# Guide To F-Engrave and Easel

By Phil Johnson

First there are 2 way to design for F-Engrave, Text only in F-Engrave (Engraving), and using an image or DXF file designed outside of F-Engrave (V-Carve). I prefer using Inkscape to design a black and white .PNG file.

First lets setup F-Engrave settings to ensure it will be imported into Easel for sending.

Setting->General Settings   -   Make yours match this exactly. Then Hit "Save" by configuration File.
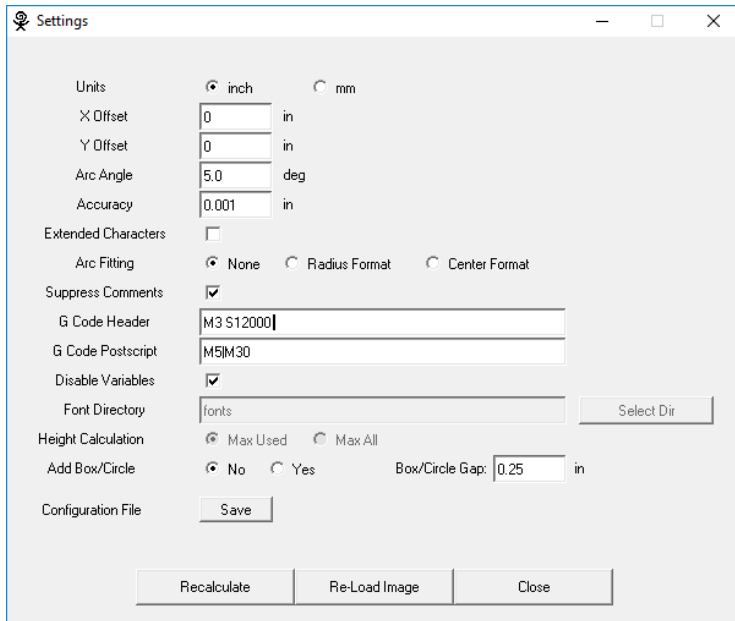
Next lets setup the files main size and origin (WORK ZERO) locations. Start by opening up a DXF or image file. I find it's easiest to use high quality .PNG files so that's what this example will show.
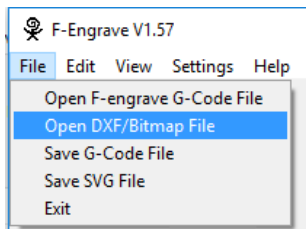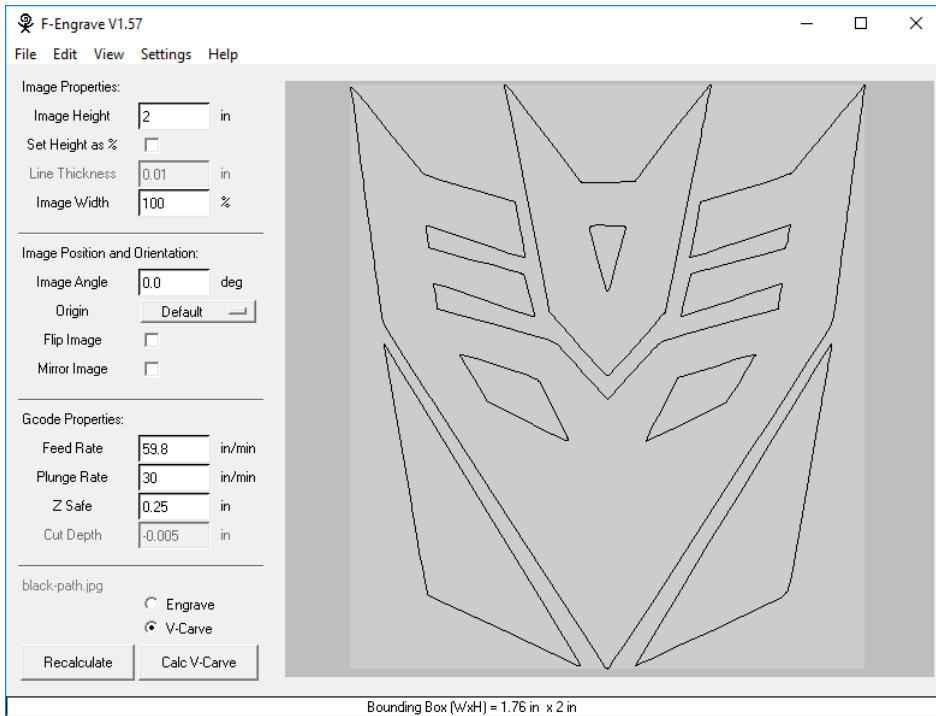

You will find the option to see and import .PNG files only after you select "Slower Image Files" from the drop down menu.
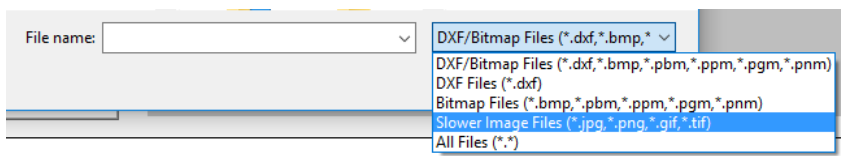
I'm going to pick an image I traced in Inkscape.

There's a few things to notice and pay attention to. Image height, Origin, Feed rate, Plunge Rate, Z Safe, and weather or not you have Engrave or V-Carve selected. We'll be concentrating on V-Carving.

Image height is how tall the actual carved image will be. It's important to note that by default, BLACK is carved, WHITE is not. So the blank white area around an image you open only affects the Origin, not the image height.

In this example, we're carving the image 2 inches tall.

NOTE THE ORIGIN!!!! There should be a green and red L in the lower left corner symbolizing our WORK ZERO mark or where our bit will be zeroed. When "Default" is selected, it is the lower left corner of the image INCLUDING ALL BLANK AREAS. In this case, it's outside of the carved image extents.

Click on the bar and select "Bot-Left". The red and green lines now appear.
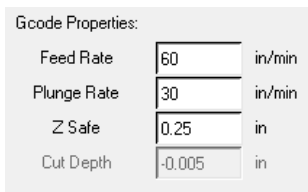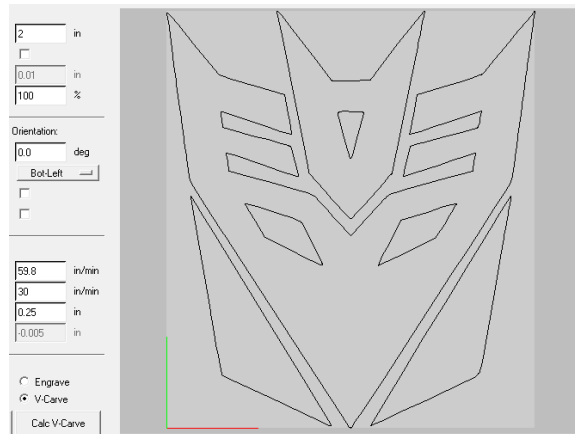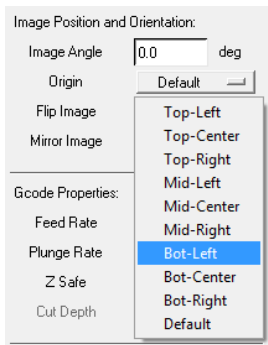
Gcode properties has to do with your machines capabilities, your bits capabilities, and your method of clamping and carving. I will give you recommendations, but it's up to you to figure out what works best with your equipment. I use the Whiteside 1540 60 degree .25" diameter V bit. I use 60 in/min Feed rate, 30 in/min Plunge Rate. Your Z safe should be high enough to clear any clamps if clamping inside the carve limits.

Lets move onto V-Carve settings

Next lets setup F-Engrave V-Carve settings. Some settings will be changed based on the carving bit you use.

Setting->V-Carve Settings

1. Specific to your bit

2. Cut depth limit controls the maximum depth the design will be cut to. If left at zero, a wide area could cut deeper than the material which can cause damage. Make sure to check the Max Cut Depth to ensure that doesn't happen. If a Cut depth is needed, it must be a **_NEGATIVE_** NUMBER.

3. The Multipass Cutting settings control the depth per pass including the last pass or "Finish Pass". Most bits, the rule of thumb is no deeper per pass than half the diameter of the bit. For V bits, it should actually be less when using a larger V bit like a 1/2" diameter. I recommend using 0.08" for Max Depth Per Pass which needs to be entered as a **_NEGATIVE_** NUMBER. The Finishing pass can either match the Max or be less than. You may want to use a very small number for the finish pass for a smooth finish, but it also may add time to the carve.

More on the next page->

AREA FOR NOTES:

V-Carve Settings — [_] [□] [×]

Cutter Type	● V-Bit
	○ Ball Nose
	○ Straight

V-Bit Angle	60	deg
V-Bit Diameter	0.25	in
Cut Depth Limit	-.125	in
Max Cut Depth	-0.125	in
Sub-Step Length	0.01	in
Flip Normals (V-Carve Side)	☐	
Prismatic (Inlay)	☐	
Prismatic Overcut	0	in
Plot During V-Carve Calculation	☐	

Multipass Cutting

V-Carve Finish Pass Stock	0.08	in (Zero disables multipass cutting)
V-Carve Max Depth per Pass	-0.08	in

Cleanup Operations

Cleanup Cut Diameter	**4**	0.25	in
Cleanup Cut Step Over		40	%
Cleanup Cut Directions	**5**	☑ P ☑ X ☐ Y	
V-Bit Cleanup Step	**6**	0.005	in
V-Bit Cut Directions	**7**	☑ P ☑ X ☐ Y	

9 [Calculate Cleanup]
10 [Save Cleanup G-Code]
11 [Save V Cleanup G-Code]

8 [Calculate V-Carve] [Close]

Cleanup Operations should be at least calculated on each design to check if it's needed. A box will pop up if there's no cleanup Gcode to save. You must first Calculate V-Carve (see 8) BEFORE you calculate the Cleanup. Cleanup Operations are only used and needed when a Cut Depth Limit is used (see 2 previous page). Skip this page EXCEPT FOR HITTING THE CALCULATE V-CARVE BUTTON (see 8) if no Cut Depth Limit is used.

4. These setting are for the Flat end mill cleanup. The diameter listed here will dictate the amount of work the V bit has to do during the cleanup. I would stick to 40% step over.

5. There are 3 choices for cleanup cut directions. P is perimeter meaning it will trace the loops in the design even around curves. X and Y are your X and Y axes. I would always select P and one axis. Selecting all just adds time with little to no benefit.

6. is the V bit cleanup step over. I would use 0.005 and leave it. You are using a fine point to clear material that the flat bit cannot reach. It needs to be fine movements to leave a clean area.

7. This is the same concept as 5 above.

8. After your settings are all entered for ALL sections above, hit Calculate V-Carve. It will pop up another dialog box with a status bar. It may run very fast for small carves or slow for large ones. I've had very complex designs with thousands of loops take 12 hours to calculate. I've had other small ones take about 2 seconds.

9. after the Calculate V-Carve is done, click the Calculate Cleanup button.

10. if there are no larger areas that the diameter of the flat bit can fit into (see 4), then when you hit Save Cleanup G-Code, it may say "you must calculate Cleanup first" or similar. If you already hit the Calculate Cleanup button, then there's nothing that the flat bit can reach. Move on to 11. If there is a flat bit Cleanup G-Code file, make sure to name the file so you know what exactly it is. I usually use the following naming convention

Main Gcode file = Transformer_MAIN_60degree

Flat Bit Cleanup = Transformer_FlatClean_025 bit

V Bit Cleanup = Transformer_Vclean_60degree

11. Click the save V Cleanup G-Code. If the design is limited at all by the Max Cut Depth, there will be a V Clean file to save. If not, the error saying

10. if there are no larger areas that the diameter of the flat bit can fit into (see 4), then when you hit Save Cleanup G-Code, it may say "you must calculate Cleanup first" or similar will be displayed.

Hit Close when done.
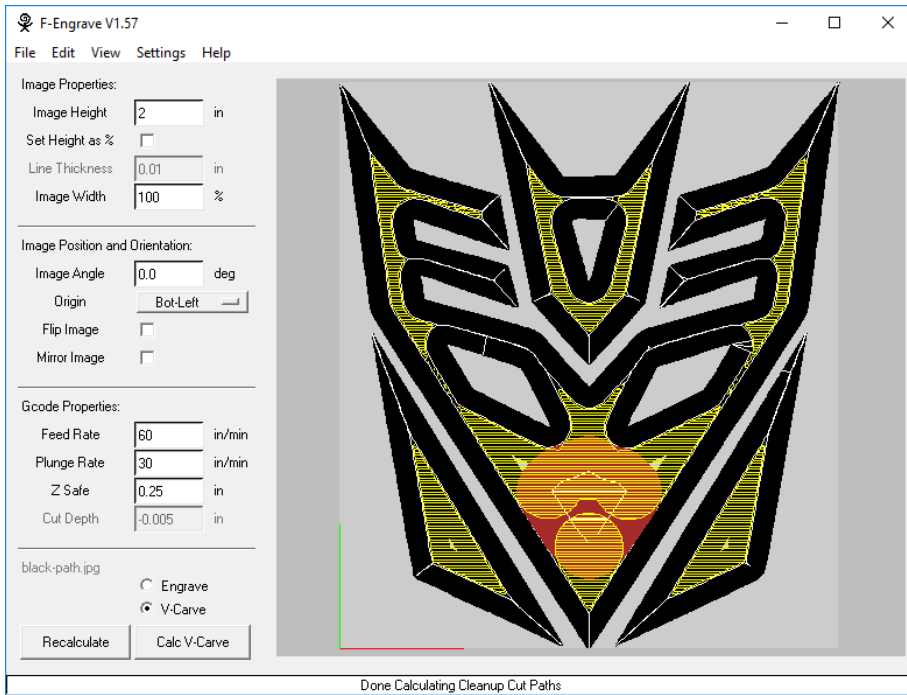
After calculating with a –.125 Cut Depth Limit, the image will look like this. Black is the carved area. White lines are the main V bit tool path. Yellow is the V bit cleanup tool path. And Red is the flat bit cleanup area.
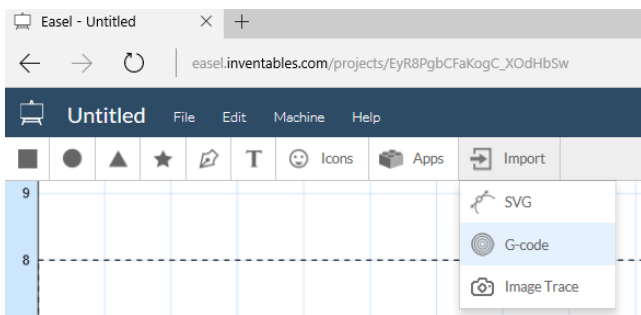
We still have one more tool path to save, the main one.
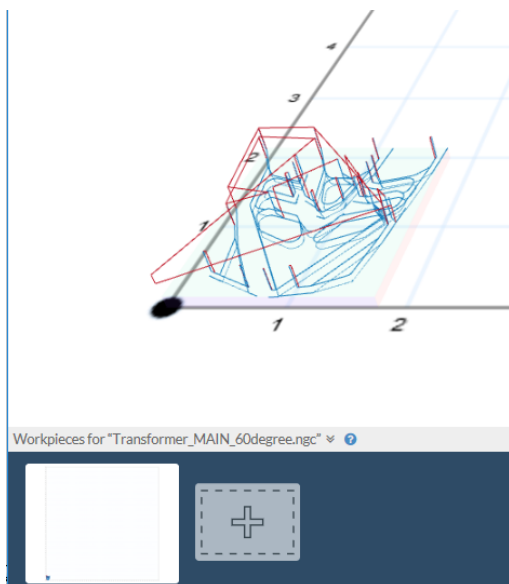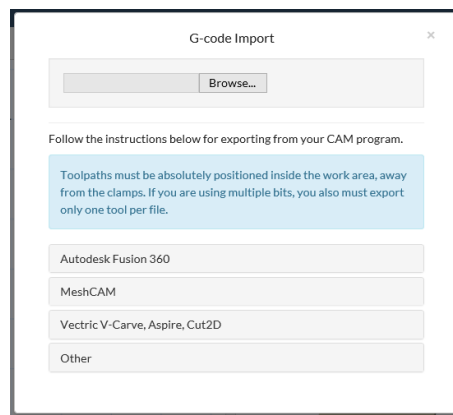
Go to File-> Save G-code File

Save it like in my example on the previous page.

You can now exit F-Engrave and open easel.

In Easel, start a new project. Then Import G-code.

Select the Main file saved before. It should show up as only one page with no "preview" section like there would be in a normal Easel project.

Use the + work page and import Gcode to add the other clean up gcode files.