

VCarvePro

Aspire

Getting Started

A quick start guide for
VCarve Pro & Aspire
users

Vetric Ltd.

Document V.6.0 V3.0



Tutorial 3

Kitchen Cabinet Door

Getting Started with Aspire & VCarve Pro

Disclaimer

All CNC machines (routing, engraving, and milling) are potentially dangerous and because Vectric Ltd has no control over how the software described in this manual might be used. Vectric Ltd or any associated Resellers cannot accept responsibility for any loss or damage to the work piece, machine or any individual, howsoever caused by misusing the software. Extreme care should always be taken and the output from the software thoroughly checked before sending it to a CNC machine.

The information in this manual may be subject to change without any prior notice. The software described in this manual is supplied under the terms and conditions of the software license agreement and may only be used in accordance with the terms of this agreement.

© Vectric Ltd
Unit 3 Dunstall Court
Astwood Lane
Feckenham
B96 6QH
UK

www.vectric.com

E-mail info@vectric.com
Phone +44 (0) 1527 460 459
Fax +44 (0) 1527 460 459

Table of Contents

Getting Started with Aspire & VCarve Pro	ii
Disclaimer	ii
Table of Contents	i
Introduction	2
What is V-Carving?	2
What the software allows you to do	3
What file formats can be used?	3
Getting Help	4
Watch the supporting tutorial videos.....	4
Overview of the interface.....	4
The Machining Logic	6
Managing windows - Auto Show / Hide.....	7
View Controls	8
2D Design Window	8
3D Window	8
Working with Vectors.....	9
Vector selection methods	9
Vector deselect.....	9
Vector editing.....	9
Drawing shapes.....	10
Tutorial 3 Kitchen Cabinet Door	11
Introduction	11
1. Opening the Vector Clipart Design	12
2. Calculating the V-Carve Toolpaths.....	13
3. Calculating the Edge Profile Toolpath	17
4. Calculating the Profile Cut Out Toolpath.....	20
5. Previewing the Job.....	22
6. Estimate Cutting Times and Save Toolpaths.....	24

Introduction

Many businesses use their CNC machine for simply cutting out flat letters and shapes from plastic sheet, or engraving standard badges and nameplates, which are all based on simple 2D machining strategies. VCarve Pro can handle all these day to day tasks, however this tutorial will show you how to use your CNC machine to route and engrave jobs that include decorative 3D designs that will be more interesting and hopefully more profitable if you run a business.

The manual takes you step-by-step through an illustrated tutorial that shows and explains exactly how to use the Aspire and VCarve Pro Software. Tips and tricks have also been included that will help you get the most from your CNC machine.

We hope you enjoy using the software.

What is V-Carving?

V-Carving produces a constantly varying and flowing 3D carved effect on the job, which is similar to how a craftsman would carve by hand. Imagine a 'hand-carver' cutting letters into a piece of wood or stone, starting at a sharp corner, pushing the chisel deeper where the font stroke gets wider and pulling the tool out to form precise, sharp corners. V-Carving, also known as 3D Engrave or Intaglio engraving allows a V shaped or engraving tool to cut at varying Z depths that are directly linked to the width of the geometry in which the cutter is moving.

This effect is difficult to describe in words, but imagine using a flat-bottom end mill to cut 3mm (1/8th) deep inside the text shown below. The tool, being round will always leave a fillet radius in the corners and will not actually cut the complete letter where the diameter of the tool is too big to pass through the small gaps.



Sign including V-Carved Text

What the software allows you to do

V-Carving is typically used in the following industries to add decoration to objects and products such as,

Sign making	House signs, Business, Restaurants, Pubs, Gold Leafed and Gilded
Woodworking	Kitchen cabinet doors, Chairs, Doors, Table tops
Engraving	Commemorative Brass plaques, Company logos
Gifts	Key rings, Personalized gifts
Stone cutting	Memorials, Commemorative engravings

What file formats can be used?

Aspire & VCarve Pro will open and import vector files that have been saved in the following formats.

DXF	Drawing Exchange Files from CAD systems
DWG	AutoCAD DWG up to and including version 2008 files
EPS	Encapsulated Postscript from Adobe Illustrator and Corel Draw etc.
AI	Adobe Illustrator
PDF	Portable Document Format for industry standard print data
BMP, JPG, TIF, GIF	Image files can also be imported and Traced / Vectorized for machining.

If the designs are being prepared with software such as Corel Draw or Adobe Illustrator we recommend that you convert the vector geometry and text to curves and switch off all patterns or colour fills before exporting, preferably as an EPS file.

Getting Help

If you need assistance when using the software there are 5 primary places to look.

1. **Program Help File** - From the Main menu select Help
2. **Video Tutorials** - These are supplied on the installation CD or can be downloaded from the Vectric website.
3. **User Forum** - The Vectric user forum at www.vectric.com/forum is a very useful resource for information on VCarve Pro along with materials, cutters etc. and also to share knowledge and experiences.
4. **E-mail Support:** - The Vectric Support Team at support@vectric.com
5. **Frequently Asked Questions (FAQ)** - The support area on the Vectric web site at www.vectric.com maintains a list of the most frequently asked questions along with the answers.

Watch the supporting tutorial videos



The video camera icon indicates there is a video file for that particular section of the manual.

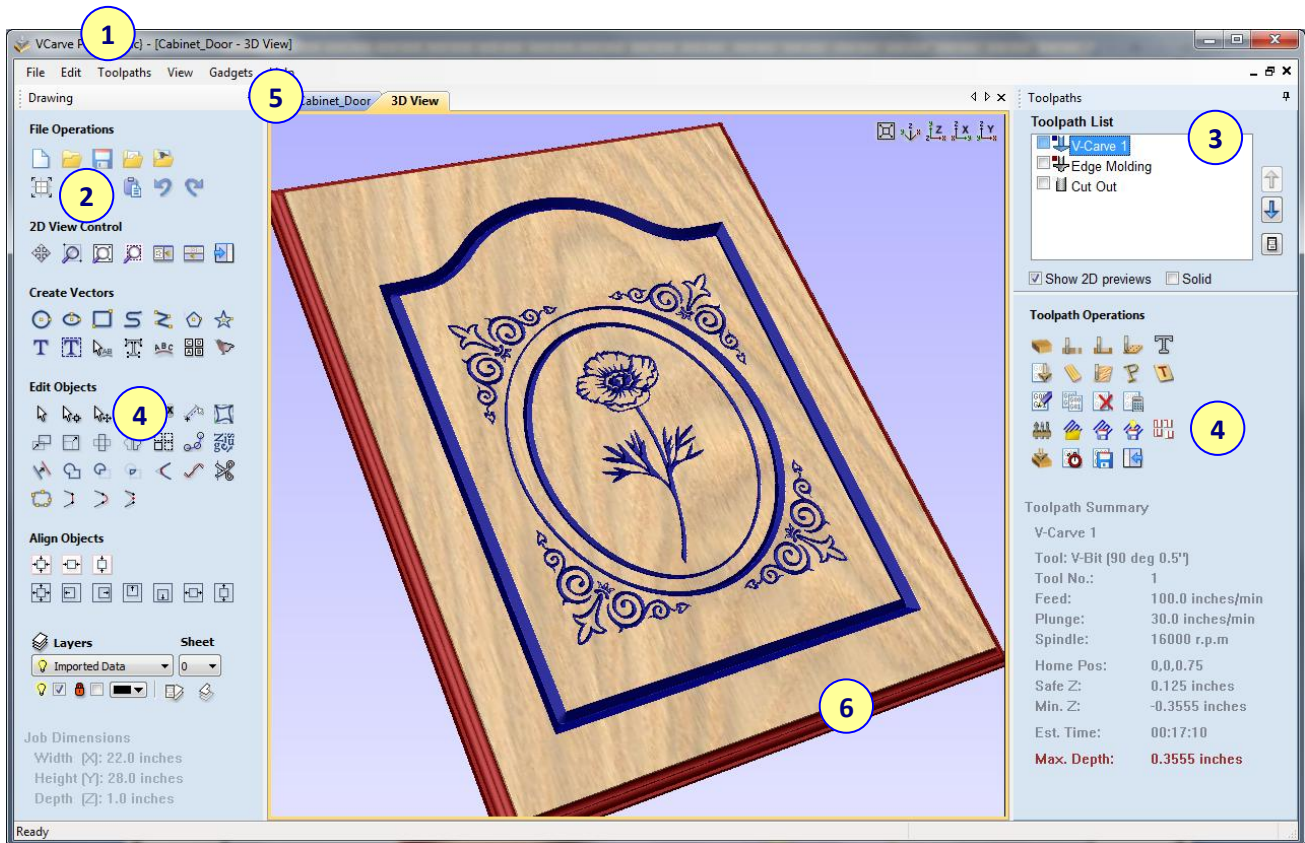
Many of the tutorials and help files have associated video footage that will make learning to use this software more interesting and enjoyable. These are available to downloadable from the web site.

Overview of the interface

The screen area is split into 6 main regions.

1. The **Main Menu bar** along the top of the screen provides access to additional, less commonly used commands available in the software. Simply click and each option will show a drop-down list of the functions.
2. The **Drawing Tab** on the left side of the screen provides general drawing tools for design modification, sizing, alignment etc. prior to machining.
3. The **Toolpath Tab** on the right side of the screen is where toolpaths are defined, calculated, edited and deleted. The Material set-up and Job Preview tools are also in this area.
4. The **Command forms** automatically appear in the Drawing window and the Toolpath tabs when tools are selected that require details to be entered such as dimensions for sizing or positioning etc.

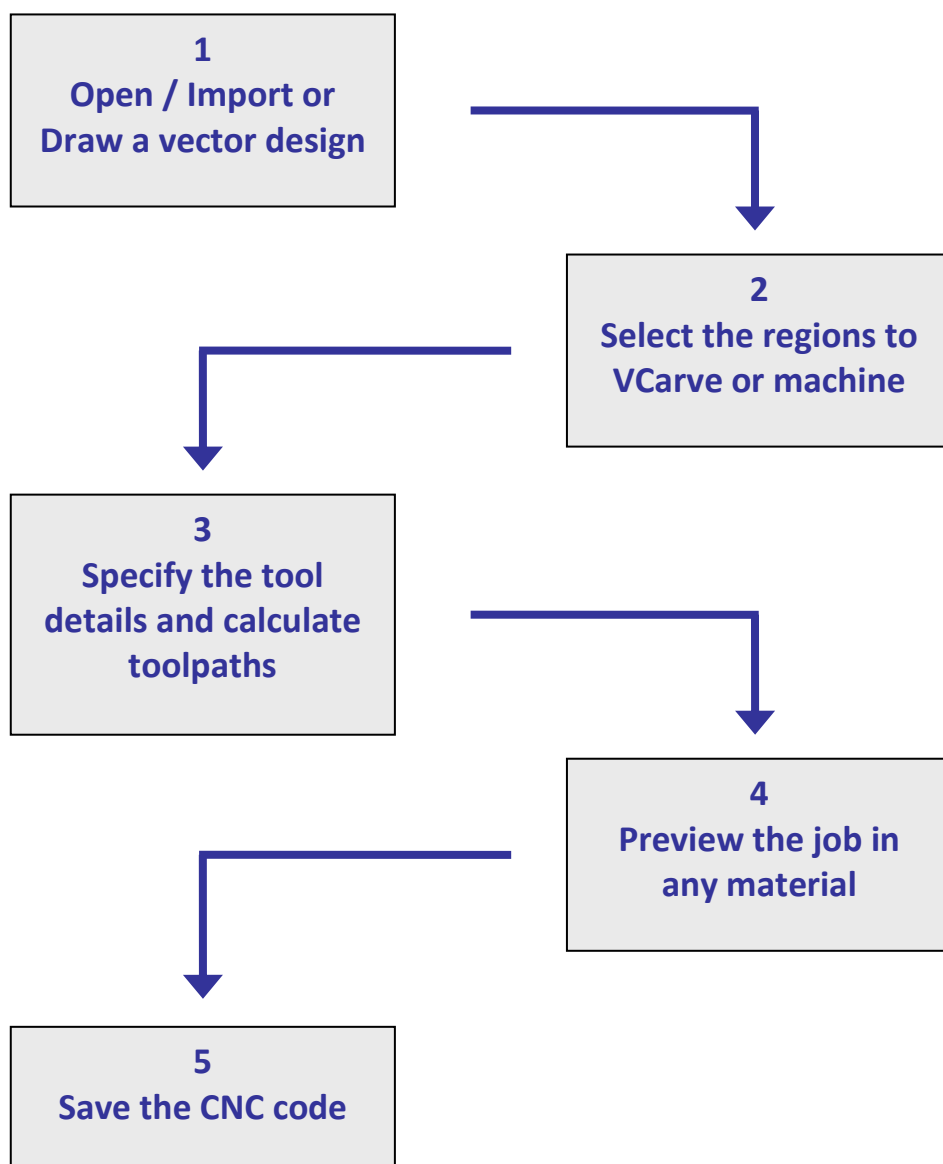
5. The **2D Design window** is where the design is drawn, edited and selected ready for machining. Designs can be imported or created directly in VCarve Pro. This occupies the same area as the 3D Preview window and the display can be toggled between the two using F2 and F3 or the tabs at the top of the window.
6. The **3D Preview window** is where toolpaths and the colour shaded job preview are displayed.



The User Interface

The Machining Logic

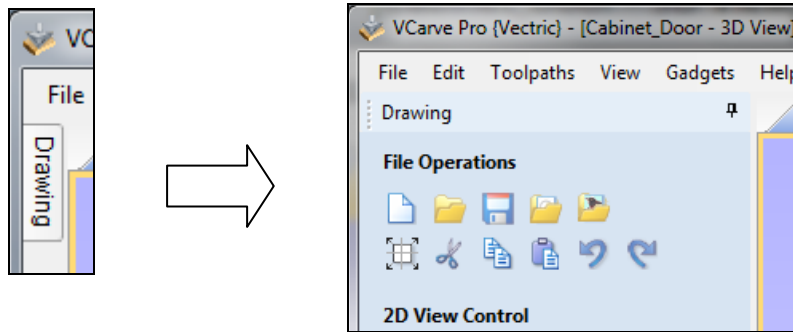
Aspire and VCarve Pro have been developed specifically to open decorative designs and calculate perfect CNC toolpaths for 3D V-Carving / 3D Engrave, profiling, pocketing, drilling, texturing etc. as quickly and easily as possible. The general work flow logic to apply to most jobs is explained in the diagram below.



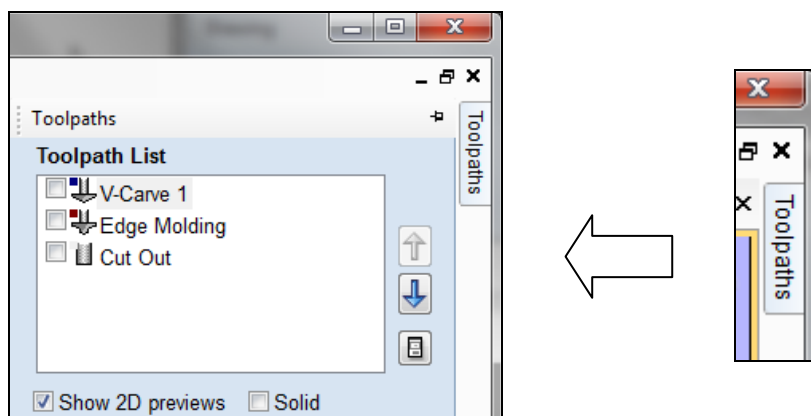
Aspire and VCarve Pro are both excellent toolpath engines that also includes drawing and editing tools that allow designs to be created and modified. Tools for drawing, positioning, alignment and node editing are very easy to use and multiple design elements can also be drawn or imported, scaled, positioned and interactively edited to make a new design. Text can also be created using any TrueType font installed on your computer or the Single stroke engraving fonts supplied with the software.

Managing windows - Auto Show / Hide

The two primary **Drawing** and **Toolpath Tabs** have Auto Hide / Show behaviour which allows them to automatically close when not being used, maximizing the working screen area. They can be opened and closed at any time by clicking the left mouse over the **Drawing** and **Toolpaths tabs** in the **top left** and **top right** corners of the interface.



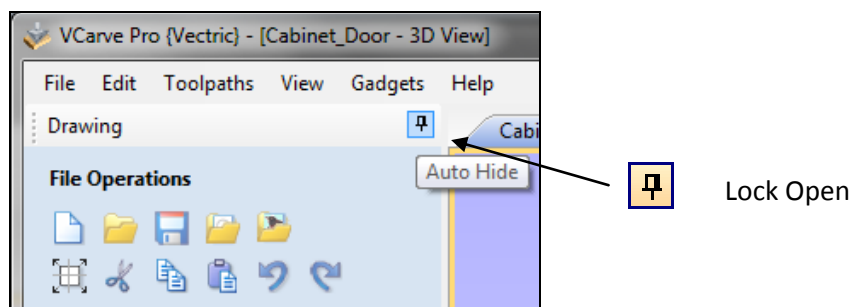
Click on the Drawing Tab to open the Drawing Window



Click on the Toolpaths Tab to open the Toolpaths Window

Note Clicking on the Drawing or Toolpath tab will also close an open window.

Both windows have Auto Hide / Show behavior and can be locked open by clicking on the **Push-Pin button** in the top right corner of the window region as shown below.








Auto Hide / Show windows

View Controls






The View Control options available when working in the 2D Design and 3D Preview windows are,

2D Design Window

	Zoom Interactive	Mouse with Middle Wheel – Push / Pull
	Zoom Box	Click top left corner Click bottom right corner
	Pan	Click and hold the Left mouse button – Esc to cancel Shortcut: Click and drag the Middle mouse button
	Zoom Extents	Zooms to show material limits in the 2D window
	Zoom Selected	Click to select an object or objects Zooms to the bounding box of the selections

Note Mouse with Middle Wheel can be used to interactively zoom in / out.

3D Window

	3D Twiddle	Click and drag Left mouse button in the 3D window
	Zoom	Right mouse button – Push / Pull Mouse with Middle Wheel – Push / Pull
	Pan	Click and drag Right mouse button + Ctrl Click and drag Right and Left mouse button
	Plan View	Looks directly down the Z axis onto the design in 3D window
	Isometric View	Shows the model in a 3D isometric view in the 3D window

Note Pressing **F2 & F3** will toggle between displaying the **2D & 3D windows**

Working with Vectors

Decorative vector designs and shapes will often be imported from another drawing package such as Corel Draw, AutoCAD etc. rather than being completely drawn in VCarve Pro. The imported vector shape(s) can be modified, moved, scaled, rotated, mirrored or deleted.

Vector selection methods

Multiple vectors can be selected in the following **4 ways**.

1. Manual multiple selection

Hold down the **Shift** key while clicking the **Left** mouse button on each vector required.

Objects can be deselected simply clicking on the object again with the **Shift** key pressed.

2. Moving the cursor from **Left to Right selects only FULLY enclosed** objects.

Click and drag the left mouse button moving from **Left to Right** selects all objects fully inside the selection rectangle.

3. Moving the cursor from **Right to Left selects all objects INSIDE** the selection rectangle and also any objects that the selection rectangle **TOUCHES**.

Click and drag the left mouse button moving from Right to Left selects all objects inside the selection rectangle + any that the selection touches.

4. Pressing the keyboard keys **Ctrl + A will select all** vector objects in the design

Note Selected vectors are displayed as dotted purple lines.

Vector deselect

Selections can be cancelled by simply,

Left clicking on an area outside the selection

Pressing the **Right mouse button** and selecting **Unselect All** (top option) from the list. You must click on the white drawing background to get this option in the menu.

Vector editing

A design is created from vector lines, arcs and bezier spans, which all have different properties that can be selected, modified and moved at any time.



Vector Selection Tool

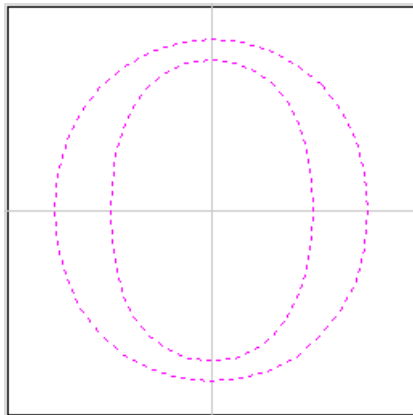
Selected from the Drawing tab on the left

Selected vectors are shown as dotted purple lines. Vectors need to be selected before any of the editing tools such as scaling and moving etc. can be used.

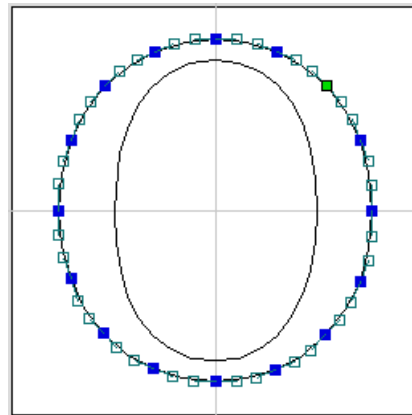


Node Editing Tool

Selected from the Drawing tab on the left



Vectors selected



Bezier node editing

When the Node Editing tool is active the cursor changes to a Black Arrow indicating that individual points (nodes) can be edited. Nodes can be interactively moved by clicking and dragging the left mouse button on a node to select and move the node to a new position.

The shape of lines, arcs and bezier spans can be edited by clicking and dragging on the nodes or control points to move them. Multiple nodes and control points can be selected and moved by using the multiple selection options such as the Shift key and dragging to make a selection.

If you right click on nodes or spans a context sensitive popup menu will be displayed which allows you to insert or delete points and nodes, cut the vector, move the start point etc.

Drawing shapes

Simple shapes and designs can be drawn using the Circle, Oval, Rectangle and Polyline options. These shapes are commonly used to create new borders for signs or as a reference plate for a kitchen cabinet door etc. Shapes can be created by either entering exact dimensions in the Command Window or simply clicking the left mouse button in the 2D window to specify the parameters and coordinates interactively.



We recommend that you watch the **15 minute Video** for this Tutorial before proceeding. The estimated time needed to complete this tutorial is around **20 minutes**.

Introduction

This tutorial will show you how to V-Carve the Kitchen Cabinet Door shown below in Figure 1, which is approximately 20" (510mm) wide by 26" (660mm) high. This design is based on decorative designs from the commercially available Vector Art clipart library (see www.vectorart.com)

The decorative detail on the job will be carved using a wide angled V-bit tool, followed by an Ogee Edge molding cutter to profile round and clean-up the outer edge to the required size.

0.5" (12mm) diameter 90 degree cutter for the flower and corner decoration

1/4" (6mm) diameter End Mill to profile around the outside edge



Figure 1. The finished carved cabinet door

There are 5 key stages in preparing toolpaths for this door panel.

1. Open / Import the design and specify the material size
2. Calculate the 3D V-Carved toolpath
3. Calculate the Profile Cut-out toolpath
4. Preview the completed job
5. Save the Toolpaths


The files required for this tutorial are installed on your PC in the folder,

C: Program Files\Aspire or VCarve Pro\Sample Files\Cabinet_Door.eps

Note If you are using the **Trial version** of Aspire or VCarve Pro and want to save the toolpaths to run on your own CNC machine. You will need to load the file **Cabinet_Door.crv** from the list of **Evaluation files** on the left side of the interface.

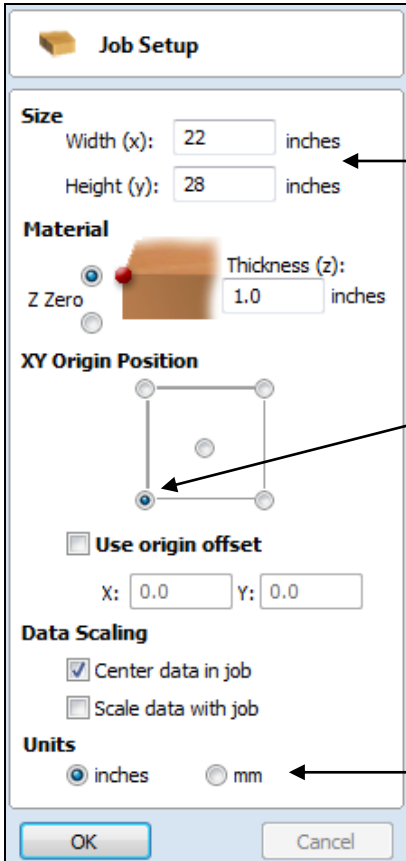
1. Opening the Vector Clipart Design

The vector artwork can be directly opened and the material size specified.

1. From the Startup Tasks tab toolbar click on the Open an existing file  icon, then navigate to the folder,

C: Program Files\Aspire or VCarve Pro\Sample Files

2. Select the file named - Cabinet_Door.eps
3. Complete the Material Setup form with the width, height and thickness required for this sign, as shown below.



The screenshot shows the 'Job Setup' dialog box with the following settings and annotations:

- Size:** Width (x): 22 inches, Height (y): 28 inches. An arrow points to these fields with the label 'Material Dimensions'.
- Material:** Thickness (z): 1.0 inches. A small image of a material block is shown next to this field.
- XY Origin Position:** A diagram shows a square with a dot at the bottom-left corner. An arrow points to this dot with the label 'XY Origin in the left corner'.
- Use origin offset:** A checkbox is unchecked. Below it, X: 0.0 and Y: 0.0 are shown.
- Data Scaling:** 'Center data in job' is checked. 'Scale data with job' is unchecked. An arrow points to the 'Center data in job' checkbox with the label 'Automatically center vectors in the material'.
- Units:** 'inches' is selected. An arrow points to this radio button with the label 'Units either Inches or Metric'.

Material dimensions are 22" wide x 28" high x 1" thick

The Z Zero origin for setting the datum for the tool is on the Material surface.

The XY Origin will be the bottom Left corner of the job.

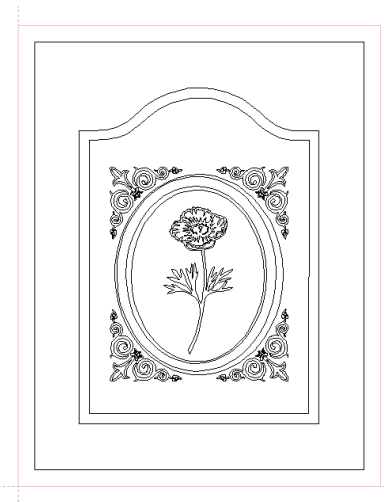
Check the option - Center vectors in material

Make sure the **Units** are set to be in **Inches**

4. Click the **OK button** and the blank job is drawn in pale pink lines in the 2D Window.

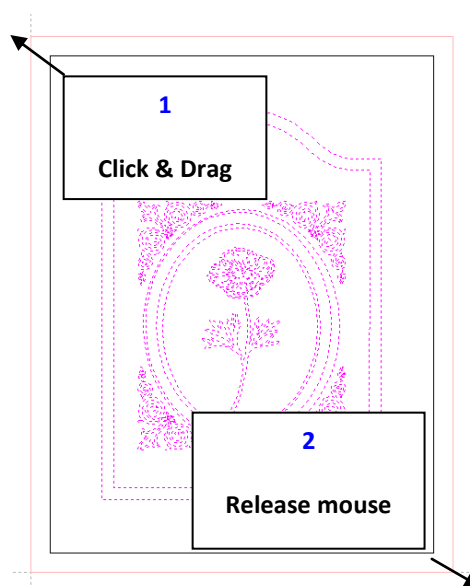
2. Calculating the V-Carve Toolpaths

The design will now be displayed in the 2D Window as shown below.

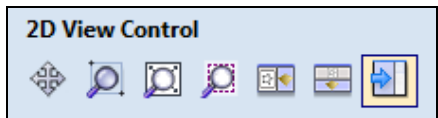


First select the decorative design vectors to be v-carved.

5. Click and Drag the Left mouse button just inside the top left corner of the outer rectangle to the bottom right corner of the outer rectangle, to select the detail on the panel as shown below.



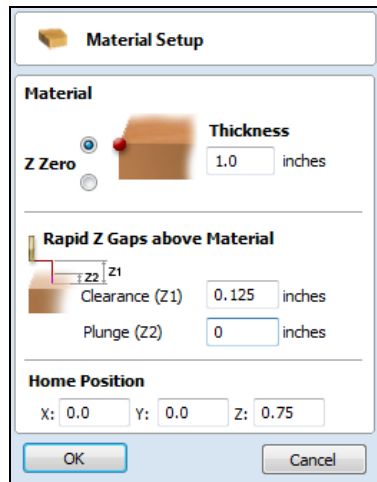
The selected vectors are shown as dotted purple lines



6. Click the Switch to Toolpaths Tab icon. This closes the Drawing Tab and opens the Toolpaths Tab on the right side of the interface



7. Click on the Setup Material icon and specify the Rapid Clearance Gap (the height above the job that the tool can move at maximum feedrate). Enter 0.125" for the Gap and 0.750" for the Home Z height as shown below.



Note the Z Zero is set to the top surface of the material.

8. Click the OK button.



9. Click the Create VCarve Toolpath icon and complete the form as shown below,

Click to **Select** a cutter from the Tool Database

V-Carve / Engraving Toolpath

Cutting Depths

Start Depth (D) 0.0 inches

Flat Depth (F) None inches

V Tool

V-Bit (90 deg 0.5")

Use Flat Area Clearance Tool

Not using area clear tool

Flat Area Clearance ...

Offset Raster

Cut Direction

Climb Conventional

Raster Angle 0.0 degrees

Ramp Plunge Moves

Distance 1.0 inches

Use Vector Selection Order

Safe Z 0.125 inches

Home Position X:0.00 Y:0.00 Z:0.75

Vector Selection: Manual

Name: V-Carve 1

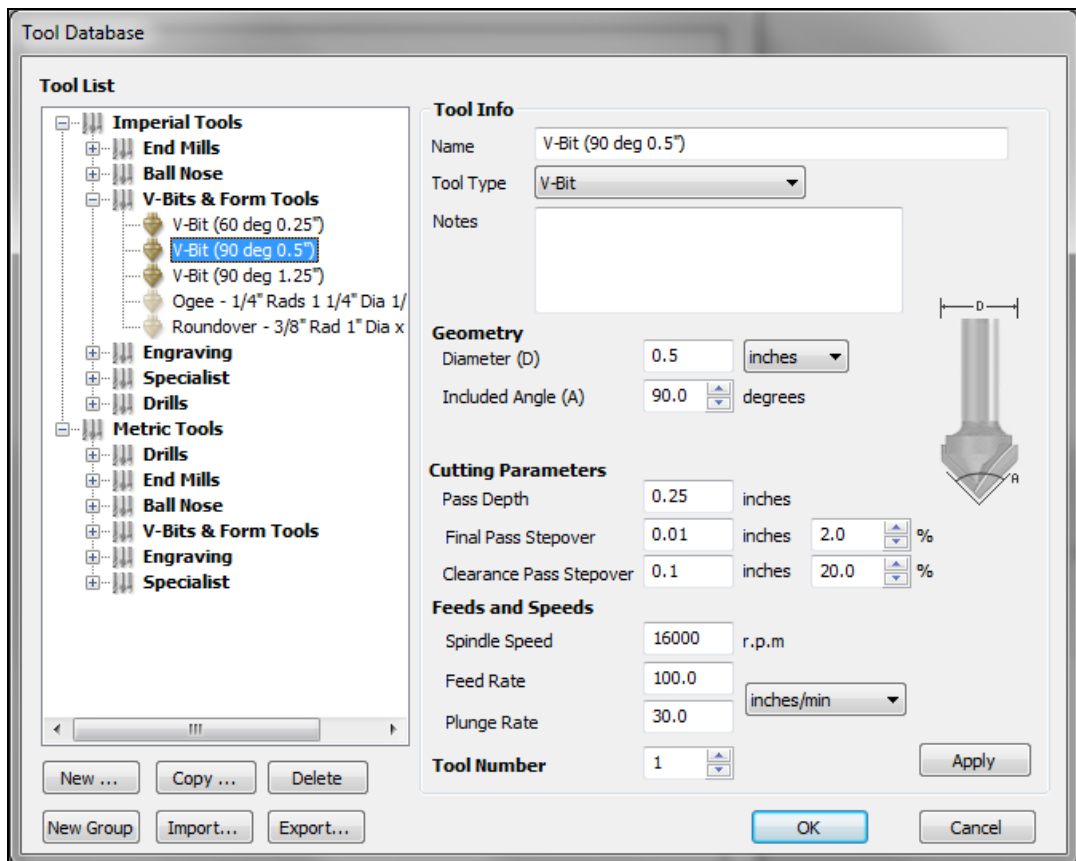
Calculate Close

A new filename can be entered manually

10. Click the **Select button** and choose the V-Bit tool from the Tool Database.


In this example we are using a 0.5" Diameter – 90 Degree included angle V-Bit tool.

Note If you are using thin material click Flat Depth (F) and enter the maximum depth you wish the carving to be routed to. For example, you could limit the depth to = 0.200".



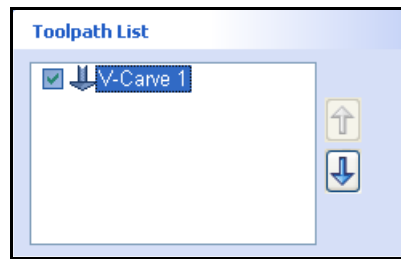
The feed rates should be set to suitable values for your machine and the material being cut.

- Click the Calculate button and the V-Carving Toolpath will be calculated. The 3D Window and the Preview form are automatically opened.

- Click the ISO View  icon and the job + toolpath will be shown as below.



13. The name of the calculated toolpath appears in the Toolpath List in the top right corner of the interface as shown below,



Simply Double click on the name to edit the toolpath or click the Edit Toolpath icon.

To limit the toolpath depth to 0.200" check the option – Flat Depth (F) on form.

The calculated toolpath will use an area clearance strategy to create a flat pocket region around the rectangular border as shown below. This will take longer to machine because the tool needs to cut multiple passes.



Not limited to a depth




Limited to 0.200" deep

Note Move the cursor over the 3D preview model and the X, Y and Z coordinates are displayed in the bottom right corner of the screen. This can be very useful to see exactly how deep the cutter will machine for different regions of a design.

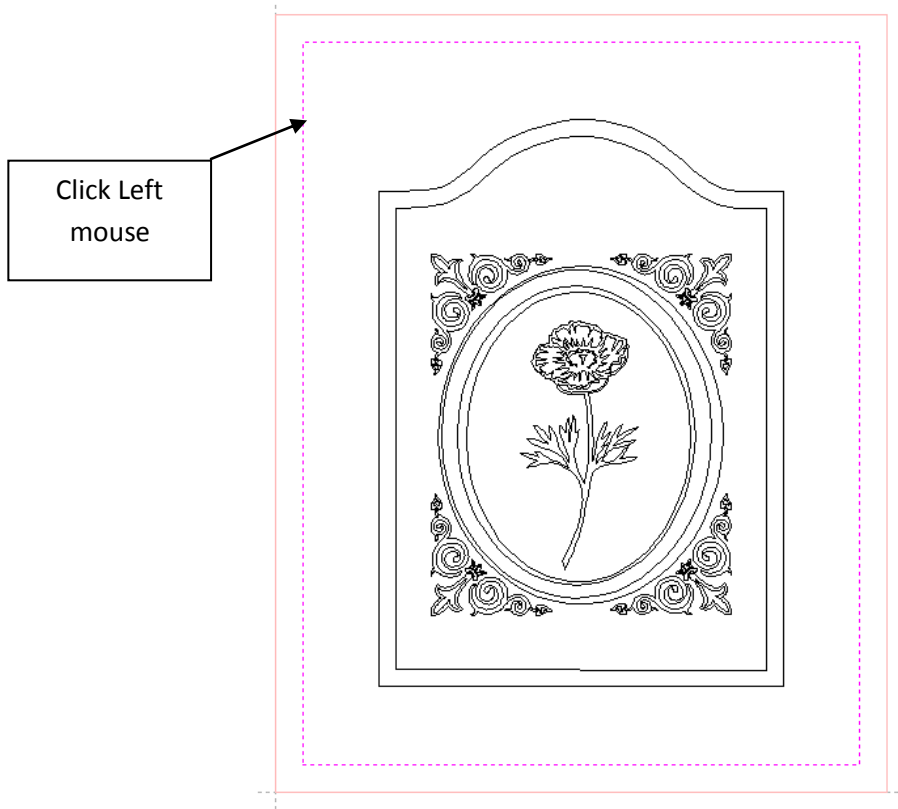
3. Calculating the Edge Profile Toolpath

An Ogee or Form cutter can be used with a Profile Toolpath to add a decorative molded edge to the panel, producing a clean machined edge.

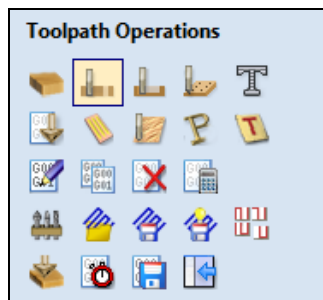
Note If you do not have an Ogee Form shape cutter go to the next section and simply cut the panel out

14. Open the 2D Window by clicking on the 2D Window Tab  or pressing the F2 key.

15. Click the Left mouse button on the outer rectangle to select it.



16. Click the Create Profile Toolpath  icon and complete the form as shown below.



The cutter will run on the selected line

Specify the Cut Depth

Ramping is used to reduce the load on the cutter

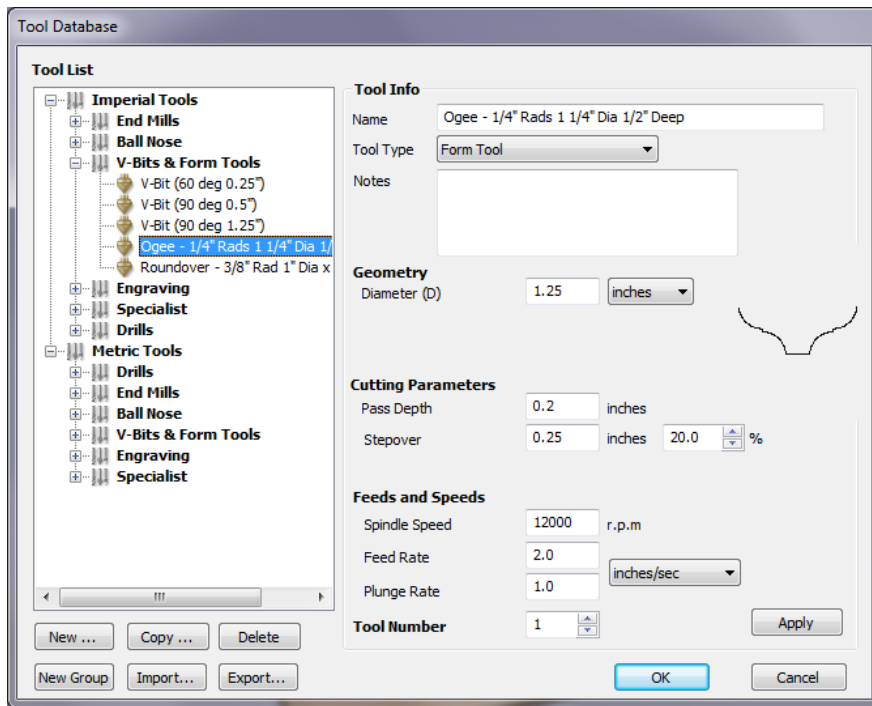
A new filename can be entered manually

This toolpath is optional as it only adds a decorative border to the door panel

In this example we are using a 1 1/4" diameter 1/4" Radius Ogee Cutter and cutting to a depth of 0.6" in 3 passes. The feed rates should be set to suitable values for your machine and the material being cut.

A unique name can be entered or simply left as the default.


17. Click the Select button and pick the Ogee cutter from the Tool Database

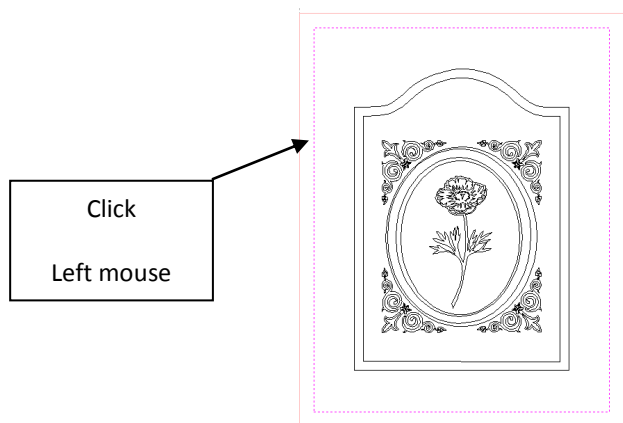


18. The Ramping option is used to reduce the vertical loading on the cutter and ease the large cutter into the wood
19. Click the Calculate button and the Profile Toolpath will be calculated. The 3D Window and the Preview form are automatically opened.

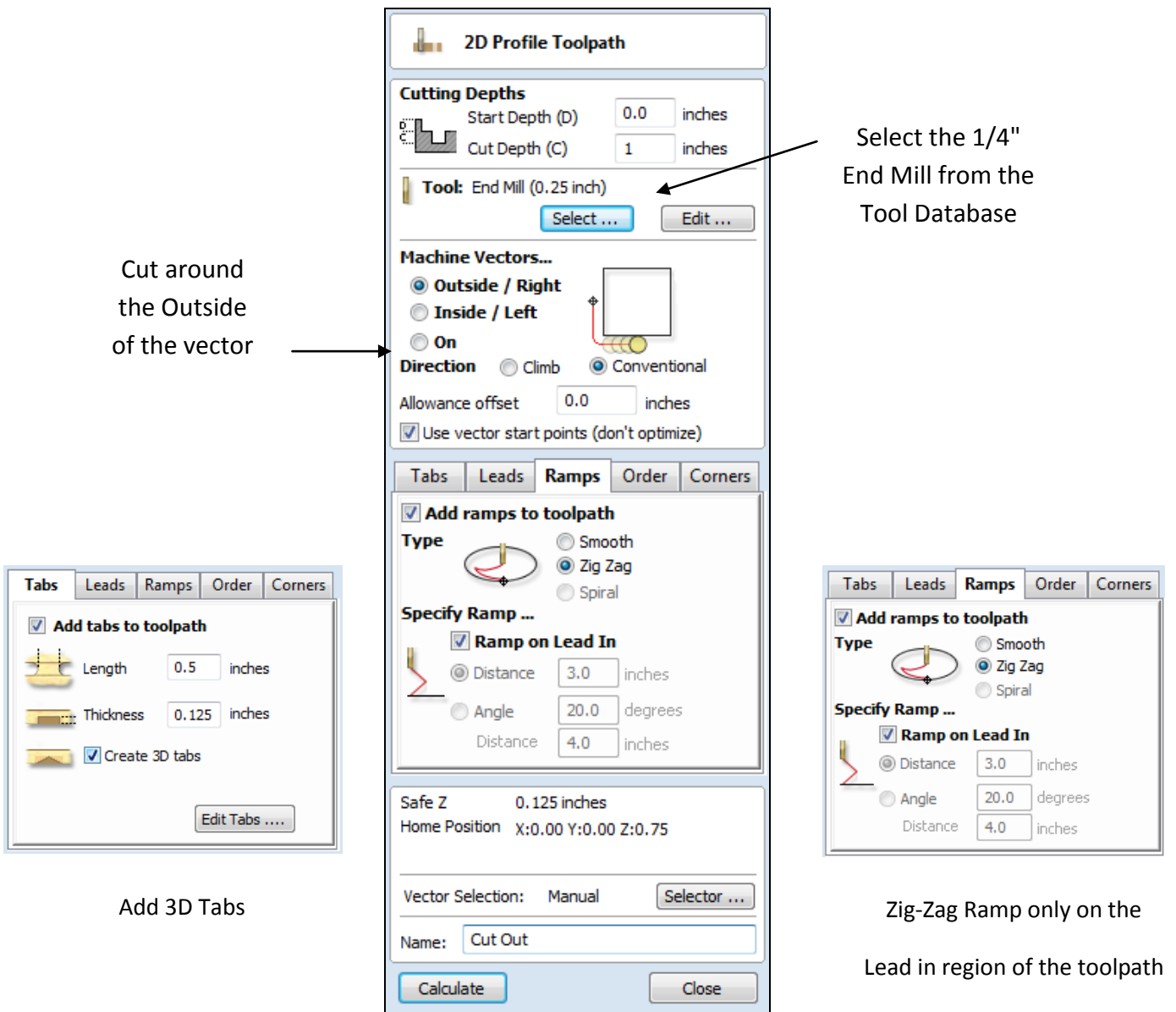
4. Calculating the Profile Cut Out Toolpath

A 1/4" diameter End Mill cutter will be used to cut the panel out of the material using Tabs to hold the job in place and using a lead in and out to ensure a high quality edge on the panel.

5. Open the 2D Window by clicking on the 2D Window Tab  or pressing F2.
6. Click the Left mouse button on the outer rectangle to select it.



7. Click the Create Profile Toolpath  icon and complete the form as shown below



Cut around the Outside of the vector

Select the 1/4" End Mill from the Tool Database

Add 3D Tabs

Zig-Zag Ramp only on the Lead in region of the toolpath

Complete the Profile Form as shown above

In this example we are using a 1/4" diameter End Mill and cutting through the material in 4 passes.

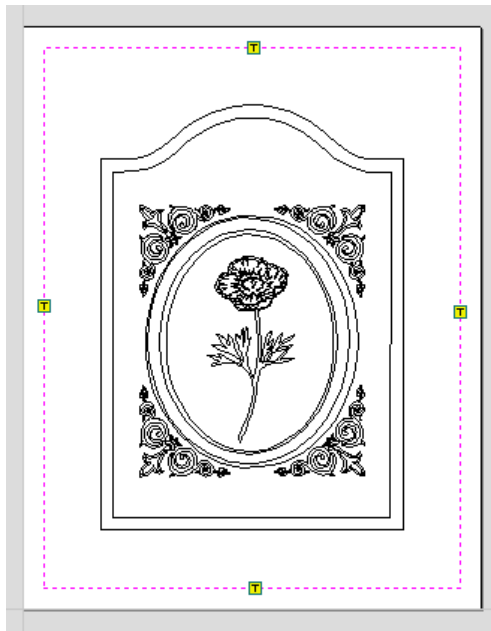
The Tabs will hold the panel in place and the Lead in and Ramp moves will help create a high quality edge on the job with minimum load on the spindle and Z axis.

The feed rates should be set to suitable values for your machine and the material being cut.

A unique name can be entered or simply left as the default.

8. Click the **Edit Tabs** button and place 4 Tabs around the rectangle as shown below.

Click the Left mouse button on the selected rectangle to add a Tab – clicking on a Tab deletes it.



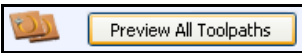
9. Click the **Calculate** button and the Profile Toolpath will be calculated. The 3D Window and the Preview form are automatically opened.

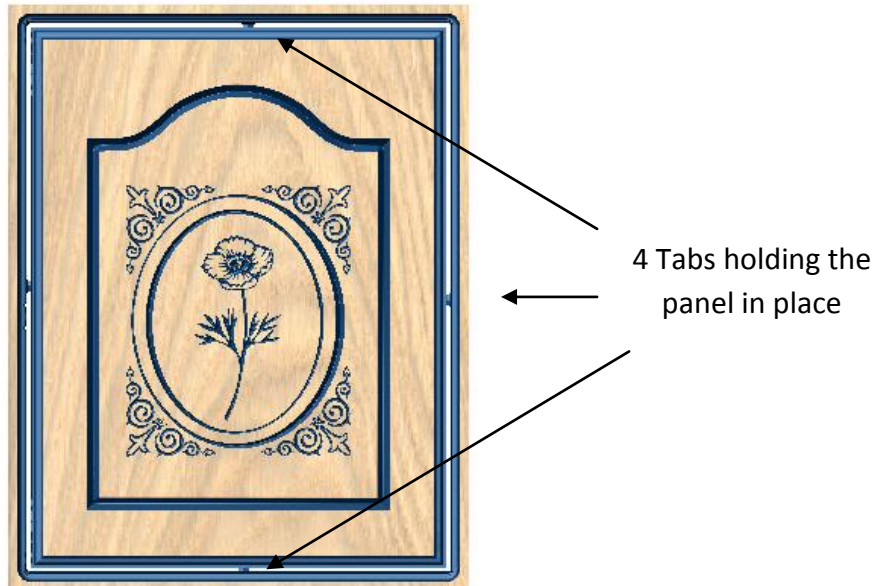



3D Preview showing the calculated toolpaths

5. Previewing the Job

A simulation of the toolpaths cutting into the material can now be shown in the 3D Window

10. Click the Preview All Toolpaths  icon and each tool will be shown cutting into the material.



Notes The Delete Waste Material  Delete Waste Material icon will **NOT remove** the outer material because Tabs are holding the inside and outside shapes together.

For presenting images to customers calculate the Cut Out Toolpath without Tabs in place, as this will allow the Waste Material to be Deleted and create preview images of the finished piece.


11. Experiment with different Material types and Fill colors to display the Preview image in the material that the job will be routed from.



Preview using Global Fill Color



Preview using different colors for each toolpath

12. When the panel looks the way it should, click the Save Preview  Save Preview icon and save the image to a file so it can be printed or e-mailed etc.

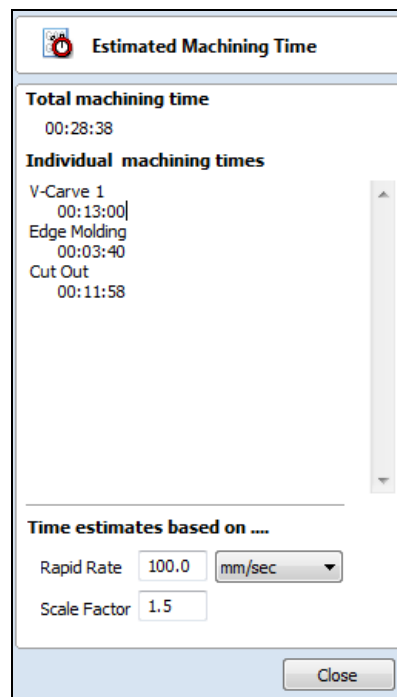
6. Estimate Cutting Times and Save Toolpaths



13. The Toolpaths by clicking the Save icon.
14. Click to select each toolpath in the Toolpath List and Save each file with a new name.
15. If you have a CNC machine that has an Automatic Tool Changer (ATC), each toolpath must e specified using a different Tool Number before the toolpaths can then be saved into a single file.



16. The machining times can be seen by clicking the Estimate Machining Times icon



Important It is very important that the toolpaths for ATC machines have different tool numbers and are saved in the correct sequence. Use the Arrows on the right side of the Toolpath list to order the toolpaths.

Note If you are using the **Trial version of Aspire or VCarve Pro** and want to save the toolpaths to run on your own CNC machine. You will need to load the file **Cabinet_Door.crv** from the list of Evaluation files on the left side of the interface.

Thank you for reviewing Aspire / VCarve Pro and we hope you enjoyed using the software

If you have any questions or need assistance visit the

Vetric User Forum - www.vetric.com/forum or E-mail - support@vetric.com