Full guidance how to connect lasers to 3D printers and CNC machines.

How to connect lasers on different CNC machines and 3D printers.¹

Table of contents
Full guidance how to connect laser to 3D printers and CNC machines..................1
Azteeg X3 Pro.................................................................................................5
  Electrical Connection ..............................................................................5
  Software ...............................................................................................5
Using PWM to control the laser ..................................................................6
CNC Shark Upgrade......................................................................................6
  Electronics..............................................................................................6
  Controlling the laser .............................................................................9
GeckoDrive 540.............................................................................................9
  Mechanical: ..........................................................................................9
  Electrical: ............................................................................................9
  Software: ............................................................................................11
Lulzbot.........................................................................................................12
  Electronics............................................................................................12
  Mechanical............................................................................................15
  Software ..............................................................................................16
MACH3 Laser Upgrade ...............................................................................19
  ***NEW UPDATE**** ..............................................................................19
  Hardware Overview ................................................................................19
  Software ..............................................................................................20
Which way to choose, Spindle or Coolant?...................................................20
  Spindle as Laser ...................................................................................20
  Coolant as Laser ..................................................................................22
  Using Spindle PWM to Control Laser....................................................23

¹ Used based on jtechphotonics.com materials
Picture Engraving with Mach3 ........................................................................................................ 25
Solution to Possible Delay in Mach 3 using M3 and M5 Commands ........................................... 26
Laser-Bot ........................................................................................................................................ 27
MakerGear M2 ................................................................................................................................. 28
  Electrical Connections .................................................................................................................... 28
  Firmware and G Code .................................................................................................................... 30
MakiBOX .......................................................................................................................................... 31
  Laser Upgrade Implementation ....................................................................................................... 31
Mostly Printed CNC Upgrade ........................................................................................................ 31
  Mechanical ..................................................................................................................................... 31
  Electrical ........................................................................................................................................ 32
  Software ......................................................................................................................................... 33
Using PWM to control the laser ...................................................................................................... 33
Controlling the Machine .................................................................................................................. 33
Using Your Machine ....................................................................................................................... 34
Openbuilds – C Beam / OX .............................................................................................................. 34
  Mechanical ..................................................................................................................................... 34
  Laser Holder ................................................................................................................................... 34
  Electrical ......................................................................................................................................... 35
Software ........................................................................................................................................... 35
Running the Laser Machine ........................................................................................................... 36
PrintRBot .......................................................................................................................................... 38
  Mechanical: .................................................................................................................................. 38
  Software: ....................................................................................................................................... 38
Replicator 2 and 2X .......................................................................................................................... 40
  Laser Upgrade for MakerBot Replicator 2 and 2X ....................................................................... 40
  Connecting the Laser ...................................................................................................................... 40
  Turning it ON and Off with G Code ............................................................................................... 41
  Mechanical Mount .......................................................................................................................... 41
RepRap – RAMPS1.4 ....................................................................................................................... 41
  RAMPS1.4 Laser Upgrade .............................................................................................................. 41
  Hooking the Laser Up ...................................................................................................................... 41
  Software ......................................................................................................................................... 44
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Using PWM to control the laser</td>
<td>45</td>
</tr>
<tr>
<td>Drawing Software with Inkscape</td>
<td>45</td>
</tr>
<tr>
<td>Rigidbot Upgrade</td>
<td>45</td>
</tr>
<tr>
<td>Electronics</td>
<td>45</td>
</tr>
<tr>
<td>Software</td>
<td>46</td>
</tr>
<tr>
<td>Mechanical</td>
<td>47</td>
</tr>
<tr>
<td>Shapeoko 2 CNC</td>
<td>47</td>
</tr>
<tr>
<td>ShapeOko 1 and 2 Laser Upgrade</td>
<td>47</td>
</tr>
<tr>
<td>Mechanical Mount</td>
<td>47</td>
</tr>
<tr>
<td>Electronics – Connecting the Kit</td>
<td>48</td>
</tr>
<tr>
<td>Software – Controlling the Laser</td>
<td>50</td>
</tr>
<tr>
<td>Set-up for Photo Engraving</td>
<td>51</td>
</tr>
<tr>
<td>Workflow Examples</td>
<td>51</td>
</tr>
<tr>
<td>Running Your ShapeOko 2 Laser</td>
<td>52</td>
</tr>
<tr>
<td>Shapeoko 3 CNC</td>
<td>53</td>
</tr>
<tr>
<td>Mechanical</td>
<td>53</td>
</tr>
<tr>
<td>Electrical Hook Up</td>
<td>54</td>
</tr>
<tr>
<td>Software – Controlling the Laser</td>
<td>55</td>
</tr>
<tr>
<td>Running your laser Shapeoko 3</td>
<td>56</td>
</tr>
<tr>
<td>ShopBot</td>
<td>57</td>
</tr>
<tr>
<td>Electrical</td>
<td>58</td>
</tr>
<tr>
<td>Software</td>
<td>60</td>
</tr>
<tr>
<td>Mechanical</td>
<td>61</td>
</tr>
<tr>
<td>Solidoodle</td>
<td>61</td>
</tr>
<tr>
<td>Laser Upgrade Implementation</td>
<td>61</td>
</tr>
<tr>
<td>Solidoodle 4, Workbench, &amp; WB Apprentice:</td>
<td>61</td>
</tr>
<tr>
<td>Stepcraft CNC</td>
<td>65</td>
</tr>
<tr>
<td>Mechanical</td>
<td>65</td>
</tr>
<tr>
<td>Electrical</td>
<td>65</td>
</tr>
<tr>
<td>Software</td>
<td>66</td>
</tr>
<tr>
<td>Mach3 Setup</td>
<td>67</td>
</tr>
<tr>
<td>UCCNC Setup</td>
<td>67</td>
</tr>
<tr>
<td>Running your Laser Stepcraft Engraving Picture</td>
<td>70</td>
</tr>
</tbody>
</table>
Vector Engraving ............................................................................................................. 72
TinyG ............................................................................................................................... 73
Electrical Connection ................................................................................................... 73
PWM Configuration ...................................................................................................... 74
Software Commands ..................................................................................................... 76
Running the TinyG Laser .............................................................................................. 77
Ultimaker 1 & 2 .............................................................................................................. 77
  Laser Upgrade Implementation ..................................................................................... 77
  Upgrading Ultimaker 1 ............................................................................................... 77
  Electrical ....................................................................................................................... 79
  Using the FAN Connection firmware mod ................................................................. 80
Velleman K8200 ........................................................................................................... 81
  Laser Upgrade Implementation ..................................................................................... 81
X-Carve .......................................................................................................................... 83
  Mechanical ..................................................................................................................... 83
  Electrical ......................................................................................................................... 84
  Original X Carve with Arduino Uno ........................................................................... 84
  Laser Commands ......................................................................................................... 85
  Running The Laser X Carve ....................................................................................... 85
Azteeg X3 Pro

The Azteeg X3 Pro is a good controller board that is relatively cheap. It has a lot of expansion slots for making all different types of machines. We have had a lot of interest in getting instructions on how to upgrade this board, so here it is.

**Electrical Connection**

The board has a lot of extra headers for expansion on it. This is the one we are going to use. Connect the laser to the header HE7. The “+” is on the left and the “-” is on the right. It is also labeled on the board. Connect this to the input screw terminal H2. Here is the diagram for the board with the laser connection labeled:

![Diagram of Azteeg X3 Pro board with laser connection labeled](image)

**Software**

The software is the same for all of the upgrades here. The commands to turn on and off the laser are:
LASER ON: M106
LASER OFF: M107

Using PWM to control the laser

You can have power control as well with the Azteeg X3 Pro board. Simply use “M106 Sxxx” command to turn on the laser where “xxx” is a number between 0 and 255. 255 is full power and 0 is off.

- M106 S255 = Turns the laser on at full power
- M106 S0 = Turns the laser off
- M106 S127 = Turns the laser on at 50% power
- M107 = Turns the laser off as well

CNC Shark Upgrade

We have gotten requests for upgrading the CNC shark by Rockler manufacturer. It is a nice woodworking tool that can do a lot of projects in your woodworking shop. Here are the instructions on how to upgrade your CNC shark.

Electronics

There are several versions of the CNC shark that have been developed over the years. You need to look at your controller and see if it has an output for the spindle to turn it on and off. Many of the newer
models like the MAKO have this feature. If you do, then you are in luck! The spindle control uses a relay to switch the power on and off to the spindle. This relay is controlled by a DC voltage signal. This is the signal we need to turn on and off the laser.

If you open up your controller box, it should look something like this:

You are going to use the signal from the control board to the spindle relay and jumper over to the laser driver board input. Here is a more detailed picture:
Attach a wire from the relay “+” signal to the laser driver “+” terminal H2. Attach another wire on the relay “-” signal to the laser driver “-” terminal H2. That is it, you are done!
Controlling the laser

You now can control your laser on and off using the same spindle commands of M3 to turn on the laser and M5 to turn it off.

GeckoDrive 540

We get a lot of people asking about connecting the GeckoDrive products to our laser system. It is normally pretty easy with most motor controllers, but with the Gecko Drive there is a little trick. It is a pretty easy setup and will work great once it is all set up! Adding a laser to your CNC setup with your GeckoDrive 540 is super easy and fun!

Mechanical:

Well, you have a CNC so you probably have some room on the spindle to mount your laser. If you have a nice aluminum mounting plate, add some thermal compound to the back and you should be ready to go! Just drill a couple of holes and mount.

Electrical:
The GeckoDrive 540 does not switch the positive, but rather sinks current via the ground. So, you will need to set up your output a bit different than normal. You will need to either use the motor power supply or another power supply you have for relays. Make sure this power supply is less than 36 volts and you are good. To hook it up:

Connect the Positive of the power supply to the “+” input terminal on the laser driver H2. Connect Position 5 (for output 1) on the GeckoDrive output terminal to the “-” input on the laser driver terminal H2.

Here is the picture of connecting using the motor power supply:

![Motor Power Supply Connection Diagram]

Here is the picture of connecting using an alternate Relay supply:

![Alternate Relay Connection Diagram]
Software:

Many people use Mach3 to control their CNC using the GeckoDrive motor controller. You will set up the software to control the laser similar to any output.

There are two outputs you can use on the Gecko. The pictures above show using terminal 5 on the Gecko header.

**OUTPUT 1:**
Gecko Header Terminal 5
Mach3 PIN 17

**OUTPUT 2:**
Gecko Header Terminal 6
Mach3 PIN 1

You can configure the Mach3 in the ports and pins section to use either output 1 or output 2. Use either pin 17 or pin 1 depending on which you choose.
Lulzbot

One of the newer open source printers that is gaining a popular reputation for being a well built printer that just plain works is the printers from Lulzbot.

Here we outline the instructions for upgrading the Lulzbot mini, but the Taz is just the same. It has the same extruder and electronics, the platform is just a bit larger. If you want to see how to build a laser only system, [check out our blog post here](#).

So here it is – Instructions on how to upgrade the Lulzbot printers. Let’s start!

****(NOTE: The pictures are mainly for the Lulzbot Mini and the TAZ 5 printer. If you have the TAZ 4, there is a separate mount and a bit different electrical connection).

**Electronics**

**Mini**

We are going to use the fan1 connection on the Rambo board to control the laser. It has PWM control needed to do picture engraving and is easily controlled with M106/107 commands in software. We
decided to mount the laser in front of the extruder so you can print then laser without any changes. We choose to mount the driver on the side of the printer because there was already a hole for the driver to attach to and the existing cables in the laser upgrade kit will reach it. We routed the laser and fan cables up the wiring harness chain to the Z axis and made it all nice and tidy with zip ties. You can cut the fan cable and either splice the wires to the laser driver input H2 or maybe even put a nice little switch in to change between laser and printing.

The Mini uses the RAMBOmini board.
Open the electronics enclosure to see the miniRAMBO board.

It is the Black and White cable.

**Taz 4, 5 and 6**

The Taz is pretty much the same, but it will be using the FAN0 connection on the Rambo board. It is:
**Mechanical**

**Mini and Taz 5**

We designed a nice little mount for the laser to attach to in front of the extruder. The good news is it just needs one additional screw! Get a small M3 screw and it will mount to the existing little hole in the aluminum plate on the extruder mount. Take the other screw from the left of the assembly off and put it through the new mount.
Taz 4

A very helpful customer, Russel Bogner designed a mount to fit on the previous Taz printers. It is very similar and mounts in front of the extruder. You use the same screws that hold the extruder plate down to put the mount in.

To protect the build plate we put some plastic with a sheet of aluminum on top of it and taped it in place. You can use whatever material you want for a sacrificial material and tape it or secure it somehow to the platform.

Software

***** You might not need to do this part. Some of the Lulzbot printers were shipped with firmware that will work. Check in Repetier Host after you install your laser whether it will work with the FAN SLIDER in the manual tab. If you can slide the slider bar and the laser gets brighter the more you slide, you are good. No changes needed. If it just kinda flashes and gets bright then dark again, then you need to update your firmware. ****

We had to modify the firmware of the Lulzbot because the PWM frequency was set way too high for it to control the laser for our machine. Here is how:

You can get the firmware from Lulzbot here:

Lulzbot Mini Firmware

or for the TAZ printer it is in the software folder here:

Lulzbot TAZ folder

Download the .tar file and open it with a program like 7-zip and extract all of the files.
Make sure you have the newest version of the compiler downloaded from Arduino HERE. Open the file called “marlin” that is an arduino sketch file.

Once you double click it, it should load up the entire sketch for you. Set your board as Arduino Mega or Mega 2560.
Then in config.h, comment out the line:

```
//#define FAST_PWM_FAN
```

Then make sure your are connected to the correct COM port. Make sure your printer is connected to the printer.

Compile your new firmware and send it to your printer:
We used Repetier Host to control the printer. We like the “preview” feature for the G Code and it does a good job with everything else as well. You need to adjust some of the settings because the table will be slightly smaller. Here are instructions for the configuration:

Lulzbot Laser Setup Instructions

Ok, now let’s get to the upgrading with pictures!

MACH3 Laser Upgrade

MACH 3 is a very popular software package for CNC machines that enables motion control on a normal computer via a parallel port without the need for fancy extra controller boards running motion control applications. This simplifies the interface boards and allows for many manufacturers to make custom widgets.

We get a lot of questions every day on whether our upgrade kits work with MACH3. The answer is a simple YES! But the details behind it depend on the interface board you buy from a third party. There are plenty of outputs on MACH3 that you can use to control the laser and every interface board we have seen implements at least one of these outputs.

***NEW UPDATE****

Many people are asking for an easier way to set-up the laser with MACH3. We have just introduced a new way to connect the laser using a “pass through” DB25 break out board. Check out the section under Hardware Overview.

Hardware Overview
The implementation is pretty simple, but first we need to understand the details of what is going on behind the scenes. Basically you have four things in the picture in terms of hardware:

1. A computer running MACH3 software with a parallel port.
2. An interface board with terminals for motors, inputs, and outputs.

The computer connects to the interface board, the interface board to the laser driver board, and then to the laser.

The end goal is to be able to turn on and off the laser with G Code in software on the computer. Here is an example of a TB6560 4axis controller board with MACH3 connecting a laser:

If you have other controller/interface boards, it is pretty similar. Look at the manual and find where they have their outputs and connect to the appropriate one.

Many motor controllers for use with MACH3 have poor documentation or the outputs just do not work correctly. All of the needed signals for the motors are still passed through to the computer, but the signals the laser needs are plucked from the cable using the breakout board and terminal strips. It is all set-up like this now:

**Software**

Now that you have your hardware connected, how do I get the laser to turn on and off with a G Code program? That is a good question! You will need to configure your MACH3 software to tell it which output will be the laser. In this, there are three ways to go. You can either make the laser a Spindle, make the laser a Coolant, or write your own custom VB script and make the laser whatever output you want. We will cover the first two options here.

**Which way to choose, Spindle or Coolant?**

Well, if you already have everything all set up for your machine and you don’t want to mess with changing the spindle connections and outputs you already have, maybe the Coolant is the way to go. The Spindle option however has the built in ability to control via PWM! Yes, that’s right, you can have power control built right into the MACH3 software. The other good news is that MACH3 supports multiple machine configurations, so you can make a new one and call it “Laser” and build a completely spindleless configuration. Also, many G Code translation programs already output in terms of M3 and M5 commands for the spindle start and stop, so you won’t have to change anything in the G Code if you go with Spindle.

**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”

In the “General Parameters” section, change all of the delays to “0” seconds. They will default to a “1”, which will leave a mark on your object you are engraving because it will pause for a second. If you are still having issues with a delay, look at the last section in this article with a possible solution.

**Coolant as Laser**

So let’s say you don’t want to set up a whole other configuration for your machine, or you already have a spindle on the machine but not coolant and you don’t want to change anything. Well, adding the laser to the coolant output is just as easy as the last step with the spindle.

In the “Output Signals” tab, map the Output #1 to Port 1 and Pin 17. You can make this another output you want, just make sure you are consistent with the setup. Then in the “Spindle Setup” tab (see the last picture), uncheck the “Disable Flood/Mist Relays” and put a 1 in the “Mist” box and a 1 in the “Flood” box. This will now enable pin 17 whenever a M Code “M7” or “M8” is in the program.

To turn on and off the laser it will be:
Laser ON: Either “M7” or “M8”  
Laser OFF: “M9”

**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 “General Parameters”, make sure all of the delays are set to Zero.

In the “Motor Outputs” tab, map the pins for the spindle to 1 and 0. Remember, this number is specific to your set-up and may require a different pin out. Just check in your instructions on which one it is.
In the “Config” —> “Spindle Pulleys” 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.

**Picture Engraving with Mach3**

Note that this PWM method normally produces a slight delay in each command. If you want to use the PicEngrave software, we have developed an encoder solution that will allow you to control a 4th axis to produce the commands needed to run full speed with picture engraving. This method was developed by Jeff and John at picengrave.com and uses the 4th axis step and direction signals to produce the PWM
output needed to engrave pictures in 8bit resolution. The PicConvert™ board converts the step and direction outputs from mach3 into a PWM signal for the laser driver.

Setting up the board is not trivial, so we recommend you read the manual first before purchasing so you understand the details involved with getting it set up. The manual is located here:

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

Some customers are experiencing delays in the start of the laser when doing vector drawings 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](http://www.machsupport.com/forum/index.php?action=printpage;topic=23636.0)

***If you are using the Ethernet Smooth Stepper you will also need to set the output in the “General Config” page on the ESS plugin. In this example it is set to output #1. Also, turn off any dwell associated with this output method by unchecking the boxes in the dwell section.
**Note this does not fix picture engraving. You still need the PicConvert™ board mentioned above for this**

### Laser-Bot

**Laser Bot™ Personal Manufacturing System**

**Laser Upgrade your 3D Printer or CNC machine! Upgrade instructions for many popular brands!**

Who doesn’t want to add laser capability to their 3D Printer?

This can be used for many fun projects as well as useful manufacturing projects like cutting solder stencils for PCBs (as seen in the video above).

**Setting up the Hardware:**

The signal for the automated build platform motor on the MakerBot heated build platform circuit board is connected to the input of the driver board using a custom cable.

**How it All Works – The Workflow Diagram**

Once you get all of the mechanical figured out, it is pretty easy to get everything going. The basic workflow is:

- **Draw**: Use a drawing program that will output G Code files based on path objects. Corel Draw and other fancy programs will do this. Inkscape has an open source free program that does the job.
- **Modify**: Use an editor to modify the G Code file to turn on and off the laser. The “On” command is M106. The “Off” command is M107. It also might be useful in developing a process to have a “pause” command after turning on the laser and the next command to move. This gives the material a bit of time to heat up before moving.
- **Load**: Load the G Code file into Replicator G and you are ready to go! Press the “build” button and the process should start.

Check back for more projects and instructions on how to create your own solder stencils using this laser cutter!

Our fourth generation laser kit offers even more than ever! New safety features make it not only easy to integrate and use but also now the safest way to use your new class 4 laser. These are not toys, so we
added safety features like interlocks, a key switch, power off reset, coupled with the safety features already there like visual indicators.

**MakerGear M2**

The MakerGear M2 printer is the top selling 3D printer on Amazon.com and it is their 3D generation of their printers so it comes with great accuracy and build quality. That being said, what if you want to put a laser on it? Good question! These are the instructions on how to get the M2 set up with the electronics to control the laser driver board via G Code.

The M2 uses the Rambo board from the RepRap project (info here: [http://reprap.org/wiki/Rambo](http://reprap.org/wiki/Rambo)). The board has two fan outputs as well as a header for accessory PWM outputs. Considering it has a nice header for the PWM outputs, I think this is the way to go.

Of course, you will still need to figure out a mechanical design to mount the laser to the printer Z axis, but that should be a fun project on the printer!

**Electrical Connections**

You will have to remove the top of the electronics box to get to the Rambo board. The header for the PWM signals are just right above the processor chip in a single row called “PWM EXT”. There are 6 pins, starting with pin 1 on the left. A picture is shown below where the header is. It is labeled “Laser Upgrade Header”.

---

**NDURANC**

**EnduranceRobots.com**
**EnduranceLasers.com**
**Gf@EnduranceRobots.com**

US Phone: +1 239 302 0053
International: +7 916 225 4302
(whatsapp, viber, telegram)
Skype: George.fomitchev

---

**28**
The M2 has the feature to add an LED to the printer, so we are going to use this output. There are two of the pins that are needed to connect to the laser driver board:

- Pin 2 = GROUND. Use this to connect to the "-" input on the laser driver board.
- Pin 3 = POSITIVE. Use this to connect to the "+" input on the laser driver board.

This is shown in the following picture:
If your board has a header on it, then you can either solder a wire to it, or you can get a cable here at Sparkfun (https://www.sparkfun.com/products/8672).

**Firmware and G Code**

The Marlin firmware has included in it a method for controlling the pin 13 LED signal to be able to turn it on and off with PWM. The command is “M42 P13 SXXX” where XXX is a value between 0 and 255 for the power level. Examples of some commands would be:

- **M42 P13 S255** = Turns the laser on at full power
- **M42 P13 S0** = Turns the laser off
- **M42 P13 S127** = Turns the laser on at 50% power

So, when you make a program file, just replace the “M03” code with “M42 P13 S255” to turn the laser on and replace “M05” with “M42 P13 S0” to turn the laser off.

Then run this file in the software of choice (including in simplify3D) and get lasering!
MakiBOX

Laser Upgrade Implementation

The MakiBOX is a really cool new printer out there that is super cheap. They are also using a version of the PrintRboard rev B from the RepRap project that includes an implementation of the fan. As far as I can see, they are not using a fan in there product, but it should still be part of the electronics and the firmware.

To upgrade the Makibox, just add a Molex style connector to the PrintRboard fan output and connect it to the laser driver input. Then use the M106 command for “laser on” and M107 command for “laser off”.

If you ever want to add a fan as well, you can use a SPDT switch and then can move between 3D printing with a fan and laser cutting by flipping the switch.

Mostly Printed CNC Upgrade

Ryan Zellars came up with a pretty cool design for a CNC that runs on conduit to win a design contest. Well, he won. The reason was because it is a pretty cool idea on how to make a very inexpensive multi purpose CNC platform. All you really need for this is some conduit, bearings, nuts/bolts/screws, and 3D printed parts. Mostly 3D printed parts…

The main website for the machine is located here: [www.vicious1.com](http://www.vicious1.com)

You can find out details on the BOM, how to assemble it, and there is a nice forum of users.

So there are several people who have upgraded their mostly printed CNC’s so far, so we thought we would put up some of the instructions based off of their designs here. The mechanical parts are all the same, but the controller is chosen by you. We will lay it all out here on this page on how to get started.

Mechanical

There have been a few people who have put up some mounting techniques for the laser.
Electrical

You have a choice on what you want to do for your electronics on this machine. Ryan suggests using the RAMPS 1.4 controller with the Arduino MEGA and the MARLIN firmware. It is a pretty easy and well-established solution. We will cover the RAMPS 1.4 here.

The laser driver input will need to be connected to the fan output on the RAMPS board. It is on connector D9. Here is the electrical diagram for the connections based on a 3D printer. You can ignore all of the extruder and heatbed stuff, just look at the Fan output:

Here is the detail of the D9 connection:
Simply connect the “+” wire from terminal D9 to the “+” input of H2 on the laser driver board. Then, connect the “-” wire from terminal D9 to the “-” input of H2 on the laser driver board.

If you use the Molex Mini fit jr. cable then connect the black wire to the D9 “-” terminal and the red wire to the D9 “+” terminal. This cable connects to the H4 Terminal of the driver board.

**Software**

The commands to turn on and off the laser in G Code are:

**LASER ON:**  M106  
**LASER OFF:**  M107

**Using PWM to control the laser**

You can have power control as well with the RAMPS1.4 board. Simply use “M106 Sxxx” command to turn on the laser where “xxx” is a number between 0 and 255. 255 is full power and 0 is off.

- M106 S255 = Turns the laser on at full power  
- M106 S0 = Turns the laser off  
- M106 S127 = Turns the laser on at 50% power  
- M107 = Turns the laser off as well

**Controlling the Machine**
First make sure you download the Arduino software to make sure you have the drivers installed before you plug in your board. Download it here: www.arduino.cc/en/Main/Software

Many people use Repetier Host for controlling the machine. Some settings can be found here: www.vicious1.com/repetier-host/

Using Your Machine

There are several ways to generate G Code for your machine. We have a few, but these are certainly not all of them.

- **Inkscape Plugin (FREE)**: Vector Based Outlines
- **Laser Etch**: Rastor Based Optimized Black and White
- **PieLaser**: Rastor Based Variable Intensity Picture Engraving

Once you have your G Code created then load it into Repetier Host and start engraving!

Openbuilds – C Beam / OX

Openbuilds has created a nice kit available on their website for a standard machine based off of their slot system and rails that they sell called the C Beam. It is a very nice sturdy system that is not very expensive.

Besides the mechanical portion, you have the choice of a lot of controllers for the electrical section and also the software control. An easy choice is to use GRBL and a GRBL based controller. This being said, you can also set this up as a mach 3 machine. So let’s get started!

Mechanical

The Z axis has a mount for the spindle that can also accommodate a laser. You can decide to put the laser in front of the spindle by tapping a couple of holes in the front of the spindle mount so you don’t need to remove the spindle or you can put the laser in place of the spindle. In this case, the machine is a dedicated laser machine, so the laser will accommodate the position where the spindle normally is. You can make your own mount or use the design that Jürgen created.

Laser Holder
There are two ways to mount your laser on the Openbuilds platform using their spindle holder. You can either make a mount to fit in place of the router or you can put it in the front of the router. We show both methods here. You can print them out with a 3D printer using the provided files, or you can buy the front mounted kit in our store here.

**Electrical**

There are many controllers you can use with the openbuilds C Beam machine. Jürgen decided to go with the [CNC-xPro V2 controller](https://www.openbuilds.com/products/cnc-xpro-v2) using GRBL for his machine build.

You can use the standard version of GRBL from the Github site, or our version [HERE](https://github.com/OpenBuilds/openbuilds-control-system) for photo engraving.

![CNC X Pro Laser Connection](https://www.openbuilds.com/assets/images/cnc-xpro-laser-connection.jpg)

Simply connect screw terminal D11 on the CNC xPro V2 board to the laser driver input terminal H2 or H4.

**Software**

Using the GRBL version that is installed on the CNC xPro you can set your Min and Max RPM values (which control the laser intensity). Add the settings:

- $30 = 255$ (RPM Max)
- $31 = 0$ (RPM Min)
To control the laser, the commands will then be:

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

Obviously if you set up your machine with something other than GRBL (like mach3) then use the appropriate commands for that software setup.

**Running the Laser Machine**

You do not need to switch firmware between laser engraving and routing, all firmwares will work with both.

There are a lot of different “sender” programs that will work with your machine. In the inside of your machine is something called “GRBL” that directs how the machine moves and interprets the G Code file. The G Code file is the “instructions” and GRBL just needs some program to “send” it to the GRBL that is in the firmware of your control board.

Here is the wiki page with all of the sender program available:


We like to use the “universal G Code sender” as it is the standard for the previous shapeoko machine and has a very large user base. It is located here:

https://github.com/winder/Universal-G-Code-Sender
If you are planning on doing pictures, then PicSender will work great to control the machine. You can get it here: PICSENDER PROGRAM
PrintRBot

The PrintRBot is a cool printer that you can either assemble yourself or get an assembled kit. The machine uses the PrintRBoard which has been a part of the Rep Rap project for some time. The details are located here on the Rep Rap wiki:
http://reprap.org/wiki/Printrboard

Just like the other upgrade implementations, the easiest way to upgrade the PrintRBot is to use the fan output from the control board. This way, the firmware is already done and you are ready to go. Just make sure to connect the wires to the correct side on the laser driver.

Or to be super cool, put a SPDT switch on the positive side of the fan output and connect it to both the fan and the laser driver. Connect the ground and you are good to go. Easily switch between 3D printing and the laser with a flick of the switch!

Mechanical:

As for mounting, we put ours on the side of the printer towards the front just right of the extruder. We suggest drilling the holes with the printout of the mount (or the laser mount itself) to make sure your holes are correct. Just hold it up to the side and mark it with a marker through the holes. Drill and then install. This shows the fan mount as well attached to the top of the laser. This allows the metal frame to help with heat sinking and have a fan at the same time.

Software:

You shouldn’t need to change anything with the marlin firmware that ships on the motherboard of the printer. Use the following commands in our inkscape plugin (or whatever G Code SW you want):

- **M106 Sxxx** – Turn the laser ON at power XXX, where XXX is a number between 0 and 255.
- **M106 S0** – Turn the laser OFF
- **M107** – Another way to turn the laser OFF

Examples:

M106 S127 (Turns on the laser at a power of 50%)
M106 S255 (Turns on the laser at a power of 100%)
M106 S25 (Turns on the laser at a power of 10%)

For running your programs, we generally like using Repetier Host software for our printer/laser control. It has a nifty preview window and good manual control. You can set
up the offset for the laser in the EE Prom settings and move the 0,0 point by creating an offset as well. Here is where to put your offset:

And then you can use the manual tab to control the laser to turn it down for focusing. Use the fan slider to adjust the power and turn it on by pressing the fan.
Replicator 2 and 2X

Laser Upgrade for MakerBot Replicator 2 and 2X

This was a tougher one to crack because the new MakerBot Replicator 2 and 2x are not open source. But do not fret! A good customer has tested out this solution and it works great!

Connecting the Laser

The Replicator 2 and 2X come shipped with either a REV G or a REV H of the mightyboard. This will work on both versions. We will be connecting the laser to the J11 (Fan_Cool) connector. Disconnect the fan that is there and then put in a new connector (it looks like it is a simple Molex style) to go to the laser driver. Connect to the laser driver terminal H2 with the + to the + and the – to the -. Either use a meter to see which pin on the fan connector is positive or just try to connect it. If it does not work then switch the wires because you have it backwards...

The location on the board for this is shown in the following picture:
Turning it ON and Off with G Code

To turn the laser on and off in your program use the following G Code:

- **M126** (Laser ON)
- **M127** (Laser OFF)

Note – The Sailfish firmware does not support PWM changes during the running of a file. However, Sailfish firmware beta release 7.8 has the ability to set the PWM fan level *before* the program runs. This is nice if you want to mess with the power levels of a particular G Code file without adjusting the feedrate.

**Mechanical Mount**

Of course part of the fun is figuring out how to mount the laser to the machine. Some people have already done some mounts for their builds and shared them. For the Replicator 2X, one customer has milled out the middle of the two extruders and placed the Laser between them to maximize the cutting area.

**RepRap – RAMPS1.4**

**RAMPS1.4 Laser Upgrade**

The RepRap project has quite a bit of different electronics. Many of the popular machines use a variation of just a few boards. Here are the details for upgrading the RAMPS1.4 controller to output laser control.

The details of the RAMPS controller can be found at:

[http://reprap.org/wiki/RAMPS_1.4](http://reprap.org/wiki/RAMPS_1.4)

**Hooking the Laser Up**

So it is pretty easy to do this upgrade because the terminal that is used already has nice screw terminals to connect wires. We are going to use the output associated with the extra FAN output to control the laser. This is terminal D9. Simply connect the “+” wire from terminal D9 to the “+” input of H2 on the
laser driver board. Then, connect the "-" wire from terminal D9 to the "-" input of H2 on the laser driver board. You are all ready to go!

If you are using the Molex Mini Fit Jr. Cable, then connect to the input laser driver connector H4. Connect the red wire to the positive terminal D9 on the Ramps1.4 board and the black wire to the negative terminal on the Ramps1.4 board.

Picture showing the terminal D9 connection.
Close up of terminal D9
Software

The software is the same for all of the upgrades here. The commands to turn on and off the laser are:

LASER ON: M106
LASER OFF: M107
Using PWM to control the laser

You can have power control as well with the RAMPS1.4 board. Simply use “M106 Sxxx” command to turn on the laser where “xxx” is a number between 0 and 255. 255 is full power and 0 is off.

- M106 S255 = Turns the laser on at full power
- M106 S0 = Turns the laser off
- M106 S127 = Turns the laser on at 50% power
- M107 = Turns the laser off as well

Drawing Software with Inkscape

You can use Inkscape to do your drawings and then use the LASERENGRAVER plugging to convert it to G Code. Download the laserengraver plugin that converts using the M106 and M107 commands.

Rigidbot Upgrade

The Rigidbot was a successful kickstarter project that created a printer that could be expanded based on the frame, so you could theoretically make whatever size you want. Just use your imagination and you can build a GIANT printer (if you have the time and money…). The basic design is not giant, but easy to build and affordable. So, many people bought them, and you still can at Inventapart.com. The machine is based on the Reprap RAMPS 1.4 and the marlin firmware, so it will be a snap to upgrade. So let’s start!

Special thanks go to Isaac Braud for testing it all out on his printer!

Electronics

Setting this up should be very easy. The technical details on the electronics are located on the wiki HERE. You will be using the FAN 1 connector which is located on the extruder breakout board. This is fully PWM capable as well. The laser will be on the extruder Z axis assembly, so the cord to the laser should be kept wound up and zip tied. You can use the FAN 2 connection to cool the extruder instead of the FAN 1 you just used for the laser.
Software

You shouldn’t need to change anything with the marlin firmware that ships on the motherboard of the printer. Use the following commands in our inkscape plugin (or whatever G Code SW you want):

- **M106 Sxxx**  – Turn the laser ON at power XXX, where XXX is a number between 0 and 255.
- **M106 S0**  – Turn the laser OFF
- **M107**  – Another way to turn the laser OFF

Examples:

- M106 S127  (Turns on the laser at a power of 50%)
- M106 S255  (Turns on the laser at a power of 100%)
- M106 S25  (Turns on the laser at a power of 10%)
Mechanical

Isaac designed a mount to fit on the extruder assembly.

Shapeoko 2 CNC

ShapeOko 1 and 2 Laser Upgrade

The ShapeOko project is one of the fastest growing open source do it yourself CNC machines ever. It is made from the popular makerslide and standard Arduino electronics. Check it out at ShapeOko.com. So, if you are one of the thousands who own a ShapeOko and are having fun making stuff but want more, this is your answer! Let's put a laser on it and do some engraving and cutting!

These instructions show how to electronically connect the laser driver and the commands needed to control the laser in G Code. So let's get started!

Mechanical Mount

Many people have integrated the laser onto the Z axis of the ShapeOko with success. Just look at the mounting dimensions on the documentation page for the hole locations. It is also advisable to provide a fan for active cooling behind the laser as well. Mounting a fume extractor is also beneficial when cutting materials like plastic as the fumes can be dangerous.
We have made a custom mount for the shapeoko that fits into the existing spindle holders. If you have a 3D printer you can download it here:

**Electronics – Connecting the Kit**

The standard ShapeOko2 electronics are the Arduino Uno and the GShield. If you bought from the Inventables and got the entire kit, you got an original Arduino Uno. If you bought the electronics from Synthetos then you will have a different Arduino board. No matter, the connection is the same either way!

The Shapeoko uses the popular open source machine control firmware GRBL for its operation. They now have different versions which require you to hook up the laser differently. We will cover both versions here.

**GRBL Version 0.8 and lower (Shapeoko 1 and older 2’s)**

If you bought your Shapeoko before January 2015 and have not upgraded your firmware then you probably have GRBL 0.8 on your machine. The connection is pretty easy. You can either connect to the GShield on the top or to the bottom with the Arduino. Soldering a wire into one of the holes is the easiest way to get going. Or you can solder a piece of a standard breakaway header and use a jumper wire cable. Either way you are going to need to connect the following:

- Pin 12 Labeled “12” on either Arduino or GShield to Laser Driver Input “+”
- Pin 14 Labeled “GND” on either Arduino or GShield to Laser Driver Input “-”

Here is the picture of the GShield with the hookups:
And the Arduino Uno:

The laser driver board has an input connector for input control. You can use either a Molex mini fit Jr. and use H4, or connect to the screw terminals on H2.

- **Pin 11 Labeled “11” on either Arduino or GShield to Laser Driver Input “+”**
- **Pin 14 Labeled “GND” on either Arduino or GShield to Laser Driver Input “-”**

Here is the picture of the GShield with the hookups:
And the Arduino Uno:

![Arduino Uno](image)

The laser driver board has an input connector for input control. You can use either a Molex mini fit Jr. and use H4, or connect to the screw terminals on H2.

**Software – Controlling the Laser**

Now that you got everything ready with the wiring, now it is time to turn on the laser with GCode!

The GRBL firmware supports a feature to turn on a spindle and turn it off. Typically, most of the spindles are Dremel type ones with no external trigger and require a manual turn on, so this pin probably is not used. Even if it is, there are numerous connection points so the two can share (but make sure to turn off the spindle power before you run!).

So, what are the commands? The standard command to turn on the spindle is M3 and to turn it off is M5. So, to turn on and off the laser it is:

- **M3** – Turn Laser On
- **M5** – Turn Laser OFF

If you want to use PWM for power control (and have GRBL version 0.9 or higher installed) then the commands are:

- **M03 SXXXXX**
  Where the “XXXXX” is a number between 0 and 10000. So, some examples are:
Laser ON 100%: M03 S12000
Laser ON 50%: M03 S6000
Laser ON 10%: M03 S1200
Laser OFF: M03 S0 (or M05 will work as well…)
M03 SXXX
Where the “XXX” is a number between 0 and 255. So, some examples are:

- Laser ON 100%: M03 S255
- Laser ON 50%: M03 S127
- Laser ON 10%: M03 S25
- Laser OFF: M03 S0 (or M05 will work as well…)

<table>
<thead>
<tr>
<th>M-code</th>
<th>Meaning</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>M0 (v0.8 or later)</td>
<td>Program Pause and End Stop</td>
<td></td>
</tr>
<tr>
<td>M2 (v0.8 or later)</td>
<td>Program Pause and End End</td>
<td></td>
</tr>
<tr>
<td>M3</td>
<td>Spindle direction</td>
<td>Clockwise</td>
</tr>
<tr>
<td>M4</td>
<td>Spindle direction</td>
<td>Counterclockwise</td>
</tr>
<tr>
<td>M5 (v0.8 or later)</td>
<td>Spindle Control Stop spindle rotation</td>
<td></td>
</tr>
<tr>
<td>M8 (v0.8 or later)</td>
<td>Coolant Control Flood coolant on</td>
<td></td>
</tr>
<tr>
<td>M9 (v0.8 or later)</td>
<td>Coolant Control All coolant off</td>
<td></td>
</tr>
<tr>
<td>M30 (v0.8 or later)</td>
<td>Program Pause and End End and rewind</td>
<td></td>
</tr>
</tbody>
</table>

Set-up for Photo Engraving

The firmware that comes with your X Carve will not be able to do very good picture engraving. It stops for EVERY pixel to adjust the intensity. Now, this might work if you turn your laser power on your driver really low and you are willing to wait hours for your engraving to finish but I think you might have better things to do with your time. This is why the new firmware was developed so the intensity of the laser can change without stopping for every pixel. So, in order to set up your X carve for photo engraving you will need to follow a few steps.

Workflow Examples

Consider the new laser as an “interchangeable toolhead” on the machine. Instead of the spindle, you now have a laser. It is exactly the same way to control it, but now you are drawing in 2D and not 3D. The ShapeOko website has a good tutorial on how to make 2D drawings with several programs, then send them to a toolpath generator and process it. Here is the link on how they say to complete this workflow:
Running Your ShapeOko 2 Laser

Running your new laser upgraded system will be exactly like when you were running it with a spindle on it. Just make sure you have all of your safety gear in place, the laser focused, power set, and everything powered on and ready to go. Here is the link to running your machine on the ShapeOko website. Instead of using a spindle, just replace all of it with a laser. We set our focus for the laser at about 3” for the shapeoko. This will be about at the bottom of the Z travel using our mount.

There are a lot of different “sender” programs that will work with your machine. In the inside of your machine is something called “GRBL” that directs how the machine moves and interprets the G Code file. The G Code file is the “instructions” and GRBL just needs some program to “send” it to the GRBL that is in the firmware of your control board.

Here is the wiki page with all of the sender program available:


We like to use the “universal G Code sender” as it is the standard for the previous shapeoko machine and has a very large user base. It is located here:

https://github.com/winder/Universal-G-Code-Sender

Here are some instructions on running the shapeoko3 with Universal G Code Sender:
The new shapeoko 3 is now out, so why don’t we put a laser on it! We have had great success with the original shapeoko 1 and 2 upgrades, so it is not much different. We have had the opportunity to upgrade a Shapeoko3 in the lab and it runs GREAT! It is a really stiff machine and the components are all built well and in the USA! Good job to the guys over at Carbide 3D on putting out another great Shapeoko kit. Now let’s get started!

We will walk you through the electronics hook up, the mechanical assembly, and the G Code needed to run. It should take only a few hours total from unboxing to running.

*note* You will need some minor soldering skills to complete the electrical portion of the instructions.

**Mechanical**

*note* We are upgrading the SMALL version of the shapeoko3 in these instructions. If you want to upgrade the larger versions then you will need to mount the driver on the Z axis differently as they moved the electronics enclosure to the side of the machine.

We have made a custom 3D printed mount for the shapeoko 3 so the laser can fit right next to the Spindle without needing to take the laser off whenever you want to carve. This is in ADDITION to the laser kit you ordered and is specific for the shapeoko 3 CNC. It includes:

1. **Power Extension Cord**: For extending the power to mount the driver on the electronics box.
2. **Laser Mount**: To put in the spindle holder for the laser.
3. **Laser Mount Screws/Nuts**: To mount the laser on the laser mount.
4. **Laser Driver Mount Screws/Nuts**: Screws to attach the driver to the electronics enclosure.
5. **Zip Ties**: To clean everything up for cabling.
6. **Zip Tie Holder**: To place on the Z axis gantry to hold the laser cable in place and out of the way.
Electrical Hook Up

The new board from Carbide is closed source, but they have provided some details on how to set it up. You just need to solder a couple of wires to the board, or put in a connector on the space for the header under the FAN connector on the right of the board. The left middle hole is the LASER "+" signal and the bottom right hole is the LASER "-" signal. Give yourself enough wire to go all the way to the laser driver, screw it in the terminals of H2, and you are good to go!

Here is the carbide board with the connections. Connect the laser driver input to the "Laser +" and "Laser -" signals on the Carbide board:
For newer Shapeoko 3 machines, Carbide is shipping new boards. The new PWM connection is:

**Laser Driver Signal Connections**

- **Laser Driver "+" connection**
- **Laser Driver "-" connection**

**New Carbide Motion Electronics Board**
**PWM Connection for Shapeoko3**

Here is the back of the laser driver. Connect the carbide board to the control signal input H2 screw terminal.

**Software – Controlling the Laser**
To run your shapeoko 3, use whatever software program you want to generate your G Code. You can use our [software on our page here](#) or any other G Code generating program. Your shapeoko 3 probably comes with 0.9 GRBL firmware installed. The G Code needed for the laser is the same as for the spindle:

- **M03** (turn laser ON)
- **M05** (turn laser OFF)

For power control, you will use the command, “M03 SXXXXX” where XXXXX minimum will be 1400 and maximum will be 12000.

- **M03 S12000** (turn on 100%)
- **M03 S6000** (turn laser on 50%)
- **M03 S1400** (turn laser on minimum)

You can also do basic power control by varying the feedrate with this method. When you are engraving, use the speed of the machine to determine how “dark” the engraving is. Example:

- Faster -> Lighter Engraving
- Slower -> Darker Engraving

You can use the [free inkscape tool](#) to make vector engravings or you can use Laser Etch to make raster engravings.

**Running your laser Shapeoko 3**

Your machine came with the Carbide control software to run you machine. Unfortunately, this produces a slight delay for the spindle on command of M03. They are expecting the large spindle to need to speed up before the machine moves. This is not the case for the laser, so it will leave marks at the beginning and ending of your lines.

There are a lot of different “sender” programs that will work with your machine. In the inside of your machine is something called “GRBL” that directs how the machine moves and interprets the G Code file. The G Code file is the “instructions” and GRBL just needs some program to “send” it to the GRBL that is in the firmware of your control board.

Here is the wiki page with all of the sender program available:


We like to use the “universal G Code sender” as it is the standard for the previous shapeoko machine and has a very large user base. It is located here:

Here are some instructions on running the shapeoko3 with Universal G Code Sender:

**ShopBot**

Shopbot Tools are known to be some of the best CNC tools for many years running. They have a great following from users and many over the years have asked us how to upgrade them. We talked to the folks over at Shopbot about integrating into their control electronics and it is pretty straightforward. So let’s get started!

There are many types of ShopBot CNC machines. One of the most popular is the Deskop Max machine. While there are several machines with different mechanical configurations, they all have the same (or similar) control system on them.
Electrical

Every ShopBot has a control box. In this box, there is an interface board called the Control Card. This has the connections for the motors and various other outputs for the machine control. If you open up your control box it will look something like this:
In the case of the ShopBot Desktop Max, we took the blue cover off the box and opened it up. The control card looks like this:

![Control Card Outputs](image)

There will be a section on the control card for general purpose outputs. These are labeled:

- GND
- OP5
- OP6
- OP7
- OP8

In this case, they have a Wago connector that has a push clamp device to let you attach tin wire directly to the outputs. Since none of them are being used on this machine already, it is easy to use the OP5 connection and the GND connection. (*Note* if the OP5 connection does not produce a signal, try the OP6, OP7, or OP8. We have seen some boards have faulty outputs).
Connect the Red + wire to the OP5 connector and connect the Black – wire to the GND terminal. Like we said earlier, the control card might look a bit different, but the OP5 to OP8 connections will be in every machine.

Connect this Molex Mini fit cable to the H4 control signal input on the back of the laser driver. You can also use your own wire as well. In this case, connect it to the screw terminal H2.

**Software**

The general purpose outputs on the ShopBot are controlled with the following syntax:

SO,[Output#].1 for ON  
SO,[Output#].0 for OFF

Where OUTPUT# is the number of the output. In our example machine we used OP5. It would have the following G Code for control:
SO,5,1 (ON)
SO,5,0 (OFF)

We also have the Vectric Post Processor written for OP5 located on the post processor page. It has how to set up the tool and run the file. Check it out here:

Mechanical

There are many machines with different mechanical configurations for the ShopBot tools. Many people make a plate that has the laser on the bottom and the driver above it that can be removed easily when not being used. Since the electrical connections are all quick release, this process should only take a couple of minutes to complete the change over.

On the ShopBot Desktop max, there are a couple of screw holes in the front of the tool holder. The customer utilized these holes and made a plate with the laser and driver on it.

Solidoodle

Laser Upgrade Implementation

Solidoodle

First things first, you need to figure out what you have on your particular machine. Solidoodles have been shipping for awhile, and they are always changing and improving them. So, take a look and follow the instructions for whatever hardware you have.

Solidoodle 4, Workbench, & WB Apprentice:

Upgrading the new solidoodle machines is super easy! We are going to take the fan that is connected to the board and replace it with the laser connection. The fan is located here:
When you want to use the laser, replace the connector for the fan for another molex style connector that attaches to the laser driver board control input screw terminal H2. Then use the M106 Sxxx command for “laser on” and M107 command for “laser off”.

You might consider putting in a SPDT switch instead of swapping the cables. Just cut the fan positive wire and put the part coming from the PrintRBoard in the center, the one going to the fan on one side and a new wire going to the laser driver on the other. Just also connect the ground to the laser driver and you are set! Then you can have a nifty switch to go between 3D printing and laser cutting!

Software for using your new laser upgrade is located here in the software section:

It is pretty straight forward. The Solidoodle printers use a version of the PrintRboard from the RepRap project. It has an output for a fan, which Solidoodle uses on top of their extruder. When you want to use the laser, replace the connector for the fan for another molex style connector that attaches to the laser driver board control input. Then use the M106 command for “laser on” and M107 command for “laser off”. You might consider putting in a SPDT switch instead of swapping the cables. Just cut the fan positive wire and put the part coming from the PrintRBoard in the center, the one going to the fan on one side and a new wire going to the laser driver on the other. Just also connect the ground to the laser driver and you are set! Then you can have a nifty switch to go between 3D printing and laser cutting!

Of course, you will need to print out a mechanical mount for the laser as well. The dimensions and solid models are located in the documents section of the website. If you remove your extruder and only use it as a laser, then the mounting holes for the extruder are matched exactly to the mounting holes on the diode. You will need to add about 1” of some material to the right side of the laser so the head will trip the limit switch correctly.

Here is the graphical version of the instructions:

If you look at your machine and it does not have a PrintRBoard, then you probably have one shipped with a sanguinololu board. It still can be upgraded, but is a bit more of an effort. Your board probably looks like this:
The board has some extra stuff in it like analog outputs and PWM. The IO header has a PWM output as well as 5 analog outputs you can use. A Solidoodle 2 owner, Ian Johnson, has made a pretty good write-up on how to add a controllable fan to the sanguinololu board here:

http://solidoodletips.wordpress.com/2012/10/26/gcode-controlled-extruder-fan/

You won't need to bother with the mosfet driver, because the input for the laser driver is a digital isolator that only needs a couple of milliamps to turn on. So, all you need to do is connect up the IO header to the input for the laser driver.

The cool part is that you are using the PWM output from the board. This means you have power control! Adjust the laser driver potentiometer to have the maximum power level you want, say 1amp. Then, use the speed command as part of your G Code. The laser power can then be adjusted on the fly from 0 to 255 levels. For all on, use M106 s255. To turn it on half power it is M106 s127. To turn off the laser, use M107. Just modify your G Code file for your part to put in the levels you want. Repetier host also has a button and a slider to control the laser (it will be marked “fan”).

Ian mentions that you might need to modify the firmware as well. He writes from his post:

“The fan may not already be activated in firmware, so follow the directions here for how to download and update the firmware. When you have the firmware open in Arduino, change to the Pins.H tab. This will probably be too far down to appear at the top of the screen, but on the right side there will be an arrow that pulls down the full list. Scroll about halfway down, looking for the section for Sangiunololu. In that section look for

#define FAN_PIN -1

and change the -1 to 4.”
Stepcraft CNC

Stepcraft CNCs are very versatile tools that have a lot of additional toolheads to be able to do a lot of different things. The only thing missing is the laser! While we understand they are developing one themselves, we will show you here how to upgrade you machine using our lasers and electronics. It is pretty straight forward, so let’s get started!

**Mechanical**

We have made a spindle mount laser holder for the 43mm spindle bracket on the StepCraft machine.

**Electrical**

There are two ways to connect your laser to the stepcraft machine:

1. **ON / OFF control** – This will be using the spindle relay to turn on and off the laser. It is the easiest way to get started.
2. **PWM control** – This will be using a different pin to produce a PWM signal for power control.

**ON / OFF Control Setup:**

Now that you have it all mechanically mounted, let’s look how to set it up to fire using your stepcraft controller. On the machine there is a DB 15 connector. On this connector there is a relay that you can control via software. The 15 pin connector is the one on the right.
From the instruction manual:
To connect the laser you will need to use the following on the DB15 connector:

- **PIN 13** = LASER + (H2 + on Driver)
- **PIN 2** = LASER – (H2 – on Driver)

**PWM Power Control Setup**

If you want to have power control, then instead of using pin 13 in the previous section for the laser “+” signal, use pin “7” instead.

To connect the laser you will need to use the following on the DB15 connector:

- **PIN 7** = LASER + (H2 + on Driver)
- **PIN 2** = LASER – (H2 – on Driver)

We will show how to set this up in the next section.

**Software**

**Mach3 Setup**

Now that you have it all hooked up, you will need to get it configured in Mach3 or WinPC-NC. Basically, you will need to set up the RELAY control from pin 13 and map it to an output. In mach 3, it is easy to map it to output #1. Once you do this, you can set it up in the Spindle setup tab. Make sure the Relay Control “disable spindle relays” box is unchecked. M3 map to output #1.
If you want to use PWM control, follow the instructions on the mach 3 page under the section “using spindle PWM to control the laser”.

Then, to turn on the laser you will use:

Laser ON command: M3
Laser OFF command: M5

**UCCNC Setup**

Many people are using the UC100 and the UCCNC software to run their machine. Here is the quick guide to setting up the machine.

You will need to go into the “configuration” -> “axis setup” -> “spindle” tab. This is the screen that you will use to set the laser up as a “spindle”.
In the “Configuration -> I/O Setup” page, make sure to set the “Laser Pin” like below:

In the previous section, you either set your laser driver to be connected to Pin 13 (Relay) or Pin 7 (PWM). When setting up the UCCNC controller there will be a “mapping” between the pins in the software and the pins in real life on the DB15 connector.

UCCNC PIN 1 = STEPCRAFT DB15 PIN 13
For ON/OFF relay control:

- Uncheck the box on “PWM spindle” and check the “Spindle relay output enabled”.
- Set M3 relay pin to “1” and port to “1”. This will map to the stepcraft Pin 13 that you have connected to the laser driver.

You can now turn on and off your laser with the “M3” command. Test it on the run screen by pushing the clockwise spindle button.

For PWM control:

- Check both the boxes for “PWM Spindle” and “Spindle relay output enabled”.
- Set M3 relay pin to “1” and port to “1” in the spindle relay section.
- Set PWM pin to 17.
- Set your Frequency to a number you want. 100 or 500 is pretty good.
- Set minimum duty % to 0.
- Set maximum duty % to 100.

Now you will have control over the power using the command:

M3 SXXX

where the “XXX” is a number between 0 and 100.

Running your Laser Stepcraft Engraving Picture

Ok, so this is probably the coolest thing you can do with your stepcraft laser – engrave images! Jim Arft, a customer of ours found this new awesome feature. We will walk through how to set up your machine here.

You are going to have to set up a “plugin” to be able to engrave pictures. You can find it on the “Configuration -> General Settings ->” tab. On the bottom left click on “configure plugins”.
A page will pop up. Click on the one labeled “Laserengrave”. Click on “Enabled” to enable the plugin.

You can click on the show button to show the plugin. You can then load an image into the plugin that you want to engrave. You have three options:

1. Greyscale – This will make a 256 level image of the photo with the laser by varying the intensity.
2. Halftone – This will make a “dithered” engraving of the photo with the use of dots.
3. Black and white – This is used for making just black and white engravings. This is best for logos and images that have no greyscale levels.
After you have all of the settings on the right figured out as well (Feedrate, etc…) then press the “create code and send to UCCNC”. This will start the progress on the engraving.

**Vector Engraving**

You can generate your G Code the same as you would for your spindle, but you now are using it in 2D and not carving in 3D.

Just make sure you wear your safety goggles when running your machine and that everyone in the area has laser protection gear.
TinyG

The TinyG project is an open source CNC control system for embedded CNC motion control. Many people build CNC machines with this controller. It is very similar to the GRBL controllers, but just with different firmware. For more information check out the TinyG wiki here: TinyG Wiki

We have many customers using the TinyG with our lasers with success, so we now have decided to put the instructions up for everyone to use. It is pretty simple to get up and running and only needs a small configuration for the PWM function. In the end, you will have laser connected with PWM power control enabled. Let’s get started!

**Electrical Connection**

On the side of the board with the three screw terminal connectors there is a signal called “PWM”. It is on the screw connector closest to the reset button. The “PWM” terminal will be the laser driver “+” signal. On the same screw terminal, there is a “GND” connection. This will be the laser driver “-” signal.

Here is a diagram showing these connections:
**PWM Configuration**

Now that you have the electrical connections completed, you need to configure the PWM function of the TinyG.

From the TinyG wiki:

*There is currently only one PWM channel (p1), but the configs are structured for multiple PWM groups. The PWM channel is set up to act as a remote control Electronic Speed Controller (ESC), but can be used for other PWM functions using these settings.*
<table>
<thead>
<tr>
<th>SETTING</th>
<th>DESCRIPTION</th>
<th>NOTES</th>
</tr>
</thead>
<tbody>
<tr>
<td>$p1frq</td>
<td>Frequency</td>
<td>in Hz, e.g. 1000</td>
</tr>
<tr>
<td>$p1csl</td>
<td>Clockwise speed low</td>
<td>In RPM – arbitrary units unless you calibrate it, e.g. 1000</td>
</tr>
<tr>
<td>$p1csh</td>
<td>Clockwise speed high</td>
<td>In RPM</td>
</tr>
<tr>
<td>$p1cpl</td>
<td>Clockwise phase low</td>
<td>0.000 to 1.000, e.g. 0.125 for 12.5% phase angle</td>
</tr>
<tr>
<td>$p1cph</td>
<td>Clockwise phase high</td>
<td>0.000 to 1.000</td>
</tr>
<tr>
<td>$p1wsl</td>
<td>Counter clockwise speed low</td>
<td>In RPM</td>
</tr>
<tr>
<td>$p1wsh</td>
<td>Counter clockwise speed high</td>
<td>In RPM</td>
</tr>
<tr>
<td>$p1wpl</td>
<td>Counter clockwise phase low</td>
<td>0.000 to 1.000</td>
</tr>
<tr>
<td>$p1wph</td>
<td>Counter clockwise phase high</td>
<td>0.000 to 1.000</td>
</tr>
<tr>
<td>$p1pof</td>
<td>Phase off</td>
<td>0.000 to 1.000 used to set OFF phase for PWM devices that are not off at 0 phase</td>
</tr>
</tbody>
</table>

Ok, this all seems a bit complicated, but you don’t need to worry about most of these settings. We will need to now configure your TinyG board to the correct settings to run the laser.

Here are the settings you need to enter:
Follow these steps to enter the codes to your TinyG:

1. Connect to the TinyG using the control panel software.
2. Click the Gcode tab (upper left)
3. Enter PWM settings line by line in command line prompt on bottom of screen
4. Confirm settings after each line, they will be echoed above

Software Commands

Your commands for running the laser will be the same as the spindle on command with RPM. It is:

- **M03 SXXX**
  Where the “XXX” is a number between 0 and 255. So, some examples are:
  
  - Laser ON 100%: M03 S255
  - Laser ON 50%: M03 S127
  - Laser ON 10%: M03 S25
  - Laser OFF: M03 S0 (or M05 will work as well…)
Running the TinyG Laser

Most people use chilipepper to run the TinyG controller. There is also a version for running a laser. You can find it here:

http://chilipeppr.com/tinyglaser

LaserWeb is also a program that will run with the TinyG controller. You can learn about it here:

http://cncpro.co/

Ultimaker 1 & 2

Laser Upgrade Implementation

Ultimaker is a very popular open source printer, with a large print platform and great accuracy, it makes for an even better laser engraver! These are the details on how to upgrade the Ultimaker printer to become a cool laser engraver with any of the laser upgrade kits.

This is the basic integration of the electronics to get the laser driver controlled by the Ultimaker with commands in G Code. Of course, you will need to print out a mechanical mount for the laser as well. The Ultimaker 1 we are looking at to upgrade has the V1.5.7 version of the electronics board located here: http://reprap.org/wiki/Ultimaker%27s_v1.5.7_PCB using the Marlin version of the firmware located here: https://github.com/Ultimaker/Ultimaker2Marlin/tree/master/Marlin

The Ultimaker 2 has a version 2.1.1 of the electronics board and the files are available at GitHub via open source here: Main%20Board%20V2.1.1.pdf It will also use the Marlin version of the firmware from above.

So, let’s get started! There are two different ways to hook up the laser driver to the Ultimaker board. One uses the PWM output for the Fan and the other uses another PWM output for an LED. Either one will work, it just depends on which way you want to go.

Upgrading Ultimaker 1

Method 1: Using the PWM output for the FAN for Ultimaker 1

If you are not already using this for an additional fan on your Ultimaker, then this might be the easiest way to go for you. The fan control on the Ultimaker Board is pin 7. On the board it is a standard Molex connector on the top middle of the board labeled “PWM”. If you don’t have a fan attached, simply
connect the “+” to the laser input “+” and the “-” to the laser input “-”. You then control it in G Code with M106 SXXX command where XXX is a value between 0 and 255. Then use M107 to turn the laser off. Here are pictures of the board showing the FAN option location.
Method 2: Using the PWM output for the LED for Ultimaker 1

Ultimaker added a cool upgrade to be able to put an LED in the printer to be able to see better and just for a cool factor. The neat thing is that they also have a provision to control this output as well.

Looking at the board, it is in EXP3, pin number 13. You can solder a wire to this pin for the “+” signal and then solder a wire to the “GND” pin next to it for the “-” signal for the laser control. You then control it in G Code with M42 P13 SXXX where XXX is a value between 0 and 255. Use M42 P13 S0 to turn the laser off. If you want to have the laser all the way on, then it is “M42 P13 S255”. For 50% power it is “M42 P13 S127” and so on.

The pictures above also show the placement of P13 on the left side of the board. Here are the details of the firmware showing this implementation:

```c
break;
case 42: //M42 - Change pin status via gcode
        if (code_seen('S')) {
            int pin_status = code_value();
            int pin_number = LED_PIN;
            if (code_seen('P') & pin_status >= 0 & pin_status <= 255) {
                pin_number = code_value();
                for(int8_t i = 0; i < (int8_t)sizeof(sensitive_pins); i++) {
                    if (sensitive_pins[i] == pin_number) {
                        pin_number = -1;
                        break;
                    }
                }
            } #if defined(FAN_PIN) & FAN_PIN > -1
                if (pin_number == FAN_PIN)
                    fanSpeed = pin_status;
            #endif
            if (pin_number > -1) {
                pinMode(pin_number, OUTPUT);
                digitalWrite(pin_number, pin_status);
                analogWrite(pin_number, pin_status);
            }
        }
break;
```

Electrical
You again have the same choice of which way you want to upgrade, either using the fan or the led output on the board. The new 2.1.1 main Ultimaker board is even EASIER than the previous version. Now they put in two nice Molex connectors so you don’t need to do any messy soldering to get to your signals!

To control the laser it is the same as the Ultimaker 1 version.

**Using Fan J14:** Control it in G Code with M106 SXXX command where XXX is a value between 0 and 255. Then use M107 to turn the laser off.

**Using LED J15:** Control it in G Code with M42 P13 SXXX where XXX is a value between 0 and 255. Use M42 P13 S0 to turn the laser off.
When using some versions of firmware, there will be a “kickstart” for the fan that creates a burst of power in the beginning. This will cause the laser to make a spike in the beginning of the run and leave a dot in the material. To remove this feature, you need to do a quick modification of the firmware.

From the Ultimaker users forum:

*In Configuration_adv.h look for this:*

```c
// When first starting the main fan, run it at full speed for the
// given number of milliseconds. This gets the fan spinning reliably
// before setting a PWM value. (Does not work with software PWM for fan on Sanguinololu)
#define FAN_KICKSTART_TIME 200
#define FAN_KICKSTART_MINPWM 20
```

Now, just set the time to 0 or minpwm to 255. Change it to this:

```c
// When first starting the main fan, run it at full speed for the
// given number of milliseconds. This gets the fan spinning reliably
// before setting a PWM value. (Does not work with software PWM for fan on Sanguinololu)
#define FAN_KICKSTART_TIME 0
#define FAN_KICKSTART_MINPWM 255
```

This will remove the “flash” the laser will produce when using the fan output and everything will work correctly.

**Velleman K8200**

**Laser Upgrade Implementation**

The K8200 printer uses a variant of the RepRap project for the control board and the software and firmware from Repetier. The hardware includes a fan and control for the fan in the software. The laser driver board will use this fan output from the control board and it’s input. When you want to use the laser, replace the connector for the fan for another molex style connector that attaches to the laser driver board control input. Then use the M106 command for “laser on” and M107 command for “laser off”.
You might consider putting in a SPDT switch instead of swapping the cables. Just cut the fan positive wire and put the part coming from the PrintRBoard in the center, the one going to the fan on one side and a new wire going to the laser driver on the other. Just also connect the ground to the laser driver and you are set! Then you can have a nifty switch to go between 3D printing and laser cutting!

Of course, you will need to print out a mechanical mount for the laser as well.

Here is the graphical version of the instructions:

Here is a picture of the machine:
Here is a link to some instructables showing more detail of an installation:


**X-Carve**

The X-Carve is the new upgrade for the very successful Shapeoko product by the Inventables company. Keeping up with their success with the Shapeoko, they have made some improvements to the entire system, making this a very powerful CNC for hobbyists and industry alike.

There are now three machine configurations that Inventables offers, the 1000mm large version, 750 medium version, and the 500mm version. We opted for the 1000mm kit and have made some cool parts to make the upgrade seamless. We did ours in under 1 hour and were laser engraving test pieces that turned out perfect! We have instructions here on how to complete the full build and are also offering the

*Update 10-5-2015* New X Carve Model Changes for Laser

Inventables have started shipping the new X Carve to customers. The instructions here have been updated to reflect changes with the new machine. The only difference is the electrical portion with the addition of the X Controller and the lack of the terminal block in the back of the X Gantry. We have modified our X Carve mounting kit with a longer 11’ cable for the laser control.

Mike Merzke at Merzke Custom Woodworking has made a great video showing his installation of the laser kit to his X Carve. View it here to see how easy it is to upgrade.

Now that you have seen how easy it is, let’s get started!

**Mechanical**

We have plenty of 3D printers in the lab cranking out parts 24/7, so we decided to design a quick mount for the laser utilizing them. The mount can either be placed in the hole where the router sits, or it can be placed on the side of the router so everything can stay on the machine. It is up to you which way to go.
The mount can be used either way, just turn it over. If you want to remove your router you will get a little extra eye protection as the holder will block some of the laser light.

We decided to go with the “keep the router on” way of mounting it next to the router. We will show you pictures of how to put it together in the build section of the instructions.

**Electrical**

There are now two versions of the X Carve.

1. **Original X Carve Machines with the Arduino Uno** – Shipped Pre-October 2016
2. **New X Carve Machine with X Controllers** – Shipped after October 2016

We will cover the electrical instructions for both here in this section.

**Original X Carve with Arduino Uno**

The X Carve still uses the Arduino UNO and the G Shield electronics like on the Shapeoko 2. The Arduino that the Inventables ships comes with a pre-installed version of GRBL 0.9j, which will work fine for most applications using the laser. We will need to hook up the output from the Arduino G Shield to the laser driver to provide the signal to turn it on and off. You will need to connect the following:

- Pin 11 Labeled “11” on either Arduino or GShield to Laser Driver Input “+”
- Pin 14 Labeled “GND” on either Arduino or GShield to Laser Driver Input “-”

The X Carve already has these cable connected for the spindle control for the 24V spindle. The wire for the Laser “+” is YELLOW and the Laser “-” is the BLACK wire.
Laser Commands

Using the pre-loaded GRBL from Inventables

If you bought the X Carve with the electronics included (like most people do), then they shipped an Arduino UNO with GRBL installed already. You either have GRBL version 0.9i or 0.9j which Inventables branched from the main GRBL github development site. Both of these will work fine for most applications, but will leave a slight delay when trying to engrave pictures with PicLaser software. If you just want to cut things and do black and white engraving, then you will have no problems.

To turn on and off the laser the command is:

- **Laser ON:** M03
- **Laser OFF:** M05

If you want to use power control for the laser (via Pulse Width Modulation) the laser command is:

```
M03 SXXXXX
```

Where the “XXXXX” is a number between 0 and 10000. So, some examples are:

- **Laser ON 100%:** M03 S10000
- **Laser ON 50%:** M03 S5000
- **Laser ON 10%:** M03 S1000
- **Laser OFF:** M03 S0 (or M05 will work as well…)

Running The Laser X Carve
We recommend not using Easel to control your X Carve when using the laser. If you are using the Inventables firmware in your machine (the firmware that came with your X Carve) then you can use Easel when routing.

There are a lot of different “sender” programs that will work with your machine. In the inside of your machine is something called “GRBL” that directs how the machine moves and interprets the G Code file. The G Code file is the “instructions” and GRBL just needs some program to “send” it to the GRBL that is in the firmware of your control board.

Here is the wiki page with all of the sender program available:


We like to use the “universal G Code sender” as it is the standard for the previous shapeoko machine and has a very large user base. It is located here:

https://github.com/winder/Universal-G-Code-Sender

If you are planning on doing pictures, then PicSender will work great to control the machine. You can get it here:  PICSENDER PROGRAM