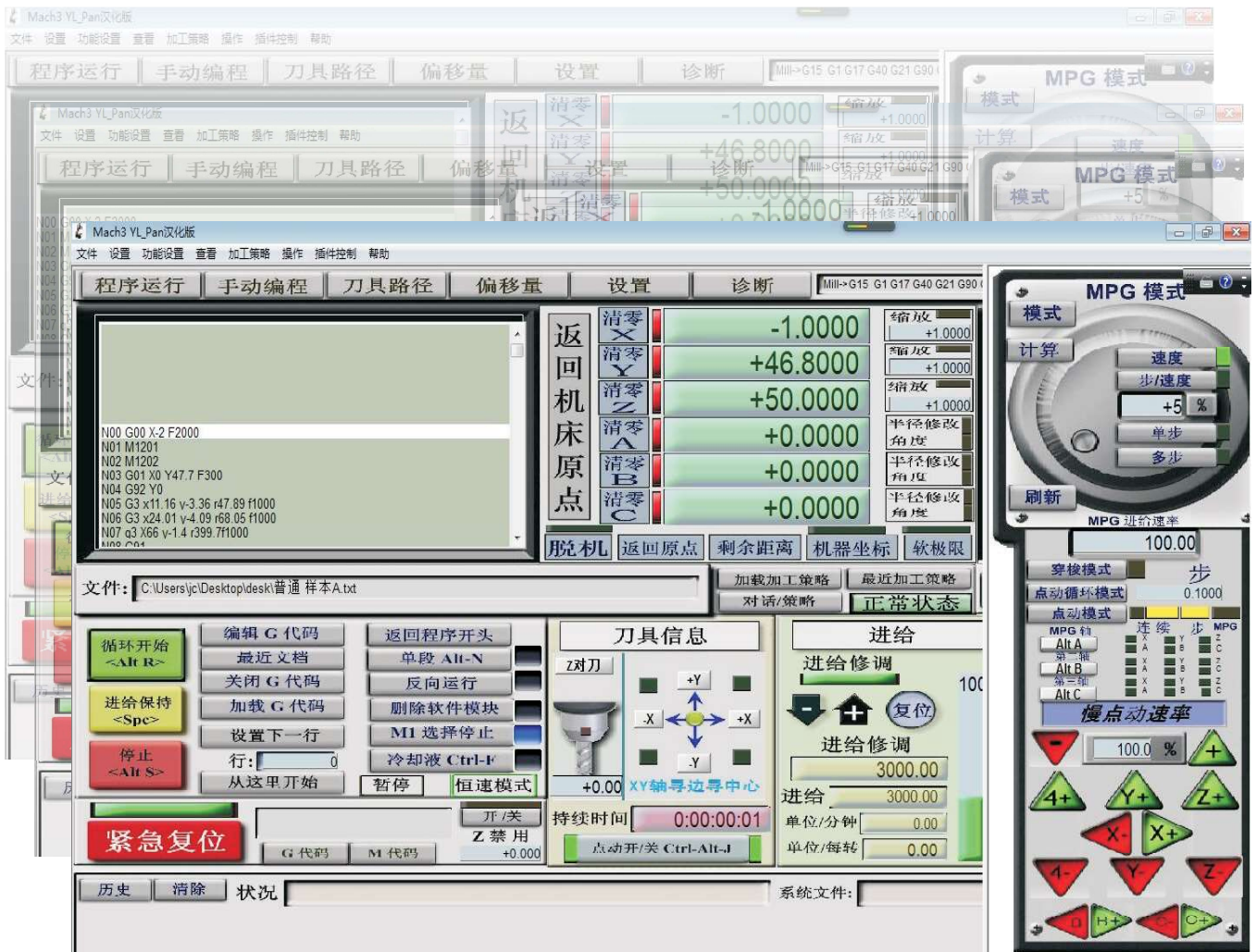




MACH3 Basic software operation manual (For MKX board)





1. How to connect to the computer

1.1 How to connect the USB board



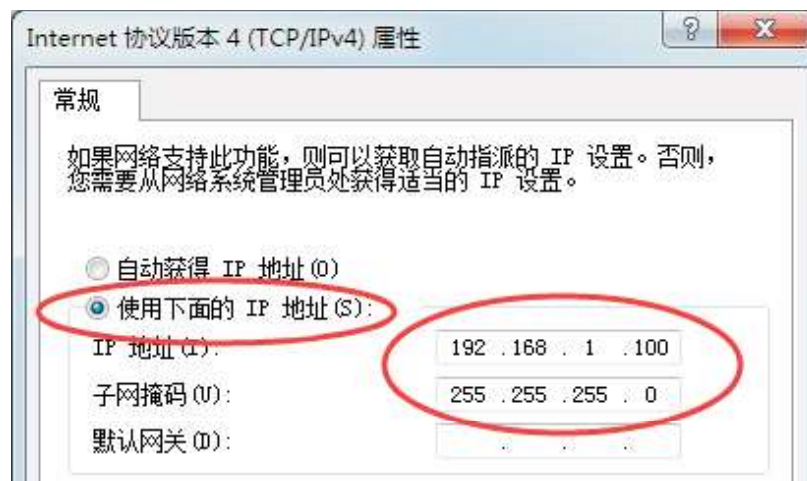
Please use the cable provided by the manufacturer. Do not add extension lines or transfer lines without authorization!



1.2 How to connect Ethernet board

Use a network cable to connect directly to the computer

Then set the IP address manually on your PC: Open the network and sharing center >> local connection >> Properties >> Internet Protocol version 4 (TCP / IPv4), and manually set the IP address in the format of 192.168.1.0 ~ 179 (as shown below)

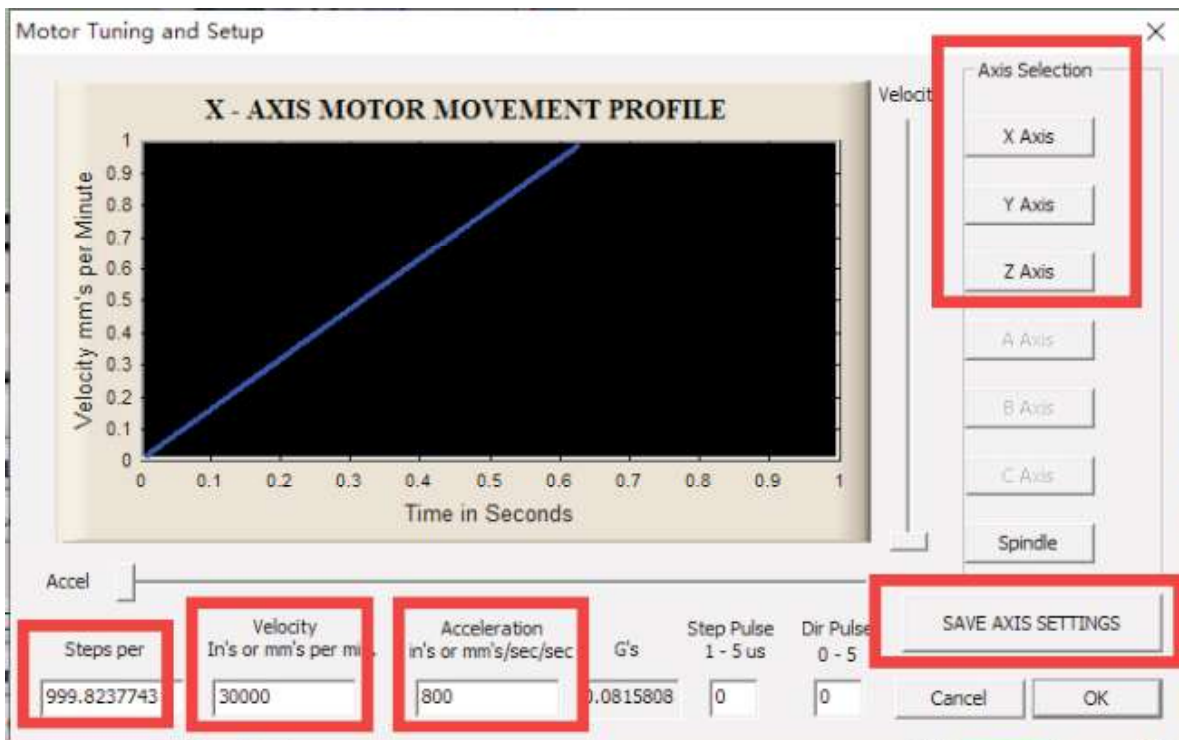




2.MACH3 Basic software parameters

2.1 Motor tuning

Open the software and select "setting" > > "motor tuning"
(as shown below)



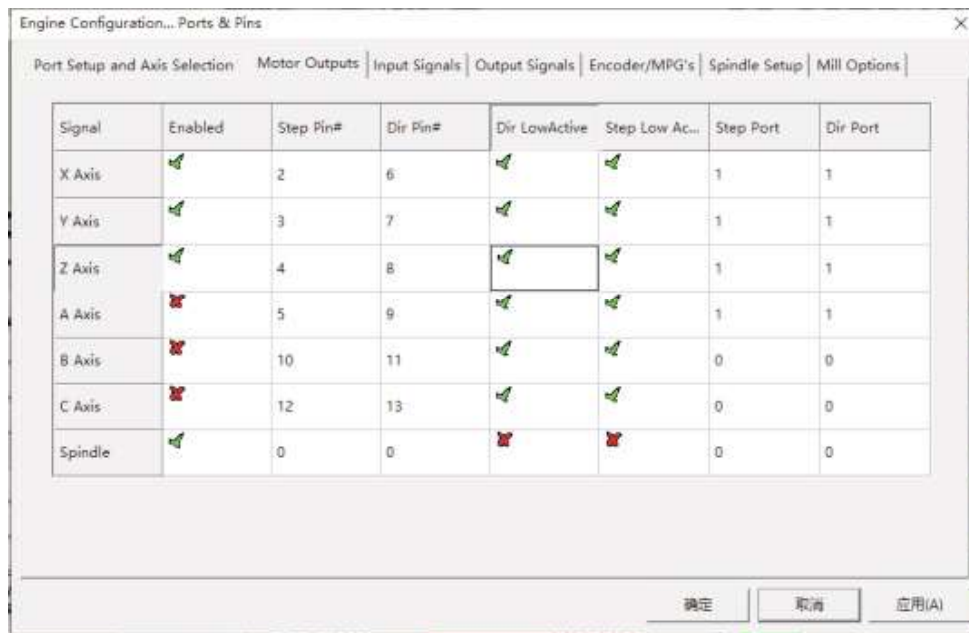
In the window, first select an axis you want to configure, next enter parameters and click „SAVE AXIS SETTINGS” . Now you can select and set the next axis. If we forget to click on „SAVE AXIS SETTINGS” all the changes will not be saved.

- 1、 **Steps per:** For correct axis scaling you must know how many steps falls on a unit (millimeter, inch or degree). To understand how to calculate it,
- 2、 **Velocity:**Maximum motor speed
- 3、 **Acceleration:**The greater the acceleration, the faster the motor accelerates. However, it should not be too large, which will make the motor unable to run.The ratio of maxspeed and acceleration is 20:1

2.2 Ports and pins

2.2.1 motor output

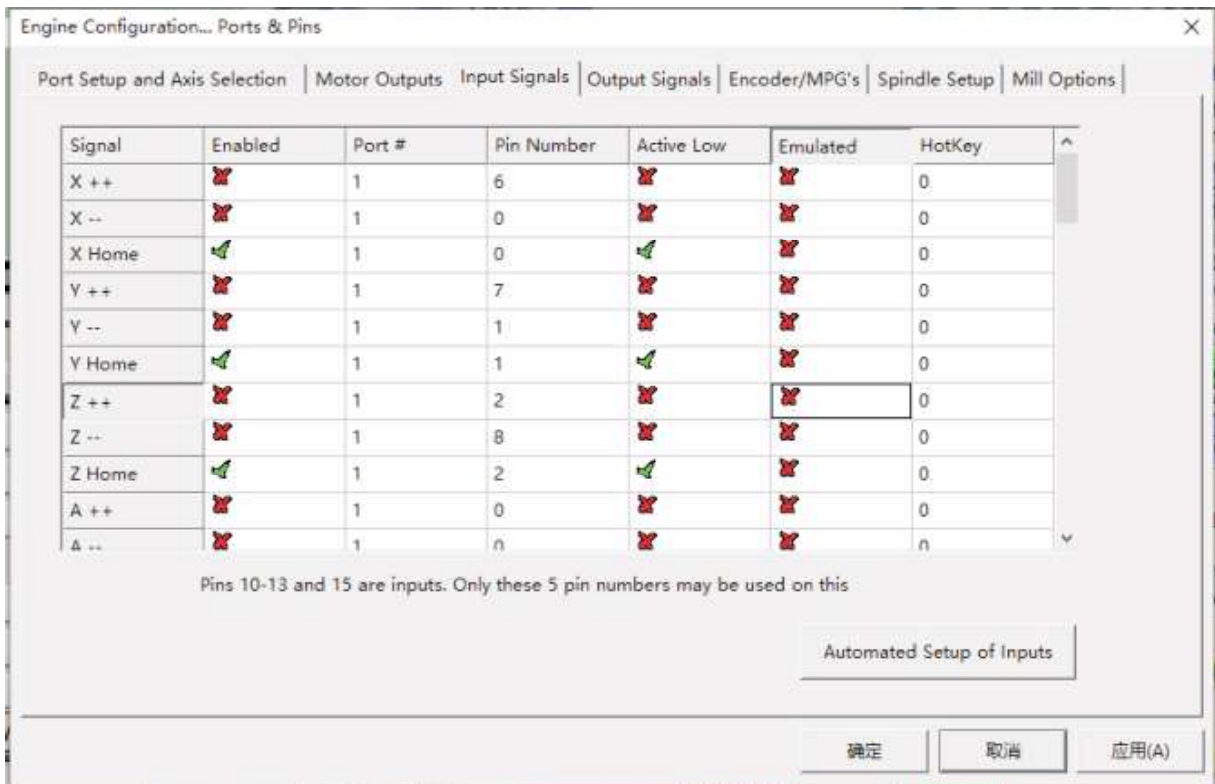
Open Mach3 Software , select "Config" >> "Ports and Pins"
>> "Motor tuning"



1. **Enable:** Tick to make the axis output
2. **Dir Lowactive:** Motor direction selection
3. **Step Lowactive:** Tick to make the axis output

2.2.2 Input signal

Open Mach3 software , select "Config" >> "Ports and Pins"
>> "Input signals "

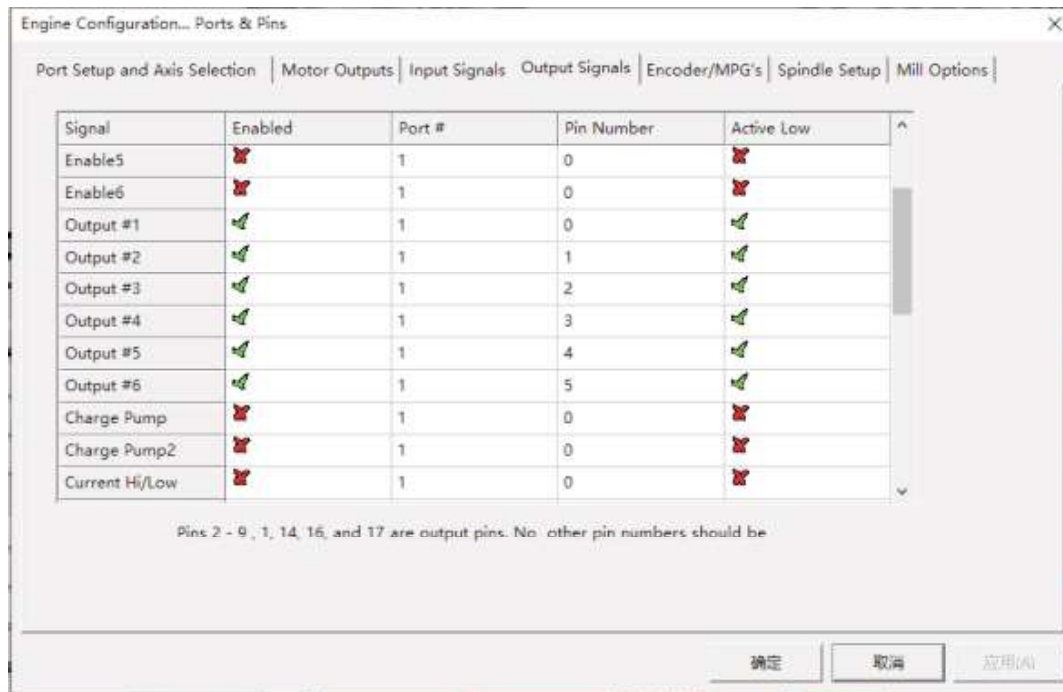


1. **Enable:** Tick to enable output ports
2. **Pin Number:** The control card input port
3. **Active Loww:** Low level active, tick to enable

2.2.3 Output signal

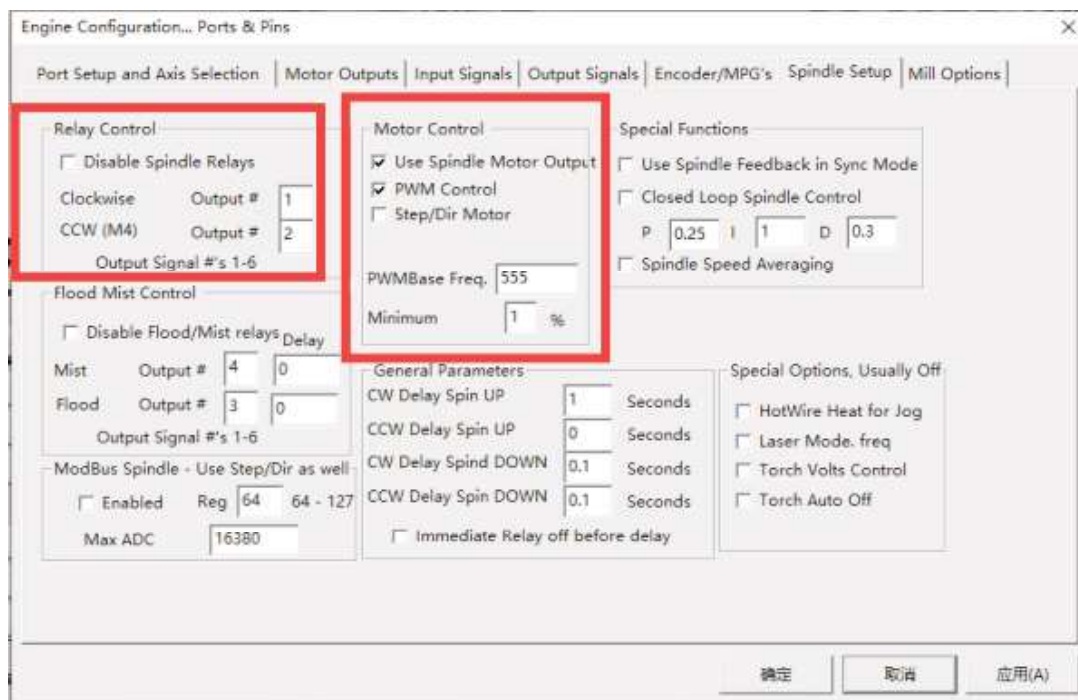
Open Mach3 software , select "Config" >> "Ports and Pins"
>> "Output signals "

1. **Enable:** Tick to enable output ports
2. **Pin Number:** The control card output port
3. **Active Loww:** Low level active, tick to enable



2.2.4 Spindle setup

Open Mach3 software , select "Config" >> "Ports and Pins" >> "Spindle setup"



1. **Disable spindle relays:** Cancel the tick
2. **Motor control:** if your spindle is VFD, tick "PWM" ; if your spindle is driver, tick " Step /dir motor"

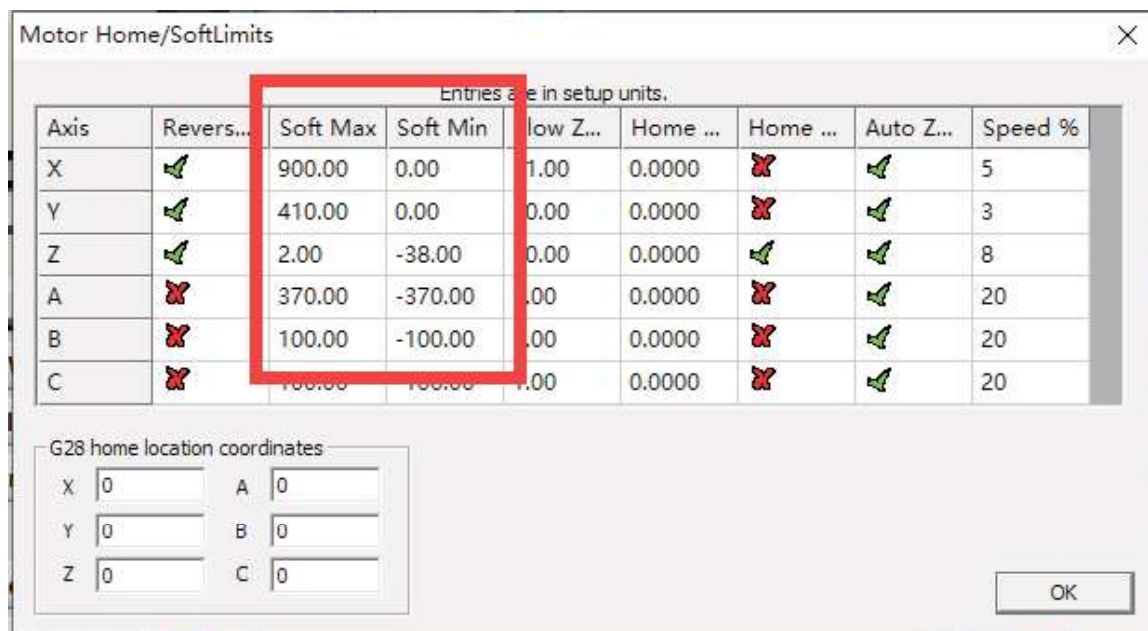
2.3 Software limit

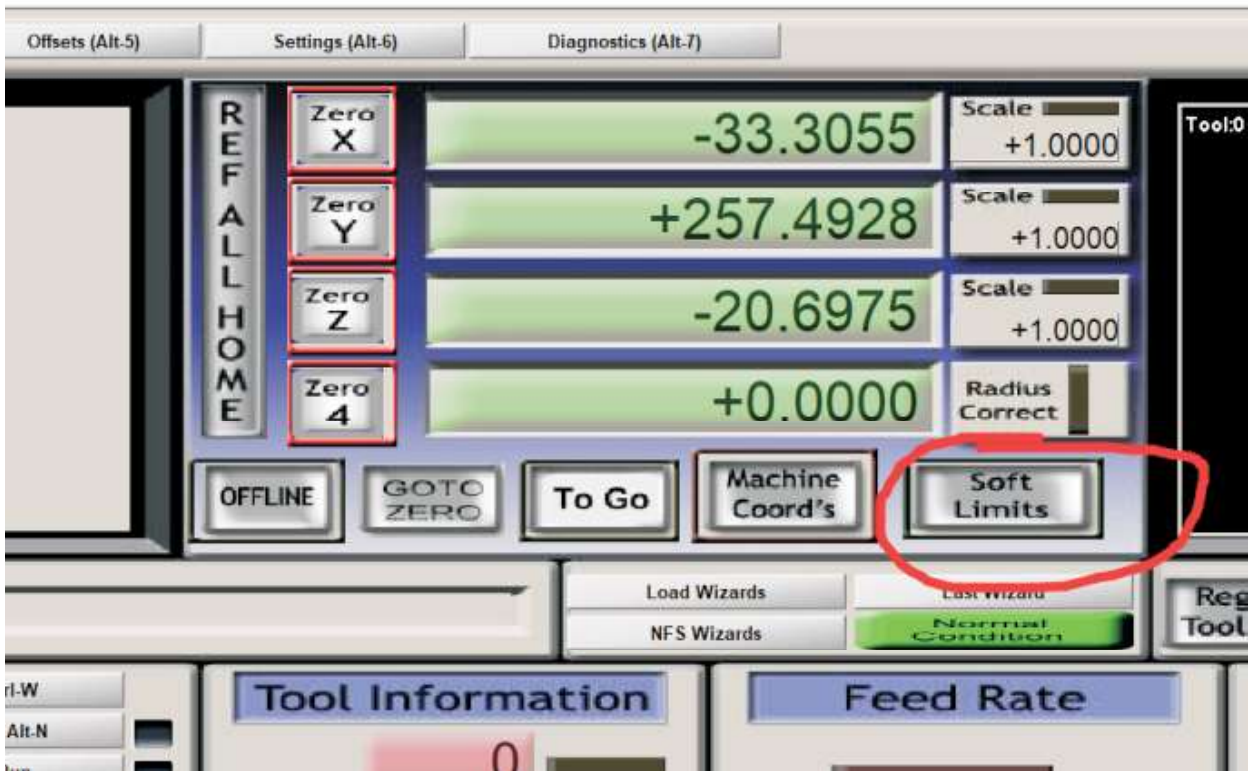
Open Mach3 software, select "Config" >> "Homing/limits" option.

"Soft Max": Software limit maximum machine coordinates

"Soft Min": Software limit minimum machine coordinate

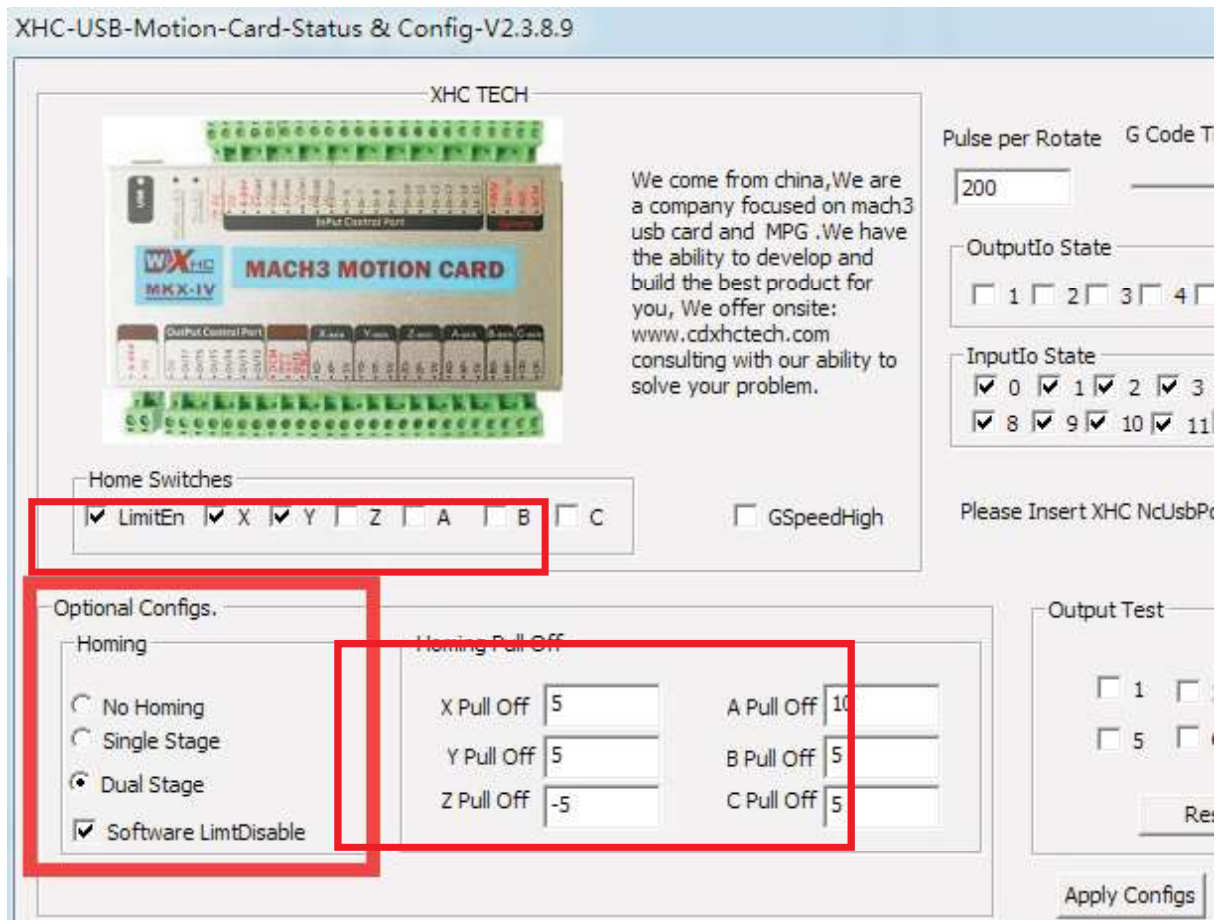
After setting up, click the "soft limit" button on the software to open the software limit function. When normal, the button will have a green light, indicating that the software limit is valid. It is important to note that if your machine's current mechanical coordinates are not within the limit of the software you set, the press button can't turn on this function. Please move the machine to the limit and press the button.





2.4 Driver plug-in control

open "Plugins control" >> "XHC Ncpod Config"



No Homing: not go to machine origin

Single Stage: go to machine origin only once

Dual Stage: go to machine origin twice

Homing Pul off: Origin fallback distance

Home Switches: tick "LimitEn" enable , and tick X.Y.Z and so on to allow it to have a limit function

GSpeedHigh: Tick, Increase the arc running speed

2.5 Z safety height

Open Mach3 software , select "Config" >> "Safe Z setup" option.

Tick "Allow safe Z move" enable;

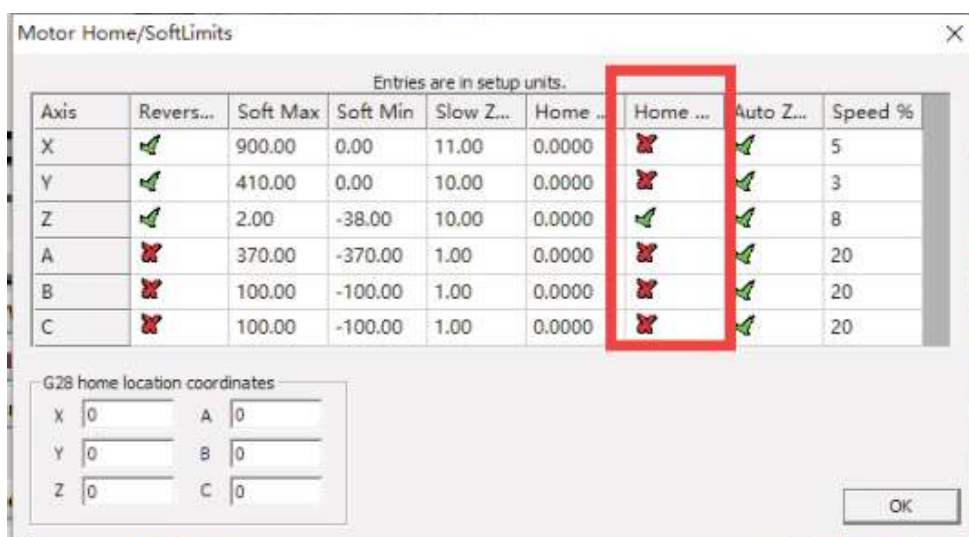
tick "Safe Z DRO is in work coordinates" enable;

Fill in the "safe Z" height value;



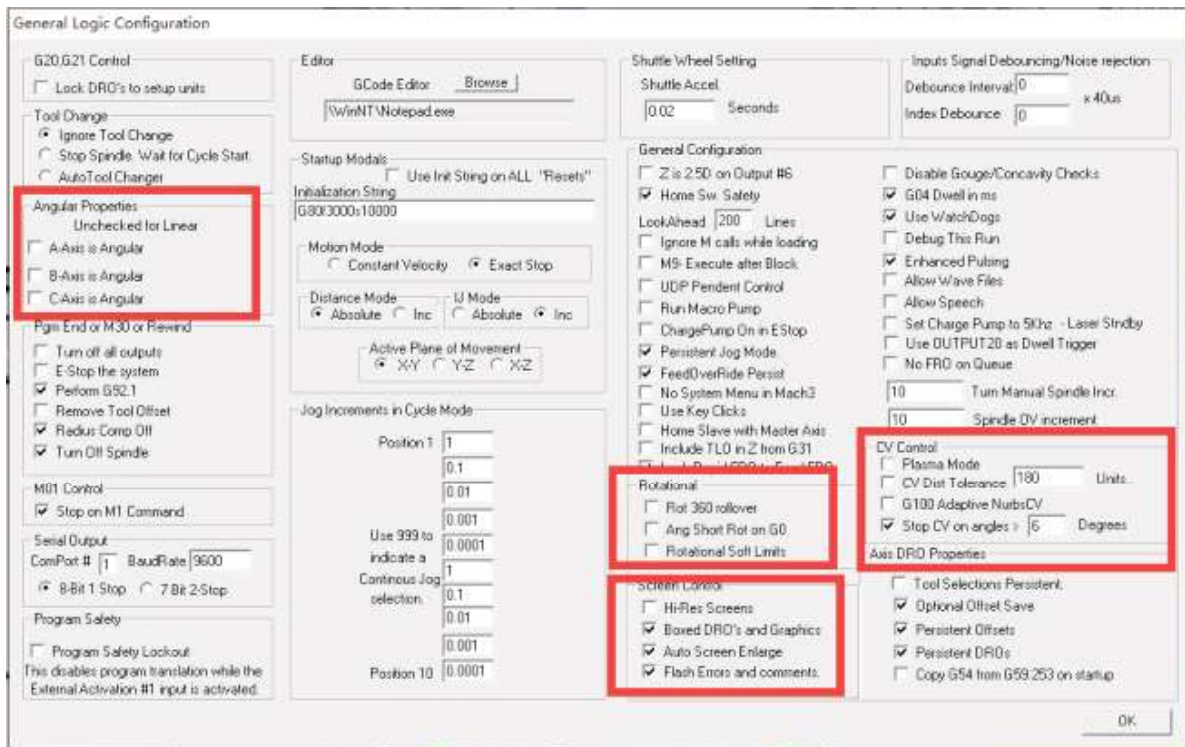
2.6 Motor direction of go to Home

Open Mach3 soft,select "Config" >> "Homing and Limits" option,when you click "Ref all home" button, the origin of the direction of an axis error , you can change "Home Neg" option,tick or cancel tick to make it correct.



2.7 General Config

Open Mach3 soft,select "Config" >> "General Config"option



A-Axis angular: If your A axis is the axis of rotation, and the angle of rotation is 0-360 degrees, please tick it.

"Rot 360 rollover": angle of rotation is 0-360 degrees, please tick it. "Ang Short Rot on G0": if you tick "Rot 360 rollover", please tick it too. Shortest path of circumferential motion.

Cv control: If your arc is not good, it turns into a right angle or a straight line, please modify the value of "Stop CV on angles", and tick it. Until you find a suitable value for you.

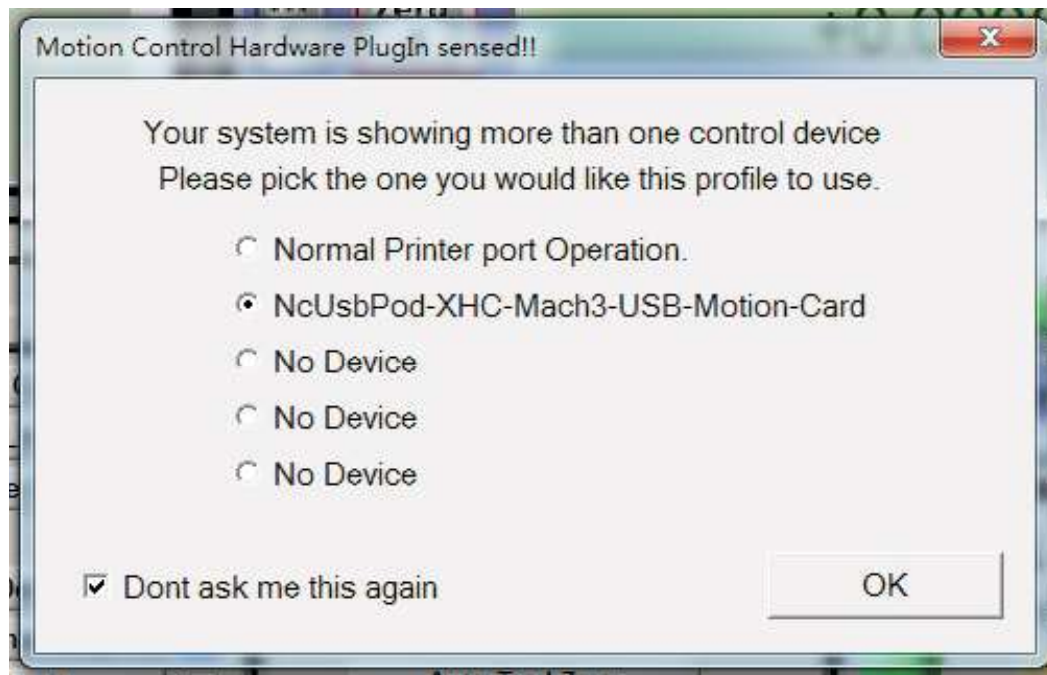
Screen control: When you first open the software, the screen is not full screen display, please tick "Hi-Res screens" and "Auto screen enlarge", Save the settings and reopen the software.



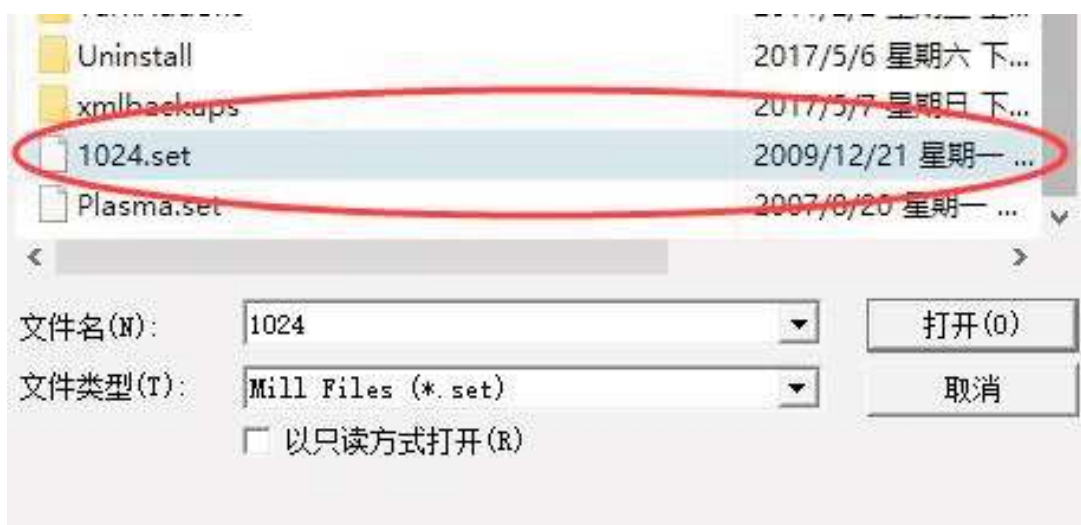
3. Mach3 software operation

3.1 Open and Loading interface

Click on “Mach3mill” icon to open software, If there is an interface to select the plug-in , please chose “NcusbPod-XHC-Mach3-USB-Motion-Card” (see Figure).



If you open the software, there is a blank interface , please chose...View>>Load screens,then chose screen file(see Figure)



3.2 Spindle ON/Off

You can press the "Spindle CW F5" button to open or close spindle.(see Figure), and you can manual input G code : M3 (Spindle on),M5(Spindle off)

Spindle CW F5 :

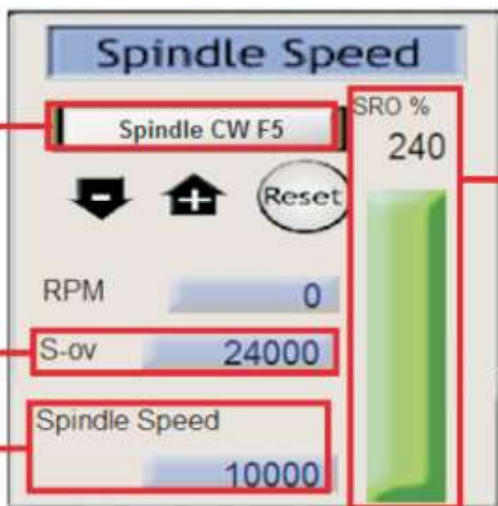
Press the button Spindle on or off.

S-oV :

Spindle current speed

Spindle speed :

Spindle initial speed



SRO% :

Spindle speed percentage,

3.3 Jog move

Press the computer keyboard "Tab" key,

Cycle Jog Step :

press the button,
chose Step value :
1mm'0.1mm'
0.01mm .etc.

Jog Mode :

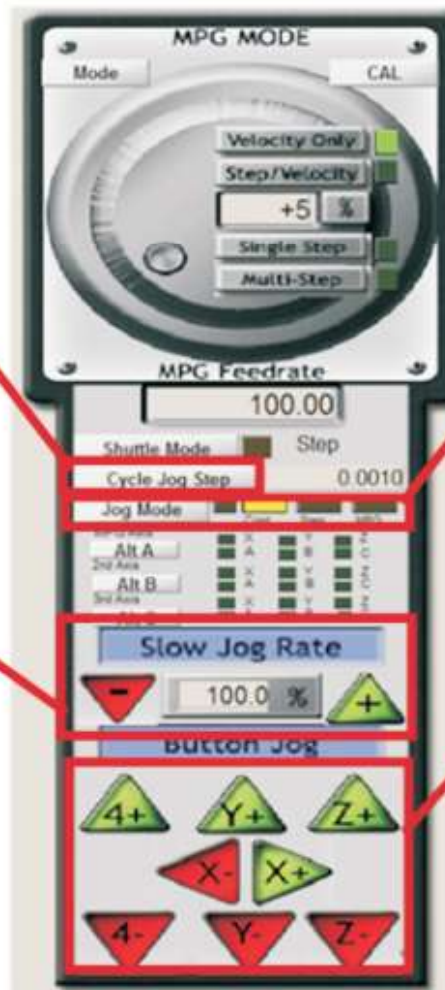
Press the button,
chose jog mode.
"Cont" , " Step"
," MPG"

Slow Jog Rate :

Percentage of Jog
move speed.

Button Jog :

X.Y.Z.4 axis Jog
move buttons,



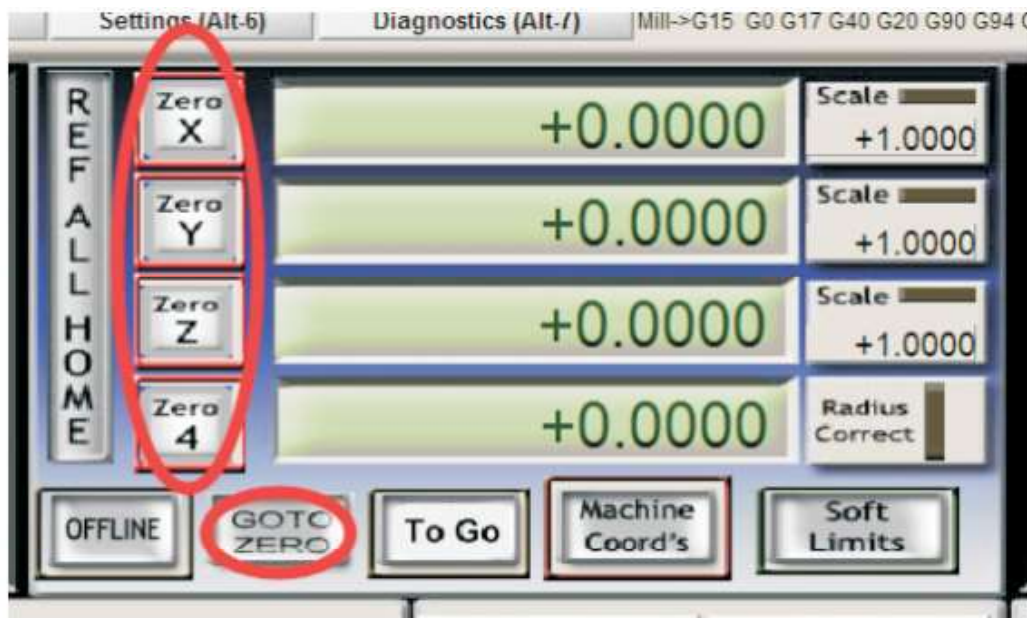
3.4 Go to Home of machine

Press "REF ALL HOME" button, all axis will going to home of machine and coordinate clearing. You can click the "Machine coord`s" to see machine coordinate.



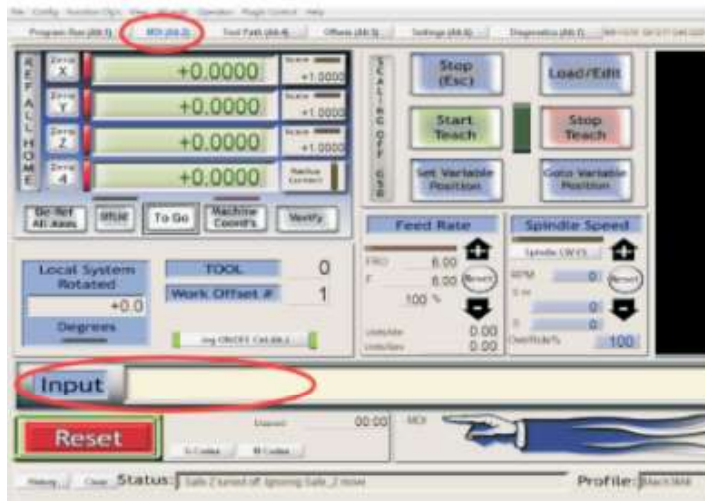
3.5 Go to Zero

Press the "Zero X" " Zero Y" " Zero Z" " Zero 4" buttons, The workpiece coordinates are cleared and set to the Zero of the workpiece. Click the "GOTO ZERO" button go to zero of workpiece.



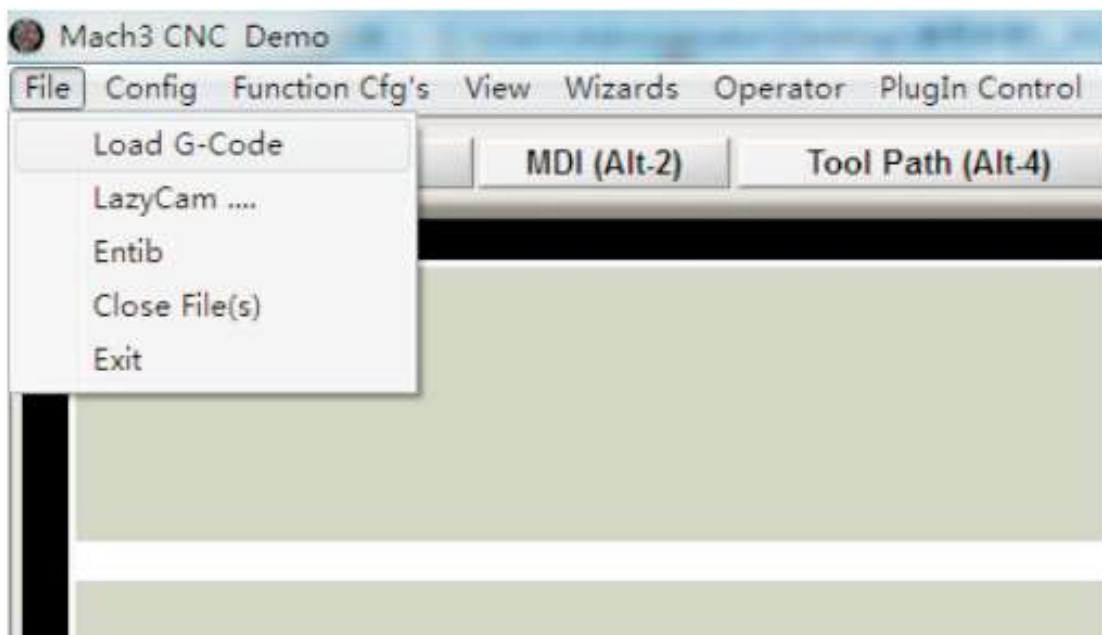
3.6 Manual input G code

When you need input G code, please click the "MDI(Alt-2)" ,then click the "Input" and input G code.

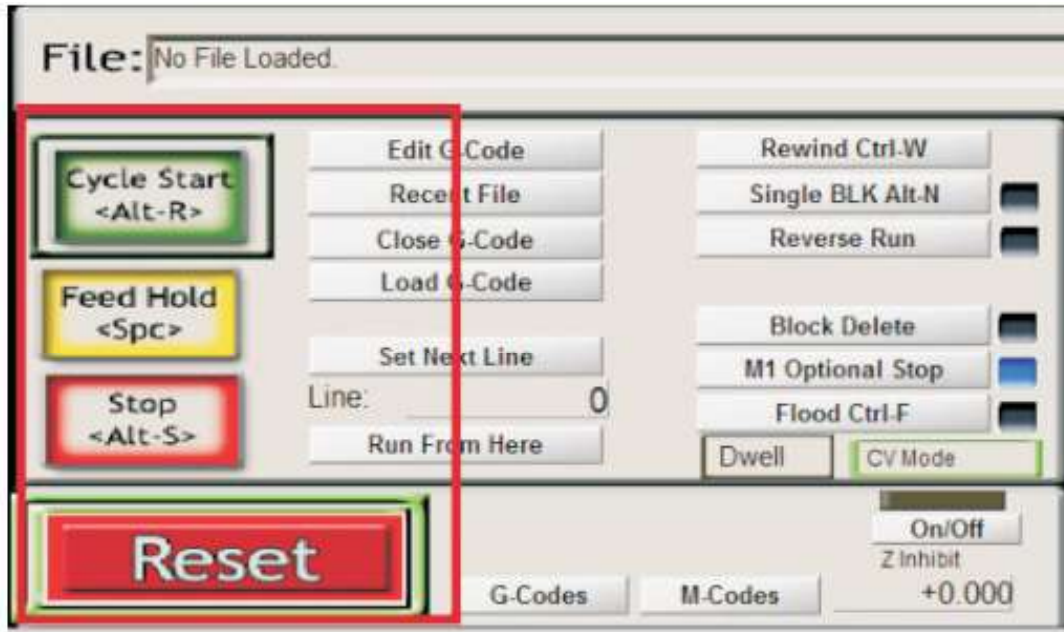


3.7 Load G code

MACH3 software only supports G code files.
Please choose "File" >> "Load G-code", and choose your processing file.



After the program is loaded, click “Cycle Start” and the program starts running; Click “Feed Hold” and program pause; Click “Reset” and software emergency stop.

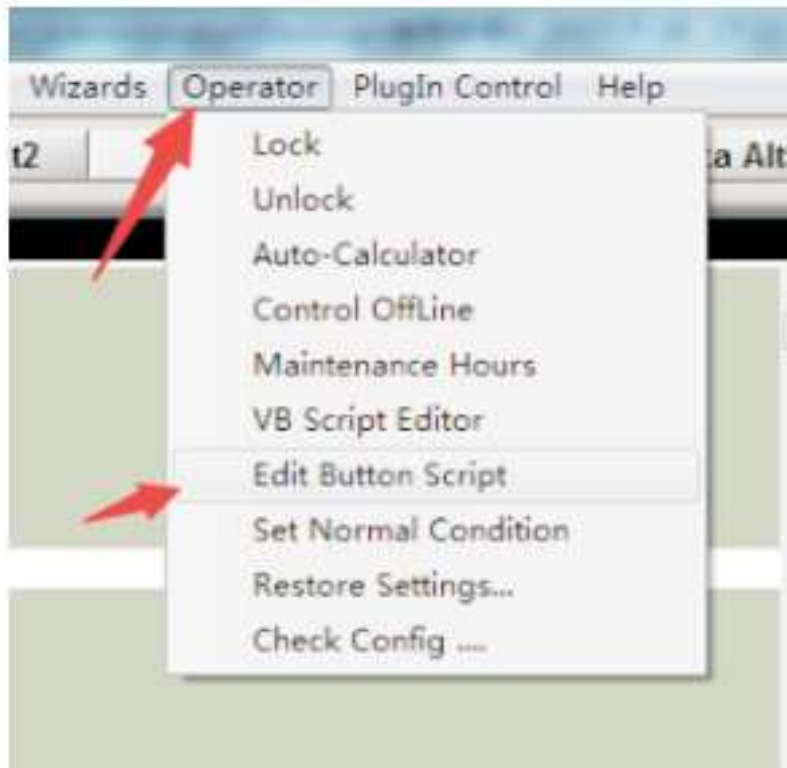


3.9 Z Auto Tool Zero

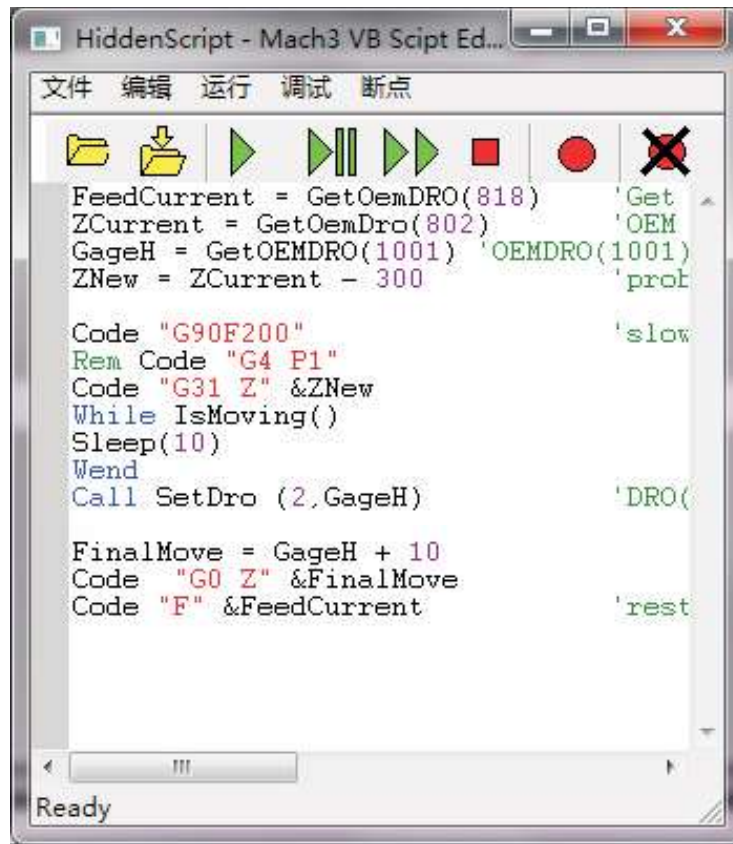
1. Open our CD... English Manual > > macro folder, and copy M930 code to...C:/Mach3/macros/Mach3Mill folder.

2. Open the M930 file in TXT format, then copy all program code. Open MACH3 software, click the "Operator" option, click "Edit button script", (Figure 1). Then click "Auto Tool Zero" button, delete the code in the button script, and then paste the copied M930 code program. (Figure 2)

3. In "Offsets" interface to set Gage Block Height (Figure 2) After the setting is completed, click "Auto tool zero" .



Software operation

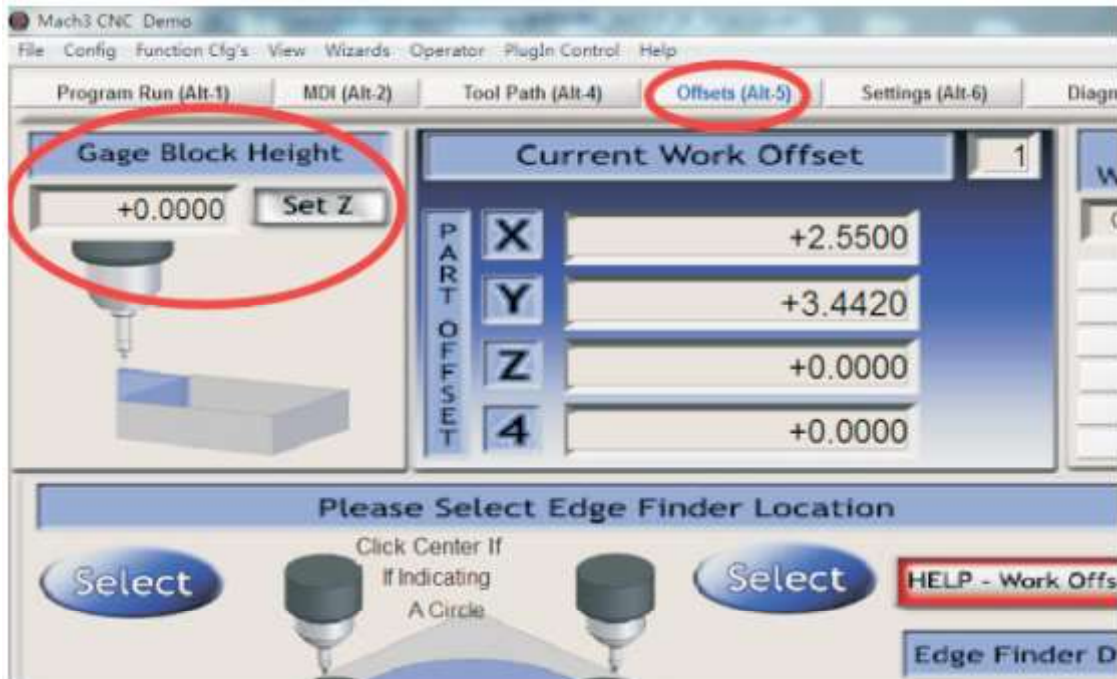


```
HiddenScript - Mach3 VB Script Ed...
文件 编辑 运行 调试 断点
FeedCurrent = GetOemDRO(818) 'Get
ZCurrent = GetOemDro(802) 'OEM
GageH = GetOEMDRO(1001) 'OEMDRO(1001)
ZNew = ZCurrent - 300 'prob

Code "G90F200" 'slow
Rem Code "G4 P1"
Code "G31 Z" &ZNew
While IsMoving()
Sleep(10)
Wend
Call SetDro (2.GageH) 'DRO(

FinalMove = GageH + 10
Code "GO Z" &FinalMove
Code "F" &FeedCurrent 'rest

Ready
```



Mach3 CNC Demo

File Config Function Cfg's View Wizards Operator Plugin Control Help

Program Run (Alt-1) MDI (Alt-2) Tool Path (Alt-4) **Offsets (Alt-5)** Settings (Alt-6) Diagn

Gage Block Height

+0.0000 Set Z

Current Work Offset 1

PART OFFSET	X	+2.5500
	Y	+3.4420
	Z	+0.0000
	4	+0.0000

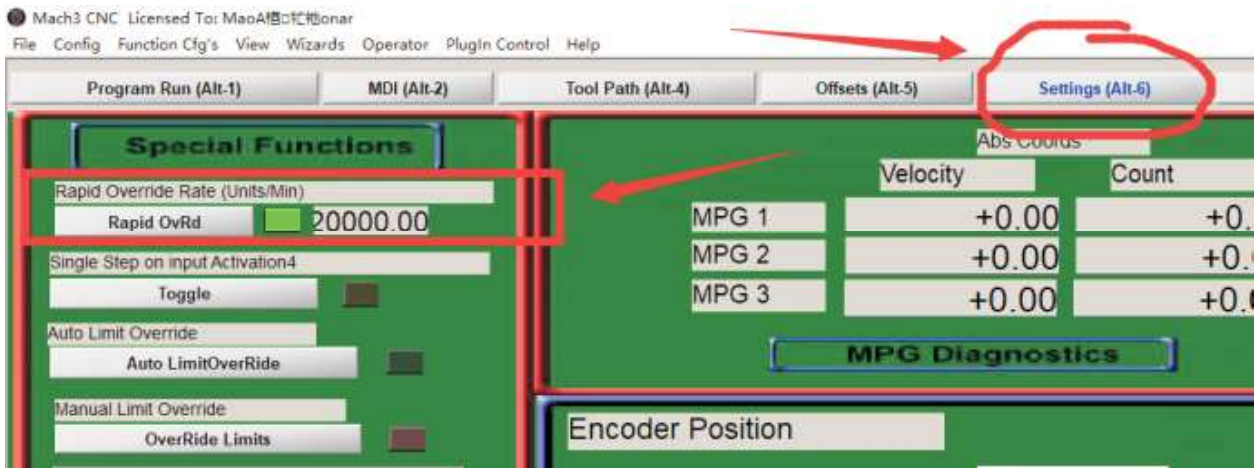
Please Select Edge Finder Location

Select Click Center If Indicating A Circle Select

HELP - Work Offs

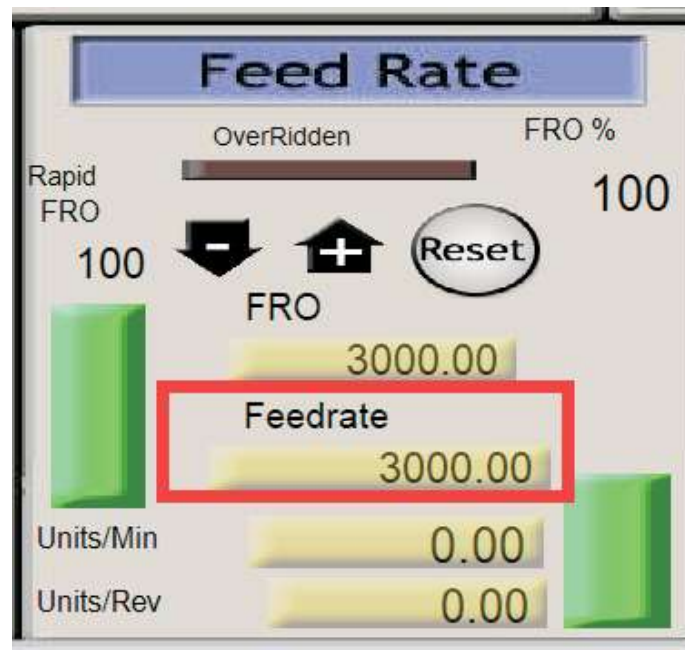
Edge Finder D

3.10 Set G0 speed



3.11 Set Feed speed

- 1、 In G code, F code to set feed speed.
- 2、 In Feed Rate to set feed speed.





Chengdu XinHeCheng Technology Co.,Ltd

<http://www.cdXHctech.com>