User Manual

Table of Contents

Getting Started

Included in the documentation with this kit are the instructions for how to set up the control board. After going through this manual several times myself, I didn't need it anymore, but I don't find it very suitable for beginners.

I will continue writing manuals for those who want to go more in depth, and learn how all the components work together. However, the multiCNC kit is meant for the average person, who would likely be put off by the "made in China" manual

The kit comes with 4 stepper motors, control board, power supply and a cable, which plugs into the PC's printer port. Yes, it has to be the 25 pin connector most computer no longer have. A USB connector, likely all new printers come with, doesn't work, not even with an adapter cable.

There are several switches on the control board, which are specifically set for the supplied motors and should not be changed unless you go through the more detailed manual.

There are two very important steps in order to operate the multiCNC safely. You should plug the power supply for the control board into a separate power strip with its own switch and separate from the PC. That way, just in case anything goes wrong, you can cut the power to the CNC machine and don't have to reboot the computer or even worse lose your CNC program. The one thing you should never do is disconnect anything from the control board while the power is on. If you do, it will most likely destroy the control board.

The Motors

The motors in this kit are stepper motors, which are quite different from the ones in your car for power windows, seats and mirrors. As the name says stepper motors rotates in small steps with a total of 200 full steps per revolution. In order to do that, the controller board turns the motor coils on and off very quickly. Imagine the poor electrons in the coils running back and forward all the time, they are rebelling a little by heating up the motor. It also means stepper motors are not particularly fast. Usually at 4-5 revolutions per second the motor stops spinning and the motor shaft just vibrates.

Stepper motors also have more than 2 wires to connect, that is because each coil has two ends of the wire. If you like to think in terms of North and South pole magnets and each coil has 2 wires, than this makes 4 wires, which are indeed connected to the control board. You should also know, that these motors operate from the 24 V power supply and it is safe to touch any metal parts on the machine or even an exposed wire. On the other hand it is a motor with quite a lot of power and you should not try to stop the motor by hand.
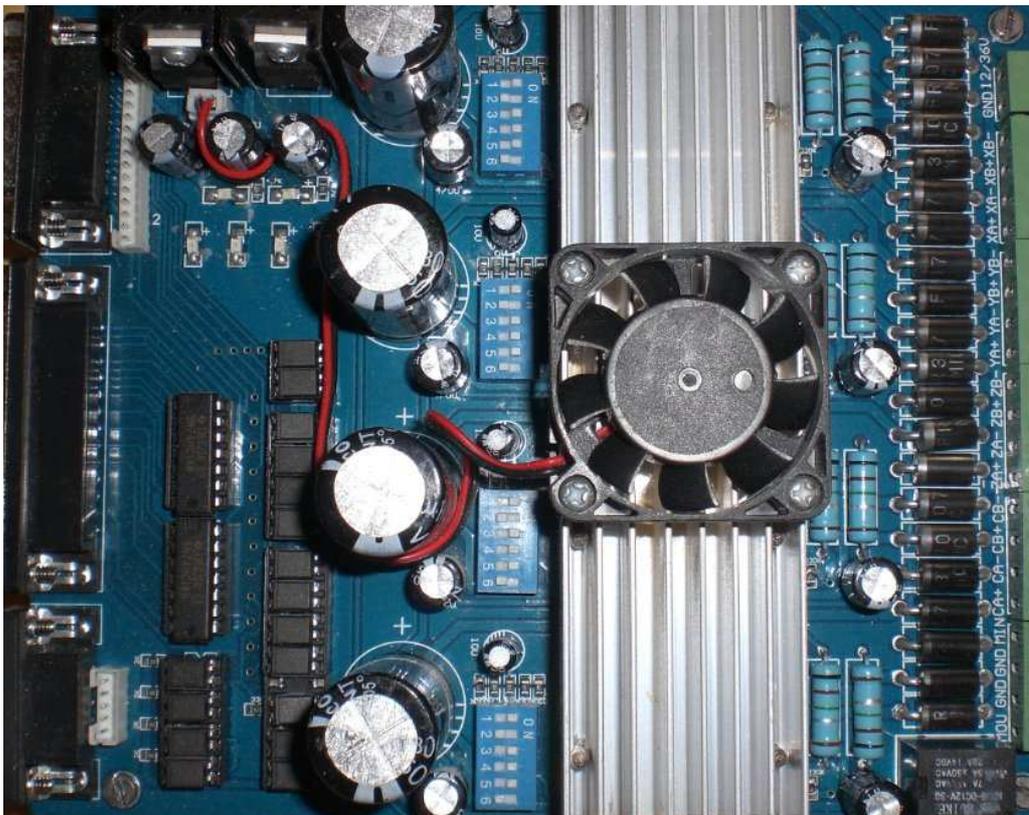
The Control Board

In the final version of this CNC kit the control board is hidden together with the power supply in a case. All you are exposed to are the connectors for the 4 motors, which are labeled X, Y, Z and A.

In the CNC router-mill version we are using Y and A motor together, in the hot wire cutter version all 4 axis operate independently.

Some LEDs on this board indicate that the board is connected to the computer. They are even on when the 24 V power supply is off, because they are powered through the cable from the PC. 4 more LEDs come on when the motors become active.

Some other options may be added to the CNC machine, like limit switches and E-Stop, which are described in the extended manual.

Switch settings: ON, OFF, ON, ON, ON, OFF
(75% current, fast buffer, ½ step mode)

Setting up the Computer with Mach3

Although other software packages claim to work as well with the controller board, I only have experience using Mach3. I know it works and has done so extremely well.

Download the demo version of Mach 3
http://www.machsupport.com/downloads.php
and install it on your PC.

You will be asked if you want to create a new icon for your new machine und you should do so. You should also be reboot when requested to do so.

After rebooting you will get some new icons on your PC.

You will not need the mach3 loader, turn and plasma. LazyCam is a very useful tool to create a CNC program even from an image, but the program requires quite some clean-up before you can use it on the CNC machine. On the other hand, I have to admit that I have to spend more time with it to get it to work right.

Ok, click on your newly-created icon, I named mine plywood2.
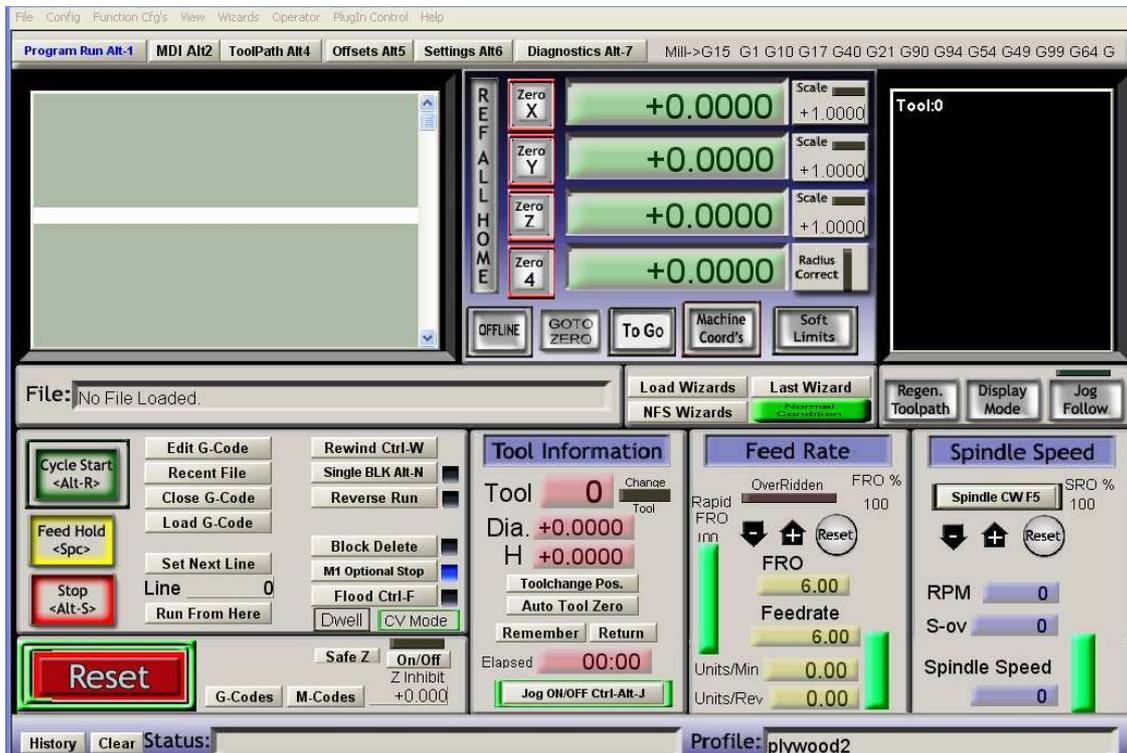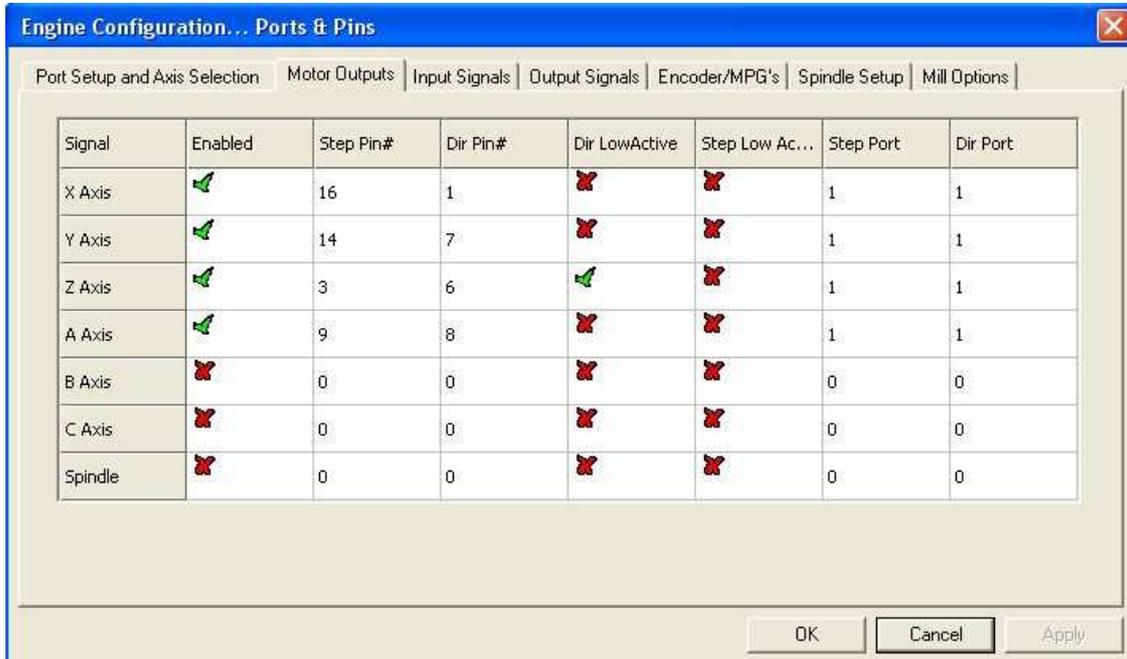
The reset button should be flashing, don't click on it yet.

Instead go to the configuration at the top bar and click on ports and pins you should be seeing this window:
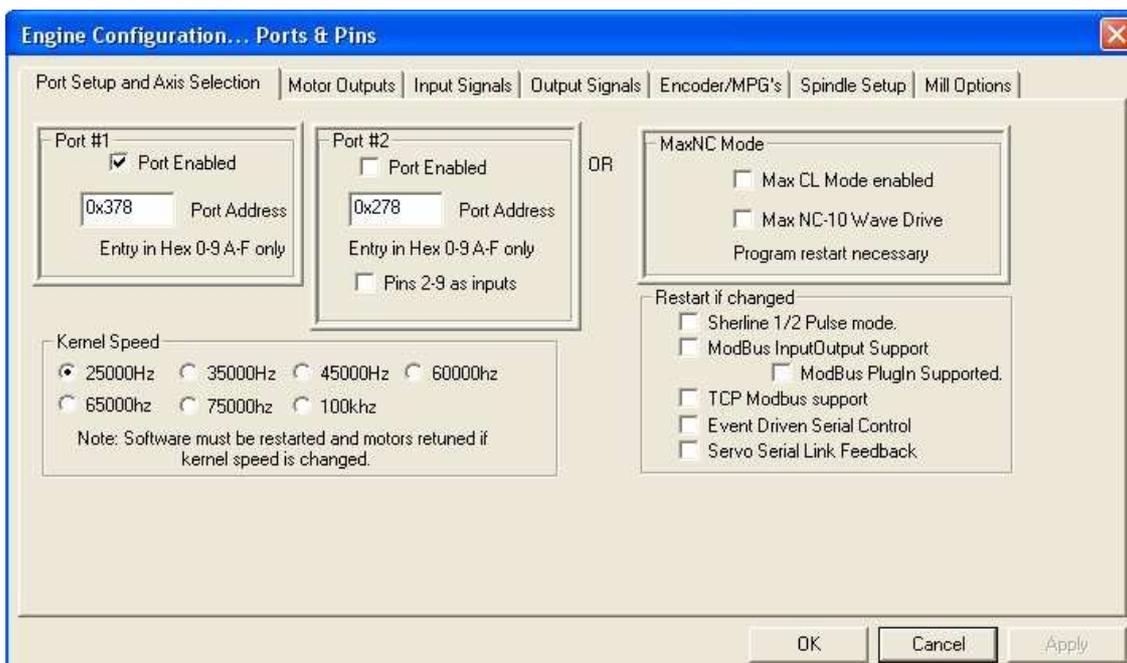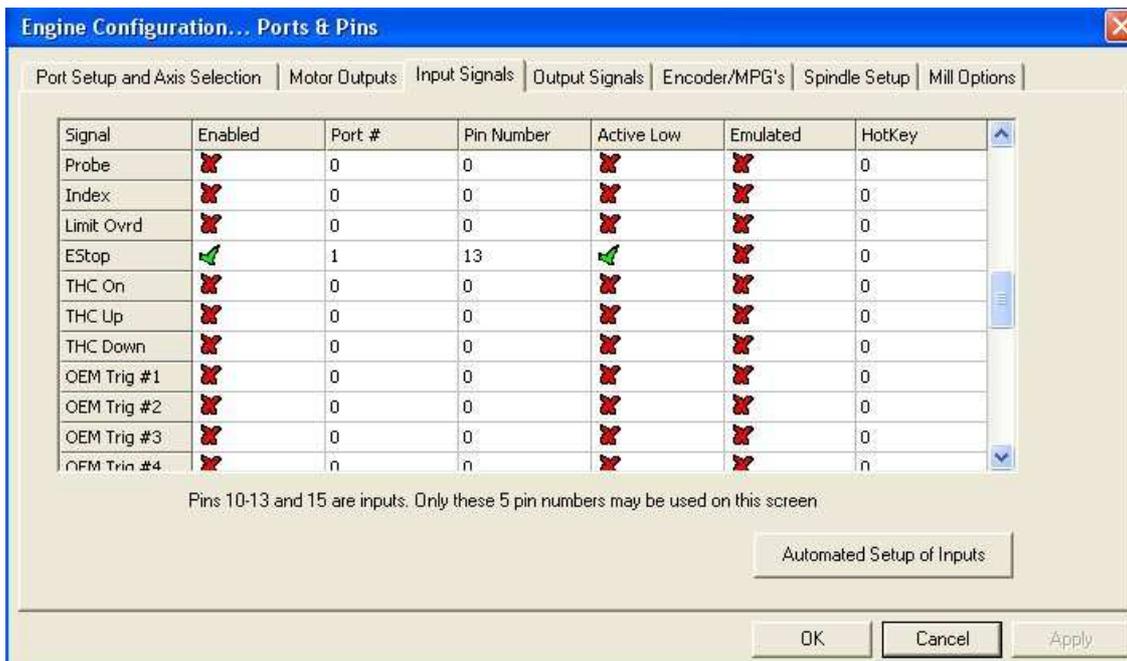
Click on the *motor outputs* tab
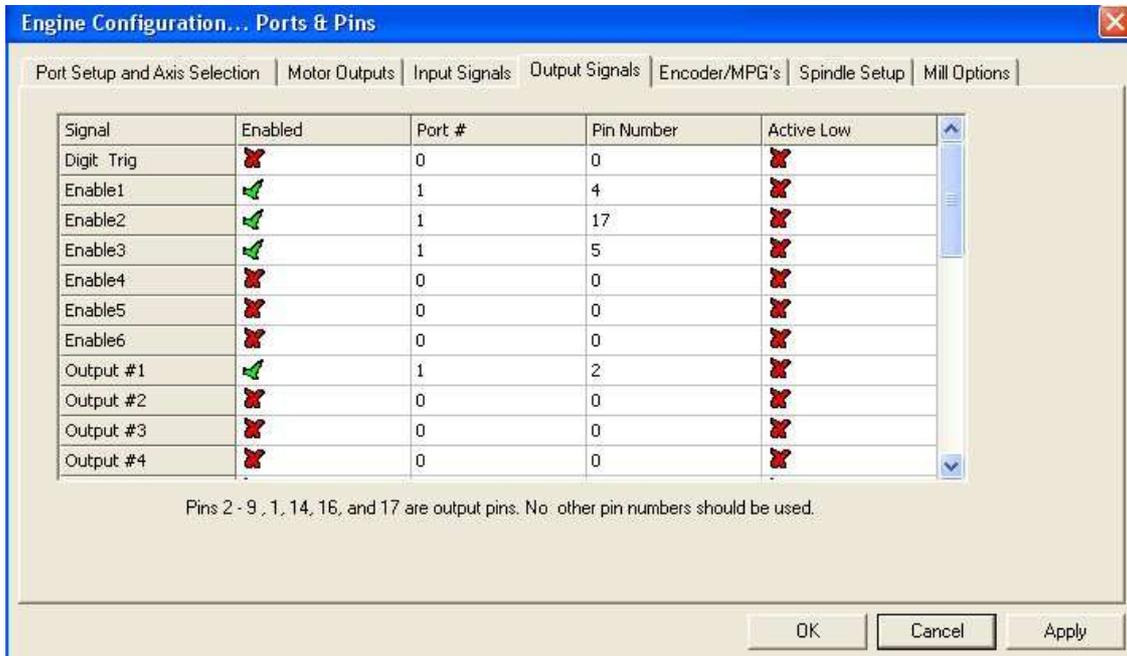
Make all boxes look like the ones shown below:

Click on the *input signal* tab

Scroll down to *EStop* and activate it just like this:

Click the *Output signal* tab and make it look like this:

You may raise the question why we enable only 3 motors, when we actually have 4 of them. Here you have to keep in mind, that we are using the printer port and on a printer we have 1 motor for printer head and one for paper feed. Plotters use a third motion to get the pencil up and down. The bottom line is that pin 5 enables the 4th motor as well. (for electronic experts, this is scary because you have to have all 4 motors connected, otherwise you have an open output and may fry the MOSFETs)

Click the tab mill option and activate the flood relay
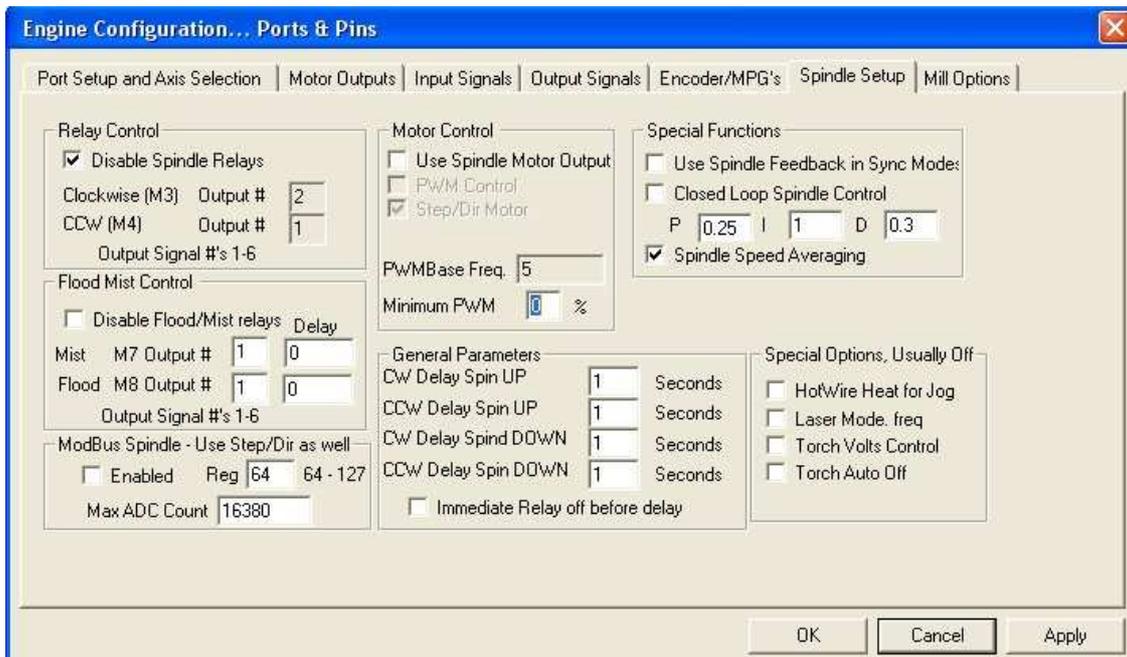
Now go back to configuration and go to *motor tuning*

Make the menu for all 4 motors look like this:
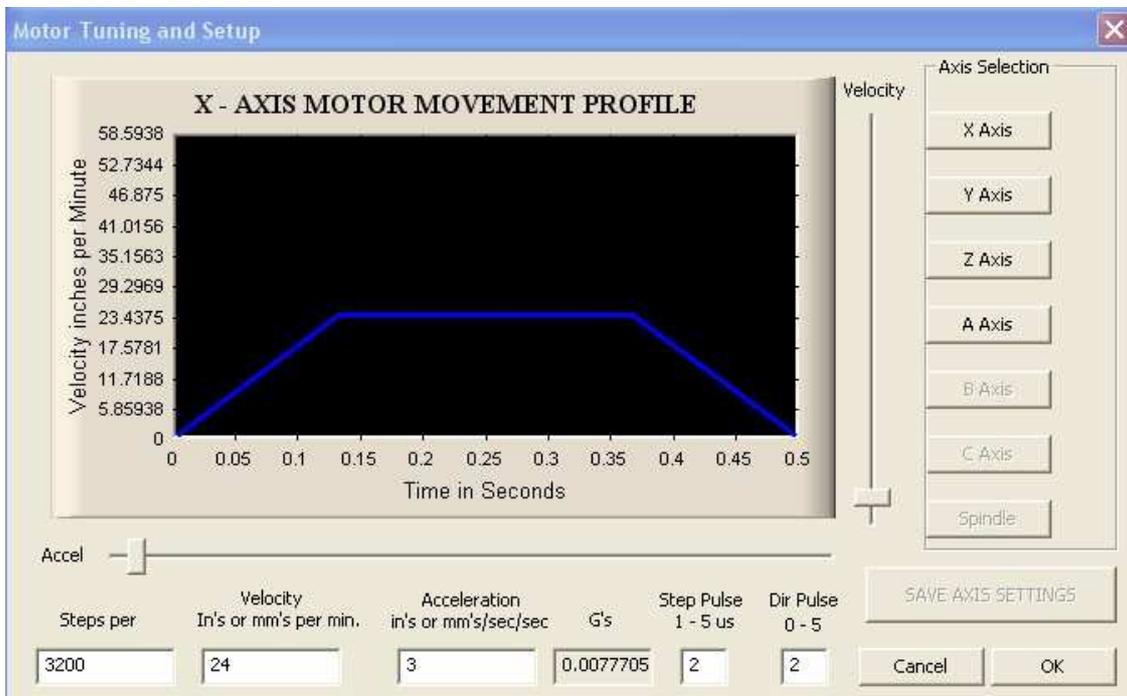
You have to click *save axis setting* for each axis, before you click on the next axis.

Note for the velocity I entered 24 in the box and the software made 0.0077 out of it (likely re-calculating G's)

(Hint for the expert users, we are using a 3/8 by 8 screw, which means we travel 1" with 8 revolutions and 400 steps per revolution makes 3200 steps per inch. 24" per minute computes to 2 rev per second, which is what these motors can do easily. Also, I'm using a very conservative ramp up and down. If you don't like it feel free to change it!)

Almost done. Go back to *configuration* and open the *slave axis* window.
Activate the A axis to be the slave of the Y axis:

(If you are setting up your machine as the hot wire cutter, don't use the slave option)
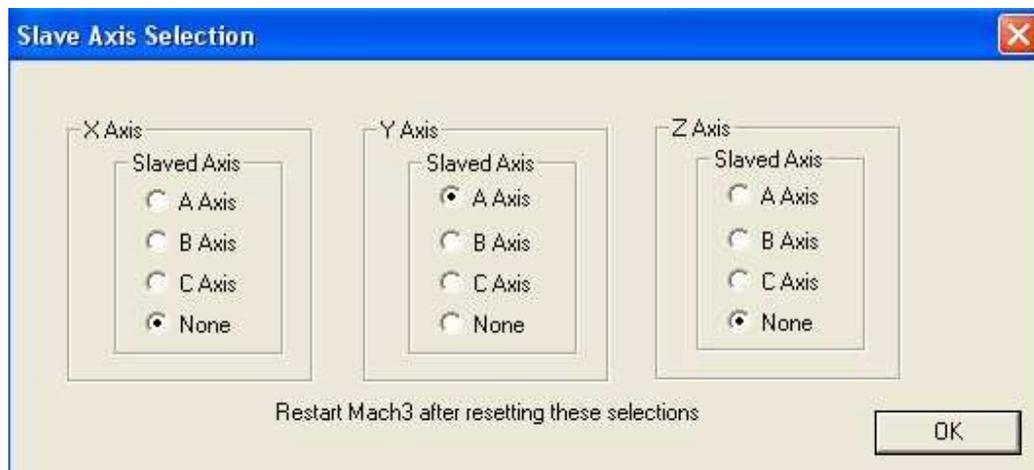
You will need to close and re-open the Mach 3 program in order to send this information to the controller.

Starting up the Equipment

Assuming everything is connected, the power is on to the controller board and the printer cable is connected to the PC, activate the reset button. You should hear a high pitch motor noise.

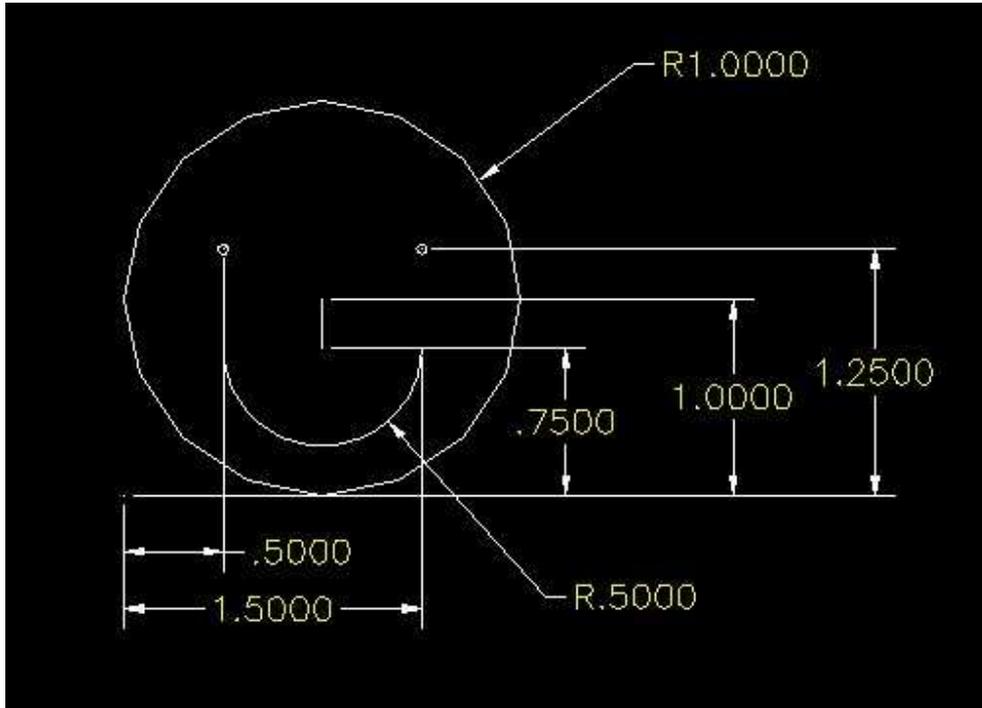Activate the tab key on your computer, which will give you manual control of the machine:
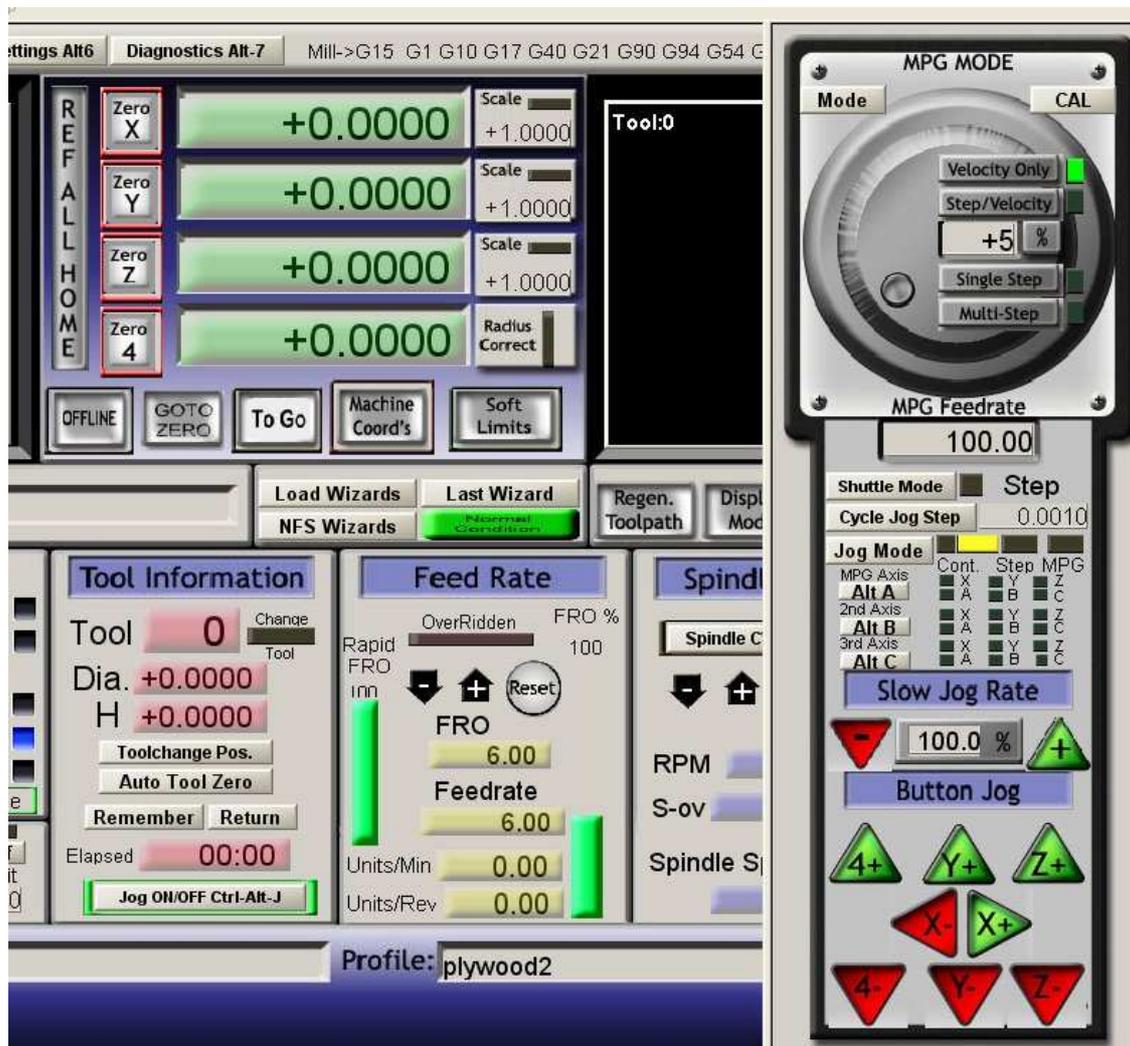
Press the X, Y and Z keys to your liking, congratulations; you are now in control of your own CNC machine.

G-Code Example

New to machine programming? Let's do a smiley together. The dimensioned smiley is shown below:


Click on the *edit G code* bar, and a notepad window should open. Enter or copy the following text into this window

```
N1 G0 X0.5000 Y1.2500
N2 G1 Z-0.0500
N3 G0 Z0.1000
N4 G0 X1.5000
N5 G1 Z-0.0500
N6 G0 Z0.1000
N7 G0 X1.0000 Y1.0000
N8 G1 Z-0.0500
N9 G1 Y0.7500
N10 G0 Z0.1000
N11 G0 X0.5000
N12 G1 Z-0.0500
N13 G3 X1.5000 R-0.5000
N14 G0 Z0.1000
N15 G0 X0.0000 Y1.0000
N16 G1 Z-0.0500
```
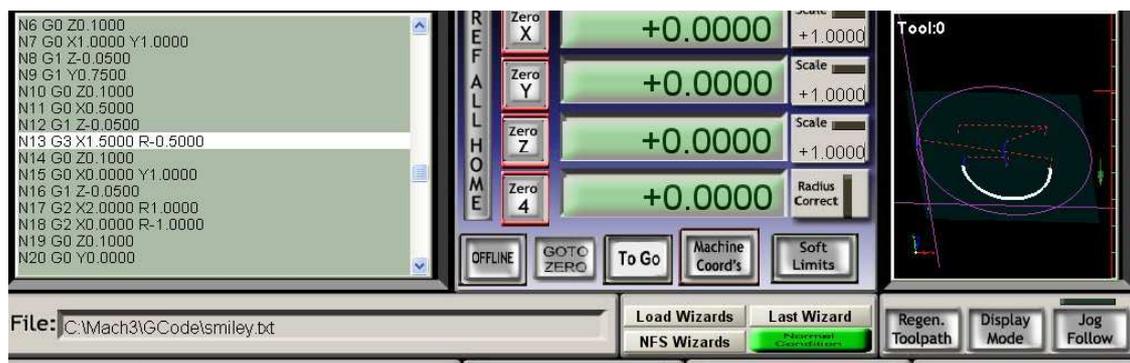
```
N17 G2 X2.0000 R1.0000
N18 G2 X0.0000 R-1.0000
N19 G0 Z0.1000
N20 G0 Y0.0000
```

Save and exit this file.

Now go back to the main window and click on the *load G-code* bar and load the saved file:
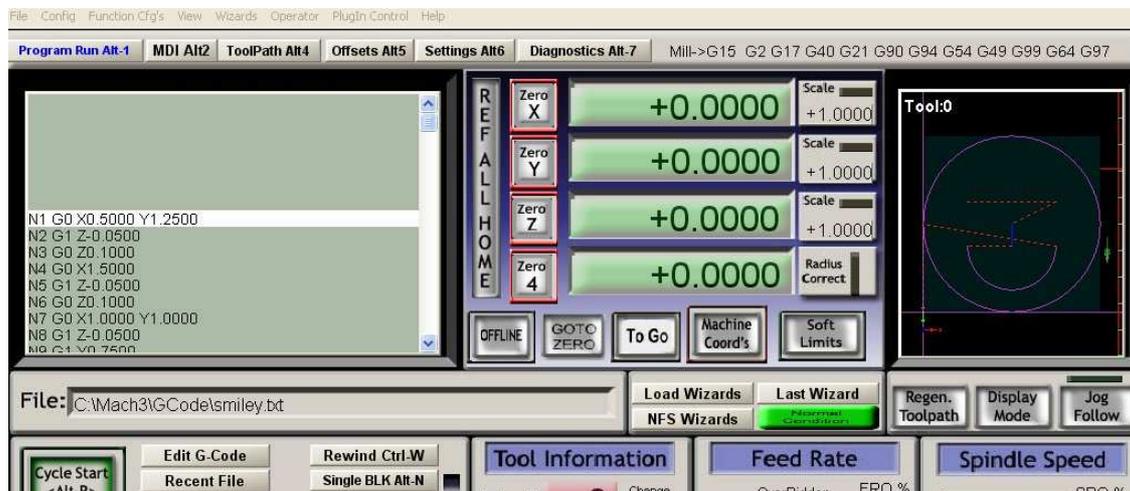
Use the right scroll bar in the G-code window to see what the machine is doing in the right display window.

Note: you can tilt the window by keeping the left mouse button pressed and moving the mouse forward.

As you can see in this example we are using only 4 different instructions:
  G0: go quickly to a position. Note positions are absolute, meaning you only have to enter the value which changed.

G1: mill something at milling speed, usually accompanied by an F value for feed speed in percent of max speed.

G2: mill clockwise. Enter the value where the circle ends and the radius of the circle.

G3: mill counter clockwise. Enter the value where the circle ends and the radius of the circle.

Note: the radius is also absolute value, meaning it is negative if it is pointing in negative direction of the start position.

Since we are not using a fixed zero position on the machine, we zero out each axis at the start position of the work piece, than we raise the Z-axis (+) by 0.1000 (doesn't have to be exact), before we press the start key.

When you are ready re-load the program, press the *reset* button and the cycle starts and watch what the machine is doing. You can also switch to *tool path*, which gives you a larger window of the action.