The following Instructional documentation is specific to vcarve settings needed to render gcode for use with the crossfire Pro 24 X 24 CNC. It assumes that you already have an understanding of working with vector images and creating profile toolpaths in vcarve. For tutorials on installing and using vcarve, go to our resource page: https://www.cedarvalleymakers.org/plasmacnc
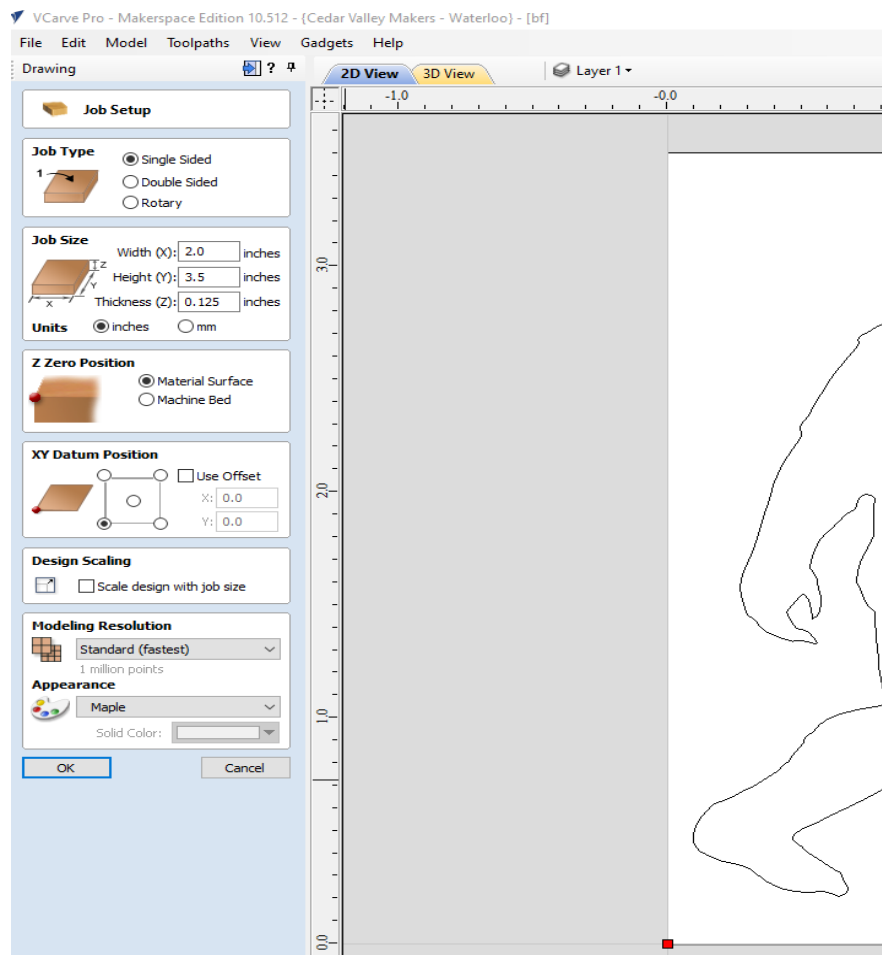
Using Vcarve with the Plasma CNC:
Vcarve is mainly used for CNC machines with spindles. We have programmed a post processor to get the right gcode for the plasma cnc, but there are settings that are set to defaults or are not applicable and should be ignored.

Job Set-up for Langmuir 24 x 24:
Width and height in job setup should be set not greater than 24 x 24 bed size.

Thickness is not important in job set-up, since cutting material thickness will be determined by gas air, and amperage settings at the plasma torch.

XY Datum position should be set depending on where you want your X0, Y0 in relation to your vector image.

Pictured left will start the vector in the lower left corner as shown on the graphic with the red dot.

ToolPaths for the Plasma CNC:
Vectors are cut by creating a vector tool path. Important settings include:

Cut depth: The depth of cut is determined by the plasma torch settings, but vcarve requires a depth of cut. We use .01 and look to ensure the tool path only makes a single pass.

Tool Database

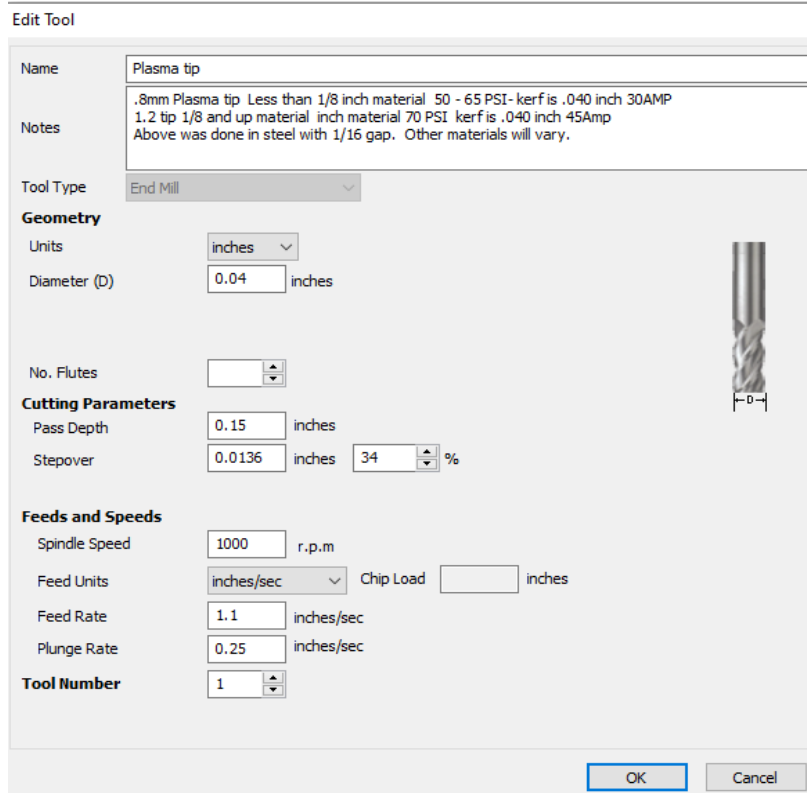Material Steel

Plasma Torch
  Plasma tip
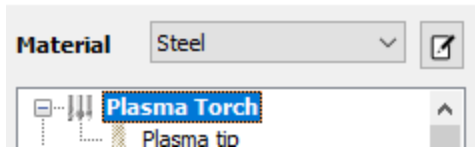
Select the Plasma tip from the tools database. In the notes you will see recommended tip sizes for different thickness of material. These are the recommended torch setting that will be set-up on the torch itself.

Other settings in the plasma tip tool of note:
Diameter .04 is the kerf cut in steel we found with both the .8 and 1.2mm tips. The kerf may vary depending on material and torch settings, so it can be adjusted(D) as needed.
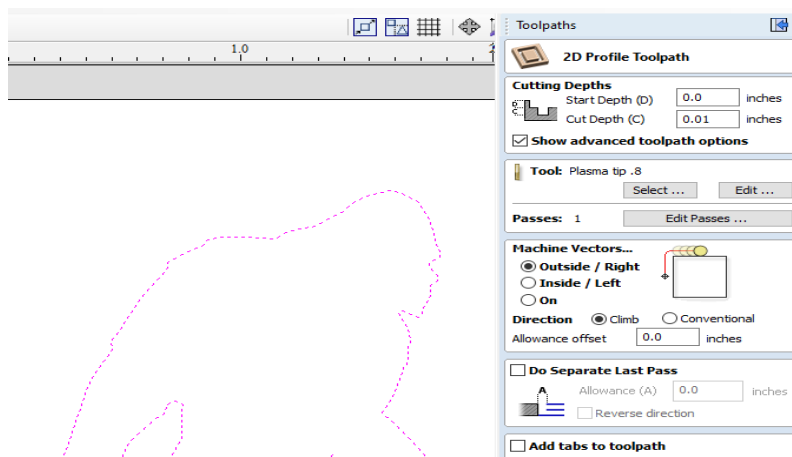
Pass depth is set at .15. This is arbitrary, but the depth of cut on tool path should be less than pass depth, so that there is only a single pass.

Edit Tool

Name Plasma tip

Notes .8mm Plasma tip Less than 1/8 inch material 50 - 65 PSI- kerf is .040 inch 30AMP
1.2 tip 1/8 and up material inch material 70 PSI kerf is .040 inch 45Amp
Above was done in steel with 1/16 gap. Other materials will vary.

Tool Type End Mill

Geometry
Units inches
Diameter (D) 0.04 inches

No. Flutes

Cutting Parameters
Pass Depth 0.15 inches
Stepover 0.0136 inches 34 %

Feeds and Speeds
Spindle Speed 1000 r.p.m
Feed Units inches/sec Chip Load inches
Feed Rate 1.1 inches/sec
Plunge Rate 0.25 inches/sec
Tool Number 1

OK   Cancel

Stepover applies to raster and pocket tool paths, so not applicable for plasma operations.

Spindle speed cannot be zero in vcarve, but not applicable for plasma. The gcode created is ignored by the plasma cnc.

Feed rate of 1.1 inches/sec (equates to 70 in/min) has been a good speed for the tips and materials tested to date.

1.0

Toolpaths

2D Profile Toolpath

Cutting Depths
Start Depth (D) 0.0 inches
Cut Depth (C) 0.01 inches
☑ Show advanced toolpath options

Tool: Plasma tip .8
Select ...   Edit ...

Passes: 1   Edit Passes ...

Machine Vectors...
⦿ Outside / Right
○ Inside / Left
○ On
Direction ⦿ Climb   ○ Conventional
Allowance offset 0.0 inches

☐ Do Separate Last Pass
Allowance (A) 0.0 inches
☐ Reverse direction

☐ Add tabs to toolpath

Select climb for direction

TABS - Tabs might be necessary when nesting parts to prevent them from moving in the z height when cut, potentially jamming the torch

head, or otherwise to keep small parts attached to the sheet. When used, two tabs per part .0625 length should be sufficient and relatively easy to break free and grind or file.

Leads are important with Plasma cnc. We have seen good results with a circular lead of .1 inches. Leads can be reduced for small inside diameters but the holes will not be as clean as a drill, so you will want to plan to drill if you want holes that can be tapped or a more exact specification. After calculating the toolpath, run the simulation to ensure cutting operations are performed in a single pass with appropriate leads and tabs. If everything looks as it should;

Click Save toolpath

Select Langmuir Crossfire (inch) from the post processor list.

Laguna Swift Cut3D (mm) (*.mmg)
Langmuir Crossfire (inch) (*.nc)
Larken (inch) (*.nc)
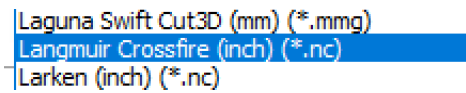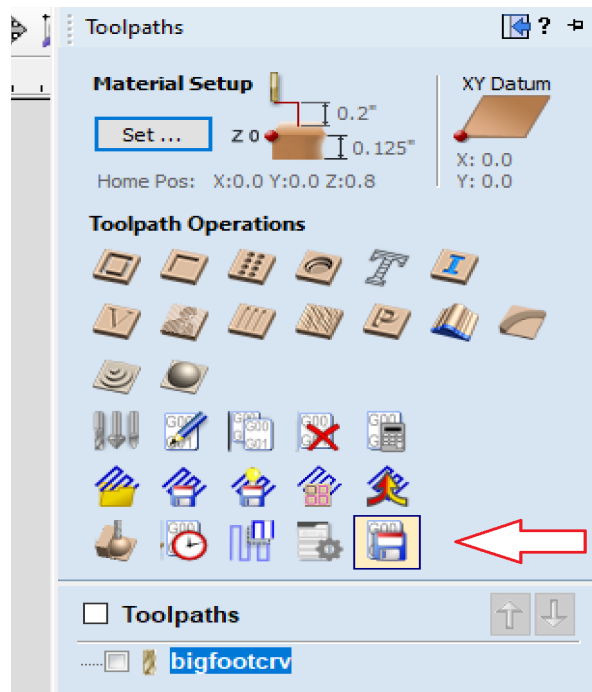
Click on your created toolpath and then save the toolpath to a usb drive, which can be accessed using the crossfire application, which interprets the gcode for the plasma cnc.

If it is your first time using the weld room, or the cnc at CVM. Request an orientation by e-mailing info@cedarvalleymakers.org.