

# Setting up the JTECH Laser System on MillRight CNC machines

version 1.03

## Safety:

You must understand that this laser is dangerous. It is meant for adult use only. It can blind you and others. The laser light should never be viewed from any distance without the red tinted safety glasses being worn. The blue light that this laser produces can damage the naked eye even if it doesn't feel like it hurts or is burning. This can create long term damage.

Only focus the laser while it is at a reduced power setting. The laser can burn your skin if the beam hits you. Exercise caution when turning the lens to focus it. Never turn the laser on when it is unmounted or it may burn unintended objects. Never allow children or pets to be within line of sight of the laser while it is operating. Ensure that you are in a well ventilated area. Research materials for suitability prior to attempting to burn them with a laser. Certain materials, such as some plastics, produce toxic fumes when burned. Finally, the laser could potentially start a fire if allowed to dwell in one place for too long while burning. You should **always** keep a fire extinguisher nearby.

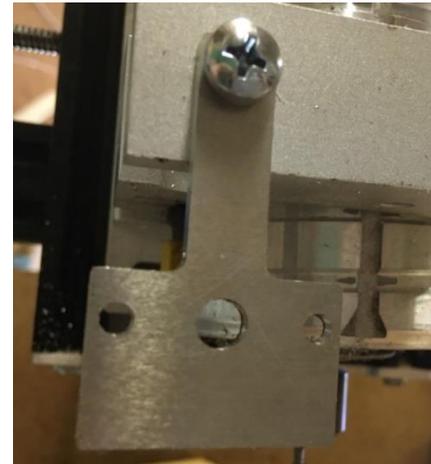
## Mounting:

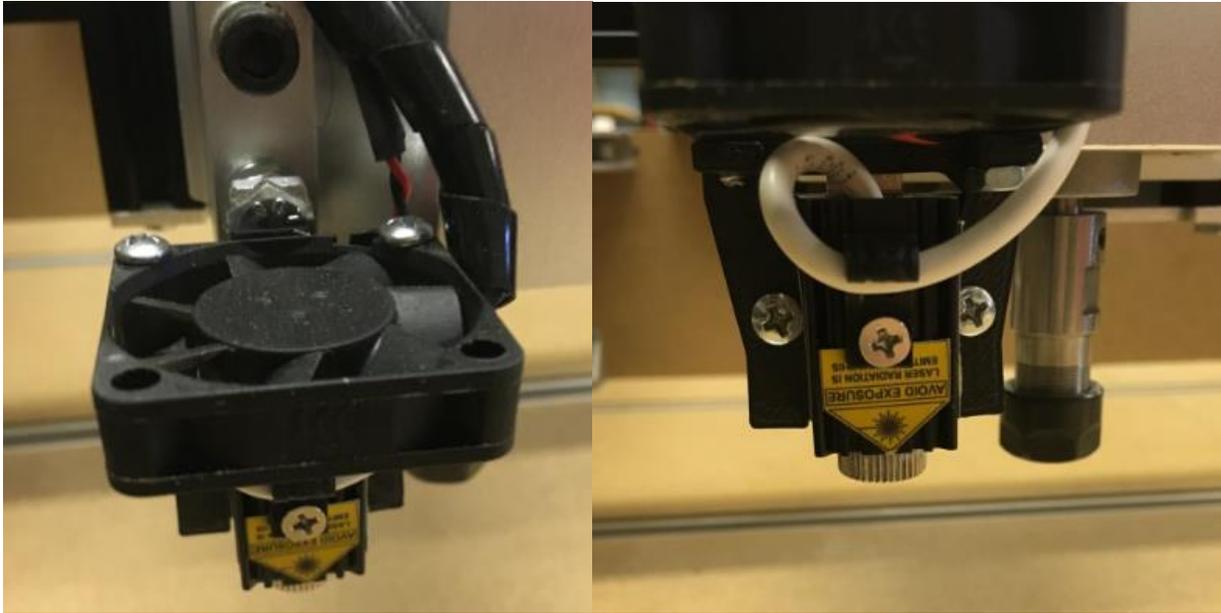
This laser kit comes with a mount to hold the laser. This mount will attach to your router mount or spindle mount that you purchased from MillRight CNC.

If you have our router mount, you will attach the laser mount to it as shown in the picture to the right using the provided M5 screw and split-lock washer. On the front face of the router mount, there are two holes. One of the two holes is for the screw that tightens the mount to the router. You will put this into the other hole. This hole is tapped for the M5 screw.

Those with the spindle will attach the mount to the bottom hole of the spindle mount. Be sure to install it on the side that does not clamp the spindle into the mount. Start by removing the existing bolt. Slide the aluminum laser mount onto the new bolt, then screw on one of the 1/4 hex nuts. This nut is being used to space the laser off from the spindle mount for proper clearance. Now slide a split-lock washer onto the bolt and insert the bolt into the hole in the spindle mount. Secure it with the other nut that was provided with your laser kit.

The laser diode is already enclosed in a black, finned aluminum heat sink. Mount the black plastic fan bracket and the black aluminum heat sink to the laser mount now using the provided hardware. After that, mount the fan onto the black plastic fan bracket. The label in the center of the fan should point downward. See the below pictures.





The cable lengths are appropriate for the Carve King and are okay for the M3 as well. If you have the Carve King, route the fan wires and the laser wires through the drag chains. The laser comes with a fairly large connector on the end that is difficult to push through the drag chain without creating unseen damage to inside of the connector. Basically, trying to push the connector through the drag chain may pull the wires out of the connector slightly and you won't get a signal to the laser. It's recommended to cut the connector off the end of the laser wire. There's a screw terminal on the laser wire that we will use with the expose wire. Do **not** cut the connector off the end of the fan wires.

### Connecting the laser:

See the picture on the next page that labels the connectors that you'll plug into. If you cut off the connector at the end of the laser wire to fit it through the drag chain, strip the end of the wire to expose some copper. Plug the red wire into the + H1 screw terminal. Plug the black wire into the - H1 terminal and screw them down. **DO NOT PUT THESE WIRES IN BACKWARDS! STUDY THE PHOTO CAREFULLY PLEASE TO AVOID DESTROYING THE LASER DIODE.** Plug the fan motor into the port labelled "laser fan".

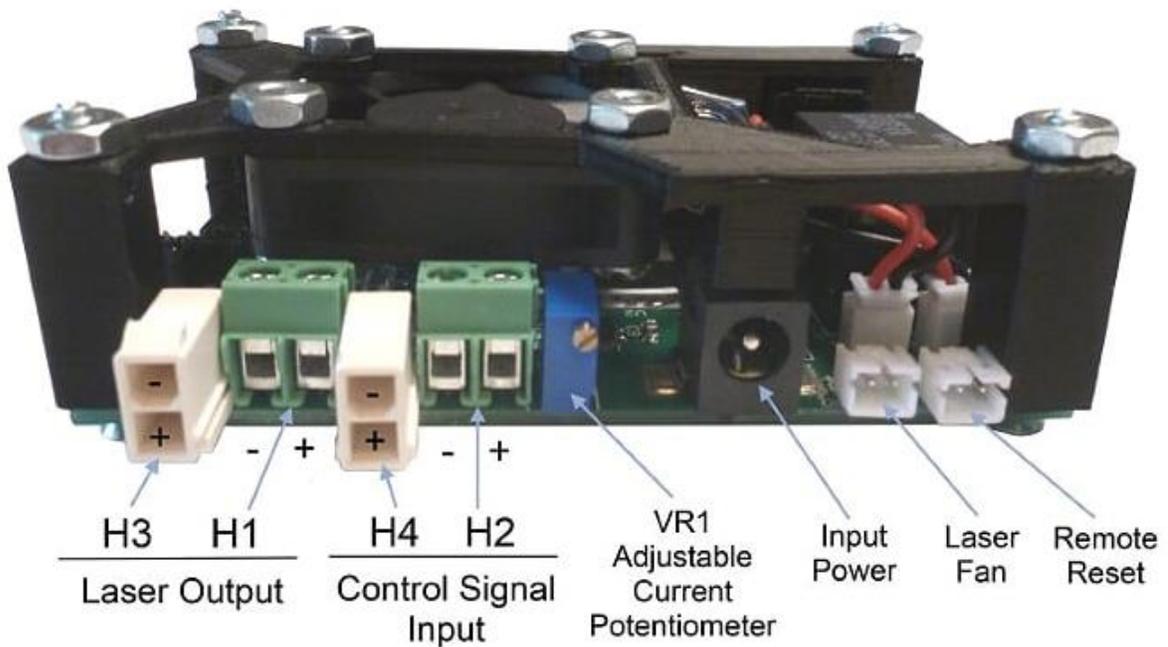
Now we'll connect the laser control signal. Get the wire in your laser kit that has a 2 point Dupont connector on one end, as shown at the bottom right of this page. This is your control signal wire. It either has an open wire on the end other end, or a connector that fits into terminal H4 shown below. If it has a connector for H4, plug it in there. If it has an open end, strip the end of the wire and plug the red wire into the + of screw terminal H2 and black wire of screw terminal H2.

Please do not get this backwards. The connector shown to the right will plug into the red stepper driver board at "Z+". This is on the right side of this board in the section labelled "END STOPS". This seems odd, but it is correct. The red wire should be to the left and the black wire should be to



the right. This is normally where the Z homing switch would plug in, but the firmware (Grbl, the control program that runs on your Uno board) that you are going to use to operate the laser has reassigned some pins so it can take advantage of special pulsing features only available on this pin. See the highlighted diagram to the right that indicates where you should plug this in. Those with homing switches will move their Z switch to the pins just below this section of the board labelled "Sp En." If you purchased your machine kit at the same time you purchased this laser, the firmware on your blue Uno control board is already set up to run this laser. If you bought this kit as an add on for your machine, see the next section for instructions to reconfigure your firmware to run the laser.

	Z+	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
END STOPS	Z-	<input type="checkbox"/>	<input type="checkbox"/>
	Y+	<input type="checkbox"/>	<input type="checkbox"/>
	Y-	<input type="checkbox"/>	<input type="checkbox"/>
	X+	<input type="checkbox"/>	<input type="checkbox"/>
	X-	<input type="checkbox"/>	<input type="checkbox"/>



**Software setup:**

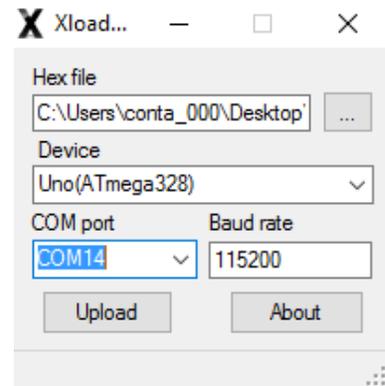
As a reminder, if you bought this laser with your machine kit you do not need to reload any firmware. It was shipped to you with the appropriate firmware. If you have the Carve King, get the new firmware by clicking this link:

<https://sabercathost.com/9auD/Carve King Homing and No Homing Laser Spindle.hex>

You must load this .hex file into your control Uno board using XLoader, which you can download here:

<http://russemotto.com/xloader/XLoader.zip>

Download and unzip XLoader. Open it, then point to the file by clicking the ellipse (...) under the window with the heading "Hex file." The device must be "Uno." Drop down the box for COM port, and make sure you select the COM port that the Uno is occupying. In most cases, only one COM port will show. If not, try both. Baud rate must be 115200. Click upload and wait for the confirmation.



### Focusing the Laser:

After you've mounted it, plugged it in, flashed your control board with the appropriate firmware, and PUT ON YOUR RED TINTED SAFETY GLASSES, you can focus the laser. Start with the bottom of the laser two to three inches from your work pieces.

You should have your glasses on at this point. Warn others that a laser is coming on! Plug the laser power supply into the driver, turn the key on, turn the rocker switch on, and press the reset button on the laser driver board.

On your G code sender command line, set the G1 mode. **G1 1.1, which is what your Carve King is using, will not turn on the laser when G0 mode is active. Example: G1 F600**

Give the intensity setting of 33% by typing S4000 (the range is S0 to S12000) into the command line and pressing enter. Now give the command M3 and the laser will come on. The laser should be on now. Turn the focus ring on the laser (it's the bottom ring just barely protruding out the bottom of the black finned heat sink), clockwise or counterclockwise in order to get a point as small as possible. You should be able to get about a 0.2 to 0.3 mm spot size. It will probably be a somewhat rectangular spot. That's okay, it's just the nature of this laser diode. If you can't get a spot that small, adjust the Z height up or down a bit.

### Software resources:

JTech has an excellent page that introduces different software packages. Their Inkscape plug-in is free!

[https://jtechphotonics.com/?page\\_id=1980](https://jtechphotonics.com/?page_id=1980)

Most laser operations will require there not to be any Z movements. The below applet will remove all Z movements from a g code file that wasn't produced with a program designed for laser operations. You should review the output to make sure it's ok.

<http://qcgeek.com/laseroko/>

LaserWeb is an open source laser program being actively developed. It's free. You can get it here:

<https://github.com/LaserWeb/LaserWeb4>

A program that we like for engraving pictures is Pic Laser by PicEngrave. It can be purchased here:

<http://picengrave.com/PicLaser.htm>

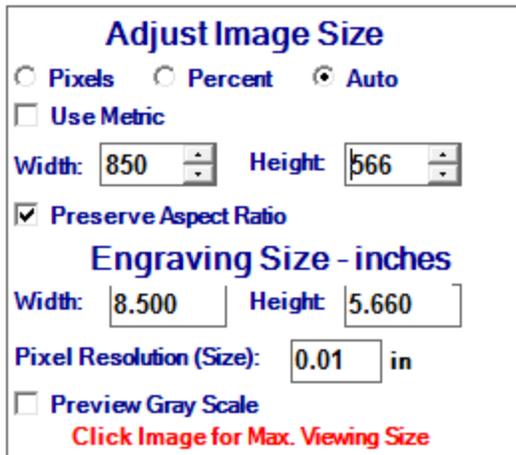
We will provide a very brief overview here, but we are not responsible for this program in any way and we have no relationship with PicEngrave other than being their customer. Direct all technical inquiries to them.

When you first open PicLaser, it will have buttons to Select Image and Edit Image. PicLaser works with .bmp files, so you will first have to click edit image to allow you to use things such as .jpg picture files that are probably most common on your computer. Select an image that you would like to use. Once loaded, you may need to click on the picture in the PicLaser screen so it will fit it properly. This seems to be a glitch.

At the top you will see this bar that allows you to add different effects to the image. You can play with this to see what you like, but most often you'll probably use Original Image.



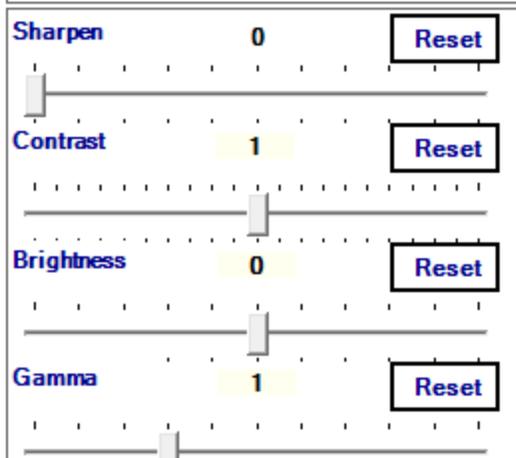
Now, on the left side is a toolbar to further manipulate the image. Click on "Auto" so you can see what size the final image will be. Leave "Preserve Aspect Ratio" checked and adjust the width and height to the size you want, keeping in mind the movement area available to you on the machine and the level of detail you want to be able to achieve. You may need to click on the image again to get it to fit in the window correctly like you did before.



You can click Preview Gray Scale to get a little more of an idea of what it will look like engraved.

Once you are satisfied with the size and other settings, click the Save button at the top left. Just rename it something easy to recognize.

Now at the top, click Return then Select Image and point to the file that you just saved. This will bring you to a screen that shows you the picture and allows you to manipulate the actual engraving settings.



Click on Change Settings. It will be at the top.

This will allow you to set up how fast you want the machine to run, the power settings, the engraving direction, the pixel size, and a number of other settings. You will need to find what works for you, but use these as a guideline. These settings have been proven to work. Notice that we left "max laser value" blank below. With

the Carve King laser firmware, a value of 11000 or 11500 will allow the laser to approach maximum intensity without completely maxing it out.

The feed rate will impact how dark the image is. 1200mm/min is a good starting point, but you could go as faster if you can accept a lighter burn. You might want to change file extension to .txt instead of .ngc. Notice that the pixel resolution will affect how large the engraved image is. 0.25mm to 0.30mm are good values. You won't be able to see how the size of the engraved image will change until you click the Save Settings button at the bottom.

**Click Labels or Values to Change** b107

Feed Rate	<input type="text" value="1200"/>	mm/min.	<input checked="" type="checkbox"/> Metric
Pixel Resolution	<input type="text" value="0.2500"/>	mm	Start Comments <input "="" type="text" value="("/>
Max. Laser Value	<input type="text"/>	mm	End Comments <input type="text" value=")"/>
Min. Laser Value	<input type="text" value="30.0000"/>	mm	
Laser Off Command	<input type="text" value="M5"/>		
Laser Control Command	<input type="text" value="S"/>		

Left - 45°  
 Vertical  
 Horizontal

Engrave Outline    No. Passes   
 GRBL     Std. Gcode     J-Tech

File Extension   
File Directory   
PreCode Commands   
PostCode Commands

Once you have done this, click Load Image at the top left of the PicLaser program. PicLaser will advise you that it is collecting file data and ask you to wait. Then click Make G Code at the top, followed by Save G Code. It can take anywhere from several seconds to a couple of minutes to save all the G Code. The G Code file will be saved in the directory that you specified. Be advised that PicLaser does not automatically put a G90 (sets absolute positioning) at the top of the G Code file. The jogging controls on many g code programs such as Universal G Code Sender may set a G91 anytime you use the jog arrows. Before you send this file to the machine, you need to issue a G90 in the command line as PicLaser's g code files are programmed for absolute positioning. You might also want to list a G90 in the "PreCode Commands" section of the settings window so it always places a G90 in the G Code file ahead of movement commands.

