What's New in VCarve Pro 5.0

Vectric
Passionate About CNC

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

Introduction

This manual is designed to provide an overview of the new features of VCarve Pro 5.0. Each new feature will be dealt with in a separate section and include information on that tool and how to use it. Below is a summary list of the new features of VCarve Pro 5.0.

Design

Nest Vectors

The Nesting tool will automatically fit vector shapes in the most efficient way it can calculate within the 2D Job area. This is a powerful way to optimize material usage and increase toolpath efficiency when cutting a number of vector shapes.

DXF/DWG Import Library

A completely new library has been implemented which adds the ability to import vectors from DWG format files to the other vector file formats available. This also improves the number of DXF and DWG files that can be imported by VCarve Pro. It will now support files written from all versions of AutoCAD up to the 2008 release. This is much more robust than the previous DXF import option.

“Bone” Fillet Option

This option allows 2 new types of filleting for straight line vector corners. It will create a “bone” shaped area in the corner to allow clearance for another piece of material to be slotted into the part without being obstructed by material left by the radius of the tool.

Fit Curves to Vectors

This allows the user to fit arcs, Bezier curves or straight lines to selected vectors. This can improve smoothness for some toolpath options and help to simplify data for modelling purposes.

Toolpaths

Two Tools for Pocketing

There is now an option to select a second tool when creating a Pocketing Toolpath. This will be a larger tool that VCarve Pro will use to clear out the majority of the area to be cut. The smaller tool will then come in and cut areas the larger tool could not fit into and all round the shape to finish it off. On large pockets with detailed outlines this can be more efficient than using a single small tool to cut the whole shape.

Fluting Toolpath

This is a new type of toolpath which is similar to the existing option to profile “on” a selected vector. This feature adds the option to define a ramp at the ends of the selected vector to slope the toolpath up and down. This can be used for cutting standard woodworking “flutes”, for solid surface draining board markings or can be used for artistic engraving effects and even projected onto a 3D model.

Inlay Toolpath

The Inlay feature will calculate either profile or pocketing toolpaths with automatic compensation for the tool radius to allow the cut-out parts to fit into the corresponding cavities. There are four types of inlay available; Straight, Stepped, Pocket and Hole. Sign Makers, plastics fabricators and woodworkers who want to inlay one piece of material into another will benefit from this feature.

Prism Carving Toolpath	Prism carving uses an angled tool to create a raised prism shape on the top of the selected vectors. This is typically, though not exclusively used on lettering.
Multi-Colored Preview	When previewing the effects of each toolpath it is now possible to assign different colors to each one. This can be used to simulate the effects of differently painted areas both as a way to verify the toolpaths and also to provide an image than can be emailed or printed for customer approval.

Bug fixes and Minor Enhancements

A number of reported bugs have been fixed along with other small enhancements and changes which are not documented here as they do not create differences to the way the software is used.

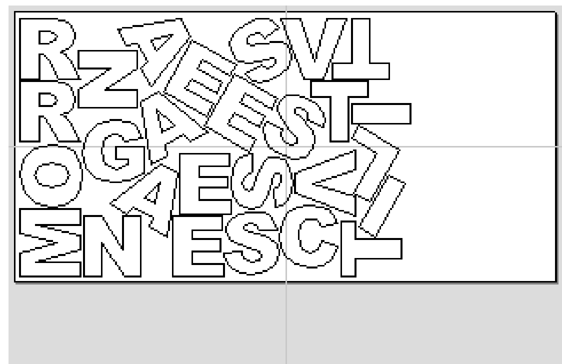
This document is intended to be used for existing VCarve Pro users to learn the new features of VCarve Pro 5.0. If you are a completely new user you should first work through the Reference Manual and tutorial videos that come with the software.

We welcome any comments on our software, this manual or the other training material, please email support@vectric.com with your feedback.

Design

Nest Vectors

The Nesting tool will automatically fit vector shapes within the user defined area in the most efficient way it can calculate (based on the user defined parameters). By default the area the vectors will be fitted is the current **Job Size** but it is also possible to select a vector as the nesting area. This is a powerful way to optimize material usage and increase toolpath efficiency when laying out and cutting a number of shapes. The image below left show a set of letters which have been typed out using the normal **Text** layout tool, the image on the right shows the same set of letters after the **Nest Vectors** function has been used to optimize their layout. The **Nest Vectors** tool will be documented in detail in this section to show how the options within the menu control the layout.



IMPORTANT NOTE: What kind of yield should I expect from the Nest Vectors command?

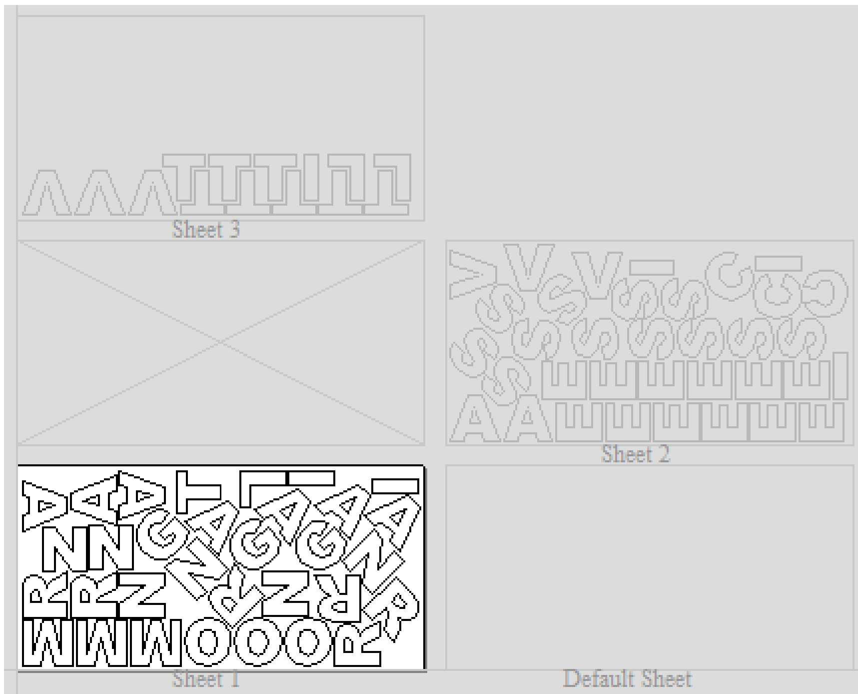
While the **Nest Vectors** function within VCarve Pro is designed to do as good a job as possible it is important to understand it will not always nest parts as well as an intelligent (and patient) human. The nesting in VCarve Pro works incrementally and does not re-arrange parts it has already placed. Therefore it does not have the ability to adjust things as the parts are being fitted that a human nester might see could be more efficient.

The **Nest Vectors** function excels when the parts are relatively small compared to the nesting area and there are a large number of parts to nest.

If you have a relatively small number of shapes to nest or you plan to cut the same set of parts many times then it may be better to take the time to manually nest your vectors. When you use the **Nest Vectors** function and see some obvious places that you could do better this is a good indicator that manual placement may be better.

IMPORTANT NOTE: Sheets

The amount of material required to **Nest** the selected vectors may be larger than the specified work area (**Job Setup**). To cope with any "overflow" VCarve Pro makes use of a new type of entity called **Sheets. Sheets** are used for any nested shapes that will not fit within the boundaries of the specified **Job Size** (or the selected vector). Additional **Sheets** will be created using the same parameters chosen to **Nest** the vectors. These are displayed to the right and above the current Job area as can be seen in the image shown below. The use of the Sheets will be explained in more detail in section below on choosing the **Active Sheet** within the N**est Vectors** menu.



The concept of **Sheets** is very specifically related to machining the finished parts and is not designed to replace layers. **Sheets** should **NOT** be used to organize vectors for modelling or to organize vectors which you intend to use for different machining operations, in those cases the **Layers** should be used to manage the vectors.

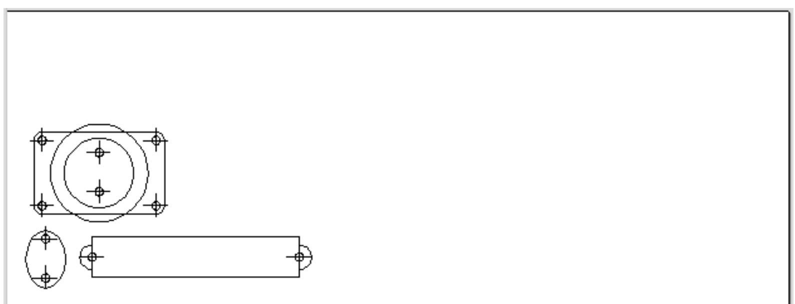
The sole purpose of **Sheets** is to allow nested parts for production type machining to be laid across many units of the same material. Only vectors on the **Default Sheet** can be nested so **Nest Vectors** should be the last command carried out on the parts before machining. As such objects should not be nested more than once, if you do not like the nesting solution then you should use the **Undo (Ctrl + Z)** command immediately and then make changes to the settings before trying the nesting again.

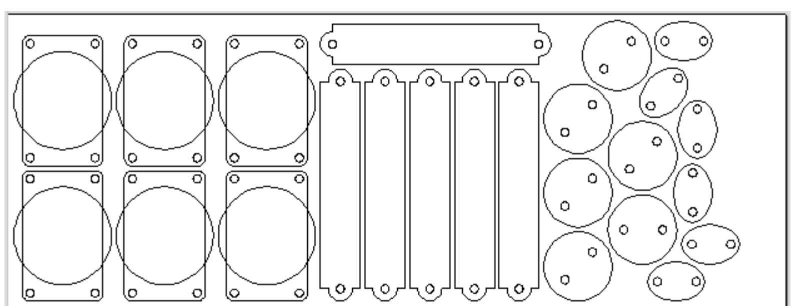
How VCarve Pro Nests Vectors

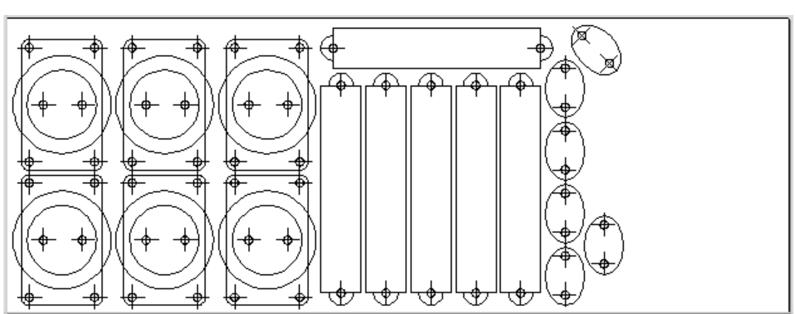
Nesting shapes is a complex calculation which requires the user to make sure the vectors are in the correct state to get the results they are looking for, this is especially important when nesting overlapping vectors or designs that have sets of vectors that need to stay in position with each other. In certain situations it is necessary to group together particular vectors to get the correct result. For simple shapes within shapes such as an "O", "B", "P" etc. there is no need to group them before nesting. VCarve Pro will keep these internal shapes in the correct position and orientation as the shapes are nested.

If the 'outer' vectors of the items being nested are overlapping (and are supposed to be overlapping) then they should be grouped together. This will ensure that the software does not try and nest other items in incorrect places inside of these objects or break the components apart; the nesting for these groups will be done using the bounding box of all these grouped items.

Below you can see an example showing the value of this. The first image shows 3 parts to be nested 6 times each, they include overlapping vectors and single lines. The second image shows the parts nested **WITHOUT** Grouping as you can see the parts are split apart and the single lines are deleted. The 3rd image shows the parts **Grouped** before nesting into 3 specific groups, these keep all the parts in the correct position and does not delete the individual lines.



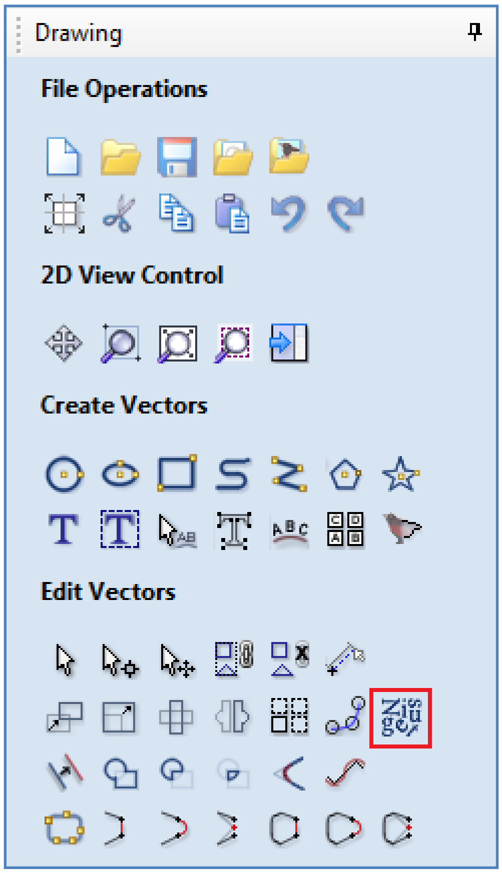




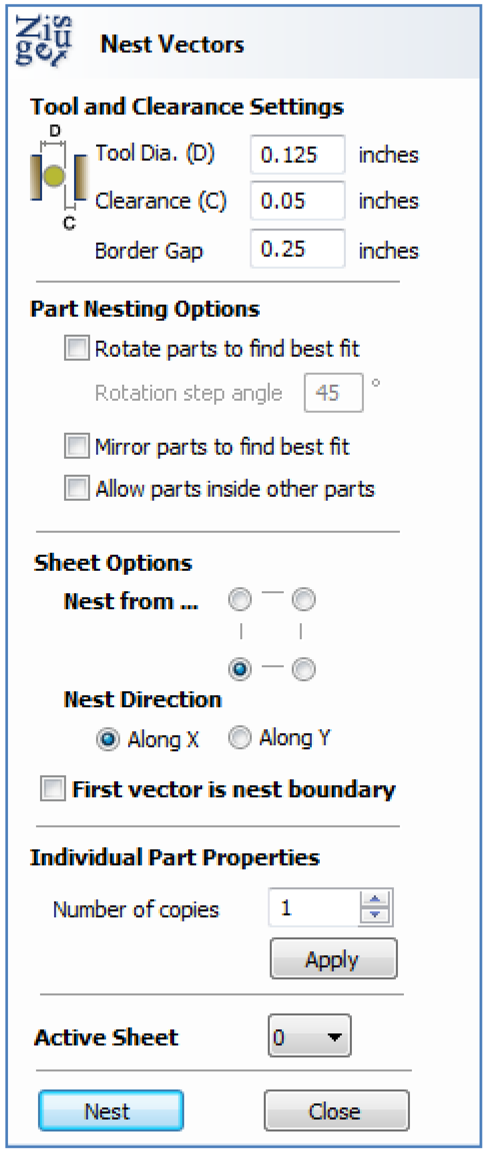
Original vector layers for grouped vectors will be remembered even if the vectors are nested. If nested and then ungrouped the objects within them will go back onto their original layers. This can be useful if you have arrangements of production parts which are on different layers that you need to nest. These might be parts which use layers for different machining operations (drilling, pockets etc.). To nest these you can do the following:

1 Group the complex component represented by vectors on different layers.
2 Nest the grouped objects to optimize placement.
3 Select all the vectors and Ungroup to get the data back on the original layers for machining.

The Nest Vectors icon is located within the **Edit Vectors** area of the **Drawing Tab**, this is shown in the image below highlighted with a red box.



Once you click the icon you will see the menu shown below. You can see there are a lot of options on this particular form, these will all have a bearing on how the parts are nested. Generally it is assumed that you are nesting parts for the purposes of machining so a number of the options are set in regard to the tooling you plan to use and parameters associated with cutting the parts out. It is therefore very important to use the correct values that correspond to the machining choices you plan to make and keep a note of these so you use the correct values when you come to actually create the toolpaths.



Tool and Clearance Settings

The settings in this section of the menu will determine the spacing which will be left between each of the nested vectors and also control how close they are to the edge of your nesting area.

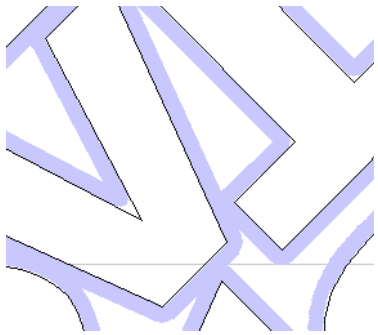
Tool Dia. (D)

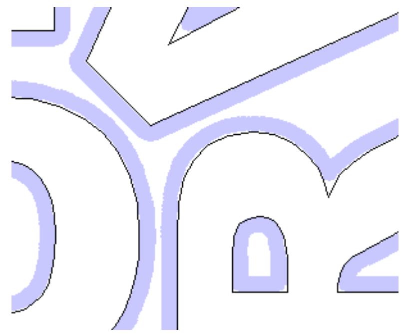
Enter the diameter of the tool that you will be using to **Profile** (cut-out) the vectors you are nesting. This is the minimum distance that will be left between shapes once they are nested.

Clearance (C)

The **Clearance** value will be combined with the specified **Tool Diameter** to create the final minimum spacing between the nested shapes. For example a **Clearance** of 0.05" combined with a **Tool Diameter** of 0.25" would create a minimum spacing gap of 0.3" (0.05 + 0.25 = 0.3).

It is important to note that if you want actual material to be left between the nested shapes once they are cut out that the **Clearance** needs to be larger than the diameter of the tool. In the example used above where the minimum gap is 0.3" (0.05 + 0.25) the area machined by a 0.25" diameter tool cutting these shapes out would overlap as shown in the image to the left below (the blue shows the area which would be removed by the tool), this would leave no material between some of the parts. If you wanted there to be material between the cut-out passes then you would need to specify a **Clearance** value larger than the **Tool Diameter**. For instance a **Clearance** of 0.3" would make a total gap of 0.55", this is shown in the image below right. This would leave a minimum of 0.05 inches of material that would be left once the tool had cut out the shapes. This would be very important if using tabs to hold your parts in place, or if you wanted to prevent the scrap from potentially jumping off the table.

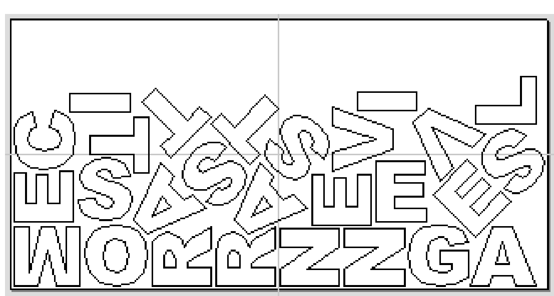


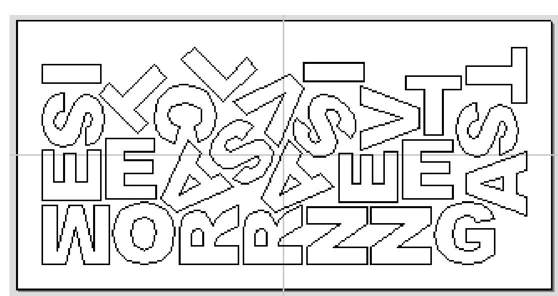


Border Gap

The **Border Gap** value is applied to the edge of the area which is being used to nest the vectors into. It will be added to the **Clearance** value around the edge of this shape to create the minimum distance that parts will be nested in respect to the nesting boundary.

Vectors can be nested either within the whole Job area or into another selected vector (see section below on **First vector is nest boundary** for more information on that option). The shapes will be nested as close to the edge as possible using the **Clearance** specified to determine the minimum distance from the edge. In many situations it is beneficial to have an extra gap from the edge of the material to ensure that the tool does not overlap into an area where there may be clamps (or other obstacles) and to ensure that some material is left for hold-down. The **Border Gap** can be used to define this extra distance. In many cases it would be defined as the tool diameter or an even larger value. Below left the image shows no **Border Gap**, below right you can see a **Border Gap** has been defined leaving a boundary area around the edge of the nesting area.



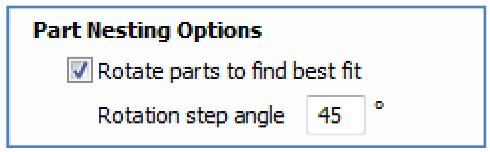


Part Nesting Options

The options in this area of the menu will all directly affect how many parts or how efficiently it is possible for the software to fit shapes into the defined nesting area. The use of these options may depend on the particular material and application you are going to be using your cut parts for. Think carefully about the effect they will have on your shapes to ensure it does not adversely affect the finished cut parts.

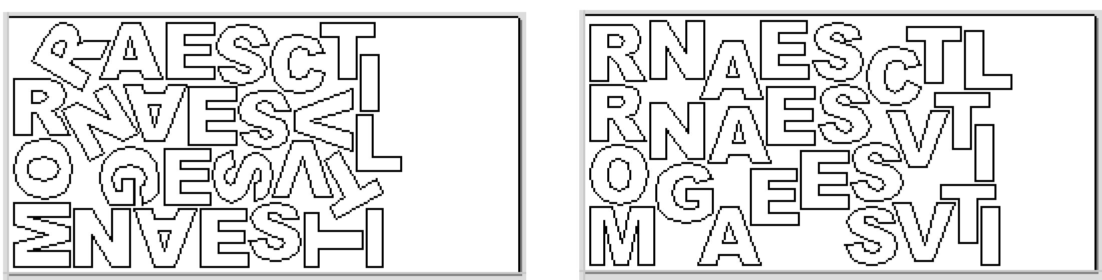
Rotate Parts to find best fit

Checking this option will allow the software to rotate the selected vectors in order to try and better fit them. The increments of rotation the software will use is based on the **Rotation step angle** which is defined in the menu box shown below.



In theory the smaller the specified angle the more options the software will have to fit the shapes together so the more efficient the nesting will be. This does depend on the shape of the vectors though. It should also be noted that the smaller the angle specified the longer the nesting will take to calculate.

Un-checking this option will ensure the parts keep the same orientation that they had when selected. This could be important if you are working with shapes that need to be oriented in a specific direction, for instance in regard to the material grain.



The image above and left shows the letters nested with 30 degree rotation and the image above right shows the same letters nested with NO rotation. As you can see by allowing the rotation the software can fit the letters into a smaller area. In this case it is not a huge difference but the more shapes there are and depending on the style of the shapes it could be a bigger margin.

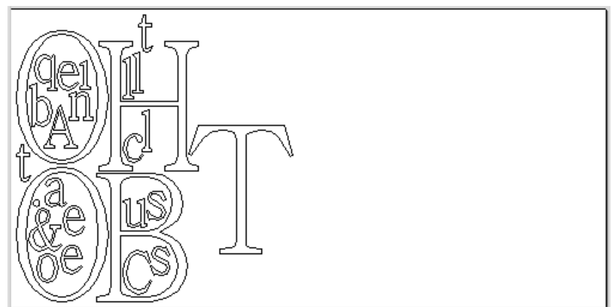
Mirror parts to find best fit

Checking this option will allow the nesting to mirror (flip) the vectors in order to try and more efficiently nest the selected shapes. This should only be checked if the direction the parts are cut in is not important. For instance if you are using pre-finished material you would always want the face of the part to the top of the material and so would not want to allow the software to mirror them. If the parts were being cut-out and then finished or depending on the material being cut this may not be an issue and so could be used to help fit more parts into the nesting area.

Allow parts inside other parts

Checking this option will allow the software to nest within the internal areas of shapes that have gaps in the middle. This would be a good way to optimize material if you were cutting out parts but would not be a good choice if the inside of the shapes was only going to be pocketed as it would not then be scrap material. **Grouped** objects will not allow shapes to be nested within them even if they appear to have space to fit the smaller items. They would need to be ungrouped to allow the software to use the internal regions. Standard typed text will allow nesting on inside areas as shown in the example below.

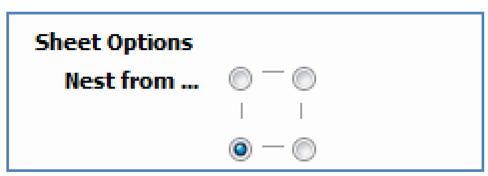
In the two images below you can see a set of letters which are going to be cut out. The larger letters have sizable internal areas that will become wasted scrap, using the **Allow parts inside other parts** option means that VCarve Pro will use these internal areas to fit any smaller parts into them. This can be seen in the right hand image which shows the letters after nesting, where the inside of the O's and the B have been used to nest the smaller shapes.



Sheet Options

Nest From

This area of the menu is used to define which corner the nesting will start in. There are four options which can be selected from the options in the menu shown in the image below:



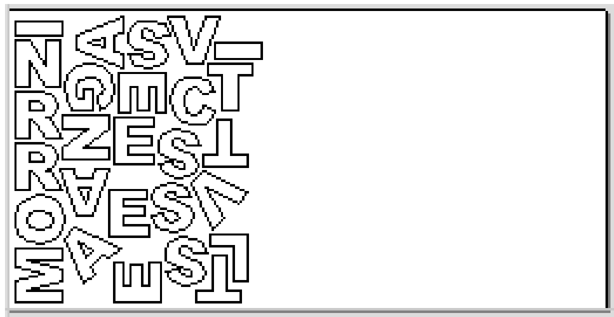
Each node corresponds to the respective corner of the Material or the selected boundary vector. The first nested parts will be placed in that corner and the shapes fitted according to the **Nest Direction** specified (see section below).

Nest Direction

The options in this area of the menu are used to select how the parts will progress as they are positioned within the sheet. The best way to think of this (for the purposes of this section) is that they "pour" out of the selected corner filling the sheet in one axis then advancing along the other defined axis (X or Y)

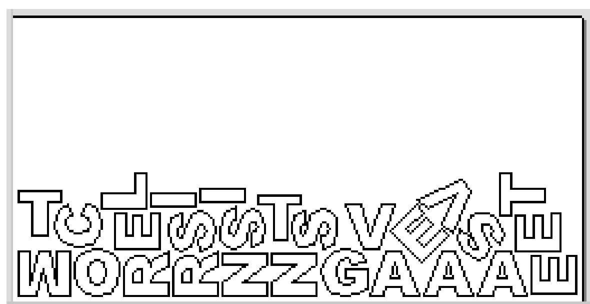
Along X

Checking this option means the nested vectors will fill the boundary area vertically then progress horizontally along the X axis, radiating from the corner selected in the **Nest From** section of the menu. The image below shows the **Along X** nesting option with **Nest from...** set to the lower left corner.



Along Y

Checking this option means the nested vectors will fill the boundary area horizontally then progress vertically along the Y axis, radiating from the corner selected in the **Nest From** section of the menu. The image below shows the **Along Y** nesting option with **Nest from...** set to the lower left corner.



First vector is nest boundary

Checking this option means the first vector selected as will be used as the boundary for the nesting area. This can be useful if you need to define a non-rectangular shape to nest vectors into, such as large off-cut from a previous job. It's important to note that using this option will not respect the currently defined **Job Area** if the selected boundary vectors goes outside of it. If there are too many vectors to fit into the first selected vector then additional **Sheets** will be created using the same boundary shape for the parts being nested, the boundary vector will be positioned on the **Default Sheet** (zero) along with any items not selected for nesting.

Not having the **First vector is nest boundary** option checked means all the selected vectors are nested into the whole of the defined Job Area (defined in the **Job Setup** area of the menu).

IMPORTANT NOTE: Using a grouped vector as the nest boundary

If you need to represent a sheet with holes or other features which can't be represented with a single selected vector as the new boundary it is possible to also use a Grouped vector for the first selected item then the shapes will be nested within the spaces in this.

Individual Part Properties

This function is very useful if you have a number of the same part to make and need to nest many parts at once. It allows multiple copies of the selected object/s to be nested without making the copies prior to the operation.

If you want more than one incidence of a particular item then select it from the 2D view. In the box where it says **Number of Copies** enter however many copies you want and hit **Apply** the selected vectors will be marked with a green number indicating how many copies of that item will be made when they are nested. Different shapes or groups of shapes can be assigned different numbers of copies. To stop an item being copied multiple times just set the **Number of Copies** back to **1** and hit **Apply**.

Active Sheet

This option lets you choose which **Sheet** of vectors is currently active, either for editing or applying toolpaths onto.

The **Active Sheet** can also be chosen when the **Nest Vectors** menu is not open by using the drop down option from the base of the **Drawing** menu – this is shown in the image below highlighted with a red box.

DXF/DWG Import Library

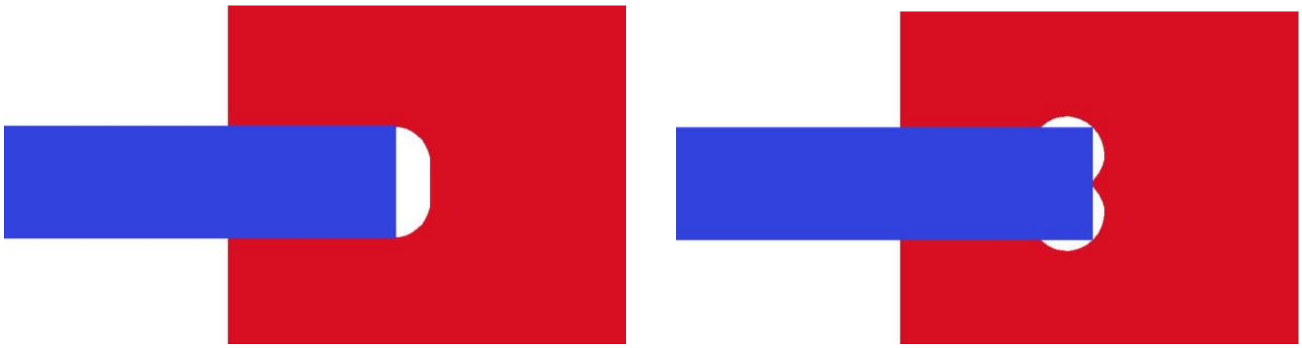
A new library has been integrated into VCarve Pro which increases the number and type of vector files that can be imported into the software. This addition adds a new file format to the list that VCarve Pro can open, allowing the importation of DWG format files. This is a format typically exported from AutoCAD or AutoCAD based software. As well as adding this file type the new library will also import more versions of DXF format files. With the new library DXF/DWG files saved from AutoCAD versions up to and including "2008" will be compatible when being read into VCarve Pro.

The new library is built into the standard Vector Import tool, the icon  is located in the **File Operations** area of the **Drawing Tab**. Once a DXF/DWG file has been imported then the imported vector shape(s) can be modified, moved, scaled, rotated, mirrored or deleted the same as vectors created within the software.

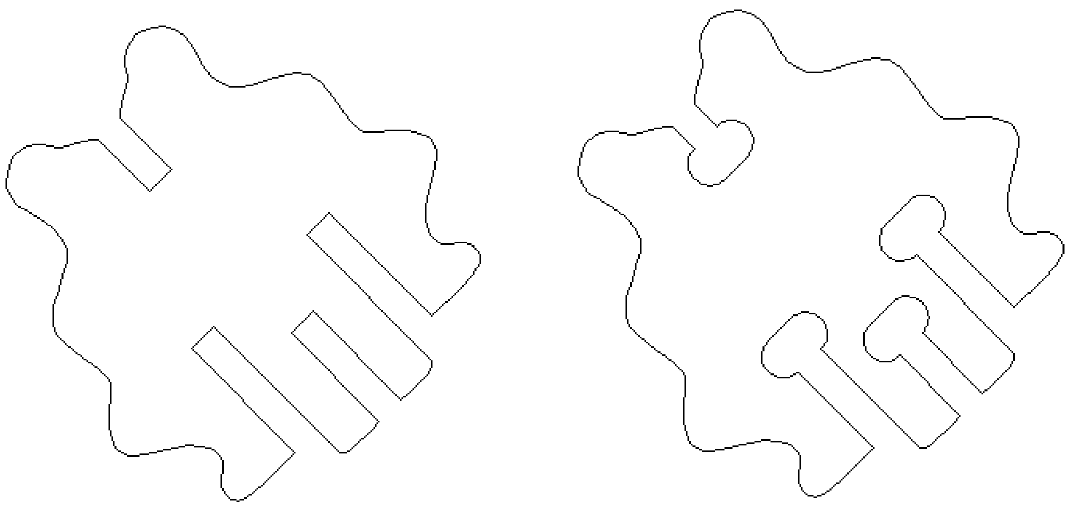
While the new import library will significantly increase the number of DXF/DWG format files that can be imported into VCarve Pro. It should still be noted though that while every endeavour is made to keep up with other software companies' changes in their file format it is possible that files in any of the supported formats may not always be readable in VCarve Pro. In that case going back to the original design systems and saving the file in an earlier version may enable it to be loaded into the software. `

"Bone" Fillet Options

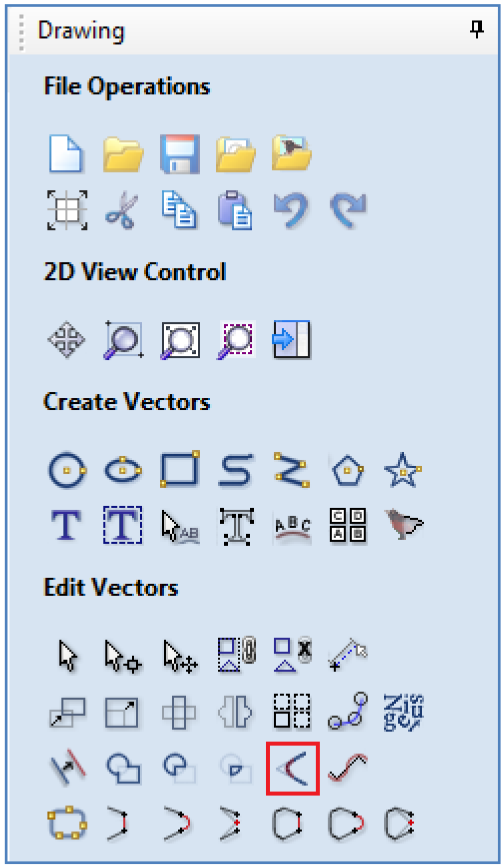
This option allows 2 new types of filleting for straight line vector corners. It will create an overcut shaped area in corners to allow clearance for another piece of material to be slotted into the part without being obstructed by material left by the radius of the tool. Below left you can see a blue piece of material fitted into a normally cut slot, this would fit no further in than the radius of the tool would allow. Below right you can see the same situation with a slot which has the "Dog-Bone" style corners applied to it which allows the part to fit full depth into the slot.



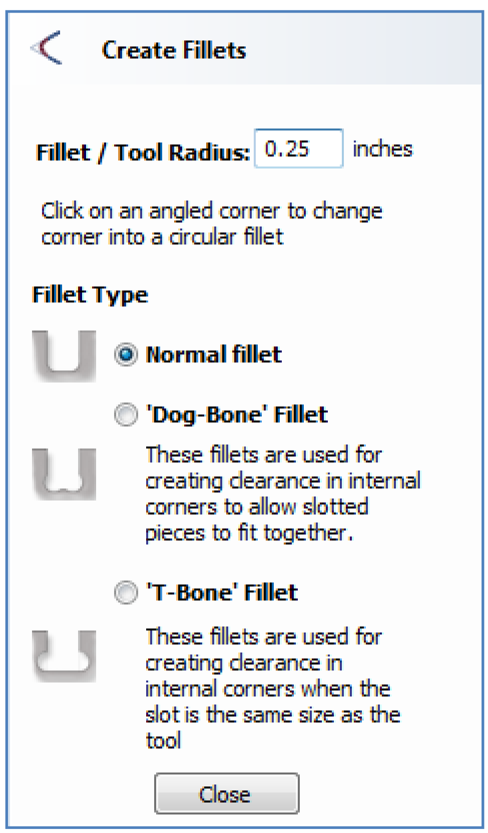
This is a very useful tool for slot-together furniture designs, model aircraft, dinosaurs etc. Below you can see a part from a slot-together dinosaur on the left is the standard part and on the right is the same part with the "T-Bone" style fillets added to these slots. This type of fillet needs to be used when the slot is close in size to the tool diameter.



These options are selected within the standard **Create Fillets** menu, the icon for which is located in the **Edit Vectors** area of the **Drawing Menu** as shown in the image below.



Clicking this will open the Create Fillets menu shown below.



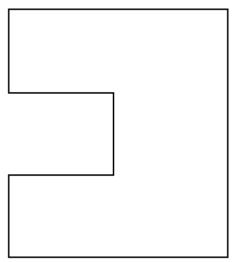
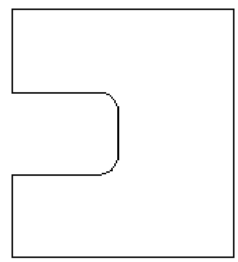
Fillet / Tool Radius

This size of this value is used to create the fillet as described for each individual type of fillet below. To create any of these Fillets you need select the type required then move the mouse cursor so it is over a corner between two straight lines, if it is a legitimate place to create the fillet then a check mark will show and the user can click the mouse button to create it. It should be noted that this will not fillet arcs or beziers even if one of sides of the node is a line.

Fillet Type

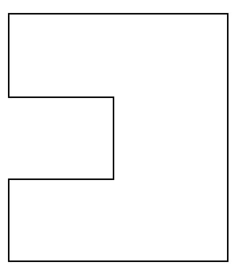
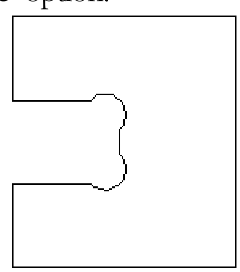
Normal fillet

This creates a standard corner fillet based on the **Radius** defined, typically this would be used for design purposes rather than editing a slot for fitting purposes. The images below are used more as a way of comparing this type with the other two new options. Below left is the vector before filleting the two inside radii, on the right is the filleted version.

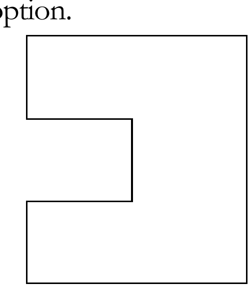
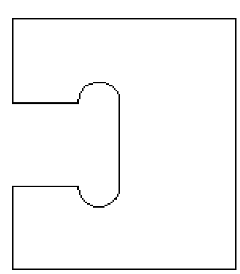
'Dog-Bone' Fillet

This creates a circular cut-out style of fillet, the circles will be placed so the upper right part of the circle touches the original sharp corner and are created with the **Radius** specified. This option should not be used if the slot width and the tool are similar in size. Below left is the vector showing the slot before filleting, on the right is the filleted version using the **'Dog-Bone'** option.

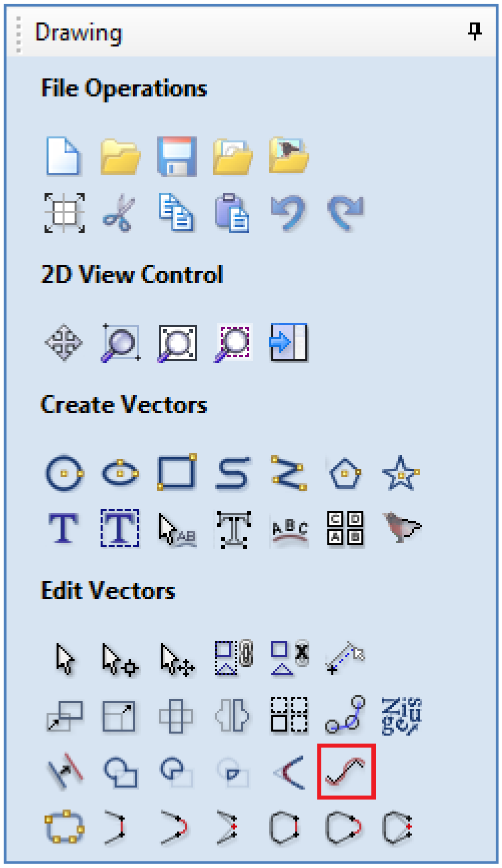
'T-Bone' Fillet

This creates a circular cut-out style of fillet, the circles will be placed so the right hand node of the circle touches the original sharp corner and are created with the **Radius** specified. This option should be used if the slot width and the tool are similar in size so the slot can "grow" out to the side to ensure there is space for them to fit. Below left is the vector showing the slot before filleting, on the right is the filleted version using the **'T-Bone'** option.

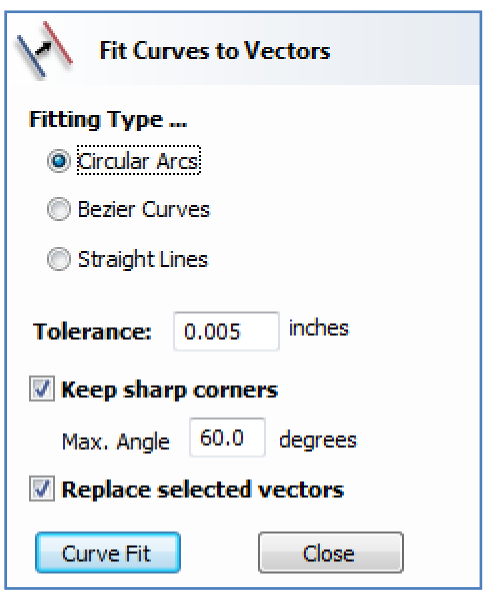



Fit Curves to Vectors

This function allows the user to fit arc, Bezier curves or straight lines to selected vectors. The newly created vectors will be approximated based on a user defined tolerance. Using this function can aid with smoothness for some toolpath options and also help to simplify data for modelling purposes. The **Curve Fitting** function can be accessed from the **Edit Vectors** area of the **Drawing** menu as shown highlighted with a red box in the image below or by using the short-cut-key combination "**Ctrl + F**".



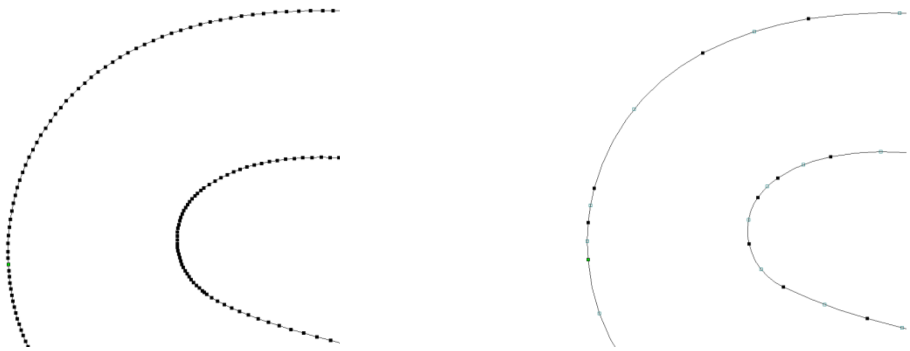
Either of these methods will bring up the following menu:



Fitting Type...

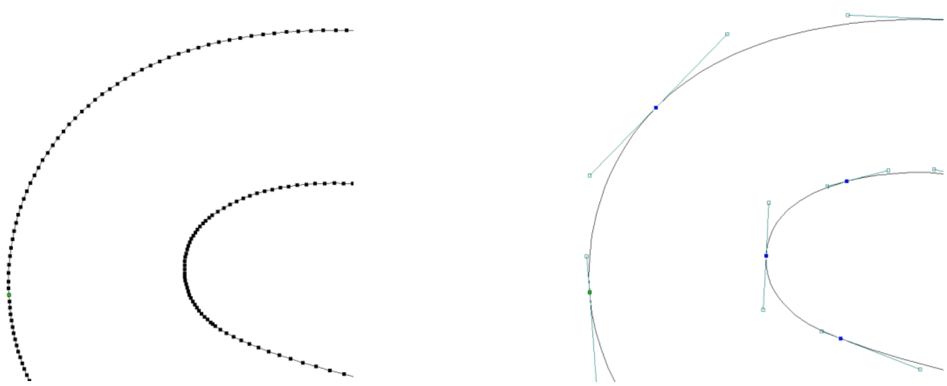
Circular Arcs

Checking this option means the selected vectors will be approximated using arcs. Below left is the vector before fitting, below right the same vector fitted with arcs.



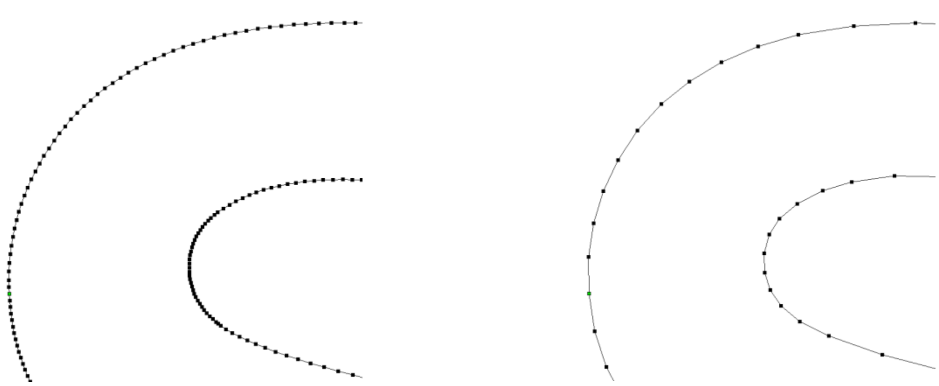
Bezier Curves

Checking this option means the selected vectors will be approximated using Bezier curves. Below left is the vector before fitting, below right the same vector fitted with Bezier spans.



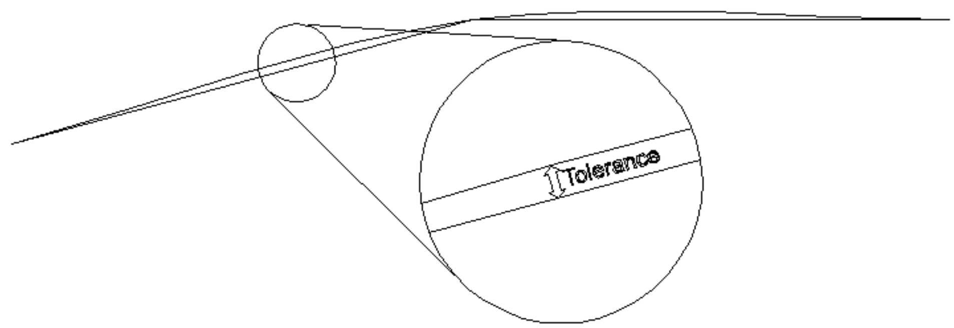
Straight Lines

Checking this option means the selected vectors will be approximated using straight lines. Below left is the vector before fitting, below right the same vector fitted with straight lines.



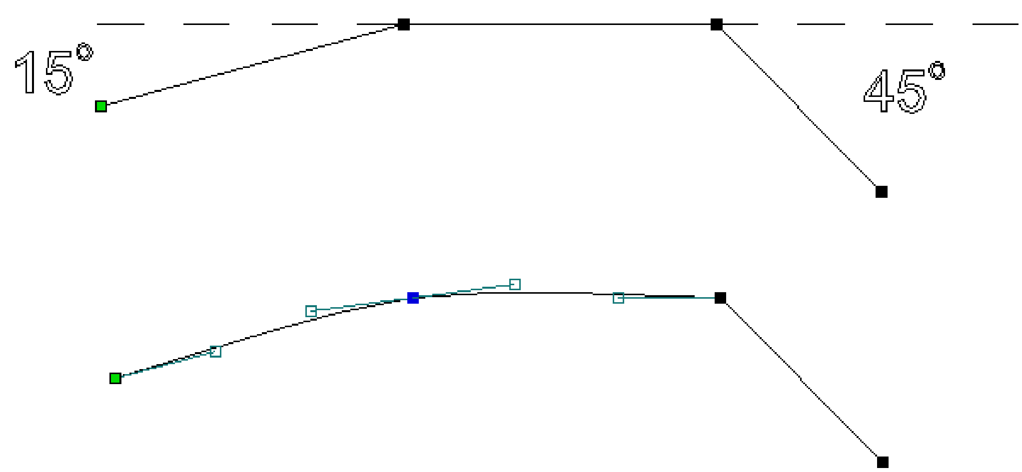
Tolerance:

The value which is set in the **Tolerance** area determines how closely the original vectors will be approximated. The newly created, **Arcs**, **Beziers** or **Lines** will be generated within a distance of the original vector which is plus or minus the specified **Tolerance** value. The smaller the value the closer to the original the new data will be but it will also mean more data points will be used. A larger **Tolerance** will not be as accurate to the original but will have less data points. The diagram below shows the curve fitted to two straight lines with the max possible distance between them defined by the **Tolerance** set.



Keep Sharp Corners

Checking this option will make the **Curve Fitting** routine keep sharp corners which have a difference greater than the **Max Angle** value specified. Any corners where the difference in angle is less than this value will be modified within the specified tolerance. The images shown below demonstrate how this works. The first image shows a set of straight lines before curve fitting with the angles shown between the spans. The second image shows this after curve fitting with the **Keep sharp corners** set with a value of 20 degrees. You can see the two lines which were under 20 degrees difference have had a curve fitted between then but the other corner has been retained as the angle is greater than 20 degrees.



Replace selected vectors

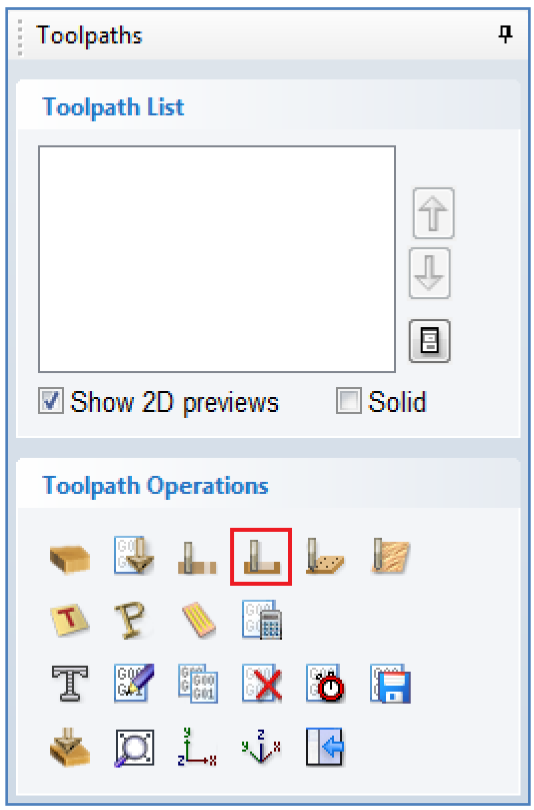
Checking this option will delete the current vectors and replace them with the new "curve fitted" vectors. Un-checking it will keep the original vectors as is and in addition create new "curve fitted" vectors. The new vectors will always be created on the currently selected Layer.

Toolpaths

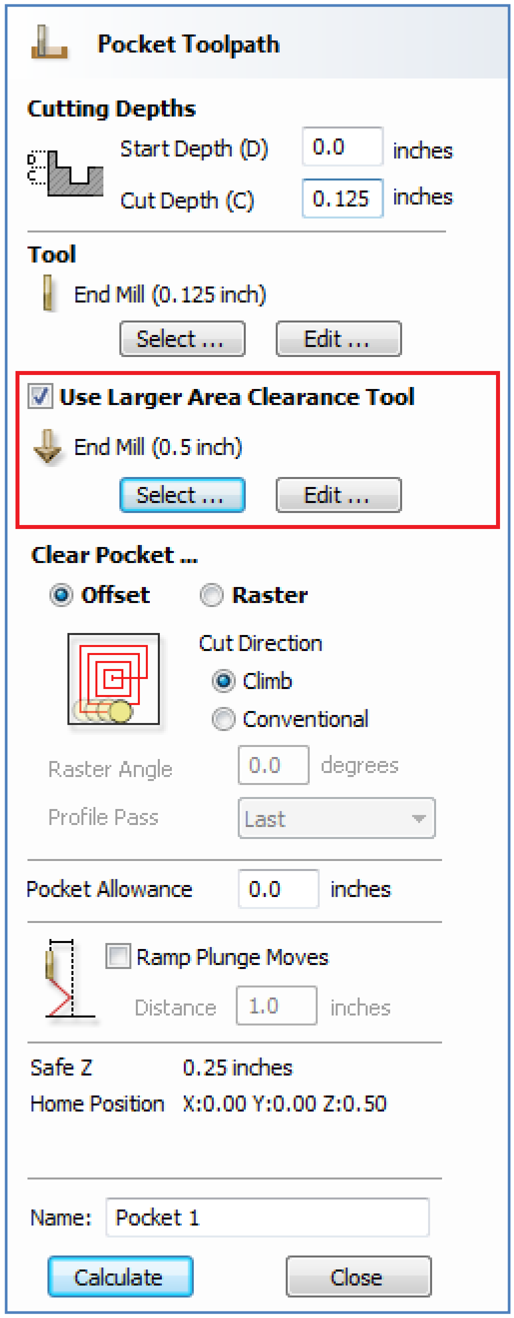
Two Tools for Pocketing

There is now an option to select a second tool when creating a **Pocketing Toolpath** within the software. This second tool will be a larger tool that VCarve Pro will use to clear out the majority of the area to be cut. The smaller tool will then come in and cut any areas the larger tool could not fit into and all round the shape to finish it off. On large pockets with detailed outlines this can save machining time over using the small tool to cut the whole shape giving the best balance of speed and detail.

It should be noted that this will mean you need to do a tool-change so if this is a manual operation on your machine you would need to factor that in when deciding whether to use this option or not. There is not a specific new icon for this feature as it is an enhancement to the standard **Pocketing** toolpath, to access it click on the icon for this tool – shown in the image below highlighted with a red box.



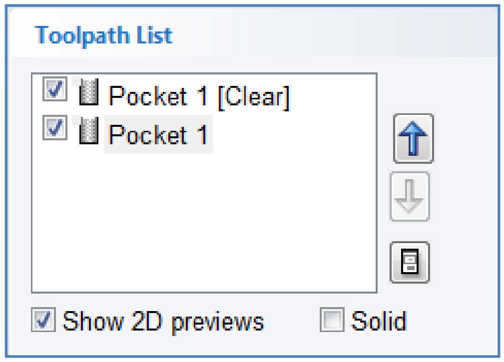
To choose the option to use two tools for the pocketing you will need to check the option which says **Use Larger Area Clearance Tool** this is shown highlighted with a red box in the image shown below**.** If you are not familiar with using the **Pocketing Toolpath** form then please see the appropriate section in the **Reference Manual** or the **Help** file (**Help – Help Contents**).



Use Larger Area Clearance Tool

Check this option to activate the second tool option. Clicking the **Select** button opens the **Tool Database** from which the required tool can be selected. See the section on the **Tool Database** for more information on this. Clicking the **Edit** button opens the **Edit Tool** form which allows the cutting parameters for the selected tool to be modified, without changing the master information in the database. Typically both tools used for this operation would be **End Mills**.

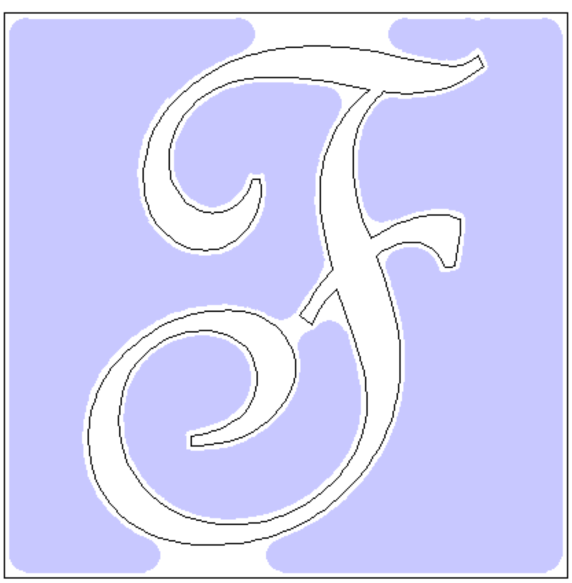
When this option is selected two toolpaths will be calculated and added to the **Toolpath List**. The first toolpath in the list will be the one created with the larger tool; it will have the same name as the other toolpath but with the suffix of [Clear] in the name area to differentiate it. You can see an example of this in the image shown below:

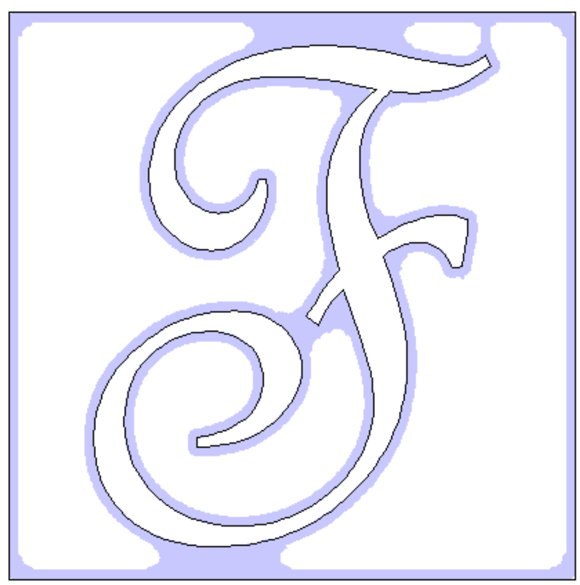


The first toolpath (created using the larger tool) will clear out as much material as it can within the selected vector. It will not go right up to the edge of the vectors as it automatically leaves a small amount of material around the entire shape that will be cleared away with the smaller tool. This is important with regard to getting a clean finished edge on the pocketed area. The second toolpath (created using the smaller tool) will clear away any areas the larger tool could not fit into and also profile around the vectors to create a clean finished edge.

The example in the images below shows how this works, pocketing out between a square and an ornate letter. Here we have shown what is being cut using **Solid** preview option (check box option just below the **Toolpath List**). This shades the area which the tool is going to remove using a blue color. In this example a 0.5" tool has been selected for the larger clearance tool and a 0.125" tool has been selected as the tool to create the final pass.

The image below left shows how much material will be removed by the 0.5" tool. As you can see this cuts the majority of the space but is not able to get into all the small areas of the part. In addition a small boundary is left all the way around the vectors for the smaller tool to finish off. The image below right shows the area that will be cut with the 0.125" (smaller) tool. This shows it cutting only areas which the larger tool was not able to access and also profiling around the complete part for a clean finished edge.



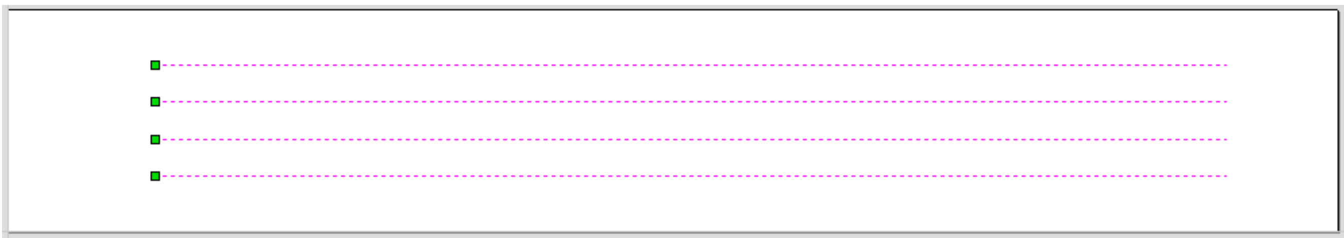


NB: The calculated toolpaths are not "tied" together so if you delete one of them the other will still be in the list, make sure to delete both if you need too. Either toolpath can be double-clicked in the **Toolpath List** in order to edit and re-calculate them both.

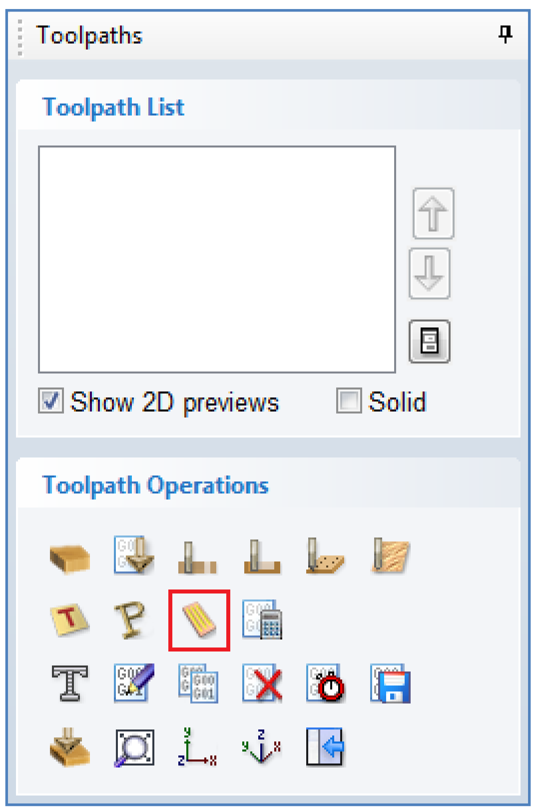
Fluting Toolpath

This is a new type of toolpath which is similar to the option to **Profile** "**On**" a selected vector. The difference is the toolpath at the end of each vector can be ramped to taper the cut. This can be used for cutting standard woodworking "Flutes" or can be used for artistic engraving and marking effects with other types of artwork.

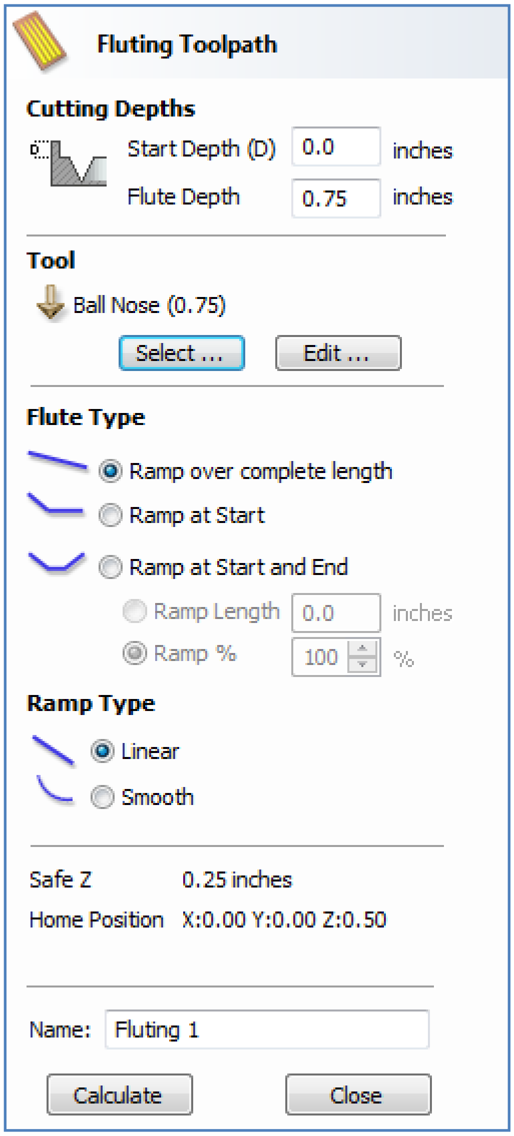
When the **Fluting Toolpath** is chosen then the selected vectors will have their start points indicated in the **2D View** by solid square green nodes, this is important as it will determine which end the ramps are added depending what modifying options are chosen from the menu. An image of this is shown below where all the start points are to the left end of the selected vectors.



The **Fluting Toolpath** icon is located in the **Toolpath Operations** area of the **Toolpath** menu. It is highlighted with a red box shown in the image below.



When you click on this icon the menu shown below will appear.



Cutting Depths

Start Depth (D)

Start Depth (D) specifies the depth at which the **Fluting** toolpath is calculated. When cutting directly into the surface of a job the **Start Depth** will often be 0. If machining into the bottom of an existing pocket or stepped region, the depth of the pocket/step that you are starting from must be entered.

Flute Depth

This is the depth of the **Fluting** toolpath relative to the **Start Depth**; the total depth will be the combination of the **Start** and **Flute Depth**.

Tool

Clicking the **Select** button opens the **Tool Database** from which the required tool can be selected. See the section on the **Tool Database** for more information on this. Clicking the **Edit** button opens the **Edit**

Tool form which allows the cutting parameters for the selected tool to be modified, without changing the master information in the database.

Flute Type

Ramp over complete length

Checking this option means the tool will ramp over the whole length of the toolpath. At the start of the selected vectors/s it will be at the **Start Depth** and at the end of the selected vectors/s it will have cut down to the **Fluting Depth**.

Ramp at Start

Checking this option means the tool will ramp down only at the start of the vectors to the **Fluting Depth**. The distance of this ramp can be specified using the **Ramp Length** or **Ramp %** options.

Ramp at Start and End

Checking this option means the tool will ramp down at the start of the vectors then will ramp up again at the end of the vectors. The distance of these ramps can be specified using the **Ramp Length** or **Ramp %** options.

Ramp Length

Checking this option means that the length of the ramp can be specified by a specific distance entered into the box. The ramp distance is measured from the start and the end of the vector/s depending what **Flute Type** you have selected. If the distance entered is greater than the possible length of the ramp then the maximum length will be used, this would be the same as choosing **Ramp over complete length**.

When you choose **Ramp at Start** it is possible to specify a ramp length which is up to the length of the vector/s. When **Ramp at Start and End** is checked, the maximum length possible would be half way along the vector/s as after that it would start to ramp up again.

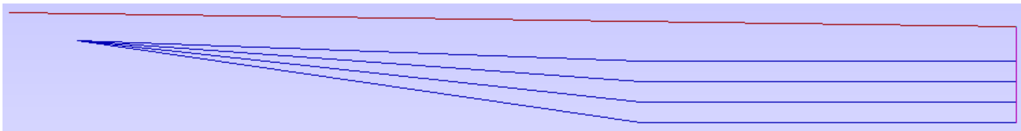
Ramp %

Checking this option means that the length of the ramp can be specified as a percentage of the maximum possible ramp length (controlled by the length of the selected vector/s and chosen **Flute Type)**. When you use this with **Ramp at Start** selected then 100% would be the whole length of the selected vector/s, the ramp length would be a percentage of this distance for each one. When you use this with **Ramp at Start and End** then 100% would be the half length of any of the selected vector/s. The ramp length would be a percentage of this "half" length. In this situation using a 50% value would give you a **Ramp** from the start which was ¼ of the vector length and a ramp from the end which was also ¼ of the vector length.

Ramp Type

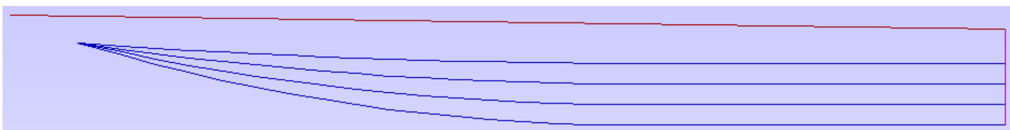
Linear

Selecting the **Linear** type will create a ramp which is a diagonal line (following the vector) from the **Start Depth** to the **Flute Depth**. Below you can see a Linear **Ramp Type** shown from the side. This ramp is set to only ramp from the start and to go 50% of the flute length.



Smooth

Selecting the **Smooth** type will create a curved ramp (following the vector) from the **Start Depth** to the **Flute Depth**; this will smoothly transition from the ramp into the full depth of cut. You can see an example of this shown in the image below.



NOTE: Affects of the Pass Depth on Ramps

Note in the two images shown above how there are multiple passes to get to full depth. These occur when the **Flute Depth** exceeds the **Pass Depth** specified for the selected tool. In these cases the tool will make multiple passes no deeper than the **Pass Depth** of the tool. As you can see from the images the ramps are graduated based on how many passes are needed. This ensures that the final pass will always be cutting material along its full length to give a nice even finish on the part.

Safe Z

The **Safe Z** value defined the distance above the job at which it is safe to move the cutter at rapid/max feedrate. This dimension can be changed by opening the Material Setup form.

Home Position

This is the position that the tool will travel to before and after machining. This dimension can be changed by opening the **Material Setup** form.

Project toolpath onto 3D Model

This option is only available if a 3D model has been defined. If this option is checked then after the toolpath has been calculated, it will be projected (or 'dropped') down in Z onto the surface of the 3D model. The depth of the original toolpath below the surface of the material will be used as the projected depth below the surface of the model. This can be used for some particularly interesting effects with the **Fluting Toolpath** – see below in the Applications section for more information on this.

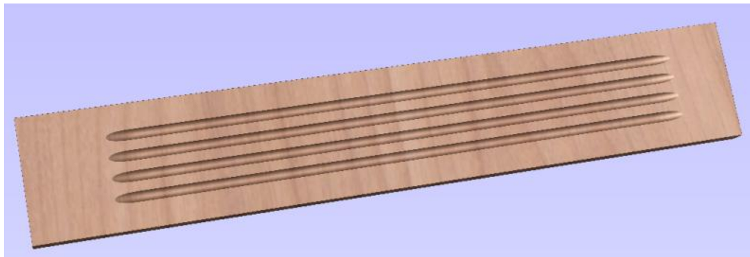
Name

A name for the Fluting toolpath can be entered or the default name can be used.

Applications for Fluting Toolpaths

Standard Straight Flutes

One of the most common applications for Fluting is straight decorative details using a large radius Ball nose tool for columns and posts, such as the one shown in the image below.



Another good application for straight flutes is the markings on a draining board (for Solid Surface fabricators) or on a cutting block (for carving meat) such as the one shown below, sloping down to allow liquid to be directed.

Decorative Artistic Designs

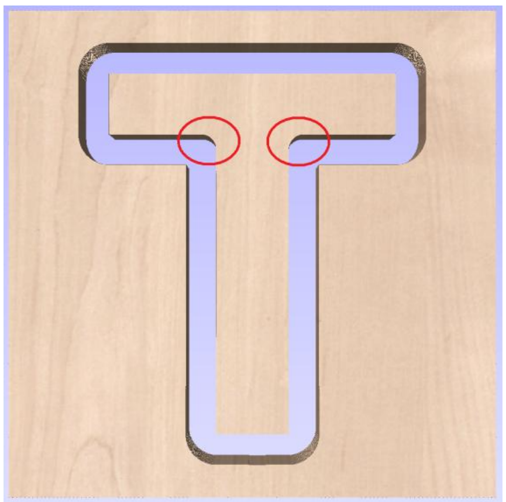
The fluting can be used to produce some interesting effects with artistic and decorative applications. The abstract leaf pattern below left was cut with a V-Bit into a flat surface using single vector lines.

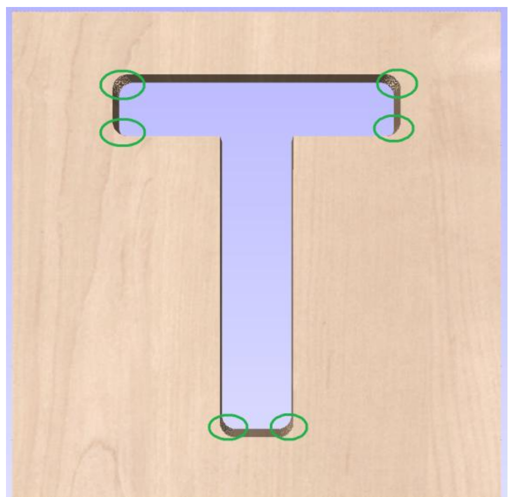
Inlay Toolpath

Inlay Toolpaths can be used to calculate either profile or pocketing toolpaths with automatic compensation for the tool radius, this allows the cut-out parts to fit into the corresponding cavities. This is a good feature for creating decorative woodwork and is also particularly useful for sign makers for creating inlayed letters and graphics.

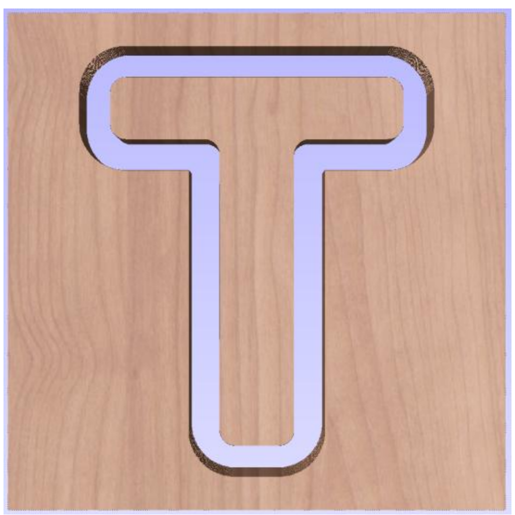
How Inlays work

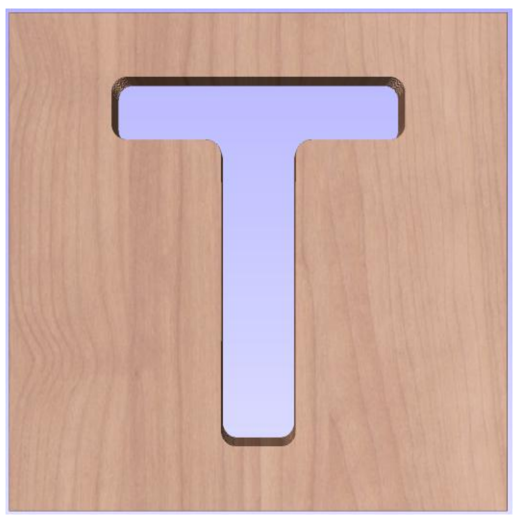
When using a CNC machine to cut out shapes then the tool will always leave a radius on any internal corner. When cutting holes or pockets the tool conversely leaves a radius on the external corners. If no changes are made to accommodate this then there would be no way to fit one part into the other. This is shown highlighted on the letter "T" shown in the image below. On the left you can see the internal corners with a radius on the part being cut out (highlighted with red ovals). On the right you can see the radiused external corners on the hole (highlighted with green ovals). As you can envisage trying to slot the "T" into the hole would not work as it is the proverbial square peg in a round hole!





It is not possible to avoid the added radius as it is formed by the tool size and shape. The Inlay function though, will create toolpaths which take the tool radius into account and compensate for it by rounding off the sharp corners so the resulting parts will fit together. This can be seen in the images shown below where the same letter has been cut using the Inlay toolpaths, you can see all corners (internal and external) now have the same radius so they will slot together.





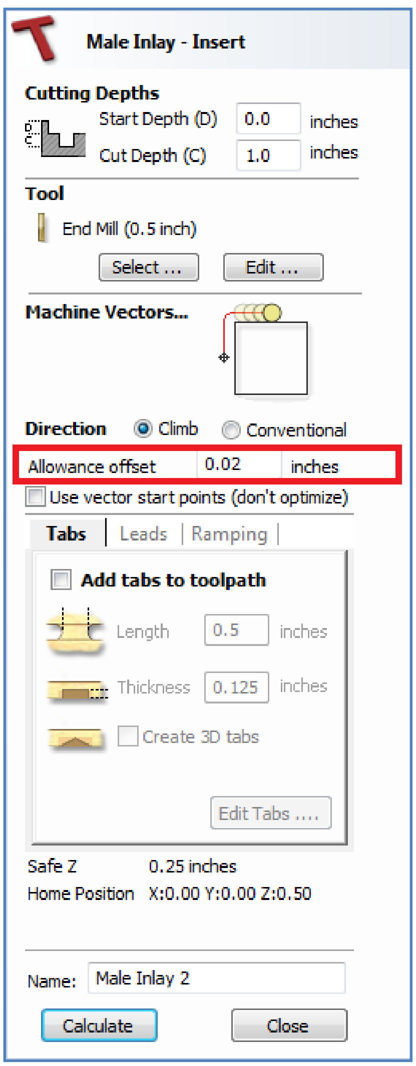
IMPORTANT NOTE: Using the same tool for Inlays

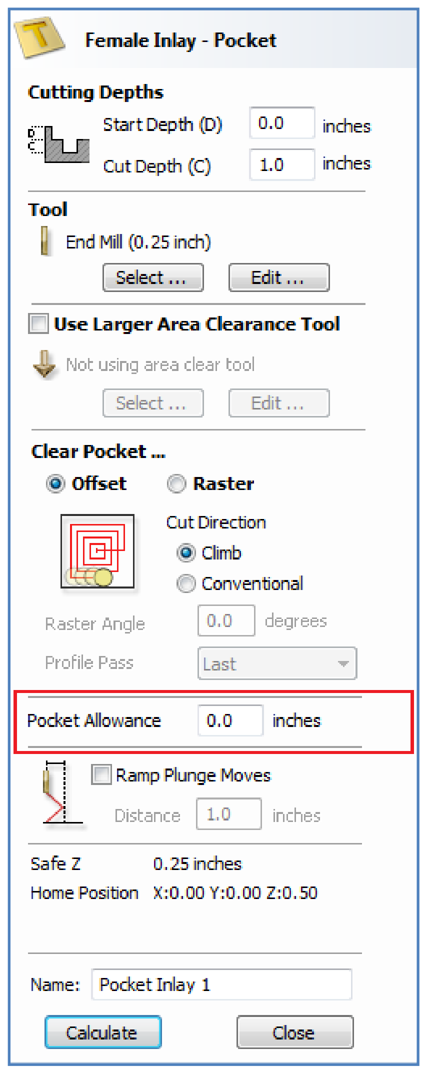
When creating an Inlay toolpath the radius is automatically compensated for, so it is very important to make sure you specify the same tool for both parts of any inlay (male and female). If you do not do this the **Inlay** will not fit together. If creating a **Pocket** then the main **Tool** (the finish tool) not the larger (clearance) tool should be the same as the one as used for creating the **Male Inlay**.

IMPORTANT NOTE: Allowance for Inlays

Although the radius of the tool is compensated for when using the inlay tool, this in itself will almost always not be enough to ensure the two parts will fit together correctly. The parts will be exactly the same size and so would not fit together without a lot of force which would damage the part or without some kind of post-CNC hand work. Cutting the parts exactly the same size also does not allow for any kind of finish to be applied to either side.

To enable the parts to fit extra material either needs to be cut from the **Male** side, the **Female** side or in some applications both. This additional distance is added using the option in the toolpath menu to add an Allowance. On the toolpaths which are based on a **Profile** then this is done with the **Allowance Offset** (Shown in the image below left – highlighted with a red box). For **Pocket** style toolpaths use the **Pocket Allowance,** (shown in the image below right highlighted with a red box). When a value is entered it will overcut the selected shape by this distance, this will reduce the size for male parts and increase the hole/pocket for female parts.

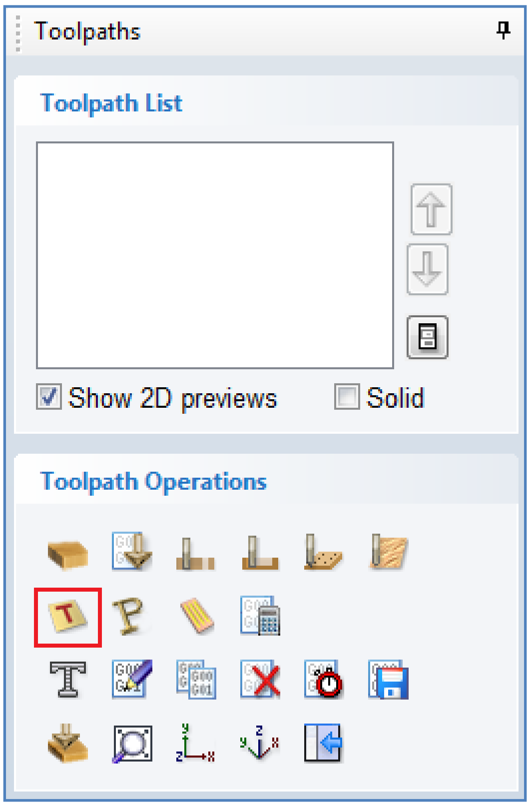




The size of the **Allowance** required will depend on the type of material (how much it may expand or contract between being cut and inlayed), the accuracy of your tooling, the accuracy of your machine and finally any finish you are planning to add to the finished parts (such as paint or varnish which will have a thickness). In most situations where no finish is being applied before the parts are inlayed then an allowance of 0.01” or 0.02” will be sufficient. If you are not sure what value to use then you should experiment with this on a test part to get the correct sizes for your particular setup and application.

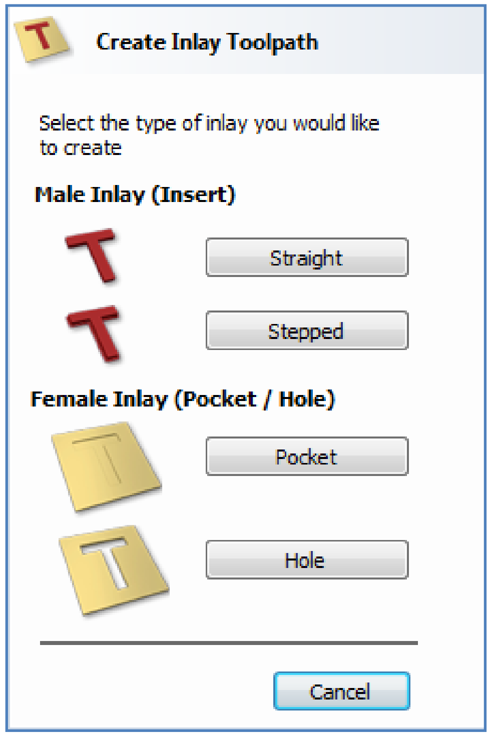
In the majority of cases the **Allowance** is applied to the **Female** side of the inlay as it is typical to not want to alter the actual vector shapes (the **Male** side) any more than is required for the tool radius. This means the **Hole** or **Pocket** will be cut over-sized to provide the additional allowance for the parts to fit.

The icon for the **Inlay Toolpath** is located in the **Toolpath Operations** area of the **Toolpaths** menu, this can be seen highlighted with a red box in the image shown below:



When you click on this the menu shown below will appear, you have a choice of four types of **Inlay**, two **Male** and two **Female**.

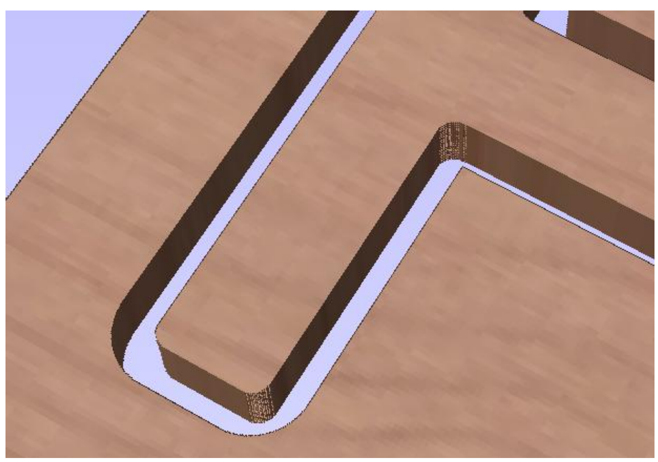
Both the **Male Inlay** options and the **Female Hole** option use variations on the standard **Profile** toolpath, the **Female Pocket** option uses a variation of the **Pocket** toolpath. In this section we will not document the complete menu for each of these but just where they differ from the standard versions of each toolpath menu. To learn more about the standard **Profile** and **Pocket** options please see the relevant sections in the **Reference Manual**.



Male Inlay (Insert)

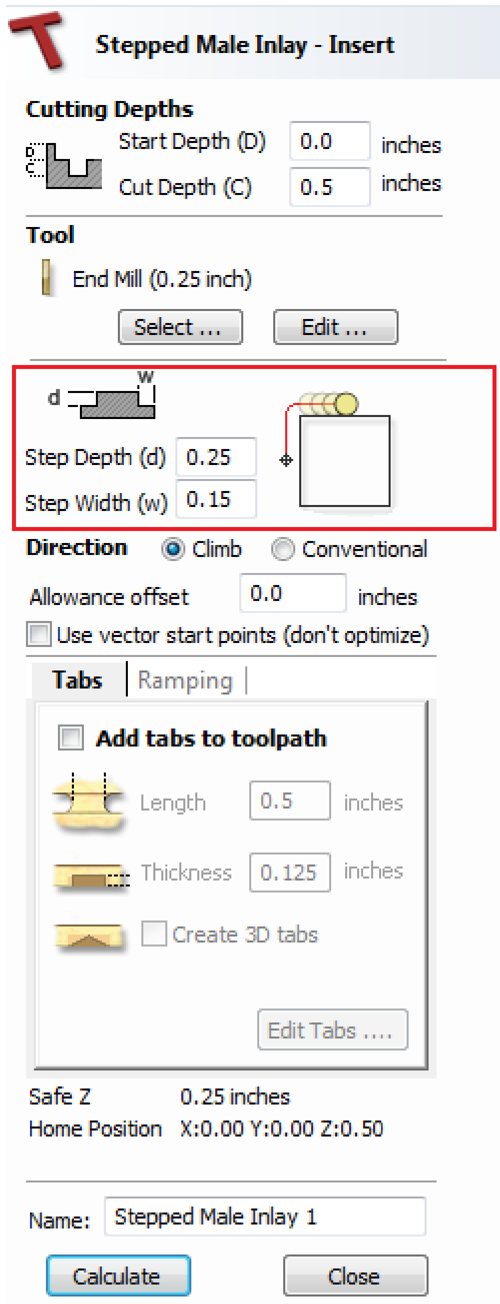
Straight

This option is for cutting out straight sided parts to act as the inserted piece of the inlay. It uses a variation on the **Profile** toolpath which will automatically round the external corners of the part to allow for the radius of the tool being used. All the standard **Profile** options are available in this menu except there is no option to **Profile Inside** or **On** as this does not apply to this inlay type as it has to cut outside of the vector. The other standard option not available is the ability to add sharp **Corners** as again this would not apply to this application. Note in the image shown below how the external corners are rounded based on the tool radius being used.



Stepped

This option is for cutting out stepped sided parts to act as the inserted piece of the inlay. This style of inlay is typically used for what are referred to as “Push Through” letters and shapes. These are parts which are inserted from the back and use the step as a shelf to invisibly mount them to the back of a sign. As with the **Straight Inlay** option it uses a variation on the **Profile** toolpath with the addition of the ability to specify a **Step Depth** and **Step Width**. These are defined in the area of the menu highlighted with a red box in the image below:



As the diagram shows in the menu the **Step** Depth is the vertical height of the step from the **Start Depth** down, the **Step Width** is how far from the edge of the original vector the part will be cut out (creating the step).

The image shown above shows a Stepped Male Inlay created in 1 inch thick material with a **0.75" Step Depth** and a **0.3" Step Width**.

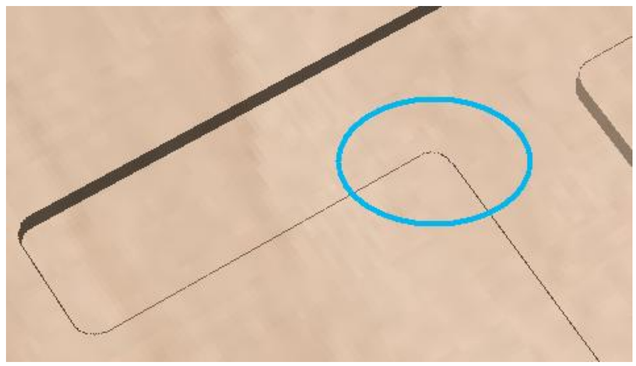
All the other standard **Profile** options are available for **Stepped Inlays** except there is no option to **Profile Inside** or **On** as this does not apply to this inlay type and this option does not allow the use of either the **Corner** or **Lead** functions as these are also inapplicable to this type of toolpath.

IMPORTANT NOTE on Stepped Inlay Spacing
It is very important when working with **Stepped Inlays** that you allow enough space in between the shapes you are cutting out for the **Step Width** to be added. If the parts are too close then they will not be cut-out and the steps will not be properly formed.

Female Inlay (Pocket / Hole)

Pocket

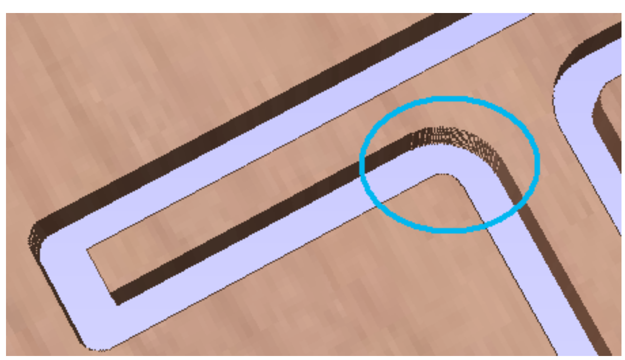
This option is selected for cutting out a **Pocket** to act as the cavity for the corresponding **Male** shape to inlay into. It uses the same options as the standard **Pocket** style toolpath including the new option where you can use a larger and smaller tool to clear the pocket (smaller tool size should match that used for the Male Inlay). In order to make it work as an inlay pocket any internal corners will be rounded based on the tool radius to allow the male parts to fit into them (shown in the image below).



As has been previously discussed it is common to leave an allowance on one or both sides of the inlay, typically when working with letters the allowance would be left on the **Pocket** so the size and shape of the lettering itself is not affected any more than necessary.

Hole

This option is for cutting out a **Hole** to act as the cavity for the corresponding **Male** shape to inlay through. The standard **Profile** options are available for **Female Hole Inlays** except there is no option to **Profile Outside** or **On** as this does not apply to this toolpath type and this option does not allow the use of the **Corner** options as these are also inapplicable to this type of toolpath. In order to make it work as an inlay any internal corners will be rounded based on the tool radius to allow the male parts to fit into them (shown in the image below).

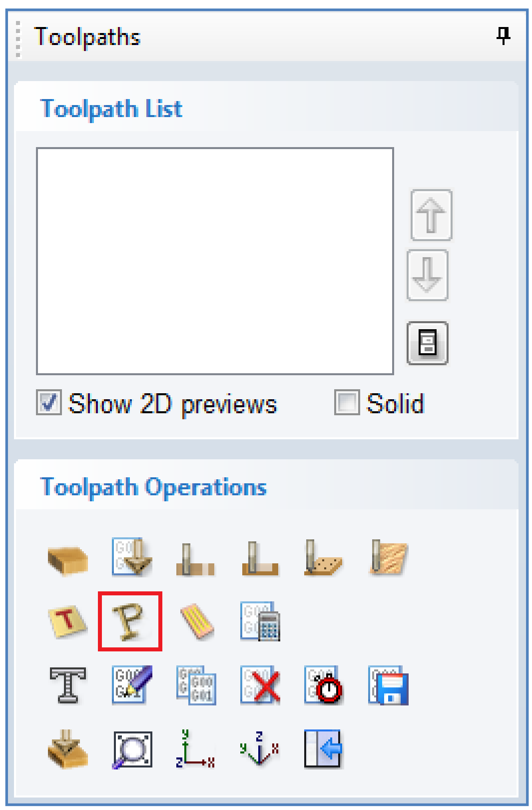


As has been previously discussed it is common to leave an allowance on one or both sides of the inlay, typically when working with letters the allowance would be left on the **Hole** so the size and shape of the lettering itself is not affected any more than necessary.

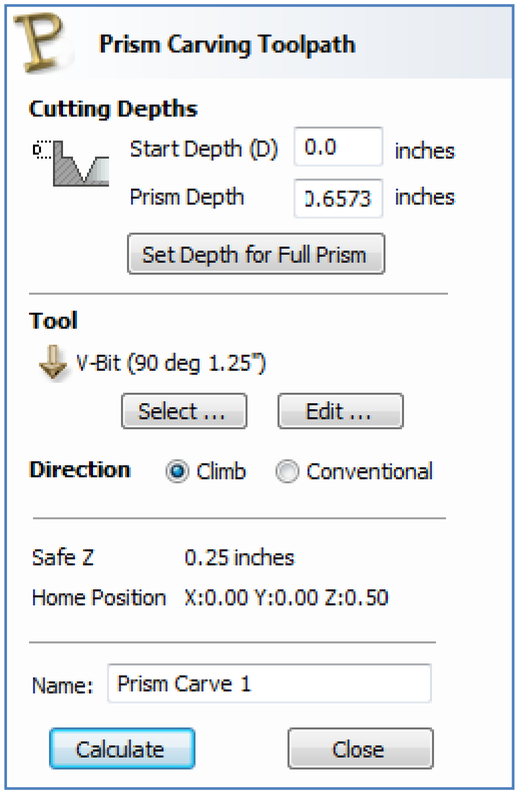
Prism Carving Toolpath

Prism Carving uses an angled tool to create a raised prism shape on the top of the selected vectors. The tool will profile at a given depth creating a nice sharp finished shape such as the lettering shown in the image below. This is often paired with a **Profile** or **Pocket** toolpath to carve the vertical edge around the shapes or clear out the material between them. This type of toolpath is often though not exclusively used on lettering.

The **Prism Carving** option can be found on the **Toolpaths** menu under **Toolpath Operations** area, the icon is shown in the image below highlighted with a red box.



Clicking this icon will open the **Prism Carving Toolpath** shown in the image below.



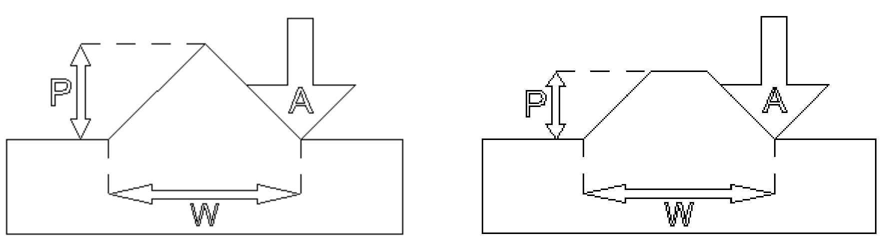
Cutting Depths

Start Depth (D)

Start Depth (D) specifies the depth at which the **Prism Carving** toolpath is calculated from. When cutting directly into the surface of a job the **Start Depth** will often be Z0. If machining into the bottom of an existing pocket or stepped region, the depth of the pocket/step that you are starting from must be entered here.

Prism Depth

This sets the depth of the **Prism Carving** toolpath relative to the **Start Depth**, the total depth of the base of the prism shape (within the material) will be the combination of the **Start** and **Flute Depth**. This depth is particularly important to set correctly as if it is too shallow then the prism shape may be truncated so it will have a flat top (shown in the image below right). The minimum depth needed to avoid this is determined by the widest point on the vector/s selected (W) and the angle of the tool (A). This can be calculated automatically by using the **Set Depth for Full Prism** function (see below for more details).



Set Depth for Full Prism

For this button to work you both need to select the vectors you plan to toolpath and also have selected the tool you are going to use, then when you click it the **Prism Depth** will be set to the minimum required to ensure a full point on the prism for the current selection/tool.

Tool

Clicking the **Select** button opens the **Tool Database** from which the required tool can be selected. See the section on the **Tool Database** for more information on this. Clicking the **Edit** button opens the **Edit Tool** form which allows the cutting parameters for the selected tool to be modified, without changing the master information in the database.

IMPORTANT NOTE: Tooling size and quality for Prism Carving

Prism Carving is predominantly cut using a V-shaped cutter, having sharp tooling which is accurately sized is very important to getting good results. You should measure your cutters to make sure the size and angle of the v-bit are as per the manufacturers specifications as a variation of even 1 or 2 degrees on the angle can make a big difference to the quality and precision of **Prism** carved shapes.

Direction

Can be set to either Conventional or Climb machining, the choice for this will largely be dictated by the material being machined and type of tool being used. See the section on Profile Toolpaths in the Reference Manual for more information on the differences between these.

Safe Z

The **Safe Z** value defined the distance above the job at which it is safe to move the cutter at rapid/max feedrate. This dimension can be changed by opening the **Material Setup** form.

Home Position

This is the position that the tool will travel to before and after machining. This dimension can be changed by opening the **Material Setup** form.

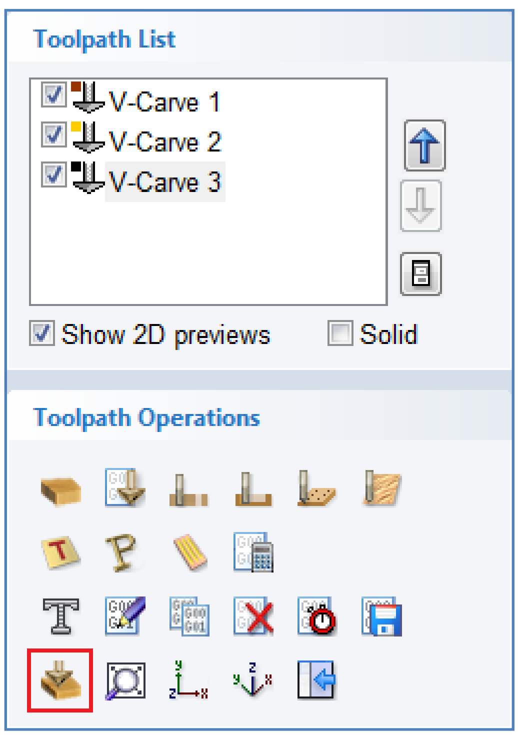
Name

A name for the **Prism Carving** toolpath can be entered or the default name can be used.

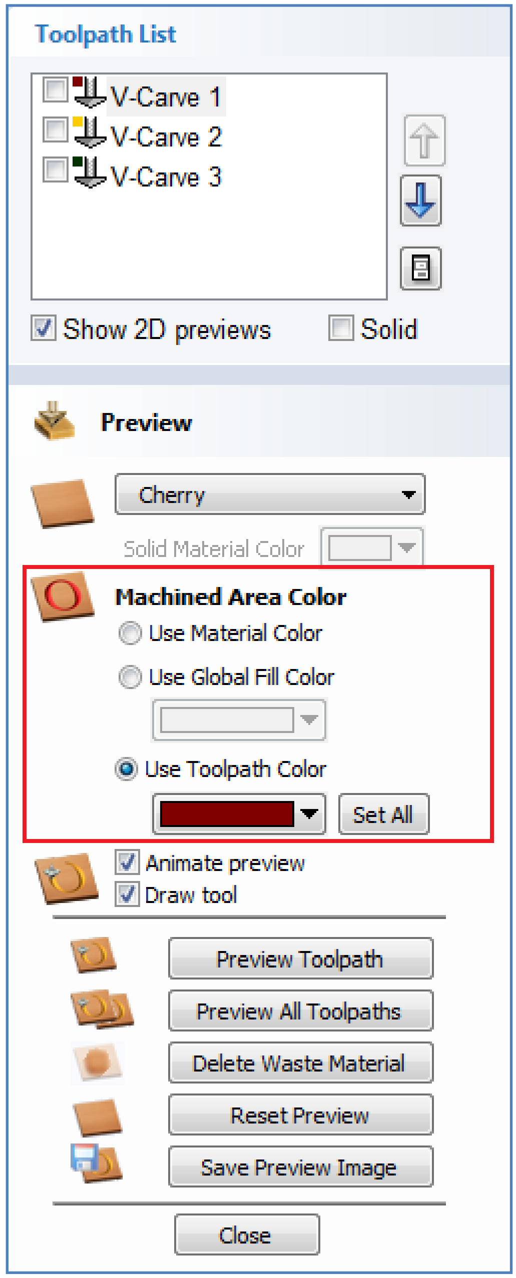
Multi-Colored Toolpath Preview

When previewing the effects of each toolpath it is now possible to assign different colors to each one. This can be used to simulate the effects of differently painted areas both as a way to verify what is being cut by the toolpaths and also to provide an image than can be emailed or printed for customer approval. An example of the new shading in use is shown in the image below of the Howling Wolf Sign, with the new preview on the left and the actual finished sign on the right.

The different colors can be set from the **Preview Toolpath** menu – this appears automatically when a toolpath is calculated or can be accessed anytime by clicking the icon under the **Toolpath Operations** area of the **Toolpath** menu, shown below, highlighted by a red box.

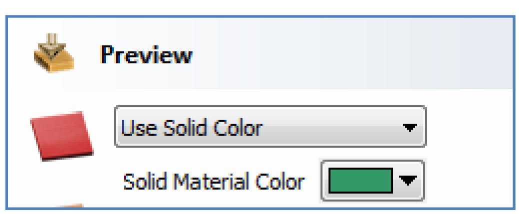


When this icon is clicked (or immediately after a toolpath is calculated) then the **Preview** menu will be shown. The new options for defining the fill color are contained in the **Machined Area Color** area of the form highlighted with a red box in the image below.



IMPORTANT NOTE: Solid Material Color

Before the individual toolpath color fill is covered it is worth pointing out another new feature within the shading menu. It was not previously possible to choose a solid color for the **Material** surface that was different from the **Fill Color**, it was only possible to choose a material from the list then solid color fill. There is now the option at the very top of the **Material List** to select a **Solid Color** for the surface of the part. Once this is selected the color choice can be made from the drop-down **Solid Material Color** area. This allows material with a different solid color surface and core to be easily represented. The **Use Solid Color** option is shown in the image below:



Machined Area Color

Use Material Color

Checking this option will use the same color/material for the fill as you have selected for the material itself as shown in the image below.

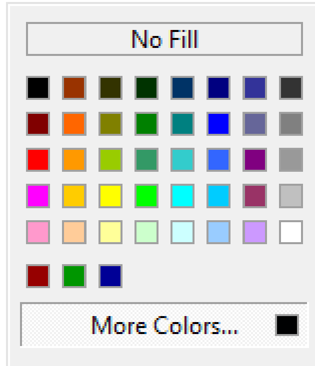
Use Global Fill Color

Checking this will use the same single fill color for all toolpaths as shown in the image below.

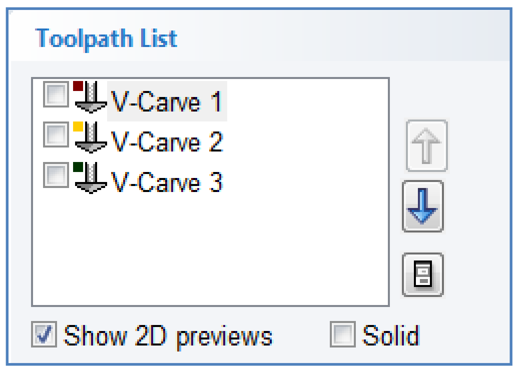
Use Toolpath Color

Checking this will let you select different colors for each toolpath in your list as shown in the image shown below.

To set the color individually, first select the toolpath from the **Toolpath** List and then click on the drop down arrow next to the color block to show the menu shown below:



Choose the color you want for the fill of that toolpath and it will be applied to the areas that the toolpath has carved when they are previewed. Once you assign an individual color a small square of that color will be displayed next to the name in the toolpath list. This can be seen in the image shown below just to the left of each tool icon.



If you click the option at the top of the color selection form where is says "**No Fill**" then this will leave that toolpath in the selected **Preview Material Color**.

Set All

Clicking the **Set All** button will set all the toolpaths in your **Toolpath List** to the currently selected color. ONLY click this if you want to change all toolpaths to have the same color as you cannot undo this operation.

Where can I get 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 – Help Contents**

2. **Video Tutorials** - These are available for download from the Vectric website http://www.vectric.com under the **Support – Training Materials** section of the 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.