Installation and configuration the Control Software

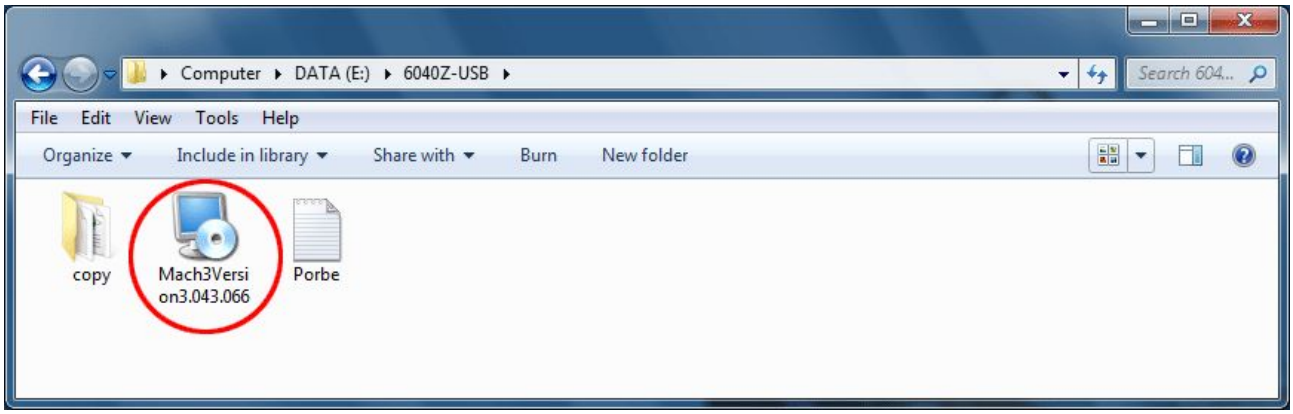
●Requirements for the configuration of master control computer:

①The desktop computer or Laptop must possess USB2.0 ports. (Does not support USB3.0), USB port output voltage can not be less than 4.8V.

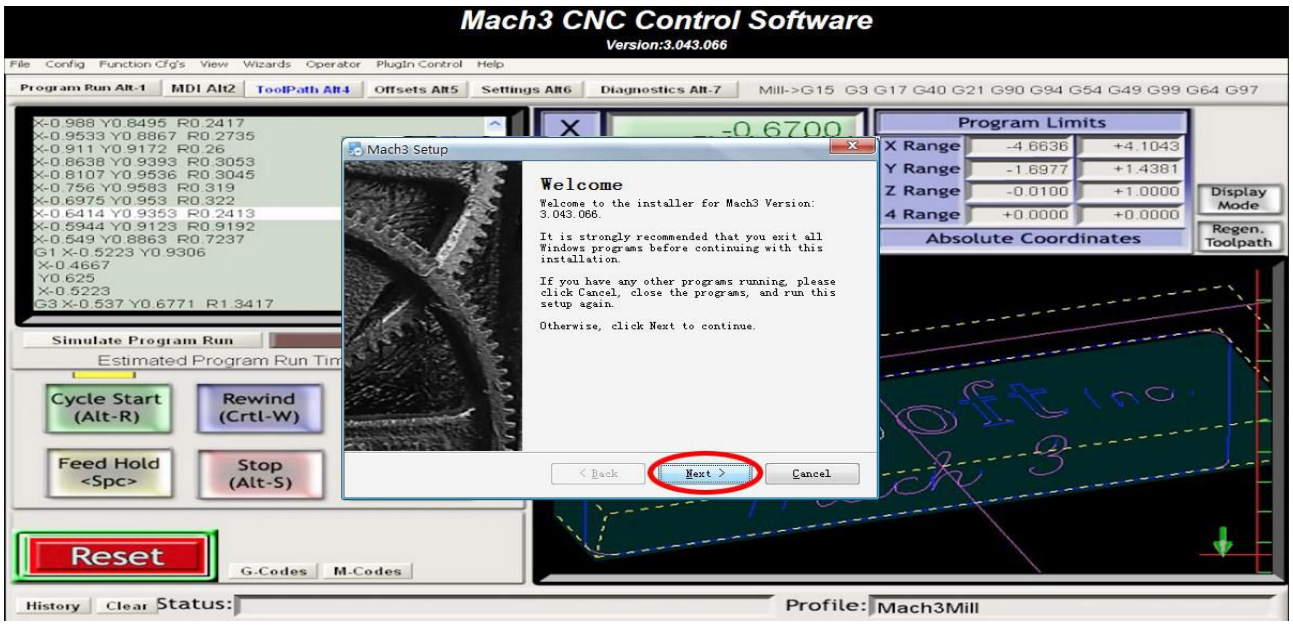
②Use only Windows XP and Windows 7 operating systems. (Currently does not support Windows8 and Windows10.)

●Installation the control software【MACH3】

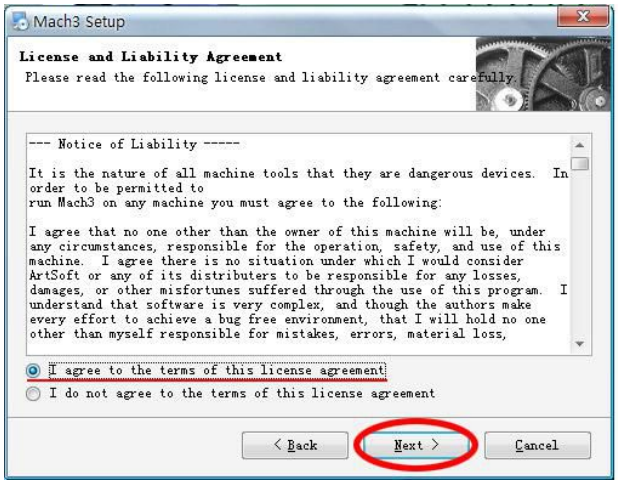
1.Double click on the icon to open the file(Mach3.exe).



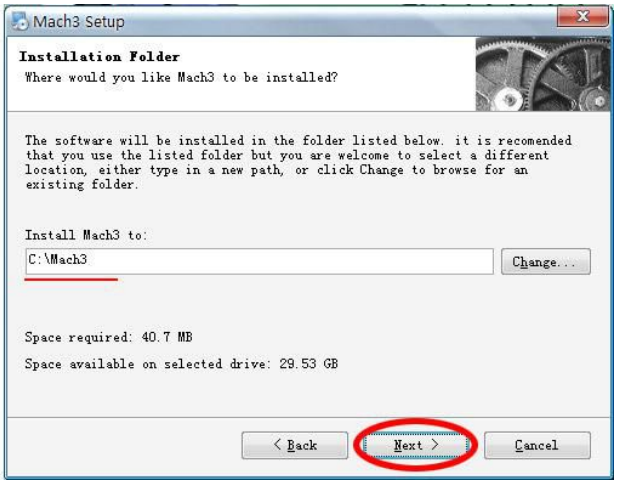
2.Click the "Next".



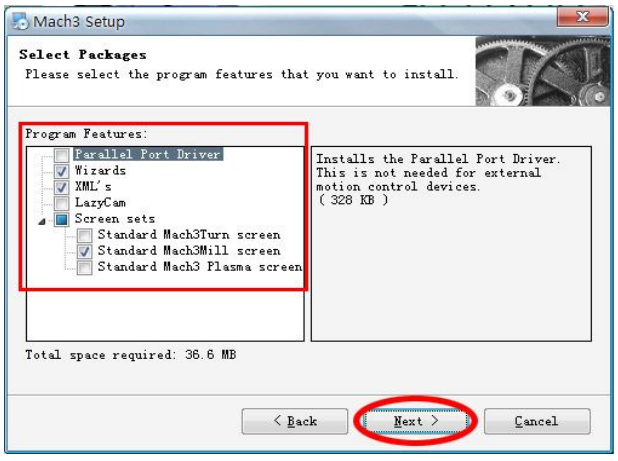
3.Click the "Next".



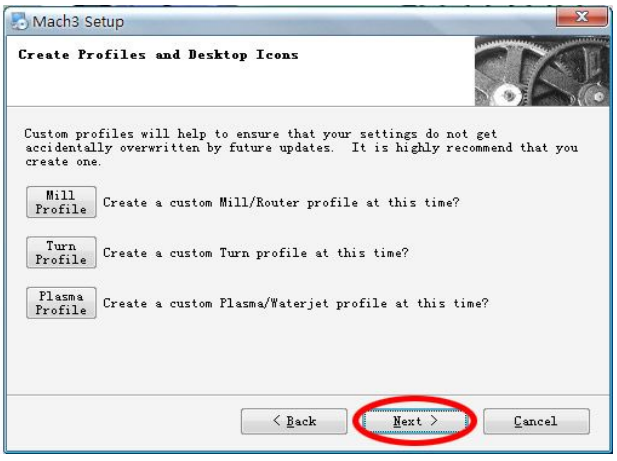
4.Click the "Next".



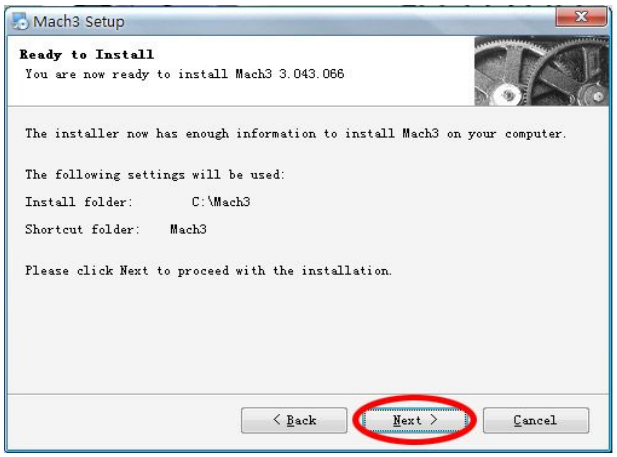
5.As shown selection,Click the "Next".

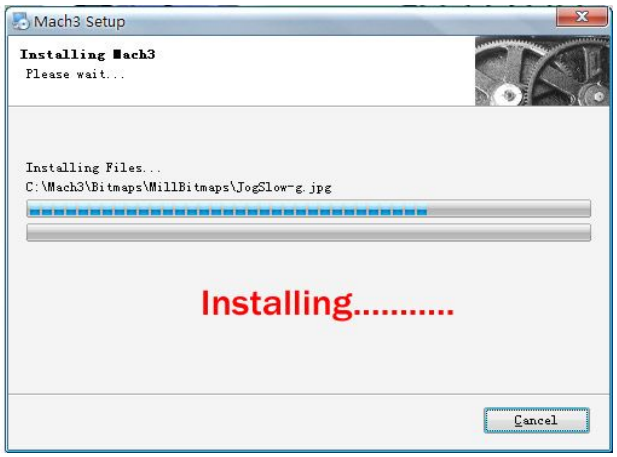


6.Click the "Next".

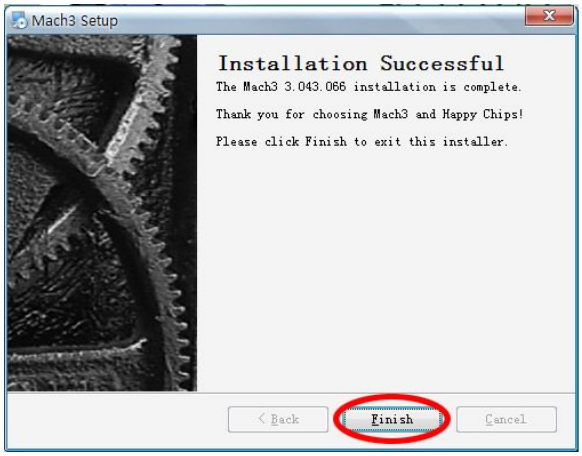


7.Click the "Next".





8.Click the "Finish". Mach3 installtion is complete.

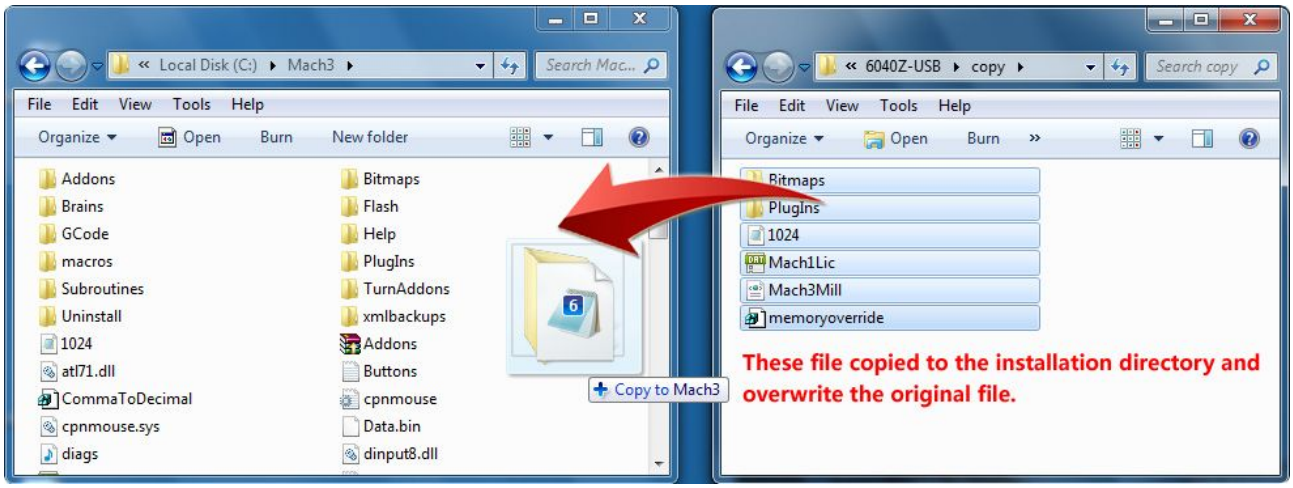


Software installation is complete,

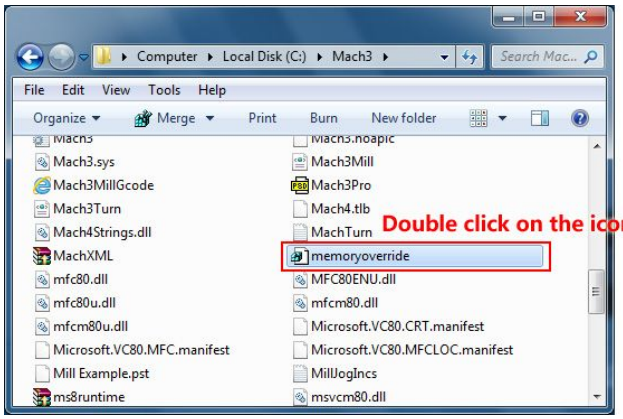
Now restart your computer.

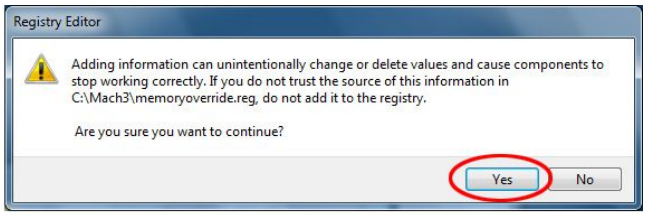
9.Restart the computer.

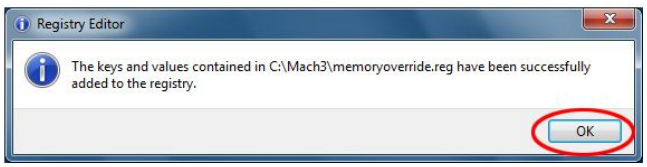
10.then we need to setup the Mach3 software configuration, so that it could be able to control the engraving machine to work.
Open the 'C' disk MACH3 installation directory, follow as shown below .



Note:memoryoverride.reg is a Registry file for Windows 7.
if you use Win7, double click this file to run it . if you use WinXP, ignore the file.

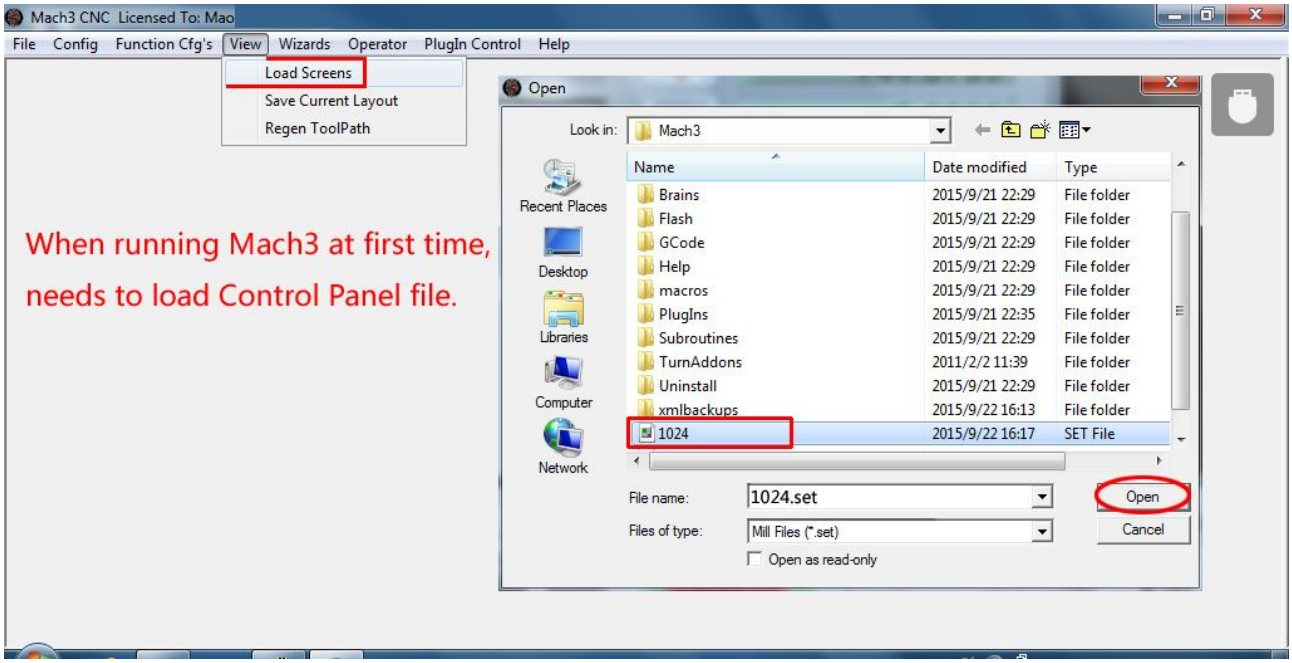




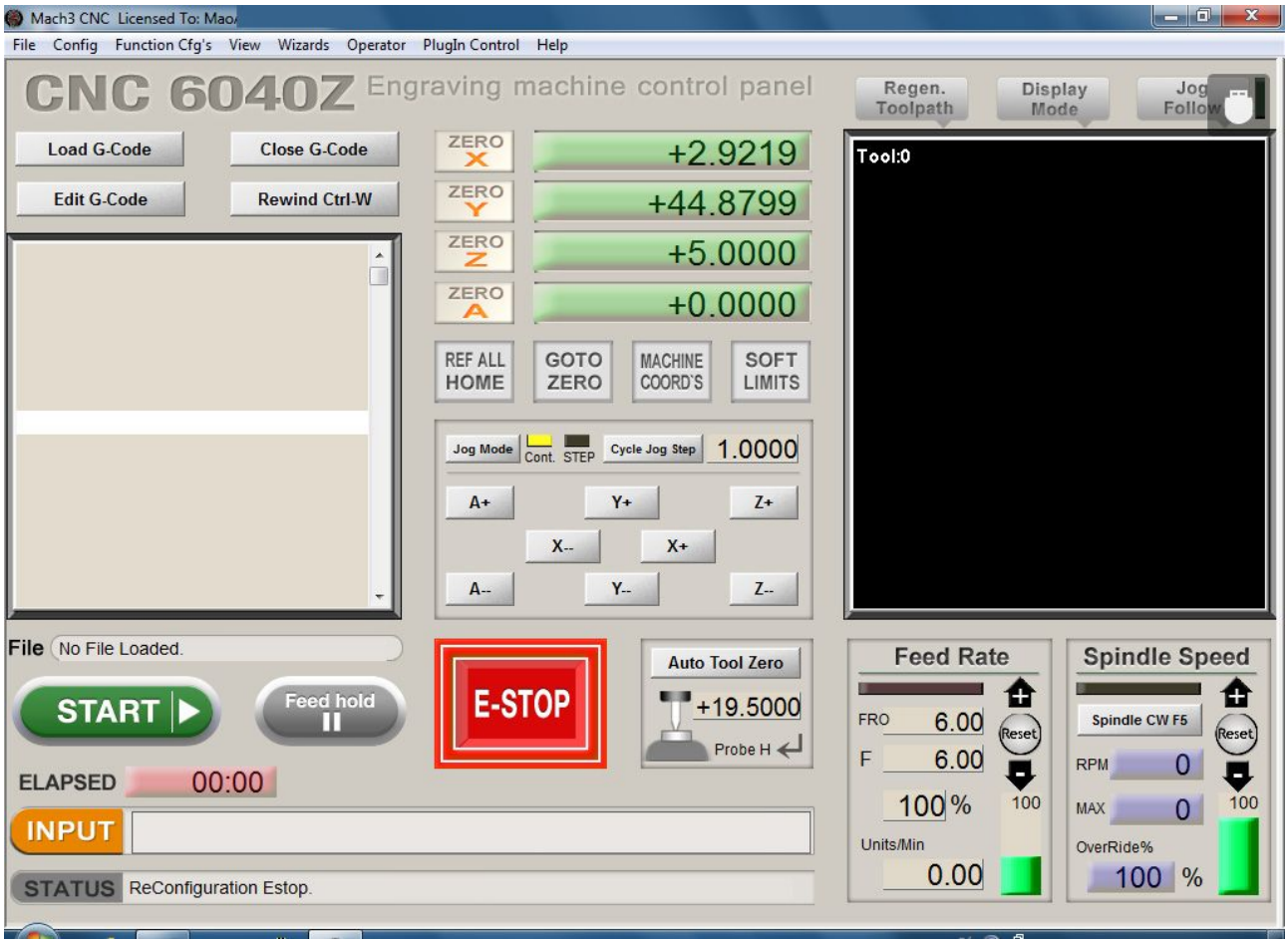


11.Double-click "Mach3Mill" to run the software. 

when running Mach3 at first time, needs to load 6040Z Operation interface file.



Now you seeing is 6040Z CNC engraving machine dedicated Operation Interface. All operations of the 6040Z will be controlled through this Operation interface.



The STATUS field will display useful status and error messages.

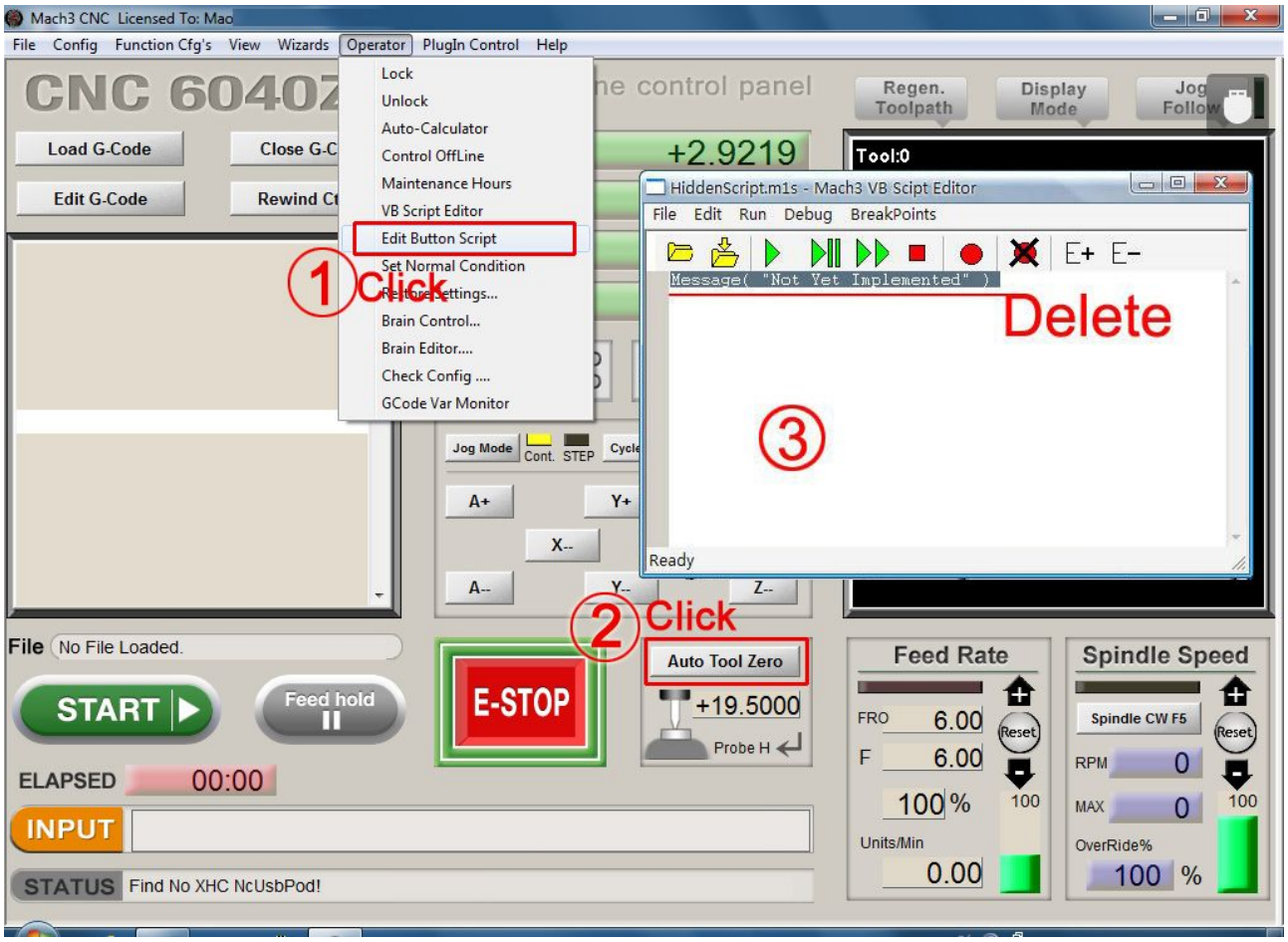
If STATUS field displayed: "Find No XHC NcUsbPod!", please check that:

1) USB cable connection.

2) Computer's USB port. (must be 2.0, and check whether good or bad)

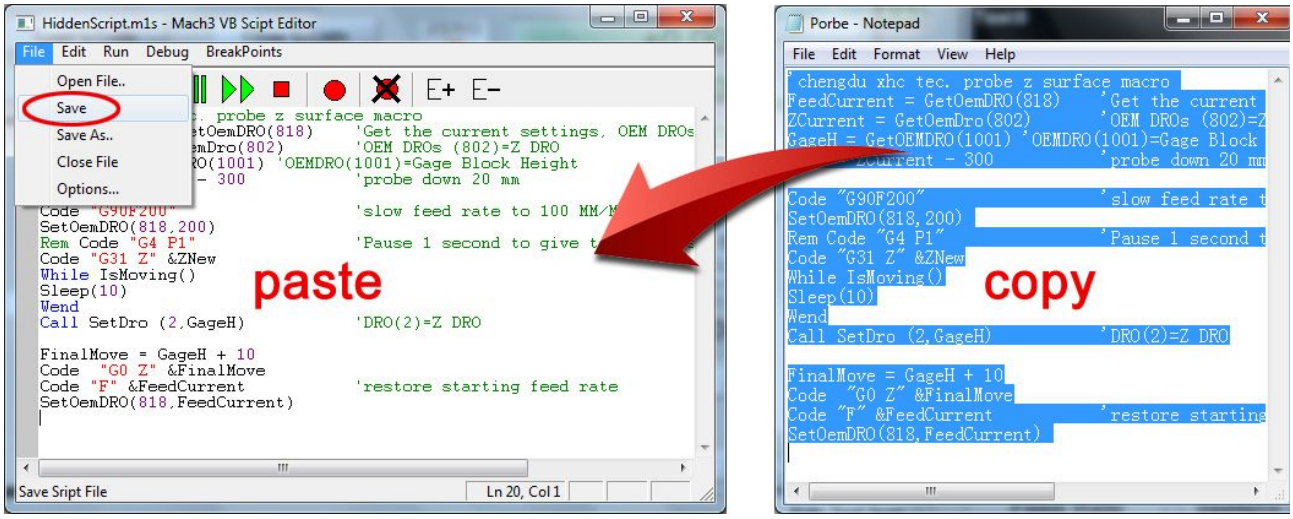
when running Mach3 each time, the "E-STOP" button is flashing, click it, it'll be turned to green frame and stop blinking, you can start using Mach3 now.

12. Now let`s setting up “Auto Tool Zero”.



Open "Porbe.txt", copy all to "HiddenScript.m1s" , then save.

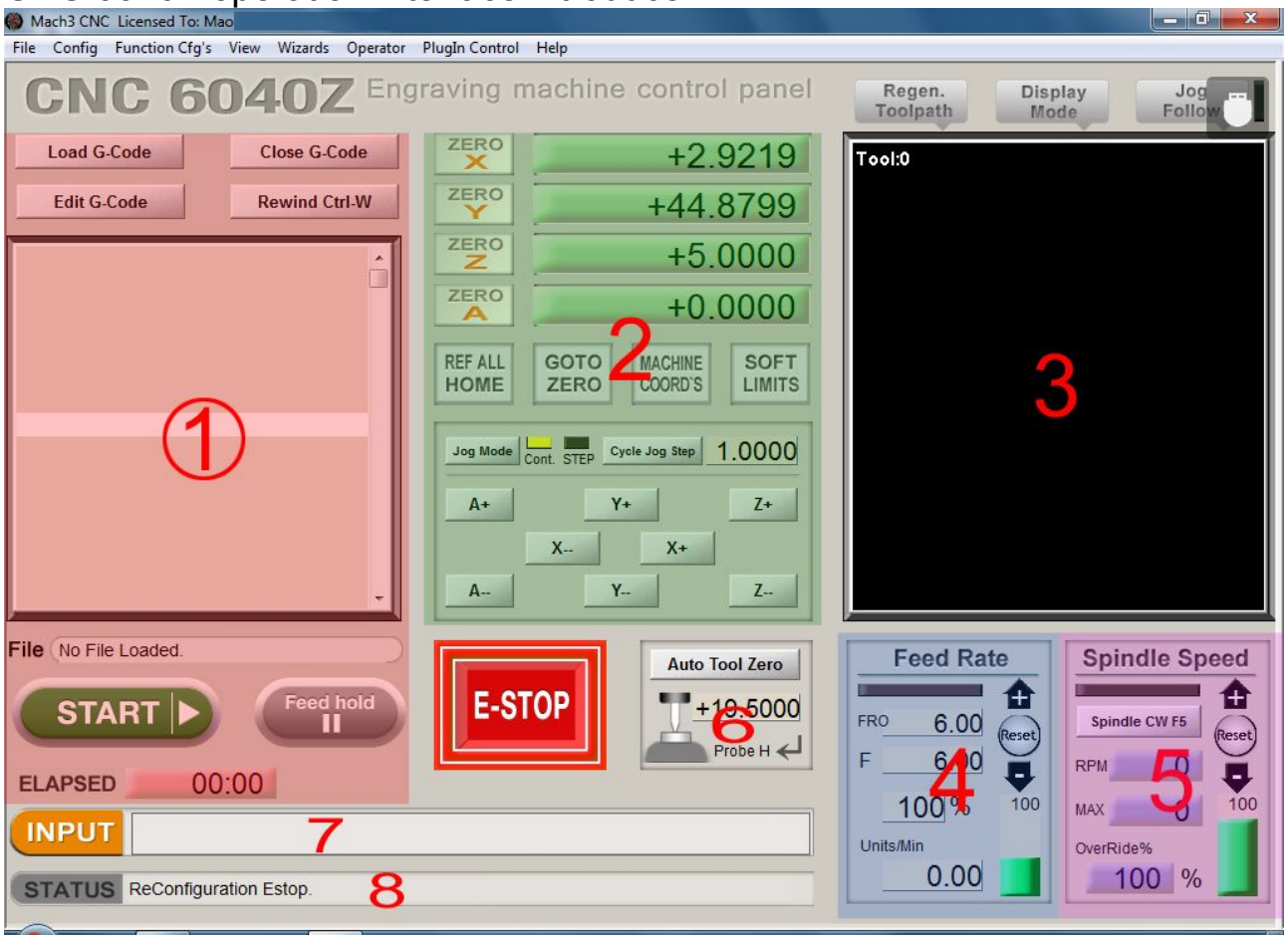
Copy all contents of "probe.txt" into "HiddenScript.mls" and save it.



Thus far,MACH3 installation and configuration has been completed，now you can use it.

Double-click on the desktop icon "Mach3Mill" to run the software.
(Other icons generated by Mach3 on the desktop can be deleted, they are not used for the engraving machine.)

CNC 6040Z operation interface introduce:



1, Code field: load G-code, modify or run. when the professional CNC design software , you can load the tool-path file file in this field,then start to engraving process.
supports most standard G-code files and ".cnc",".nc",".tab" etc file.

2, Coordinate area: the position information of each axis coordinate, manually operation for control each axis , positioning workpiece origin.
[The "REF ALL HOME" and "SOFT LIMITS" functions only can be used for the customized machine which has been installed limit switches.]

3, Trajectory field: display the running track of toolpath.

4, Feed Rate field: Feed speed adjustment;

5, Spindle Speed field: control spindle motor to run/stop, speed adjustment.

6, Probe(tool-presetter) field: preset tool for Z-axis and cleared.

7, handwritten code field: If you are proficient in G-code, you can directly type G-code and run in this field.

8, STATUS field: status information and troubleshooting tips;