**V-Carve Tutorial**

1. Open V-Carve and choose create new file
2. V-Carve defaults to imperial units. On the left hand side in the Job Setup menu, set the size of your material. (ex. W: 12" H: 12" T:.75") Z Zero is always to top of the material, left corner. <OK>
   * 1. If you need to access this menu again, it is under the edit tab – Job Size and Position
3. File > Import Vectors > Find your DXF or DWG\*\*\*\*\*you must make the DXF/DWG in CAD with layers\* See beginning of tutorial > OK
4. Under the drawing tool bar on left side, use the Transform Objects/Edit Objects/etc. options to move, center, and duplicate your drawing. Set this up perfectly how you want to machine it. > OK
5. JOIN OPEN VECTORS > Select entire dwg, find select open vectors in EDIT OBJECTS > Set tolerance to 0.5" > OK
6. Remember to save!!!

**TOOLPATHING**

1. Under the toolpaths tab – Show toolpaths tab
2. Material Setup > Set >
   * 1. Z Zero – top left corner > Material Thickness – 0.75"
     2. XY Datum – Bottom left
     3. Don't change anything on Model Position in Material – Gap above model = 0.0"
     4. Don't change anything on Rapid Z gaps – Clearance and Plunge = .2"
     5. Home/Start Position – X = 0, Y = 0, Z = 0.8
3. The two most common tool paths = Profile and Pocket. (anything else, eg engraving will need to be practiced and worked on)
   * 1. PROFILE
        + 1. Start Depth = 0.0, Cut Depth = .25" as an example
          2. Tool > Select > End Mill (.25 inch) > Copy

Rename as End Mill (.125 inch)

Diameter > .125"

Cutting Parameters > Pass Depth 0.125, Stepover > 0.05 and 40%

Feeds and Speeds > Spindle Speed 12000RPM, Feed Rate > 40 Inches/min, Plunge Rate > 15

Apply > OK

Passes > Edit Passes > Ensure 0.125 is selected > OK

* + - * 1. Machine Vectors > Choose Outside/Inside or On
        2. Last Pass – Nothing to change
        3. Tabs – Unless cutting all the way through, nothing to change

If cutting all the way through, Select the Add tabs to toolpath button. (note the depth of cut should go through the thickness of the whole material)

* + - * 1. Vector Selector > Click Project toolpath onto 3D model

Selector > Select open and closed Vectors on the appropriate layer you made > OK

Name your toolpath (ex: pocket eyes, outer contour, etc.)

**OR** you could simply just window all if not doing separate toolpaths.

* + - * 1. Name the tool path appropriately with the size of the bit in the extension, for example: owlouterprofilevbit, or owlinsidepocket.125
        2. Calculate
        3. In the toolbar tool bar, you should be able to select tool paths and do previews.
        4. Close to create new toolpath or to continue on

REPEAT THIS PROCEDURE FOR ALL TOOLPATHS

**SAVING THE G-CODE**

9. Each toolpath with a different bit will need to be saved individually

* + - * 1. Check off the boxes that you wish to use in your tool path \*\*\*\*Remember, different bits all need separate tool paths.
        2. Select the SAVE TOOLPATH icon
        3. Double check all toolpaths are correct
        4. POST PROCESSOR > CamTech CMC3 (inch).(\*.nc) > SaveToolpath(s)
        5. Name your toolpath so that we can correctly ID it and the toolbit, for example: FREEMANOWLvbit.nc or FREEMANOWL.125.nc

Remember your save location (I drive)

* + - * 1. Continue saving toolpaths associated with different tool bits.

**EDITING THE G-CODE (THIS MUST BE DONE OR MACHINE WILL CRASH)!!!!!**

* + - * 1. Find your saved toolpath g-code on idrive and open
        2. You can open it with WORD PAD
        3. Scroll down to the very end of the tool path code and **erase** the final three lines of G-Code

M5

G53Z0

G0X0.0000Y0.0000

* + - * 1. Save your toolpath
        2. Do this for all toolpaths
        3. Copy all your newly edited toolpaths to your USB Memory Device.

**THE CNC MACHINE SETUP**

* + - * 1. Open WinCNC from the desktop
        2. Setup and install material so that it will not move or crash
        3. Install the proper bit into the router (.125", .25" or V-bit)
        4. Using the remote control, move bit to where you would like to set up x=0 and y=0. You need to hit the > SET x/y on both the remote control AND the computer WinCNC.
        5. Moving the bit over with the remote control, slowly jog z down to it just barely touches your material. \* you can use the paper towel method here\*
        6. Hit the > SET z on both the remote control AND the computer WinCNC.
        7. Raise the bit with the remote control that it is not touching your material
        8. On the computer, load up the toolpath you are planning on running on WinCNC.
        9. Manually start the router
        10. Drag the toolpath to about 30% on the computer. You can increase this after but it's always good practice to start slow and ensure everything is running correctly.
        11. Click the green button on WinCNC to start tool path.
        12. Have hand on emergency stop button while tool path is running just in case the tool crashes or does something expectantly.
        13. Ensure the dust extraction system is working as well
        14. Someone must always be present beside the CNC machine while it is running.
        15. When finished, you can run more tool paths or remove piece. If you change the bit for another toolpath on the same part, DO NOT reset x/y. You only need to reset your z depth.

1. Congrats! CNC machining is complete! Paint your little hearts out!

**AUTOCAD DRAWINGS FOR VCARVE TOOLPATHING**

If you are planning on doing several different toolpaths (ex: contours, profiles, pockets, engraving etc), each tool path will NEED to be on a separate layer to avoid future problems on VCarve. You will therefore have overlapping tool paths but that's ok! Use separate colors so that they are distinguishable. DXF 2007