

Using a Simple Laser with your Wham Bam MUTANT

by Andrew McClary

Using a Wham Bam MUTANT tool changer opens up a whole new world for 3D printers. One of those worlds is the ability to quickly swap out your hotend for a diode laser to do simple engraving and cutting projects. Setting up a laser on a MUTANT is a pretty easy procedure. Mounting one on a MUTANT is also much safer than doing so with magnets.

There are a number of new diode lasers out there on the market. In this tutorial we are going to look at using a simple inexpensive Blue Violet laser engraver heads which are easily available both from Creality, Amazon, and many other sources. We are using a simple 12/24V 500mw laser that is suitable for doing simple engraving on a number of materials such as wood, leather, acrylic, paper, and other materials. It can easily cut a number of craft materials like EVA foam and construction paper. It is also very easy to hook up to most 3D printers without having to do any modifications if you have a MUTANT.

There are a number of other higher powered diode laser that are out there on the market that can be mounted easily to your printer with a MUTANT. Some of these lasers are powerful enough to cut through up to 1/2" of wood and cut and engrave a much wider variety of materials. In our lab we have been working with the NeJe A40640 dual beam laser which is extremely powerful for a diode laser. Installation of this laser is a little more advanced and can be approached in another tutorial. If your interested in seeing that tutorial, leave your comments in the comment area. What I can say is I was delightfully surprised at how incredibly useful even a simple low power laser was to add to my toolbox of tools.

Laser Safety

Lasers, even these inexpensive low power ones are extremely dangerous to your eyes. DO NOT TURN ONE ON WITHOUT WEARING APPROPRIATE PROTECTIVE GLASSES. I can't really stress this enough. All it takes is a moment of the laser reflecting off something into your eye to do permanent damage and cause blindness. While the beam may not be strong enough to burn at a longer distance, it is strong enough to damage the nerves and lenses in your eyes. Sometimes this damage is not instant but can manifest itself many years later as cataracts. While diode lasers are much safer than gas lasers, they still can pose a danger to our eyes. You must also be aware of pets and children in the area who may inadvertently look at the laser. Be safe with your eyes.

Using a laser actually means you are burning material. While we are using a pretty low power laser in this tutorial, it can still start a fire. You should keep a fire extinguisher nearby. When using your laser you always want to be present and keeping an eye on it when it is in operation. This is especially true when working with any new material that you haven't tested before.

Fumes are also another thing that you want to consider. If you don't have some kind of vacuum system to exhaust the fumes, then lasering should be done in a well ventilated area. Some materials shouldn't be lasered at all because of the highly toxic fumes they give off. This includes PVC, Lexan, ABS, HDPE, and Fiberglass.

Step 1. Installing the laser on your MUTANT mobile plate.

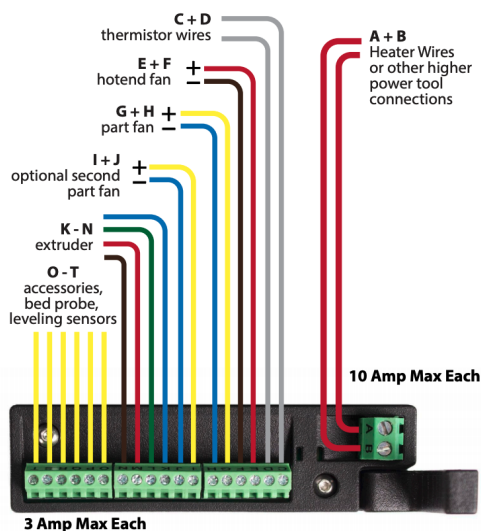
The laser mounts to the MUTANT mobile plate via two 4mm bolts from the back. Line up the laser so it is straight and as far down as you can go on the plate. The creality laser has a lens that can be turned to focus the beam. If your laser has a fixed focus, some are 30mm, you may want to adjust it up or down in the slots to get it as close to that as possible.



The

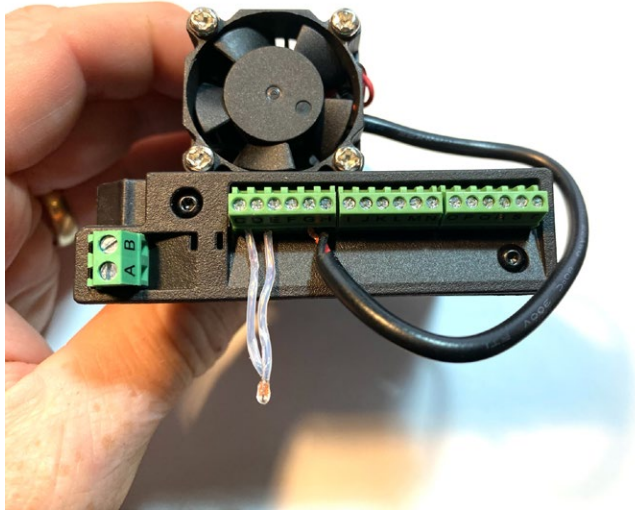
power for this low power laser can be connected directly to the connections used by the parts cooling fan. Be sure to check that you have either 12V or 24V available at this connection when the parts cooling fan is turned on.

Identify and confirm which connections you are using for your parts cooling fan. If you are using the wiring standards suggested by Wham Bam then your wiring looks like the following and you would want to use terminals G and H. Be very careful to confirm the polarity.



WHAM BAM Systems Wiring Layout Best Practice

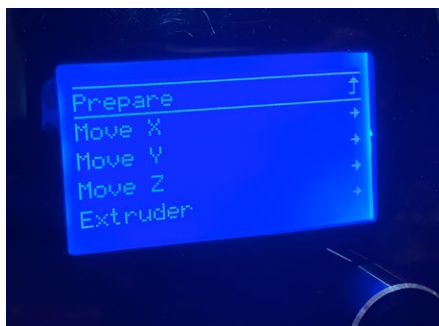
On many 3D printers, the firmware will throw an error if the thermistor is missing. The easiest way to solve this is to wire in an extra thermistor or a resistor that matches your thermistors resistance at 25C. For most printer hotends a 100K resistor could be used. (Check the 25C resistance of the thermistor used in your printer.) In our case we buy packs of replacement thermistors and have lots on hand. You want to wire this thermistor into the terminals that are used by the hotend thermistor. If you are using our standards, this is C and D.



Step 2 - Setting up the laser and focus

Now that you have your laser mounted on a MUTANT mobile plate it is really easy to swap out your hotend and put on the laser. If you have one of our magnetic Flexi Plate systems, its also a good idea to flip the plate over so that bare metal is facing up. This way the laser won't hurt your build surface. The metal will protect your build surface, but you don't want to use the laser on the flat metal because it will reflect the laser back up and into itself potentially causing it to burn out. It can also reflect into someones eyes. So you want to put what we call in the industry a "spoil board" on the build plate. This is a surface we can burn into and not worry about and easily replace. I bought a quarter inch thick, one foot square piece of MDF from the hardware store for this purpose.

Now its time to test out the laser and to focus it. To do this go into the menu on your printer, home the printer, then move the X to 100 and the Y to 100 and Z to 20. For our example we are using a CR-10V2.



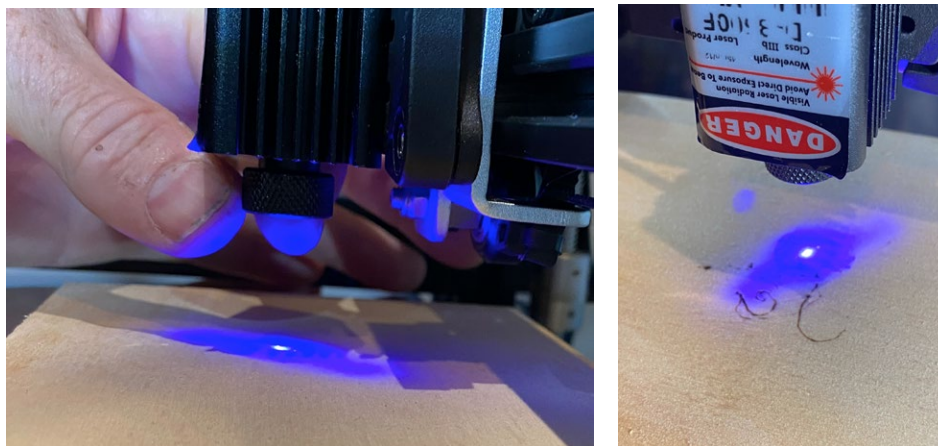
Once you have moved your laser into position. Put your spoil board on your build surface and secure it with some clips.

Now it is time to PUT ON YOUR LASER SAFETY GLASSES before continuing. Keep them on for all of these next steps.

The laser's power is controlled by the parts fan cooling setting. So to turn on the laser for testing we are going to go into the menu and turn on the cooling fan. We are going to set it to 100%.



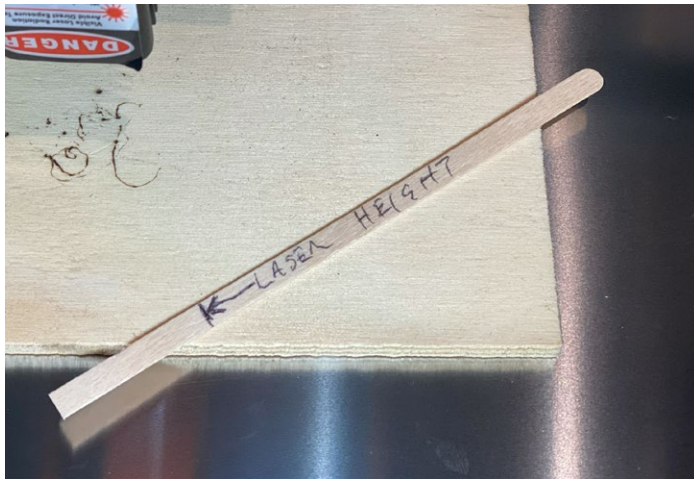
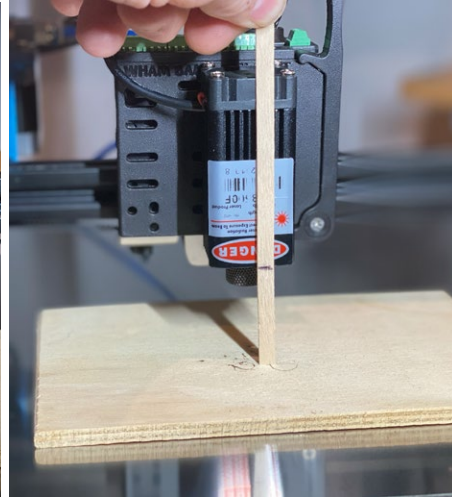
When you turn on the laser, the beam will probably make a small oval or rice shaped blue dot on the board. Now you are going to use your fingers to carefully turn the focus adjustment ring until the laser point is a tiny round dot and you start to see smoke rising off the wood. The wood will turn to black carbon at that point and then smoke will stop rising. Move the wood a bit and it should burn another small dot. If it doesn't, adjust the beam till you get the dot as small as possible and moving the wood very slowly will burn a line as you move the wood.



Once you have it set to a nice point that will burn. Turn off the laser in the menu by setting the cooling fan back to 0. **Only when this is done is it safe to take off the laser safety goggles.**



Now we are going to measure this distance from the wood to the edge of the laser housing. Write this measurement down. This is the measurement we are going to need to set the height of the laser as we put other items under the laser to be etched. In my example its 33mm. Another easy way is to create a measuring stick with a mark on it. You can easily use this to quickly set your laser's height.



3 - Setting up software

Now that our laser is all setup, we need to install and setup some software to control our laser to make cool things. This software works very similar to your slicer software except it is 2D and outputs g-code to move the laser and turn it on and off at different settings.

For this tutorial we are going to use a free open source software called Inkscape and a free extension created by J Tech Photonics. Using this combination gives you basic vector etching and cutting capabilities. For more advanced features I would highly recommend LightBurn, which is a commercial software that I can cover in another tutorial.

Start by installing version 1.1 of Inkscape. You can download versions for both the Mac or for PC from this link. <https://inkscape.org/release/inkscape-1.1/>

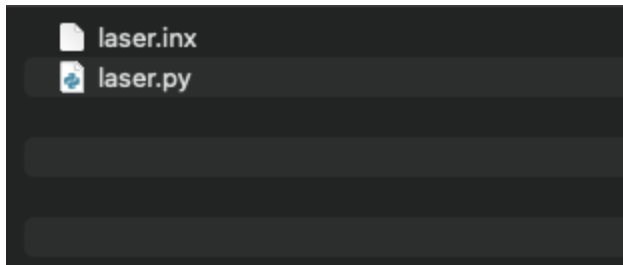
Now its time to install Inkscape on your computer. This part is pretty straight forward.

The next step is to download the Inkscape extension from J Tech Photonics. You want to make sure you have the correct one for the version of Inkscape you are using. If you are using version 1.1 like we are doing in the tutorial use Beta for Inkscape Version 1.0. You can download the software from this page https://jtechphotonics.com/?page_id=1980

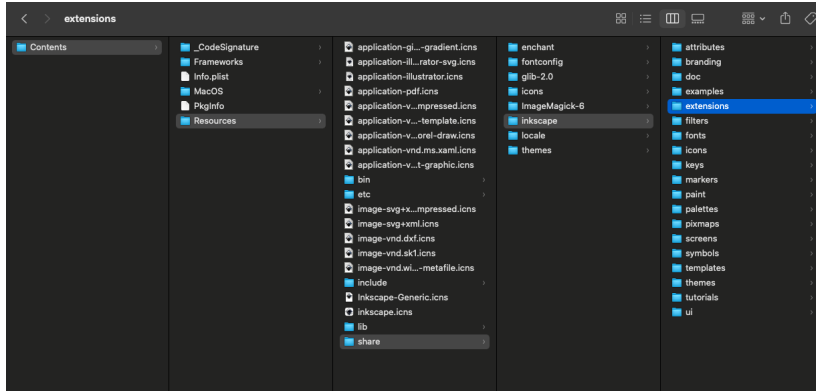
The install of extensions in Inkscape is a bit different depending on if you are on a Mac or PC and isn't as easy as using a prepacked installation wizard.

Here is how to do in on a Mac:

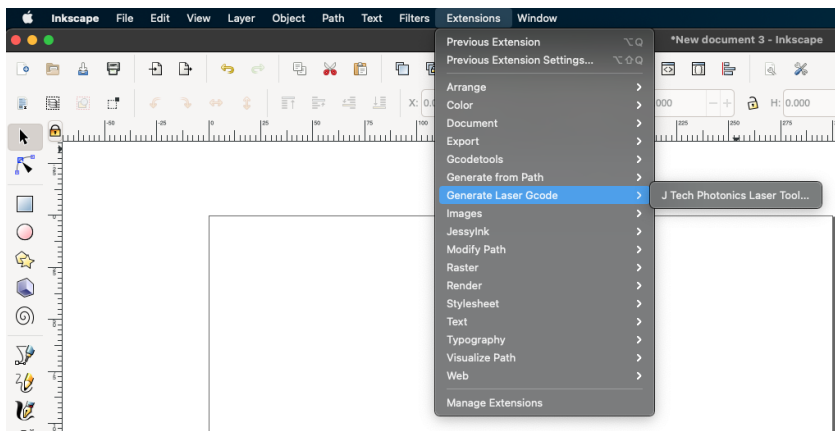
1. Start by downloading and extracting the files from the zip file. There should be 2 files - laser.inx and laser.py. Select them and hit copy.



2. Using Finder, go to your Applications directory on your computer and find Inkscape.app. Hold down <Control> key and select the app. A menu should pop out and you want to choose "Show Package Contents". Now navigate in Finder to "Contents/Resources/share/inkscape/extensions" and then paste the two files into the extensions directory.



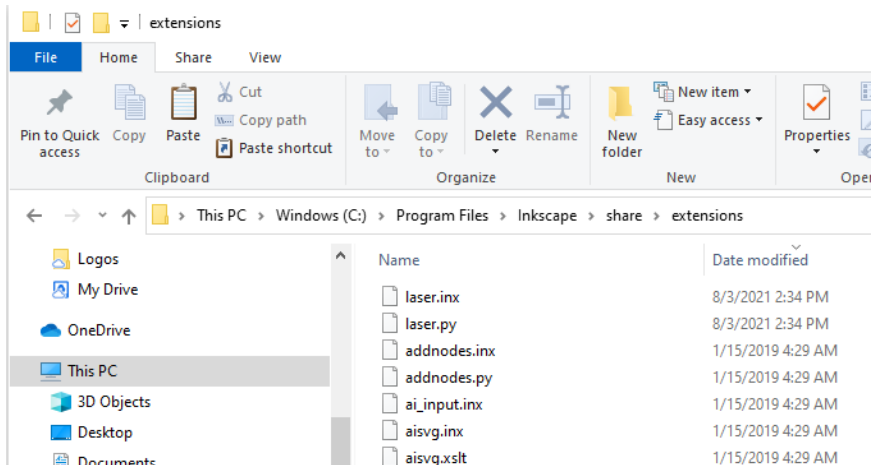
3. Open Inkscape and look for Extensions - Generate G Code of Laser - J Tech Photonics Laser Tool.



The extension is loaded and you are ready for the next step.

Here is how to do in on a PC:

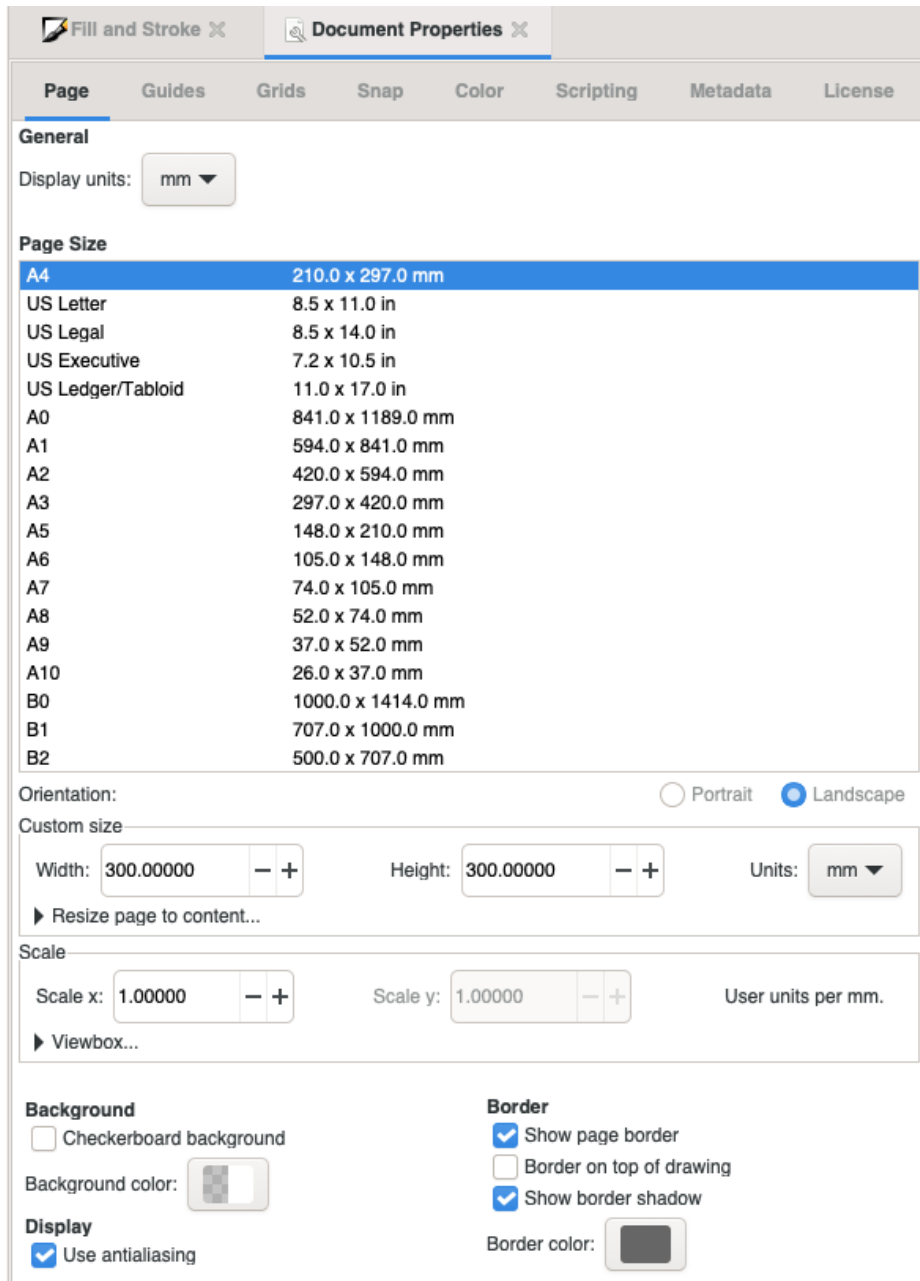
1. Start by downloading and extracting the files from the zip file. There should be 2 files - laser.inx and laser.py. Select them and hit copy.
2. Now go open the File Explorer and go to C:/Program Files/inkscape/share/extensions and paste the two files in to that directory.



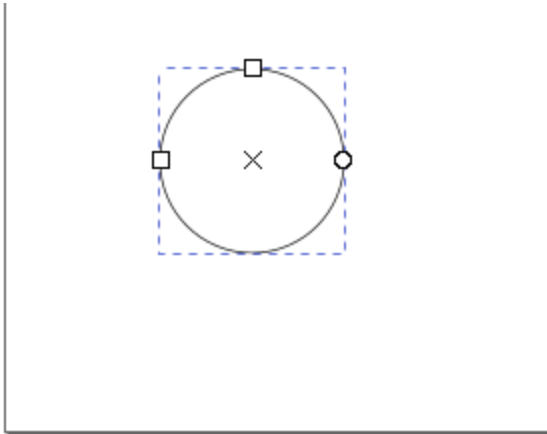
3. Open Inkscape and look for Extensions - Generate G Code of Laser - J Tech Photonics Laser Tool.

Now that we have the plug in loaded we want to setup your Inkscape document and the plug in.

When you open up Inkscape choose New Document. It is going to show you the document properties. Start by setting the Display units to mm. You are going to want to use Custom Size to set the document size to the size of your build plate. In our case I am setting it to 300mm x 300mm.



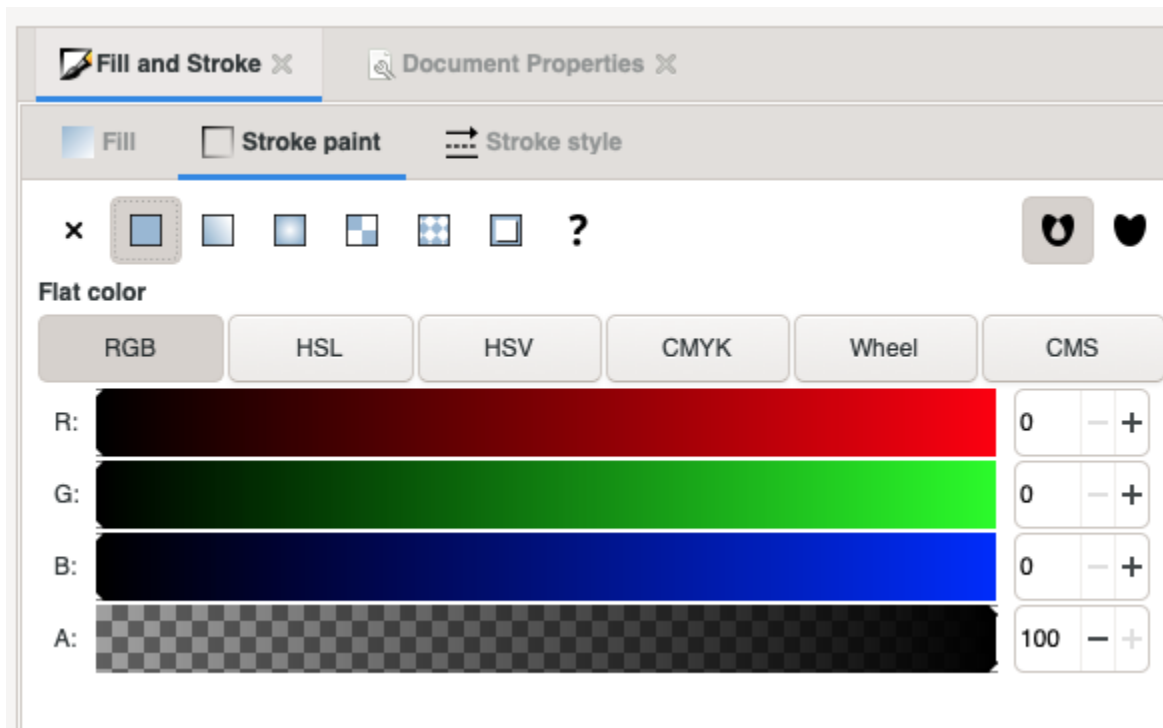
You can now use the drawing tools to create a number of vectors and text for your design. I am going to just do a simple circle for this tutorial as a test. Create a circle like this.



With the circle selected go up to the top and find Path ->Stroke to Path.



Now go to the right and choose the tab Fill and Stroke. Select the Fill sub tab and select X. Then click the sub tab Stroke paint and select the box for a solid. Then make your color black.



Click on the sub tab Stroke style and make the width .2mm. This really doesn't matter but you don't want it zero or your path won't show up.

Now open up the J Tech Photonics Laser Tool from the Extensions menu at the top.

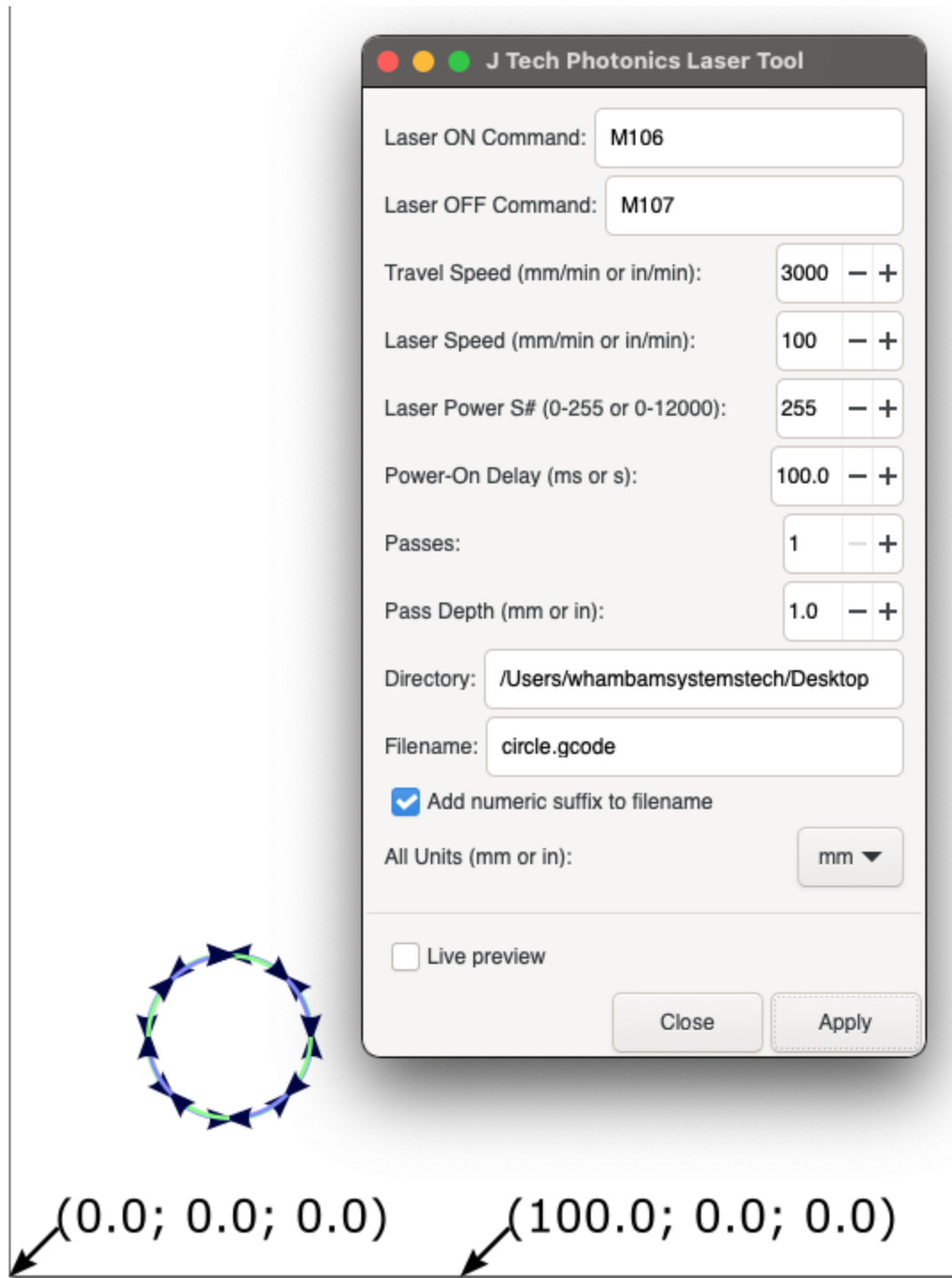
The first thing we are going to want to set is the Laser ON Command and the Laser OFF Command. Since we are using the cooling fan output to control our laser we are going to want to put in M106 for on, and M107 for off which are the g code commands for turning the fan on and off.

Your travel speed can be left at 3000 mm/m. If you find this to fast you can change it. The Laser speed though is very important and will change depending on the laser and material you are using. The stronger the laser or easier the material to engrave or cut the higher this number can be. We are going to start with setting this to 100mm/m for this test with our Creality laser.

The Power on Delay is another one you can play with. When you turn on the laser, it can take a moment before it starts actually burning into the material. This setting will turn on the laser and wait a moment before the laser starts moving. If you are seeing a light area at the start of your lines then you can increase this. If your getting a darker spot at the start of your lines you can decrease it. A good starting point is 100ms.

Next you are going to want to set your output Directory and the Filename.

Now its time for the magic to happen. Click Apply and it should output your gcode. It will also create text on the screen showing you the origin.



Now it it time to use the file with your laser.

Copy the gcode file to your SD card and put it in your 3D Printer with the MUTANT and laser in place. Use the menu on your printer to Home the printer.

Put a small piece of wood, or other item you want to engrave with your design on the bed. Now using your menu controls move the laser up and over the block of wood. You may have to do the z height first to make sure the X gantry rollers clear the wood.

Using the measurement that we took before when we were setting up the laser, use your menu controls to adjust the z height so that it matches the same measurement from the top of the wood. Now Home ONLY the X and Y, NOT THE Z.

Put on your safety glasses.
Run the gcode file.

Working with Inkscape and the last Extension

Inkscape and the laser extension being used here are free and open source so they are only supported by volunteers in the community. I do recommend you give JTech Photonics a visit since they so graciously made a nice extension. As such there are some limitations that we need to work around. Most of this has to do with preparing your vector artwork before output.

1. All of your artwork must be converted to Paths. Paths are the only elements that this extension looks at. So all of your elements including text must be converted to paths.
2. You must ungroup everything. If you have any elements grouped the the software will either throw an error and not do them, or it may not translate them correctly.
3. All text must be converted to curves and ungrouped. This is kind of a pain to do but must be done.
4. You can not do fills, only lines. If your text or graphic has a fill, then it will not laser.
5. If you do get an error when you run the extension, there is probably something that is a group or has some other advanced operation on it.
6. If you are still getting errors, select your graphic, hit copy, and then open a new drawing and paste it into the new drawing. This sometimes can fix the error.

Links

Inkscape

<https://inkscape.org/>

Video by DIY3DTech on setting up Inkscape with the laser extension.

<https://www.youtube.com/watch?v=qxroxysbisM>

Plugin for Inkscape to output laser g-code

https://jtechphotonics.com/?page_id=1980

Lightburn Software - A great commercial software that \$40-80 a year.

<https://lightburnsoftware.com/>

Output box designs

<https://www.makercase.com/#/>

Another Free laser control software

<https://hci.rwth-aachen.de/visicut>

Vectrix Software output

<https://jtechphotonics.com/?p=3851>

Cool software for creating repeating patterns for lasering

<https://polygonia.design/>

Which materials are safe to laser

<https://cpl.org/wp-content/uploads/NEVER-CUT-THESE-MATERIALS.pdf>

Some cool Thingiverse Files

<https://www.thingiverse.com/thing:2821011/files>

<https://www.thingiverse.com/thing:20681/files>

<https://www.thingiverse.com/thing:14518/files>

<https://www.thingiverse.com/thing:9331>

<https://www.thingiverse.com/thing:80577/files>

<https://www.thingiverse.com/thing:1929105/files>

Some Sources for Laser Cut art

<https://www.vecteezy.com/free-vector/laser-cut-invitation>

Materials and Speeds

Here are some settings that we used to give you a starting point for your laser. We recommend that you do some experimenting with your laser to get the perfect settings.

Laser Type	material	speed mm/s	speed mm/m	power	passes	result
Creality	PSU Leather Black	5	300	70	1	dark
	PSU Leather Black	10	600	70	1	light
	2mm EVA Foam yellow	30	1800	70	1	engrave
	2mm EVA Foam yellow	3	180	70	1	cut
	7mm EVA Foam Black	3	180	70	3	cut
	7mm Polyethylene foam	3	180	70	3	cut
	bamboo	10	600	70	1	engrave
	PSU Leather tan	20	1200	70	1	engrave
	PSU Leather tan	10	600	70	1	dark engrave
	PSU Leather tan	3	180	70	10	cut
	cork	10	600	90	1	engrave
	pine wood	3	180	70	1	dark engrave