

# ZEN CNC

## Laser Attachment Quick Start Guide

Included In The Box



# Mach3 Installation

USE THIS INSTRUCTION IF YOU HAVE OUR MACH3 CONTROLLER. FOR ARDUINO CONTROLLED MACHINES SCROLL DOWN TO THE ARDUINO INSTRUCTION.

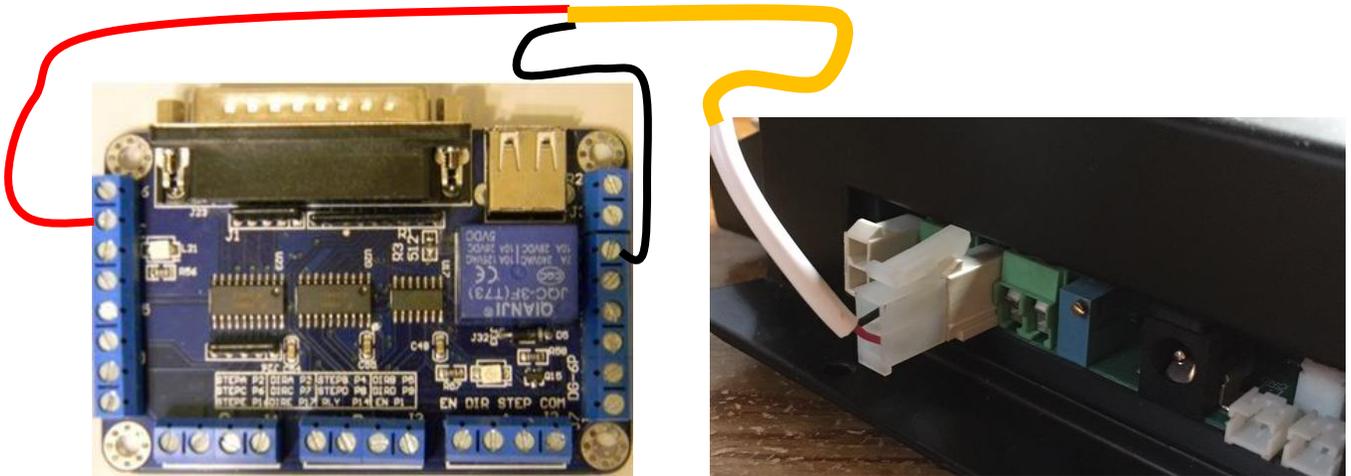
- 1.) Because PVC plastic releases toxic fumes when it burns, you must use some type of cover over the top of your Zen CNC Work Table. We typically recommend just a thin sheet of plywood or MDF. Before you turn on your laser, be sure this is in place
- 2.) Remove your spindle assembly from your Zen CNC via the 4 allen head bolts and nuts. Save the bolts and nuts as they will be used to mount the laser assembly.
- 3.) Using the provided spacers and your spindle mounting bolts, mount the laser assembly to the toolbase via the slotted mounting holes alongside the housing.
- 4.) When working with the laser at any point in the process is a very good idea to be wearing the Laser Safety Glasses. Laser Radiation can cause blindness, so this is a matter to take very seriously.
- 5.) Connect the provided **Power Wire Connector's** stripped ends to your 12V power supply used for your CNC electronics. Connect the other end of the harness via the round receptacle to the Laser Assembly.



- 6.) Install the **Safety Switch Key** carefully and turn to the ON position.



7.) Connect the **Signal Wire Connector** to your controller's signal wires (In This example we are using Pin 17). You will need to extend the harness as needed. Plug the other end (white connector) into the signal plug on the laser assembly.



8.) Be Sure the main switch on the left side of the laser is in the OFF "O" Position.



9.) Turn "On" or plug in your CNC's 12V Power Supply

10.) Click the red "reset" button on the left side of the laser assembly (beside the green connector). This reset button must be pressed every time the 12V power is turned on/off to the laser.



11.) Be sure the toggle switch is in TTL mode. TTL mode is used for controlling the Laser with your CNC and through Gcode. CW mode is for turning the laser on manually.



12.) With your glasses on, switch the main rocker switch to ON ("I"). Place a piece of scrap under the laser head and switch the laser to CW mode via the small toggle on the left side. The laser should illuminate. Go ahead and switch the toggle back to TTL to turn it off.



13.) You are now ready to calibrate your laser. Calibration instructions can be found in the Jtech user manual starting on page 15.

[http://jtechphotonics.com/wp-content/uploads/2013/05/Instruction-Manual-HCDBSAFDA-J-Tech-Photonics-Inc-V1\\_2.pdf](http://jtechphotonics.com/wp-content/uploads/2013/05/Instruction-Manual-HCDBSAFDA-J-Tech-Photonics-Inc-V1_2.pdf)

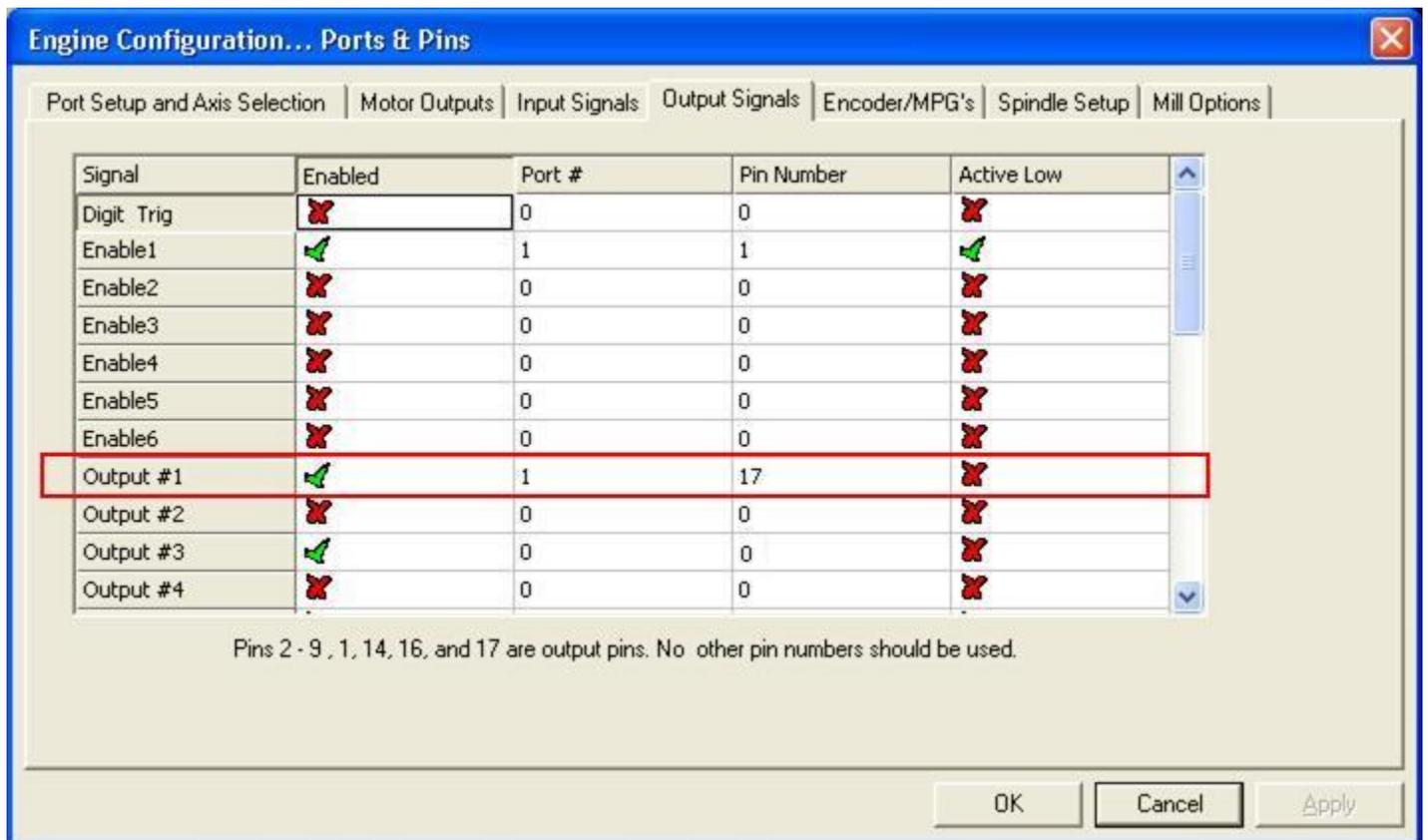
# Configuring Mach3

The following instructions are provided thanks to Jtech Photonics. You can find the original copy here: [http://jtechphotonics.com/?page\\_id=1436](http://jtechphotonics.com/?page_id=1436)

## Spindle as Laser

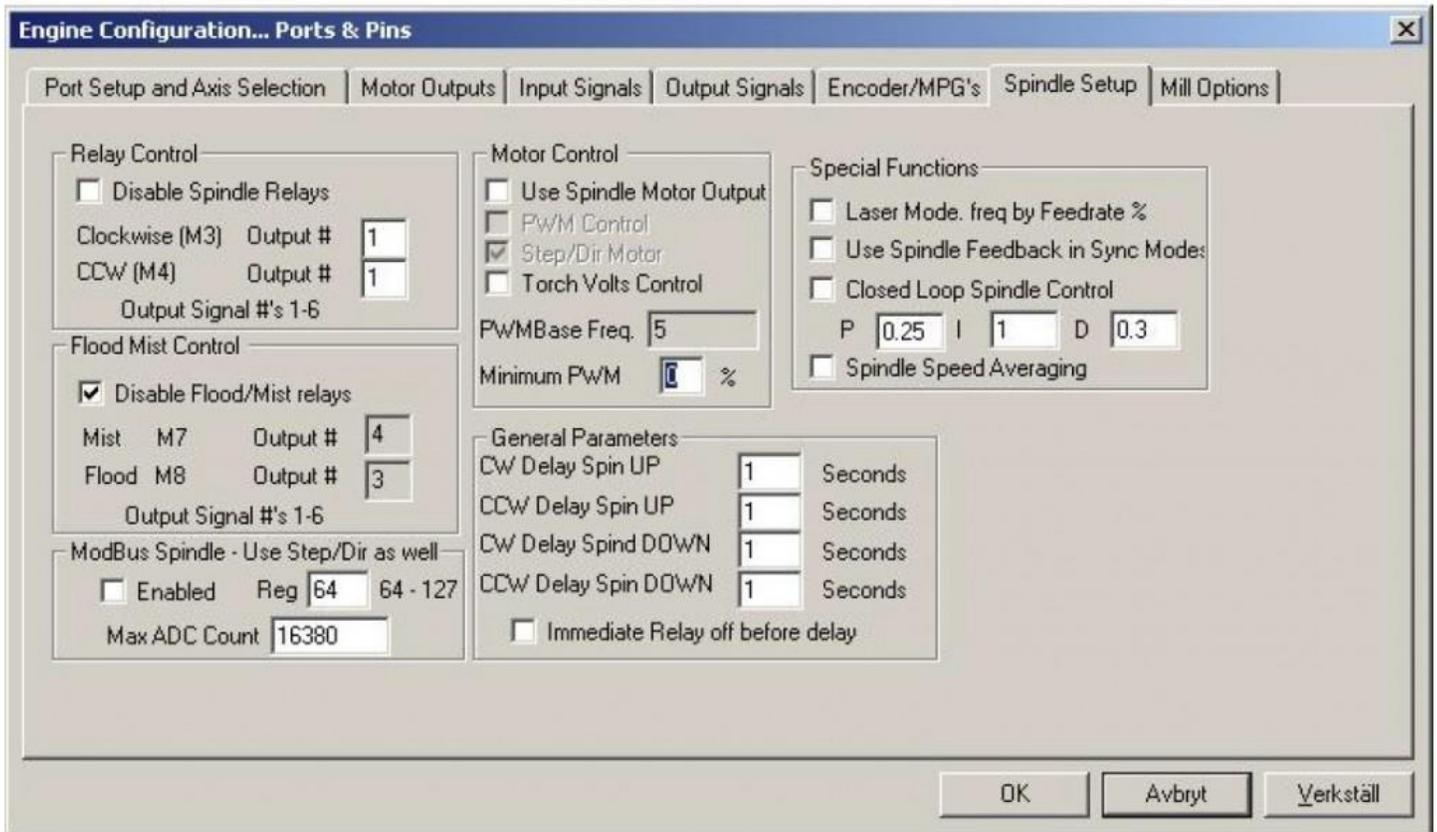
If you have chosen Spindle, you will have two options to choose from to control the laser. You can either just turn it on and off or you can turn it on with PWM and turn it off. We will first through the just simple “On” and “Off” first. To start, open up the “Ports and Pins” menu by clicking “Config” -> “Ports and Pins”.

To get the spindle to just turn on and off, ignore the spindle set-up in the “Motor Outputs” tab for now. Instead, we are going to configure it just as a standard output. The first thing to do is enable an output and map the pin on the parallel port to the software. Go to the “Output Signals” tab:



In the “Output Signals” tab, put a 1 in the “Port #” box and a 17 in the “Pin Number” box. This will enable the output (green checkmark next to it) and map it to Pin 17 of the DB25 connector.

Now go to the “Spindle Setup” tab:



In the top left corner, uncheck the “Disable Spindle Relays” box and put a 1 in both of the “Clockwise” and “CCW” boxes. Since we just mapped the output #1 to pin 17 in the last step, whenever a M3 (clockwise) or a M4 (CCW) command is sent in G Code, it will now enable output #1 (pin 17).

So, the M Code to turn on and off the laser now is:

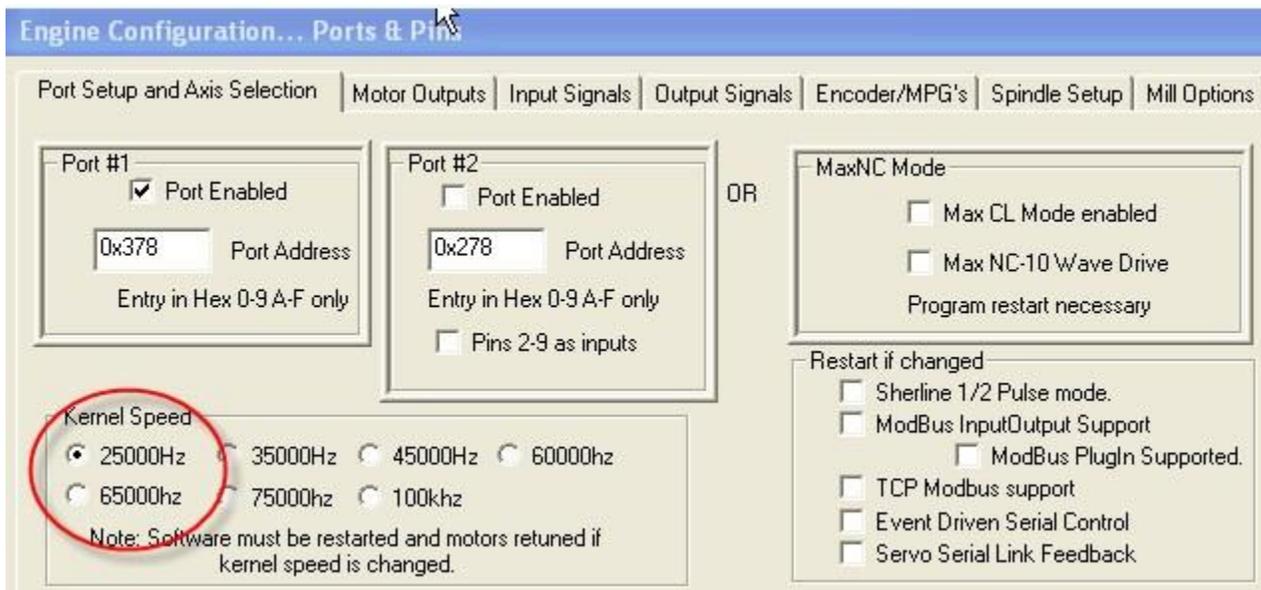
- Laser ON: Either “M3” or “M4”
- Laser OFF: “M5”

## Using Spindle PWM to Control Laser

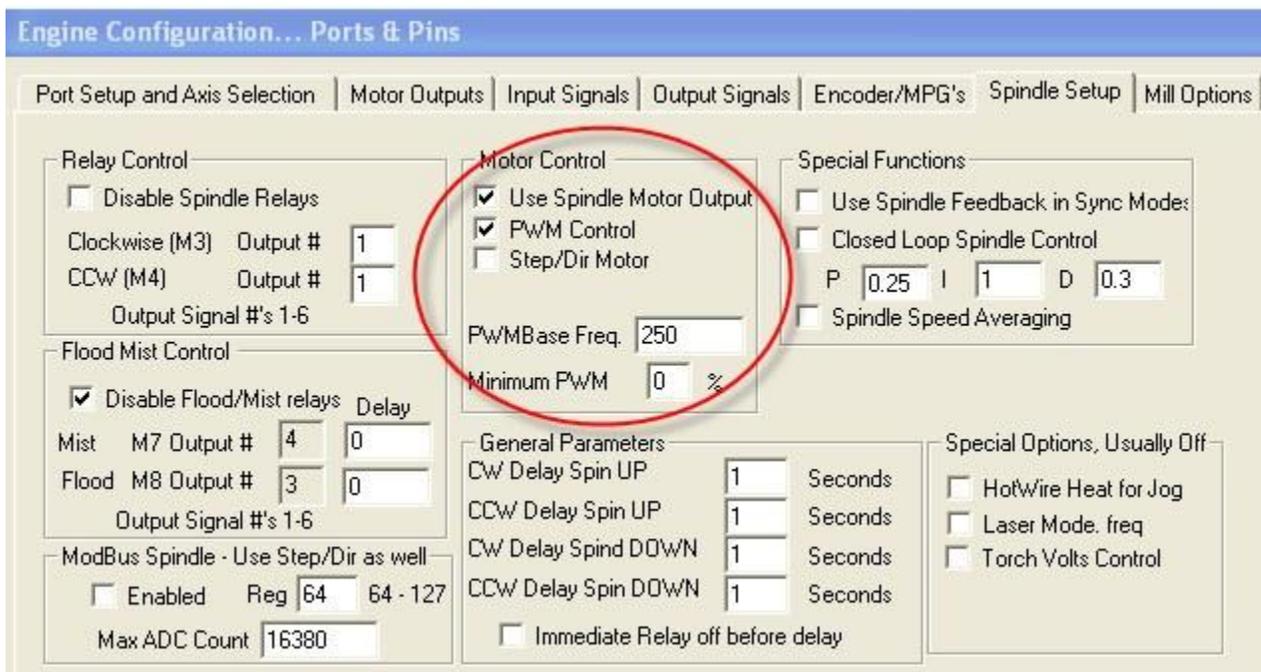
Now this is a bit more advanced to do, but in the end you should have power control via PWM of the laser as well and the general On and Off signals from before. I am really just repeating the instructions from [http://buildlog.net/cnc\\_laser/mach\\_laser\\_power.html](http://buildlog.net/cnc_laser/mach_laser_power.html) so I will give him credit and hopefully it is correct... Anyway, we continue on!

The theory is that you want a specific frequency to run the laser at and then you can adjust the duty cycle to increase or decrease the power of the laser. It would be nice to adjust the power of the laser in easy 1% increments, so we will set it up that way.

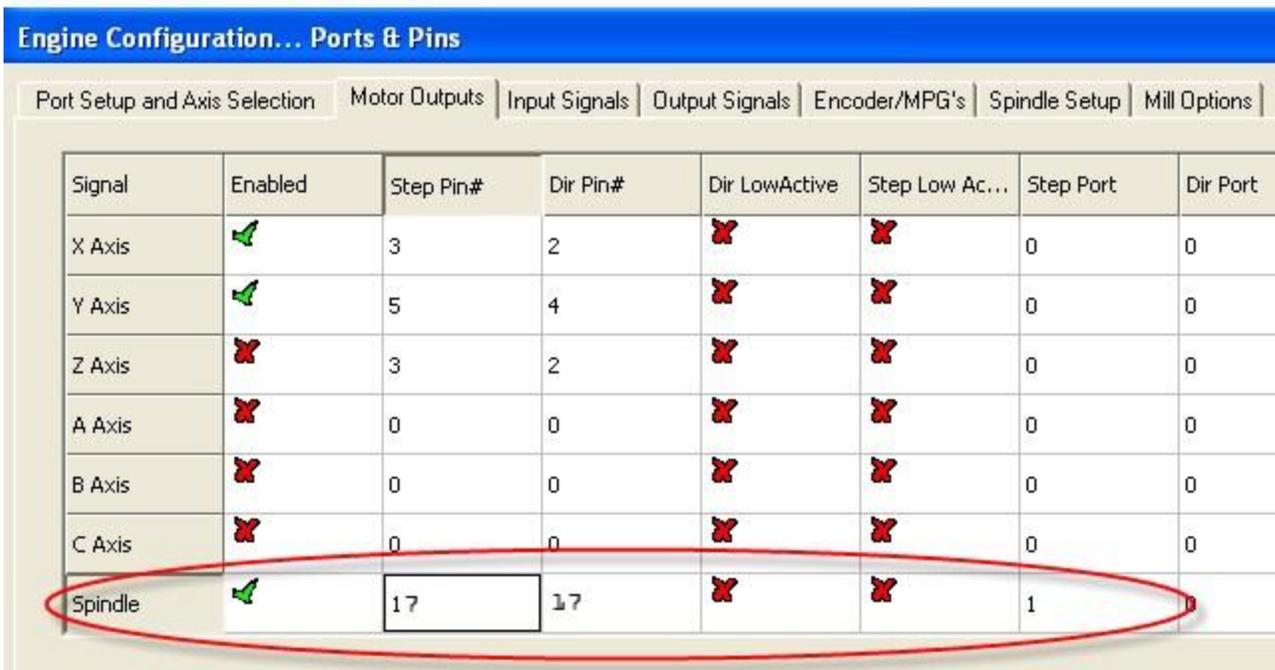
We are first going to change the kernel frequency and the Spindle frequency.



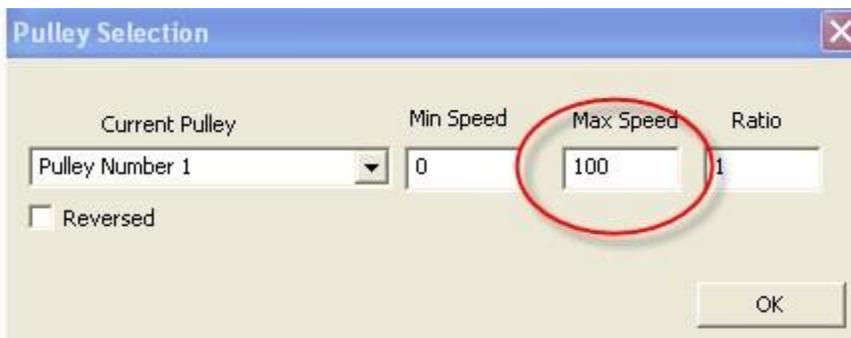
In the "Port Setup and Axis Selection" click the Kernel speed to 25000Hz.



In the "Spindle Setup" tab, check Use "Spindle Motor Output" and "PWM Control". Also set the base Freq. to "250" and the Minimum PWM to "0".



In the “Motor Outputs” tab, map the pins for the spindle to 17. Remember, 17 is for this specific controller using the TB6560 board, your set-up may require a different pin out. Just check in your instructions on which one it is.

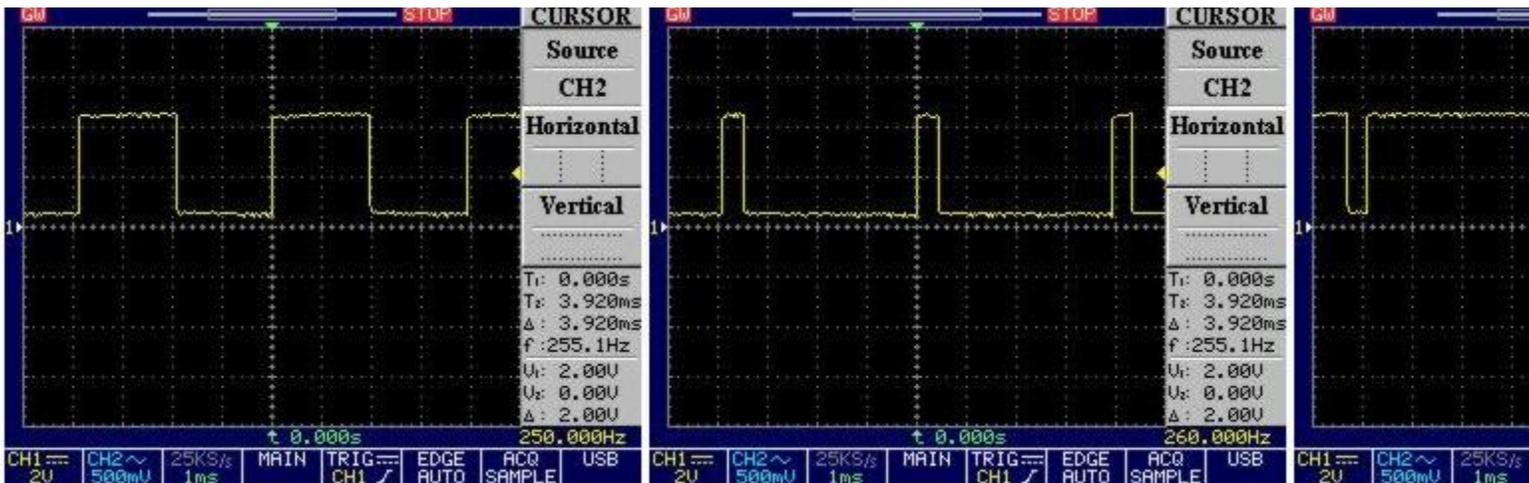


In the “Config” —> “Spindle Pullies” selection, enter “0” for Min Speed and “100” for Max Speed and a Ratio of “1”. This will then equate the spindle speed setting for laser power with a one to one ratio. Full power is 100 and off is 0 (or better to use M5).

To control this new PWM output in G Code, simply add an “SXXX” command to the normal spindle M Code of M3 or M4 where the “XXX” is the power of the laser. Some examples of the code are:

- M3 S100 – Laser ON at full power
- M3 S90 – Laser ON at 90% power
- M3 S50 – Laser ON at 50% power

The image below shows different duty cycles for 50%, 10% and 90% respectively.



## Solution to Possible Delay in Mach 3 using M3 and M5 Commands

Some customers are experiencing delays in the start of the laser due to the M3 and M5 commands being linked to axis move commands. A way to get around this problem is to use the M code to turn on an output instantaneously. These commands are:

- M11p# – Turn on the output the laser is on, where # is the number of the output.
- M10p# – Turn off the output the laser is on, where # is the number of the output.

So, if your output is set to 1 the command to turn on the laser is now M11p1. To turn it off it is M10p1. The post on the machsupport forum about this is

here: <http://www.machsupport.com/forum/index.php?action=printpage;topic=23636.0>

# Arduino Installation

USE THIS INSTRUCTION IF YOU HAVE OUR ARDUINO CONTROLLER (GRBL)

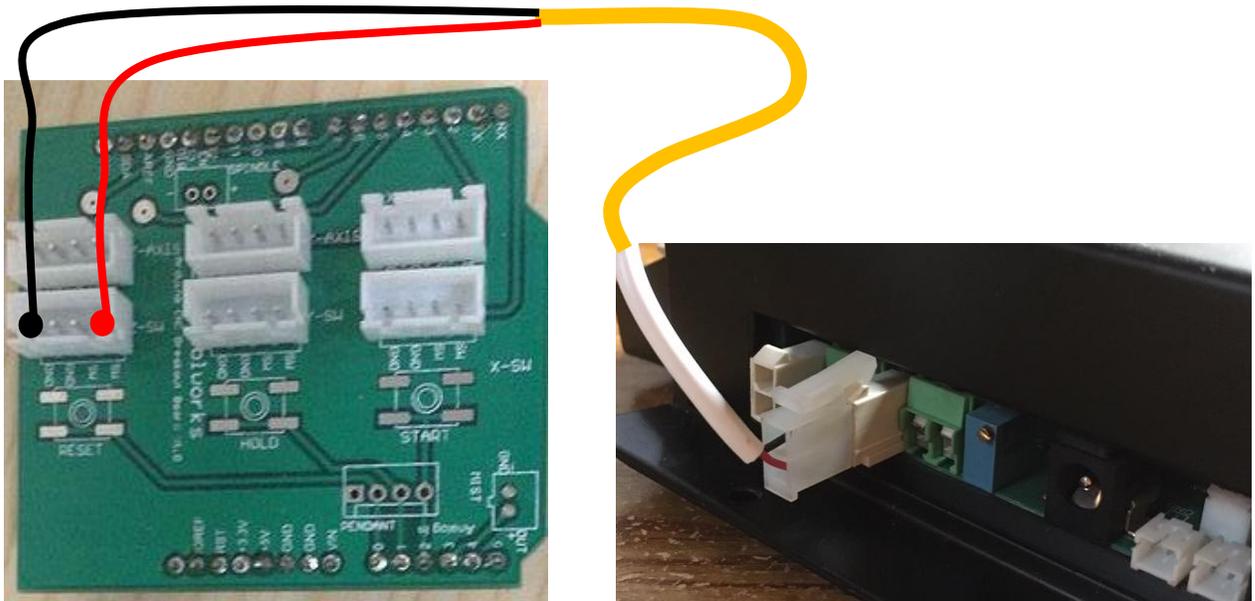
- 1.) Because PVC plastic releases toxic fumes when it burns, you must use some type of cover over the top of your Zen CNC Work Table. We typically recommend just a thin sheet of plywood or MDF. Before you turn on your laser, be sure this is in place
- 2.) Remove your spindle assembly from your Zen CNC via the 4 allen head bolts and nuts. Save the bolts and nuts as they will be used to mount the laser assembly.
- 3.) Using the provided spacers and your spindle mounting bolts, mount the laser assembly to the toolbase via the slotted mounting holes alongside the housing.
- 4.) When working with the laser at any point in the process is a very good idea to be wearing the Laser Safety Glasses. Laser Radiation can cause blindness, so this is a matter to take very seriously.
- 5.) Connect the provided **Power Wire Connector's** stripped ends to your 12V power supply used for your CNC electronics. Connect the other end of the harness via the round receptacle to the Laser Assembly.



- 6.) Install the **Safety Switch Key** carefully and turn to the ON position.



7.) Connect the **Signal Wire Connector** to your controller's signal wires. The Positive Signal Wire goes to the MS signal for the Z-Axis (originally the Z-Axis limit switch). The negative goes to the GND on the Z-Axis limit switch plug. You will need to extend the harness as needed. Plug the other end (white connector) into the signal plug on the laser assembly.



8.) Be Sure the main switch on the left side of the laser is in the OFF "O" Position.



9.) Turn "On" or plug in your CNC's 12V Power Supply

10.) Click the red "reset" button on the left side of the laser assembly (beside the green connector). This reset button must be pressed every time the 12V power is turned on/off to the laser.



- 11.) Be sure the toggle switch is in TTL mode. TTL mode is used for controlling the Laser with your CNC and through Gcode. CW mode is for turning the laser on manually.



- 12.) With your glasses on, switch the main rocker switch to ON ("I"). Place a piece of scrap under the laser head and switch the laser to CW mode via the small toggle on the left side. The laser should illuminate. Go ahead and switch the toggle back to TTL to turn it off. Flip the main rocker switch back to OFF "O"



- 13.) You are now ready to calibrate your laser. Calibration instructions can be found in the Jtech user manual starting on page 15.

[http://jtechphotonics.com/wp-content/uploads/2013/05/Instruction-Manual-HCDBSAFDA-J-Tech-Photonics-Inc-V1\\_2.pdf](http://jtechphotonics.com/wp-content/uploads/2013/05/Instruction-Manual-HCDBSAFDA-J-Tech-Photonics-Inc-V1_2.pdf)

# Configuring Your Arduino and GRBL

- 1.) We will need to upload a special version of GRBL to your Arduino in order to take advantage of the Laser's PWM intensity variation.
- 2.) The next steps are taken from Jtech Photonics website:  
[www.JtechPhotonics.com](http://www.JtechPhotonics.com)

## 1. Get PicLaser Software

Buy the software PicLaser in our shop to convert your .BMP file into G Code to run on your machine. You can get the [software here](#):



## 2. Get PicSender

You will need this sender program because it can handle the large files that are required for picture engraving. It also has a nifty way to save your parameters, which we will discuss next. You can get the program from Picengrave.com here: [PICSENDER PROGRAM](#)

## 3. Save Your GRBL Settings

When you re-flash your Arduino with the new firmware, you will have to re-enter your settings for your machine into the Arduino again. PicSender has a nice feature that allows you to save and load parameter files into your Arduino with one button. Click on the top right orange/yellow "GRBL" button to access this feature.

## 4. Put the J Tech Photonics 0.9g GRBL Firmware on your Arduino

This version of firmware was developed so you can do variable intensity photo engravings using the PicLaser software. The G Code generated from the PicLaser program will give a different power level for each pixel of an image. Note that this version will still control your router as well. Everything will still be the same.

To Download our version click here: [DOWNLOAD J TECH GRBL 0.9g](#)

You will need an uploader program to load the firmware on the Arduino. We use [X Loader and can find it HERE.](#)

When using Xloader be sure to set the board to Arduino Uno 328 and 115200 Baud.

When using our version of GRBL, the intensity value will be a number between 0 and 255. So the laser command will be:

- **Laser ON 100%: M03 S255**
- **Laser ON 50%: M03 S127**
- **Laser ON 10%: M03 S25**
- **Laser OFF: M03 S0 (or M05)**

## 5. Upload your old GRBL settings

Since you just re-flashed your arduino, you will need to put your parameters back on it. Hopefully you saved them from step 3!

## 6. Use PicLaser to generate your G Code file

Import your image into PicLaser and make sure you have the correct settings. Under the “change settings” tab you will have”

- FeedRate: The speed of the engraving
- Pixel Resolution: This is the spot size of the laser. The laser will be turning on for each pixel, so each pixel needs to be the spot size. For the 2.8W laser I use 0.008” or (0.2mm).
- Max Laser Value: 255
- Min Laser Value: 0
- Laser OFF command: M5 (or you can use M03 S0)
- Laser Control Command: S (you can put M03 S, but really the firmware is only looking for the S) Choose if you want it horizontal, vertical, or 45 degree, outlines, and where the file is going to be saved. Make sure you choose GRBL button for the format.

Save your settings. You can look at the original image size and the engraved image size. If you want the engraved image size different, don't change the pixel resolution but rather change the original image. If you want it larger then start with a larger image. Save your G Code and your are ready to load.

## 7. Load into PicSender and run

Set your 0,0 location and press send to run the file. Your machine should have a constant feedrate motion without stopping for every pixel. You will need to play around with the feedrate for different materials, but soon you will have the hang of it and making really cool pictures on wood and other materials.