



X-Carve CNC router guide

Revision 1.0RC1
April 2024

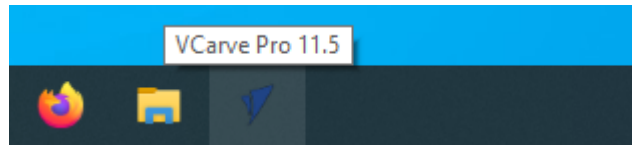
Nolop Makerspace
Tufts University

Step 1: log in to the X-Carve computer

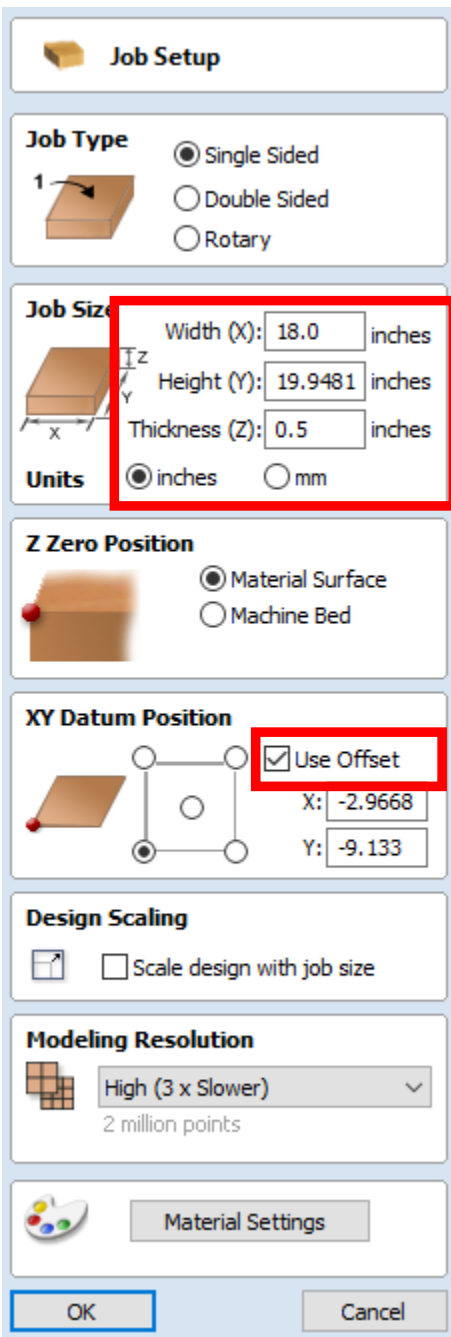
username: nolop
password: xcarve4ever

Step 2: import a DXF file into VCarve

Start the VCarve software.

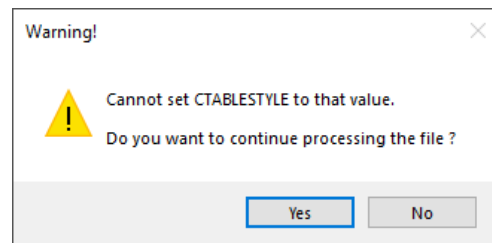


Here is the icon you click.



In VCarve, open a DXF file by navigating to File > Open and selecting your file. You can export a DXF from Onshape or other CAD program.

You might see a warning that says something about "CTABLESTYLE." Just click "Yes."



Nobody knows what this means; just click yes.

Step 3: set up your workpiece in VCarve

Set the width, height, and thickness of your material before it is cut. Uncheck "use offset."

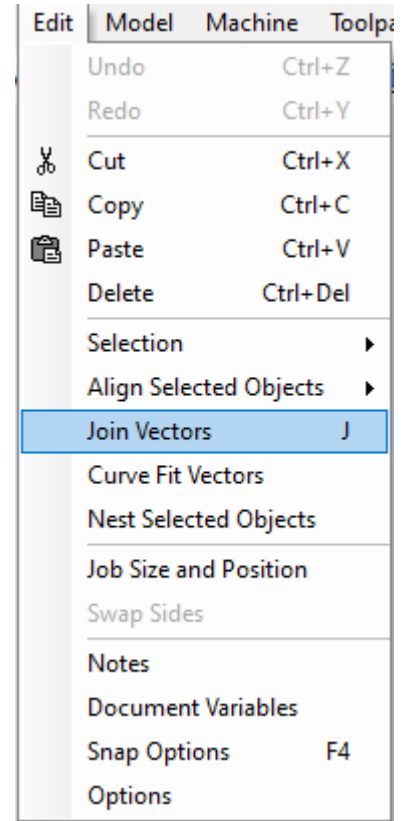
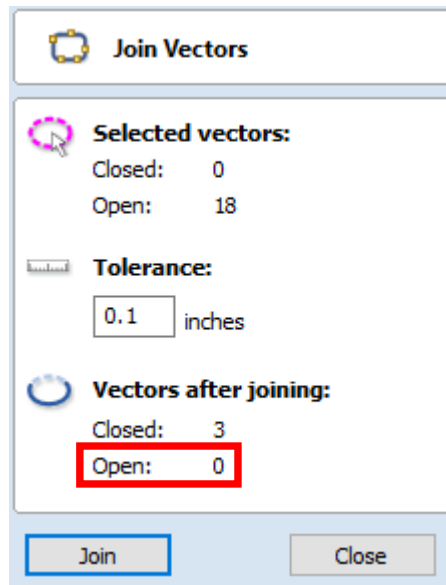
The rest of the default values in "Job Setup" are okay.

Step 3: join open vectors

Your DXF file probably has a bunch of disconnected line segments or vectors. Use the mouse to select all vectors that should be continuous loops in your design. They'll turn pink when selected.

Choose Edit > Join Vectors.

You probably want 0 open vectors when you're done.

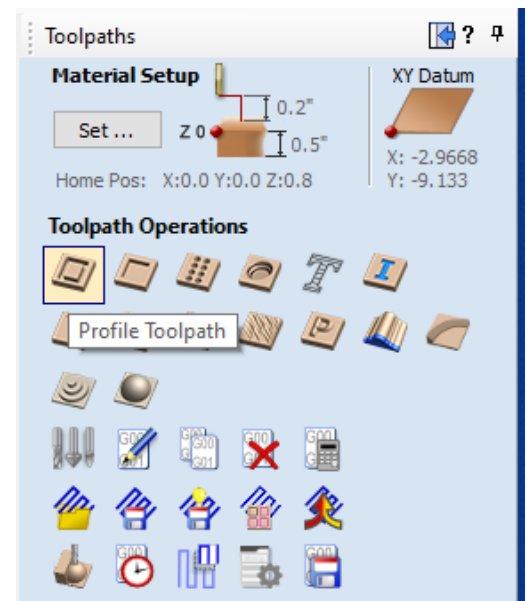
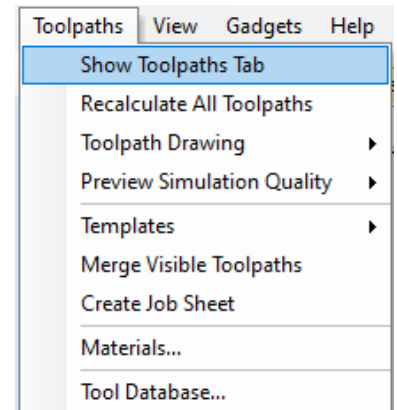


Step 5: create a toolpath

Choose Toolpaths > Show Toolpaths Tab

With all vectors selected, choose the appropriate tool path operation for your project. You should cut holes first, then pockets, then profiles.

If you're not sure what you want, choose the profile toolpath for now.

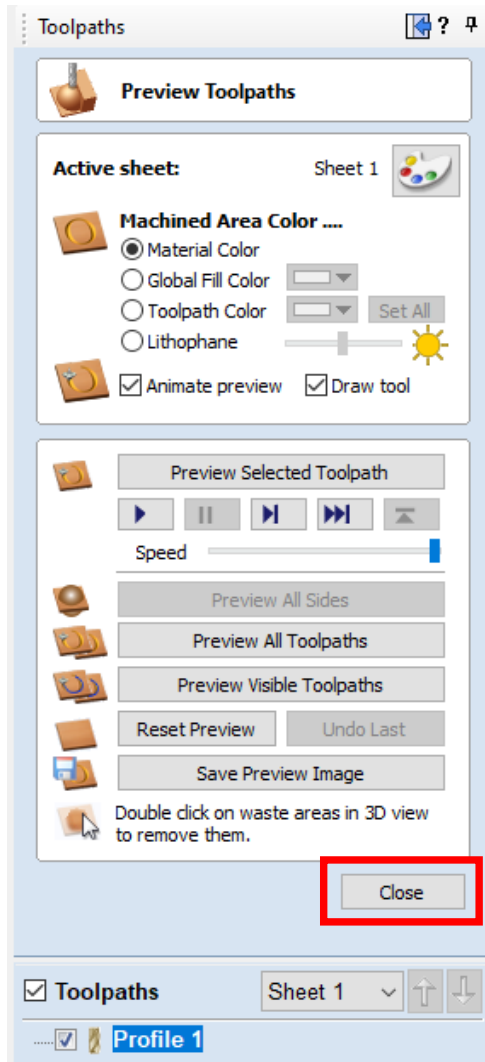


Step 6: set toolpath parameters and add tabs

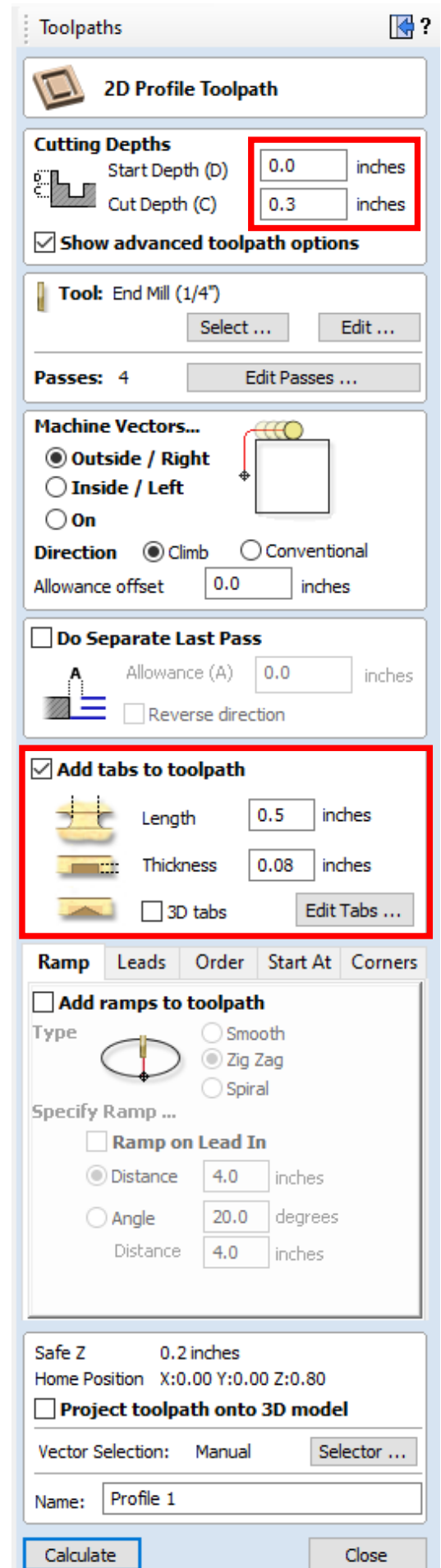
Adjust depth per pass and cutting speed. With a 0.25 inch bit, you probably shouldn't go deeper than 0.075 inches/pass or faster than 45 inches/min. You can finetune the speed in Easel later on.

Click Edit Tabs... to add tabs to your toolpath. The automatically chosen locations are usually fine.

Click "Calculate" to generate the tool path based on your settings which will also show a preview.



After you preview the toolpath to make sure it looks right, close the preview. (You can't get to the Save Toolpath button while the preview is open.)

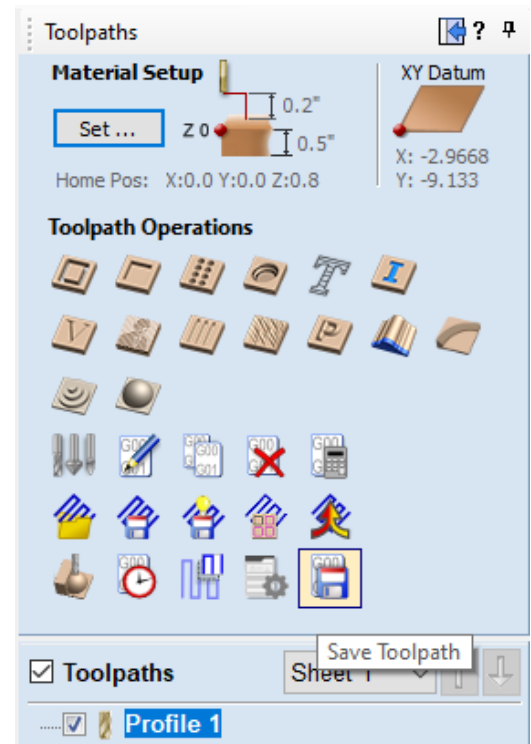


Step 7: save the toolpath as G-code

Save the toolpath as a G-code file. Oddly, you have to use the Save Toolpath button highlighted below.

File > Save will not work.

The file extension should be .nc



Step 8: run the G-code in Easel

Go to <https://easel.inventables.com>, create an account or log in, and start a new project.

Load your .nc G-code file into the new project by clicking Project > import g-code.

When asked about adding a "G-code header," click "I understand," even though that may well be a lie.

Importing G-code

Secure material

Work zero

Start carving

Importing G-code

As a reminder, Easel relies on the g-code file's settings for feed rate, depth per pass, safety height and material thickness. If you need to change these settings, you'll need to make the changes in your CAM software and upload the new g-code file.

By default Easel will add the following header and footer to your g-code.

Header	Footer
<pre>G20 ; Set inches G90 ; Absolute programming G1 Z0.15000 F9.0 ; Custom position</pre>	<pre>G20 ; Set inches G90 ; Absolute programming G0 Z0.15000 G0 X0.00000 Y0.00000 ; Rapid 0,0 G4 P0.1 ; Dwell & sync</pre>

Add the header and footer to my g-code file
 DO NOT make changes to my g-code file

I understand

Jog Machine

Keyboard Controls

Y

X

UP

Z

DOWN

Interval in mm

XY	0.001	0.01	0.1	1
Z	0.001	0.01	0.1	1

Custom XY 0.1 in

Home

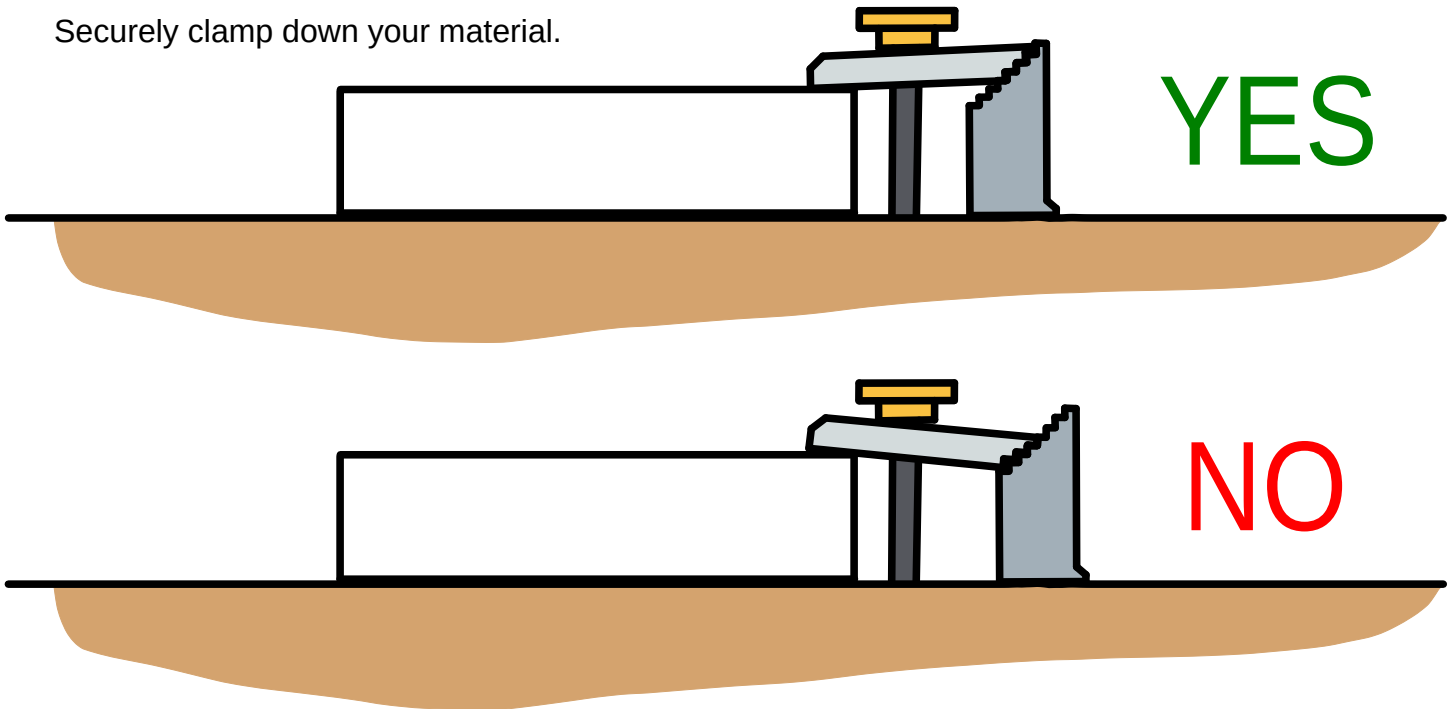
Work Zero *

*Homes the machine first to ensure correct origin

Layout: Old New


Step 9: prepare the machine

Securely clamp down your material.




Use the machine controls to jog the bit to the correct starting position above your material. You can jog using the arrow keys. Adjust the Z-axis resolution to 0.001 when you're close to the material. Jog the bit down along the Z-axis until it just touches the surface of your material. Use a scrap of paper to check how close you are.

Double-check that the material is secure and click Carve to start the carving process.



Carve!



Warning: Never leave the X-Carve unattended while carving, always stay within sight of the machine and near the emergency stop button.

Watch the machine while it's carving. Do not leave the room; do not go to Kindlevan. Text a friend to cover for you if you really have to pee. If something goes wrong, hit the big red emergency stop button and turn off the spindle.

Once the carve is complete, turn off the spindle. E-stop the machine before you unclamp and remove your part-- you don't want the machine to jog if someone bumps the arrow keys.

Reminders

1. **ACTUALLY measure the thickness of your material with calipers.** Your half-inch plywood is probably 0.450-0.470 inches thick. If you get the thickness wrong, either your parts will not be cut all the way through or the bed will be damaged.
2. Before you start cutting, triple-check that your material can't slide sideways.
3. Turn on the orange vacuum under the table when you start cutting. Airborne wood dust will eventually give Brandon lung cancer.
- 4, Wear ear protection: ear muffs, ear plugs, or both.
5. Please clean up when you're done. Sweep up dust, put away clamps, and throw away scraps.

FAQ

The router cut too much in the negative direction, near the lower left corner of the table.
You probably forgot to uncheck "Use offset" in V-Carve, so V-Carve assumed your origin was at the center of your design.

Why is VCarve cutting outside of some curves but inside others?

You need to join your curves into one closed vector.

The Carve button in Easel won't turn green.

Check whether the X-Controller shows up in Device Manager. Try replugging the USB cable. Try turning on the X-Controller. The power switch is hidden on the back.

What settings do I need to know if I'm setting up a new machine in Easel?

Our machine is pre-November 2021 upgrade (but it has been upgraded).

The motion controller is an X-Controller.

The bed is 750 mm x 750 mm.

The Z-axis is the direct-drive Z-axis (2021 upgrade).

The spindle is a Dewalt DW611.

The belts are 3GT 9 mm (2021 upgrade).

We also have extra large stepper motors.

We have homing switches, but they are not useful. We also have a Z-probe, but it also is not useful. It seems easier to disable those features and set your work zero manually.

Should turn the router up to its highest speed? Will that help?

No, the router should be set between speeds 1 and 2. Speeding it up means you will take more cuts, and each cut will be thinner. In total, this will take more strain energy and overheat the bit. Let the edged tool work its magic.