

# Setting up the JTECH Laser System on MillRight CNC machines

version 1.05

## Safety:

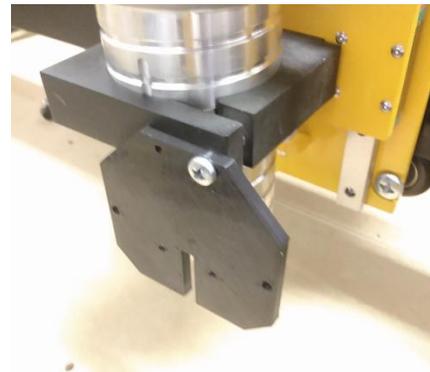
You must understand that this laser is dangerous. It is meant for adult use only. It can blind you and others. The laser light should never be viewed from any distance without the red tinted safety glasses being worn. The blue light that this laser produces can damage the naked eye even if it doesn't 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. 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 3D printed mount to hold the laser. This mount will attach to your router mount from MillRight CNC through the pinch bolt.

You will attach the laser mount to it as shown in the picture to the right using the provided M5 screw. Some mounts will have two holes in the front. Use the pinch hole screw on those mounts, too. The 3D printed piece has holes that are M4 while the pinch bolt fastener in the router mount is M5. Use the head of your Phillips head screwdriver to open up the hole some (just press it and turn and it will widen the hole). Then take the pinch bolt (M5 screw) out of the router mount and screw it into this hole. It will cut threads right through it. Then you can screw it into the router mount. The 3D printed quick disconnect piece is compressively quite strong, so you'll be able to tighten right against it while the router mount tightens against the router.



The laser diode is already enclosed with the fan already installed. The laser mount has magnets that allow the laser to snap to it. There are pins on the back of the laser that fit into corresponding holes on the laser mount.

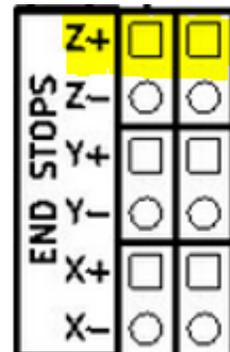
If you have the Mega V, Carve King, or Power Route, put the fan wires and the laser wires through the drag chains. The laser comes with a fairly large connector on the end that is difficult to push through the drag chain without creating unseen damage to inside of the connector on the **Carve King**. 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 if you have a **Carve King**. 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.



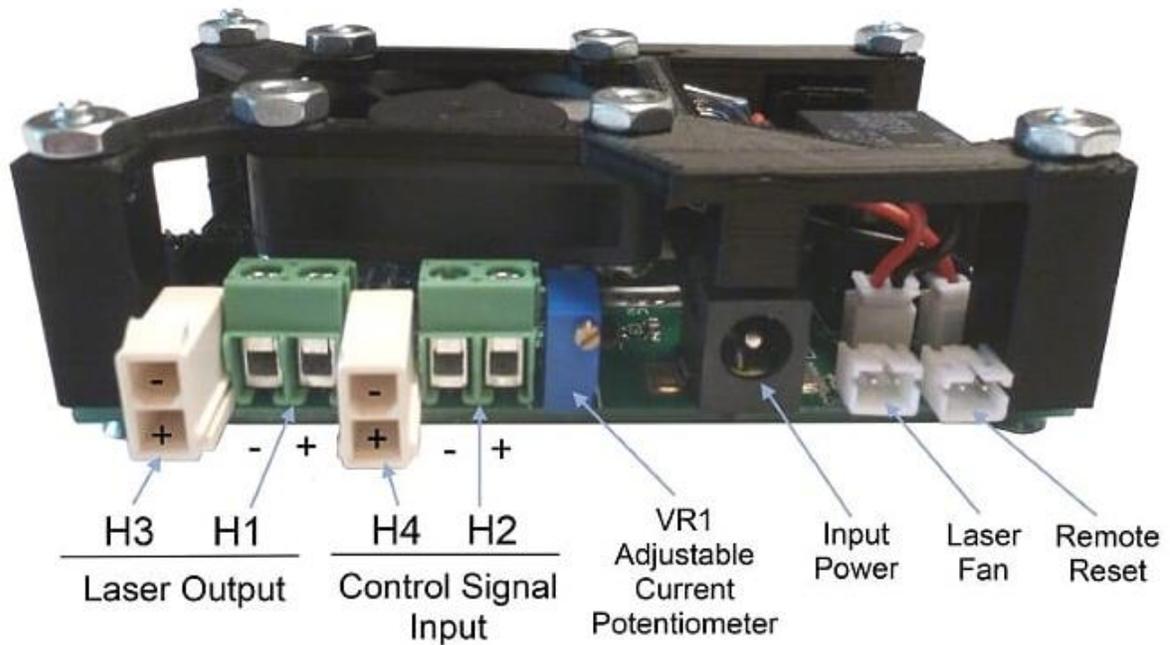
### Connecting the laser:

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”.

This section applies to M3s and Carve Kings shipped before 11/30/2020. Now we'll connect the laser control signal for the M3 or Carve King. 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+” pictured to the right. 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 on the previous page 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. **With the Mega V and Power Route, you will just plug the AC connector in the last port on the right if looking at the back of the electronics box. Pictured to the right.** If you bought a laser for your Power Route or Mega V at a later date, you will have to manually connect the wires to the board. For the Power Route and 3 axis Mega V, connect the red wire to the port labeled PWN on the CNC shield (Green board on top). The black wire can go to any port labeled GND. For 4<sup>th</sup> axis Mega V, the red wire will go to the same port, but the black wire cannot go to the GND port to the left of the PWM port. All other GND ports are ok.



**Software setup:**

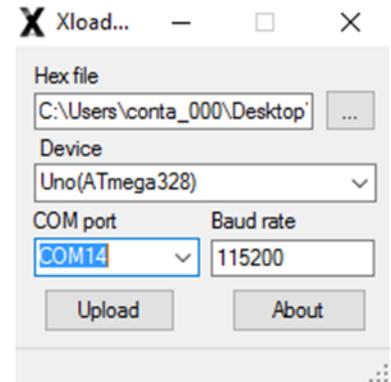
As a reminder, if you bought this laser with your machine kit you do not need to reload any firmware. It was shipped to you with the appropriate firmware. If you have the M3 please email us at [support@millrightcnc.com](mailto:support@millrightcnc.com). If you have the Carve King, get the new firmware by clicking this link: <https://sabercathost.com/9auD/Carve King Homing and No Homing Laser Spindle.hex>

**Note: Carve Kings shipped after Nov 30<sup>th</sup> 2020 will already have the laser firmware installed. The Power Route and Mega V already have the correct firmware also.**

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

<https://millrightcnc.com/wp-content/uploads/2020/08/XLoader-zipped.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.



### Testing 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 start with the bottom of the laser 1/8" from your work pieces. That is the focus height of the laser.

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. **Grbl 1.1, which is what your machine 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. **Note that on the Mega V, the S value that corresponds to 100% laser intensity is "S100". Be sure to use the 0 to 100 S value scale which directly relate to the percentage of output. On some other MillRight models, the S value is scaled to correspond to a 12,000 RPM spindle. Type \$\$30 into the command line. If it is set to 12000, then the machine is set to the 12000 scale. If \$\$30 is set to 100, it is set to the 100 scale. All machines will eventually be set to the 100 scale.** To set the Mega V, or any 100 scale machine to 33% type S33 into the command line and press enter. Now give the command M3 and the laser will come on. The laser should be on now. Give an M5 command to turn off the laser. Do not leave the laser on too long at one impact point as it could start a fire.

### 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>