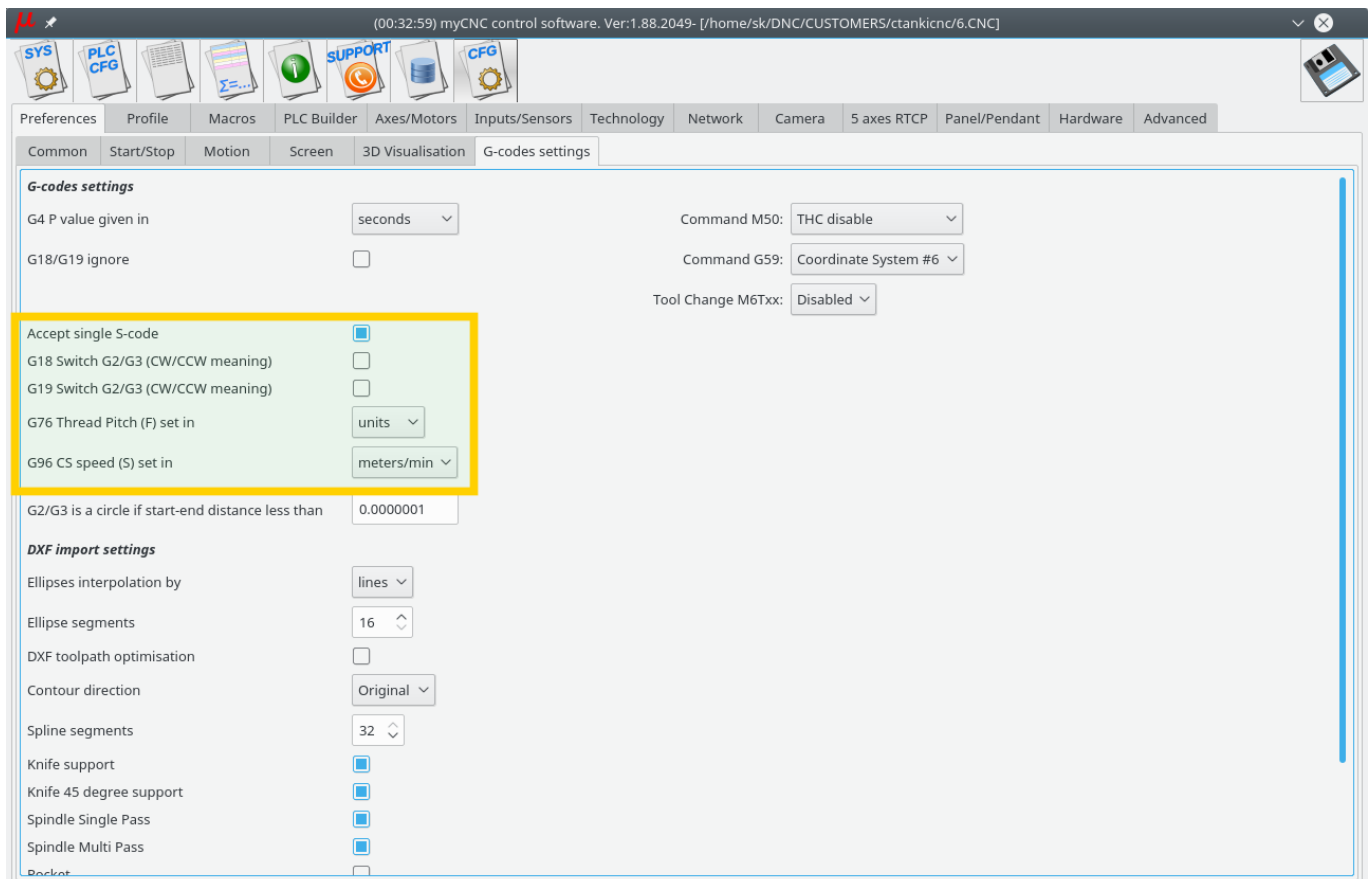


2. Select “Lathe Visualisation” in **Cfg - Preferences - 3D visualisation** configuration dialog

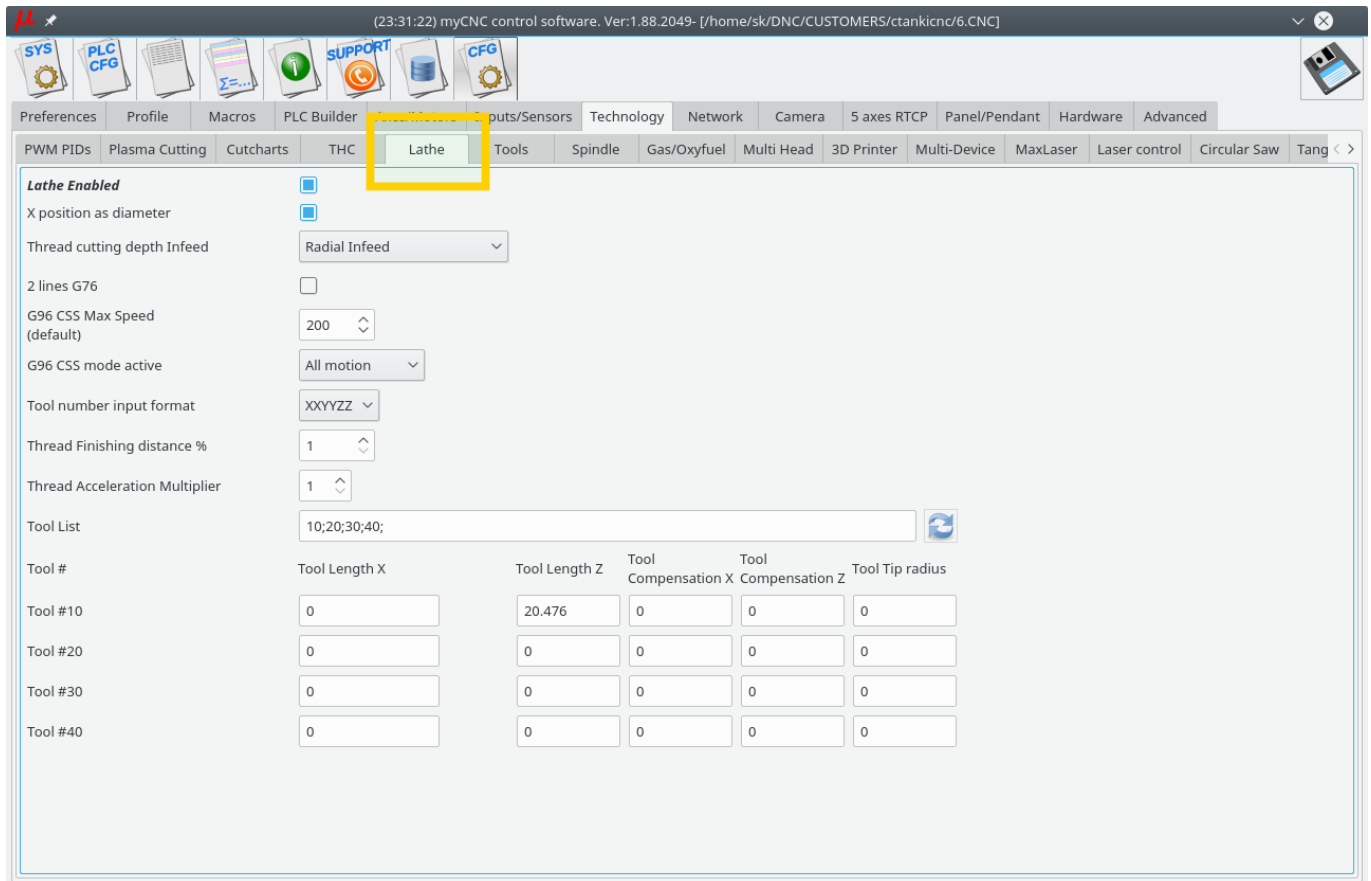
3. Select axes **X, Z** for visualisation and deselect the rest axes.



4. Check G-code settings related to Lathe operations in **Cfg - Preferences - G-codes settings** configuration dialog



5. Goto **Cfg - Technology - Lathe** configuration dialog and setup appropriate settings



How to set up a Multi-Tool router

We will put here some process of setup multi-tool router. The router has -

- 2x motors on X axes.
- 2x separate spindles installed on separate Z heads.
- Tangential knife head

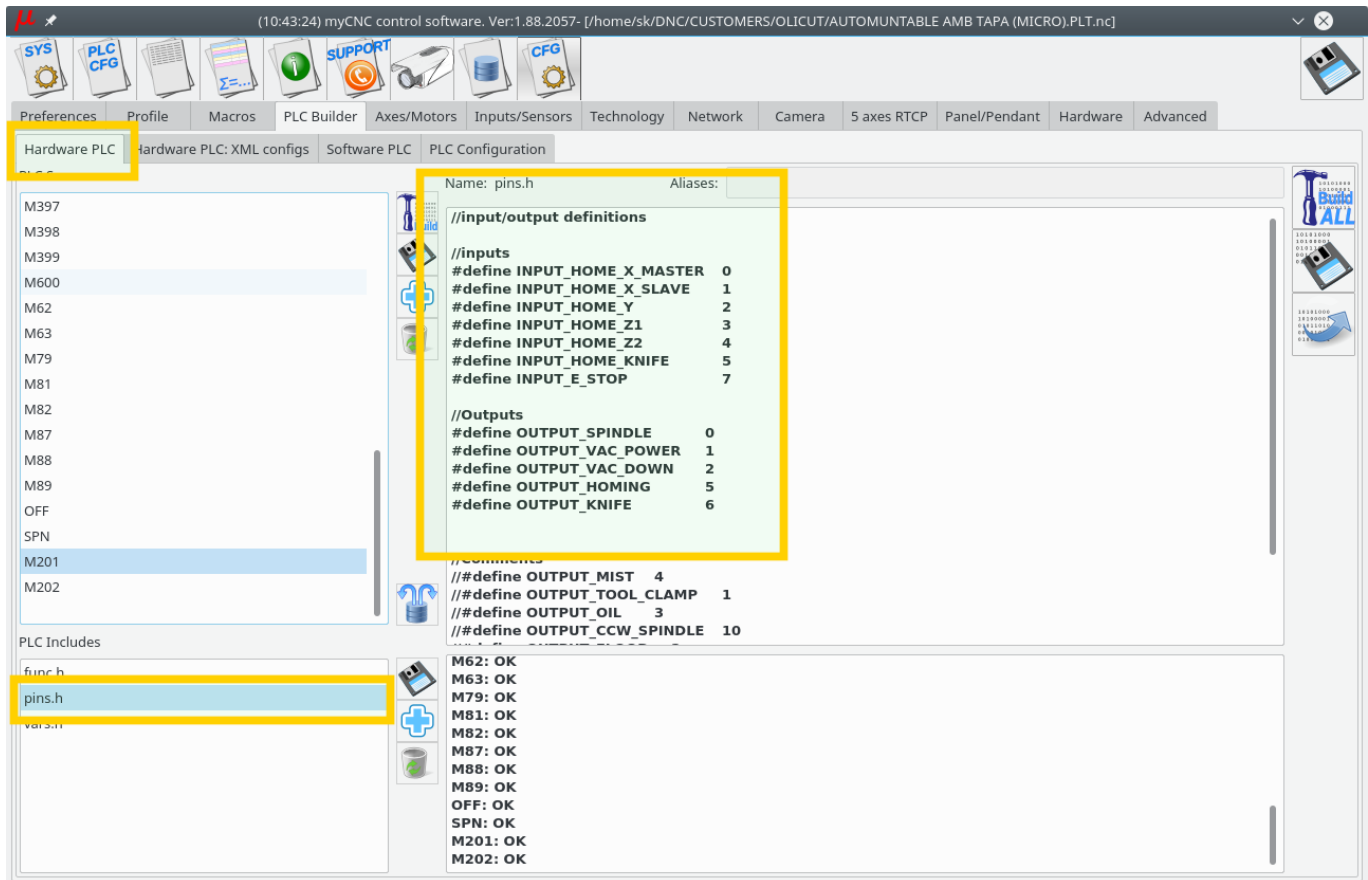
1. Inputs/Outputs assignment described on "pins.h" includes the file in **Hardware PLC Builder** area -

[pins.h](#)

```
//input/output definitions
//inputs
#define INPUT_HOME_X_MASTER 0
#define INPUT_HOME_X_SLAVE 1
#define INPUT_HOME_Y 2
#define INPUT_HOME_Z1 3
#define INPUT_HOME_Z2 4
#define INPUT_HOME_KNIFE 5
#define INPUT_E_STOP 7

//Outputs
#define OUTPUT_SPINDLE 0
#define OUTPUT_VAC_POWER 1
#define OUTPUT_VAC_DOWN 2
```

```
#define OUTPUT_HOMING      5
#define OUTPUT_KNIFE       6
```



2. How to switch between spindles (Z1/Z2) heads.

Axis pulse-dir signal can be connected/disconnected from **Motor output** by writing to CNC registers 0x70...0x75 (112...117)

- 0x70 (112) - Motor output #0
- 0x71 (113) - Motor output #1
- 0x72 (114) - Motor output #2
- 0x73 (115) - Motor output #3
- 0x74 (116) - Motor output #4
- 0x75 (117) - Motor output #5

Low 4 bits (0..3) of the writing value represent Axis to connect -

- 0 - X
- 1 - Y
- 2 - Z
- 3 - A
- 4 - B
- 5 - C
- 15 - disconnected

pulse-dir Direction will be changed (**DIR** signal inverted) if Bit #4 is set.

This way we add PLC procedures M201 and M202 to switch Z axis between Motor outputs #3 and #4

M201.plc

```
#include vars.h
main()
{
    parameter=15;    //OFF
    command=112+3;   //channel 3
    message=PLCCMD_SET_CNC_VAR;
    timer=2;do{timer--;}while(timer>0);

    parameter=2+16;    //Attach to Z
    command=112+4;   //channel 4 turning off
    message=PLCCMD_SET_CNC_VAR;
    timer=2;do{timer--;}while(timer>0);

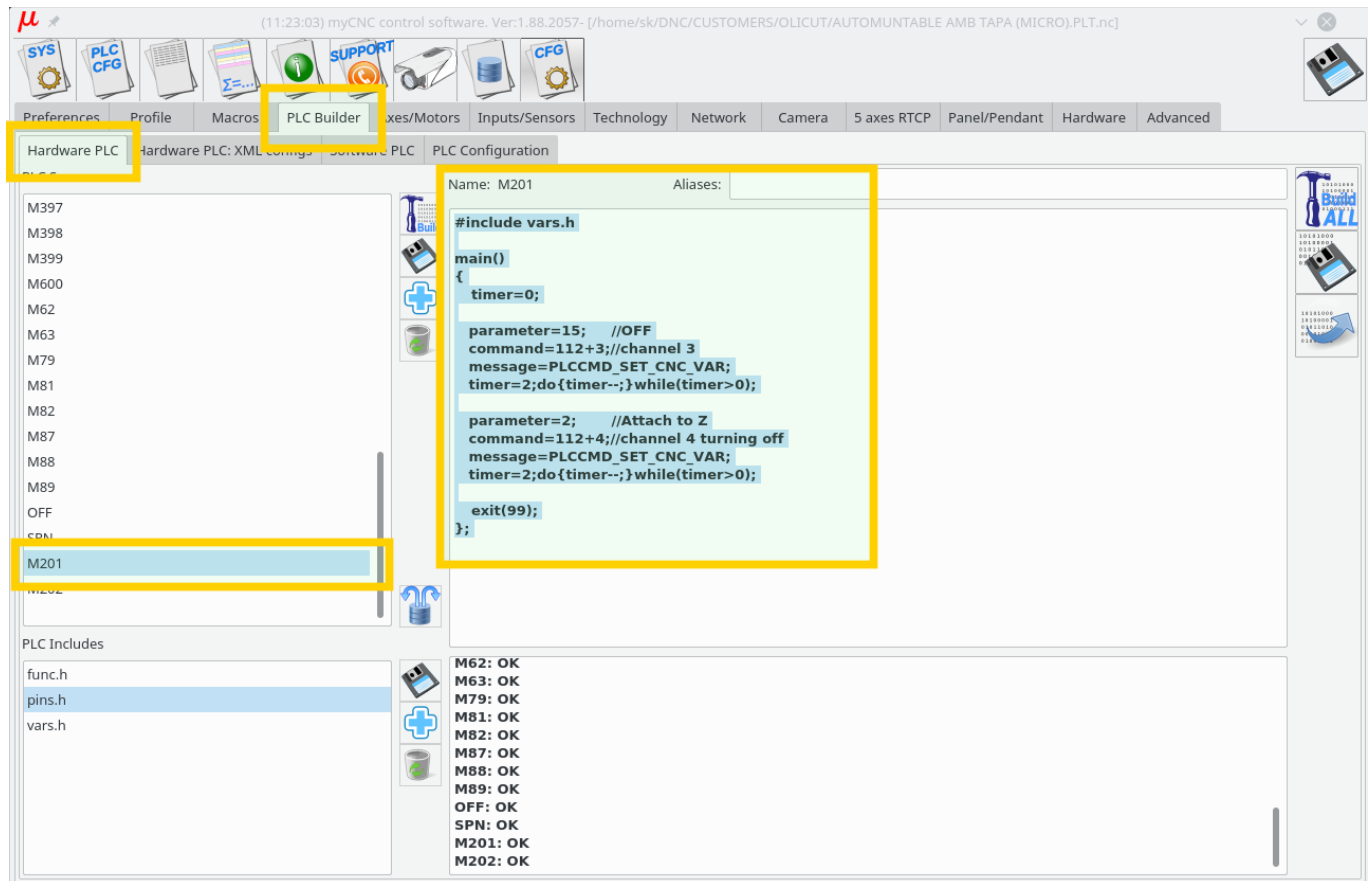
    exit(99);
};
```

M202.plc

```
#include vars.h
main()
{
    parameter=15;    //OFF
    command=112+4;   //channel 4
    message=PLCCMD_SET_CNC_VAR;
    timer=2;do{timer--;}while(timer>0);

    parameter=2+16; //Attach to Z
    command=112+3;   //channel 3
    message=PLCCMD_SET_CNC_VAR;
    timer=2;do{timer--;}while(timer>0);

    exit(99);
};
```



3. Homing for Z1, Z2 axes can be configured in Macro Wizard. **M133** macro is usually used for **Homing Z** procedure. We will use macro names M1331 and M1332 for 2 homing procedures for every Z axis.

M1331

```
M201 (Turn ON Z1 axis, OFF Z2 axis)
G10 L80 P5521 Q1
G10 L80 P5525 Q1
M88 L0 P3(Soft stop when sensor triggered)
G91 G0 Z 200.0000 F 600.00
G04 P0.1
M89 L1 P3(Quick stop when sensor triggered)
G91 G0 Z -200.0000 F 30.00
G04 P0.1
G91 G0 Z 1.0000 F 500.00
G90 G10L70 P0 Z #5453
G10 L80 P5521 Q0
G10 L80 P5525 Q0
G10 L80 P7393 Q0 (Homing Flag)
```

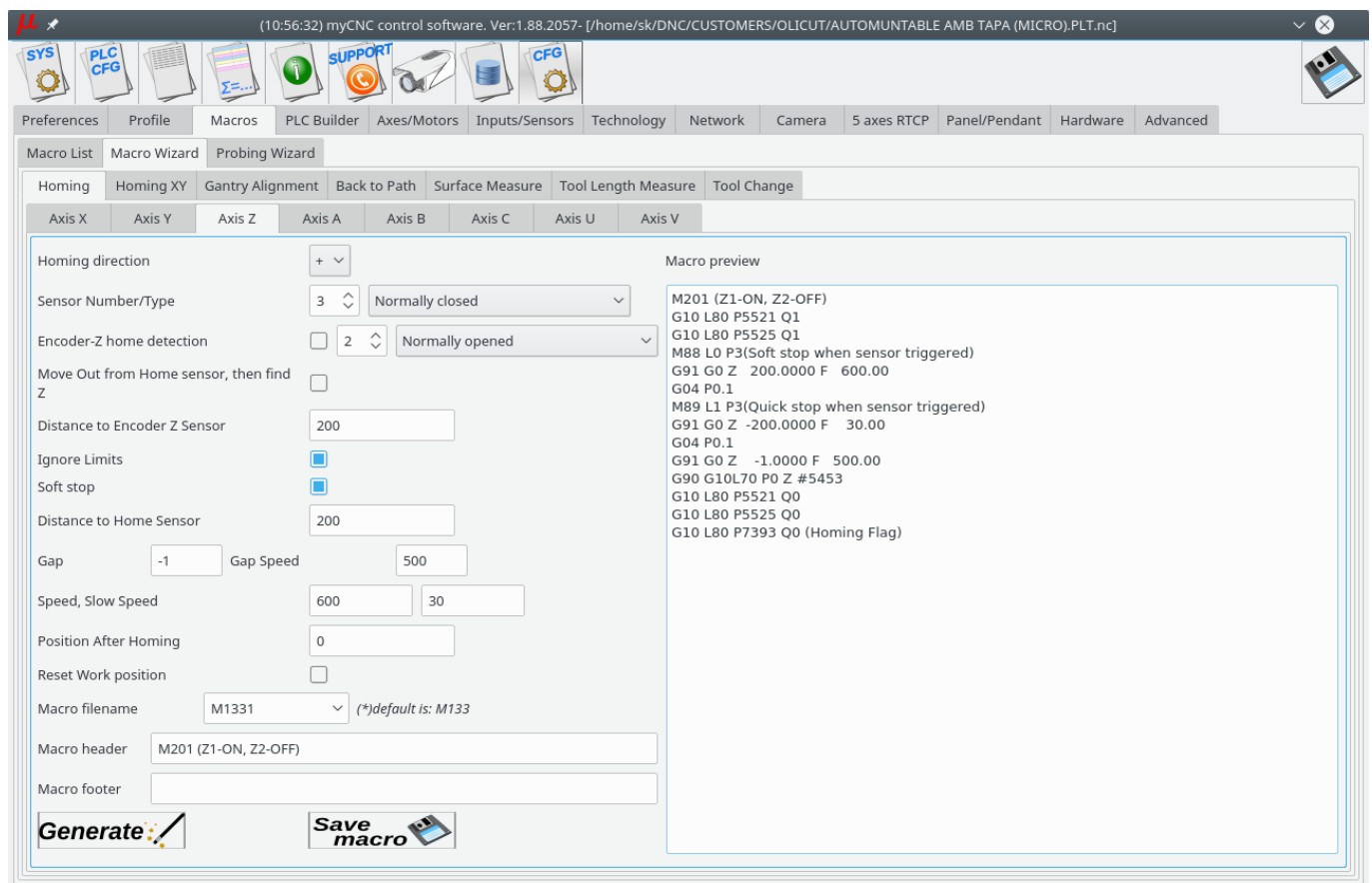
M1332

```
M202 (Turn OFF Z1 axis, ON Z2 axis)
G10 L80 P5521 Q1
G10 L80 P5525 Q1
```

```

M88 L0 P4(Soft stop when sensor triggered)
G91 G0 Z 200.0000 F 600.00
G04 P0.1
M89 L1 P4(Quick stop when sensor triggered)
G91 G0 Z -200.0000 F 30.00
G04 P0.1
G91 G0 Z 1.0000 F 500.00
G90 G10L70 P0 Z #5453
G10 L80 P5521 Q0
G10 L80 P5525 Q0
G10 L80 P7393 Q0 (Homing Flag)

```



4. **M138** macro is used for **Home-All** procedure. It should be fixed to support Z1, Z2, tangential knife homing and **OUTPUT_HOMING** output described in "pins.h"

M138

```

M62P5 (OUTPUT HOMING ON)

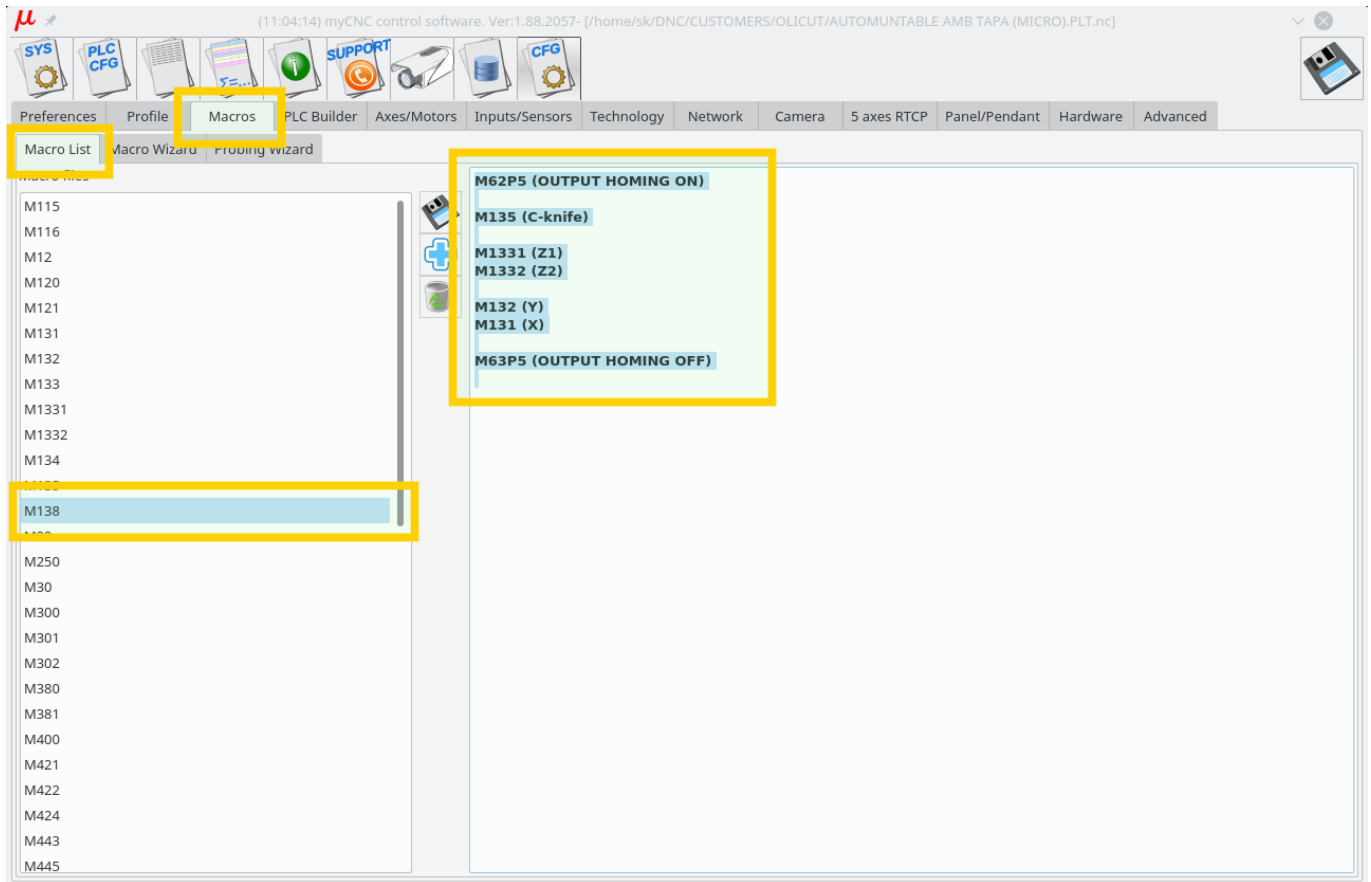
M135 (C-knife)

M1331 (Z1)
M1332 (Z2)

M132 (Y)
M131 (X)

```

M63P5 (OUTPUT HOMING OFF)



5. M6 - Tool Change macro for multitool configuration.

M6

```

M600 P#5409

if [ #5409 NE 1 ] 100
M150
GOTO 1000

N100
if [ #5409 NE 2 ] 200
M151
JUMP 1000

N200
if [ #5409 EQ 3 ] 300
G10 L80 P7005 Q0
JUMP 1000

N50
G10 L80 P7005 Q1
  
```

```
N300
```

```
N1000
```

```
G10 L81 P5400 Q5409      (set current tool number)
```

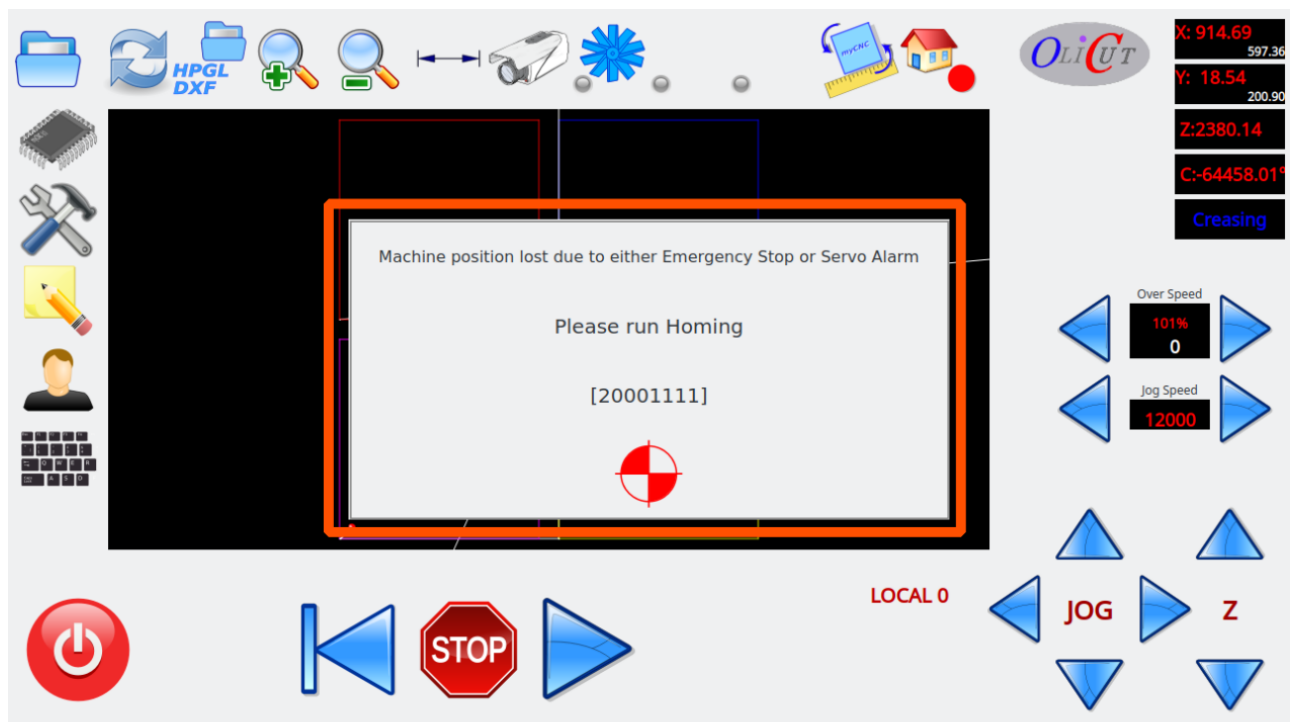
How to setup Homing procedure

Homing is a procedure to find the initial machine position (home position) by using Home sensors. The home procedure is a macro file. The macro filename is absolutely flexible, but we usually use a number of default macro names -

- M131 - Homing for the X axis
- M132 - Homing for the Y axis
- M133 - Homing for the Z axis
- M134 - Homing for A axis
- M135 - Homing for B axis
- M136 - Homing for C axis
- M138 - Homing all the axes (custom macro which does all the axes homing in series)

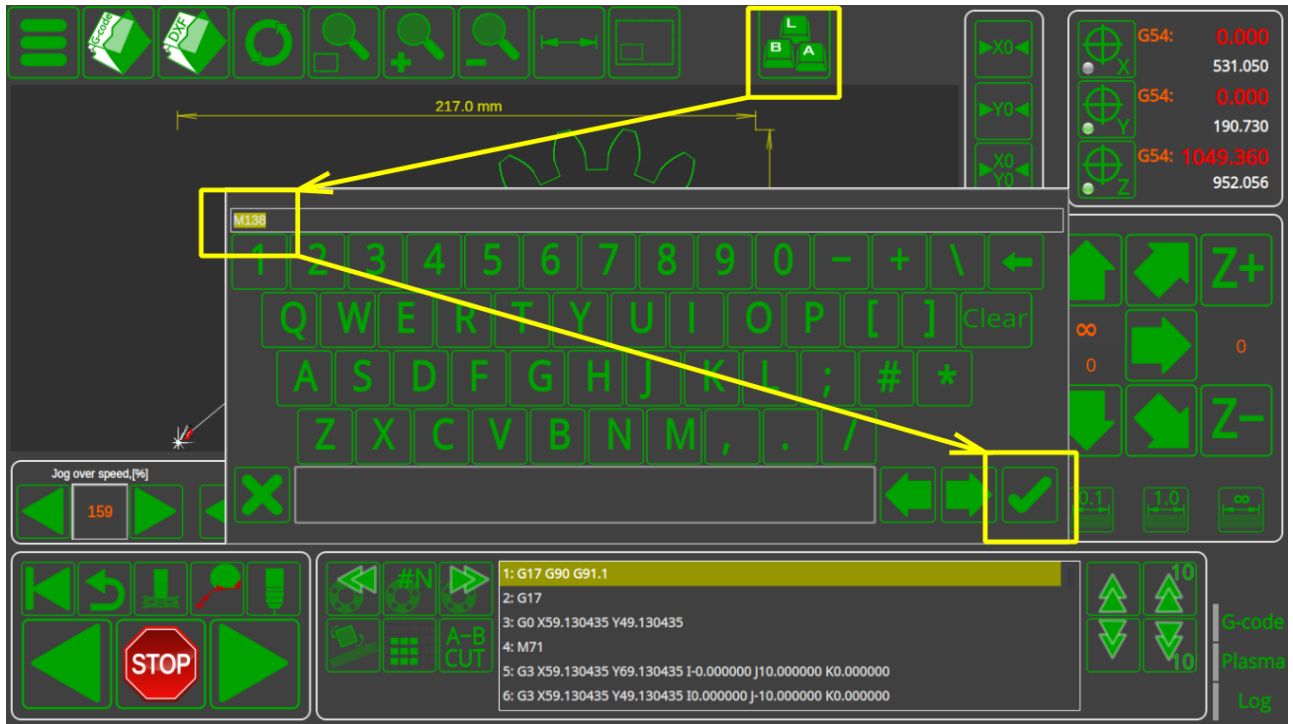
There are a few ways to send the machine to find homes

- It's possible to configure mandatory homing if CNC control system started and/or an Emergency or Servo drivers alarm event happened.

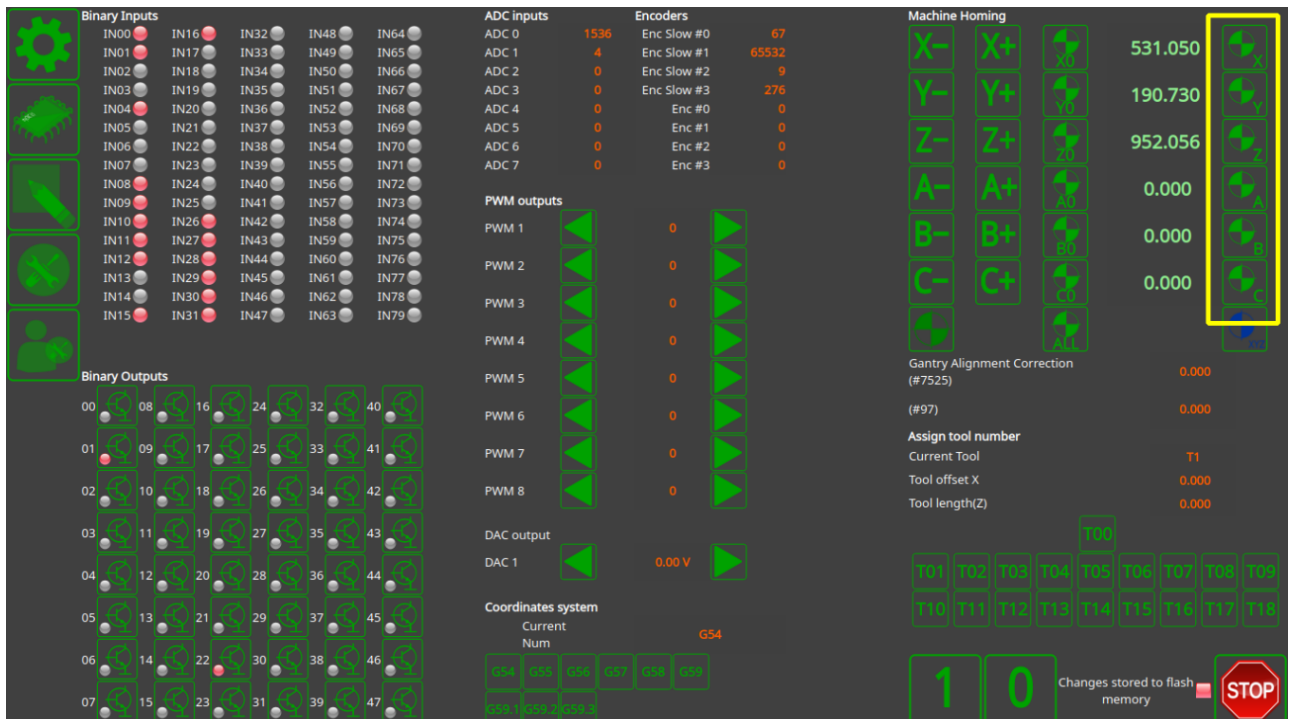


The red button on the message popup widget should be pressed to send a machine to the home position. The popup message will be hidden when all home flags are ready.

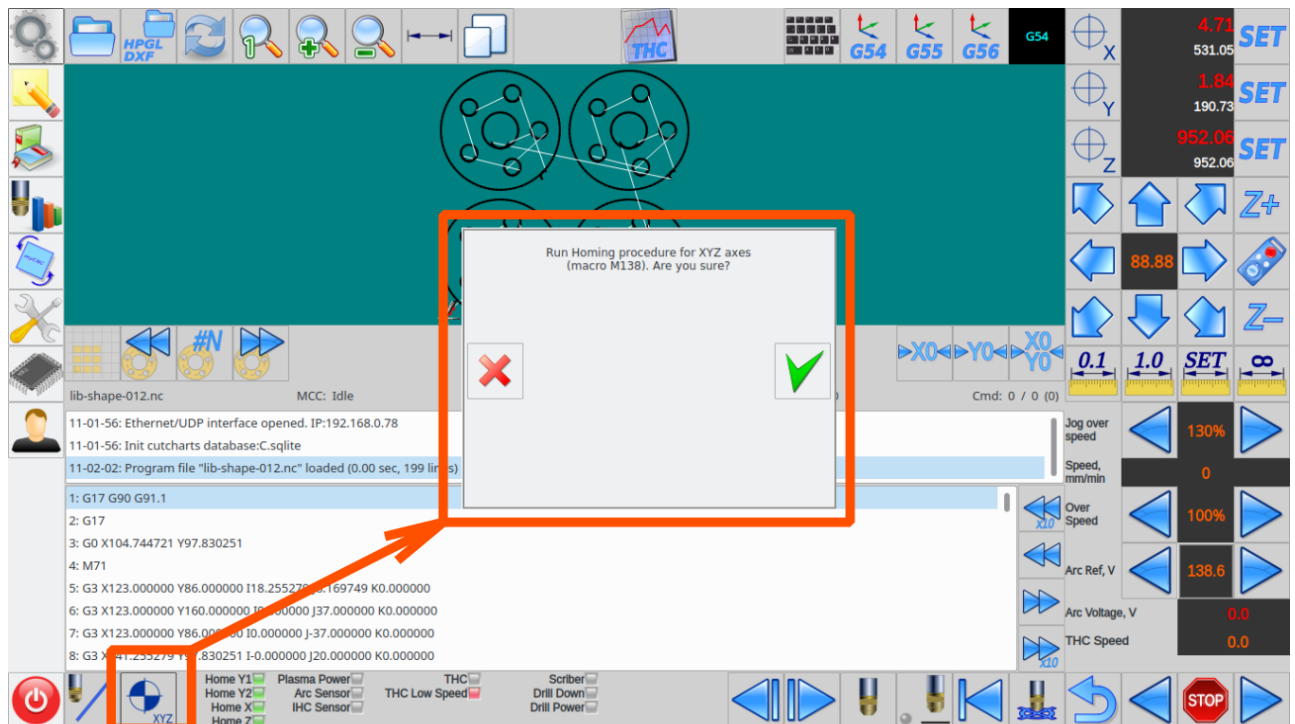
- Home macro procedure can be started from the MDI interface



- UI might contain button to send machine home. For example buttons for homing each axis X, Y, Z, A, B, C on diagnose widget

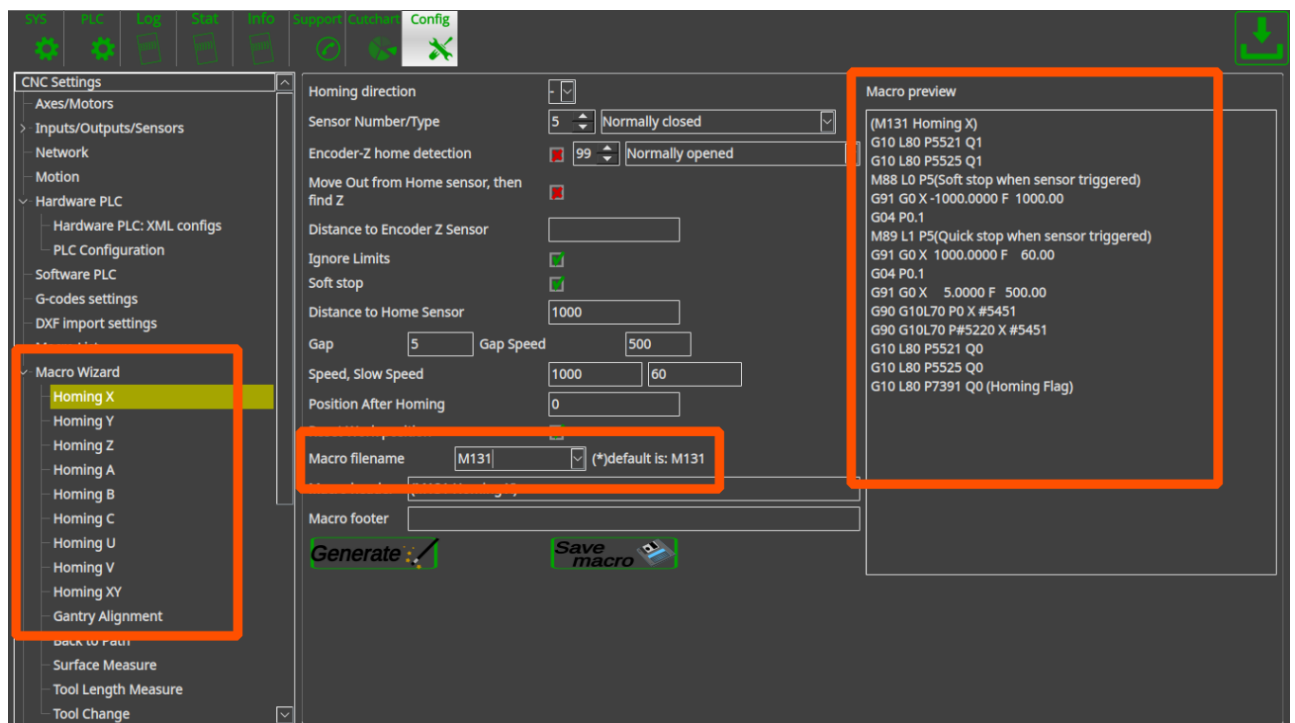


or All the Axes Homing button on the main screen of “1024P-V2” profile



Homing Macro files should be created during machine setup. Macro file can be created

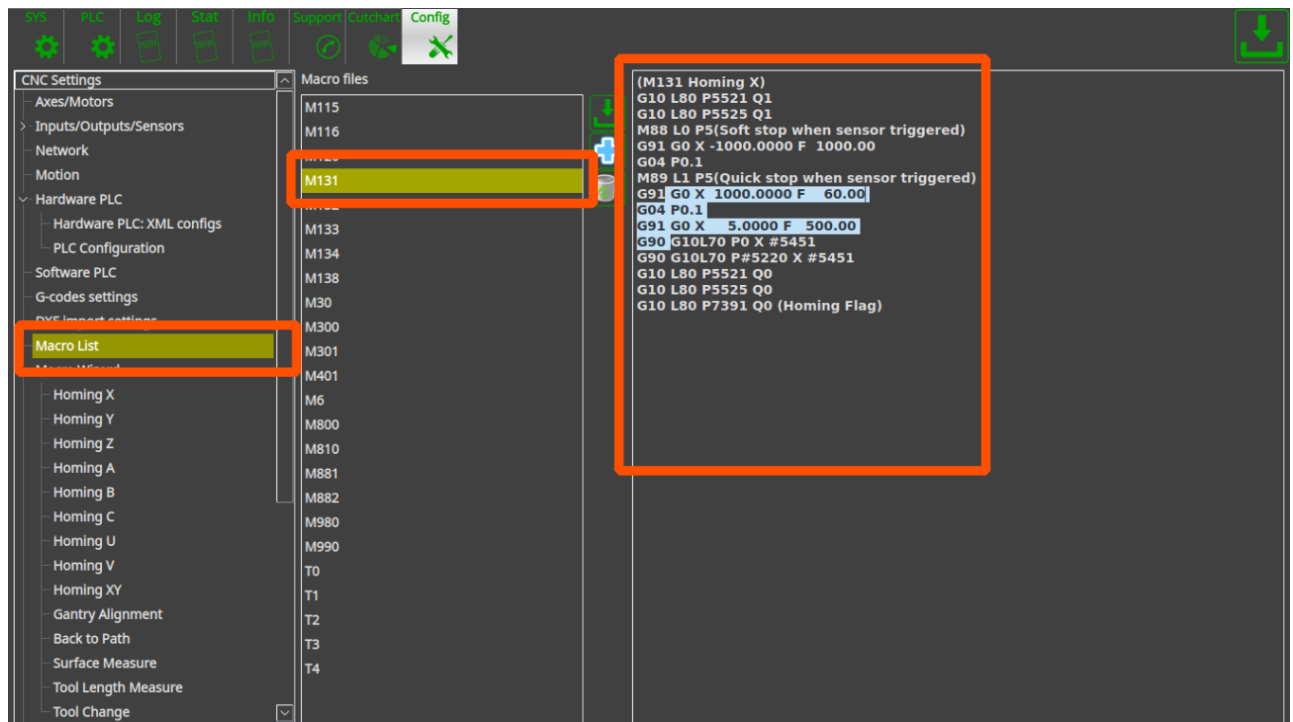
- in “Macro Wizard” conversational dialogs



By using the macro wizard you just point homing moving direction, input number used for home sensor, sensor type (normally opened/closed), homing and probing speed, parameters of Encoder Z-label (if used to find precise homing position) and some other parameters. Macro file will be generated on the widget “Macro preview” by pressing “Generate” button. If the macro is ok, the button “Save macro” should be pressed to save macro file on a storage device (HDD, SD

card etc). The dialog settings itself might be saved in the configuration file by pressing “Save” button on the top-right corner of the screen.

- manually in “Macro List” widget

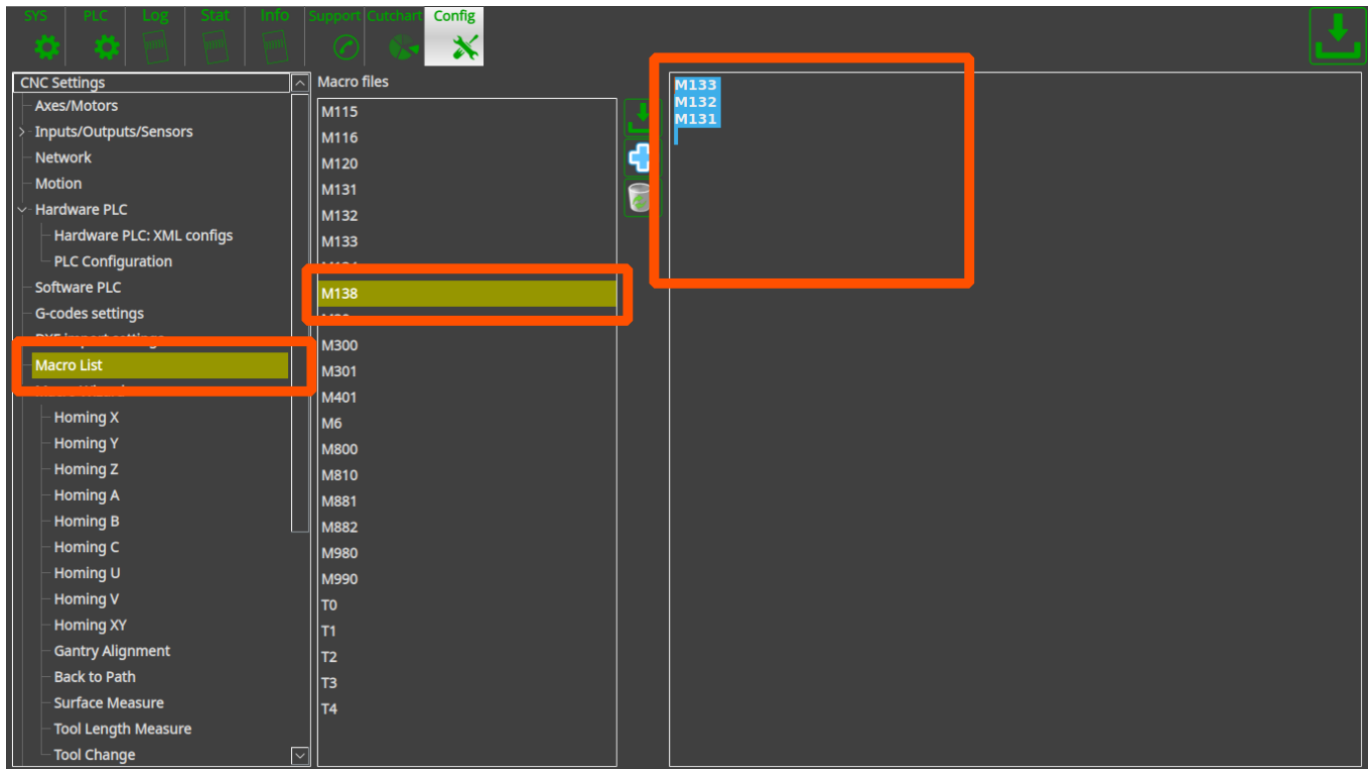


Home-All macro M138 is a simple macro file which runs in series macro files for all axes. For simple 3 axes machine it would be like this

M138

```
M133
M132
M131
```

Procedure started from Homing Z, then Y and the last is X. Any other sequence can be chosen according to your requirements.



In case Gantry alignment procedure used instead of standard homing procedure, it might have different macro file name to not be mixed-up with standard homing procedure. For example - homing Y is "M132", gantry alignment is "M1321". This is just example, any of macro filename can be chosen. The result of Home-All macro would be

M138

```
M133  
M1321  
M131
```

Home-All macro can be very different for more complicated machines with multi-spindles, 5 axes and so on. For example M138 macro for multi-spindle machine with tangential knife.

M138

```
M62P5 (OUTPUT HOMING ON)  
  
M135 (C-knife)  
  
M1331 (Homing Z1)  
M1332 (Homing Z2)  
  
M132 (Homing Y)  
M131 (Homing X)  
  
M63P5 (OUTPUT HOMING OFF)
```

How to set up Homing Handler

If the machine has stopped unexpectedly, due to things like an emergency stop button press/power failure/etc, it is sometimes necessary to prompt the user to run the Homing procedure again before allowing any further actions to be taken in order to safely calibrate the machine. In order to do that, a software PLC is put in place called the **HOMING_HANDLER**

The Homing Handler can be set up by using the following instructions:

1. Go to **Settings > Config > PLC > Software PLC** and select **HOMING_HANDLER**

The screenshot shows the myCNC software interface with the 'Config' tab selected. The left sidebar shows the 'CNC Settings' tree, with 'Software PLC' highlighted. The main area displays the configuration for the 'HOMING_HANDLER' PLC. The 'PLC Sources' list includes various handlers, with 'HOMING_HANDLER' selected. The 'PLC Includes' list shows the following items:

- _BV17
- _HANDLER_EXIT
- _HANDLER_GCODE_START
- _HANDLER_GCODE_STOP
- _HANDLER_INIT
- _HANDLER_SERVO_OFF
- _HANDLER_SERVO_ON
- CHARGE_PUMP
- HOMING_HANDLER**
- LEDS
- MODBUS_TEST
- OIL_CHANGE_COUNTER
- POPUP_HANDLER

The 'Name' field is set to 'HOMING_HANDLER' and the 'Aliases' field is empty. The 'main()' function is defined as follows:

```
main()
{
    exit(99);

    a=0;
    do{

        hx=gvarget(7391); //monitor axes flags X, Y, Z and C
        hy=gvarget(7392);
        hz=gvarget(7393);
        //hc=gvarget(7396);

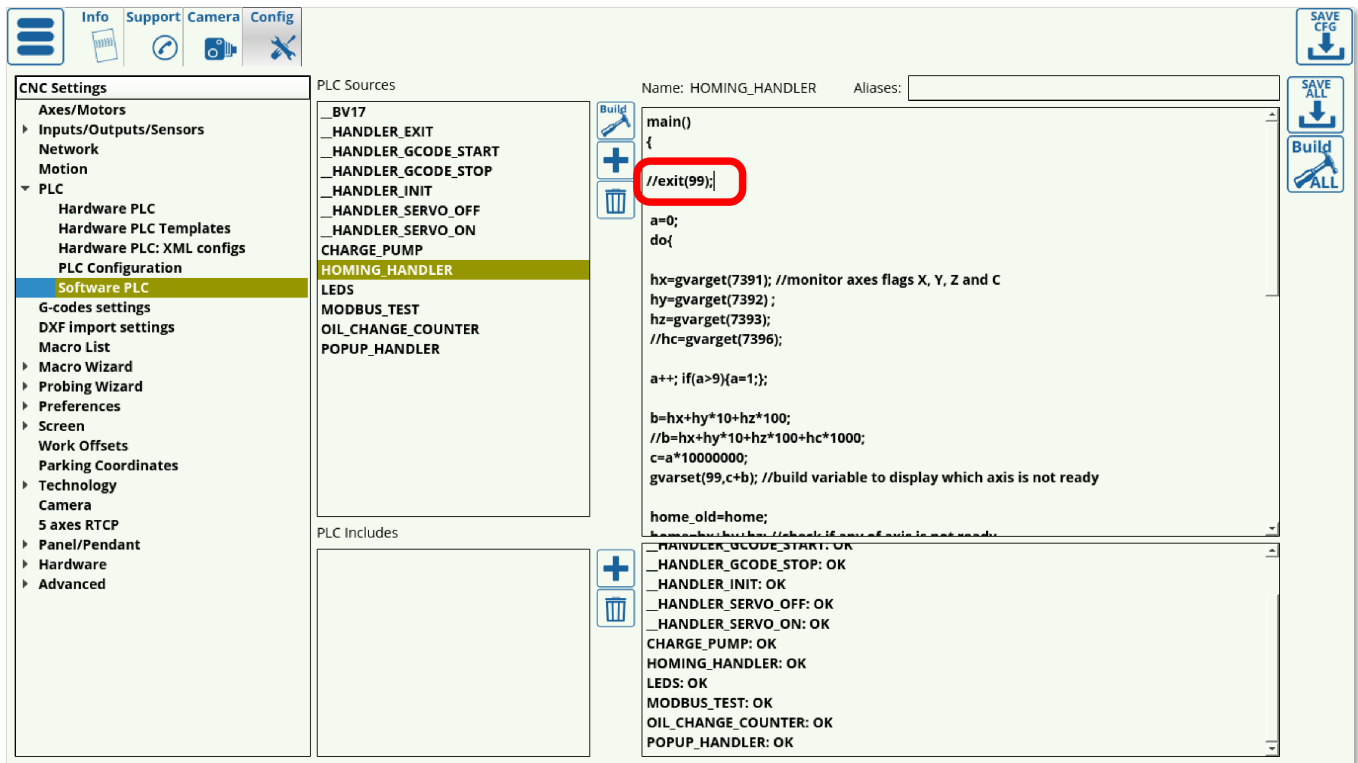
        a++; if(a>9){a=1;};

        b=hx+hy*10+hz*100;
        //b=hx+hy*10+hz*100+hc*1000;
        c=a*1000000;
        gvarset(99,c+b); //build variable to display which axis is not ready

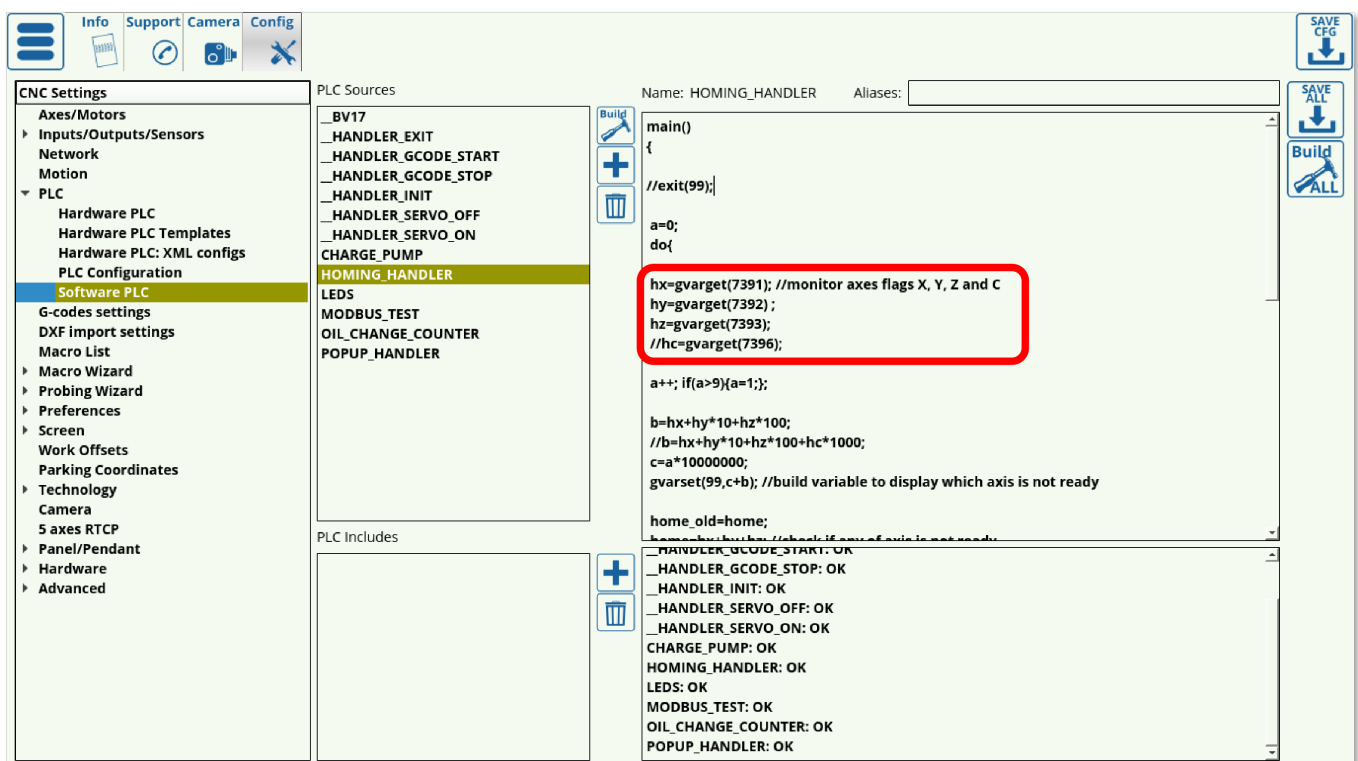
        home_old=home;
        //home=home; //check if any of axis is not ready;

        _HANDLER_GCODE_START: OK
        _HANDLER_GCODE_STOP: OK
        _HANDLER_INIT: OK
        _HANDLER_SERVO_OFF: OK
        _HANDLER_SERVO_ON: OK
        CHARGE_PUMP: OK
        HOMING_HANDLER: OK
        LEDS: OK
        MODBUS_TEST: OK
        OIL_CHANGE_COUNTER: OK
        POPUP_HANDLER: OK
    }
```

2. In the HOMING_HANDLER code, put the two forward slashes in front of the **exit(99);** line, commenting it out.



Note that the axes for which the homing is not required (Axis C in this case) have also been commented out. If only select axes need to be included in the homing procedure after the emergency stop, these can be configured in the code by adding/deleting forward slashes in front of the respective axes' commands.



At this point, in an emergency stop situation, myCNC will prompt the user to run the homing procedure for the selected axes before any other action is taken after starting the machine back up again.

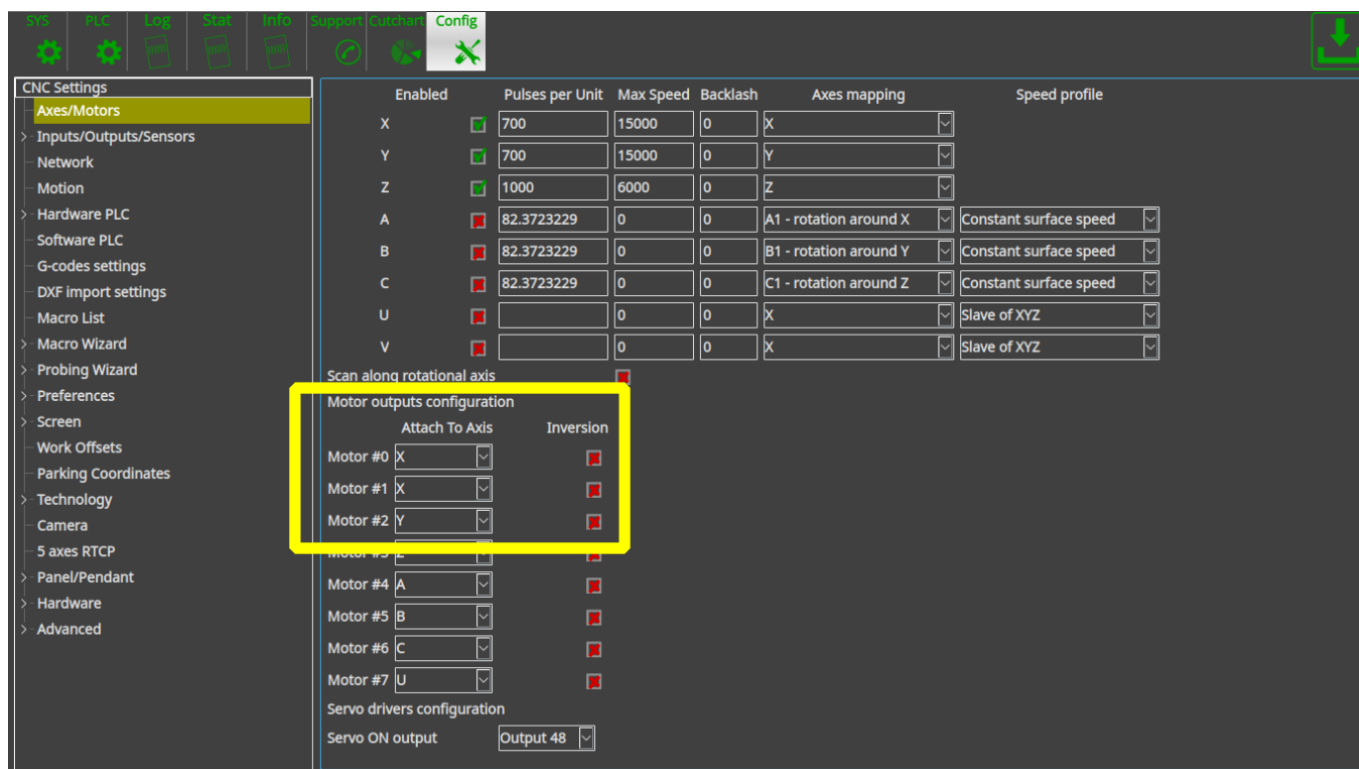
In order to disable the Homing Handler, add the two forward slashes in front of **exit(99);** again, then Save and reload the application.

How to setup Dual Axis (Slave X, Slave Y)

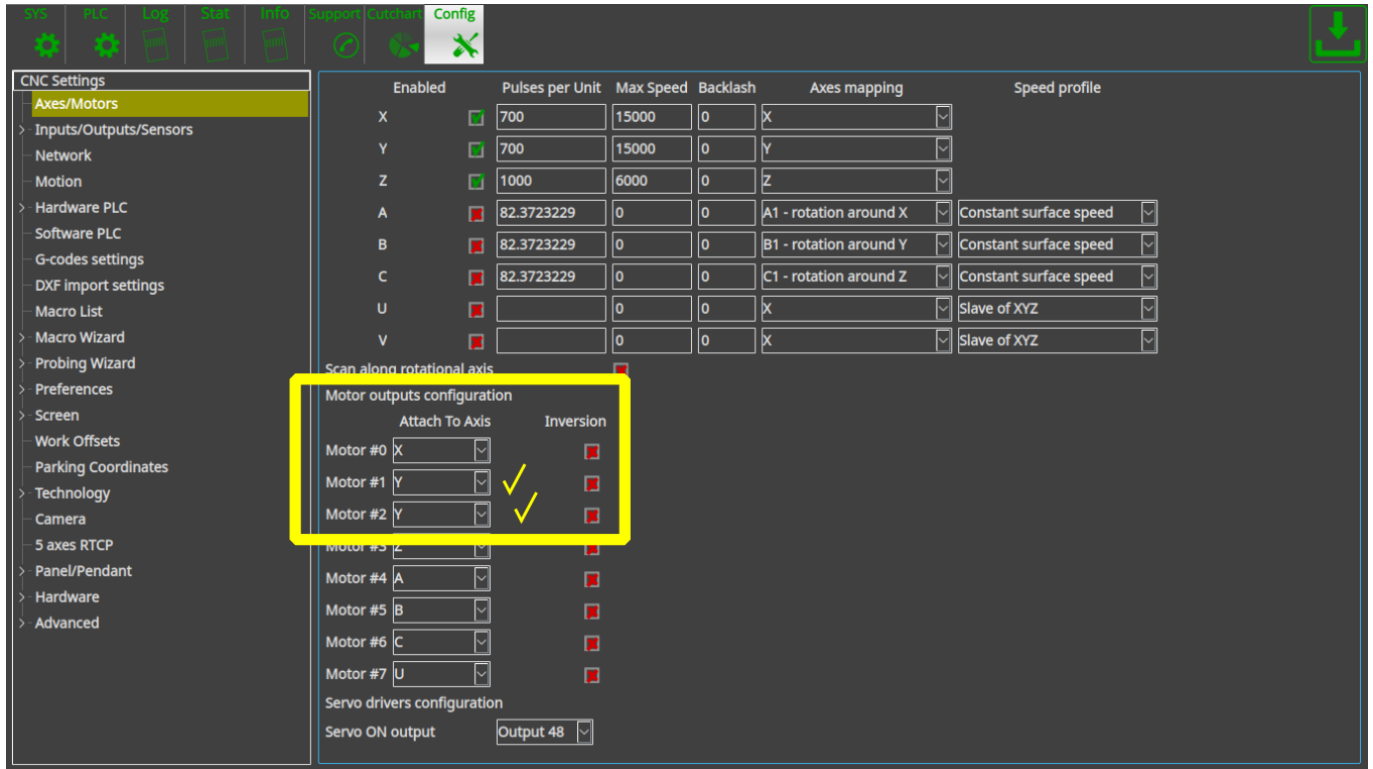
Dual axis setup is really simple.

Goto "Axes/Motors" configuration dialog and set the same axis for two motor outputs. They will work together

For example, motor outputs #0 and #1 are used for the same X axis on the picture below.



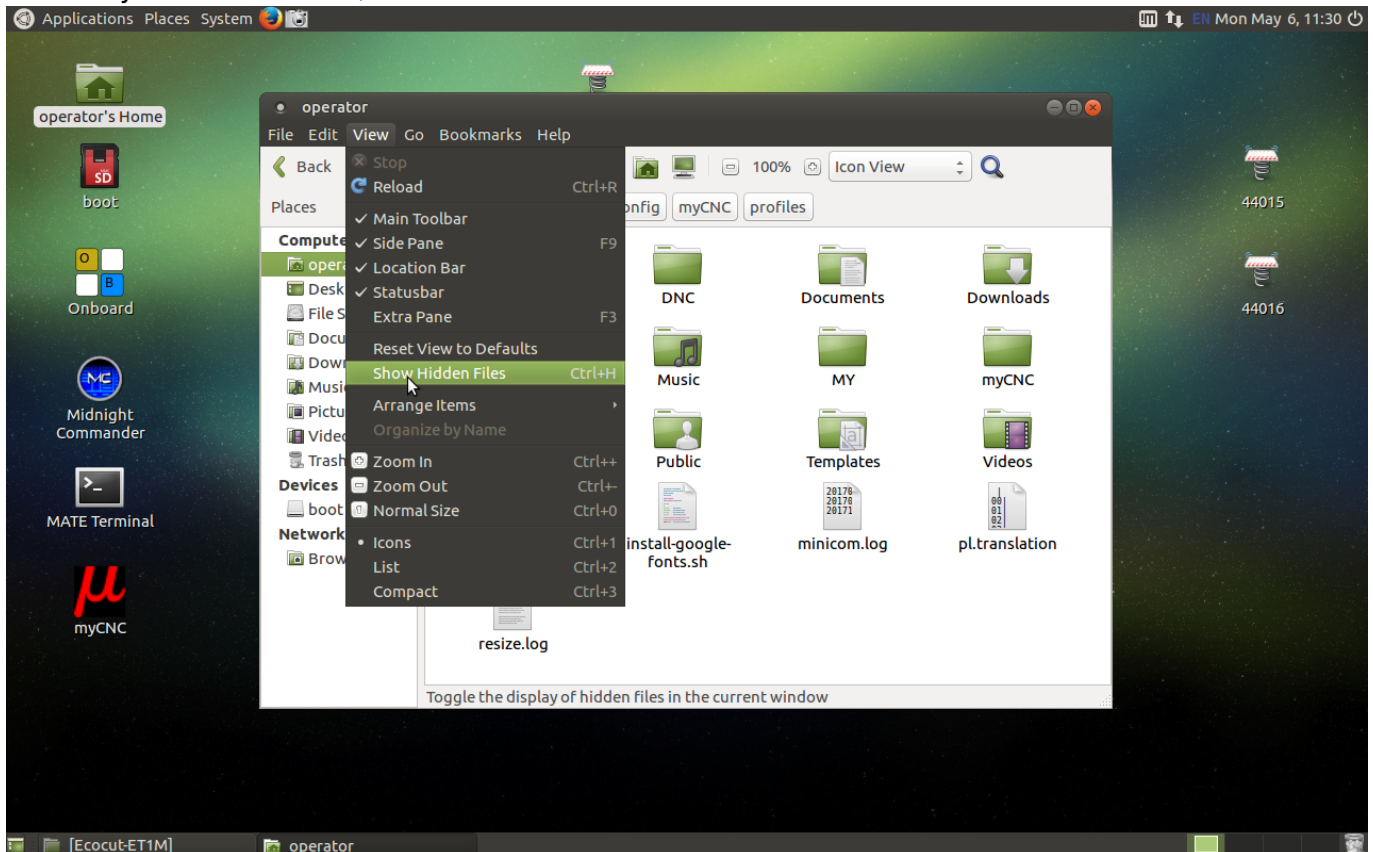
Next picture shows how to attach the Y axis to motor outputs #1, #2.



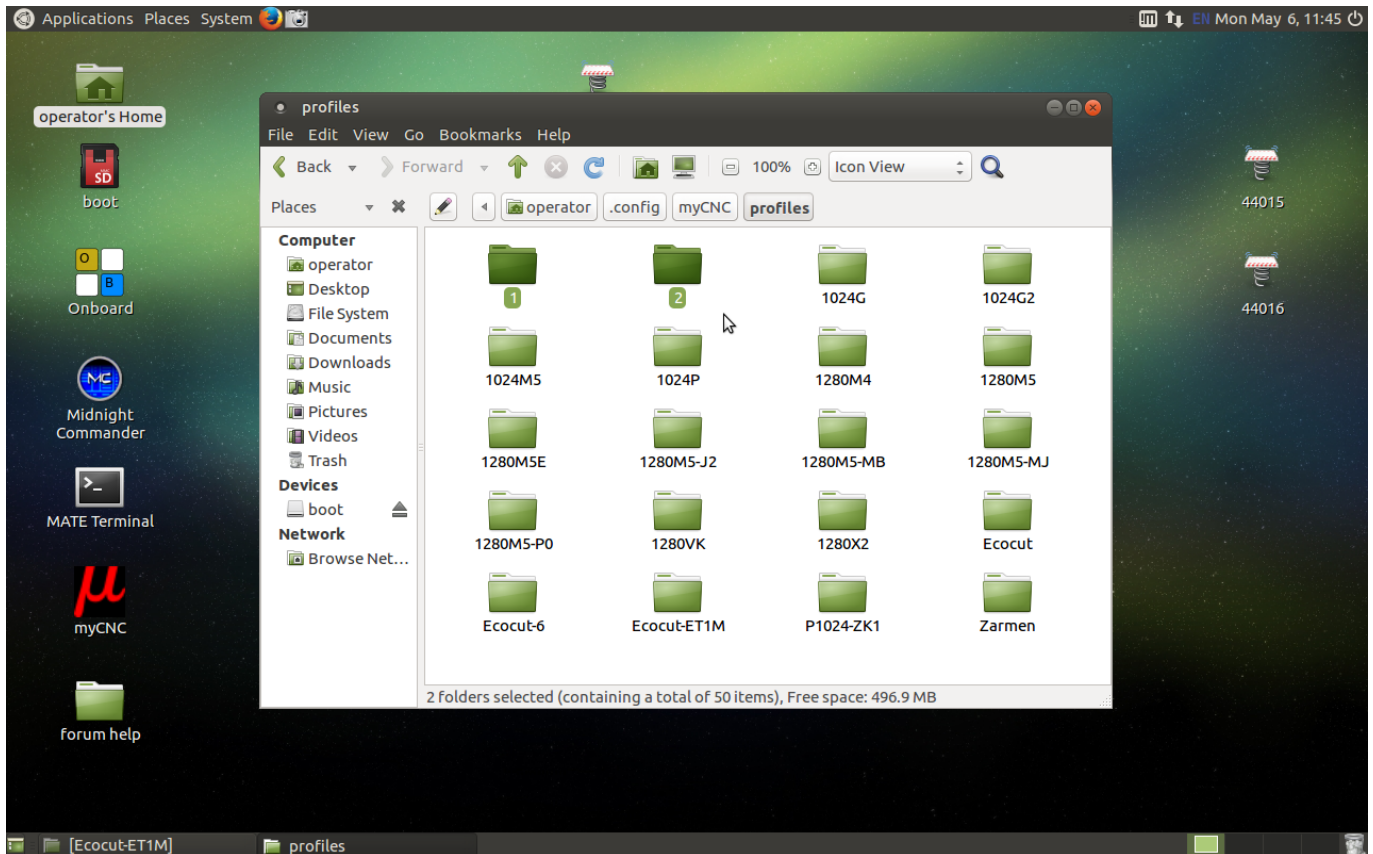
How to set up two instances of myCNC software on one machine

In order to start two instances of myCNC software on a single board Odroid-C2 computer:

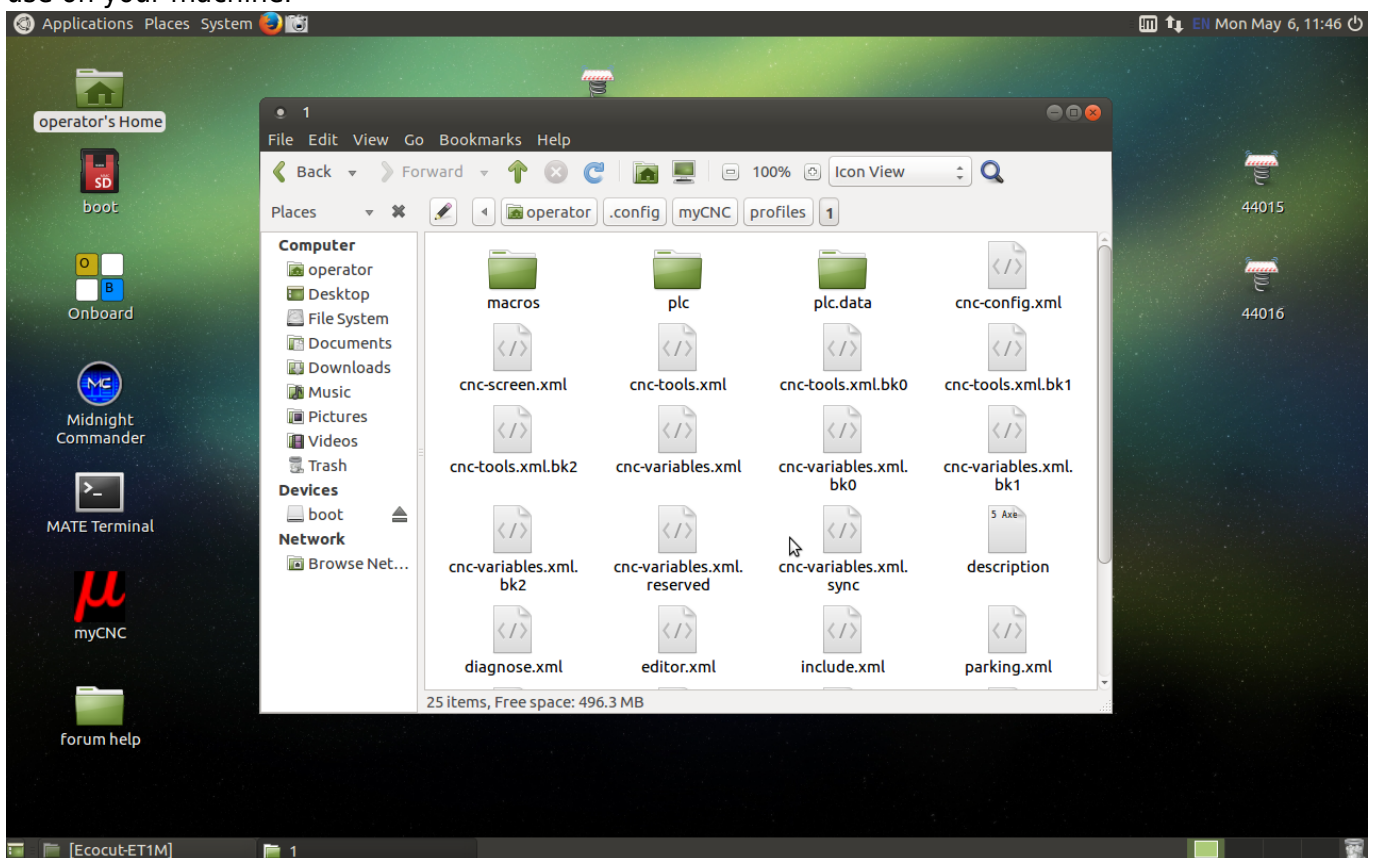
1. Go to your **Home** folder, and in the **View** menu click **Show Hidden Files**



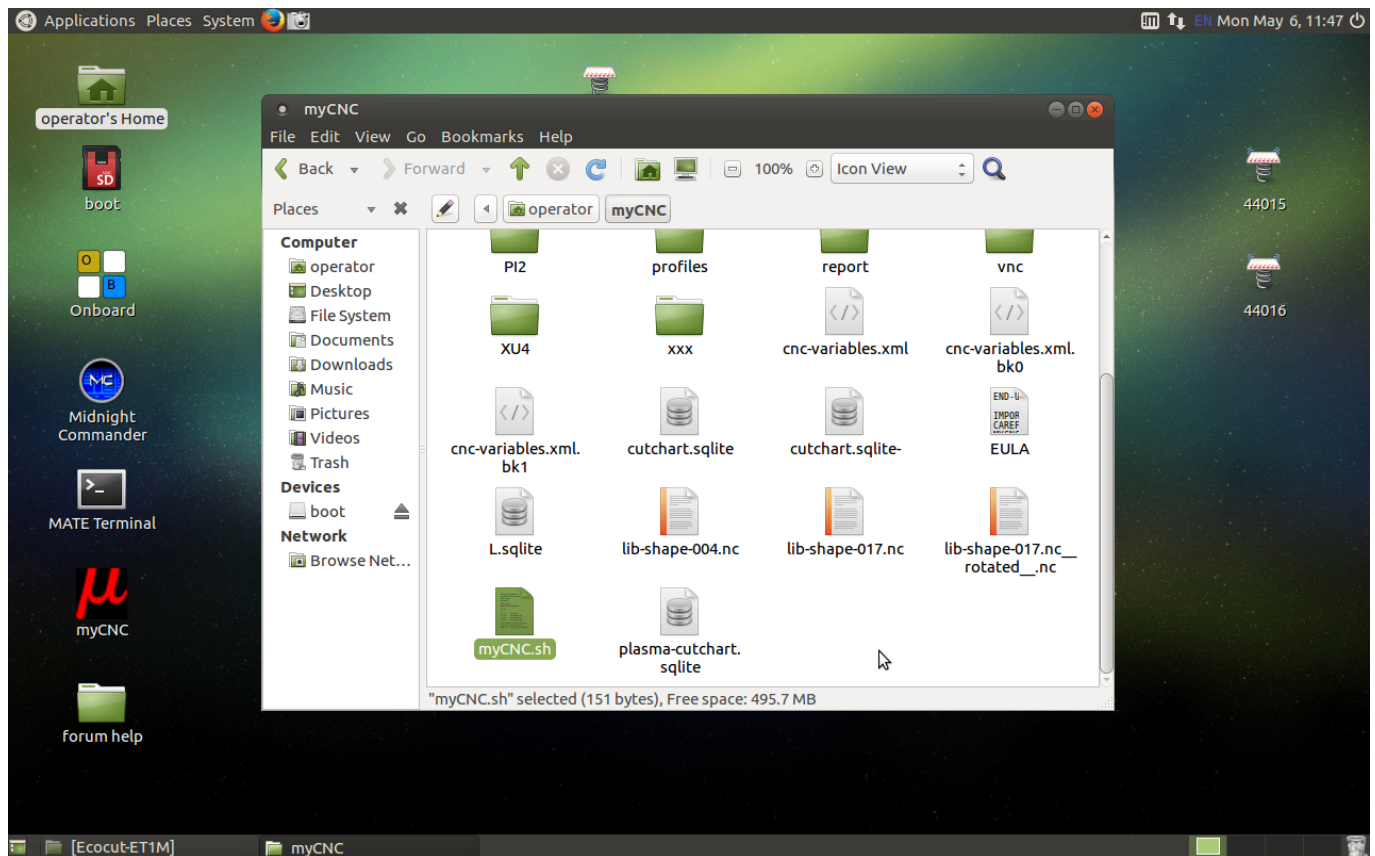
2. From your Home folder, navigate to `.config > myCNC > profiles`, and create folders named **1** and **2**



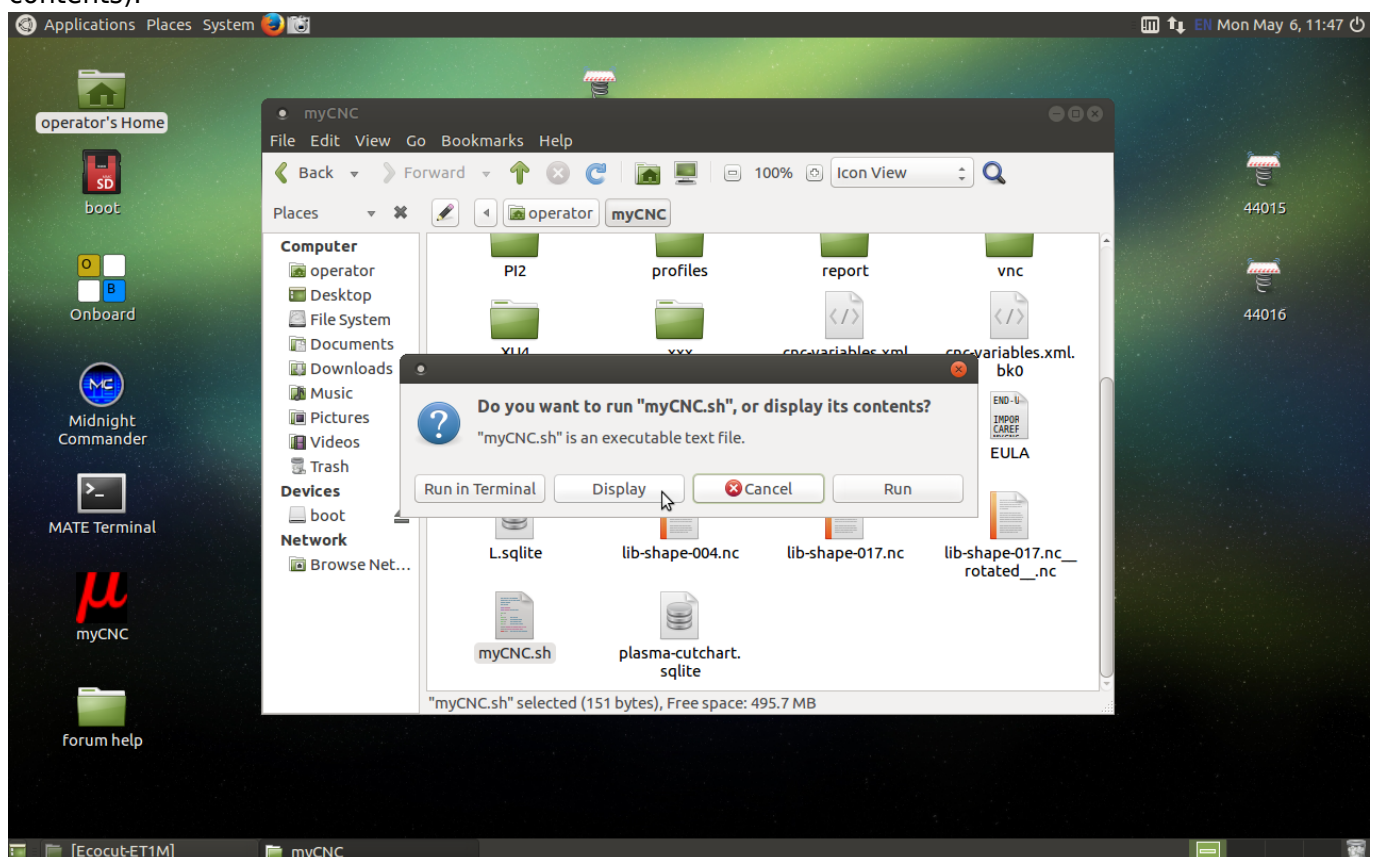
3. Copy the contents of another profile folder (for example, the contents of `.config/myCNC/profiles/12805`) into both new folders. Here, you can choose which profiles you usually use on your machine.



4. Go back to your home folder, and from there navigate to the **myCNC** folder (NOTE: this is a different folder from before, so go Home > myCNC instead of Home > .config > myCNC).



5. Open the **myCNC.sh** file (click Display when asked whether you want to run the file or display its contents).



6. Paste the following code into the file, then save it and close the editor.

```
#!/bin/bash
```

```
cd $(dirname $0)
echo DIRNAME: $(dirname $0)

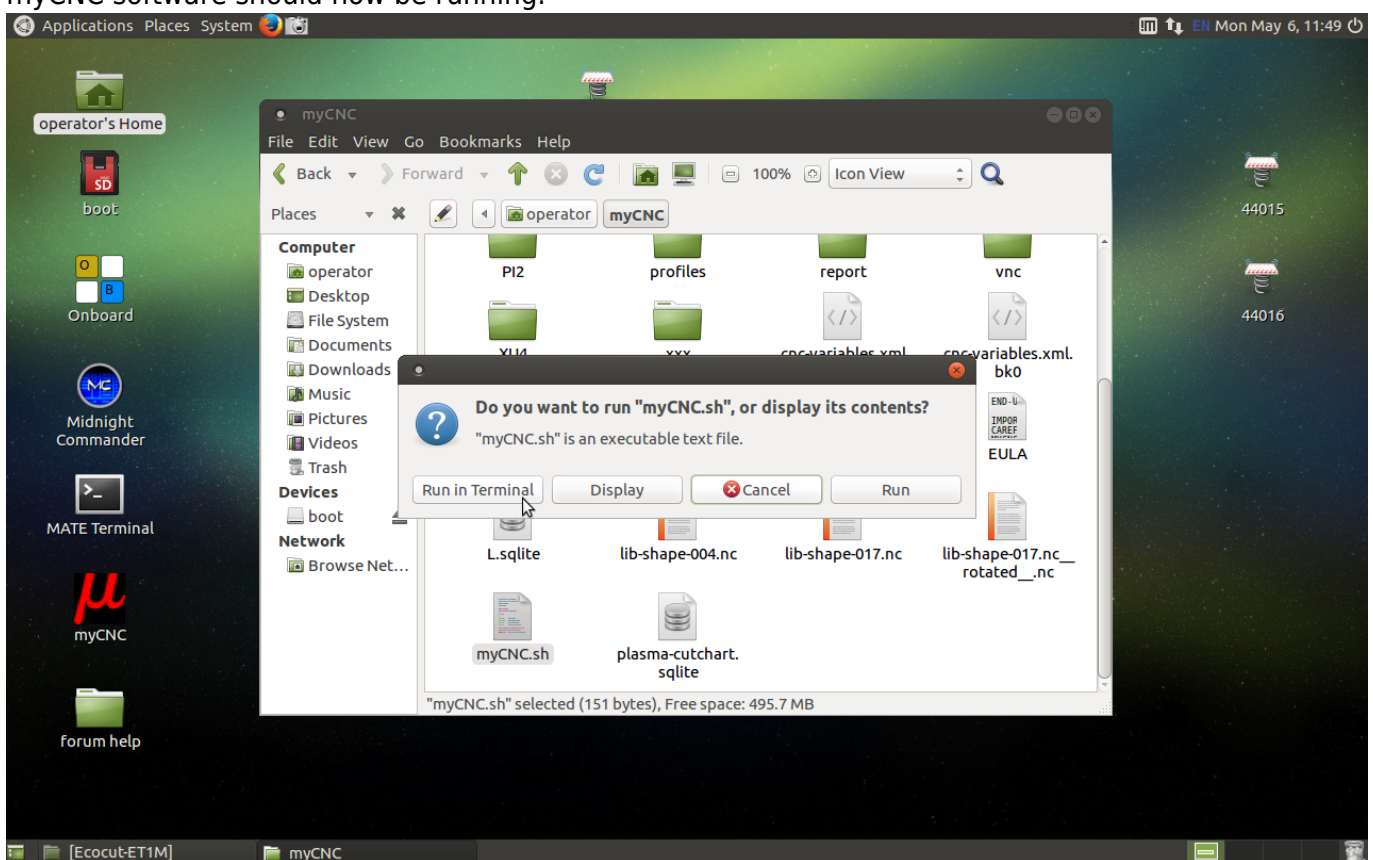
killall myCNC
killall myCNC

xset s 0
xset -dpms

./C2/myCNC -p1 &

./C2/myCNC -p2
```

7. Run the **myCNC.sh** file by selecting Run in Terminal after opening the file. Two instances of myCNC software should now be running.



From:

<https://docs.pv-automation.com/> - **myCNC Online Documentation**

Permanent link:

https://docs.pv-automation.com/ru/mycnc/mycnc_setup_examples

Last update: **2022/03/29 15:56**

