

# How to run every machine in the shop

Let's make Christmas decorations in the easiest way possible.

**fox.build**



# Before you come or before we start the class

- Make a login for Tinkercad.com. This will take our SVG and output an STL for the 3D printers.
- Make a login for Inventables to be able to use Easel for the X-Carve
- Bring a SVG picture of a Christmas ornament.

# Files Files Files

- 3D Printing → STL
  - Find on thingiverse or make your own with CAD
- Laser → Prefer .ai or DXF to input. Vector input
  - Inkscape will convert an SVG to DXF
  - Any image can be imported but scalar images will only scan
  - .ai files come from Adobe Illustrator. We do not have a copy but some of the members seem very familiar with it.
- X-Carve – Easel needs SVG. Surprisingly color is OK
- Maslow – SVG works as well in MakerCAM
  - <https://github.com/MaslowCNC/GroundControl/wiki/MakerCAM>

# Great, well where do I get an SVG

- <https://image.online-convert.com/convert-to-svg>
- [https://commons.wikimedia.org/wiki/Main\\_Page](https://commons.wikimedia.org/wiki/Main_Page)
- Google what you want and svg.
- Trent's Adobe Illustrator class may cover this better.

# Laser Cutter Files



- Files - Import dxf to Reworks on the computer between the 3D printers. Inkscape SVG → Save as .dxf or output from CAD.
- Image files like jpg, png or GIFs can be used for scanned images but the laser will not cut the lines all the way though. Used for engraving only.
- Draw you own basic shapes in Reworks
- Worry about layout in Reworks.
- Anything you bring in as a .dxf will be much easier to adjust and be at least 10 times faster to cut.
- Again SVG → Inkscape → Save as DXF → Import to Reworks

# Laser Cutter Operation

- Cutting area is 500mm long and 250mm tall. ~20"x10"
- It will cut up to ¼" material that will burn. Wood, plastic, cardboard, leather.
- Do not put in vinyl or Lexan (Polycarbonate). The fumes are poisonous.
- Focus your work. (Use the 6mm acrylic piece on the side of the machine with the red ribbon. Toggle switch on the side of the laser moves platform up and down.
- Make sure air is blowing from the nozzle and water pump is going.
- Keeping the lid on will cutting to prevent you from cutting off your fingers.
- The bow of a sheet will make the difference between cutting through and not cutting.
- If you are just cutting, check if everything cut and if not, run it again.
- The speed of a cut will change the kerf. If you want consistent parts, use the same speeds.
- Small parts that jump when cut out can hit the nozzle and move the work. Cut them last.
- You can change cut order by placing them in the lower left corner or with layers.
- There is a way to offset the laser call sew compensation. Use this if the 0.1mm kerf is an issue. You can cut on either side of the line by adding a positive number for outside and negative number for inside.

# Speeds and power settings

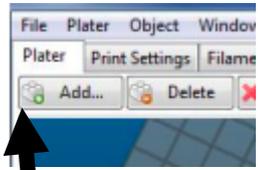
- Speed and power is trial and error. Start with a spare piece of a similar material and see what looks good.
- Acrylic – 10 mm/s @ 80% to cut 0.1”
- Wood – 8 mm/s @ 80% to cut 3mm wood. Slower for 5-6mm.
- Scan is meant to run faster 50-200 mm/s and it is run at a lower power.
- Click on a layer and you can adjust the power and speed.
- Check out this Youtube video for some great tips.

<https://www.youtube.com/watch?v=9dz8w9rlrHc>

# 3D printing files

- 3D printer needs a STL file to print. This is turned into a .gcode file but we can do that on the computer between the printers.
- Again lets start with a SVG file and import it in to Tinkercad.
- Size it as you see fit and export the STL.
- Alternately, you can download a STL from thingiverse or even make your own in Tinkercad or any other cad package. The important part is to have a STL file to bring into Slic3r

# Slicing for the Prusa



- Open the slic3r for Prusa multimaterial
- Click Add and find the STL you want to print.
- Rotate your work so it is sits in the way you want it to be printed. (Deep subject)
- Right click on the part on the right side to specify the extruder to use.
- Adjust the printer setting if needed. (Fill ratio etc)
- Slice the parts and export .gcode. Save it to somewhere convenient.

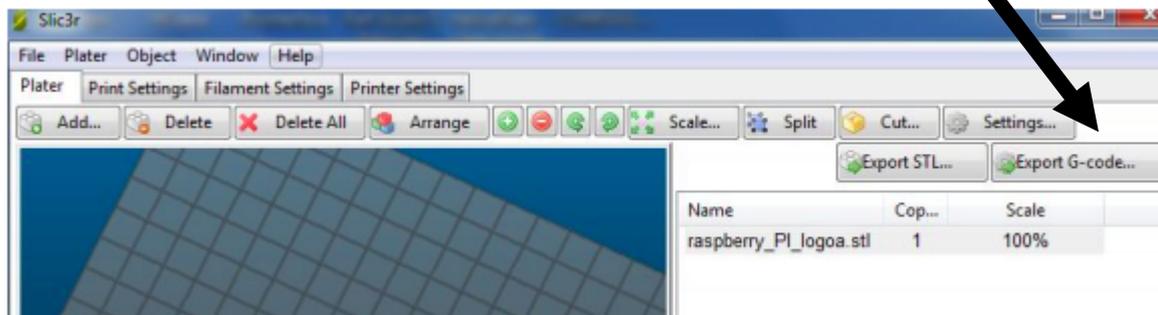
# Printing on the Prusa

- Open Octoprint by typing 10.1.10.240 in the browser.
- Login and connect.
- Drag the .gcode file you made to the left side of the octoprint screen.
- Turn on the Prusa. Check that the filament is in the extruder you chose in Slic3r. If not, use the Prusa controls to preheat, unload and load the filament.
- While preheated, try to run the extruder either from octoprint or the Prusa menu. If it is stuck, unscrew the bowden tube of the extruder you are using and make sure it is feeding properly. We will be getting a new set up. Our junction for the filament is presently Borked.
- Clean the print bed with the scraper and then alcohol.
- Once you get filament coming out when you extrude, click the printer button on the file you are trying to print in octoprint.

# Slicing for the Tarantula



- Open the slic3r.exe
- Click Add and find the STL you want to print.
- Rotate your work so it sits in the way you want it to be printed. (Deep subject)
- Adjust the printer setting if needed. (Fill ratio...)
- Slice and then Export .gcode. Save it to somewhere convenient.

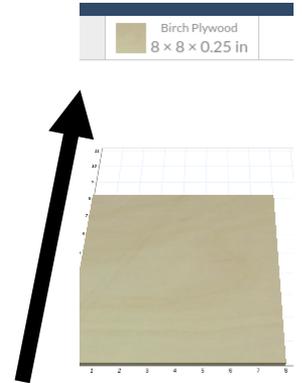


# Printing on the Tarantula

- Warning. The Tarantula is still in beta testing and has not been completely dialed in. I have printed a few things and its about 90%. Since David moved on to his next gig, this printer is desperately in need of a sponsor.
- Open Octoprint by typing 10.1.10.241 in the browser.
- Login and connect. See yellow sheet to the right of the computer for login info if it does not auto populate.
- Drag the .gcode file you made to the left side of the octoprint screen.
- Plug in the Tarantula.
- There is one manual setting that should be checked. The z-offset needs to be +0.6mm. If you print and no filament is coming out it is probably because this needs to be set.
- Clean the print bed with the scraper and then alcohol.
- Click the printer button on the file you are trying to print in octoprint.

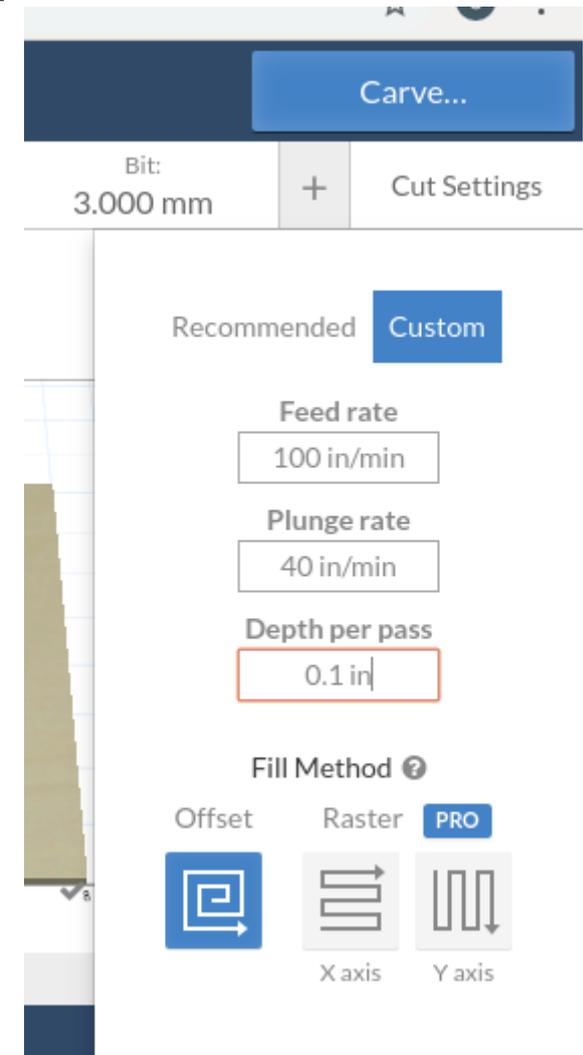
# X-Carve

- Start up the laptop and login in to Easel.
- Adjust the size of the board you will use by clicking on the board size.
- Put down a waste board under your work and clamp it in a way that the bit will not hit the clamps.
- Import your SVG.
- Adjust the depths of the areas you will cut or pocket.



# X-Carve Feeds and Speeds

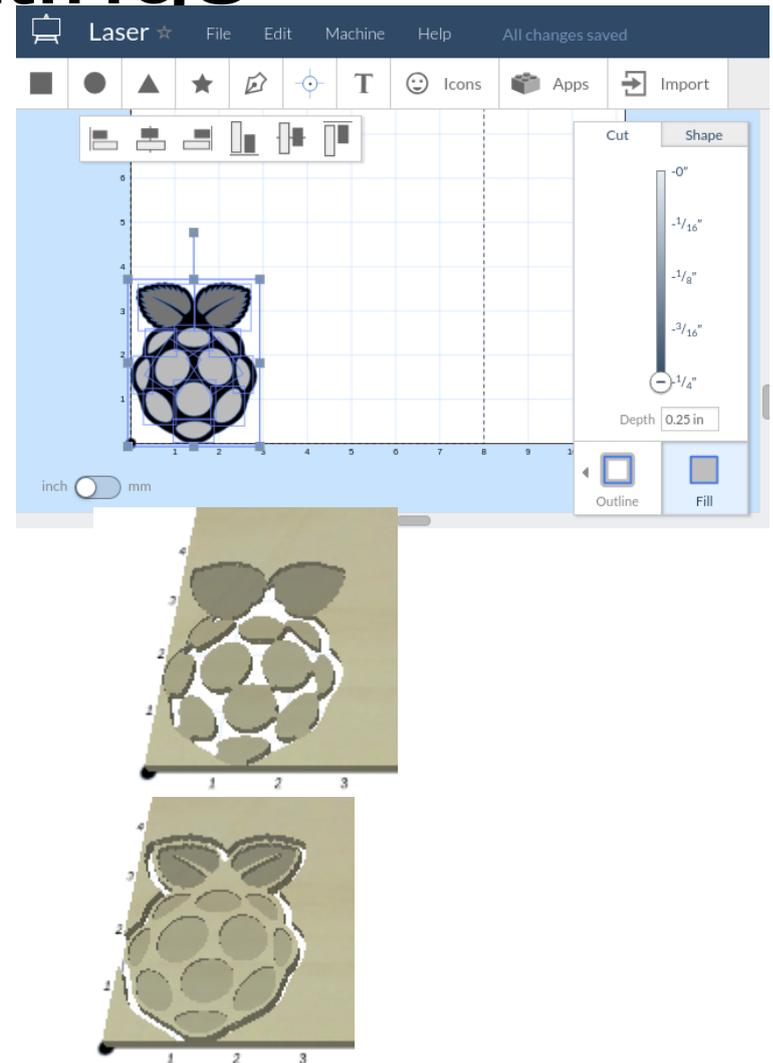
- The bit it in there now is 3mm.
- The recommended settings take forever. I have had good luck with the settings on the right.
- The depth per pass should not be 50 → 100% of the bit diameter.
- If the bit is shaking or the work squealing, slow it down a bit.



T

# X-Carve Settings

- When I imported the Raspberry Pi logo SVG it looked like this.
- There preview showed me I was a little off.
- I clicked on the black and changed it to outline instead of fill. It changed to what I wanted.



# X-Carve Carving

- Click Carve and walk through the screens.
- When moving, be extremely careful about the units for movement. 1 inch in the z is a lot and both Doug and myself have broken bits as we jammed the router into the board. I did it after I saw Doug do it.
- The origin should be the lower left hand corner. The home position is on the top of the work.
- Again make sure the clamps will not be hit. The safety height can be adjusted in the advanced settings.

# Maslow CNC



- The Maslow is really similar to the X-carve, it just handles a bigger work area.
- It takes Gcode in the form of a .nc file.
- This Gcode can be made by MakerCAM.
- Maker cam requires a SVG file and outputs the .nc gcode file. Here is a great description of how to do it.  
<https://github.com/MaslowCNC/GroundControl/wiki/MakerCAM>
- Other ways to make the gcode can be found here.  
[https://github.com/MaslowCNC/GroundControl/wiki/CAM-\(G-code-generation\)-programs](https://github.com/MaslowCNC/GroundControl/wiki/CAM-(G-code-generation)-programs)
- I have been using Kiri:Moto as an application in Onshape. It is great for waterline cuts but the pocketing seems to require facing which takes forever. The upside is I just import the part I want to cut from Onshape and start setting it up.

# MaslowCNC cutting

- Once you have your .nc gcode file you are ready to cut.
- Open ground control and load your file.
- Set your work so the top surface is 3/4" from the waste board. It is critical if the sled will hit the bottom as it cuts the lower part of the part. If you are cutting a 1/4" board add a 1/2" waste board.
- **SECURE YOUR WORK.** Not needed for a 4x8 sheet of wood but a 2x4 sheet can be moved by the sled causing chaos.
- Understand where your origin is on your Gcode. Move the sled to that position and press define home. You will then see how your carving will look in the preview in ground control.
- Make sure the z-axis is just touching the work. If not move it up or down to zero it and hit define zero.
- Save and raise the z-axis for travel with the button in the z-axis menu.
- Click the play button and let her rip.