



SETTING YOUR SOFT LIMITS

FOR INDUSTRIAL CNC PRO SERIES MODELS



Important Note

This manual assumes that you have already followed the instructions in your Installation Manual, followed the Computer System Requirements and Set Up Guide, installed and configured your Mach3 Software, hooked your CNC Router to your power supply and computer, installed the included XML file, and configured your Mach3 Software per those instructions. If you have not, please do so before proceeding.

Job Zero vs Machine Coordinate Zero

As you may know, the term “Job Zero” refers to the starting point of a particular CNC operation. In most cases, your Job Zero will be set to the surface of the bottom left-hand corner of your material or workpiece.

“Machine Coordinate Zero” refers to an absolute zero position on the table itself, often outside of the bounds of your material or workpiece. This zero location can be utilized for more advanced techniques, like establishing reference points for tool changes, and is also a key factor in setting your Soft Limits.

Put simply, your Job Zero point is the predetermined starting point of a specific job, whereas your Machine Coordinate Zero is the absolute zero position for your entire work area.

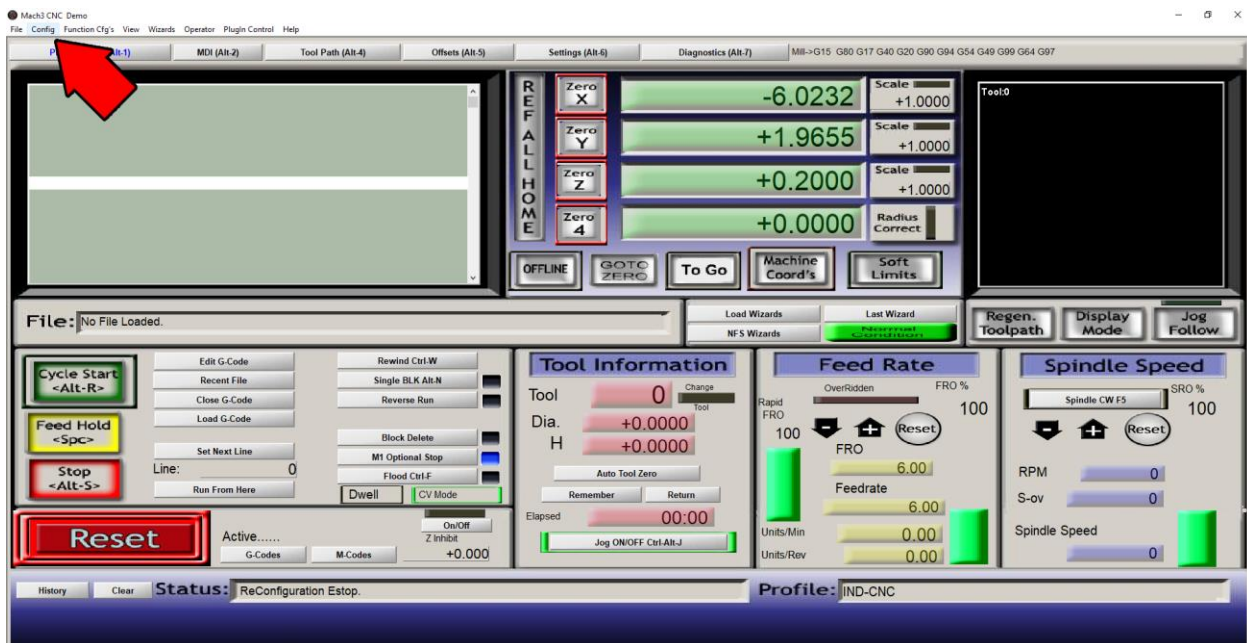
Getting Started

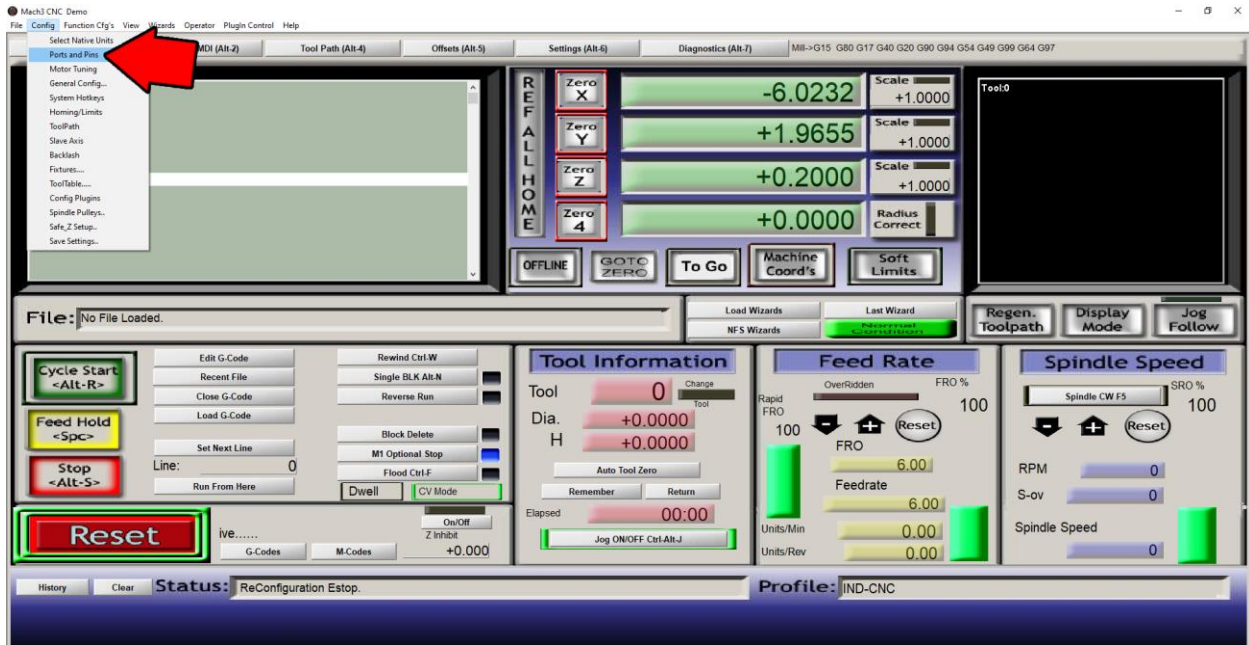
To start, make sure you have your computer connected securely to your router via your parallel cable or smooth stepper device. On your computer's desktop, open Mach3 via the "Mach3 Loader" button. Make sure you select "IND-CNC" as your profile.

Disabling Your Hard Limits

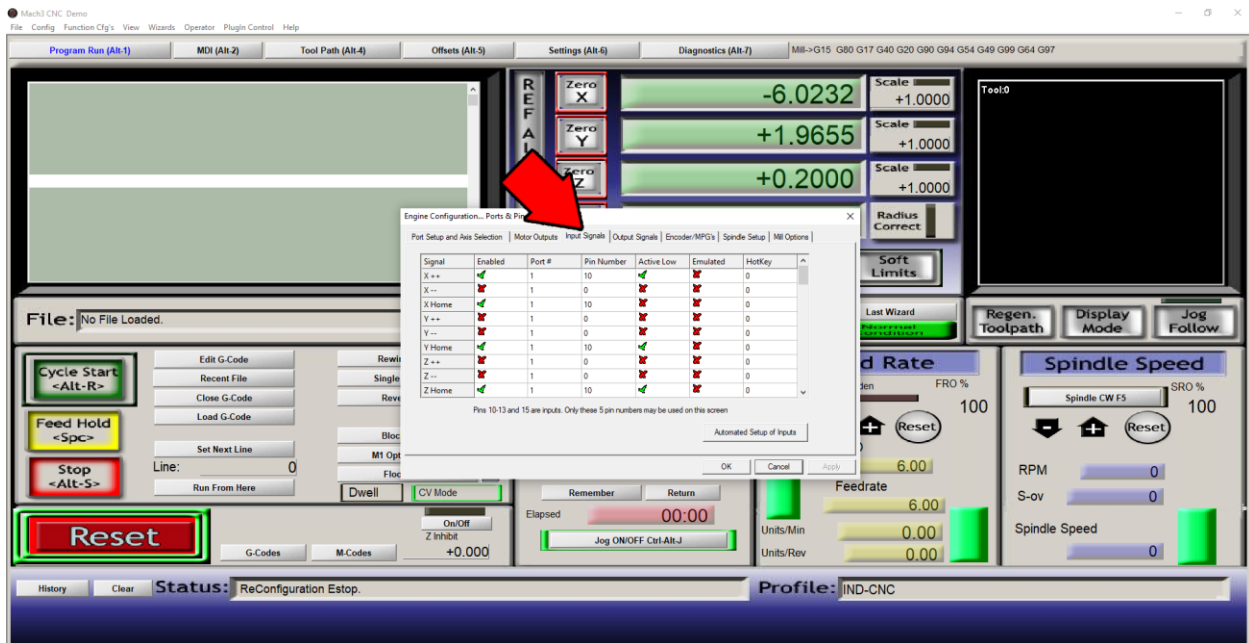
In order to run your machine with Soft Limits, you'll first need to disable your Hard Limits.

In the top left of the Mach3 window, click the "Config" button to open the Config drop-down menu, and select "Ports and Pins".





In the Ports and Pins window, click over to the "Input Signals" tab.



In the “Enabled” column, click each green check mark to change it into a red X. You can leave the “Estop” category as it is. Hit “Apply” and “OK”.

Engine Configuration... Ports & Pins

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

Signal	Enabled	Port #	Pin Number	Active Low	Emulated	HotKey
X ++	X	1	10	✓	X	0
X --	X	1	0	X	X	0
X Home	X	1	10	X	X	0
Y ++	X	1	0	X	X	0
Y --	X	1	0	X	X	0
Y Home	X	1	10	✓	X	0
Z ++	X	1	0	X	X	0
Z --	X	1	0	X	X	0
Z Home	X	1	10	✓	X	0

Pins 10-13 and 15 are inputs. Only these 5 pin numbers may be used on this screen

Automated Setup of Inputs

OK Cancel Apply

Setting Machine Coordinate Zero

Return On the “Program Run” screen, hit “Machine Coord’s to highlight it.

REF ALL HOME

Zero X	+0.3504	Scale +1.0000
Zero Y	-4.1574	Scale +1.0000
Zero Z	+0.0750	Scale +1.0000
Zero 4	+0.0000	Radius Correct

OFFLINE GOTO ZERO To Go Machine Coord's Soft Limits

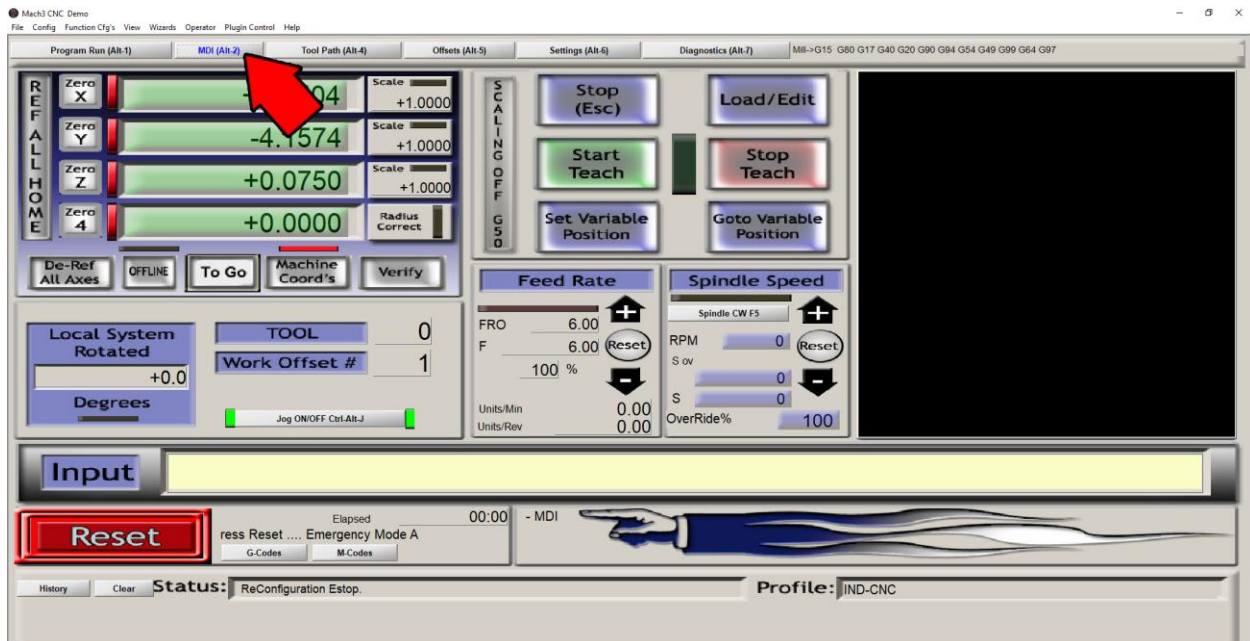
Jog to the very bottom left hand corner of your table, and then jog your spindle all the way to the top. Hit the “REF ALL HOME” button to set Machine Zero.

Hit the “Machine Coord’s” button again to deselect, and go back to standard view.

Now you can move around the table and set your Job Zero.

Running A G28 Command (Return to Machine Zero)

At the top of the Mach3 screen, click over to the MDI screen.

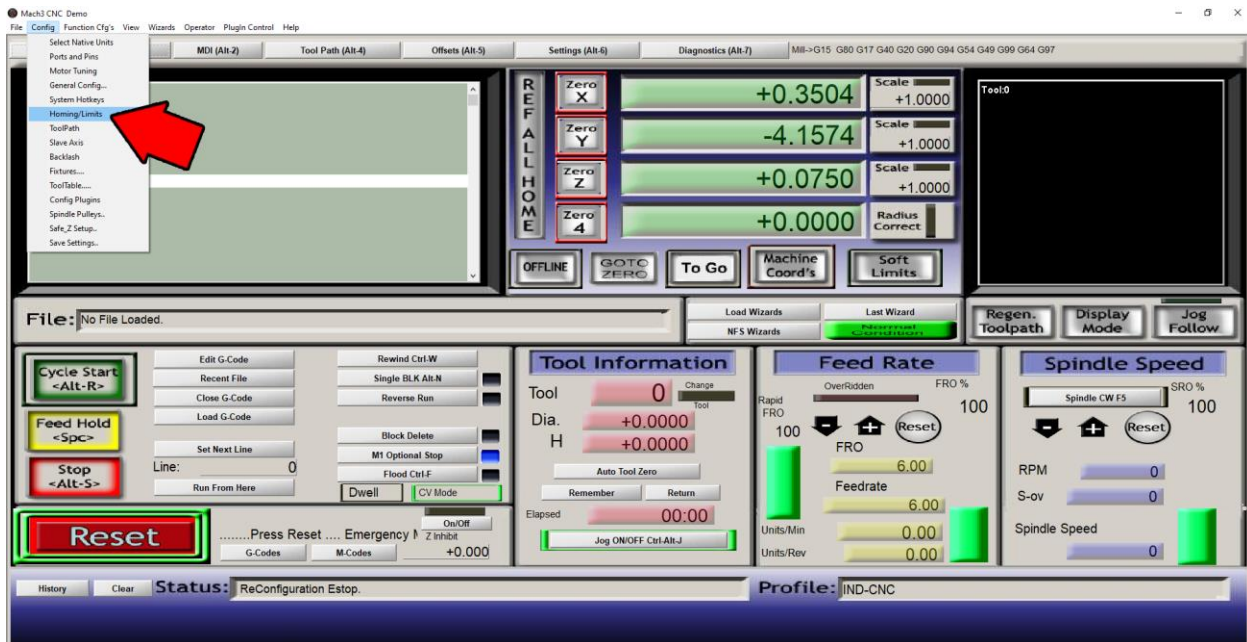


In the “Input” bar, type G28 and hit Enter on your keyboard. This will cause your machine to travel from its current position to the Machine Zero position in the bottom left-hand corner.



Setting Your Soft Limits

In the top left of the Mach3 window, click the “Config” button to open the Config drop-down menu, and select “Homing/Limits”.



In the “Soft Max” column, input the total length of your table’s cutting range for the Y and X axis, starting from your Machine Coordinate Zero point. For Z, input the total length in the “Soft Min” column as a negative number.

Axis	Reversed	Soft Max	Soft Min
X	X	48.00	0.00
Y	X	96.00	0.00
Z	X	0.00	-12.00

Apply settings and run.

Saving Your Settings

When shutting down for the day, you can run a G28 command (as explained in the [Running A G28 Command \(Return to Machine Zero\)](#) section). This will ensure that Mach3 will always start at Machine Coordinate Zero. You can confirm this at startup by clicking the “Machine Coord’s” button. When this button is highlighted, your X, Y, and Z should all read “0” in Machine Coordinate Zero. Then just click Machine Coord’s again to return to normal display.

When shutting down, you may be asked to “Fixture Save”. If so, click yes to save session settings.