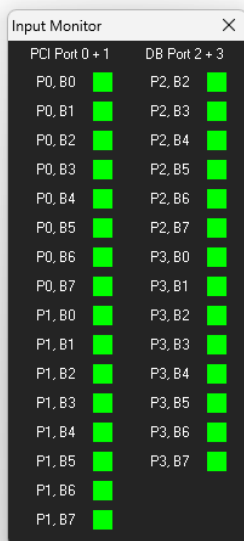# Initial Machine Setup

Please note that this is intended as a basic guide to initially setting up your CNC machine. All machines differ, and it is best to use the manufacturer's recommended set-up procedures.

Step 1: Check for functionality of all limit switches.

To do this, open WinCNC, click on "View", and then click "Inputs". For the next step, it is best to have a second person assist the tester. You should be looking at a screen similar to the following picture:

Input Monitor	✕
PCI Port 0 + 1	**DB Port 2 + 3**
P0, B0 ■	P2, B2 ■
P0, B1 ■	P2, B3 ■
P0, B2 ■	P2, B4 ■
P0, B3 ■	P2, B5 ■
P0, B4 ■	P2, B6 ■
P0, B5 ■	P2, B7 ■
P0, B6 ■	P3, B0 ■
P0, B7 ■	P3, B1 ■
P1, B0 ■	P3, B2 ■
P1, B1 ■	P3, B3 ■
P1, B2 ■	P3, B4 ■
P1, B3 ■	P3, B5 ■
P1, B4 ■	P3, B6 ■
P1, B5 ■	P3, B7 ■
P1, B6 ■	
P1, B7 ■	

Have one person continue to monitor this Input Monitor, while the second physically finds and triggers each of the limit switches. It is best to use a worksheet such as the following:

X Axis Lo switch (usually toward the left side of the table): Port ____ Bit ____
X Axis Hi switch (usually toward the right side): Port ____ Bit ____

Y Axis Lo switch (usually at the front of the table): Port ____ Bit ____
Y Axis Hi switch (usually at the back): Port ____ Bit ____

Z Axis Lo switch (there may not be one present): Port ____ Bit ____
Z Axis Hi switch: Port ____ Bit ____

Additional switches _____: Port ____ Bit ____
Additional switches _____: Port ____ Bit ____

It is important to know exactly which port and bit goes with which switch for proper limit switch functionality.

Once the switches have been identified, the correct port and bit must be entered into the WinCNC.ini file.

First make sure that your limit switches have not been commented out of the code in the WinCNC.ini file. Locate the WinCNC folder on the C: drive of the computer. Inside that folder, locate the file named WinCNC.ini file.

If you are unable to see file extensions, do the following: With the WinCNC folder open, click on the menu item "Tools" at the top of the page. Then click "Folder Options". Next click the "View" tab at the top, and scroll down until you find the heading "Hide extensions for known file types", and make sure there is NOT a check in that box. Click "Apply", and then "OK". You should now be able to see all file extensions.

Double click the WinCNC.ini file. This file should open with Notepad, or WordPad. It is an editable text file. Scroll down until you see text similar to the following:

[X Axis]
axisspec=p0 s1 d1 r500 e1 a1000
axisvel=r1800 f5000 s50 m400 h5000 c10
axishi= p3 b2
axislo= p3 b3

[Y Axis]
axisspec=p0 s0 d0 r500 e1 a1000
axisvel=r1800 f5000 s50 m400 h5000 c10
axishi= p3 b4
axislo= p3 b5

[Z Axis]
axisspec=p0 s2 d2 r100 e1 a500
axisvel=r100 f500 s5 m20 h1000
axishi= p3 b6
[axislo= p3 b7   **NOTICE this limit switch has been commented out

If any of the "axislo=" or "axishi=" lines have a square bracket "[" in front of them, then those lines have been commented out (meaning essentially that they have been disabled). If you do not have a limit switch, for Z axis lo for instance, you will want that commented out. Make sure you only have a square bracket "[" in front of any limit switches that you do not have on your machine.

The above is only an example. Your files will differ. To make sure you have the correct limit switches for the correct axis, refer back to your worksheet. Find the port and bit you recorded for the X axis, hi switch, and make sure that the same port and bit is in your WinCNC.ini file. For example, let's say you recorded your X axis hi switch as port 3 bit 1. However in your WinCNC.ini file it has "axishi = p3 b2". You need to change that line so that it says "axishi = p3 b1". Follow the same procedure for all axis hi and lo switches. Make sure to save the changes, and back up your new WinCNC.ini file.

Once you have made these changes, it is important to double-check that the limit switches are working correctly. To do this, have the second person physically go to the machine, and one at a time click each limit switch. The tester needs to be watching the WinCNC screen for the following confirmation of limit switch functionality. When the limit switch is activated, an arrow will appear to the left or right of the axis letter, depending on if it is the lo or hi switch.

The following picture represents the X lo switch being activated.
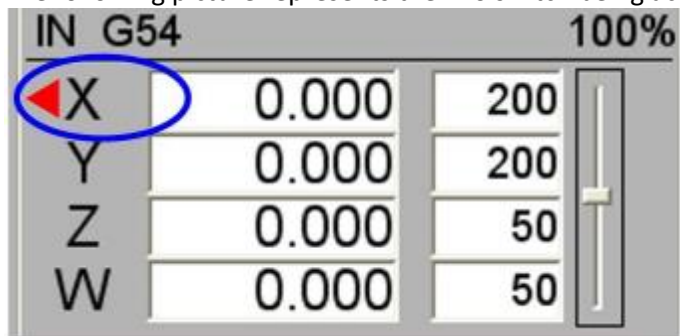
The next is a picture of the Y hi switch being activated.
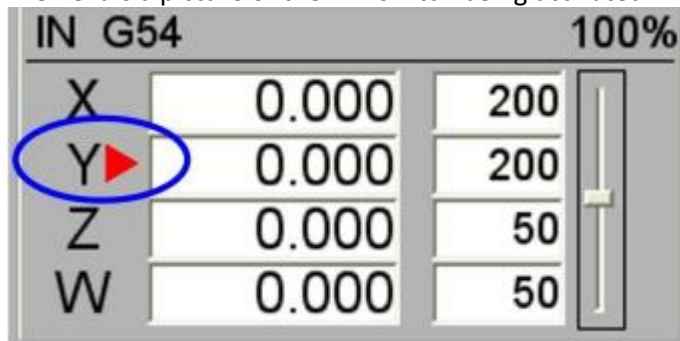
Once all limit switches have been checked, you are ready to proceed to Step 2 of the set up procedures.

Step 2: Checking for correct motor direction and travel.
Before actually homing the machine, it is important to make sure all of the axis are traveling in the desired direction. For instance, when pressing the transit key on the keyboard, the X Axis should travel to the right. And when pressing the on the keyboard, the Y Axis should travel to the front of the table. If any one, or all of the axis are traveling in the wrong direction, this must be addressed before homing the machine. To fix an incorrect motor direction, the value "o0" or "o1" must be added to the axisspec= line in the WinCNC.ini file. That value is the letter "o" (pronounced "oh"), not a zero. Lets say the X Axis is reversed and traveling in the wrong direction. Open the WinCNC.ini file and locate the lines that are similar to the following:

[X Axis]
axisspec=p0 s1 d1 r500 e1 a1000
axisvel=r1800 f5000 s50 m400 h5000 c10
axishi= p3 b2
axislo= p3 b3

As you can see, the axisspec= line does not contain either "o0" or "o1". In most cases, adding "o1" to the line will reverse the motor direction. The line would then read: axisspec=p0 s1 d1 r500 e1 a1000 o1. There have been a few cases in which adding "o1" did not change the motor direction, but adding "o0" did. If the line already contains either "o0" or "o1", then change the value to the opposite of what is already there. Make sure to save the WinCNC.ini file after making any changes, and **always restart WinCNC after any changes are made**.

After making sure that all axis move in the desired direction, proceed to Step 3.

Step 3: Checking for correct distance travel.
At this point, your machine should be traveling in the correct direction for each axis, and all of the limit switches should have been tested and proven functional. Now you need to make sure that your CNC machine travels the correct distance for each axis. The following test can be repeated for each axis. This test is proven most accurate with a bit in the router (other machine types will have other methods of testing).

Transit the X Axis nearly all the way to the left side of the table. Lower the Z Axis until it is just barely touching the spoil board. Mark the spoil board with a marker or pencil, and type G92X0. This will set a temporary 0 on X. Now raise the Z Axis high enough to clear anything on the table. In WinCNC, command X to travel nearly the entire width of the table. For instance, if the table width is 5 feet wide, and the X Axis can safely travel 4 feet, then type X48 (4 feet in inches) into the command line, and hit enter. Now lower Z back down to the spoil board, and mark that spot. Now measure the distance of actual travel between the two marks. You will have to calculate a correction for the WinCNC.ini file based on your findings. Use the following equation:

Commanded distance ÷ actual distance = x value
X value * current resolution (axisspec r#) = new resolution

Example: (based on "axisspec=p0 s1 d1 r500 e1 a1000")

36 inches commanded 31 actual inches moved = 1.6129
1.16129 * 500 = 581.45

Your new axisspec line would read "axisspec=p0 s1 d1 r581.45 e1 a1000"

Repeat Step 3 for all axes.

Once all of the axis move the correct commanded amount, proceed to Step 4.

Step 4: Homing the machine.
Manually transit the X Axis all the way to the left, until it is barely touching, or just off of the limit switch. If the axis positioning on the WinCNC screen is very close to zero, type into the command line "x0" and then enter. If the X Axis is physically positioned as close to zero as you can get, and the axis positioning on the screen reads differently, then the position in WinCNC needs to be re-set. To do this, physically

transit the X Axis to where zero should be, and then type into the command line "L28X" and hit enter. This will zero the position in the WinCNC program, but will NOT physically move the machine. This can be repeated for all axis.

Next, try to home each axis, one at a time. To do this, type into the command line G28X and enter, and let the X Axis finish homing. Then try the Y Axis by typing G28Y, and so on. Once all of the Axis have been homed one at a time with no problems, try a general homing, by typing G28 and enter.

If each axis homed ok individually, but problems present themselves when they home together, you may need to change the WinCNC.ini file so that they will always home separately. To do this, open the WinCNC.ini file, and locate the following:

[G28 Settings]
g28move=z-.25 x1.5 y1.5 f500

To separate each axis, type a new g28 move for each axis, and add "m1" to the end of each line as follows:

[G28 Settings]
g28move= x1.5 f500 m1
g28move= y1.5 f500 m1
g28move= z-.25 f500 m1

Please keep in mind that the values presented above are examples. Use the values already provided in your WinCNC.ini file.

Contact technical support at Microsystems World CNC for any further questions, or for additional help. (304) 472-7206.