



# **Laser Engraving Project**

## **Quick Engrave**

The toolpaths generated in this tutorial are for demonstration purposes only. These will need to be modified depending upon your machine, material, and the laser accessory you have.

## Create a New Project:

Open the Vcarve/Aspire software and select "Create a new file".

Enter the following parameters for the Job Setup

1. Job Type: Single Sided

2. Job Size:

Width (X): 5 inches

Height (Y): 5 inches

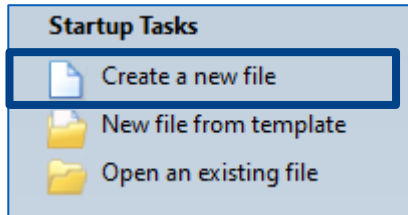
Thickness (Z): 0.75 inches

Units: Inches

3. Z Zero Position: Material Surface

4. XY Datum Position: Bottom Left Corner

Select OK

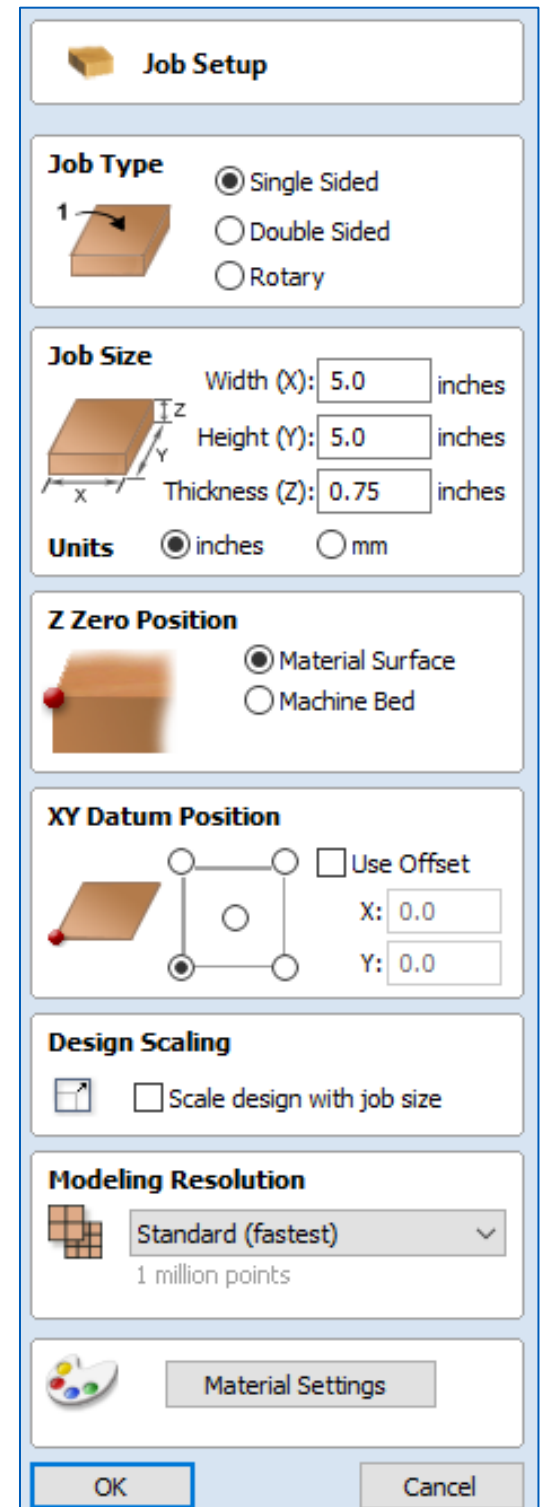


1

2

3

4

A screenshot of the 'Job Setup' dialog box in the software. It contains several sections: 'Job Type' with radio buttons for 'Single Sided' (selected), 'Double Sided', and 'Rotary'; 'Job Size' with input fields for 'Width (X): 5.0 inches', 'Height (Y): 5.0 inches', and 'Thickness (Z): 0.75 inches'; 'Units' with radio buttons for 'inches' (selected) and 'mm'; 'Z Zero Position' with radio buttons for 'Material Surface' (selected) and 'Machine Bed'; 'XY Datum Position' with a diagram of a square and a circle, a 'Use Offset' checkbox, and input fields for 'X: 0.0' and 'Y: 0.0'; 'Design Scaling' with a checkbox for 'Scale design with job size'; 'Modeling Resolution' with a dropdown menu set to 'Standard (fastest)' and '1 million points'; and a 'Material Settings' button. At the bottom are 'OK' and 'Cancel' buttons.

**Job Setup**

**Job Type**

☒ Single Sided  
☐ Double Sided  
☐ Rotary

**Job Size**

Width (X): 5.0 inches  
Height (Y): 5.0 inches  
Thickness (Z): 0.75 inches

**Units** ☒ inches ☐ mm

**Z Zero Position**

☒ Material Surface  
☐ Machine Bed

**XY Datum Position**

☐ Use Offset  
X: 0.0  
Y: 0.0

**Design Scaling**

☐ Scale design with job size

**Modeling Resolution**

Standard (fastest)  
1 million points

Material Settings

OK Cancel

## Import the Image:

Select File from the top left corner

Scroll down to Import

Select Import bitmap

From the pop-up window, navigate to and select the Shark\_Logo.png file downloaded from the Laser Project article.

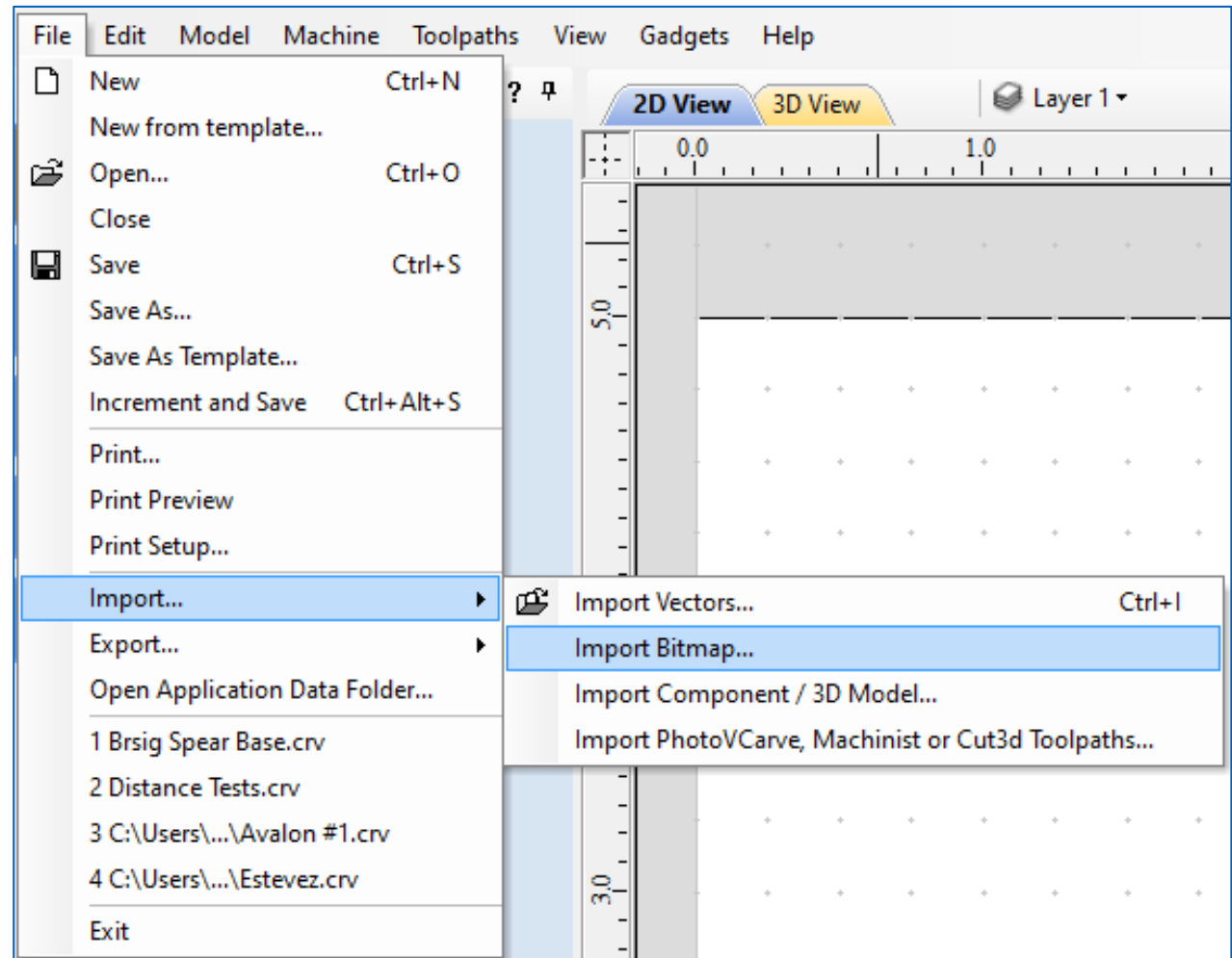
Then select Open

### Vcarve Tip:

Many other file types can also be imported into Vcarve for laser engraving or milling.  
Below are a few of the file types.

Import Vectors file types: dxf, dwg, eps, ai, skp, crv, crv3d, svg

Import Bitmap file types: bmp, jpg, gif, tif, tiff, png, jpeg



## Bitmap Trace:

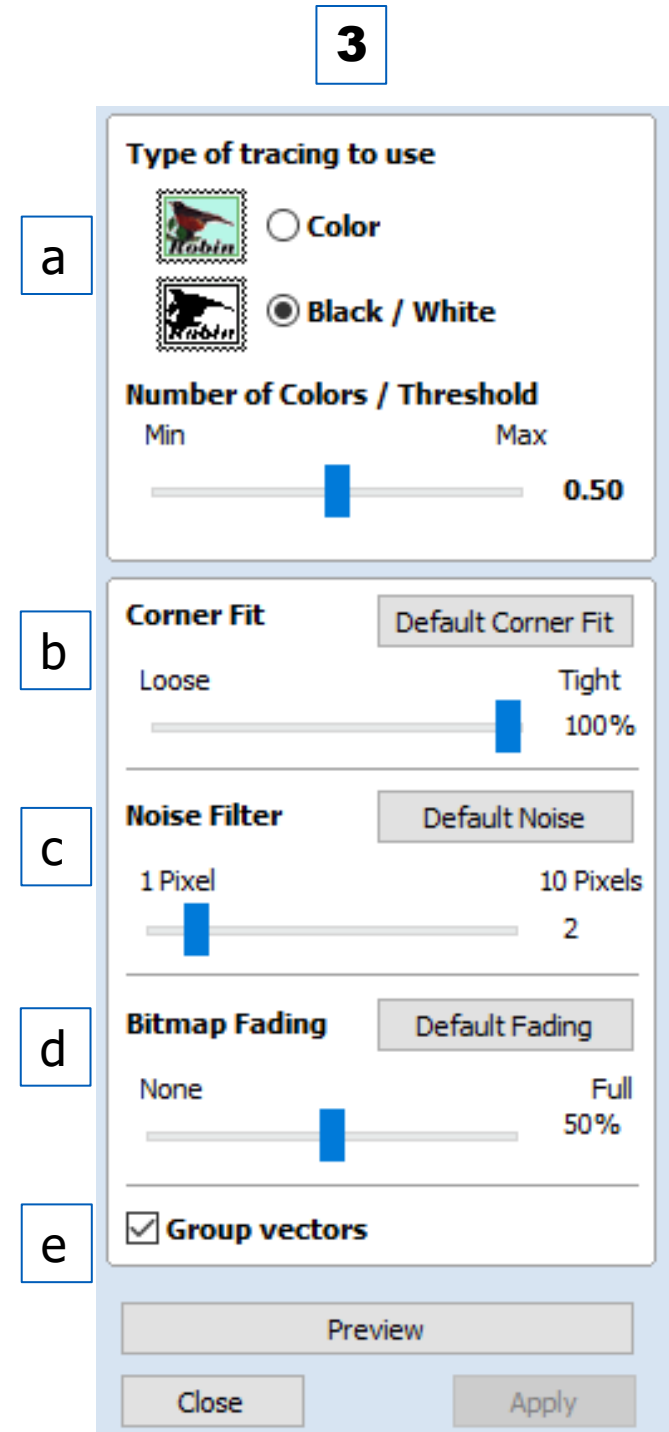
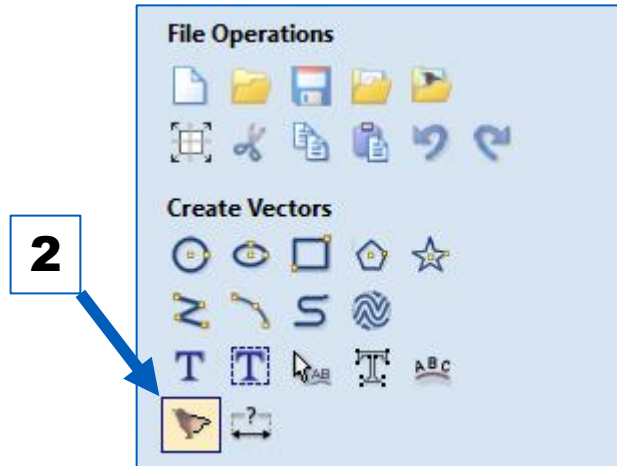
1. Left-click the Shark\_Logo image

When selected, the image will have a pink/purple dotted line around it and the colors of the image will be brighter

2. Select Trace Bitmap from the Create Vectors section on the left side of Vcarve

3. Use the Settings:

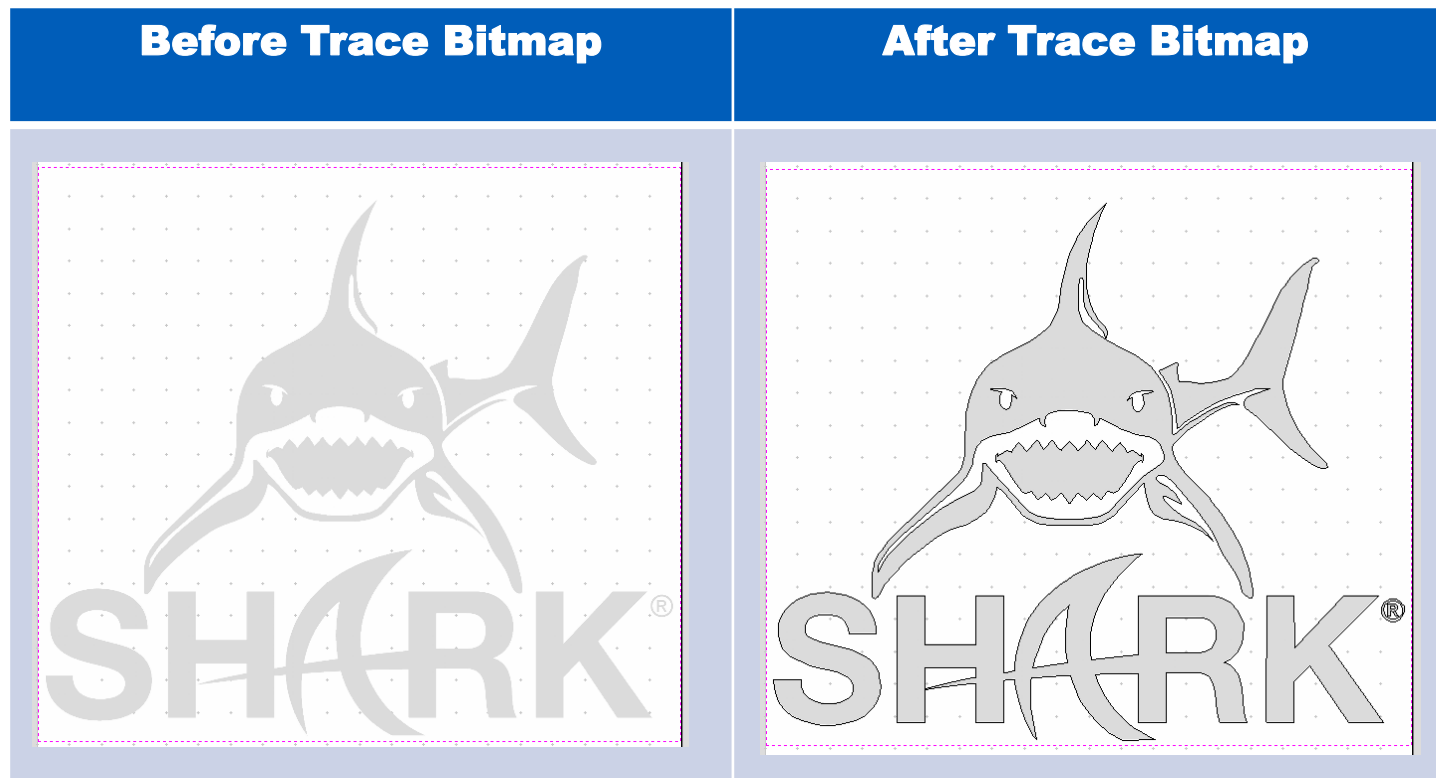
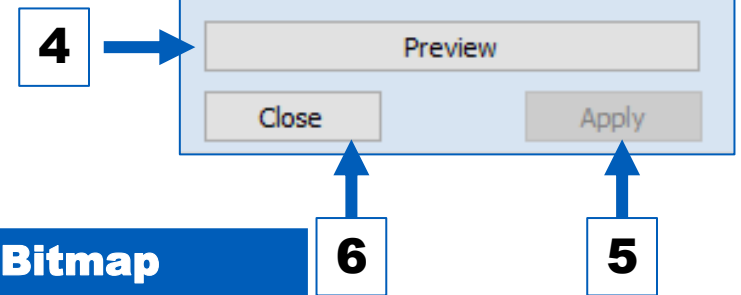
- a. Black/White
- b. Corner Fit - 100%
- c. Noise filter - 2 Pixels
- d. Bitmap Fading: 50%
- e. Group Vectors checked



## Bitmap Trace: continued

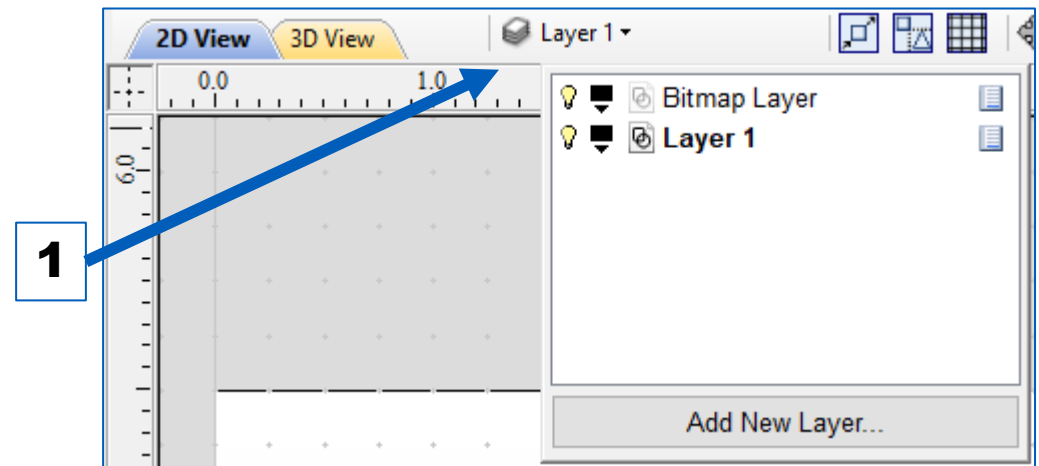
4. Select Preview to make sure the Vector lines are applied around the image
5. Select Apply to save the Vector lines
6. Finally, select Close to exit Trace Bitmap

Illustrated below is how the vector lines should look



## Hide the Image:

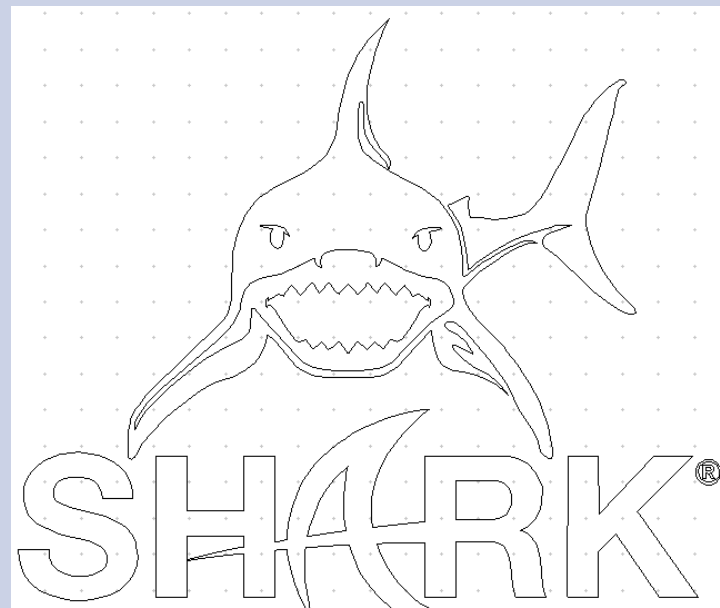
1. Go to Layer 1 at the top center, to the right of the 3D View tab and select the drop down.
2. Click on the Light Bulb next to Bitmap Layer to turn off/hide the image.



**Before Hide Bitmap**



**After Hide Bitmap**



### **Add the Laser Engraver:**

For complete steps for adding the laser accessory to your Vcarve/Aspire tool database [Click Here](#).

It is suggested the [Laser Feed and Power Grid Testing](#) should be done to determine the best Feed rate and Power setting for your laser and the material you will be engraving on. Below are the recommended starting parameters if you have not done the Laser Feed and Power grid.

<b>2-Watt Laser Accessory</b>	<b>7-Watt Laser Accessory</b>
Name: 2W Laser Engraver Tool Type: End Mill	Name: 7W Laser Engraver Tool Type: End Mill
Geometry Units Inches Diameter: 0.01	Geometry Units Inches Diameter: 0.01
Cutting Parameters Pass Depth: 0.01 Stepover: 0.01	Cutting Parameters Pass Depth: 0.01 Stepover: 0.01
Feeds and Speeds Spindle Speed: 1000 Feed Units: inches/min Feed Rate: 50 Plunge Rate: 20	Feeds and Speeds Spindle Speed: 500 Feed Units: inches/min Feed Rate: 100 Plunge Rate: 20



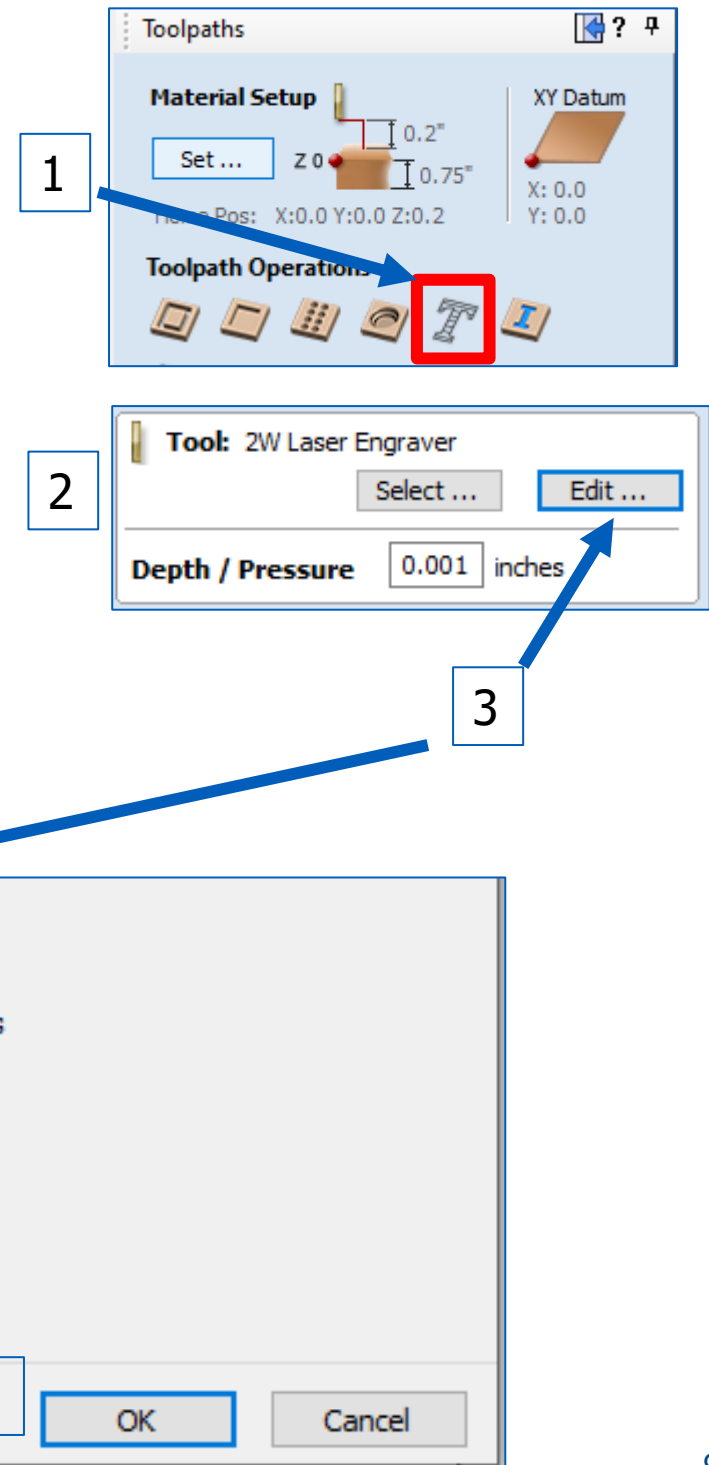
## Applying a Toolpath – Quick Engrave Outline:

1. Select the Quick Engrave toolpath from Toolpath Operations on the right-hand side of the Vcarve/Aspire software
2. Under Tool: Press Select to bring up the Tool Database Find and select the Laser Engraver you will be using
3. If you ran the [Laser Feed and Power Grid](#) test, select Edit and make any changes to the spindle speed and feed rate to best suit the material you will be engraving on.

If you have not ran the Laser Feed and Power Grid, use these basic settings laser Feed rate and Spindle Speed settings.

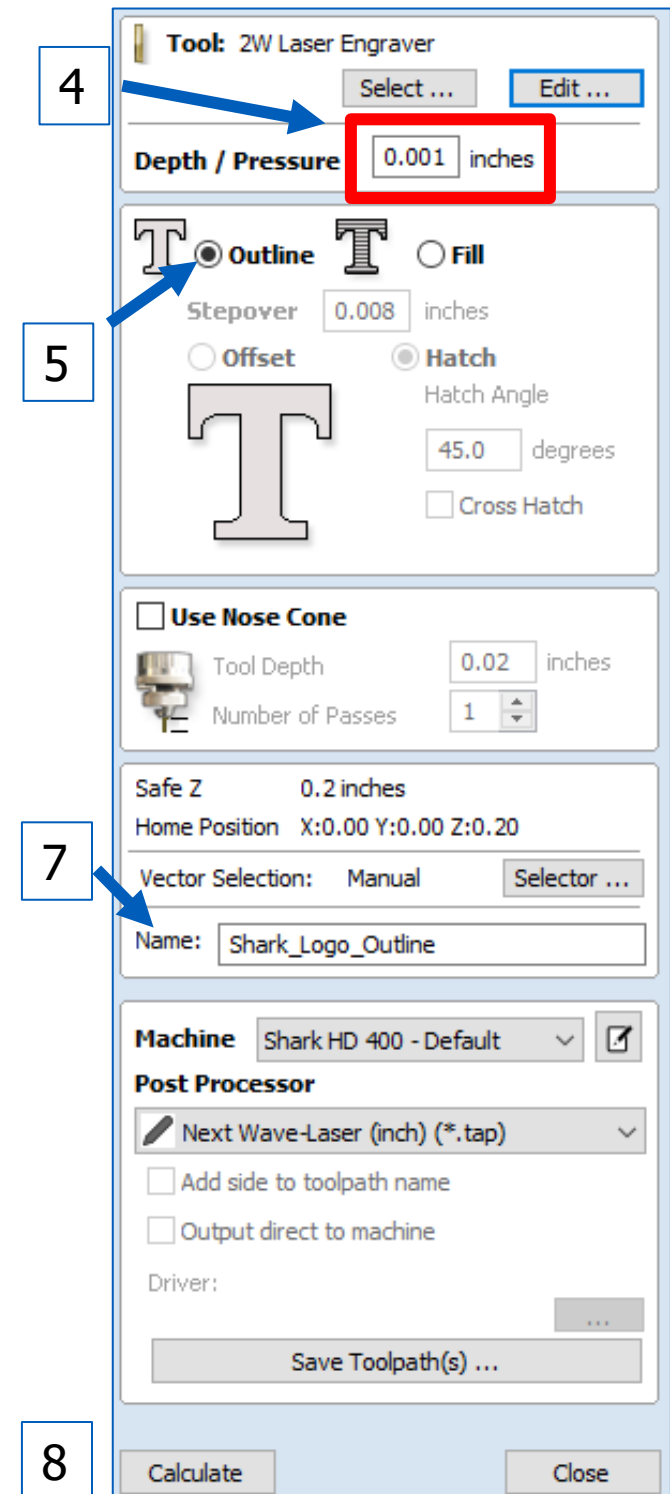
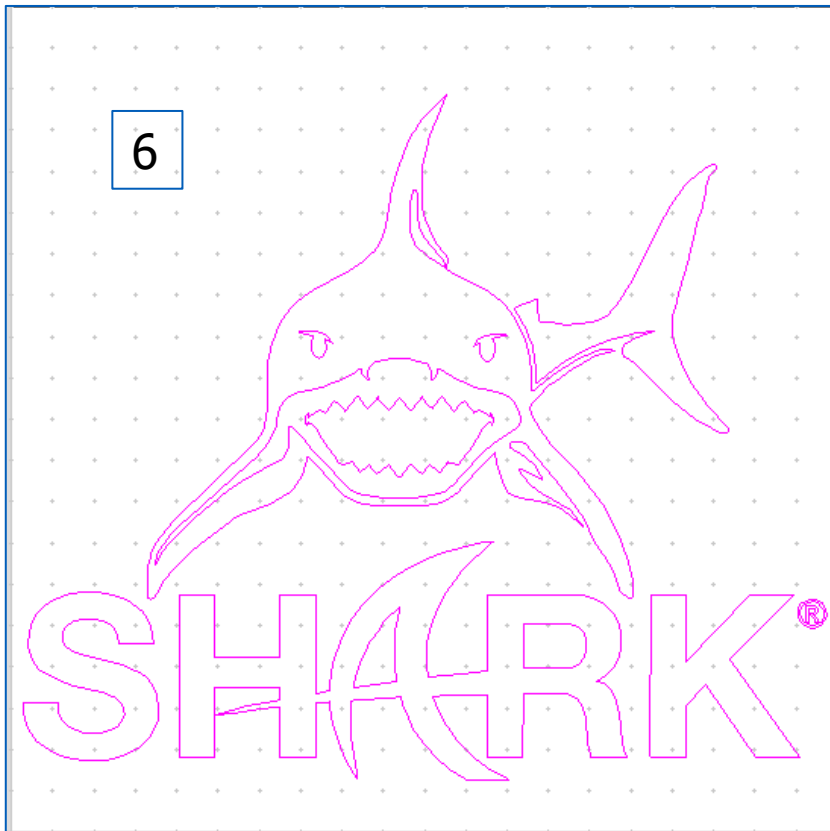
- 2W laser: 1000 rpm / Feed 50 ipm / Plunge 20-45 ipm
- 7W laser: 500 rpm / Feed 100 ipm / Plunge 20-45 ipm

4. Select OK to save these changes.



## Applying a Toolpath – Quick Engrave Outline: Continued

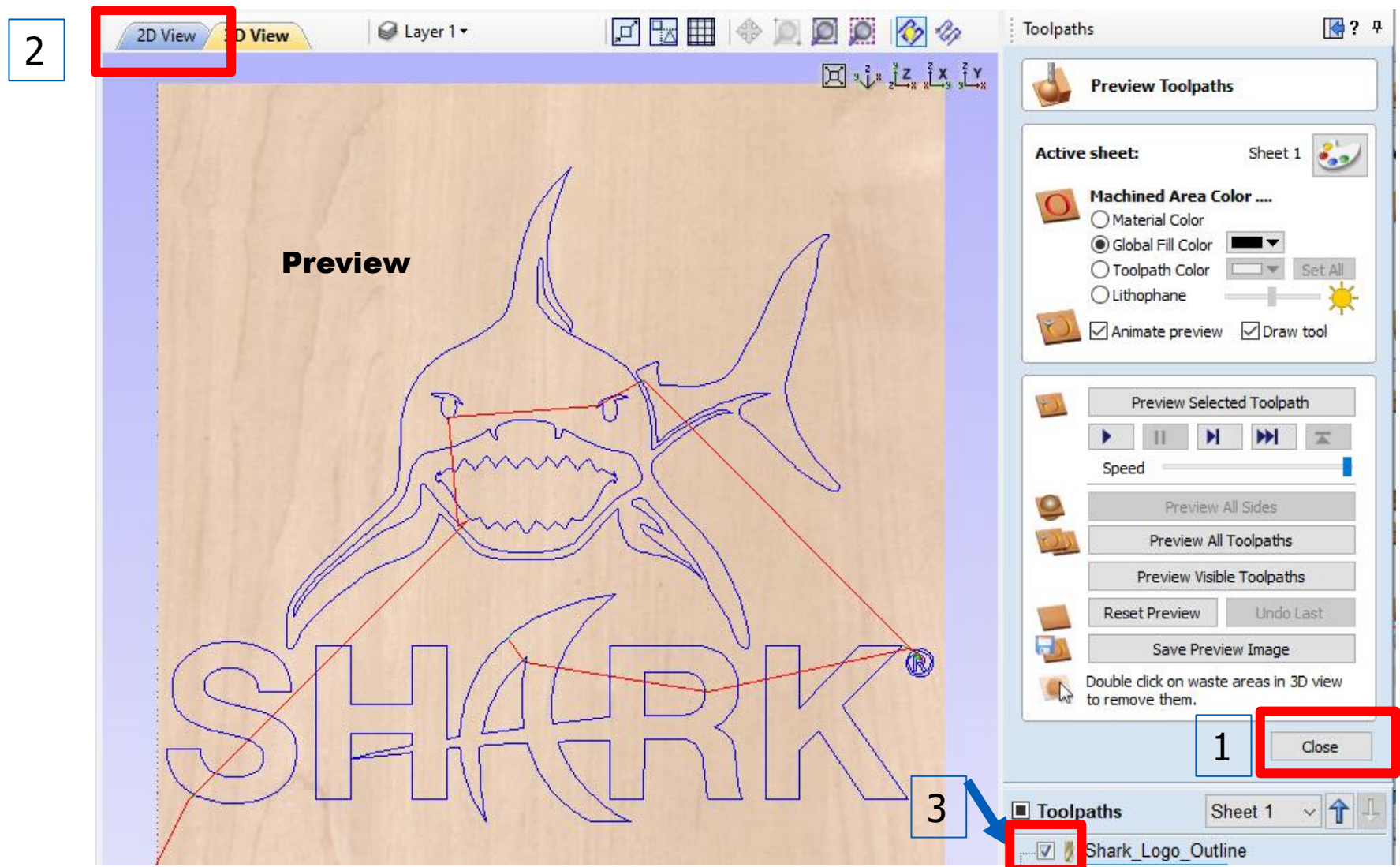
4. Set the Depth / Pressure to 0.001"
5. Select the radial button for Outline
6. Left-click on the vector lines for the Shark Logo to highlight them in pink/purple
7. Name the toolpath "Shark\_Logo\_Outline"
8. Select Calculate at the bottom to create the toolpath



## Preview the toolpath - Outline

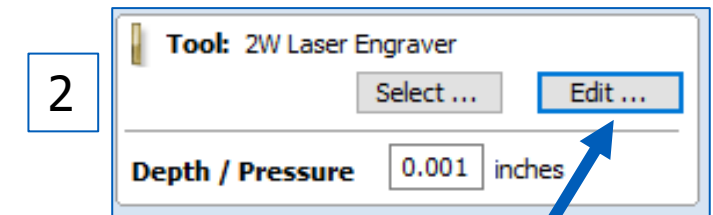
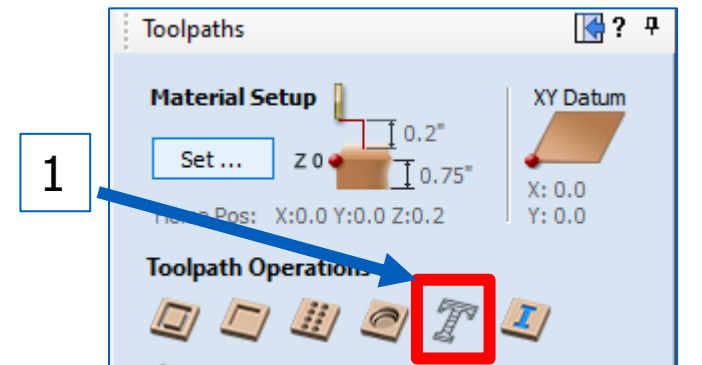
Once a toolpath has been created, the Preview Toolpaths tab will immediately come up and show the path the laser will follow to create the outline of the Shark Logo. Because the depth of cut is set to 0.001" it will be difficult to preview and see the actual toolpath.

1. Select Close to exit out of the preview.
2. Click on the 2D View to view the vector lines
3. Uncheck the box next to Shark\_Logo\_Outline to remove the toolpath indicators from the 2D view.

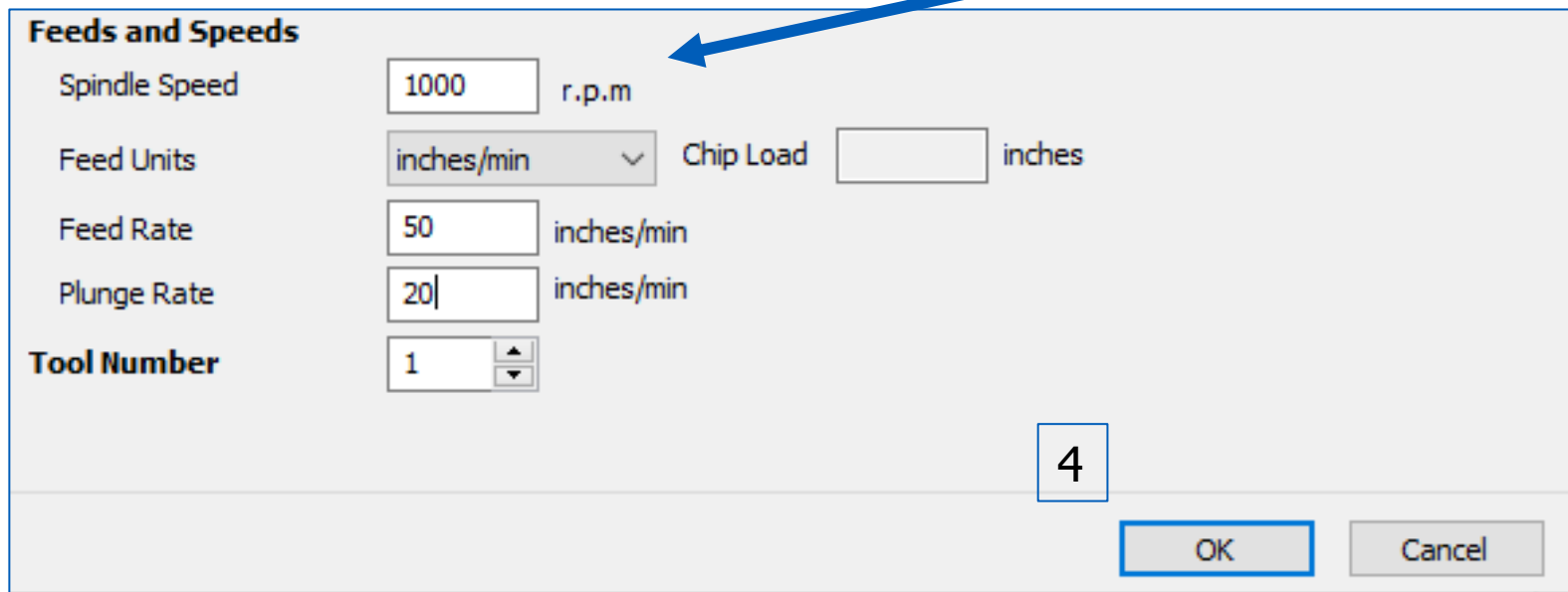


## Applying a Toolpath – Quick Engrave Fill:

1. Select the Quick Engrave toolpath from Toolpath Operations on the right-hand side of the Vcarve/Aspire software
2. Under Tool: Press Select to bring up the Tool Database  
Find and select the Laser Engraver you will be using
3. Select Edit and make any changes to the spindle speed and feed rate to best suit the material you will be engraving on.
4. Select OK to save these changes.

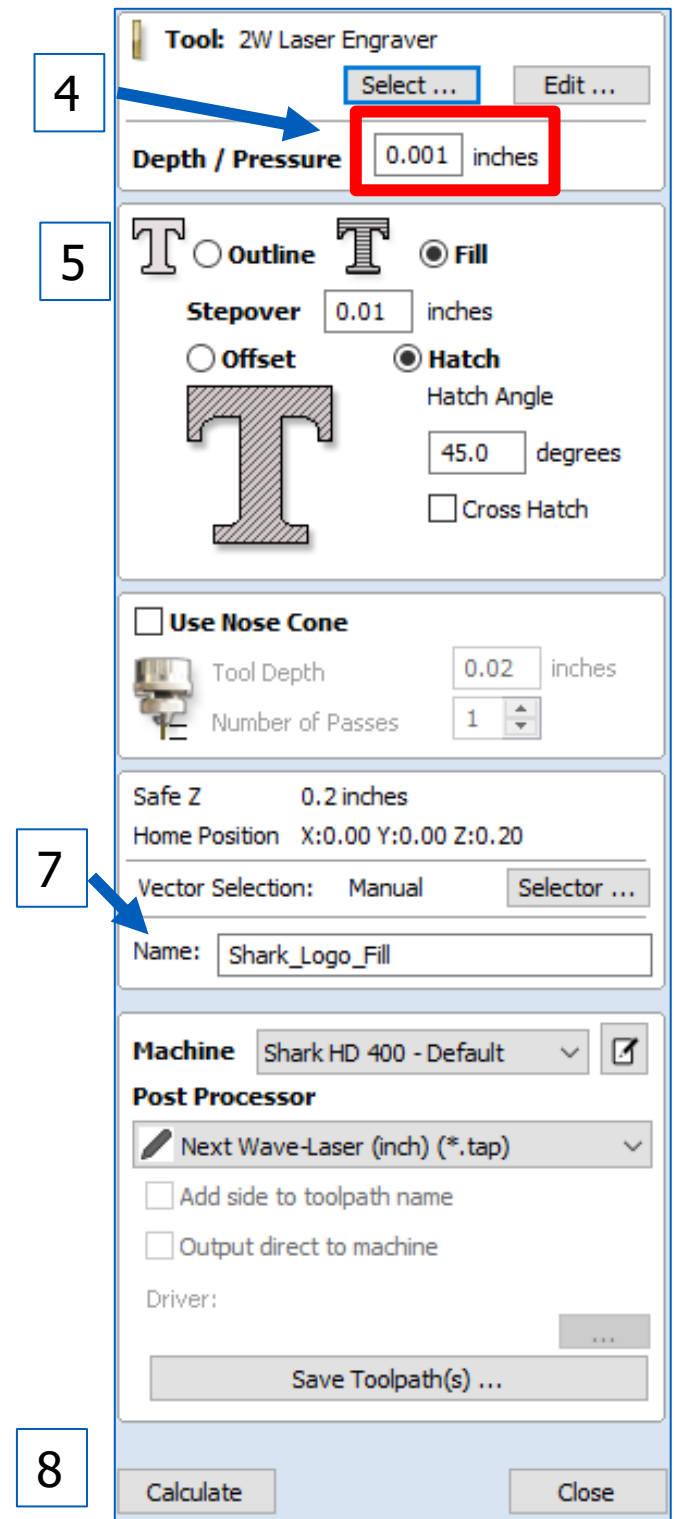
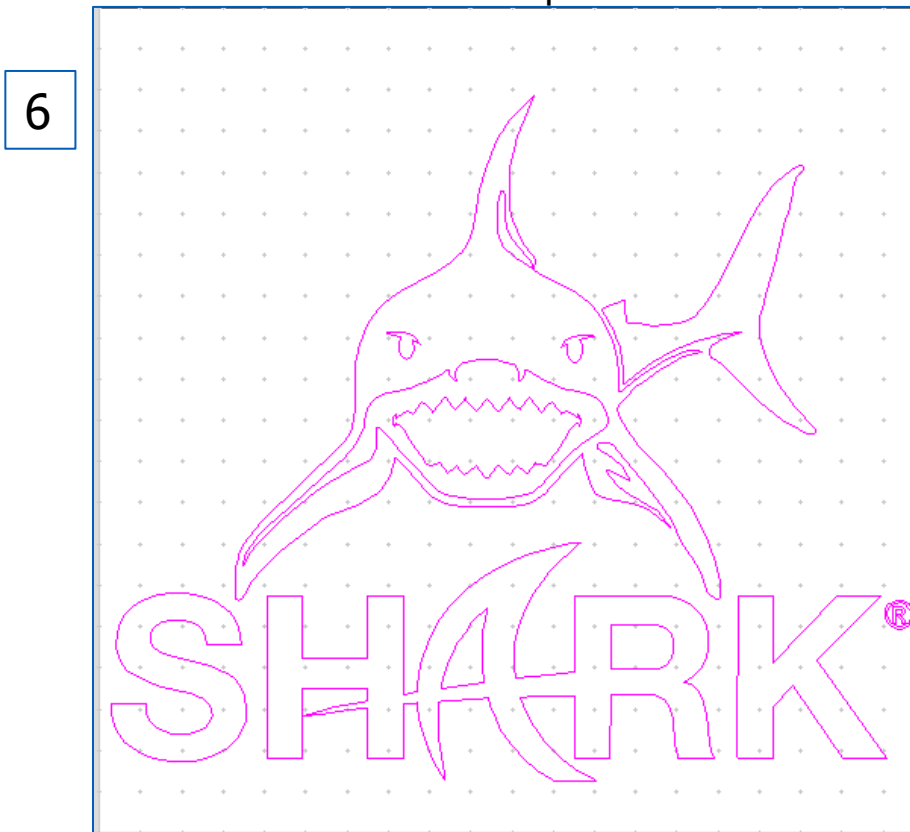


3



## Applying a Toolpath – Quick Engrave Fill: Continued

4. Set the Depth / Pressure to 0.001"
5. Select the radial button for Outline  
Set Stepper to 0.01 inches  
Select Hatch  
Enter 45 degrees for the Hatch Angle
6. Left-click on the vector lines for the Shark Logo to highlight them in pink/purple
7. Name the toolpath "Shark\_Logo\_Fill"
8. Select Calculate at the bottom to create the toolpath

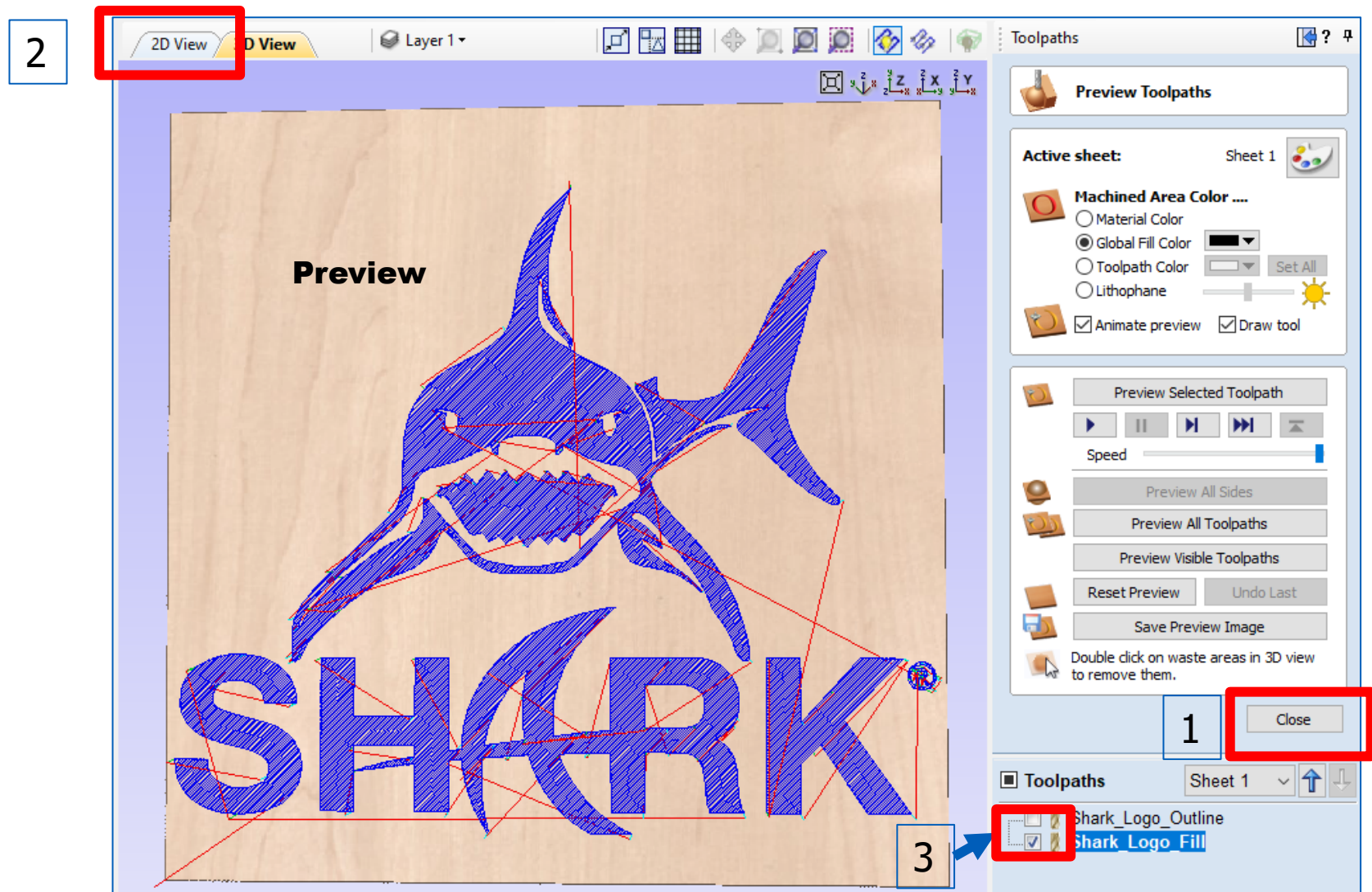




## Preview the toolpath - Fill

Once a toolpath has been created, the Preview Toolpaths tab will immediately come up and show the path the laser will follow to create the outline of the Shark Logo. Because the depth of cut is set to 0.001" it will be difficult to preview and see the actual toolpath.

1. Select Close to exit out of the preview.
2. Click on the 2D View to view the vector lines
3. Uncheck the box next to Shark\_Logo\_Outline to remove the toolpath indicators from the 2D view.



## Save a Toolpath Option 1: Individual files

1. Select the Save Toolpath button under Toolpath Operations
2. Left-click on "Selected toolpath"
3. Left-click on the Shark\_Logo\_Outline toolpath from the Toolpaths list so it is highlighted in blue.
  - a. The toolpath will display under "Toolpaths to be saved"
4. Select Next Wave-Laser(inch)(\*.tap) from the Post Processor drop down list.
5. Select Save Toolpath

Save the toolpath to your flash drive if you will be running the file through the LCD pendant.

### OR

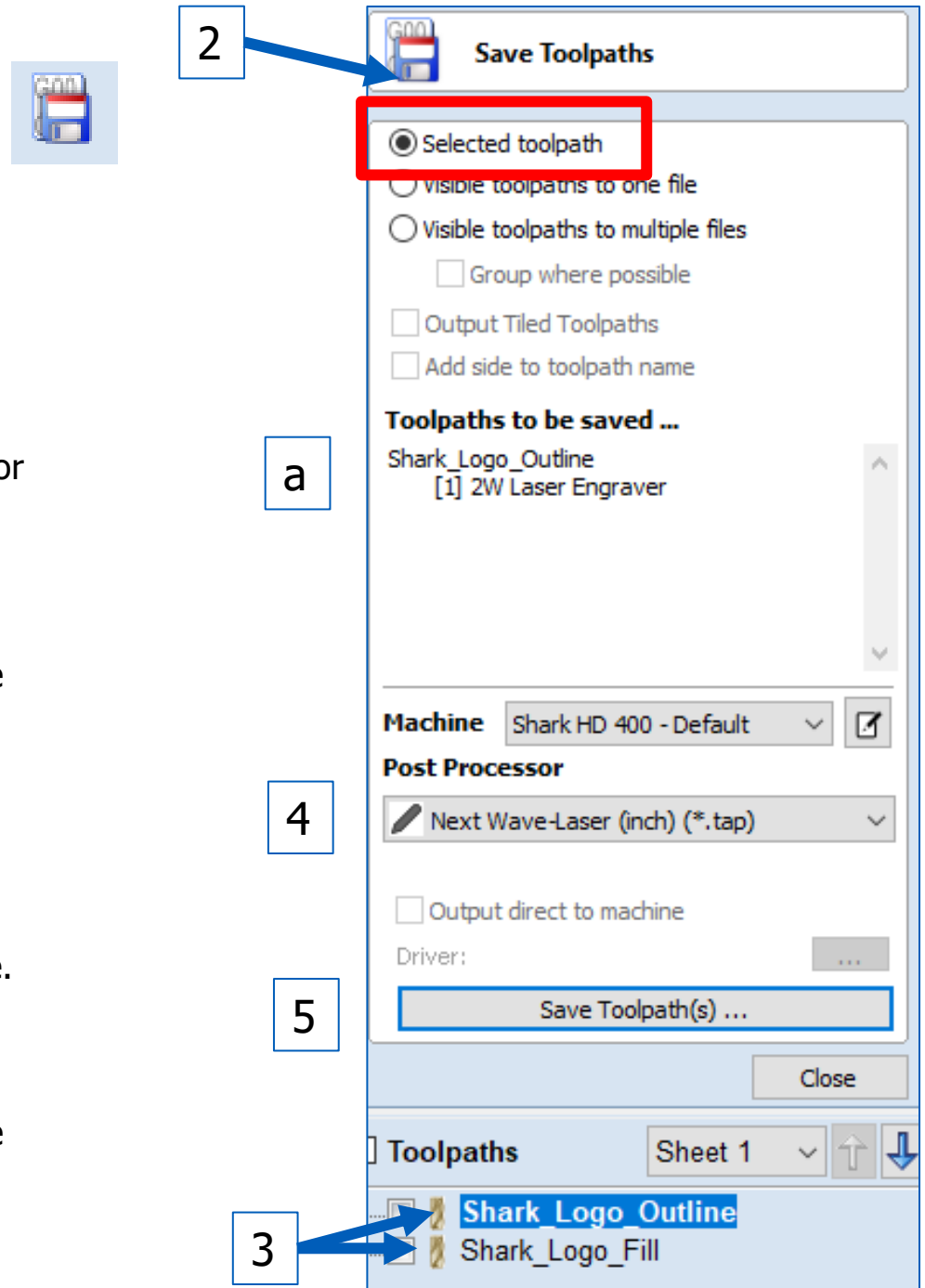
Save the toolpath to a folder or location on your computer to run the file through Ready2Control.

3. To save the second toolpath, left-click on the Shark\_Logo\_Fill toolpath name so that it is highlighted in blue.
5. Select Save Toolpath.

Save the toolpath to your flash drive if you will be running the file through the LCD pendant.

### OR

Save the toolpath to a folder or location on your computer to run the file through Ready2Control.



## Save a Toolpath Option 2: One file

1. Select the Save Toolpath button under Toolpath Operations
2. Left-click on "Visible toolpaths to one file"
3. Check the boxes next to Shark\_Logo\_Outline and Shark\_Logo\_Fill toolpath from the Toolpaths
- a. Both toolpaths will display under "Toolpaths to be saved"
4. Select Next Wave-Laser(inch)(\*.tap) from the Post Processor drop down list.
5. Select Save Toolpath

Save the toolpath to your flash drive if you will be running the file through the LCD pendant.

**OR**

Save the toolpath to a folder or location on your computer to run the file through Ready2Control.

**You are now ready to run the laser engraving file on your CNC machine.**

