



## Inkscape and Designing for the CNC Plasma Cutter

Name: \_\_\_\_\_ Block: \_\_\_\_\_ Date: \_\_\_\_\_

If you plan on modifying an existing image that you found online these tips and tricks will be helpful to get your final image

Look for the highest quality image you can in “google images”. Ideally a logo with 3 or less colors and simple shapes.

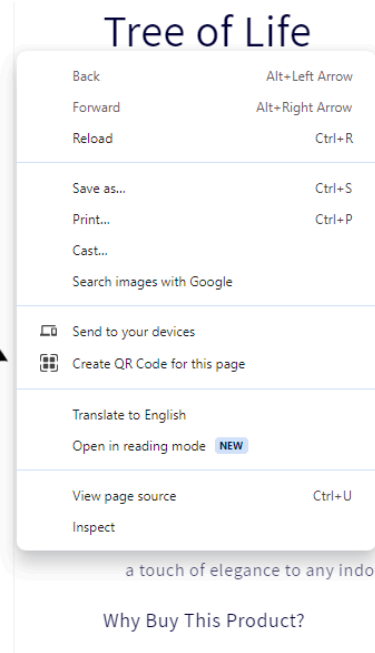
Good - simple shapes	Not Good - too complicated to cut with the plasma cutter
 <p>The image shows two examples of good designs. The top one is the KTM RACING logo, which consists of the letters 'KTM' in a bold, italicized, sans-serif font above the word 'RACING' in a similar font. The bottom one is a black silhouette of a gear with several cutouts, representing a simple shape suitable for CNC plasma cutting.</p>	 <p>The image shows a detailed line drawing of a dog's head, likely a Golden Retriever, facing right. The drawing is highly detailed with many fine lines and cross-hatching to create texture and shading, making it a poor example of a simple shape for CNC plasma cutting.</p>

### Image search tip:

If looking online for a specific type of image use key-words in your search like “bicycle gear silhouette vector” or “tree nature dxf”. Key-words like **silhouette**, **vector**, **png**, **svg**, **dxf**, or **transparent** will help narrow your search to usable images that require less editing.

To download your image you may be able to “right click on it” and save the file. If an image is made out of **pixels** (called a raster image) you will hopefully get a JPEG, PNG, or GIF image. Images that are incompatible with the software could be WEBM or TIFF.

It's a bonus if you can get a DXF, SVG images as those are already a type of **vector file** which is our end goal



Above image: I right clicked on the image and can try to “save as” to save the image to my computer.

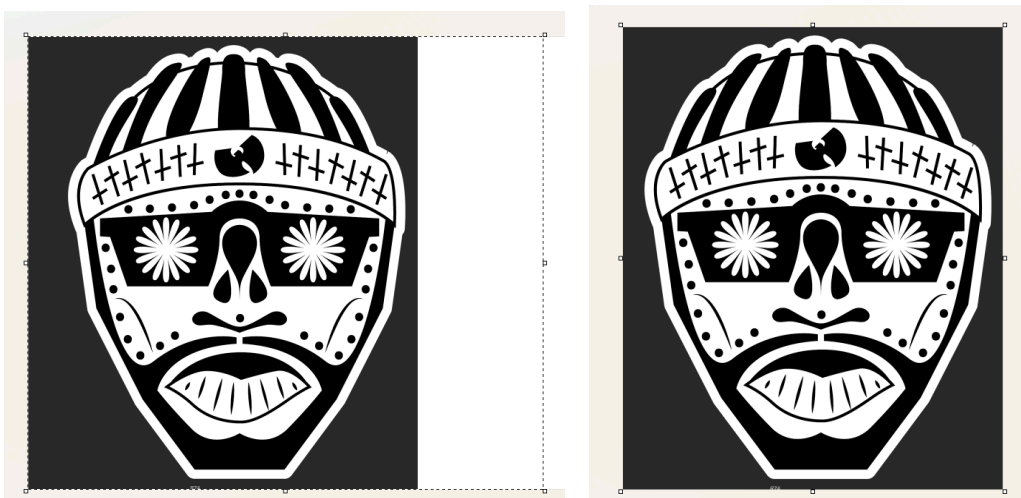
### Screenshot of your screen if you can't download:

If you found a quality image but can't save it via the “right click + save as” method you can try to

- 1) press the screen shot key on your keyboard (PRT SC, Print Screen, capture, etc).
- 2) Open the microsoft paint software and press CTRL V (shortcut for paste) and it will paste a screenshot of your screen into the software. Crop your image in Paint and “file + save as” a JPG or PNG image.



This is what the “crop tool” will look like in most types of image editing programs



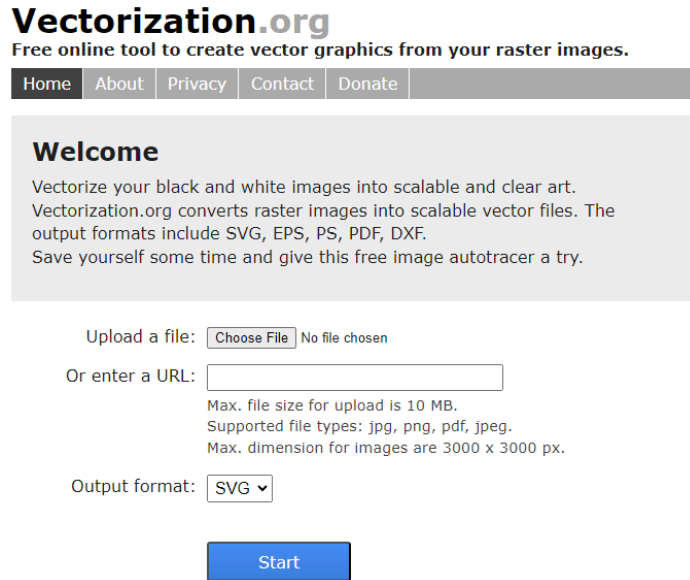
The dotted lines around the image represent the crop area. Click and Drag the dots or edges until your image is sized right. Then “file + save as” a JPG or PNG image. Don't forget where

you saved the file!

Above image credit: RZA Wu Tang Artwork by [Ali Straightsound](#)

### Converting your image to a vector:

Sometimes you get lucky and can use a free online tool found at [vectorization.org](#) if you are able to save your image to the computer or can find the correct internet URL (internet address) of the image file. Not every file is compatible with this website but it doesn't hurt to try!



**Vectorization.org**  
Free online tool to create vector graphics from your raster images.

Home About Privacy Contact Donate

### Welcome

Vectorize your black and white images into scalable and clear art. Vectorization.org converts raster images into scalable vector files. The output formats include SVG, EPS, PS, PDF, DXF. Save yourself some time and give this free image autotracer a try.

Upload a file:  No file chosen

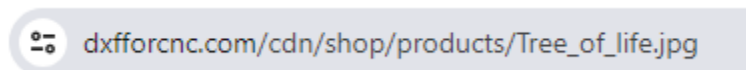
Or enter a URL:

Max. file size for upload is 10 MB.  
Supported file types: jpg, png, pdf, jpeg.  
Max. dimension for images are 3000 x 3000 px.

Output format:

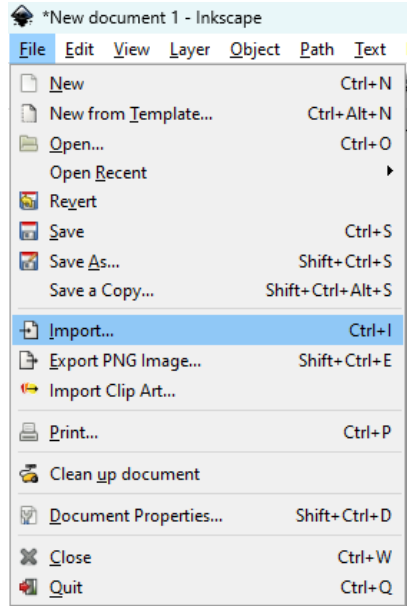
This website allows you to upload your file and convert it to a usable vector image. I recommend you convert your file to an SVG for it's output format.

**Hot tip!** You can also copy and paste a URL link to your image if you can find that link at the top of your browser window. This is not always possible for all photos as many websites will block your ability to see the source URL of their images.



### Inkscape:

Once you have your desired file you need to import it into INKSCAPE. The file can be either a regular pixel raster image (JPG, GIF, PNG, etc) or a vector image (SVG, DXF, PDF, EPS). It may not work every time so you may need to search for a new source image.



You can open up the top right menu in Inkscape and import your image directly onto the “drawing canvas” or you can also drag and drop your file from your download location in Windows Explorer right on the “drawing canvas” in Inkscape that usually looks like a rectangle in the middle of the screen. You don’t really need to worry about keeping your design inside that canvas. Think of the canvas like a piece of paper. You only need to stay inside the lines if you are printing or saving your final image as raster. Vector will know no limits other than how much your computer can handle before it crashes!

The final goal for CNC plasma cutting is to have a drawing that is black and white. Black will represent your metal and white will represent where there is no metal (cut out parts, holes, words, vastness of space, etc). Just getting your picture looking correct is half the battle. We must make sure that the vector pathways are clean, not missing any nodes, no extra nodes, no overlapping vector lines, all final objects and paths on the same layer, it is at the exact size you want cut out, and eventually outputted as a DXF file that the CNC Plasma Cutter can read. Sounds daunting? Go slow and ask for help if you get stuck.

### **Basic Tools you need to learn in Inkscape:**

You should play with



Select and transform objects (F1 shortcut key)



Edit Paths by Nodes (F2 shortcut key)



Create rectangles and squares (F4 shortcut key)



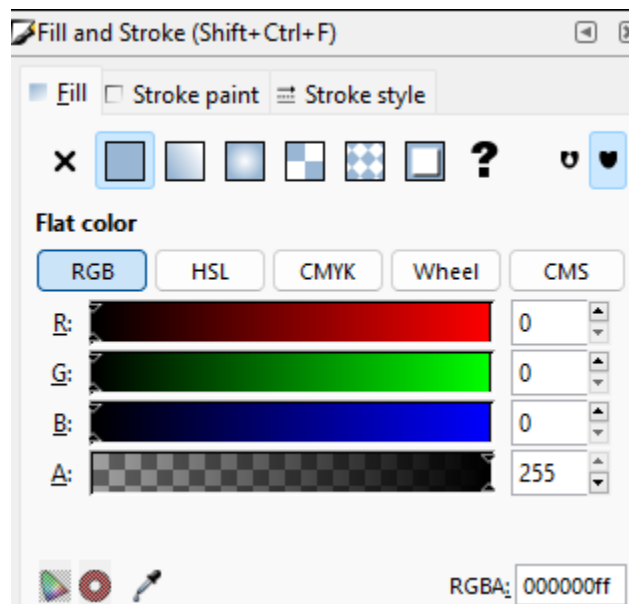
Draw Bezier Curves and Straight Lines (Shift + F6 shortcut key)



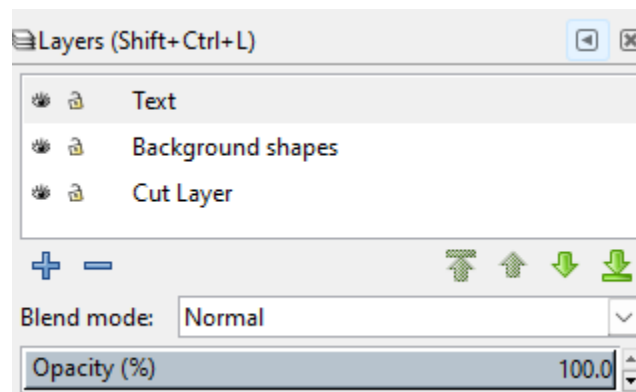
Fill Bounded Areas (Shift + F7)



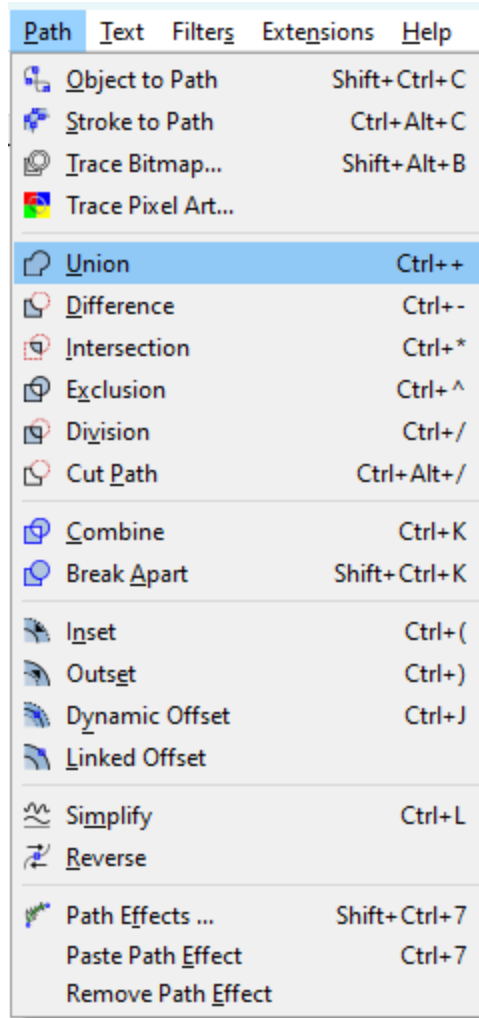
Erase existing paths (Shift + E)



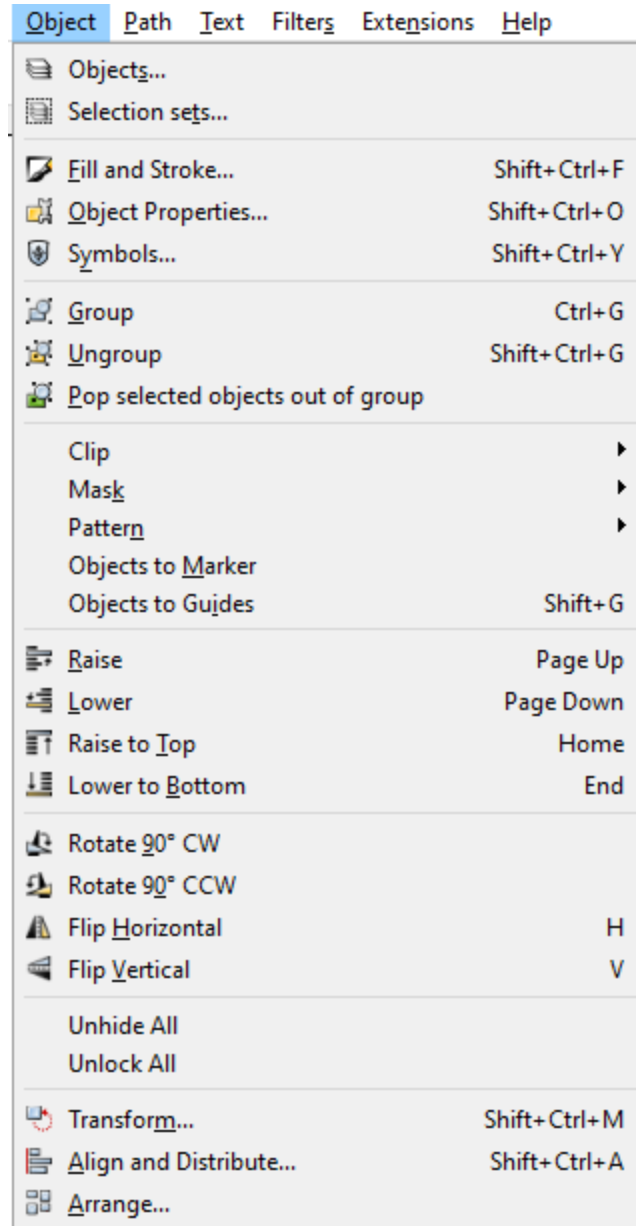
You will want your final image (that represents metal) to be all black which means the RGB values are set to zero and the A value is set to 255



The layers windows allows you to organize your objects and “Stack them” on top of each other in a specific order while working on your design. I like to name each layer so that I can organize my different shapes before I flatten everything into one compound shape and output for the CNC.



The Path dropdown menu will give you many options in manipulating your drawing. You will want to experiment with different options and see what happens! Sometimes “Union” will not work to join objects together. Maybe you have a shape that needs to be cut out instead of combining. Each word has a sample icon beside it to give you an idea of what it does. *Difference* and *Cut Path* have similar icons but will have a different result. You may need some trial and error while working. Don’t give up!!

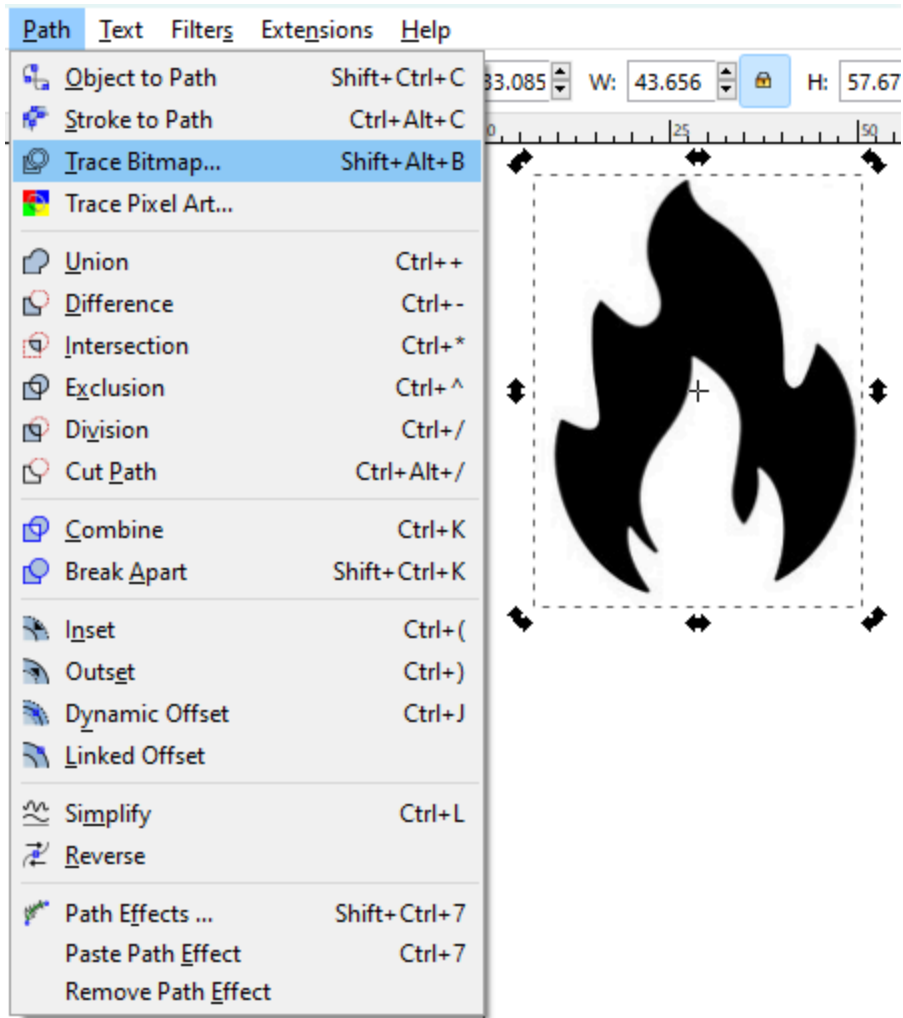


As you progress through using different tools and experimenting with them you may need to manipulate your drawing using some of these tools found in the Object dropdown menu. Fill and Stroke is very useful. *Fill* is the inside color of a shape. *Stroke* is an outline around the shape. For CNC design you should only have one or the other and not both!

### **Using Inkscape to trace a raster image and turn it into a vector:**

If your downloaded image is a JPG, GIF, PNG, or anything with pixels you will need to turn it into something that is made up of nodes, pathways and a series of coordinates that “looks like your image”.

Use the select tool (F1) to click on the raster image and after it is highlighted click on the “Path drop down menu” and choose “Trace Bitmap”.



If your picture is already black and white this is easiest but it can work with images with multiple colors. For this sample I'm using a simple black and white image with simple lines. Check that your settings match the screenshot.

Brightness cutoff threshold: 0.450

Colors: 2

Scans: 2

Checkmark on Smooth

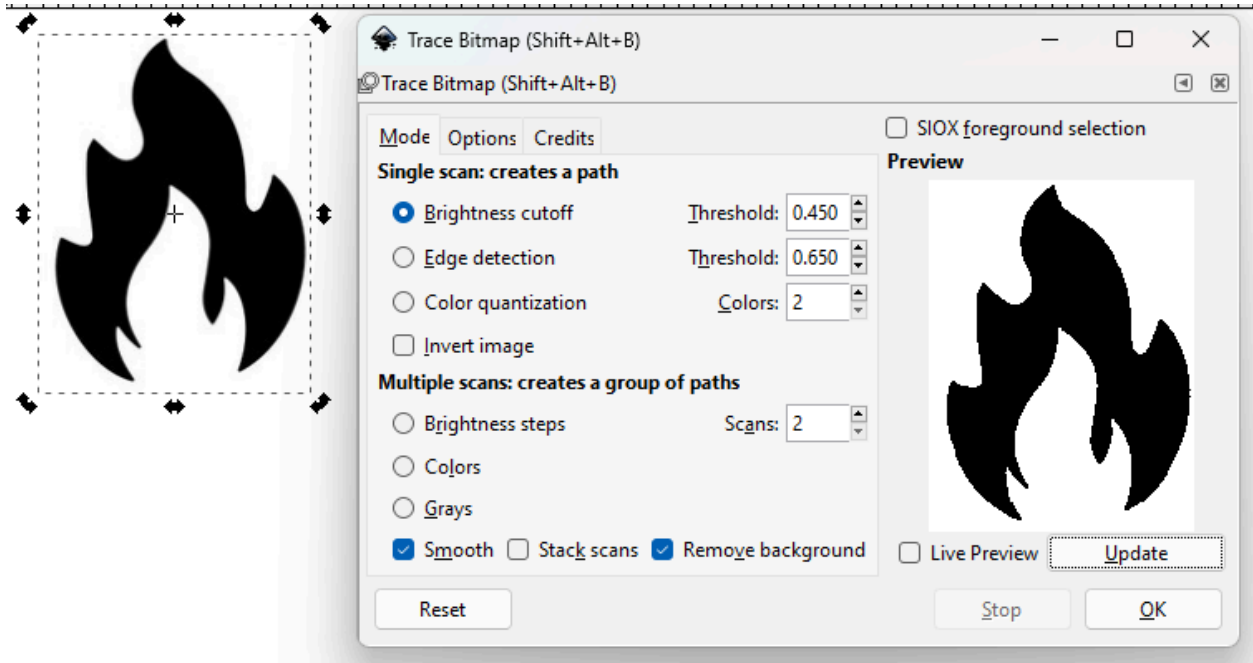
No checkmark on Stack Scans

Checkmark on Remove background

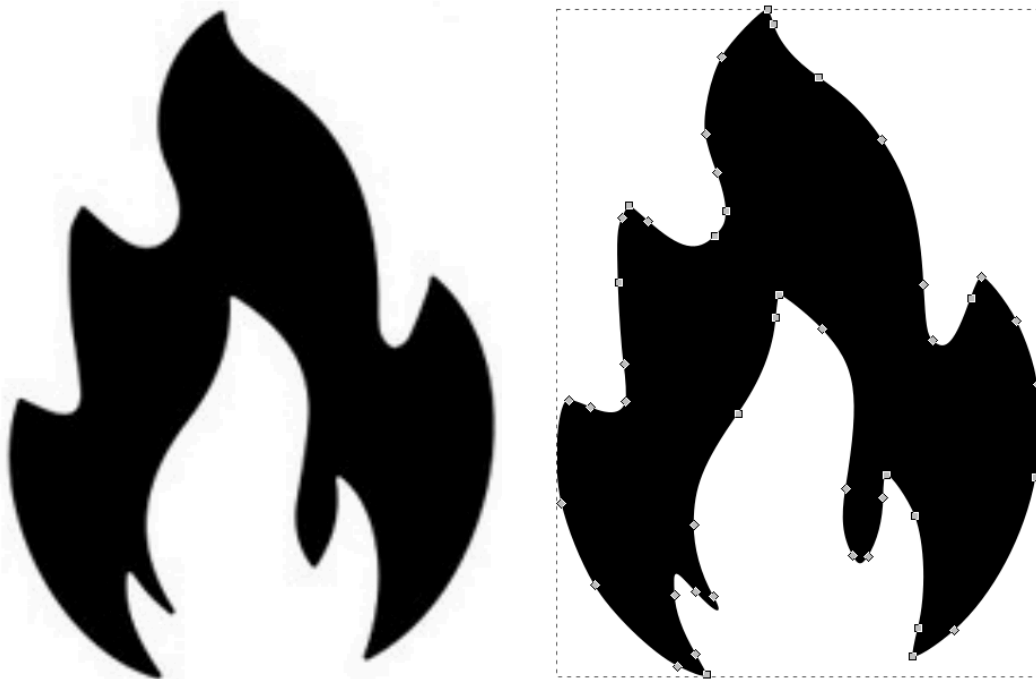
You can sometimes use the preview window and press update to give a simple example of what your converted image might look like.

Click "OK" on the bottom right of the window and then press "X" to close the window






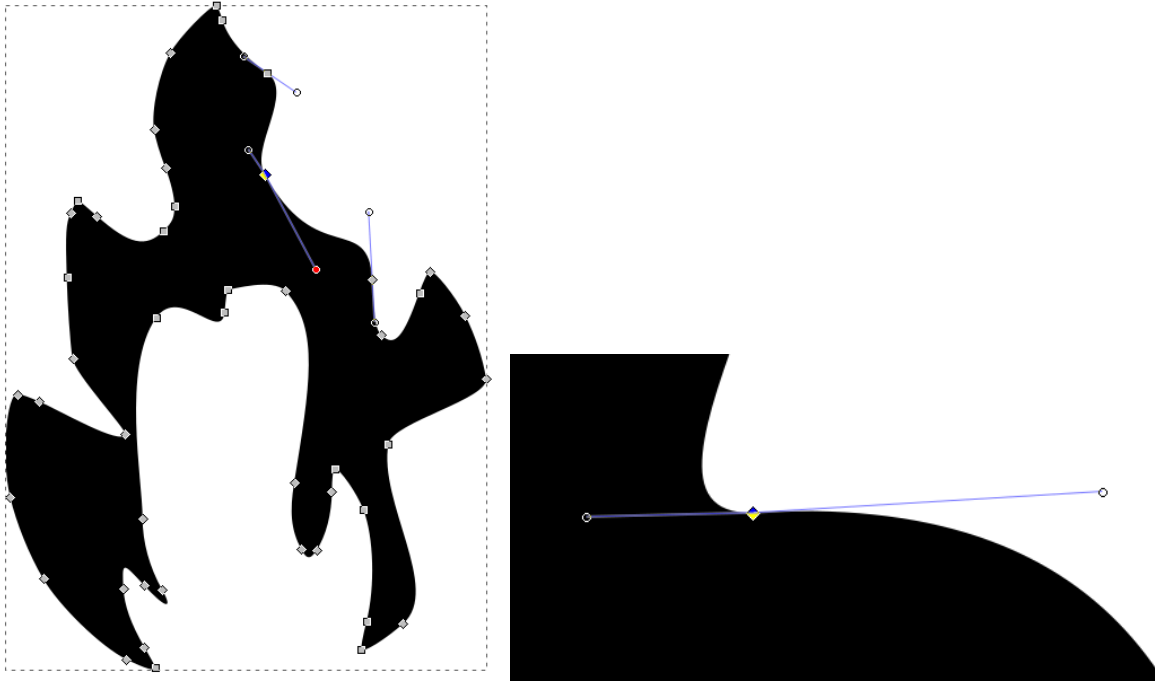
You will need to use the Select tool (F1) to drag and drop your image away from the original raster (pixel image) so you can edit it.



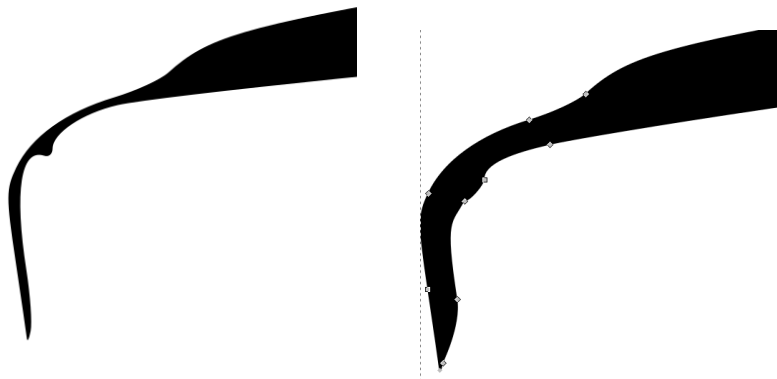
If you double click your newly converted image a bunch of dots will appear around its outline if everything works correctly. These dots are called Nodes and they are basically coordinates of the different parts of a pathway that creates what you see as an image but what the CNC

machine will see as where it needs to travel and cut. You can use the  "edit path by nodes

(F2) tool to move nodes around. If a node is part of a curve or arc shape you can also manipulate the angle of the curve by grabbing the ends of the “handlebars” and moving them around or stretching them. The below image no longer looks like a flame because I got carried away and started moving the nodes around randomly and stretching the “handlebars”.



So why would we need to move the nodes around if we already like the way the image looks? Sometimes the resulting shape will be too small for the CNC to cut out so you may have to “thicken up” lines that are too thin by moving nodes around. While a laser cutter could cut out the left image precisely our CNC Plasma Cutter may burn up the thin metal



If you have lots of shapes and holes and text there is a good chance that your final project will fall apart if lines and shapes are “floating” inside other shapes. An easy example would be the letter A or O or P or R . If the plasma cutter tried to cut out the black part of the letters then the inside white part would work if you just wanted a solid block of those letters cut out separately.

# A O P R = working!

If you wanted a rectangle piece of material with those letters cut out from inside then you have to modify the inside “floaters” of each letter so they aren’t “floating in space”

**A O P R = won't work!**

You need to edit each letter or shape or object so that the black parts are all connected in some way. I drew new black parts to connect everything to look like stencil lettering. You would need to fix any floaters in your drawing regardless of if they are shapes, artwork, letters, etc

**A O P R = working!**


## A picture example:

Here is an image I have already started editing but isn't yet finished.



The black would be metal and any white areas would be cut out. This image has over 15 issues where the inside shapes of the letters and parts of the car would fall out and not be attached.



I used the  rectangle tool and drew shapes to connect the “floating parts”. I used red in the first image to see where the parts were and then turned them black to see what the final shape would look like. But... it isn't done yet. The CNC machine won't see the 2nd image like we do. The CNC software will read the two rectangles as separate and not cut out the shape correctly due to the overlapping shapes. We need to connect the shapes and combine them so there is are not multiple pathways overlapping each other.

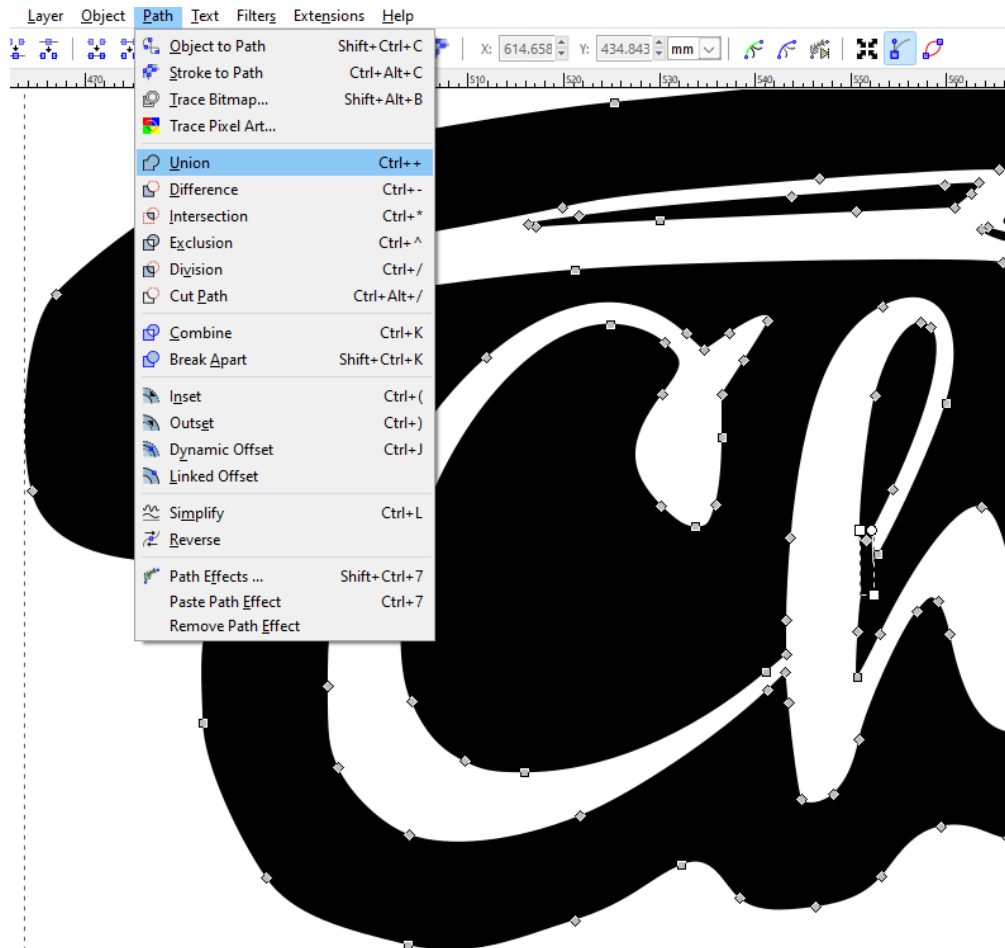


**One way to combine your overlapping vector objects with the Pathway UNION Method:**

Use the *select tool* and click on your small new shape that is bridging the parts together. Hold down the shift key and *click on your big shape* so that you have both of them selected. You can't select all the shapes and do this. Unfortunately you have to do it one object at a time using this method. The below image shows dotted lines around both shapes (including active nodes) to illustrate that they are both selected.




With both selected go to the “Path” drop down menu and try “Union”



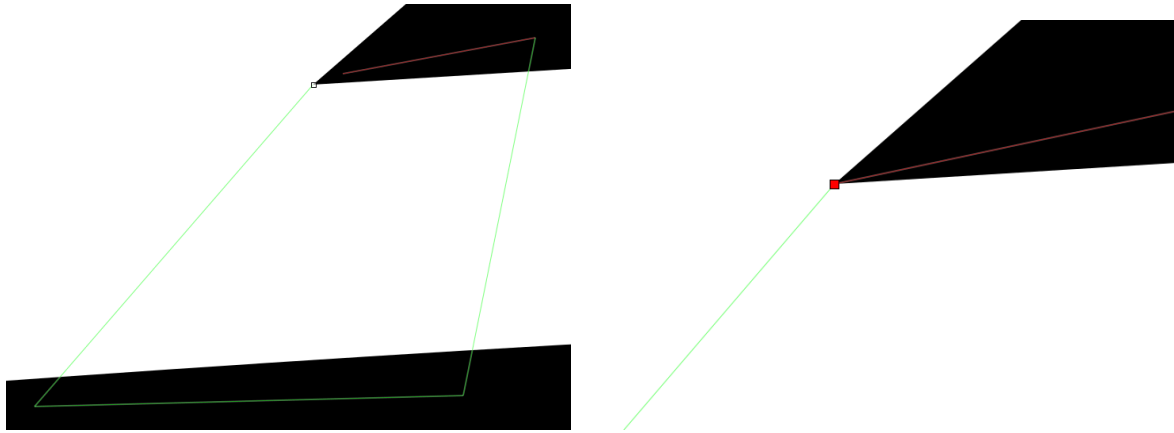
In the below image using “Union” has joined the rectangle to the letter H in the word Chevelle. It is no longer a floating part or a rectangle. You will need to do this *in some way* to every part of your drawing that has a floating shape.



Some areas that need to be joined may not work with a rectangle tool. One of my other favorite

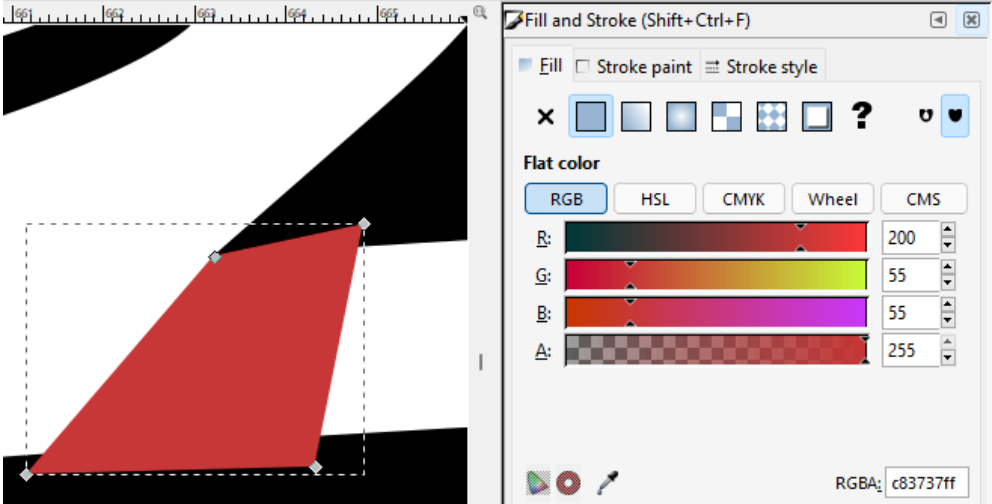
tools to draw weird shapes with is the  Bezier Curves and Straight Line Tool (Shift + F6). With this tool you will click where you want to drop a node to create a custom shape with either straight lines or curves.

Straight lines are easy. You just click, move your mouse, click, move your mouse, and eventually you **have to** finish the shape by joining back to where your first node was dropped. If you do not join your first and last node together you will have an incomplete shape and may need to start over on that shape. If you are unhappy with a shape click each node individually and press the delete button on your keyboard until the shape and loose nodes are gone.



Above: straight lines to create a bridge between the two existing black shapes. The right image shows the final dot attached to where I clicked and put the first node.

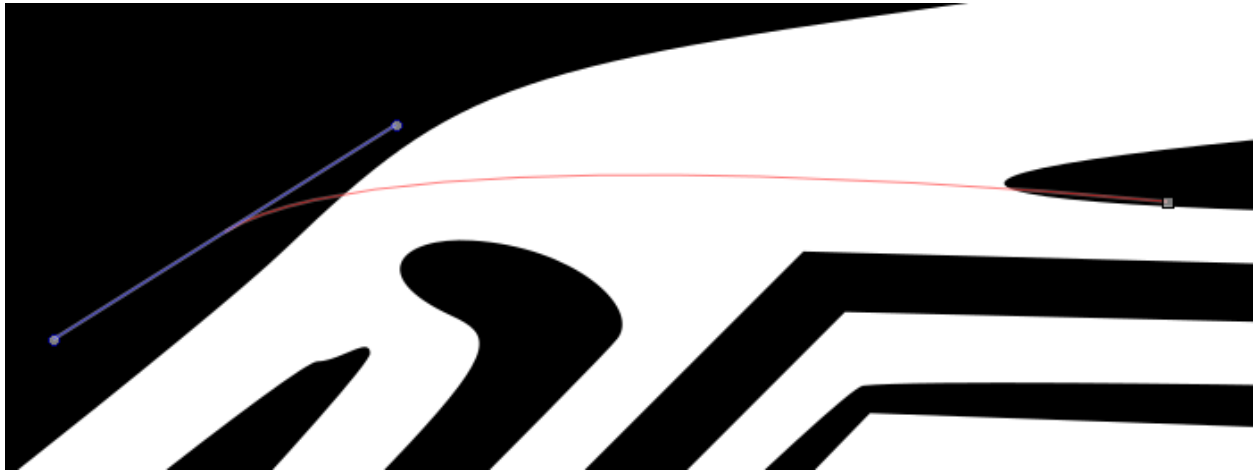
Below: I made the shape red first to make sure it was what I wanted and then made it black.



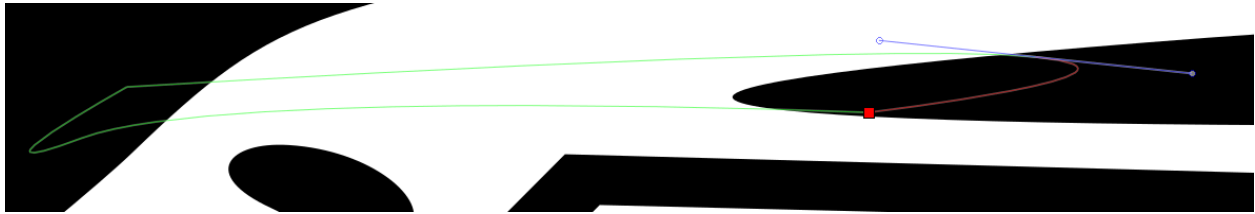
After making it black I selected the new shape, Shift + click and selected the main shape, Path dropdown menu, and then Union. Below you can see the joined shape and the new nodes that help to create the pathway that the CNC machine can follow.



To make a curve with the “Bezier Curves and straight line tool” you click and put down your first node and if you want a curve or arc you click your 2nd node location but do not let go of your mouse button yet. Hold the mouse button down and move your mouse around to see the resulting arc. Once happy with the arc or curve, let go of your mouse button and continue to where you want to drop your next node.

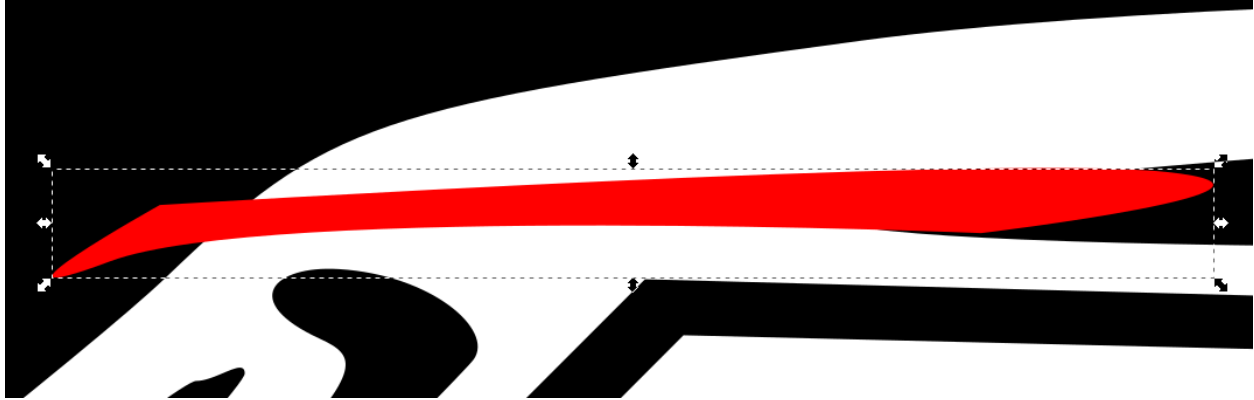


Above: On the right is the first node. On the left is the new arc node and handle-bars which you can edit later.



Above: you can see where the last node and the first node connect together.



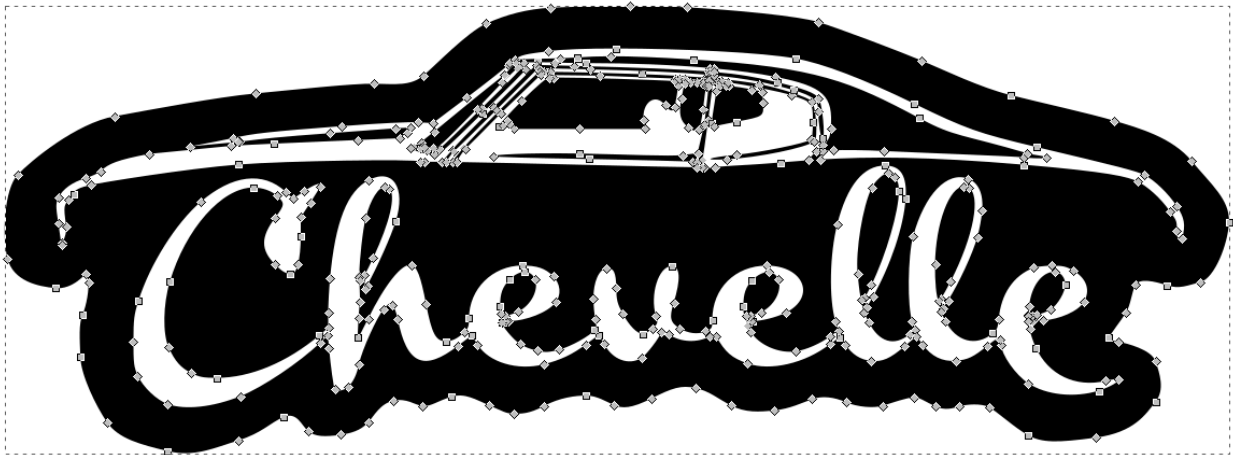


Above: I filled the new shape with red (no outline stroke) to see better using the “Fill and Stroke Window” (shortcut keys: Shift + Ctrl + F) or using the color palette at the bottom of the screen (see image below).



The two shapes are joined together using the Union method and after that I moved a few nodes around to make it look better and thicker using the “Edit Paths by Nodes” Tool (Shortcut: F2). If some lines are too thin they won’t cut well on the Plasma Cutter

Below: I went through every spot that I could find to join any floating pieces together using the rectangle method and the bezier curves tool. I carefully zoomed in and looked at every single part to make sure there were no nodes floating around that didn’t belong and everything is now connected into one “compound shape”.

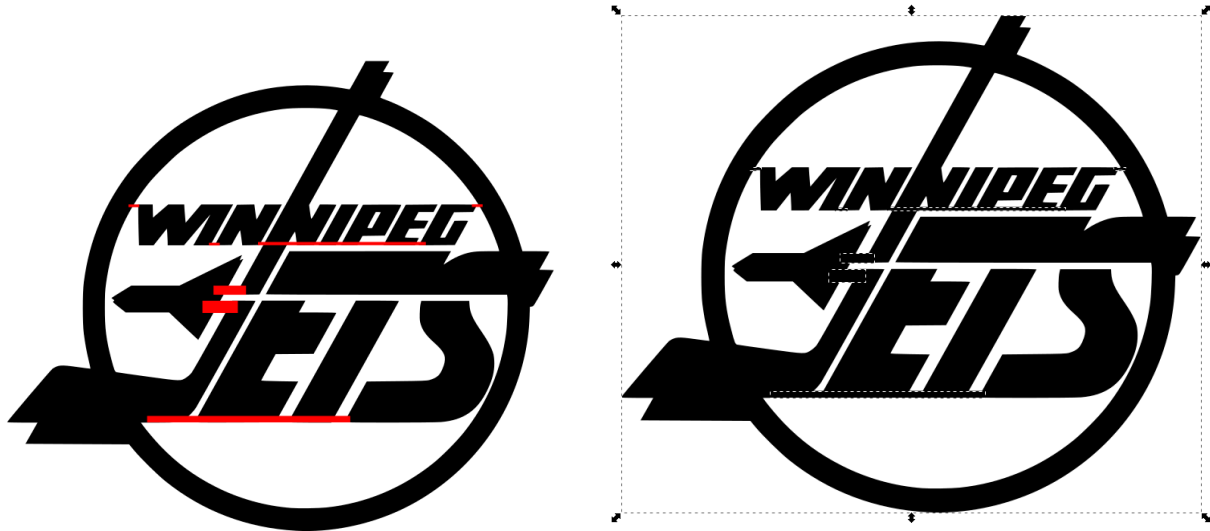


Aside from final sizing this image might be ready to cut out on the CNC. Due to the many small details this design would be best suited for a large cut or a very fine plasma cutting tip otherwise many of the small details would be burnt up by the plasma cutting tip or may not have enough space between for the G-code to even be generated correctly without errors messages (that is a whole other demonstration). Our two options are a 45amp fine tip with 16g sheet metal or 65amp tip with 10 Gauge (1/8" thick) metal.

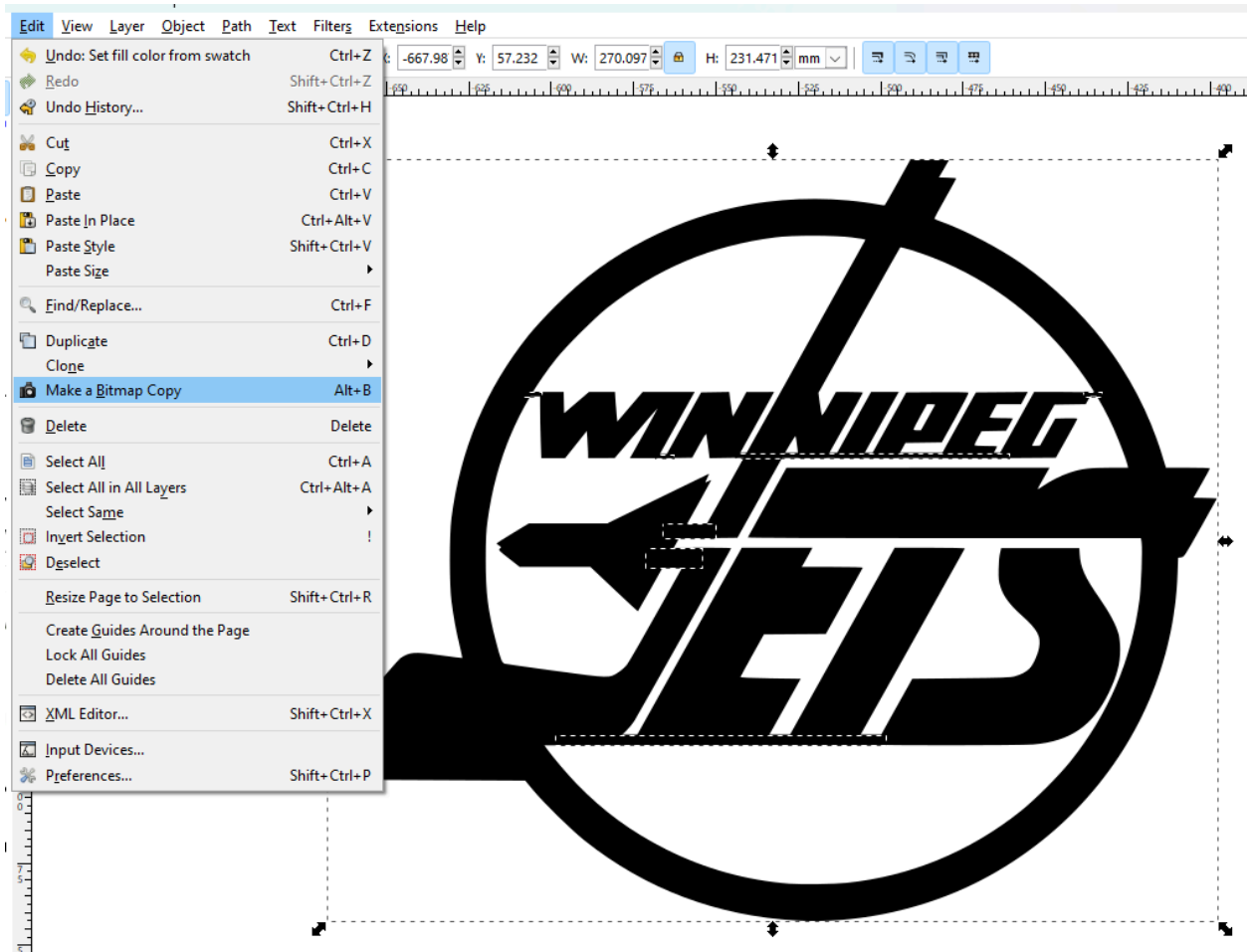
**Easy “Trace Bitmap Method” to join all your new “bridges” or shapes together that doesn’t involve joining pathways with the Union method:**

If you are impatient and don’t like the “path union tool” another method that works to join everything together after drawing your rectangles or curve shapes involves making a “BITMAP COPY” of your design and then converting back to a vector by way of the “Trace Bitmap” method.

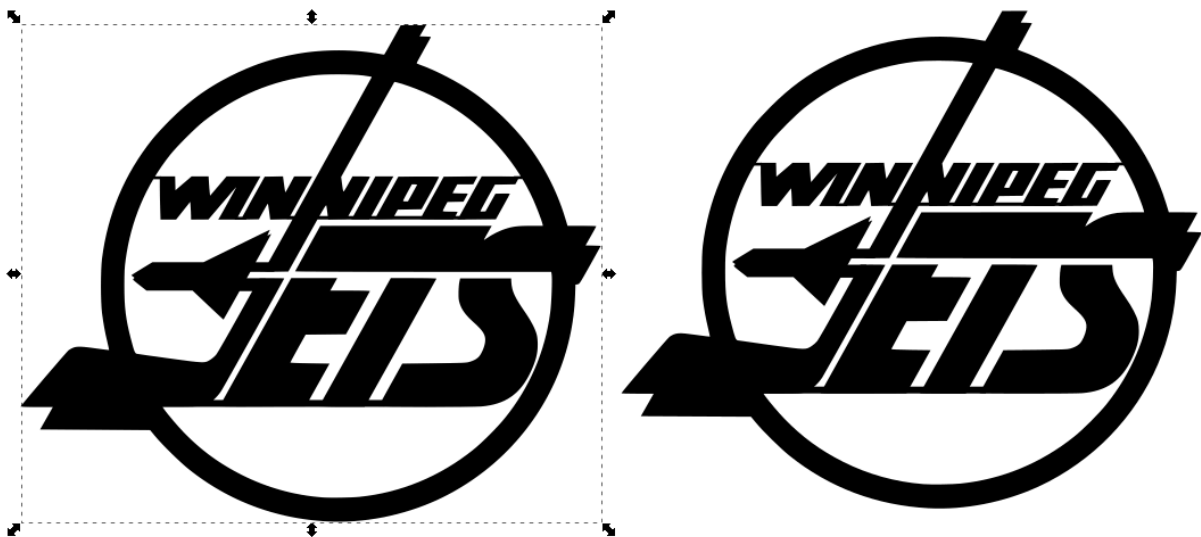
Step 1: Create all the needed “bridges” for your design and make everything black.



Above: The left drawing shows the rectangle bridges to join all floating parts together with red fill. The right image shows everything black and selected together.



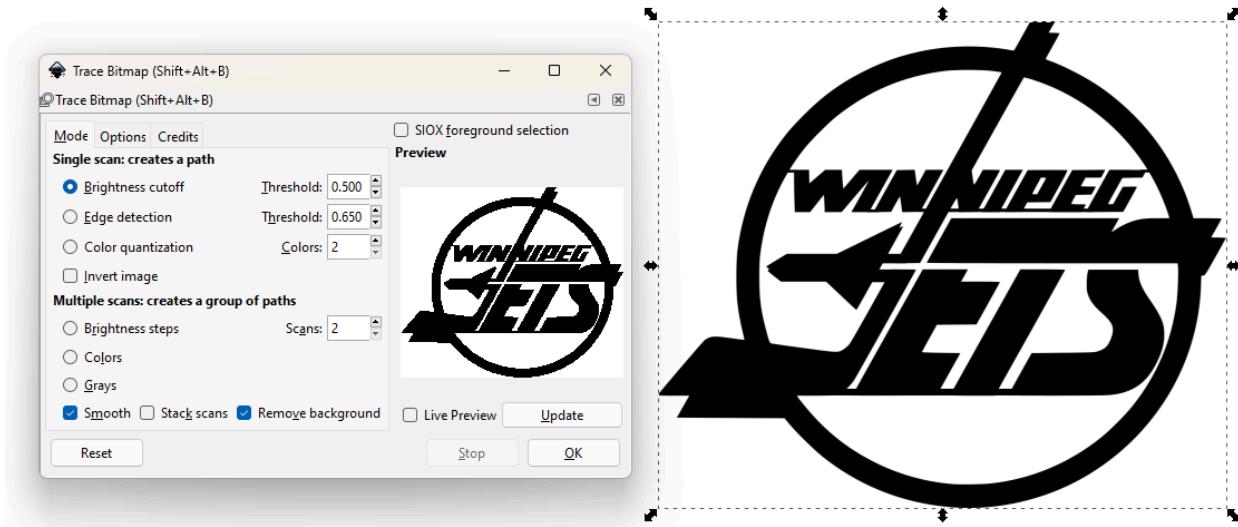
Once all the shapes are selected together go to the “Edit Dropdown Menu” and find “Make A Bitmap Copy”. This will take your selected vector drawing and objects and turn it into a raster image (like a JPG, PNG, GIF, etc).



I like to click and drag the new bitmap copy away from the original. They look the same from far away but when you zoom in you can see that the left image is made up of pixels (below image)



Above: It's ugly and pixelated but all the parts are now together. Next step is to convert it back into a vector image using the "Path drop down menu" and "Trace Bitmap"



Above: As previously shown using the Trace Bitmap Function I hit the "OK" button after checking my settings to convert the raster image into a vector image.



New vector image (on the left). Raster image in the middle. Right image shows before the bridges were joined together.

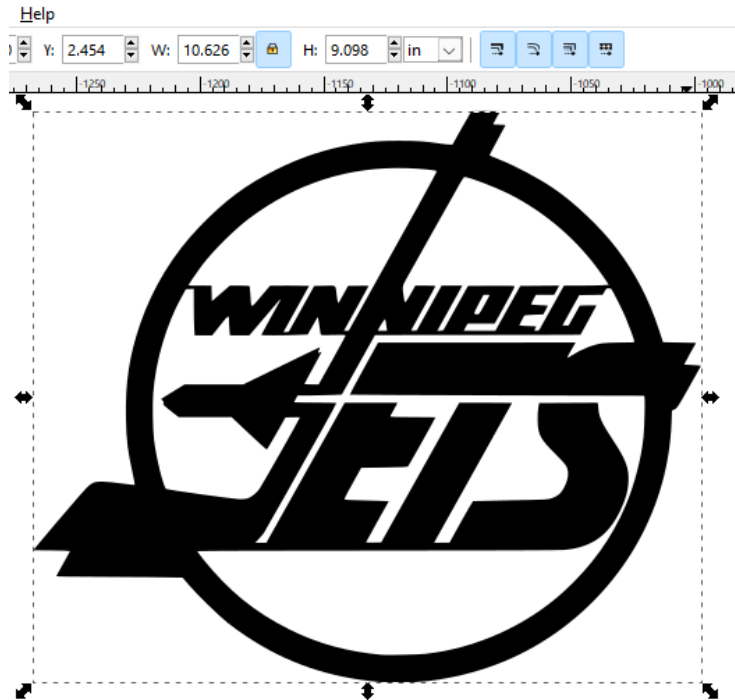


Above: Source image and the final vector design. In the future I would spend more time on this specific image to make it multi-layered and separate the original red and blue parts into 2-3 layers of metal for a 3d final layered metal design that can be welded together vs the above flat one piece design that would be cut out of one piece of metal.

**Exporting vector files that the CNC Plasma cutter software can read:**


The final output file needs to be saved as a DXF file but you should double check the dimensional size of your drawing before this.

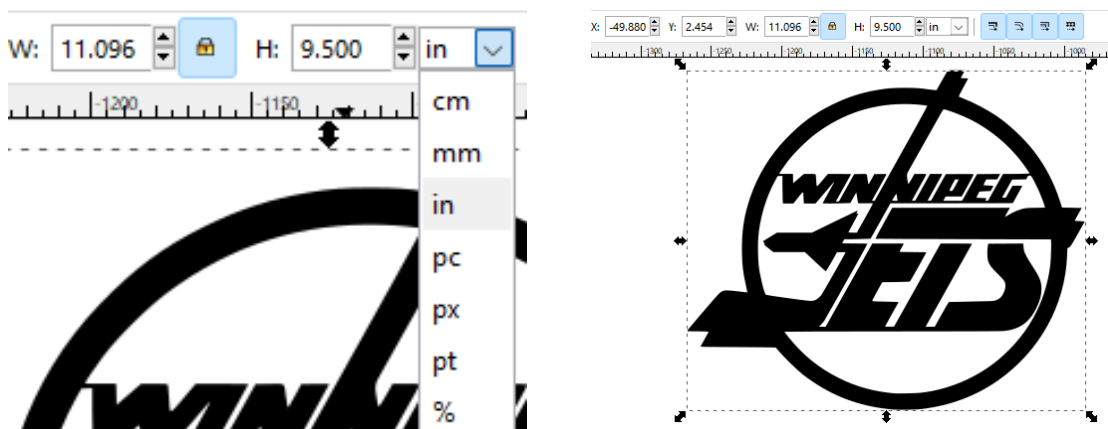
The CNC software measures everything in Imperial (inches) so we need to check the size of our drawing to ensure it is the correct dimensions we require as part of our design process. To do this you select your vector image and you can see the coordinates and size on the upper toolbar. The below selected image shows a width of 10.626 inches and a height of 9.098 inches for it's dimensions.



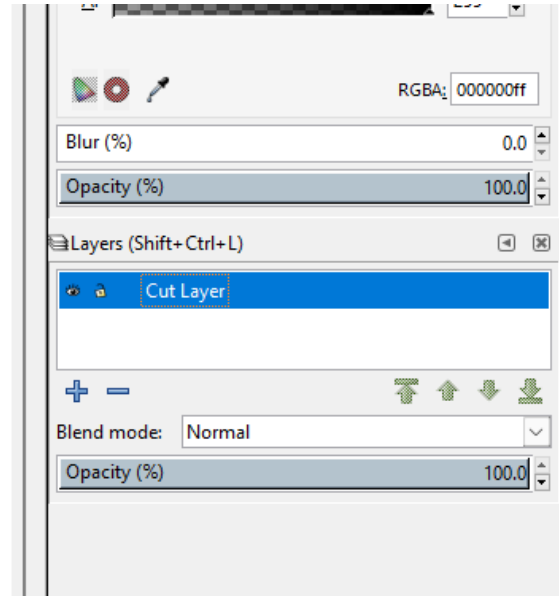
If we need to change the size of the design in order to fit a specific piece of metal you can change the numbers to get your desired size.

If you have a piece of metal that is 10 inches in height by 15 inches in width you will want to leave a small border around the image of at least 0.5cm or ¼ inch. I would change the size of

my design to have a height of 9.5 inches and make sure to have the  ratio lock button on so that a change in height will also change the width with the same ratio you started with. A squashed or stretched image does not look professional. The resulting width of 11.096 will work because I have enough material from my 10x15 inch piece of metal. **Tip:** Double check the units of measurements so they match our required units of measurement.

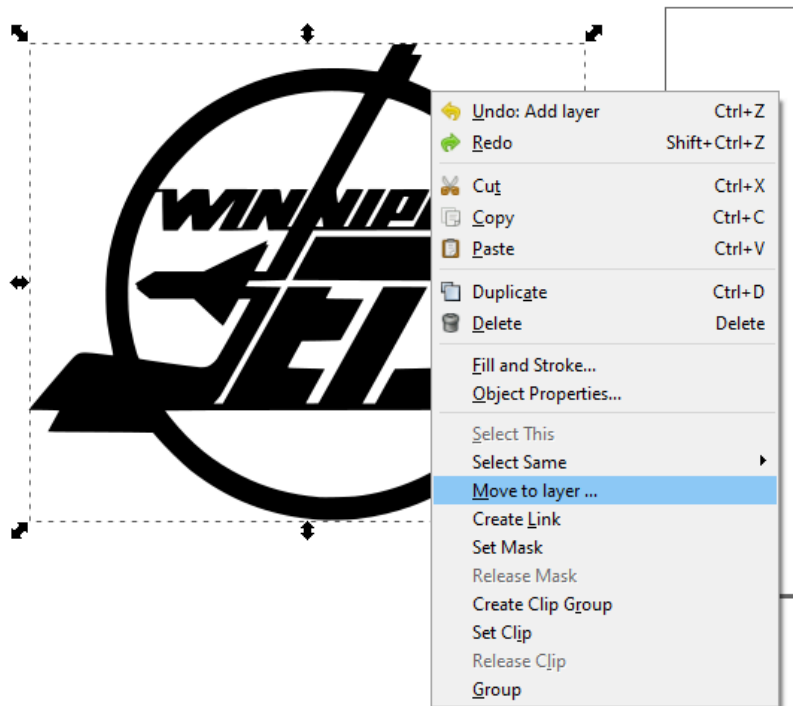


The final check before saving is that the shapes you are wanting to cut out are all on the same "Layer". If your drawing has different pieces that are not in a compound shape together on the same layer the file will not show up correctly in the CNC software. If I have multiple layers in an Inkscape document I make sure everything has been moved to one layer for my final image. I will usually save a backup before merging layers and sometimes have more than 5 different backup files as I work throughout the project.



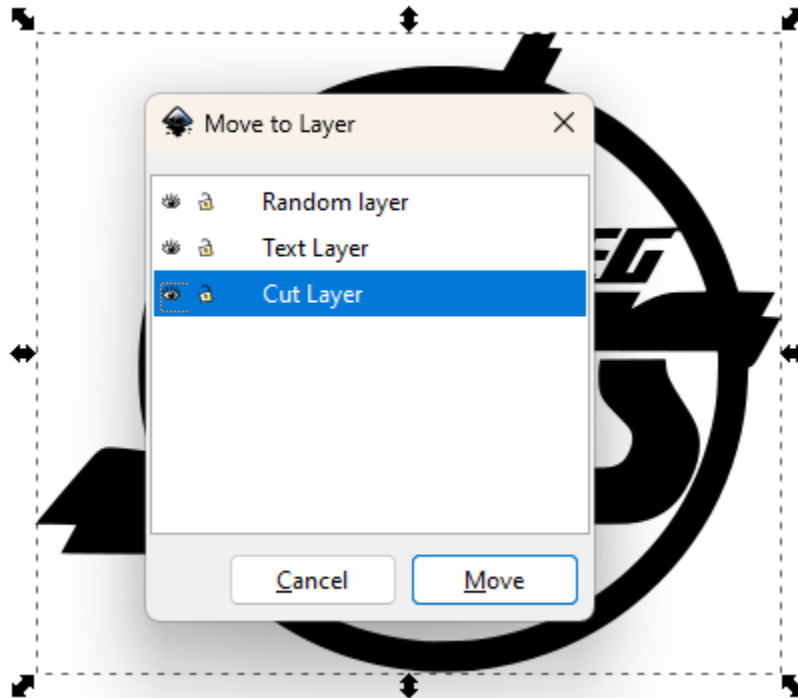
**To move an object onto a different layer:**

Select the object (picture, text, etc) and right click on it to bring up this pop up menu. Choose "Move to layer".



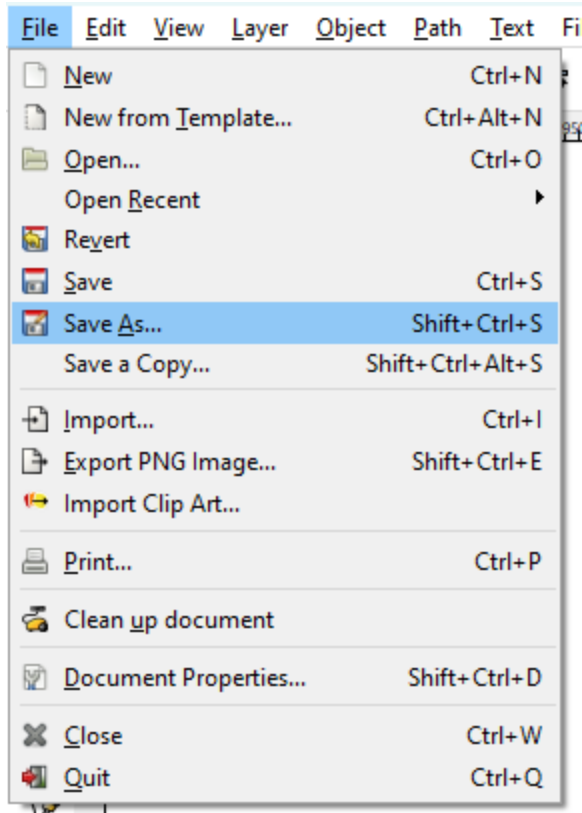


A window will pop up asking you to choose which layer you want to move that object to. In my case I like to have a layer called “Cut Layer” as my bottom layer. I move everything that I actually want cut to the same layer. Sometimes I leave my original drawings or other objects that I don’t want cut on my other layer. These could be things like layout lines or notes I have left to myself in the file.



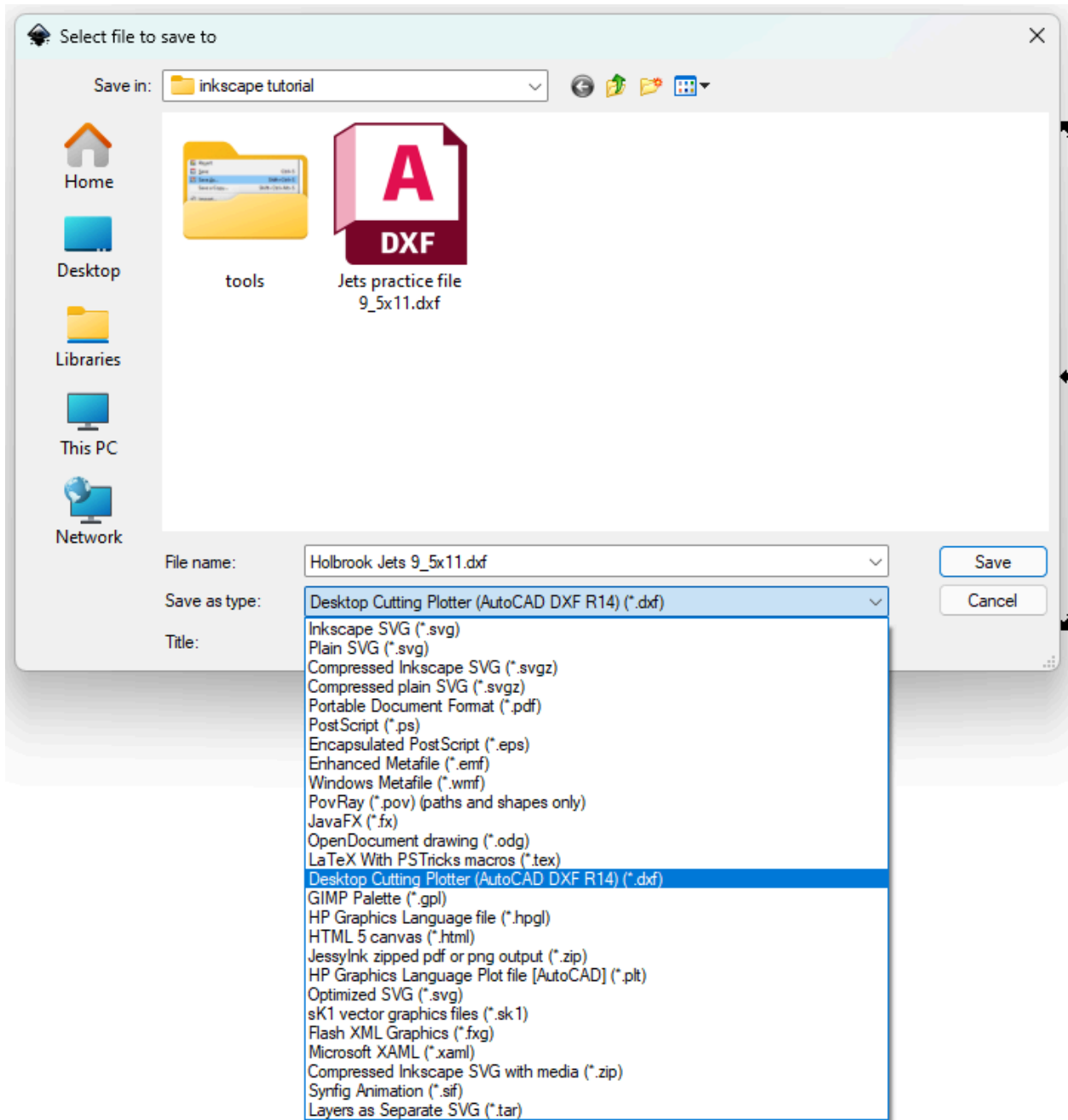
**Saving your final drawing as a DXF file that the CNC software can read:**

Once you are happy with where your object(s) are then save your file as a DXF by going to “File” and then “Save As”



Name your project file [Your Name] [Project name] [Size] . dxf

I am doing **Holbrook Jets 9\_5x11.DXF** to help me find the file later and include the size of material required to cut it out. I use 9\_5 to describe 9.5 inches in my file name and this makes it easier in the CNC software when setting up options in the SheetCam TNG software. You may choose to use a different file naming convention to fit your personal needs including things like the date, version number, project title, etc. In the classroom I ask students to use my naming convention in order for me to organize things on our shared computers and USB flash drives.



Now that our file is saved we need to transfer it to the CNC computer