Creating your 2.5D VCarve File for the ShopBot CNC Router
A Basic Step-By-Step Guide

This guide is just for creating your project file in VCarve Pro. After you complete your VCarve file and make a reservation with the ShopBot, refer to 'Running your File on the ShopBot CNC Router' to review setting up and running your file. You must have taken the CNC Routing Shop Class with Maker Playground to be certified to reserve time on the CNC.
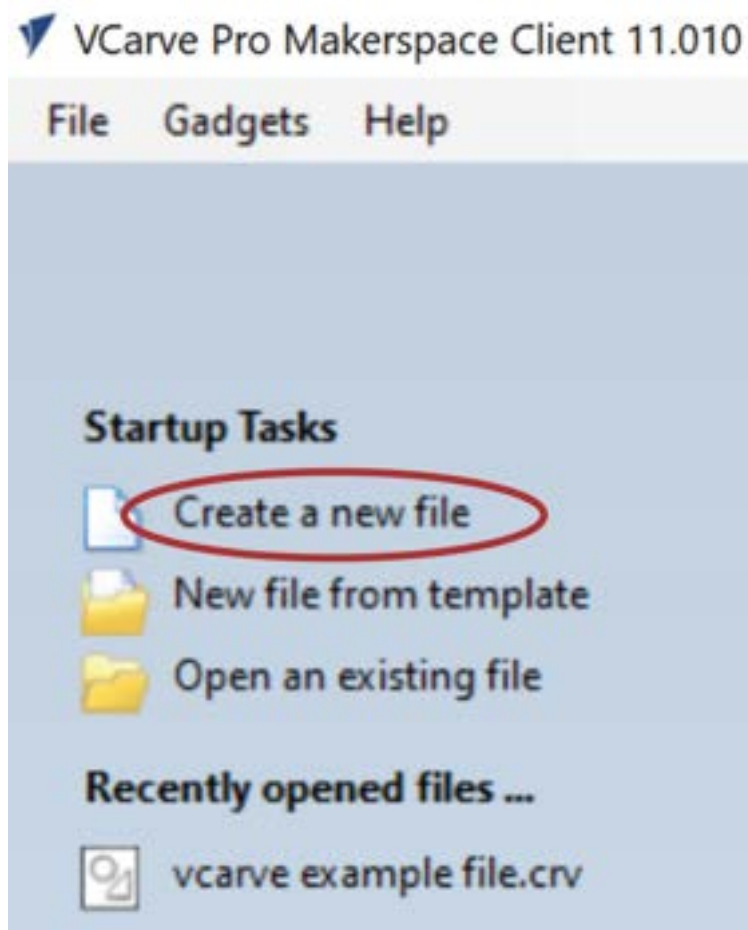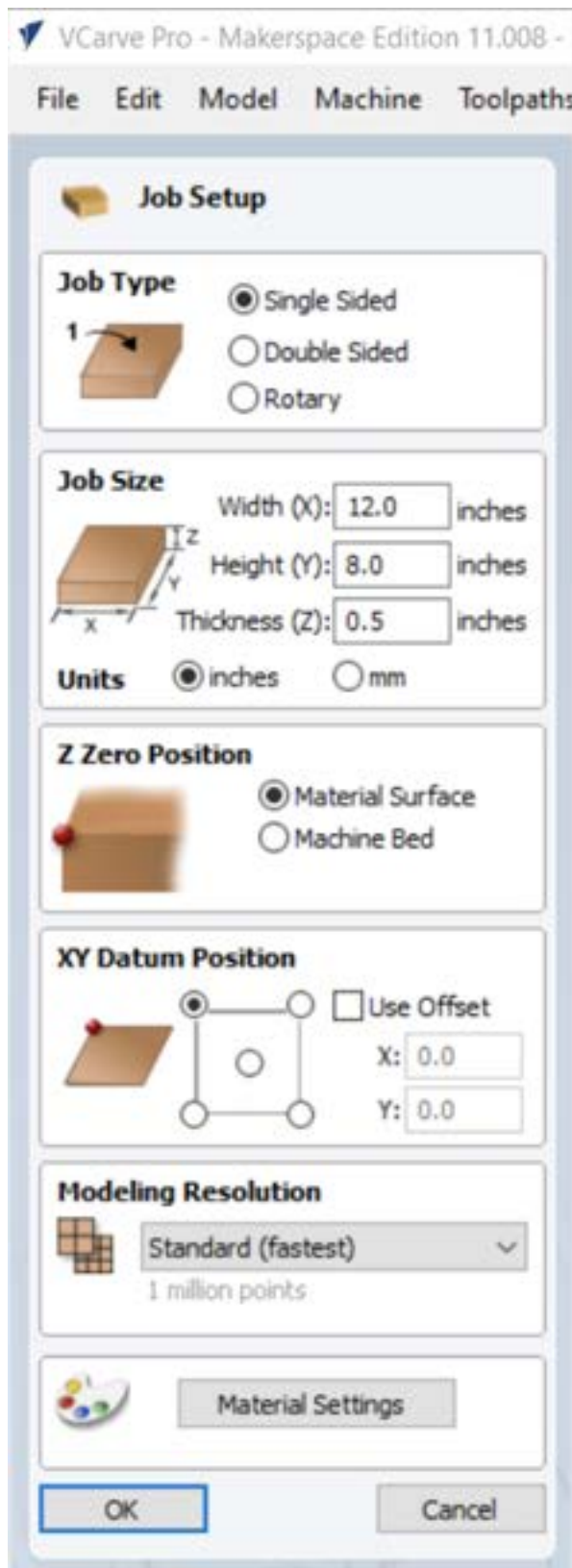
Index

Getting Started & Job Setup

Open VCarve Pro.

To the left, under Startup Tasks, Click 'Create a new file.'

A new file will open. First, you'll need to configure your Job Setup.

Job Type allows you to choose which type of job you are creating. **Most Job Types will be 'Single Sided.'** Double Sided jobs allow you to flip your material and continue cutting on the other side, which this guide will not cover. Rotary jobs are for machines that have a rotary indexer, which looks somewhat like a lathe. We don't have this (yet), so you won't need this option.

Job Size will most often be your material size. Enter the width, height, and thickness of your material. Be as accurate as possible. We recommend using inches as the unit of measure, since this is what our ShopBot is configured for. Using millimeters will require a different post-processor.

Z Zero Position determines where the machine will read your Z height. If your job is set to material surface, the machine will reference the top of your material for its Z axis. If you select Machine Bed, the machine will reference the bed of the ShopBot. There are times when referencing the bed may be more useful, but **most projects are fine to use 'Material Surface.'**

XY Datum Position is essentially your origin position. You can make your origin your upper left, upper right, lower right, lower left, or center. Use your judgment to determine which origin will be best for your particular project, as this will be the reference point the ShopBot will use. 'Use Offset' allows you to adjust your origin from any of these points.

Modeling Resolution and **Material Settings** simply affect how your project appears in the program and does not affect your actual project. These do not need to change.

Once you have these configured, click 'OK.'

If you need to change anything, you can return to the Job Setup Screen and change the settings any time by clicking the 'Set Job Dimensions and Origin' button under 'File Operations.'
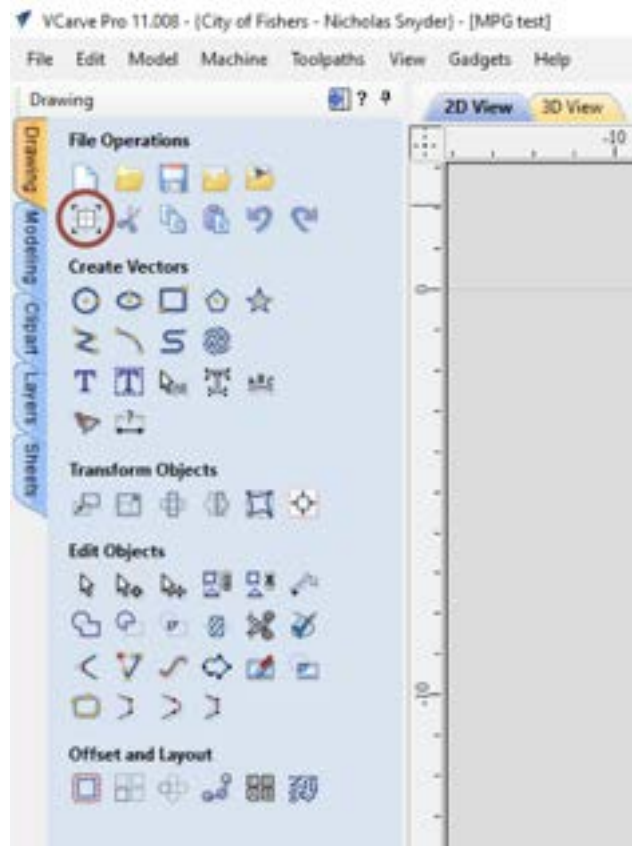

Now, we can create our vector design. **Vectors** are images that are built by mathematical formulas that establish points on a grid, rather than raster images, which use colored dots or pixels to make up the image. Vectors can be scaled infinitely and allow a tool to follow the points which make up lines or boundaries.

You can do this a few different ways: importing a vector design, importing a raster image to trace into a vector design, or creating a design in VCarve.
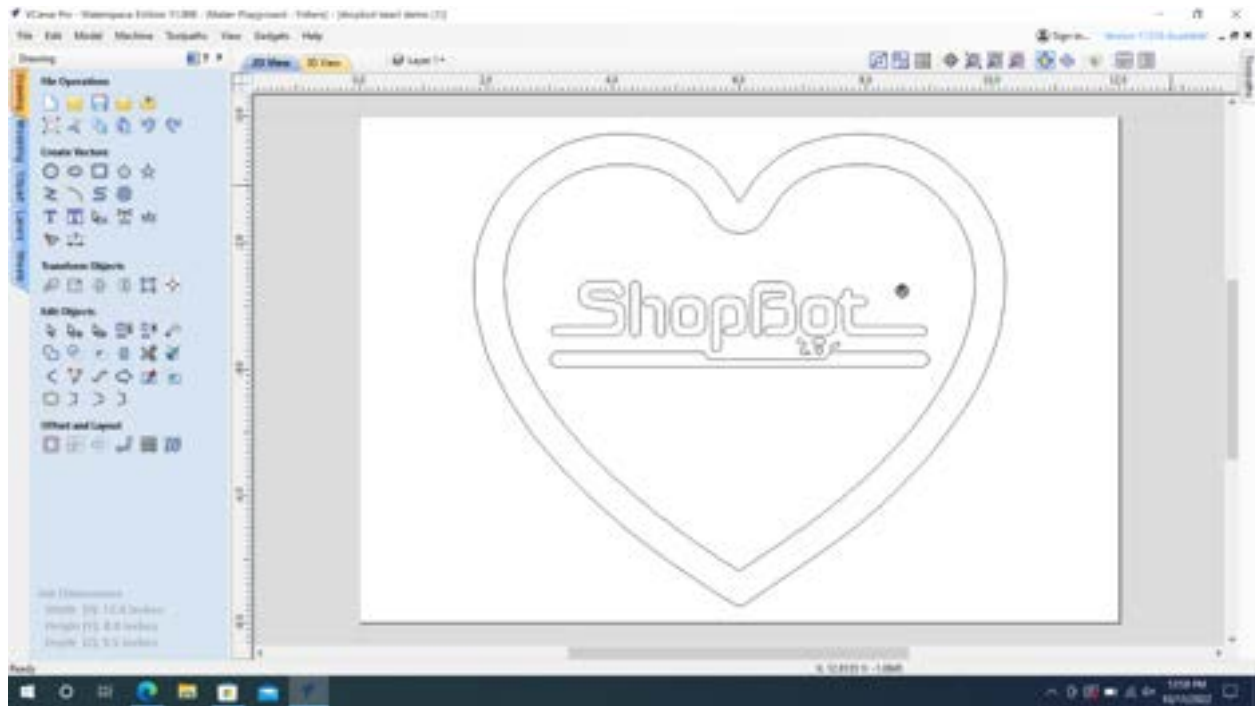
Import a vector design

If you create your designs in a different program such as Adobe Illustrator, or purchase premade vector designs on Etsy, you can simply import these designs into VCarve.
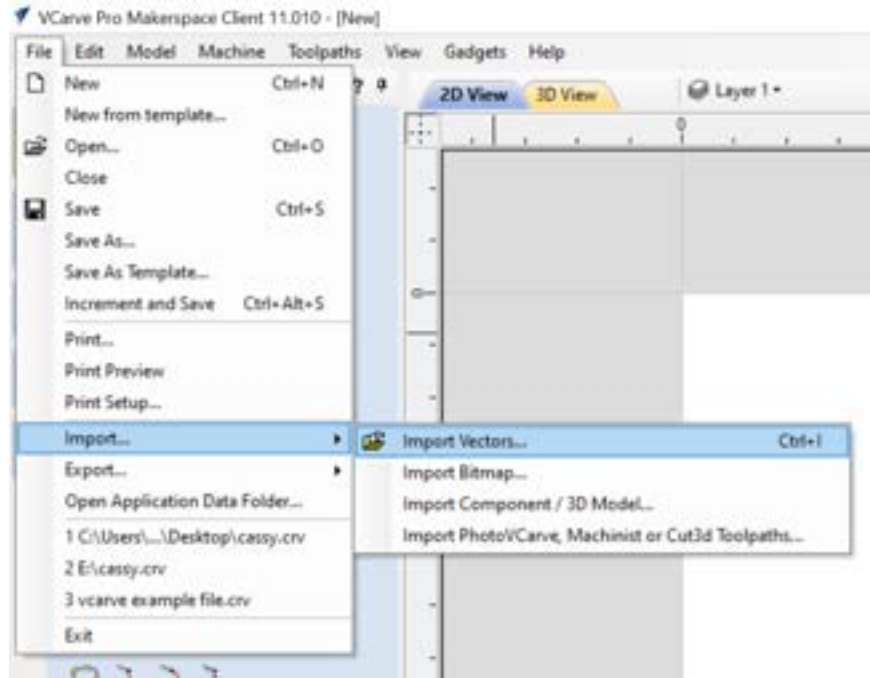
In the upper left corner, go to File > Import... > Import Vectors...
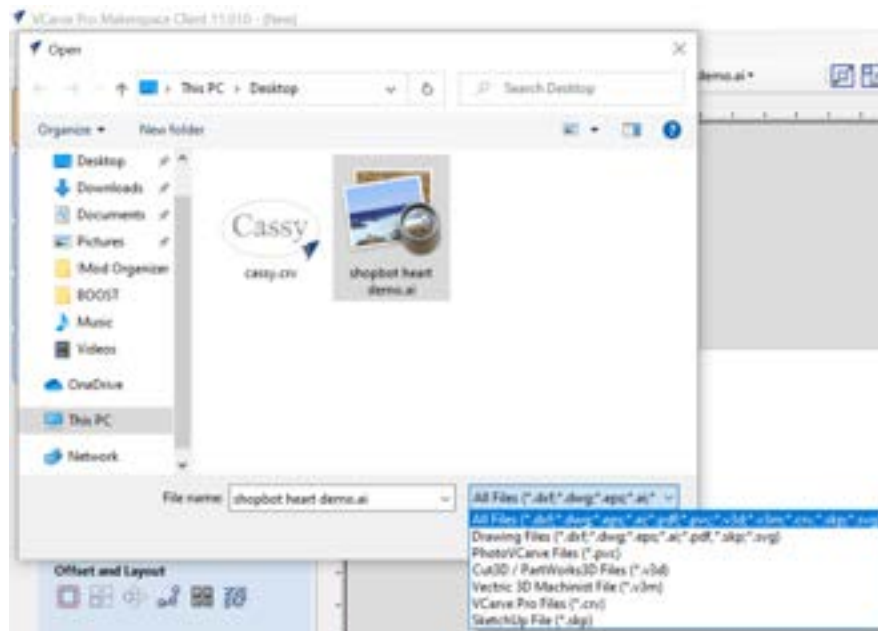
Select and import your file. You can import any of the following file types:
.dxf, .dwg, .eps, .ai, .pdf, .pvc, v3d, .v3m, .crv, .skp, .svg

Your vectors will appear in your project! They may show up in some of the gray space, so be sure to zoom out to look for them if you do not see them immediately.

If your design is complete, you will be ready to begin creating Toolpaths and can continue to page 11. If you have other shapes or text to add to your design, continue through this guide to learn how to import bitmaps to trace or draw and adjust vectors in VCarve.

Import a bitmap image and trace it into a vector design
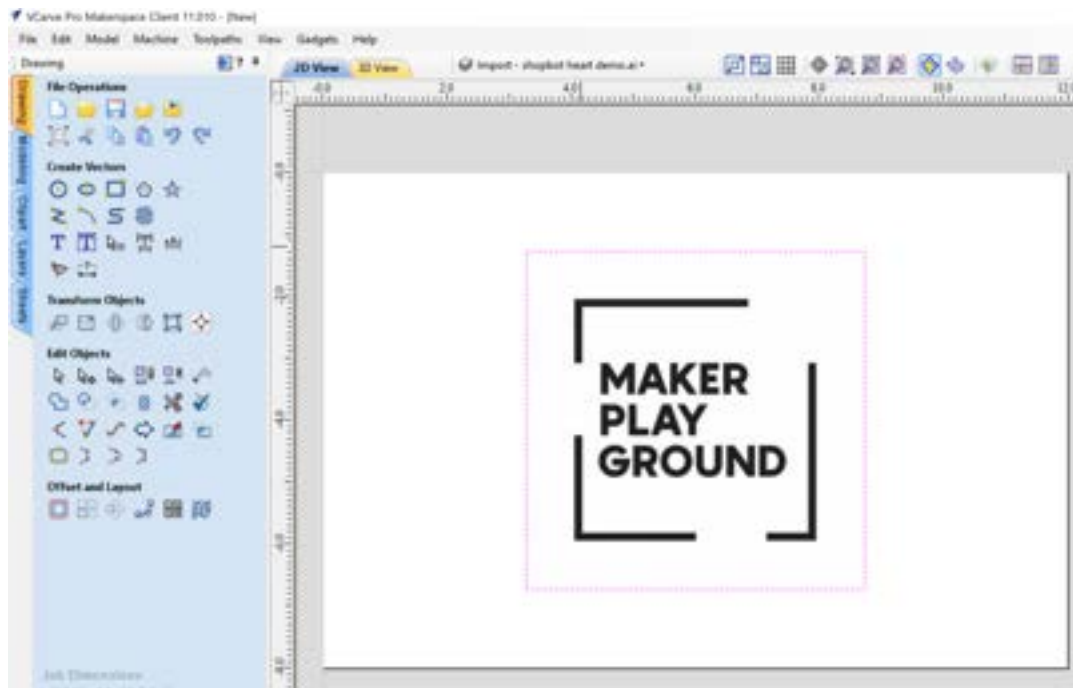
This works best with dark colors logos or shapes on a white background. Avoid images with gradients or fine text.
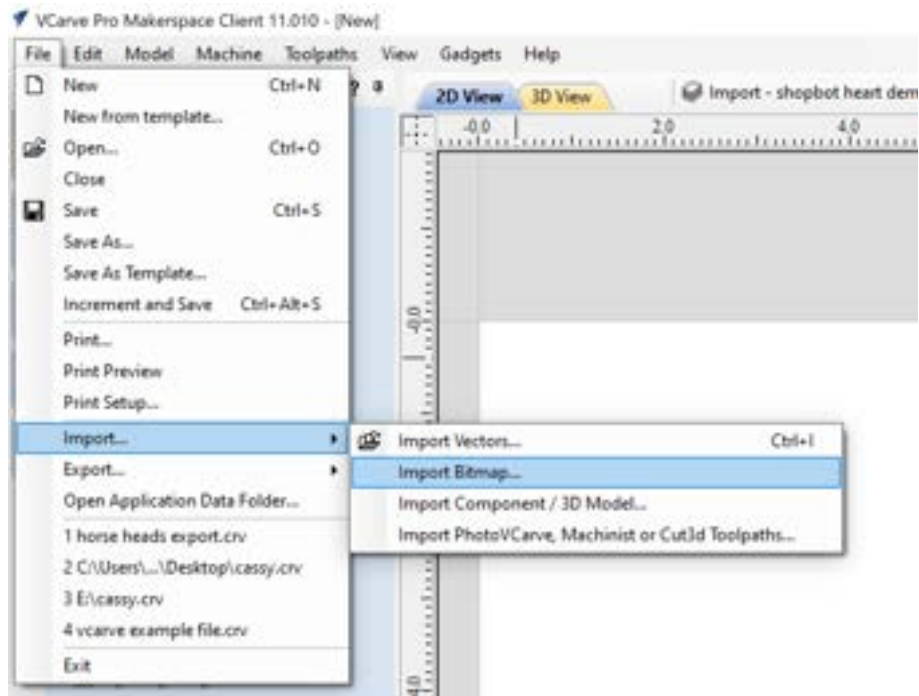
In the upper left corner, go to File > Import... > Import Bitmap...
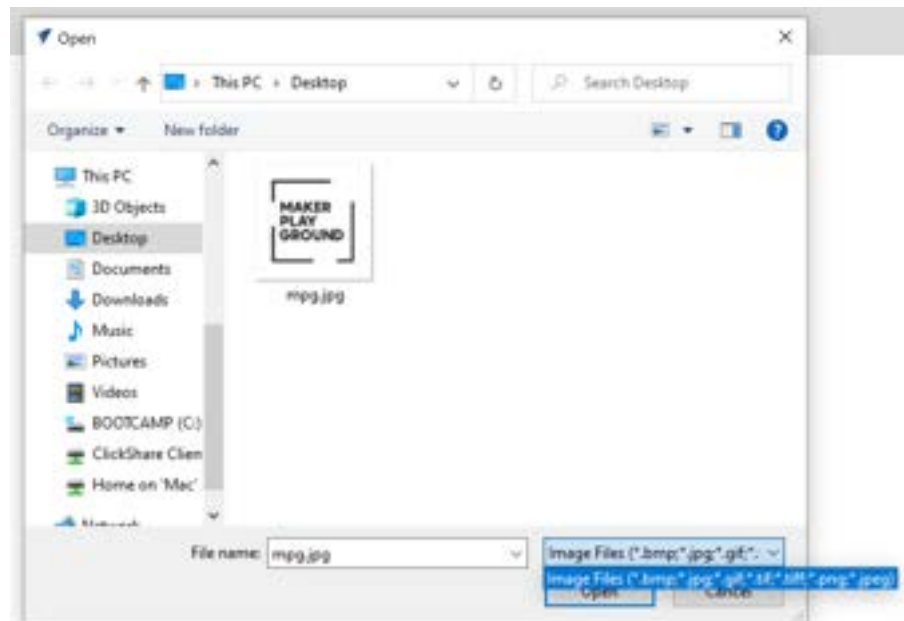
Select and import your file. You can import any of the following file types:
.bmp, .jpg, .gif, .tif, .tiff, .png, .jpeg

Your image will import into your project! However, we still need to trace the bitmap image into vectors that we can use to create toolpaths.
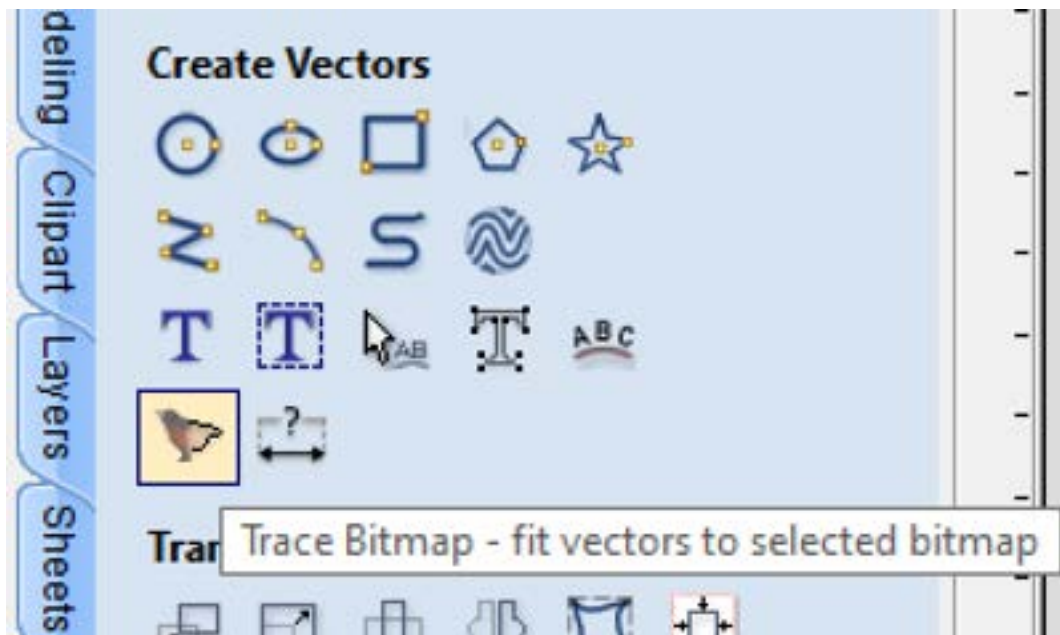
With your image selected, find and click the "Trace Bitmap - fit vectors to selected bitmap" button under "Create Vectors" on the left side of the screen. The icon looks like an image of a bird.
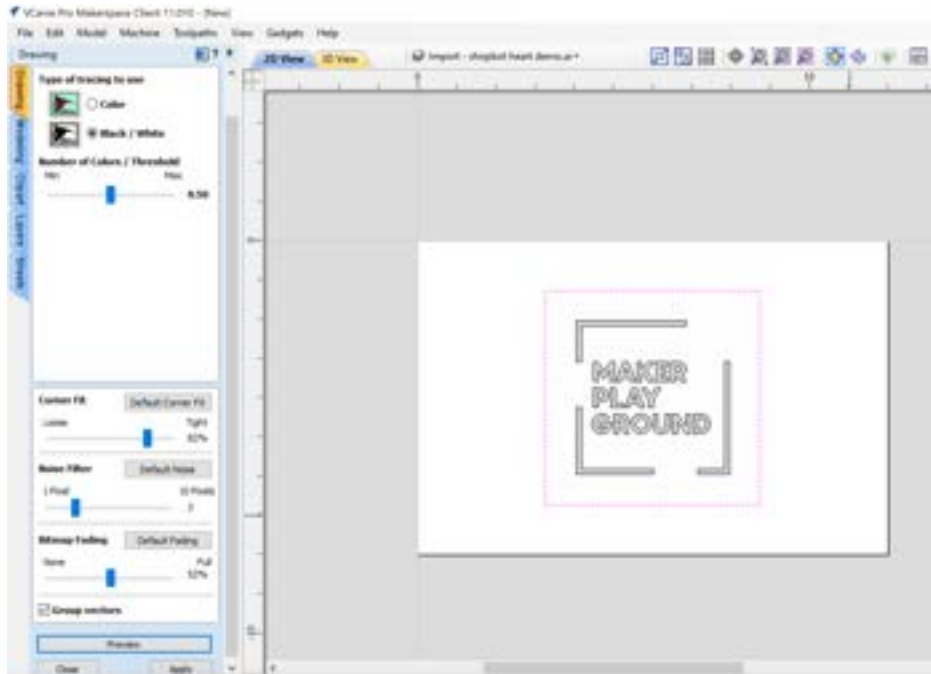
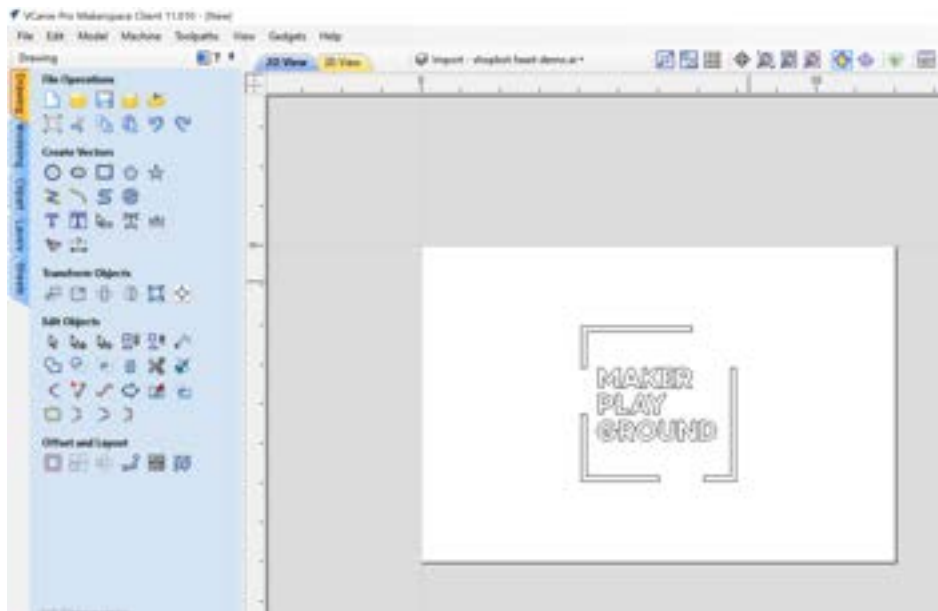You can adjust the sliders on the left to slightly change the tracing of your image. Once you are satisfied with how your vectors look, click "Apply" and "Close."

Now your bitmap image will be traced! If you'd like, you can delete the original bitmap to just show the new vectors.

If your design is complete, you will be ready to begin creating Toolpaths and can continue to step 11. If you have other shapes or text to add to your design, continue through this guide to learn how to draw and adjust vectors in VCarve.

Create a Design in VCarve

Using the tools to the left in the Drawing Tab of VCarve, you can create your own design.
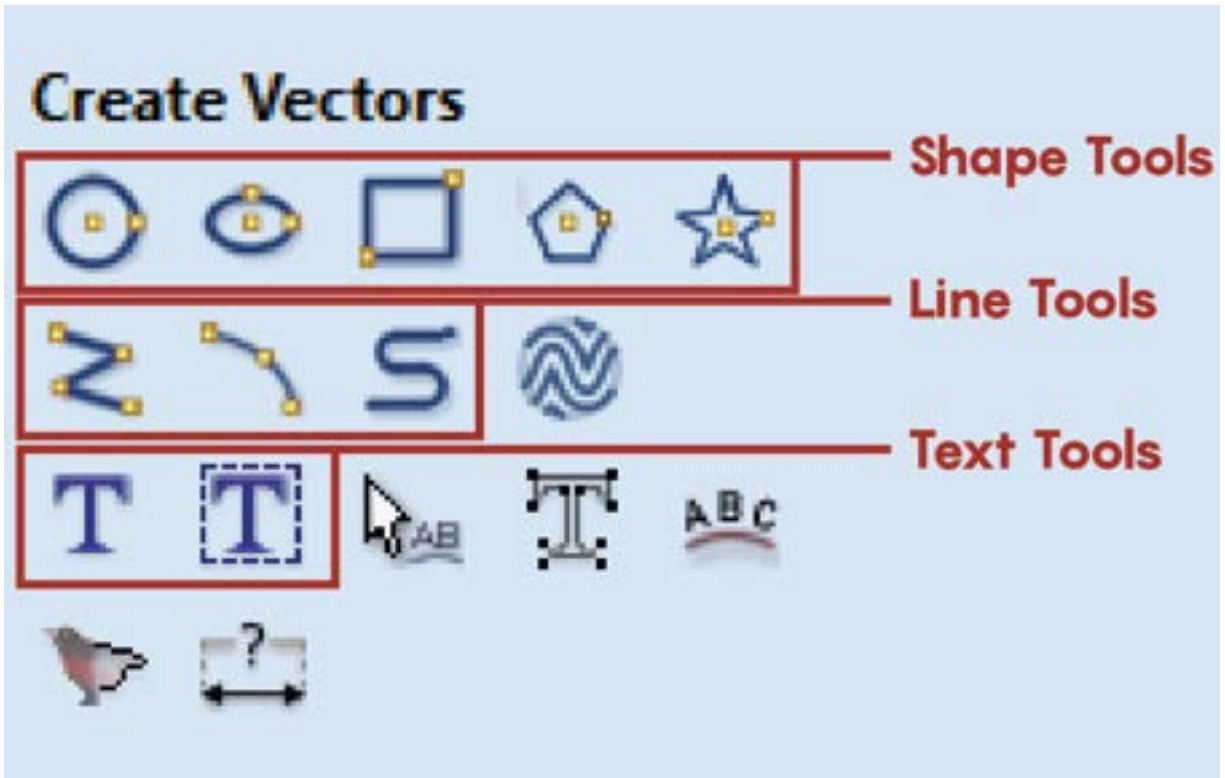
Under "Create Vectors," there are many useful tools for creating shapes, text, and lines.
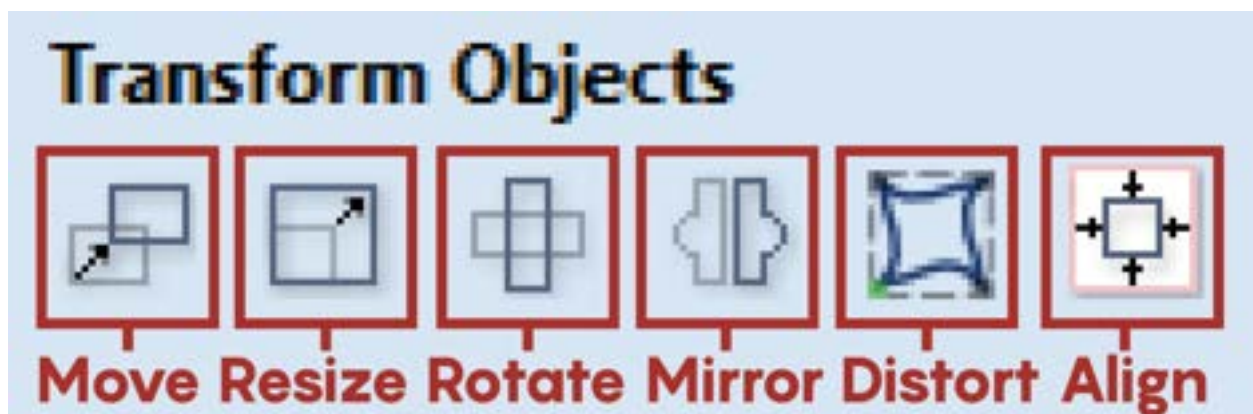
Under "Transform Objects," there are many useful tools for resizing, centering, rotating, etc.
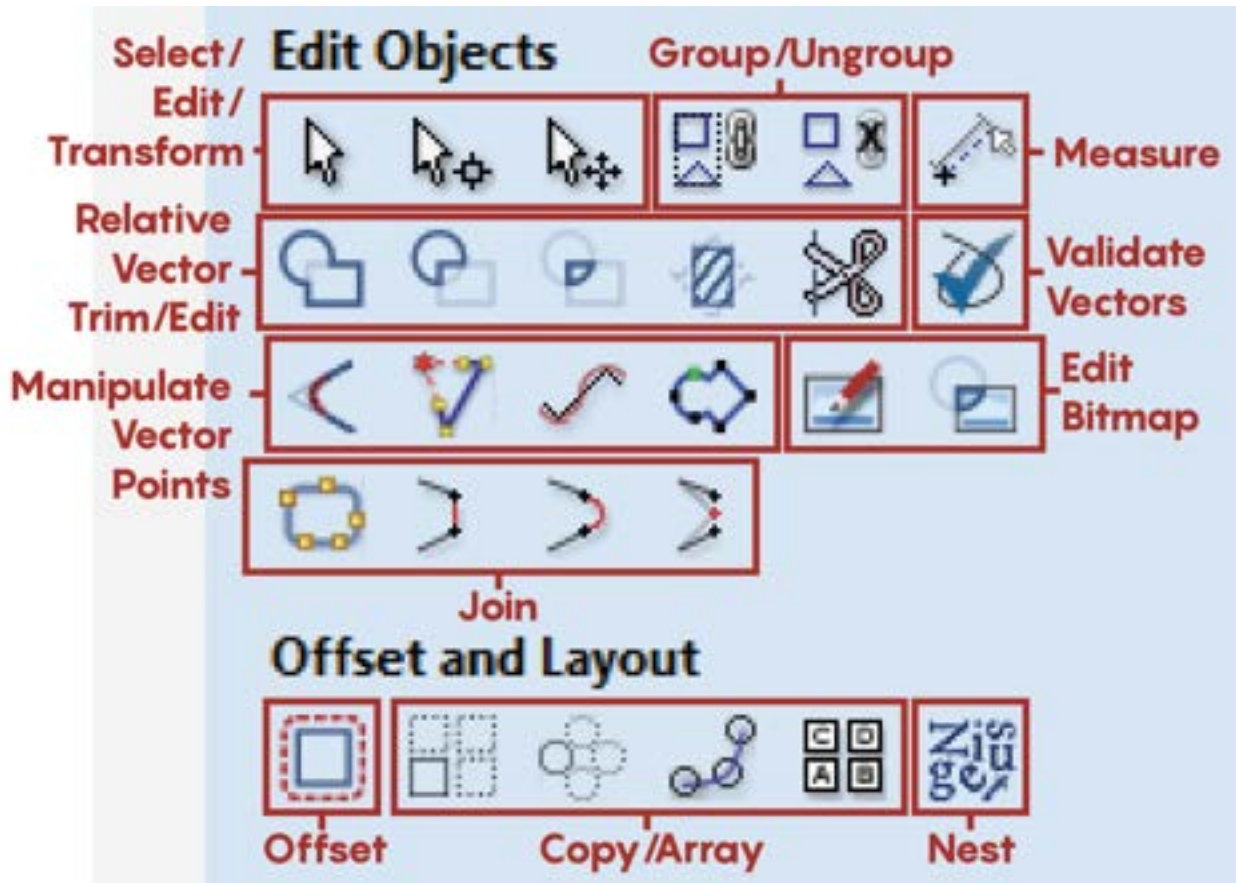
Edit Objects & Offset and Layout include tools for editing and manipulating vectors.

There are many helpful tutorials for using these drawing tools on Youtube. Since this is an abridged guide, we will not be elaborating upon all the many functions of VCarve's drawing tools. We encourage you to explore tutorials and play around with them yourself!

Once your design is complete, you will be ready to begin creating Toolpaths.

Calculating Toolpaths

Toolpaths are instructions that define the movement of cutting tools to create our design. This is where we tell the program how the machine should cut these two dimensional vectors into a three dimensional object.

Open the Toolpaths window on the right side of the screen.
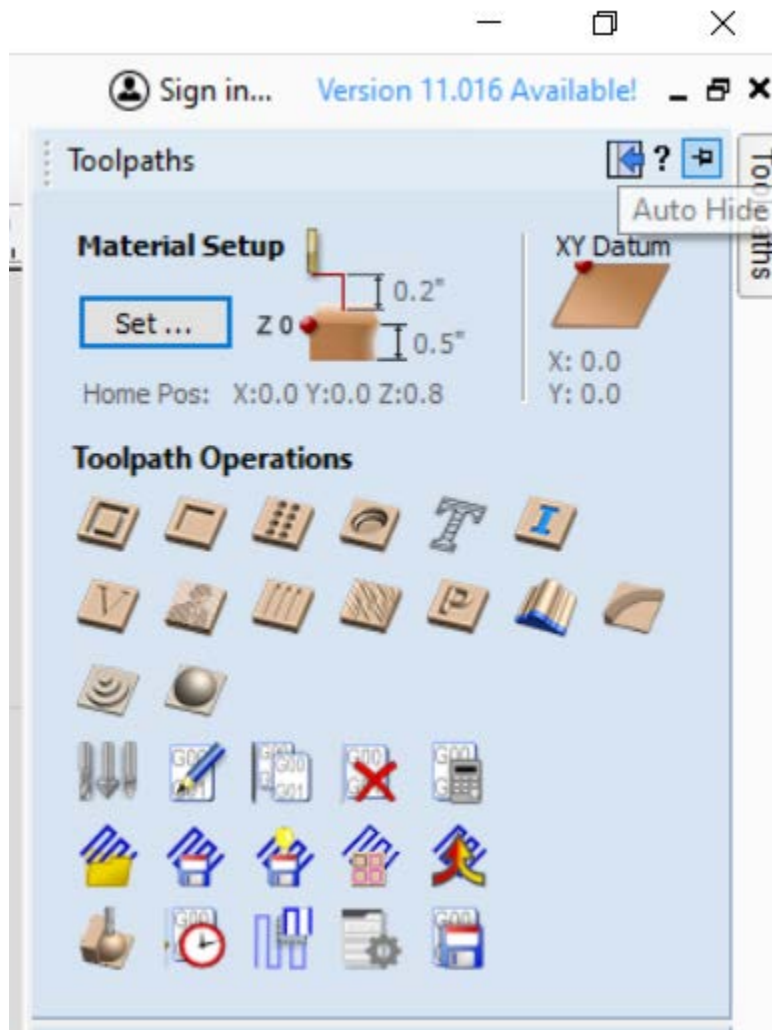
You'll want to click the "Auto-Hide" button in the upper right to keep the Toolpath window from disappearing when you move your cursor. You'll see a lot of different Toolpath Operations in this window. This guide is only going to review the three most common for vector cutting: Profile, Pocket, and VCarve.
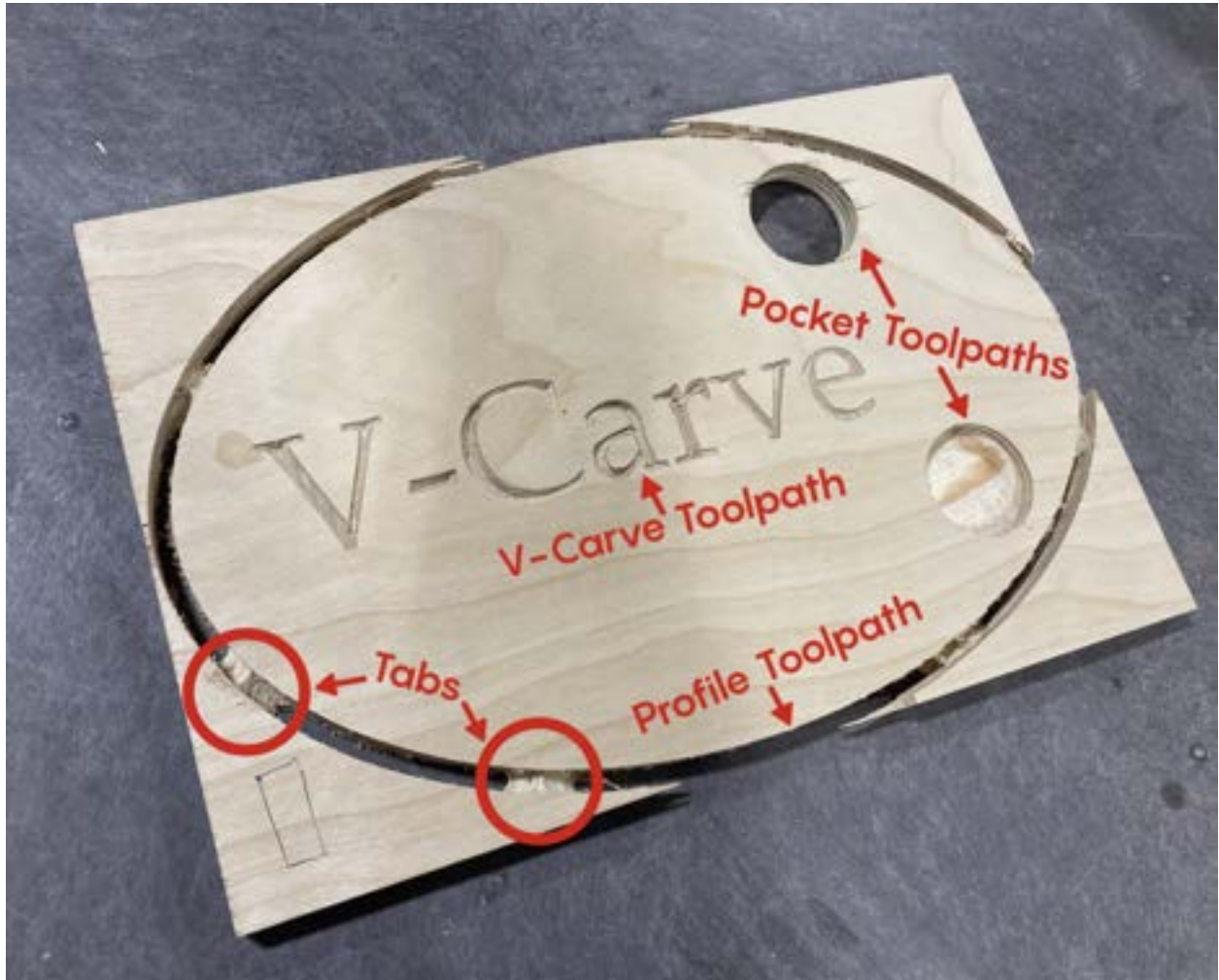
Profile Toolpaths

Profile toolpaths are used when you want your bit to cut along a line. Profile toolpaths do NOT require a closed shape.

Examples where you might want to use a profile toolpath include cutting out shapes, cutting dados the size of your bit, or perhaps a juice groove. These are all profile cuts, and vary just by changing the depth of the cut.

Make sure to select the vector(s) you want to calculate as your profile toolpath(s). Once selected they should show up magenta.
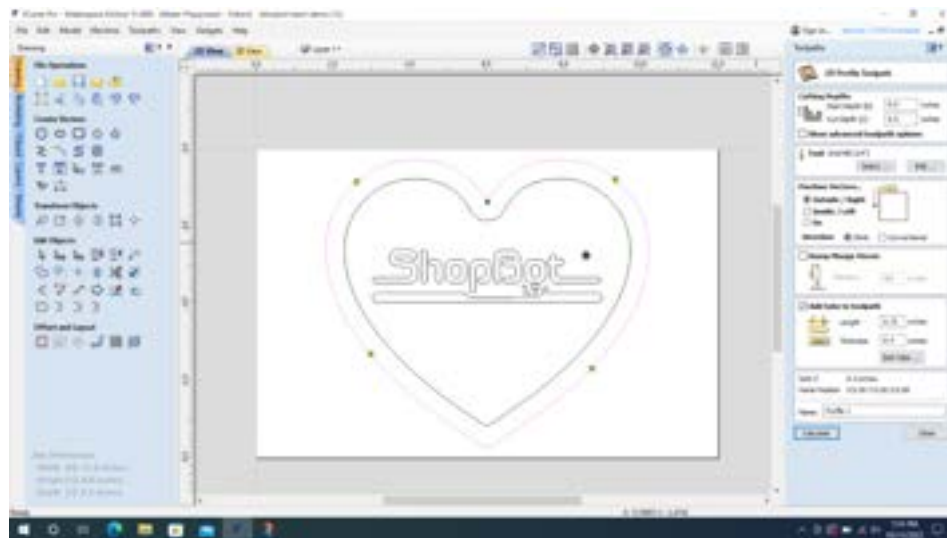
Keep in mind that if you are profile cutting a shape with sharp inner corners, the inner corners will have radii the size of your bit.

* Asterisks indicate required settings! *

*Start Depth - This is the Z height where the bit will begin cutting on your material and will usually be zero. The only time it would be a different depth is if you have already made cuts that change the width of your material where the cut is being made. This allows cutting inside of a pocketed region.

*Cut Depth - This is how deep the bit will cut your toolpath.

> Oftentimes, profile cuts are used for cutting out shapes. If you want to do this, you'll want to enter your material thickness (measure this with a caliper!), and add about .03-.06 to get your cut depth. This will make sure that your bit cuts through your material completely even if your material is warped. You'll also want to use tabs, explained more below.

> You may just want the tool to carve along the line but not cut completely through your material, perhaps for a dado or a juice groove in a cutting board. You'll simply want to enter the depth you want the tool to cut into your material.

DO NOT check "Show advanced toolpath options". This guide will not go over these optional settings.

*Tool - Click the Select button opens the Tool Database from which the required tool can be selected. For profile and pocket toolpaths, these will often be endmills for most applications. Maker Playground will typically have ¼" and ½" endmills available to use. Ask staff or see our CNC Tooling Guide for more information about bits. "Select" allows you to select a different bit, while "Edit" allows you to change the parameters of the current bit.

*Machine Vectors - This is where you decide where your bit will follow along your line, which will depend on your desired outcome.

If you are trying to cut out a shape to a specific size, you will probably want to use "Outside/Right." If you are cutting an inner shape or hole you need to be a particular size, you probably want to use "Inside/Left." If you want a juice groove precisely on a line you've made, you probably want to use "On." If your vector is not a closed shape, you will still have these options, but make sure it calculates the way you intended if you choose to cut inside or outside.

*Direction - In most instances, climb cut will be the best option for cut quality, reducing tearout, and extending the life of your router bit.

Conventional   Climb
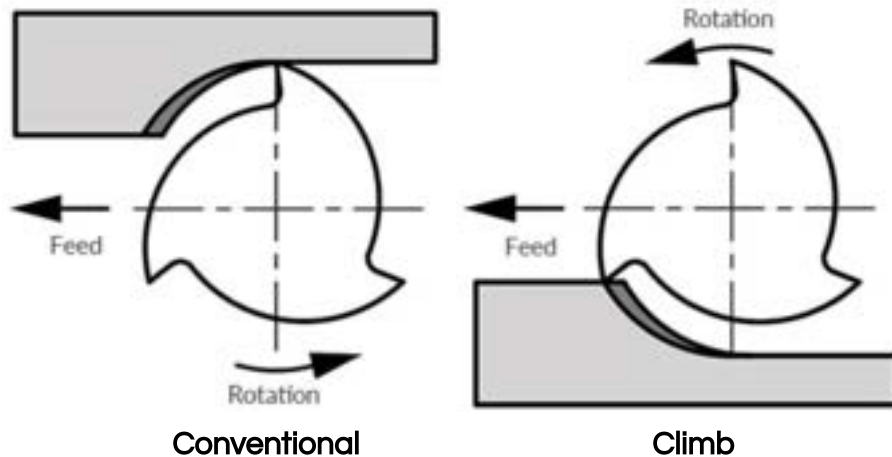
Ramp Plunge Moves - Ramp moves are used to prevent the cutter from plunging vertically into the material. The cutter gradually cuts at an angle dropping into the material significantly reducing cutter wear, heat build-up and also the load on the router spindle and Z axis of the machine. However, this is not a necessary option to create your toolpath.

*Add tabs to toolpath - Tabs are important to use when cutting completely through your material so that any pieces you cut will not become unsecured while cutting. If your piece becomes unsecured, any moving parts can break the bit, or even dislodge from the piece completely and be thrown from the bed by the spinning bit. Tabs are not necessary if you are not cutting all the way through your material.
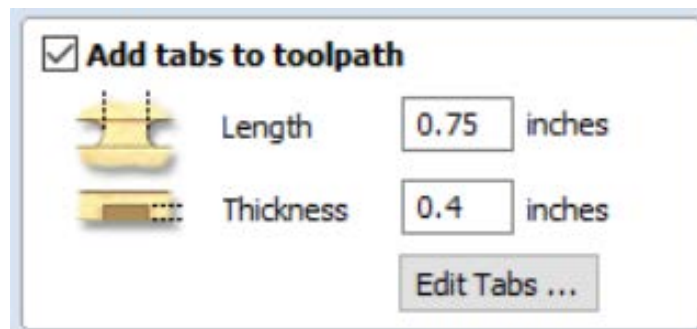
To add tabs, check the "Add tabs to toolpath" option. Enter the length you would like your tabs to be (recommended to be between .5-1") and the thickness (recommended to be just under your material thickness, though this can vary.)

Tabs can be added automatically, but I recommend adding them manually so you can decide exactly where you want them to go. Simply click along your vector where you want to add your tabs. The larger your shape is, the more tabs you should add to keep your piece secure.
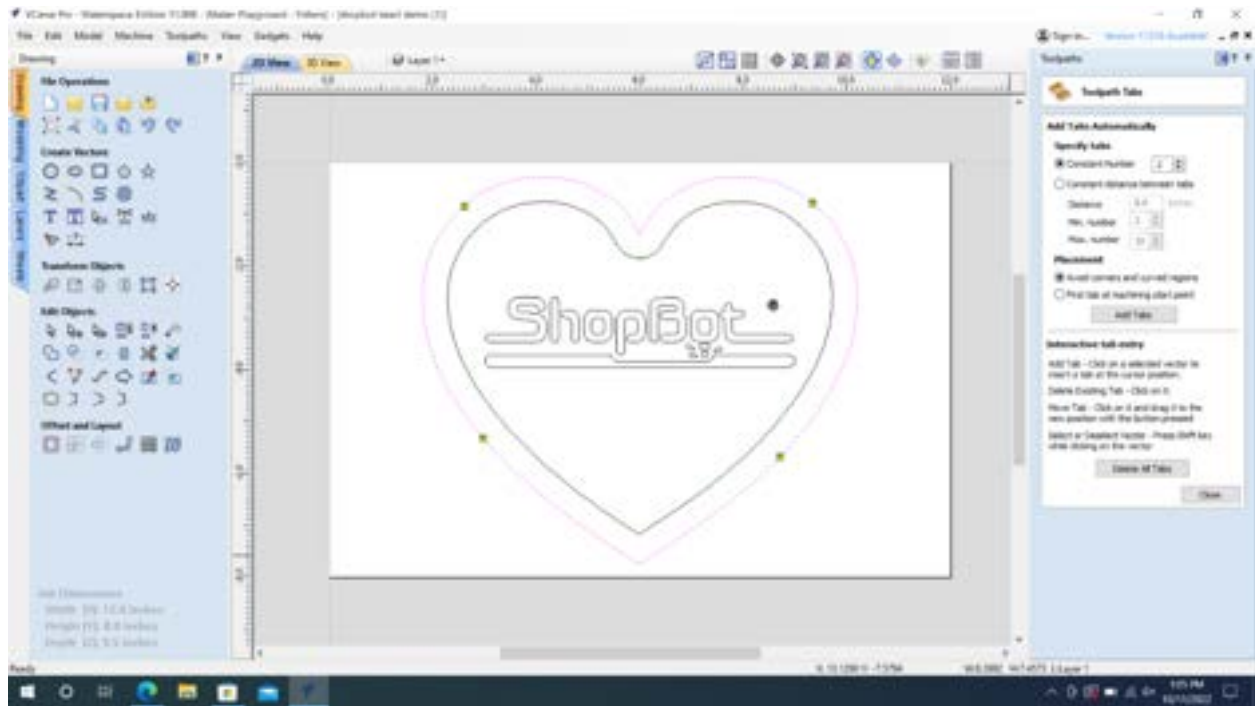
Name - The name of the toolpath can be entered or the default name can be used.

Click "Calculate" at the bottom to create your toolpath.

After any toolpath is calculated, it will automatically direct you to the "Preview Toolpaths" tool. Here you can play a simulated preview of exactly what the tool will do during your cut.

If the box next to your toolpath is checked, it will also show you exactly where your tool will go during the cut. Red lines represent jogs, where the tool moves at a safe height above the material. Blue lines are where the tool will be cutting into your material.
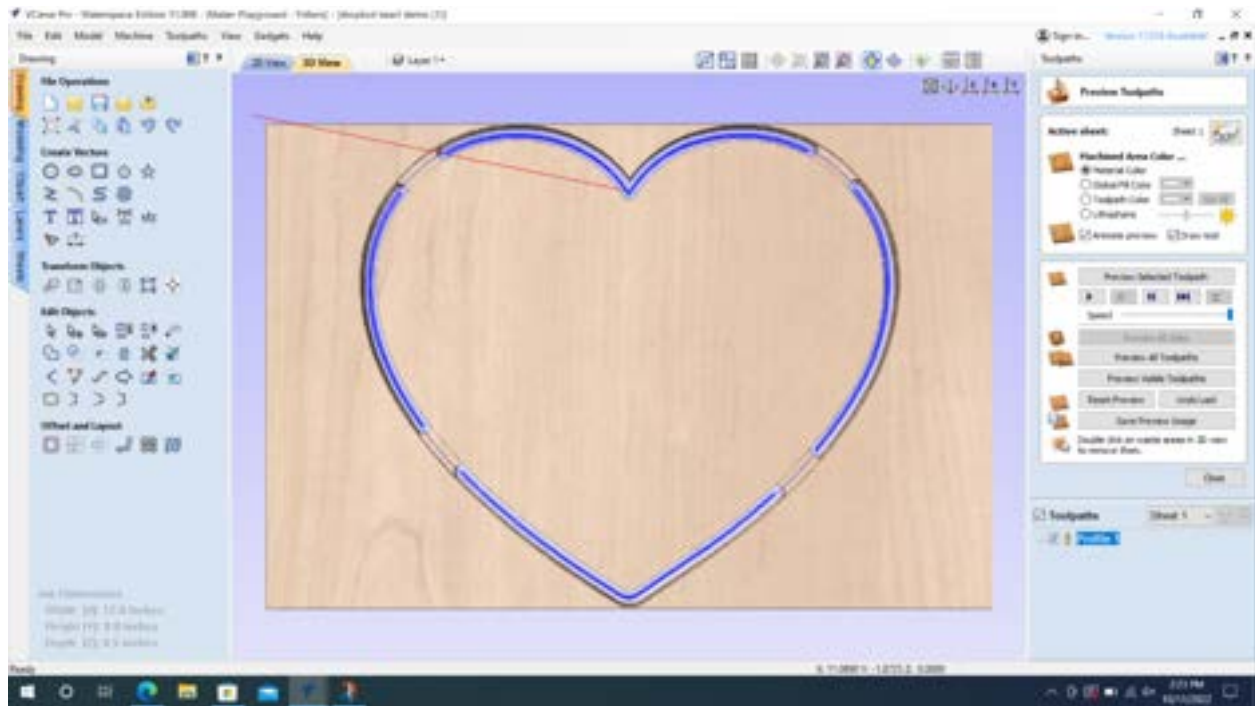
To navigate back to toolpath selection, simply click "Close" at the bottom of the panel. To navigate back to your vectors in order to select them for a new toolpath, you'll need to click "2D View" on the top of the screen.

You can always go back and edit toolpaths by double-clicking on the toolpath's name. If you change your vectors, you'll need to click on the toolpaths it affects and recalculate them to update them for any changes you've made.

* Asterisks indicate required settings! *

Creating a Pocket Toolpath

Pocket Toolpaths are used when you want your tool to remove all material inside of a shape to the depth you set. Pocket toolpaths DO require a closed shape.

These can be used to create engravings, trays, or small holes that may go through your material. These are all pocket cuts, and vary just by changing the size of the pocket and the depth of the cut. Pocket is also what you will use for surfacing slabs, simply by creating a pocket that is oversize of your piece.

Make sure to select the vector(s) you want to calculate as your profile toolpath(s).
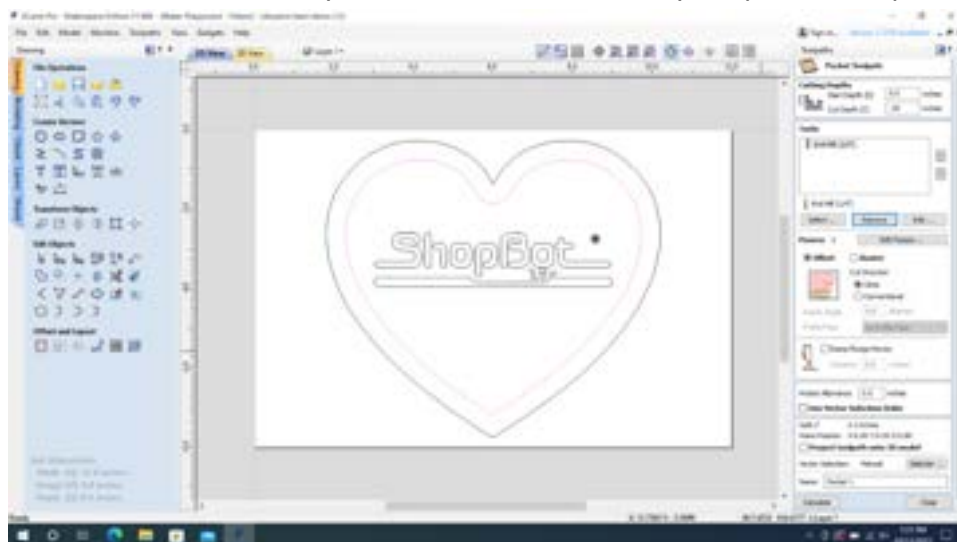
Keep in mind that if you are pocketing a shape with sharp corners, the inner corners will have radii the size of your bit.

*Start Depth - This is the Z height where the bit will begin cutting on your material and will usually be zero. The only time it would be a different depth is if you have already made cuts that change the width of your material where the cut is being made. This allows cutting inside of an already pocketed region.

*Cut Depth - This is how deep the bit will cut your pocket.

> Pockets can be cut to a desired depth to create a hollowed out shape. This can be useful for creating dishes, joinery, engraving, etc. If you are looking to achieve this, simply enter the depth you want the tool to cut into your material.

> Pockets are also useful for cutting out small holes that will go through your material. While you could also cut the shape out using a profile toolpath, it may be safer or easier to use a pocket so you don't risk the small piece detaching or having to clean up tabs afterwards. If you want to do this, you'll want to enter your material thickness (measure this with a caliper!), and add about .03-.06 to get your cut depth. This will make sure that your bit cuts through your material completely even if your material is warped.

*Tool(s) - Click the Select button opens the Tool Database from which the required tool can be selected. For profile and pocket toolpaths, these will often be endmills for most applications. Maker Playground will typically have ¼" and ½" endmills available to use. Ask staff or see our CNC Tooling Guide for more information about bits. "Select" allows you to select a different bit, while "Edit" allows you to change the parameters of the current bit.

While cutting a pocket, you may choose multiple tools to clear material quicker. For example, if you have a large cut, you may want to start with a ½" endmill to clear material quickly, then cut again with a ¼" endmill to remove material in areas the ½" endmill could not reach. "Remove" allows you to remove any bit on the tool list.

 If you are surfacing a slab, Maker Playground does not provide surfacing bits.

Passes - This shows how many passes your bit will make to cut to the specified depth. Since pass depth is already set in the tool parameters, editing this is not necessary or recommended.

*Offset/Raster - There are two choices of the type of fill pattern that will be used to clear away the area to be machined with the Pocket Toolpath, Offset and Raster.

Clear Pocket - Offset Strategy
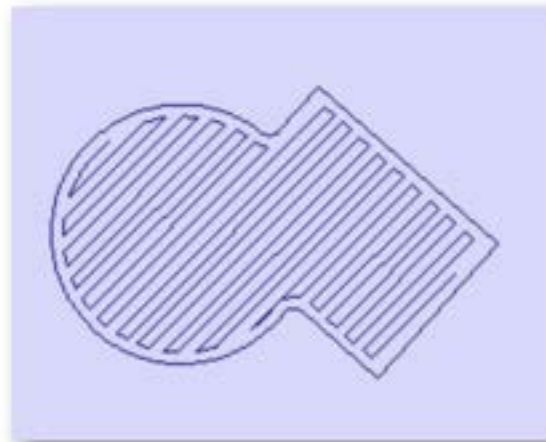
Clear Pocket - Raster Strategy

Offset calculates an offset area clearance fill pattern to machine inside the selected vector(s).

Raster calculates a Raster based area clearance fill pattern to machine inside the selected vector(s). Raster Angle allows you to choose which angle it machines inside your shape. Profile Pass will trace the profile of your interior shape. You can select whether this happens first, last, or no profile pass. This option is recommended if you are surfacing a slab.

*Cut Direction - In most instances, climb cut will be the best option for cut quality, reducing tearout, and extending the life of your router bit.
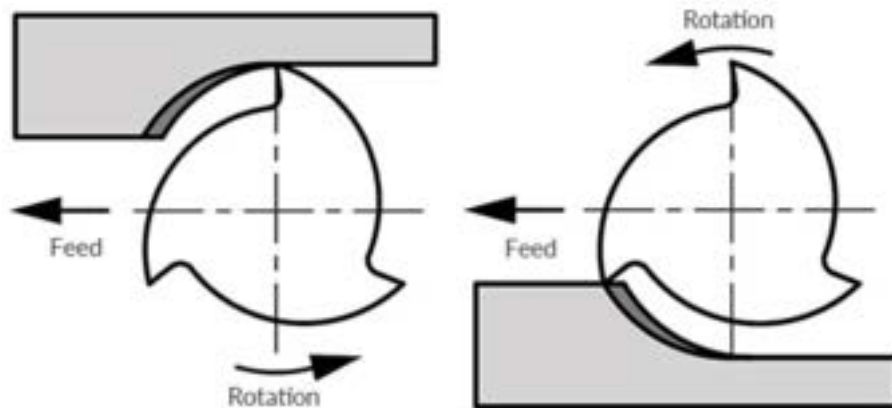
Ramp Plunge Moves - Ramp moves are used to prevent the cutter from plunging vertically into the material. The cutter gradually cuts at an angle dropping into the material significantly reducing cutter wear, heat build-up and also the load on the router spindle and Z axis of the machine. However, this is not a necessary option to create your toolpath.
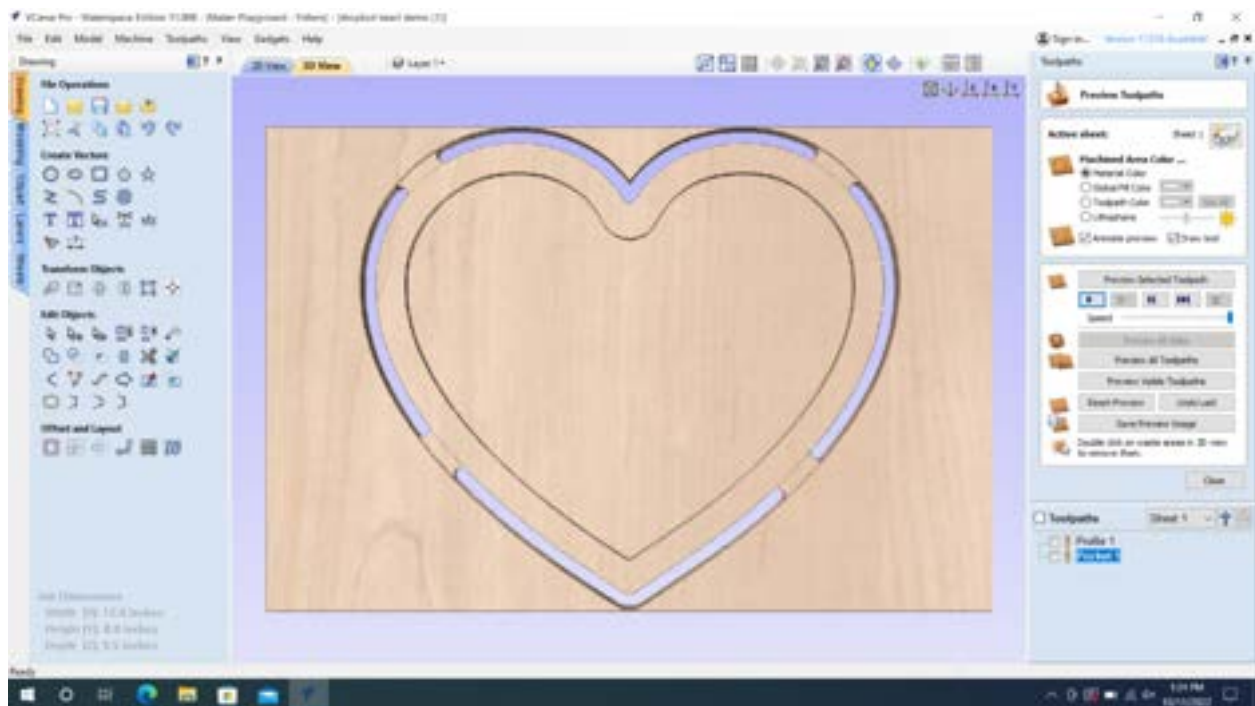
* Asterisks indicate required settings! *

20

Pocket Allowance - Offsets the size of your pocket to the specified measurement.

Vector Selection Order - If this option is checked, the vectors will be machined in the order you selected them. If the option is not checked, the program will optimize the order to reduce machining time.
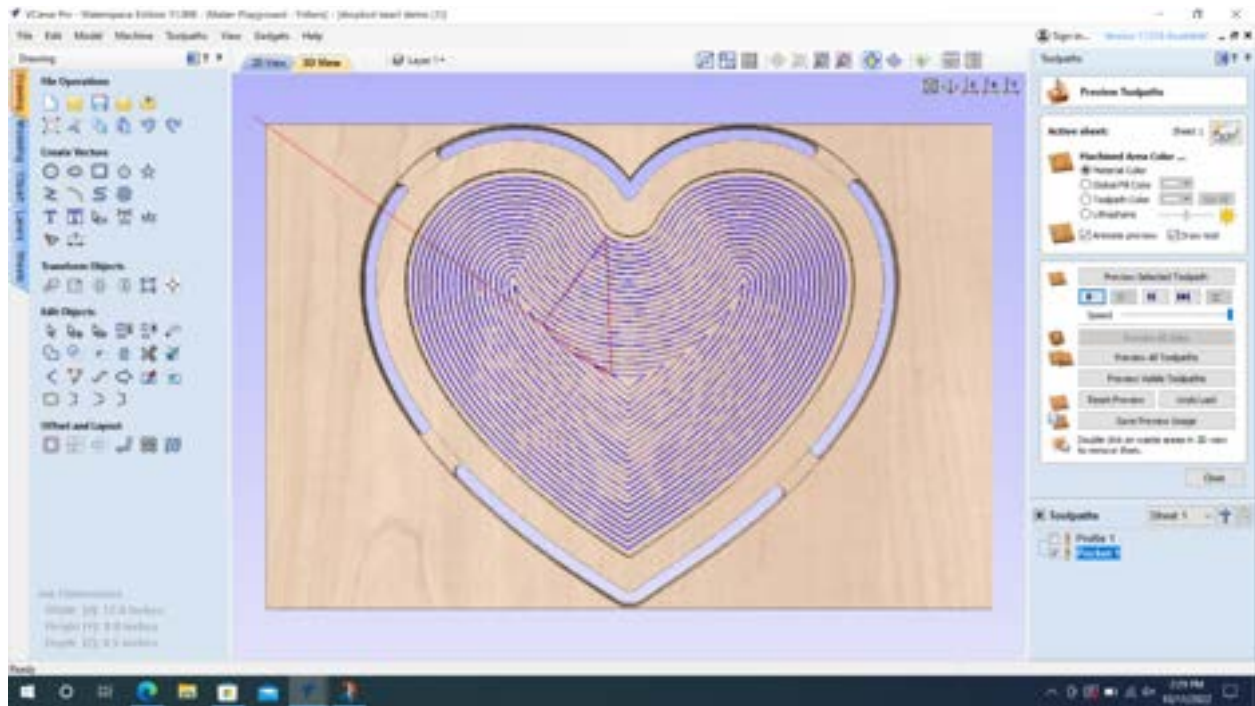
Project toolpath onto 3D Model - This setting is only used when combining your 2D vectors with 3D models. Vectors will be calculated to cut depth from the varying surface of the 3D model.

Name - The name of the toolpath can be entered or the default name can be used.

Click "Calculate" at the bottom of the tab to create your toolpath.

After any toolpath is calculated, it will automatically direct you to the "Preview Toolpaths" tool. Here you can play a simulated preview of exactly what the tool will do during your cut.

If the box next to your toolpath is checked, it will also show you exactly where your tool will go during the cut. Red lines represent jogs, where the tool moves at a safe height above the material. Blue lines are where the tool will be cutting into your material.

To navigate back to toolpath selection, simply click "Close" at the bottom of the panel. To navigate back to your vectors in order to select them for a new toolpath, you'll need to click "2D View" on the top of the screen.

You can always go back and edit toolpaths by double-clicking on the toolpath's name. If you change your vectors, you'll need to click on the toolpaths it affects and recalculate them to update them for any changes you've made.

* Asterisks indicate required settings! *

22

Creating a VCarve Toolpath

**VCarve toolpaths** are used when you want to engrave a detailed design or text in your project with a v-shaped groove bit. VCarve toolpaths DO require a closed shape.

Because the VCarve toolpath uses a v-shaped engraving bit, it is useful to carve small shapes that endmills would not otherwise be able to fit in, such as serifs.


Make sure to select the vector(s) you want to calculate as your Vcarve toolpath(s).
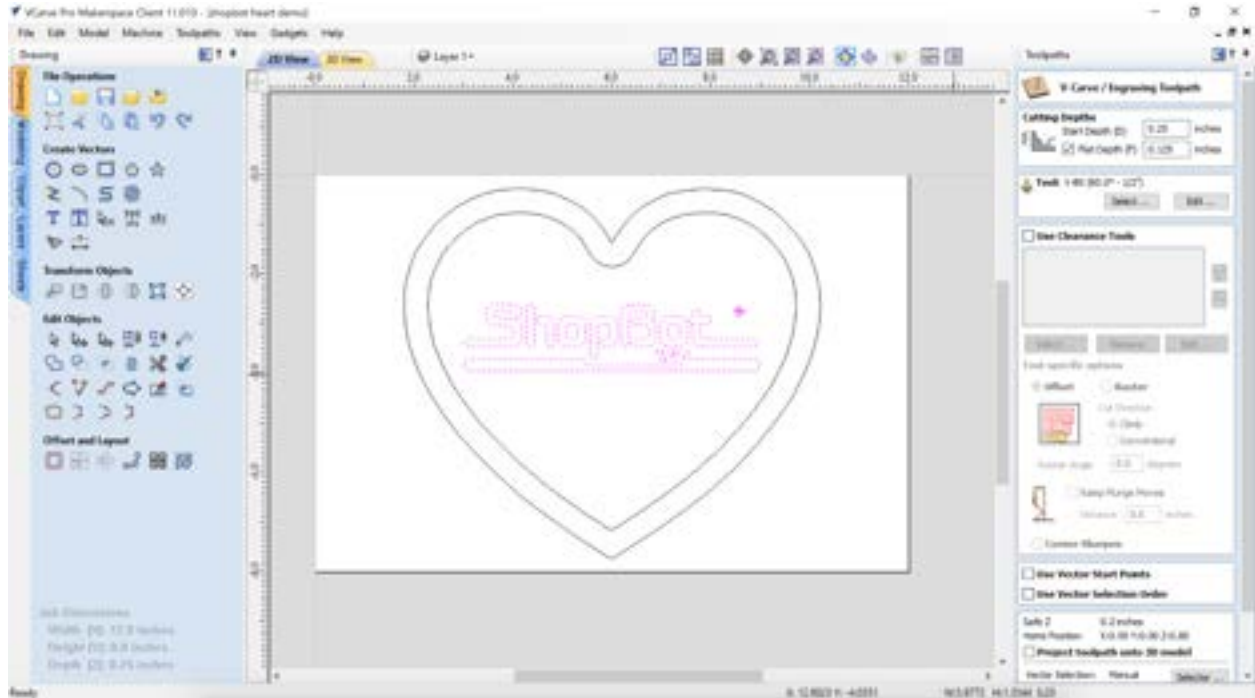
*Start Depth - This is the Z height where the bit will begin cutting on your material and will usually be zero. The only time it would be a different depth is if you have already made cuts that change the width of your material where the cut is being made. This allows cutting inside of an already pocketed region.

*Flat Depth - Flat depth may be necessary if your engraving uses large shapes and your material is not very thick. To ensure the tool never goes completely through your material, **it is good practice to add a flat depth at the deepest point you would want the tool to be able to engrave.**
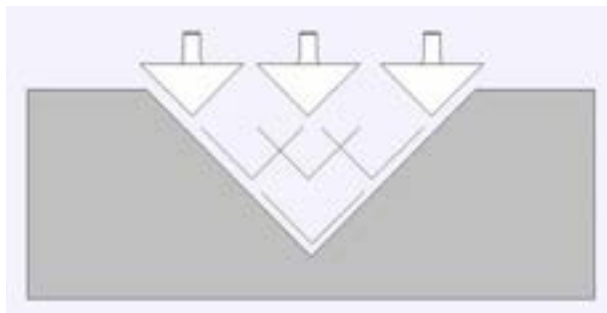
No flat depth   Flat depth

*Tool - Click the Select button opens the Tool Database from which the required tool can be selected. For VCarve toolpaths, these will be v-shaped engraving or lettering bits. Maker Playground will typically have 90 and 60 degree v-bits available for your use. Ask staff or see our CNC Tooling Guide for more information about bits. "Select" allows you to select a different bit, while "Edit" allows you to change the parameters of the current bit.

*Clearance Tools - Check this option if you wish to use an End Mill to machine the large open regions of a design. This can significantly reduce your cutting time and leave a better quality cut on the flat areas of your design. If no tool is selected here but Flat Depth is specified, then the selected engraving bit will be used to clear the flat areas as well as for the VCarving, which may take longer than expected if cutting large shapes.
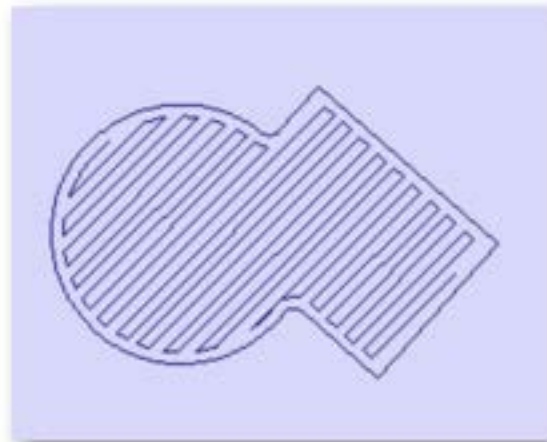

*Offset/Raster - There are two choices of the type of fill pattern that will be used to clear away the area to be machined with the Pocket Toolpath, Offset and Raster.

Clear Pocket - Offset Strategy

Clear Pocket - Raster Strategy

Offset calculates an offset area clearance fill pattern to machine inside the selected vector(s).

Raster calculates a Raster based area clearance fill pattern to machine inside the selected vector(s). **Raster Angle** allows you to choose which angle it machines inside your shape.

*Cut Direction - Both options allow you to choose your cut direction. In most instances, climb cut will be the best option for cut quality, reducing tearout, and extending the life of your router bit.
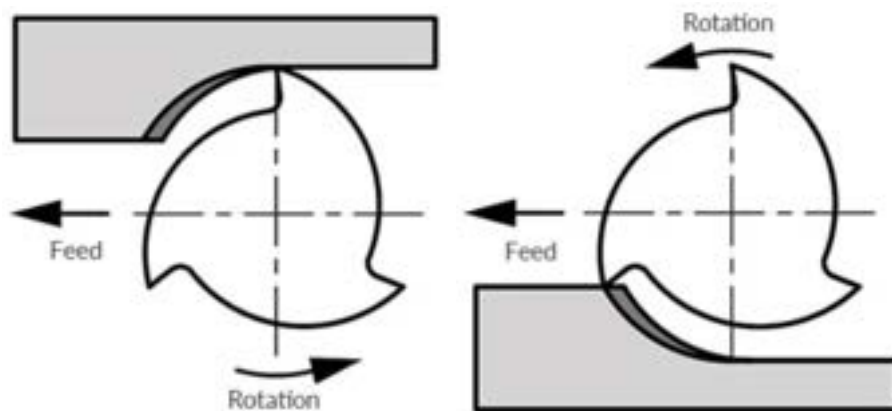
Ramp Plunge Moves - Ramp moves are used to prevent the cutter from plunging vertically into the material. The cutter gradually cuts at an angle dropping into the material significantly reducing cutter wear, heat build-up and also the load on the router spindle and Z axis of the machine.

Corner Sharpen - If this option is checked, the Engraving tool will raise in the corners to fit the small tip into narrower regions. This option is available for a tool positioned second or later in the list.

Vector start points - If this option is checked, the start point of the profile and offset toolpath segments will be as close as possible to the start point of the corresponding boundary vector. Otherwise this is left up to the program.

Vector Selection Order - If this option is checked, the vectors will be machined in the order you selected them. If the option is not checked the program will optimize the order to reduce machining time.

Project toolpath onto 3D Model - This setting is only used when combining your 2D vectors with 3D models. Vectors will be calculated to cut depth from the varying surface of the 3D model.

Name - The name of the toolpath can be entered or the default name can be used.

Click "Calculate" at the bottom of the tab to create your toolpath.

After any toolpath is calculated, it will automatically direct you to the "Preview Toolpaths" tool. Here you can play a simulated preview of exactly what the tool will do during your cut.

If the box next to your toolpath is checked, it will also show you exactly where your tool will go during the cut. Red lines represent jogs, where the tool moves at a safe height above the material. Blue lines are where the tool will be cutting into your material.

To navigate back to toolpath selection, simply click "Close" at the bottom of the panel. To navigate back to your vectors in order to select them for a new toolpath, you'll need to click "2D View" on the top of the screen.

You can always go back and edit toolpaths by double-clicking on the toolpath's name. If you change your vectors, you'll need to open the toolpaths it affects and recalculate them to update them for any changes you've made.

* Asterisks indicate required settings! *

28

Once all of your toolpaths are calculated, you're ready to move on to setting up your file at the CNC router.
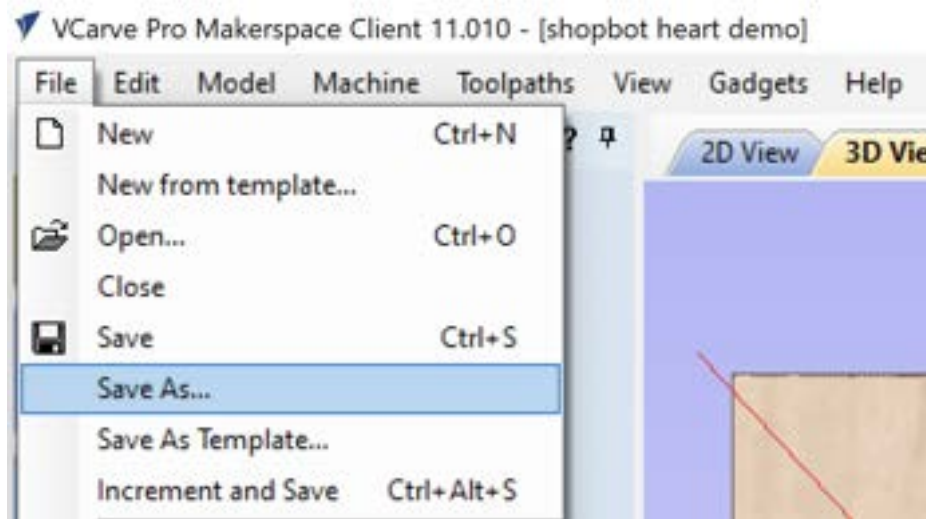
Save your file as a .crv by simply clicking File > Save As… in the upper left corner and naming your file.
We will wait to export our toolpaths when we get to the CNC so we can reference our VCarve files to see our origin points, material orientation, etc.

View "Setting up your file on the ShopBot CNC Router Guide" to proceed!

Be sure to reserve the CNC Router ahead of your visit at Maker Playground!

ShopBot CNC Router Reservation Procedure

*You must have taken the CNC Routing Shop Class with Maker Playground to be certified to reserve time on the ShopBot.

Please have your VCarve file ready prior to your reservation on the CNC Router in order to keep the ShopBot desktop dedicated to running files. This can be prepared on MPG's laptops or on your personal computer. Enter the Makerspace Client Code below in the trial version of VCarve to allow your files to be opened in VCarve on the ShopBot Desktop. Many helpful tutorials for VCarve can be found online. If you need staff assistance, complete the file to the best of your ability, and staff will be able to assist with finalizing the file for the ShopBot.

> Makerspace Client Code: 251BE-8E15B-4705B-C38B2-67D8F-6E7C8-3AA39

Check your feed and speeds for the given bit and material with Maker Playground staff before exporting your toolpaths for ShopBot. If you would like to bring your own CNC bits, make sure they have either a 1/4" or 1/2" shank for our collets.

VCarve will provide a rough time estimate for your file. Be sure that the time estimate fits within your reserved time slot. Most projects take less than an hour to complete.

Maximum Job Size: 96x48x8"

Software: VCarve Pro

Compatible File Types:

> vector: .dxf, .dwg, .eps, .ai, .pdf, .pvc, v3d, .v3m, .crv, .skp, .svg
>
> image: .bmp, .jpg, .gif, .tif, .tiff, .png, .jpeg
>
> 3D model: .stl, .v3m, .3dm, .skp, .rlf, .3ds, .asc, .prj, .x, .dxf, .lwo, .wrl, .obj

Approved Materials:

- Solid wood
- Plywood
- MDF
- Dense foam
- Plastic
- Aluminum (prior clearance by MPG staff required)

Available bits:

- 1/4 " Endmill
- 1/2 " Endmill
- 1/8 " Endmill
- 1/4 " O-Flute (Acrylic)
- 60° Engraving V-Groove
- 1/4" Diameter Up Spiral Ball Nose (3D Milling)

Staff will typically have the above bits available for your use, and we may have additional bits for specific projects. Ask staff for a complete, up-to-date list of bits. If you are looking for a more specific router bit, you may need to bring in your own. If you are surfacing slabs or river tables, you will need to provide your own surfacing bit.