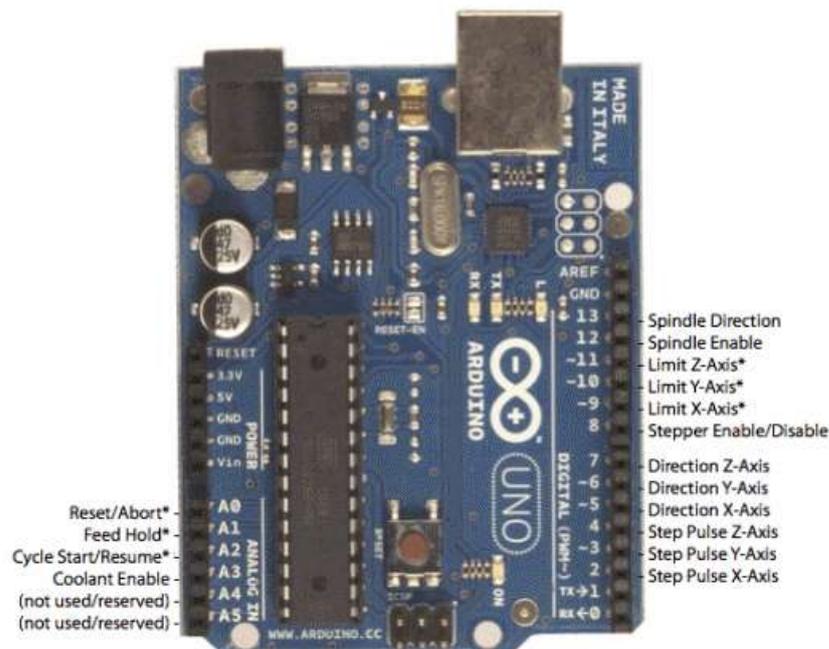
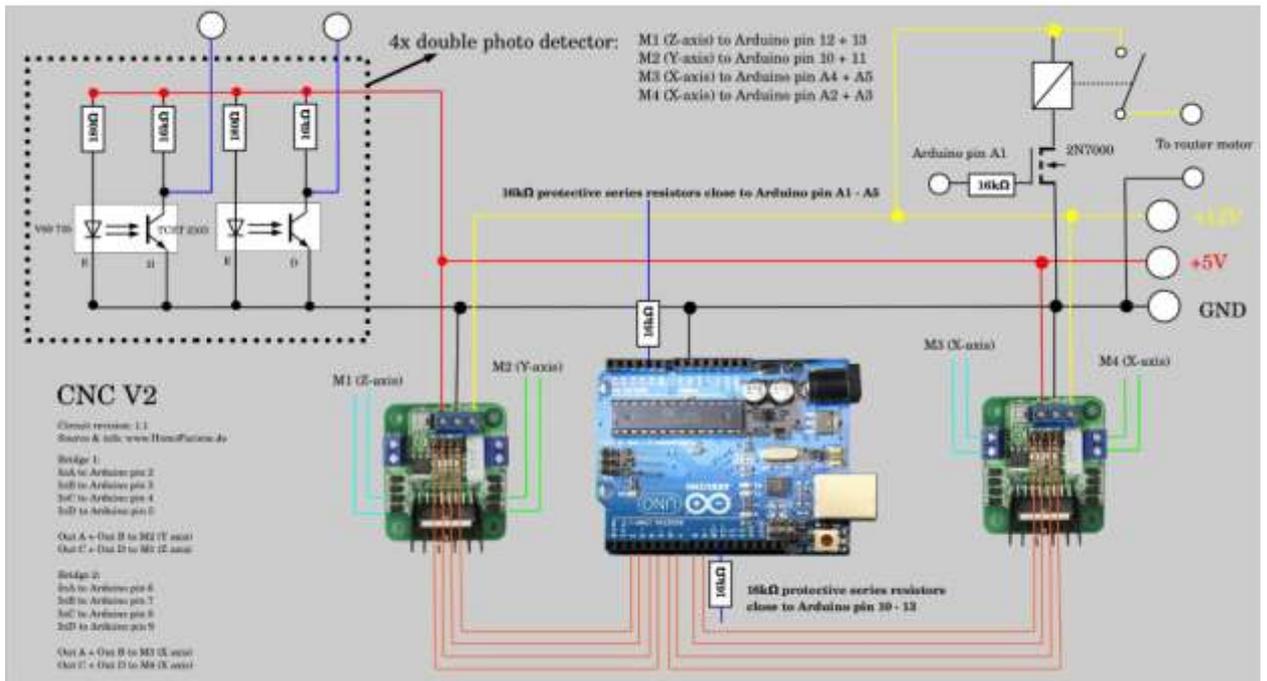


Endurance lasers with MillRight CNC machines

Endurance laser has compatible mounting holes for M3 screws. Additional mounting kits are available upon request. The laser can be mounted to your router mount or spindle mount that you purchased from MillRight CNC.

If you have our additional laser mounts, you will attach the laser mount to it as shown in the picture to the right using the provided M3 screw. The hole is tapped for the M3 screw.

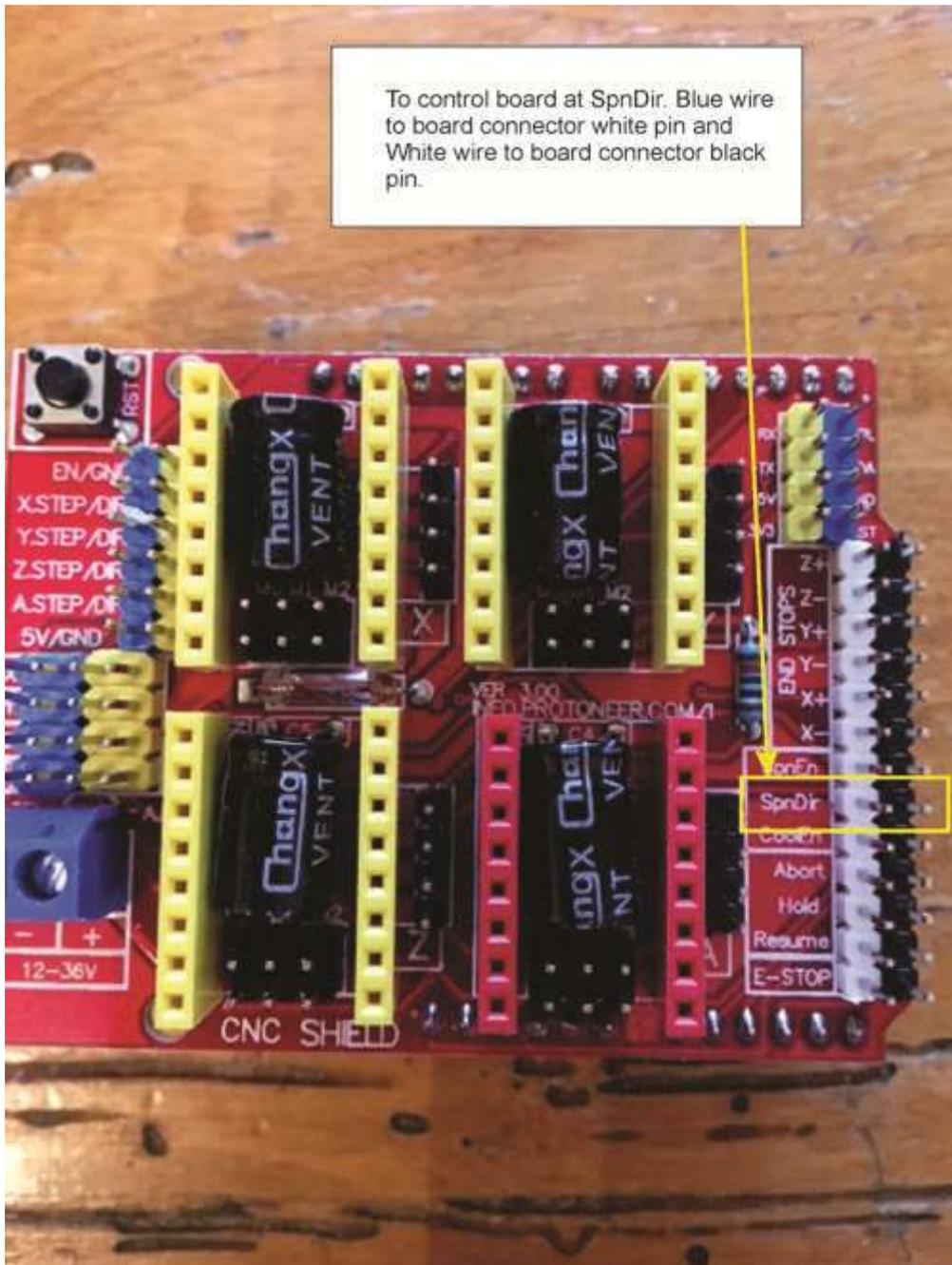
Connecting the laser:



* - Indicates input pins. Held high with internal pull-up resistors.

The laser is connected to D12 (if the laser goes without an Endurance MO1)

If the laser is connected through Endurance MO1 then D12+ GND

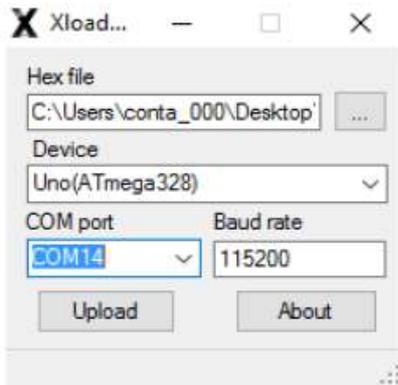


Also connection is possible directly to SpnDr

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.

Additional software resources:

T2Laser is a Benbox replacement for Grbl based laser and CNC systems.

It supports grey scale images, vector graphics and also has basic sketch features. You can add text to images, contour cut-out images or convert raster to vector and output it to the laser all in a single G-Code file.

Advanced algorithms for photo engraving and clipart / logos, as well as multi-pass vector cutting capabilities. Resolution, speed and power are user defined so you can maximize quality while reducing the engraving time.

<http://t2graphics.weebly.com/> <http://www.t2laser.org>

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:

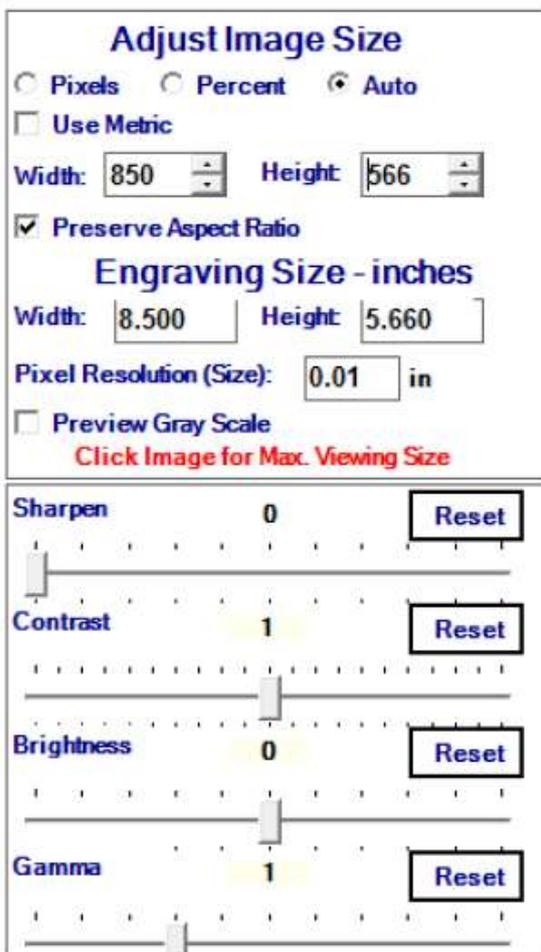
<http://picengrave.com/PicLaser.htm>

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" value="255.0000"/>	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 Engrave Outline No. Passes
 Horizontal GRBL Std. Gcode J-Tech

File Extension
File Directory
PreCode Commands

PostCode Commands

<input type="button" value="Edit Image"/>	<input type="button" value="Change Settings"/>	<input type="button" value="EXIT"/>	<input type="button" value="Help"/>
-------------------------------------------	------------------------------------------------	-------------------------------------	-------------------------------------

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.