Mach 3 and the MaxNC 10 Open Loop / 2005 Mill

One of the many posts I see on the message boards are, "How do I set it up." Or " It's not working with MaxNC wave mode or MaxNC C.L. mode." Well I will try to explain as I good as I can. First, the original open loop mills used 4 pins from the printer port for each axis. The company MaxNC knew this as a Phase controller. MaxNC no longer uses this method, and it was dropped in the first quarter of 2004, but some how it is still carried by ArtSoft when using MaxNC wave mode with Mach 3 software.

Basically any of the MaxNC 10/15 Open Loop mills made after mid 2004 use the same board (printed circuit boards) that the Closed Loop mills do. The Open Loop differs from the Close Loop in that some times MaxNC does not install the connectors for the encoders and they're maybe (I have not verified this yet) a change in the PIC 16F84A 20mHz programming, As well as a few more pin changes for limit switches and the digitizing probe.

There are two PCB boards that I know of that use the newer setting for Mach 3, one is the MaxNC CL 10108A. This is the same board used on many a MaxNC 15 Closed Loop before 2005. Now I don't know exactly how long this board has been used but my guess is since the removal of the Phased controller board in the beginning of 2004 . The newer board (as of 2005) is the MaxNC CL 10108B, and is the current board that is in the MaxNC 10 Open Loop 2005. You can tell witch board you have by simply removing the cover form the controller box on the rear of the mill, look in the bottom left hand corner of the PCB. It will say ether MaxNC CL 10108A or MaxNC CL 10108B.

Now believe it or not, you can and should use the MaxNC CL mode when using Mach 3 with the newer Open Loop mills. The only problem is Mach 3 is made to cover a wide range of machines, so it's no wonder that many people get lost in all those windows.

So here is the setup, as I know it to be. And I use it myself with my MaxNC 10 Open Loop 2005. Number one, after install reboot. After any major change to the configuration of Mach 3, Reboot. Don't just start the software, close it, and reopen it. Follow Art's instruction to the letter on the installing video on his web site. Once you have that and have set your units of measurement (Metric or English). It's time to configure Mach 3 to use your MaxNC Open Loop mill. Lets look at the thing. First, from the main window, go to the top left hand side of the screen and click open the config drop down menu, with this menu open, click on Ports and Pins. A window should open that looks like the one on the next page. In this window, you should follow Art's video but if you're in doubt, this is what you settings should be unless you printer port is different, so check your LPT (printer port) address in hardware manager before you change it.
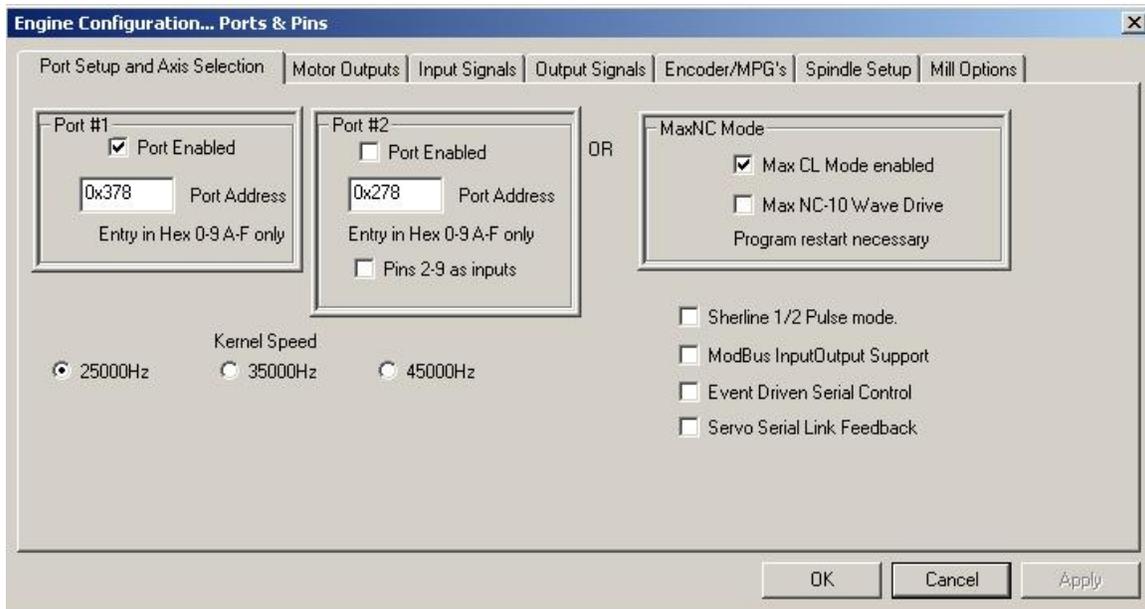
As you can see, my LPT (printer port) is set to default 0x378. Most computers will have this setting, but you better check first to be safe. Since I don't have a 2nd port, it will not be enabled. Art talks about using a second port for limit, home, and other switches. It might be worth investigating at a later date, but for now, will go without it.

And now the confusion sets in. remember I said that a long time ago (2 to 3 years ago) that MaxNC used Phase Controllers, if you have a older Open Loop made before 2004, then Max NC-10 Wave Drive is for you.

If you have a MaxNC Open Loop made around or after the first quarter of 2004, or you opened your controller box and found a circuit board with MAXNC CL 10108A or B, or later, Then Max CL Mode enabled is for you. Now some folks at this point will ask about the waveform used by MaxNC. Well in a nutshell, when MaxNC says Quadrature, it not so much of a waveform as it is how the system talks between the P.C, controller, and the stepper motors. As in, witch coil of wire is powered, and when it's turned on in the stepper motor. This is a little deep so I may get to that later as I dig into the controller at the circuit level. (One day if I have time.)

Art also talks about the Kernel Speed but I think it will be wise to start with 25000Hz for now. This is something you might want to read the Mach 3 manual for or at least see the online video at the ArtSoft – support page before mucking with this setting.

The next window of fear is the Motor Output window, as you can see I already have it populated with the correct setting for my computer. Yes it works vary well.

Engine Configuration... Ports & Pins

| Port Setup and Axis Selection | Motor Outputs | Input Signals | Output Signals | Encoder/MPG's | Spindle Setup | Mill Options |

Signal	Enabled	Step Pin#	Dir Pin#	Dir LowActive	Step Low Ac...	Step Port	Dir Port
X Axis	✔	6	7	✘	✘	1	1
Y Axis	✔	4	5	✘	✘	1	1
Z Axis	✔	8	9	✘	✘	1	1
A Axis	✘	0	0	✘	✘	0	0
B Axis	✘	0	0	✘	✘	0	0
C Axis	✘	0	0	✘	✘	0	0
Spindle	✔	1	0	✘	✔	1	0

[OK] [Cancel] [Apply]
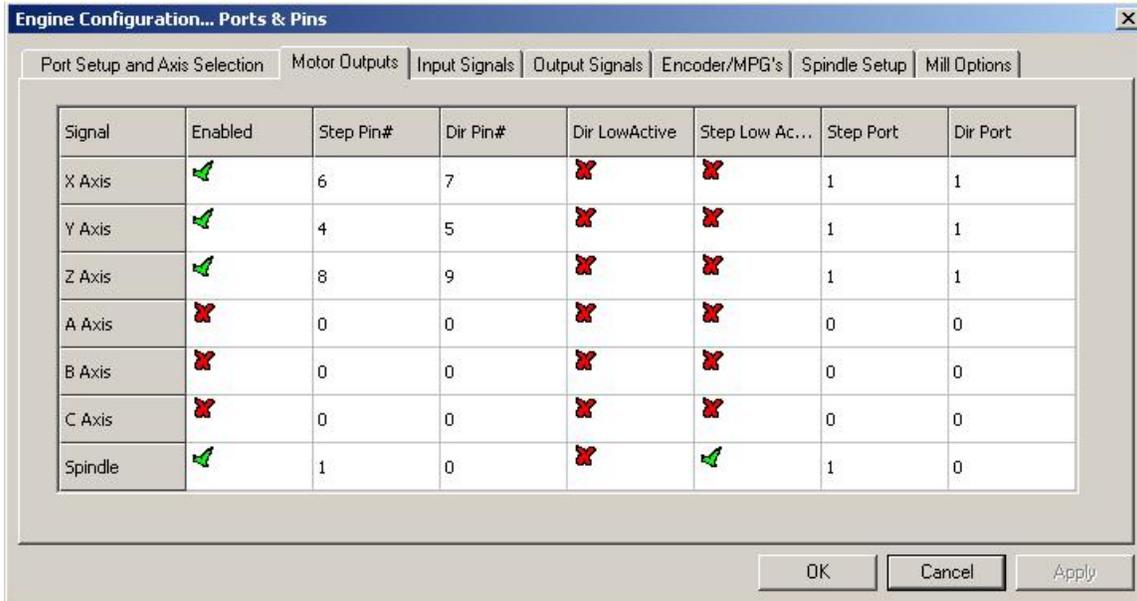
And this is a list of the LPT pins used in this window. And yes I know I'm using step/pin as well as the port pins. Hay it works, and I'll play with it later to see for my self-how the machine acts with changes to the assignments.

Partial pin out list (full list is in the back of the bus, ohh end of document.)

1.............. M3 OUTPUT (LOW=ON, PULSE FOR PWM)
2…………………………................ A AXIS (quadrature)
3…………………………............ A AXIS (quadrature)
4…………………………........... Y AXIS (quadrature)
5…………………………........... Y AXIS (quadrature)
6…………………………............X AXIS (quadrature)
7…………………………........... X AXIS (quadrature)
8...……………………….......... Z AXIS (quadrature)
9...……………………….......... Z AXIS (quadrature)
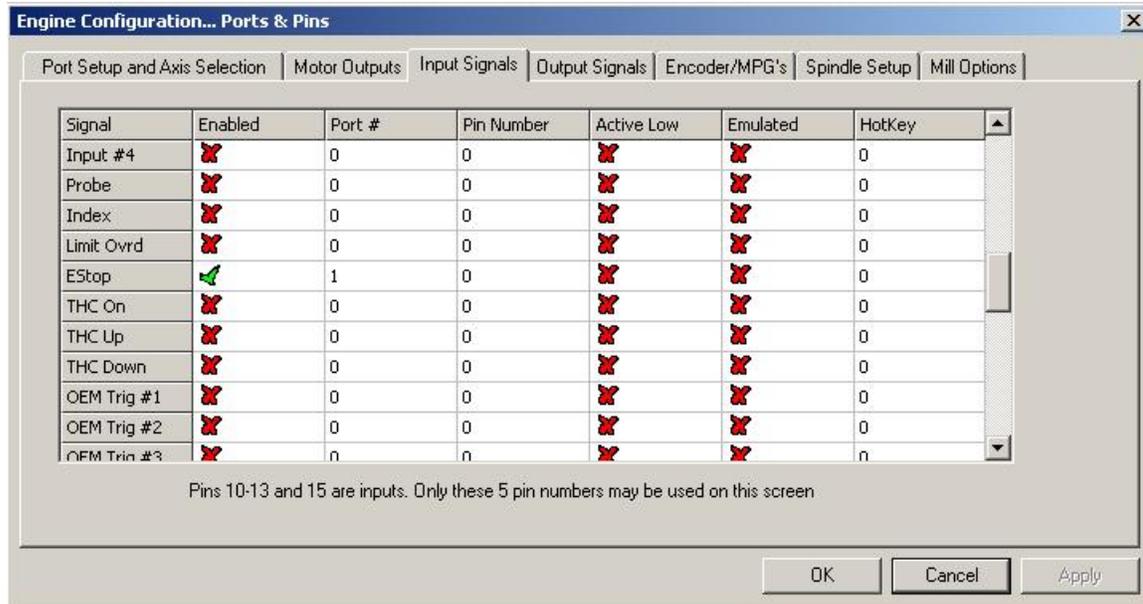
Now I don't much about Art's plan of attack with the EStop thing here but it works, you try it out and let me know what this is all about ok.

This is highly important my fellow MaxNC users, and it's the place where the biggest headaches come from. Look below and you will find some control signals that the controller must see for your MaxNC to work at all.
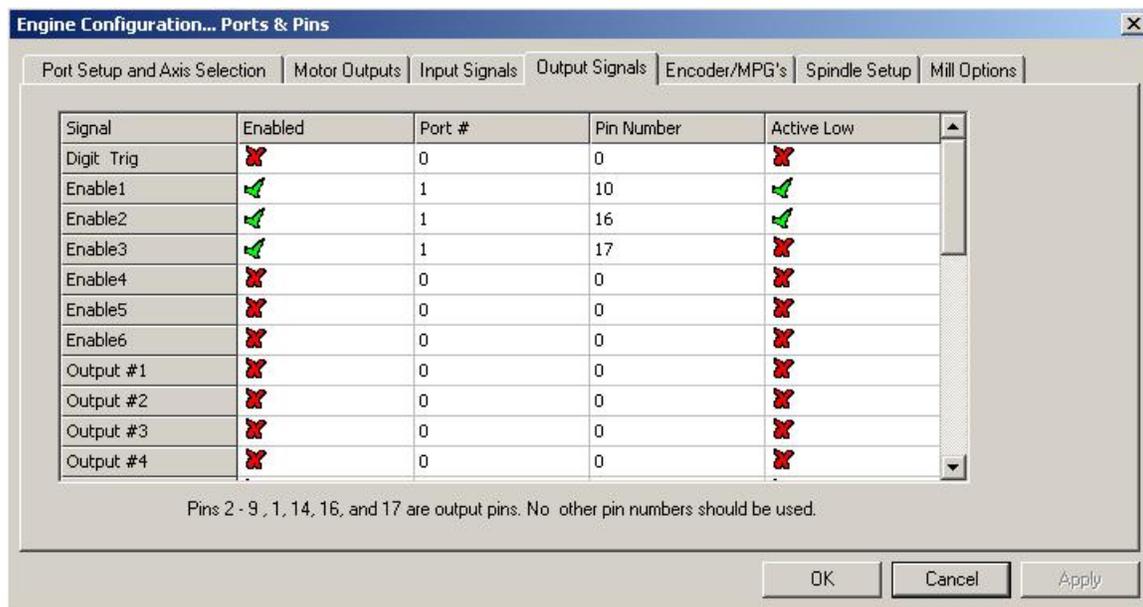
Pin 10. INPUT, MOVE ENABLE (LOW=NORMAL OPERATION, HIGH=STOP)

Pin 16. OUTPUTS, DISABLE POWER TO MOTORS (LOW = NORMAL, HIGH = DISABLE)

Pin 17. OUTPUT, RESET CONTROLLER (LOW=RESET, HIGH=NORMAL)

If just one of the control signals is missing or nor configured right, you will lose you mind tying to fix it. I know, I missed it a lot until I found that I always configured these wrong. As with all the pictures in this setup help document, I already filled out the information, you just got to copy it to the window as you see it here.
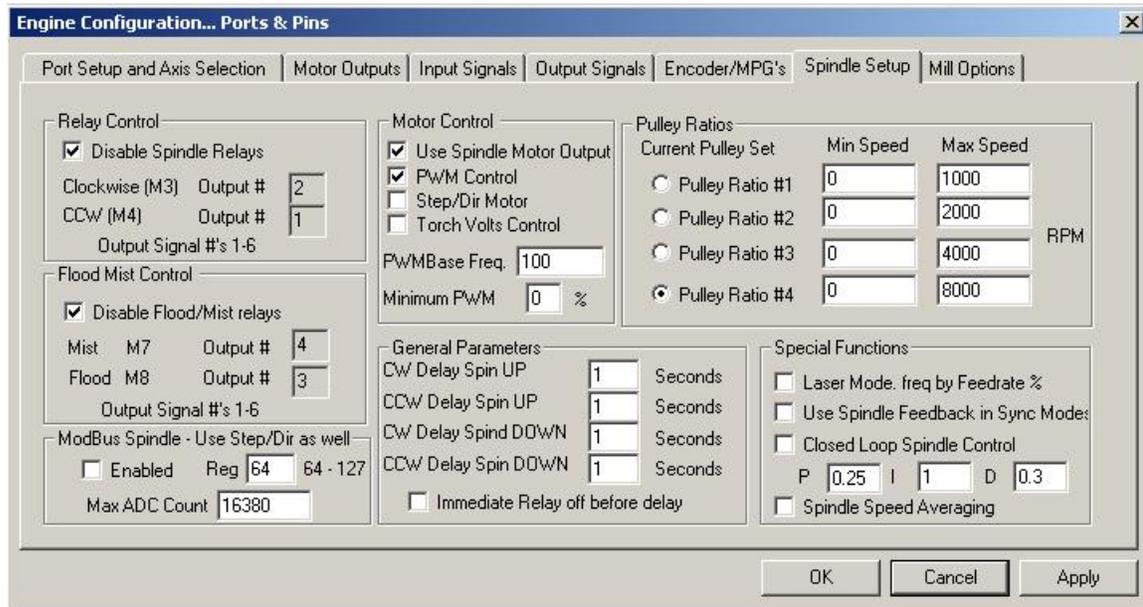
Ahh the Spindle Setup window, what you see is what you get – from now. Since I don not own a model helicopter rpm meter, I can't give you any better PWM information in the motor control box above. This will at least let your MaxNC O.L. work so you can dial it in better than I.

Yes, the MaxNC O.L. uses PWM or Pulse Width Modulation to control the speed of the Spindle motor. Art told me that you can also use step/dir to control the spindle speed, and he's right. I did it.
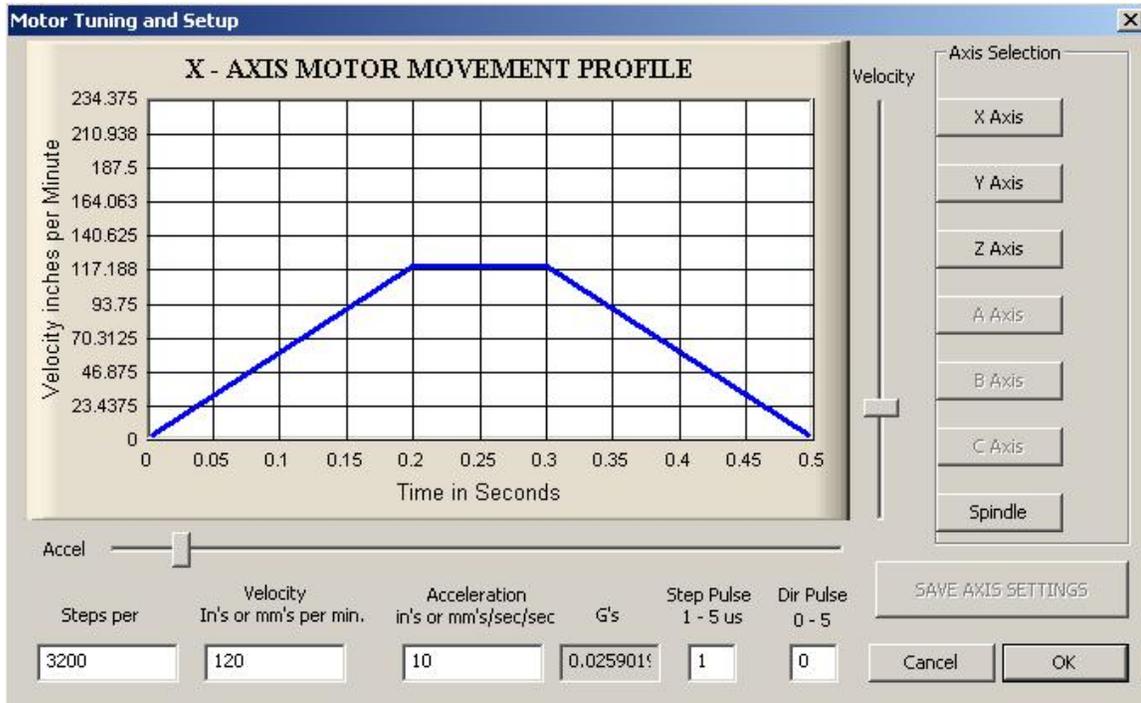
I'm not going to say this is perfect as the older MaxNC used Acme thread with 16 tpi (threads per inch) and some of the newer C.L. use Ball Screws. But if you have a Acme thread with 20 tpi then this should get you in the ballpark. Now I Know I need to fine-tune the velocity and the Acceleration to get it closer to Factory movement, but like I said, it will get you in the ballpark.
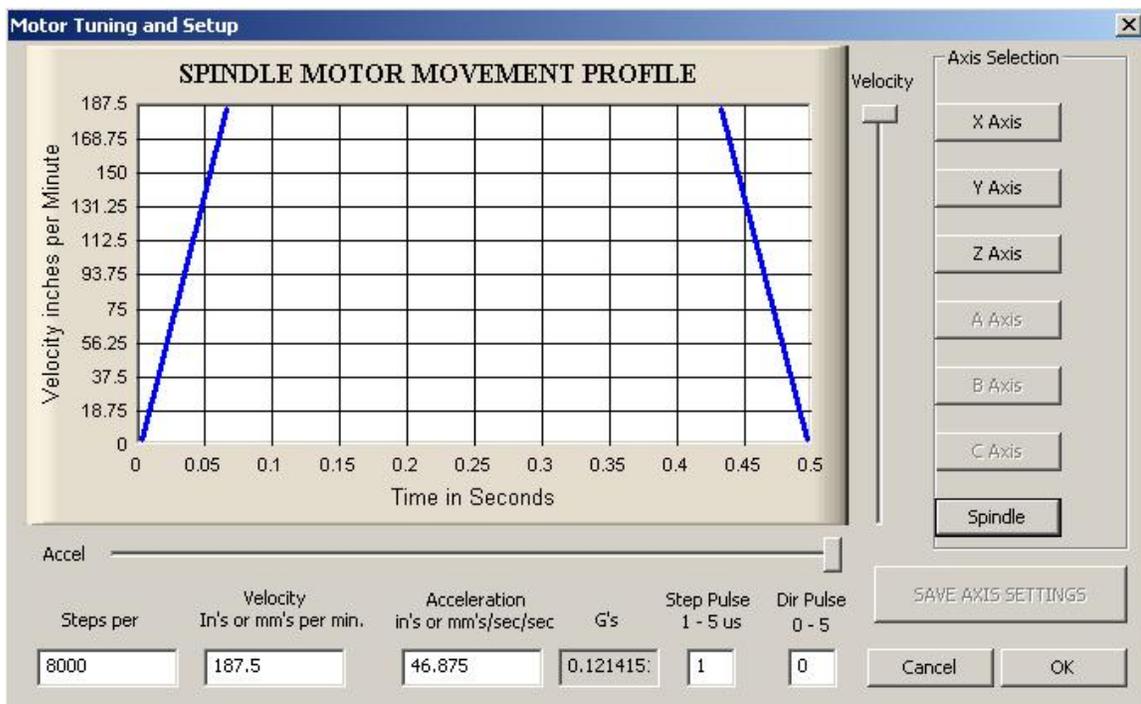
I know the spindle profile is a little messy, I'll be working on that so if you wanted to use step/dir control of the spindle, you could.

As you all can see, I'm collecting as much data on the MaxNC machines as I can. If you have some thing different for your setup, or maybe you have a newer (or older) MaxNC machine and want to share the information with others, please feel free to drop us a line, or e-mail, post, or by what ever means you can so that we all can share the wealth. It's good to help others.

I will try to update this info as I can,

Thank you.
Richard
KB7CGA