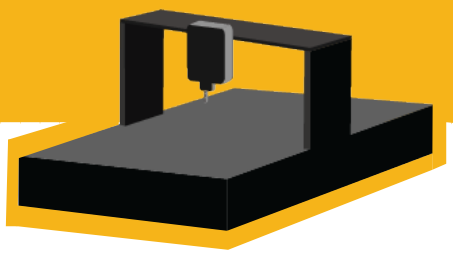
STEPS ON HOW TO OPERATE CNC SHOPBOT-PRSAlpha (ATC 120-60-8)



Data making in Cutting and Engraving (Ai)

1

Create your design in AutoCad or Adobe Illustrator and have them saved in their respective formats: .dxf, or .ai file.

Recommended: Adobe Illustrator

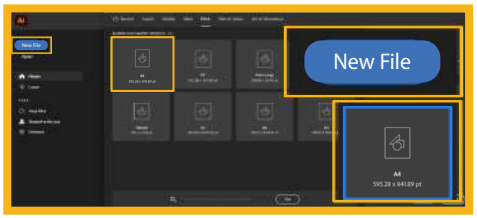
A

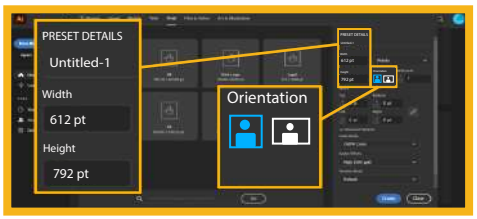
Open adobe Illustrator

Data making in Cutting and Engraving (Ai)

B

Choose custom or any size of art-board by selecting **make** new file in the upper corner of the screen.



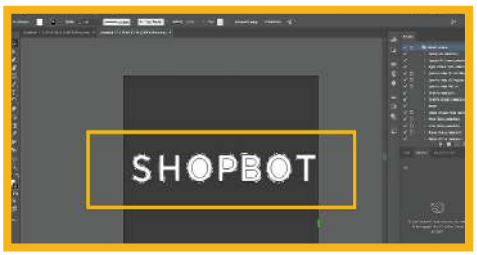


C

Layout your design in an artboard. Be sure that your design is in one stroke.

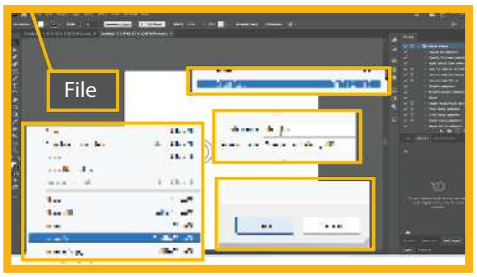
NOTE:

Optionally, you should have an allow-ance (drill bit size used) in between vectors, depending on what you want to be cut or engrave.



D

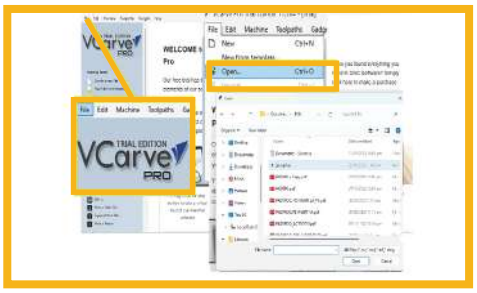
Choose the desired file extension (.ai or .dxf) under File > Save as Your Design and it is prepared for import into Pathwork/Vcarve.



Data making in Cutting and Engraving (VCarve)

2

Open your design (.dxf or .ai) with the VCarve Pro Software, **select** file and **open** your Ai design then **input** your data under Job setup and **click** “Ok” once.



The Job Set Up: To **prepare** your materials for machining, measure their length, width, and height and enter them into the “Job Sise” section. To avoid harming the machine or materials, the XY datum and Z zero must be established. What you must do is as follows:

1. **Set** Z to the material’s top.
2. **Place** the XY datum on the lower left.

NOTE:

It should be noted that the X and Y values should match that of your design files.

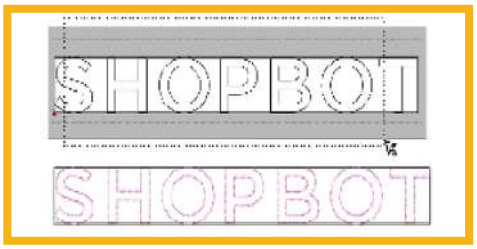
3

Prepare your Toolpaths for cutting. Toolpaths are where the ShopBot’s drill passes through when cutting, engraving, or drilling holes.

Data making in Cutting and Engraving (VCarve)

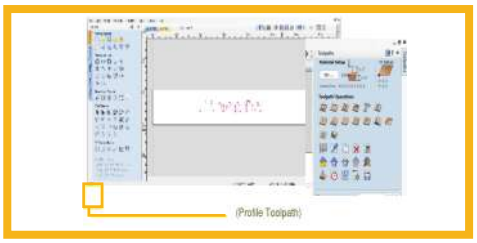
A

Firstly, **select** your design's vectors by clicking and dragging the cursor over the designs (they will be hightlighted in purple magenta).



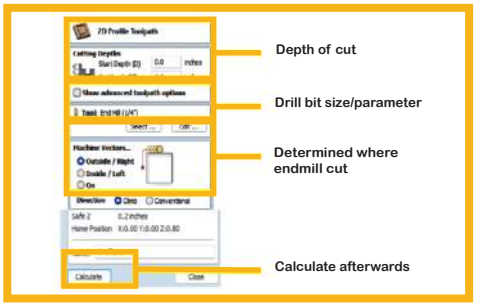
B

Next, **click** "Toolpaths" at the top right corner of your screen. For cutting, **click** the "Profile Toolpath" icon.



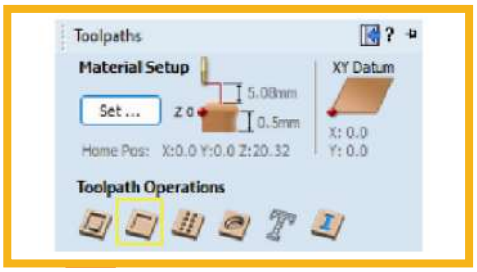
C

Input how deep the drill you want to cut, **choose** your desired tool for the job, then **click** "Calculate" once your done.

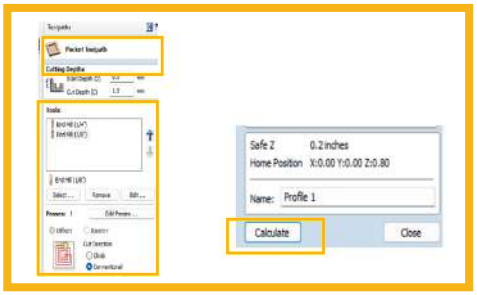


Data making in Cutting and Engraving (VCarve)

For engraving, **click** the “Pocket Toolpath” icon. **Input** how deep you want the machine to engrave.



Choose your tool preferences, then **click** “calculate” once your done.

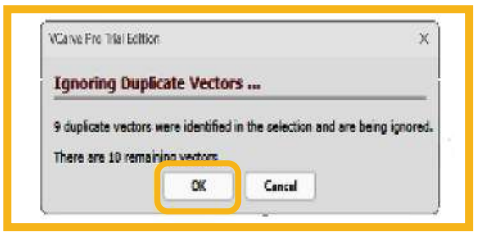


NOTE:

If your purpose is to cut material, make sure that the cutting depth is a little more than the thickness of your material.

D

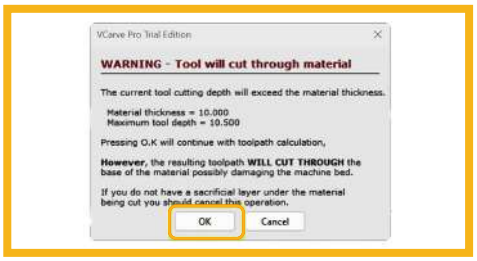
After clicking calculate, you will be greeted with two pop-out windows saying “Ignoring Duplicate Vectors...”



Data making in Cutting and Engraving (VCarve)

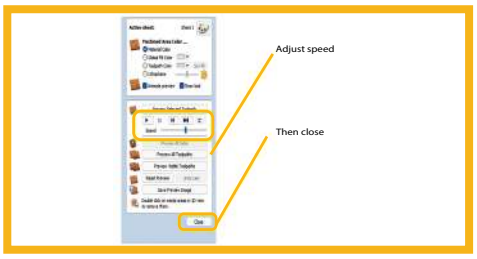
E

After closing the pop-up, another one will appear saying ‘WARNING - Tool will cut through the material’. **Press** “OK” as these pop-ups won’t affect your work.



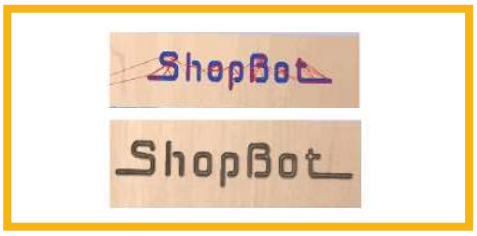
F

Preview Toolpath



G

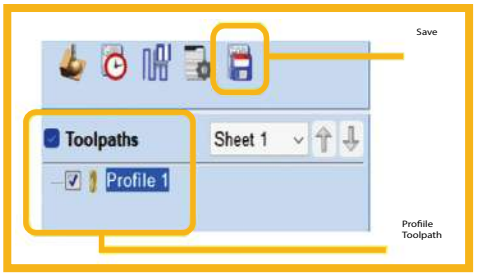
After calculating you toolpaths, you can see the preview of your design of how it’ll look like after the ShopBot is used. If you feel like making adjustments, you can until you’re satisfied, then **click** “Close”.



Data making in Cutting and Engraving (VCarve)

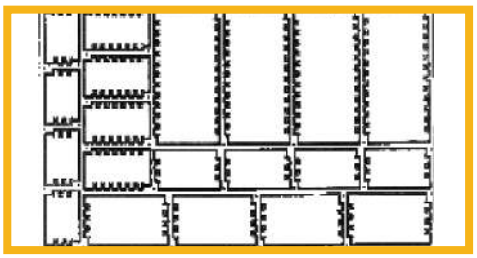
H

Save the toolpaths by clicking the “save Tool path” icon. But be sure to check the box under Toolpaths (profile or pocket) before saving. your project will be save in a .sbp file and **transfer** the file to a flash-drive, then your ready to go.



1

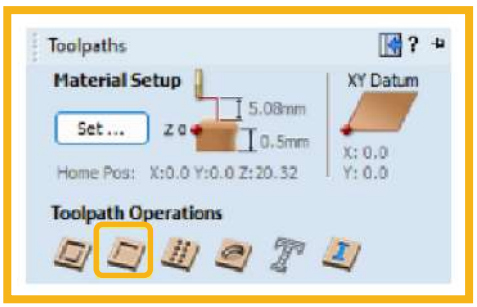
Incorporate the locations of the holes in your design using Auto-Cad, VCarve, and Adobe Illustra-tor, have them save in their respective files. These holes will be circle vectors in your project.



Data making in Cutting and Engraving (VCarve)

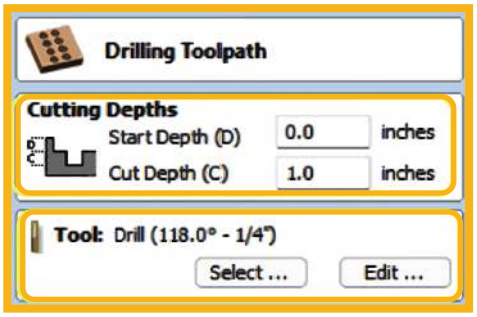
2

To create toolpaths for holes, **select** the object to be worked with then **open** Toolpaths Menu on the upper right corner of the screen.



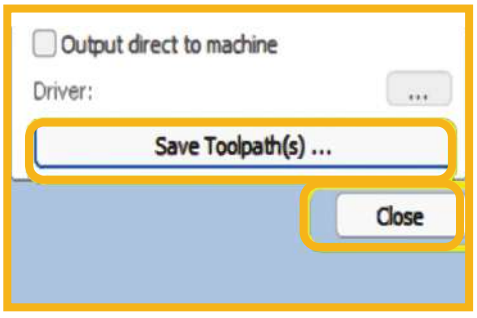
3

Click the Drilling Toolpath's icon then **input** how deep you want the machine to drill, the tool to be used, **check** 'Use Peck Drilling', then **calculate** once finished.



4

Save the toolpaths by clicking the 'Save Toolpath'. But be sure to check the box under Toolpaths (Drilling) before saving. Your project will be saved in a sbp file and transfer the file to a flash drive.



FAQ/s

What software is compatible with ShopBot?

For operation, primarily VCarve Pro. For designs however, any program that allows vectors and can be exported in .dxf, .crv, .ai file formats.

How to select or deselect individual vectors?

You do this by clicking the left mouse button, hold it, then drag the cursor over your designs. Once it's done, the vectors will be highlighted purple/magenta broken lines. To deselect them, just simply click the background.

How do I choose a bit? What Feeds and Speeds should I use?

These things are relatively complex concepts and can be learned through practice. It also depends on what your project needs for it to be done

FAQ/s

Why do we need to put allowance in our designs?

Designs with multiple separate alements need to have allowance for the machine to move between paths. Failure to add will result to an error.

What is the use of Auto Hide and Show behavior of the drawings and toolpath?

Allows them to automatically close when not used, maximizing the working screen area.

Why does the screen say 'No Vector Selected'?

This is because you haven't selected your project's vectors before making toolpaths. To fix this, click and drag your cursor above the project. They wil successfully be selected once they turn purple/magenta broken lines. Then proceed to make your toolpaths afterwards.

FAQ/s

What materials do we need to use?

The materials needed for the ShopBot is entirely what you need for your work. But using materials that are too soft may cause damage to the machine, so please exercise caution.

Why should the width and length match our design files?

This is to ensure that your project gets engraved, cut, etc. within your matrial's boundaries at the same exact location. Adding dimensions other that what is specified in your project will lead to misalignment.

What is Tool Offset used for?

By letting the machinist set z-axis datum for each part, the tool length offset enables the CNC machine to compensate for many tools of different lengths. This ultimately enables machinists to use multiple tools without having to reset the z-axis datum every time they use a tool.

FAQ/s

What is the maximum depth that the tool can cut?

A good rule of thumb is the depth of cut that shouldn't be greater than 1/2 the diameter of the tool; especially on smaller machines. Any more that this and you'll risk messing up the bearings of the machine.