Laser Cutter Instructions

(v1.0, 6/20/2017)

Erik Brunvand, School of Computing, UofU

WARNING: ONLY cut approved materials. Many materials produce dangerous and even poisonous gasses when cut/melted by this laser.

WARNING: Because this IR laser cuts with heat, things can catch on fire during the cutting! Be extremely careful when cutting - especially flammable materials like paper, cardboard, and wood. If you think you might have issues with flare-up and flame, you can put a damp paper towel on top of your material to be cut. This will not impact the cut noticeably, but can keep things from flaring.

WARNING: There are a lot of little things to get right when setting up for a cut. Please always use the "pre-flight" check-off before cutting.

Figure 1: The laser cutter in the Senior Hardware Lab. The water chiller is on the floor to the left. The filter is the grey box on the right. The compressed air is accessed through the manifold on the wall. The black PC is the driving PC.

Introduction:

Our laser is a 120w $C0_2$ laser cutter with a 24x36" bed. It is great for cutting acrylic, thin wood, paper, cardboard, and other approved materials. It will NOT cut metal though, so don't try! It also stinks a bit when things are being cut, so if you can do your cutting in less-busy hours, other users of the lab, and people whose offices are near will thank you.

Preparing your Design:

Designs to be **cut** should be prepared in a vector drawing program like Illustrator, Corel Draw, Inkscape, Rhino, or AutoCAD. Raster drawings (photoshop, gimp, etc) can be etched into materials, but they cannot be used to define lines to be cut. Lines to be cut should be drawn as vectors with a stroke width as thin as possible. Depending on the drawing tool you use, this sometimes means setting the line width to be "hairline" and sometimes means setting a small value like 0.001pt. Lines to be cut should also be a pure RGB color. That is, select the lines and make them 100%R, 0%G, and 0%B (e.g. 0xFF0000), or pure blue, or pure green. You can also use cyan, magenta, and yellow if you're in a CMYK color space.

Designs to be raster engraved should be in black and white where black means "cut" and white means "don't cut." Because the laser is either on or off, you can't directly engrave designs that have shades of grey. But, you can "fake" shades of grey with halftones. Halftones are small dots used to represent density of black: the more dense the dots, the darker things appear, and the less dense the dots, the lighter. This is how newspapers print shaded images if you look closely. Most raster drawing packages (e.g. Photoshop, Gimp, etc.) have a tool that will turn greyscale images into halftoned black and white images. But, bold black and white designs may work best.

The laser print driver will consider thin vector lines in a pure color to be lines to cut. If you have lines that might need different cutting settings, you can have multiple different colors in your design and set the cut differently for each color. If you have raster objects to etch into your design, make those pure black. This will distinguish the shapes to be cut from the shapes to be raster-engraved.

Save your design as a PDF. Currently we don't have any drawing packages loaded on the machine that the laser cutter is connected to – only Adobe Acrobat. So, you'll open your design in Acrobat and "print" from there (which means cutting/engraving the design).

Designs should be done in their final actual sizes. That is, if you want a 5" square, it should be 5" in the drawn image.

Some designs have features designed to friction-fit with other features (like finger joints on a box corner, for example). For things to fit well you need to account for the **kerf** of the laser beam. The kerf is the width of the laser beam that's making the cut. If you think about a saw – if the saw is 1/8" thick, then when it makes the cut, it removes 1/8" of material in the process of making the cut. That's the kerf. In our laser the kerf is thin, but it's still there and should be accounted for if you want snug-fitting joints.

Unfortunately, the kerf is not uniform on laser cutters – it is slightly different depending on the material (some materials get eaten away more easily than others), the speed of the cutting, and the strength of the beam. You'll have to do some tests to make sure you get

things exactly right for your materials. Depending on the material and the thickness, the laser kerf can be anywhere from 0.015" to 0.008" (0.4 to 0.2mm, but many drawing tools still use inch measurements).

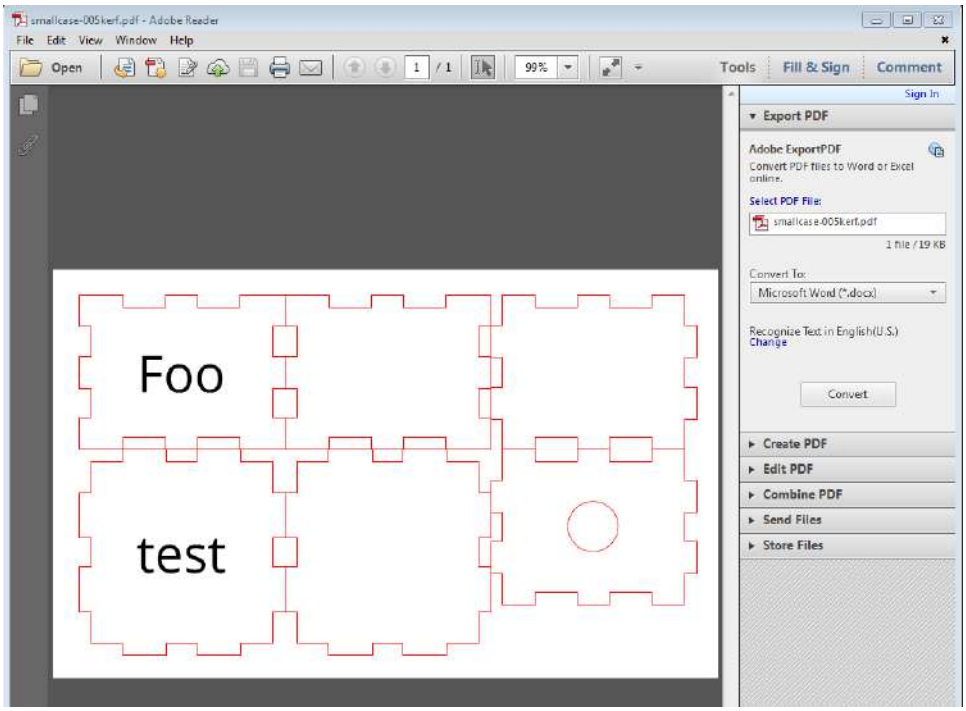

Figure 2: A box design from a box-making widget at MakerCase.com. The lines have been specified to be pure red, and the raster text is in black. BoxMaker.com outputs the design as a .svg (vector) file. I've converted this to .pdf using Illustrator.

As an example of an input file, Figure 2 shows a box made from a template at MakerCase.com. This box has finger joints for assembly, and if you look carefully you can see that they've taken the kerf into account because the slots and fingers are slightly different sizes (to account for the width of the kerf when cutting). The cuts are in thin red vector lines, and there is raster text that will be engraved into the material. Be careful with the kerf – nominally the kerf is the width of the laser beam, but in MakerCase.com they ask for "width of laser beam / 2" as the kerf, so a setting of .004" in this case makes the finger joints snug when using 1/8" acrylic.

This box was returned to me as an .svg file which is a vector format file. But, it's not a format that Adobe Acrobat can read, so I had to convert it to PDF using Illustrator. There are also web-based conversion apps you can use.

Bottom Line: Prepare a vector drawing for cutting with pure-color lines as thin as your program can make them. Raster designs should be in black and white (or halftoned black and white). Save or convert this vector file to PDF for printing/cutting on the laser.

Once you have your design as a PDF, you open it in Acrobat and "print" to the laser cutter. Before you do this, though, you should have the laser cutter application running so that the "print job" from Acrobat has an application to go to. The basic idea is that when you print your design, the printer is the laser cutter and the lines are cut instead of drawn.

Starting Up the Laser Cutter and Retina Engrave Software

The laser cutter software (Retina Engrave) needs to communicate to the laser cutter, so it's easiest if you start up the hardware first before launching the software. There are four parts to the laser cutter hardware:

1. The water chiller that cools the laser tube
2. The filter unit that tries to filter the exhaust from the cutter
3. The compressed air that blows the exhaust away from the cut
4. The laser cutter itself

When you start things up, do them in that order – first the chiller, then the filter, then the compressed air, and finally the laser cutter. *You definitely NEVER want to make a cut without having the chiller, filter, and compressed air up and running!*

Chiller: if the red LED display is on, the chiller is running. If not, there's a switch on the front plate. You'll get some loud scary beeps the first time you fire it up – let those happen and they'll stop pretty quickly (Figure 3).

Air filter: there's a red button on the front of the filter (Figure 4). You can change the speed of the fan in the filter, but the default of 70% speed is usually fine.

Compressed Air: There's a valve on the wall – move the handle to be in-line with the pipe to engage the air – you'll hear a hiss when it's on (Figure 5).

Laser Cutter: There's a key in the top compartment of the plastic tool box nearby the cutter. The metal key (without the plastic) is the key for the laser cutter. Turn key to turn on the cutter. It will take 1 minute or so to boot, and show that it is connected to IP 155.98.66.184 (Figure 6).

Figure 3: Water chiller up and running at 24.7 degrees C.

Figure 4: Air filter - up and running at 70% speed (which should be fine for our laser cutter).

Figure 5: Compressed air turned on (bottom valve).

Figure 6: Laser cutter front panel when booted. This shows the default IP address, and that the cutter's control panel is in "Fast XY" (FX) mode. This means that the arrow keys move the laser cutter head in the XY direction. This mode is useful for positioning the head on the cutting bed.

Once everything is powered up and running, you can fire up the Retina Engrave software on the PC. The software will first search for the laser cutter on the network (reporting "Connected" in green at the bottom left of the main window), and then try to connect to the Full Spectrum Laser corporate office. This seems to fail and takes a while to time out. We don't know why at the moment. But, it means that Retina Engrave takes a minute or so to start up (time out). Figure 7 shows the main screen after the software successfully connects to the laser cutter, and has finished timing out.

Once the SW is running, the first thing you should do is "home" the laser. This moves the laser cutting head to a known home position so that the SW knows where things are. The button that looks like a house is the "home" button. When the laser has been homed, you'll see a green "homed" verification in the lower left (also Figure 7). You're now ready to open a design and "print" that design to the laser SW, and cut it on the laser cutter.

As an aside, there are a few controls on the front panel of the laser cutter that you'll find very useful (see Figure 6). These are:

- F/S Z: This button lets you cycle through the "modes" for the control panel. F/S are the Fast and Slow XY movement modes, and Z is for the Z-axis movement. Note that there are lots of modes, and you have to cycle through them to get from one to the

other. So, if you're currently in FZ mode (for example), it takes a number of button pushes to get back around to FX mode. The modes will be noted on the LCD display. The ones you'll likely use are:

- FX: Fast XY mode – in this mode the arrow keys will move the laser cutter head in the X and Y direction on the bed, and move at the fastest speed that the cutter can move. You'll use this mode to position the laser head for focusing and to position the head for a cut.

- SX: Slow XY mode – same as before, but much slower for fine-tuning positions

- FZ: Fast Z mode – the up and down arrows move the cutting bed up and down. You'll use this to move the bed down before putting new stock on the bed to be cut.

- SZ: Slow Z mode – for fine-tuning the Z-axis position. You may not have to use this one

The other buttons on the control panel are mode-specific. The upper (green) icons are in use when the cutter is in FX or SX mode, and the lower (yellow) icons are in use when in FZ or SZ mode. The most useful buttons are:

- Green house – Homes the laser head to the upper right corner of the cutting head. This must be done once (at least) when the laser and the software are fired up so that both the HW and SW know exactly where the head is

- Yellow AF – Autofocus – this activates the autofocus procedure where the bed is raised until the gold-colored autofocus plunger hits the material.

- Yellow arrows – this is active in all modes, and engages the "perimeter check" of the laser. The design that is currently loaded in the Retina Engrave SW is checked for its extremes, and the laser head makes a circuit of that rectangle to show the extent of the area where it will cut. You can use this to make sure that your design will fit on your stock before cutting.

There are many parts of this procedure where you'll want to know where the laser cutting head is positioned. The red dot that you can see on your stock is the aiming laser. It's basically a laser pointer that shows you where the cut will eventually be made through the cutting head. The actual cutting laser is IR and is not in the visible spectrum so you can't see it. It's also disabled when the lid is open, so one way to see exactly what the laser will do before you actually cut is to start up the cut with the lid lifted. This will allow you to see where the cut will be made, without actually firing up the cutting laser. For cutting the lit must be closed.

When you look at the aiming laser, make sure there's a tiny pinprick bright dot in the center of the larger aiming dot. If not, the compressed air shroud may need to be re-positioned.

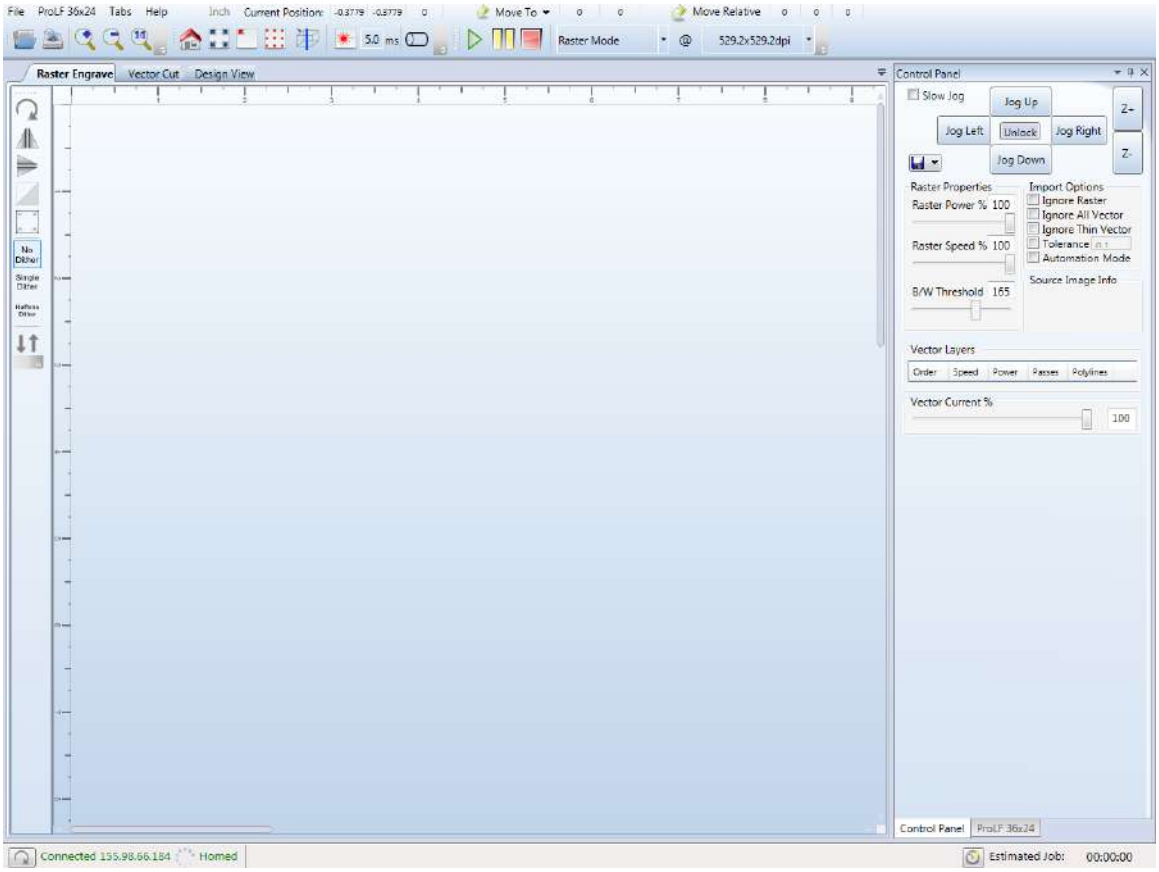

Figure 7: Retina Engrave main screen. Note the green "Connected" text in the lower left.

Loading a design in Acrobat and “printing to the laser”

The easiest way to transfer a design to the PC connected to the laser is using a USB flash drive. Once you have your PDF file on the machine, open it with Acrobat. You should see your design in Acrobat with the associated vector (and perhaps raster) images in the colors you expected.

The process of cutting your design is to “print” the design from Acrobat through the Retina Engrave SW to the cutter. So, select Print in Acrobat. The only control you need to pay attention to here is to set the page size to be FSL ProLF Series 36x24 to represent the size of the print bed on the laser.

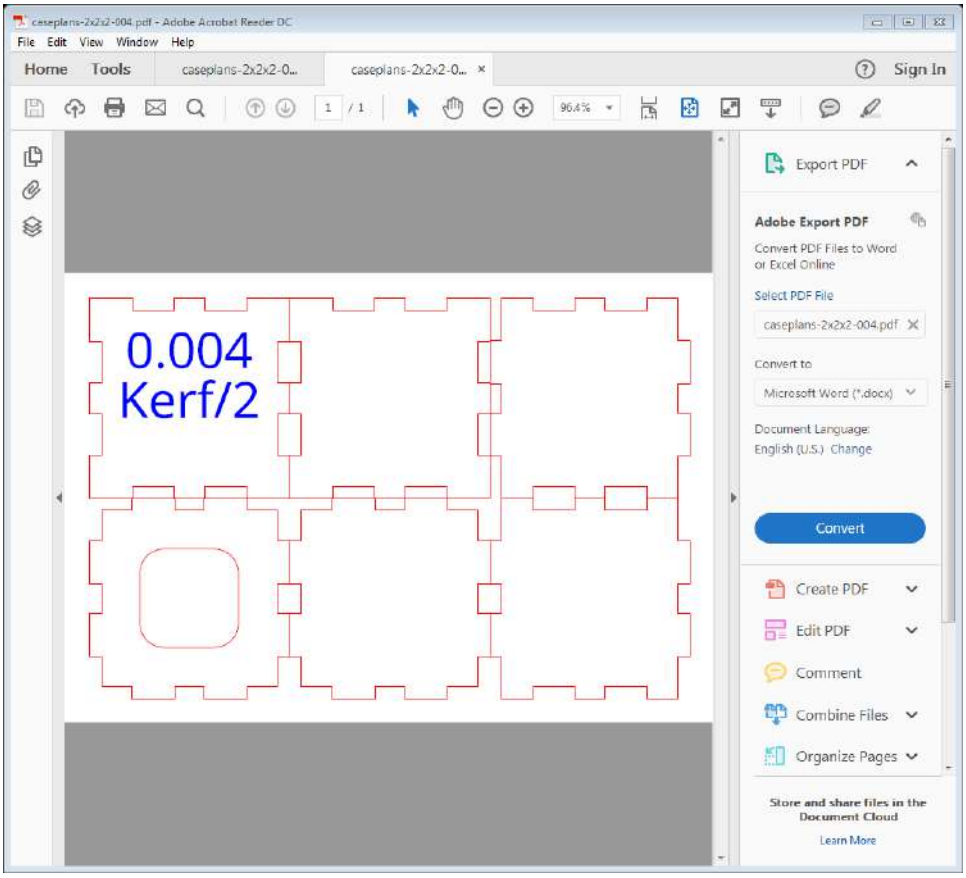

Figure 8: An example file viewed in Acrobat. The red lines are the vector lines to be cut, and the blue is that raster image.

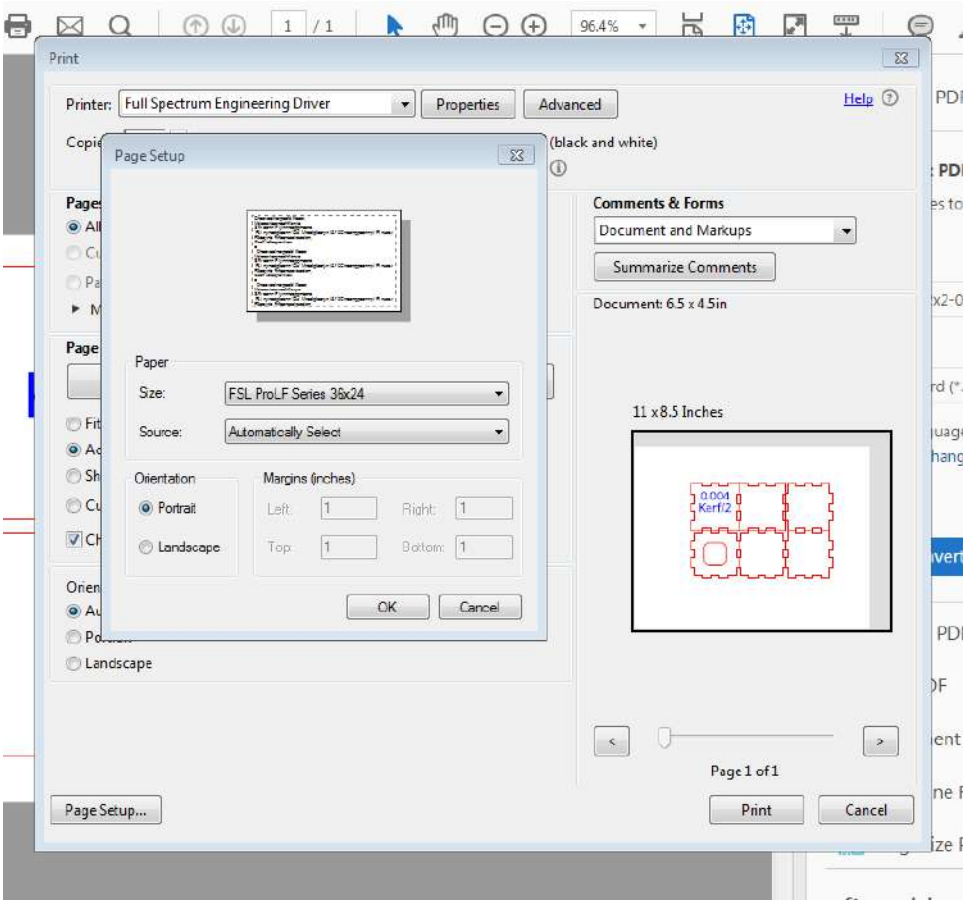

Figure 9: Setting the page size to FSL ProLF Series 36x24 through the Page Setup dialog box.

Now when you select Print, this will send the design to the Retina Engrave SW. You may see a dialog box like that in Figure 10. If you have no raster elements in your design (only vector cuts), you can safely choose No in this dialog to save time in transferring the design.

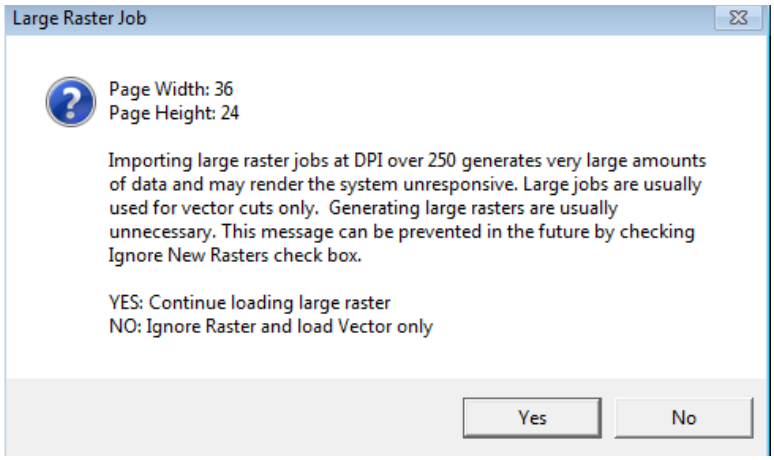

Figure 10: Dialog box that may appear, depending on your job size. If you're only cutting (not engraving a raster) you can safely click "No" to load only the vector portion of the file.

When the design has been transferred to the Retina Engrave SW, there should be two different views of the design: one in the “Raster” tab (Figure 11), and one in the “Vector” tab (Figure 12). Note that in this example design, the outline of the box has been included in the Raster tab. This is likely not what you want because although it won’t hurt anything, it will take a long time to raster the outline before you end up cutting through it. To keep the design from seeing those thin lines as raster-able, select the “Ignore Thin Vector” input setting in Retina Engrave before printing from Acrobat.

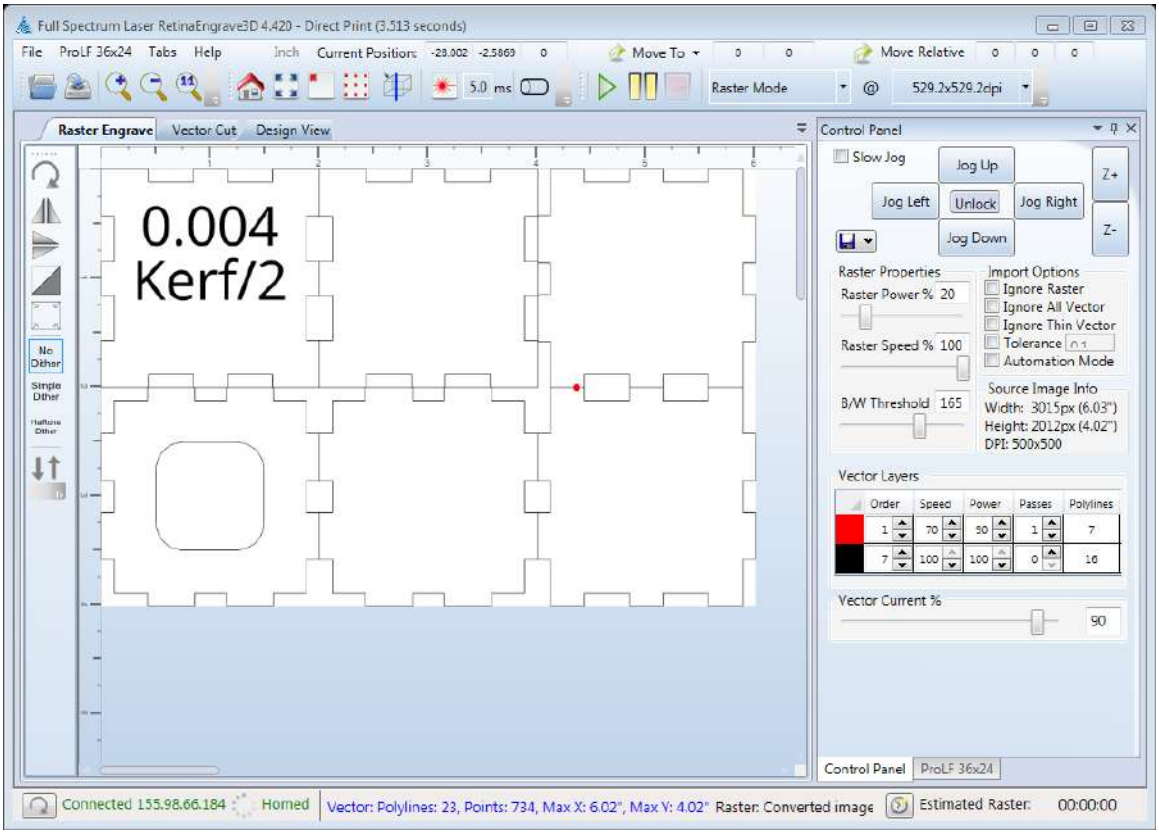

Figure 11: Raster tab after importing (printing) the design from Acrobat. Note that the outlines of the vector cuts are included in the Raster Engrave tab. This is likely not what you want. You can avoid this using the Ignore Thin Vector option in the control panel on the right before importing (printing) from Acrobat.

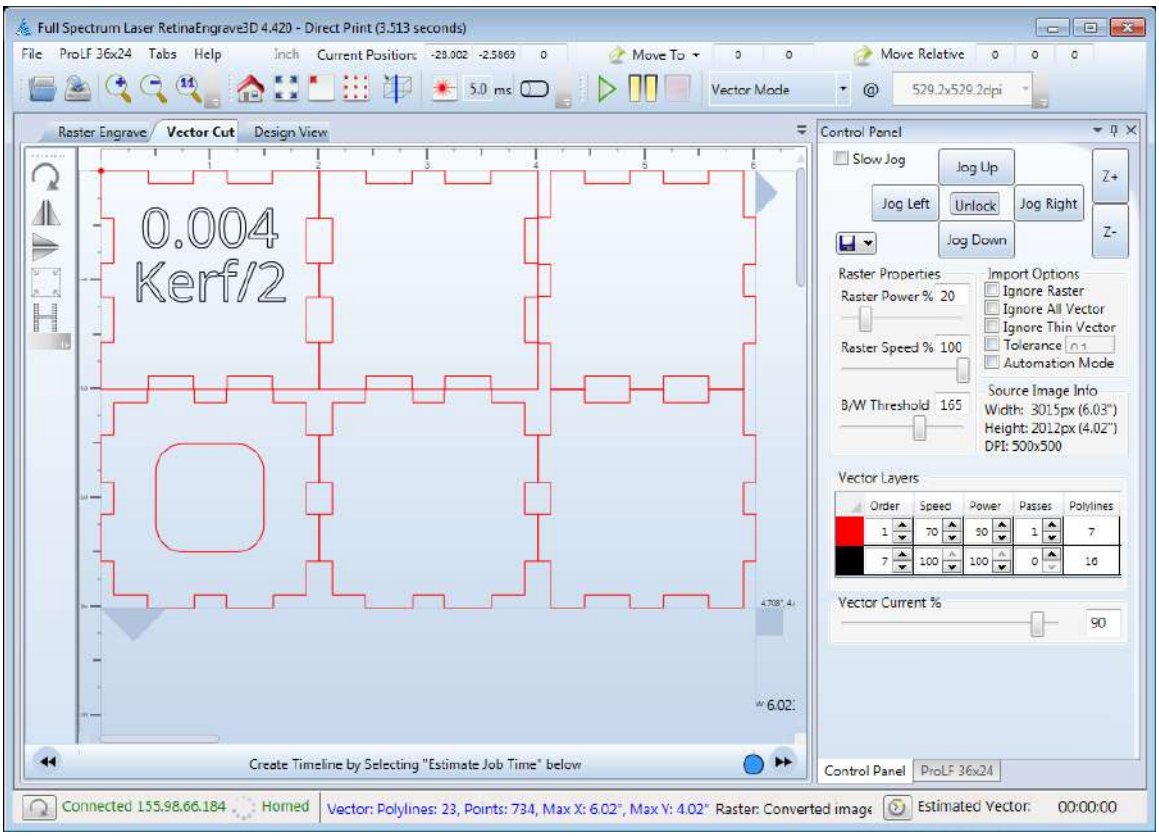

Figure 12: Vector tab after importing (printing) the design from Acrobat. Note that the outlines of the (raster) letters are in the Vector Cut tab. You can avoid cutting those outlines by selecting "0 passes" for the black lines in the Vector Layer control on the right.

The main controls for the Retina Engrave SW are seen in Figure 13. The house-shaped icon is for homing the laser. The icon with the four blue arrows will cause the laser head to "run the perimeter" of the design so that you can see if the design will fit on your stock before cutting. The outline cube with the blue line in it is the AutoFocus button. And the green "play" button starts the cut. You can hover over the other icons to see what they do.

Figure 13: Retina Engrave button/icon bar at the top of the main window.

Now that you have a design in retina Engrave and looking how you want it, you need to set the laser power and speed to make sure you don't start a fire. This is very important!

There are four settings that control how the laser makes its cuts (see Figure 14):

1. **Speed** – This controls how fast the cutter moves when it is making its cut, or rastering the image. The slower the head moves, the stronger the cutting effect.

2. **Power** – This controls how the laser beam is pulsed on and off. At 100 the beam is constantly on during cutting. At 50 the beam is pulsed so that it's only on half the time. A smaller number here means that the beam is on for a smaller percentage of the time, and therefore is a weaker beam.

3. **Current** – This controls how much current is delivered to the laser tube. At 100 the tube gets 100% of its rated current. Smaller numbers scale the current. So, 80% in this setting means that the laser tube gets 80% of its maximum rated current, and thus is a weaker beam.

4. **Passes** – Used in the Vector Cut menu, this controls how many passes over each line the laser cutter makes. For very thick material, you may need to make multiple passes to cut all the way through.

We have tried a number of materials and posted suggested settings for approved materials in the lab. For example, on acrylic, for a rastered image, we recommend 20% power and 100% speed of the laser. For cutting 1/8" acrylic we recommend 75% speed, 90% power, 90% current, and 1 pass. For other (approved) materials we may need to experiment to get the best settings. The settings for the example seen in this document (the box) are seen in Figure 14.

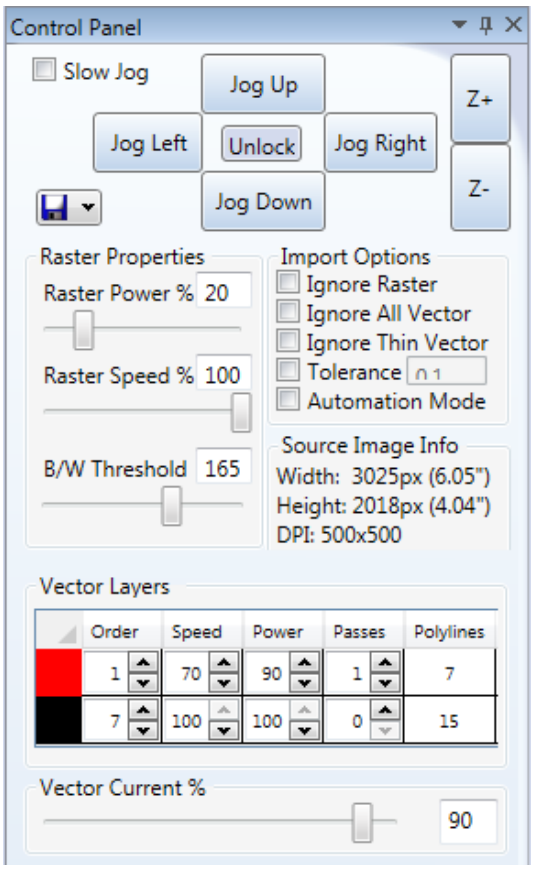

Figure 14: Settings for 1/8" acrylic - both vector cutting and raster engraving.

Cutting!

Before you cut, you need to focus the laser on your material. Follow these steps:

1. Move the bed down (Z axis) so that your material won't hit the laser head when you put it in the machine.
2. Move the laser head (XY mode) over the material
3. Execute the automatic focus. This will move the bed up until the gold-colored autofocus plunger makes contact with the material, then back off slightly. The laser is now focused on the surface of your material.

Once the laser is focused, you should move (XY mode) the laser to the upper-left of your stock. The red aiming laser is what you're looking at in terms of positioning the laser head. This is the starting point for your cut. When the laser is in position, you can check to make sure your design will fit on the material by causing the laser to "run the perimeter" of the design – that is, it will move in a rectangle showing you the extent (by watching the red aiming laser) of where it will cut. You can watch during this to make sure that the laser stays over your stock. The perimeter check is the icon with the four blue arrows in the Retina Engrave SW (Figure 13), or the button with the yellow square arrows on the laser cutter panel (if you're in XY mode).

Once again, make sure that you can see a bright pinprick of red inside the larger aiming dot. If you can't see that, the compressed air shroud over the laser head may need to be repositioned.

When your stock is focused, you have the laser head positioned in the upper left corner, and you make sure that your design will fit on your stock, you can close the lid of the laser cutter, and use the green "Play" button to make the cut.

Very important – do NOT leave the cutter unattended while it's making a cut! The laser cuts with heat (the laser is infrared), and in some cases things can catch fire! You need to be there to put it out if that happens!

When the cut finishes, remove all pieces from the bed (including the tiny pieces left over from cutting out bits), and clean up your working area.

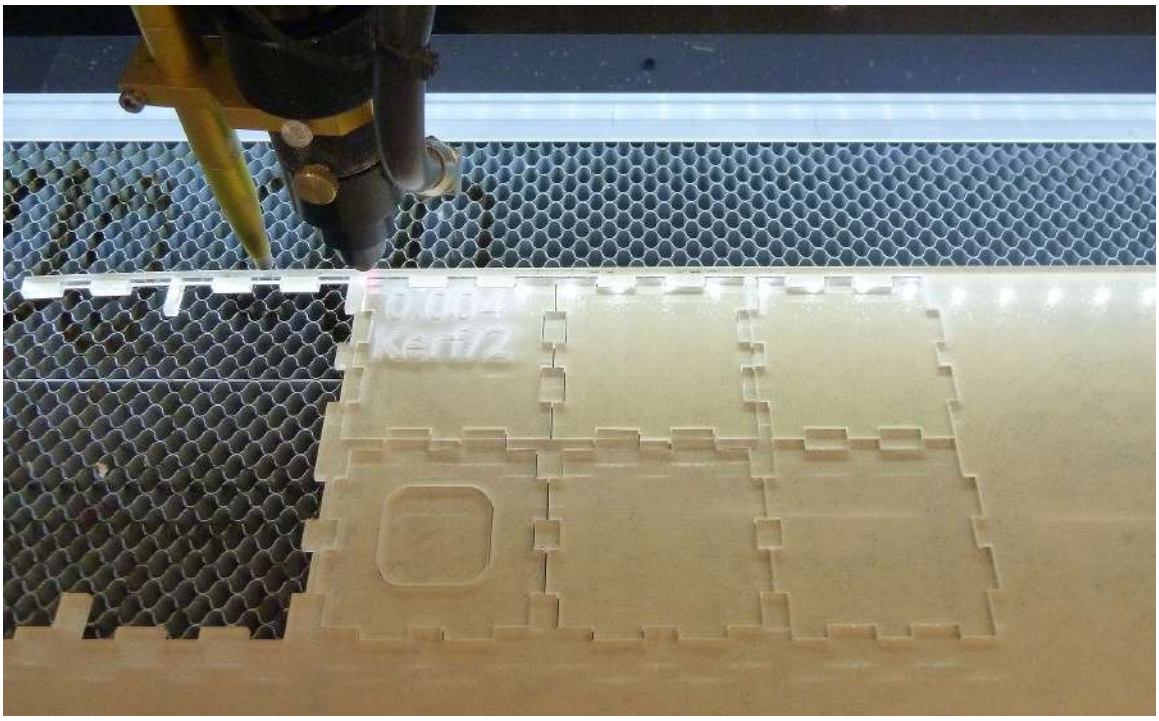

Figure 15: A successful cut!

Note that in this case I've left the protective paper on the acrylic on the bottom. The laser will cut right through the protective paper that comes on most acrylic so you can choose to leave it on or take it off – your choice. I like to take the paper off the top surface of the acrylic and leave it on the bottom. That way I can visually watch the cut and make sure that it's really cutting all the way through the bottom layer by looking through the clear acrylic.

Also, as mentioned before, if you are worried about flame-up, or about scorching of the material as the cut is being made (e.g. on cardboard), you can put a piece of slightly damp paper towel on top of the material being cut. This should have minimal impact on the actual cut being made.

Figure 16: The resulting box held together with friction-fit finger joints.

Approved Materials for the Laser Cutter

Make sure you know what materials you have!
Many materials look similar…

		Raster Etch		Vector Cut		
Material	**Thickness**	**Speed**	**Power**	**Speed**	**Power**	**Current**
Basswood	1/16”					
	1/8”	100	20	100	20	40
	3/16”					
	1/4"					
Plywood	1/8”	100	20	100	40	40
	3/16”					
	1/4"					
Wood Veneer	thin					
Acrylic	1/16”	100	20	100	60	30
Plexiglass	1/8”	100	20	70	90	90
Lucite	3/16”	100	20	50	90	90
PMMA	1/4"	100	20	25	90	90
	3/8”	100	20	20	100	100
	1/2"	100	20	15	100	100
MDF	1/16”					
	1/8”					
	3/16”					
	1/4"					
Paper, card stock	thin					
Cardboard	1/8”					
	1/4"					
Matte Board	1/16”					
	1/8”					
Cork	1/8					
	1/4"					
Mylar	thin					
	thicker					
Cloth (cotton, felt)						
Glass				NA	NA	NA

Two general rules:

1. **Make SURE you know what materials you have – many look similar**
    a. **For example, polycarbonate (banned!) looks a lot like acrylic (approved)**
2. **If it’s not on the APPROVED list, do NOT try to cut it!**

Prohibited Materials in the Laser Cutter

Don't even THINK about trying to cut these!!!!

Material	DANGER!
PVC (Poly Vinyl Chloride) **Vinyl, Pleather, Artificial leather**	**Emits pure chlorine gas when cut!** **Not only is this poisonous, it will ruin the optics of the lase cutter**
Polycarbonate / Lexan	**This will catch fire! Polycarb absorbs IR radiation so it doesn't cut in the laser cutter. Easily confused with acrylic! Make sure you know what you've got!**
ABS plastic	**Emits cyanide gas when cut!** **Not only is this poisonous, it will melt instead of cut and mess things up**
HDPE / milk bottle plastic	**Catches fire and melts!**
PETG	**Does not cut. Can look similar to acrylic! Make sure you know what you've got!**
PolyStyrene Foam (Styrofoam)	**Catches fire and melts!**
PolyPropylene Foam	**Catches fire and melts!**
Fiberglass	**Emits fumes, and doesn't cut. It's a mix of two materials that don't cut: glass and epoxy resin.**
Carbon fiber	**Emits dangerous fumes**
Foam Core board	**Emits chlorine gas and catches fires**
Food materials	**Food will melt and become contaminated, and make a big mess**
Polymer clay	**Emits chlorine gas.**
Metals	**Our laser can't cut metal!!!**

Two general rules:

1. **Make SURE you know what materials you have – many look similar**
    a. **For example, polycarbonate (banned!) looks a lot like acrylic (approved)**
2. **If it's not on the APPROVED list, do NOT try to cut it!**