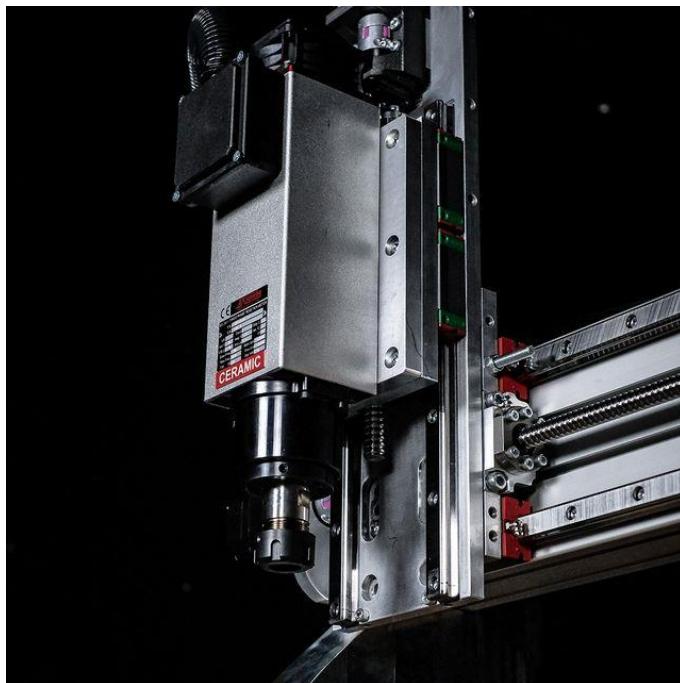




Section 5 Cut2D~Vcarve~Aspire setup



4.0 ROBOTICS



BG Precision
Version 1.0
April 2022

BG Precision PTY LTD

Unit 1/82 Brunel Road

SEAFORD

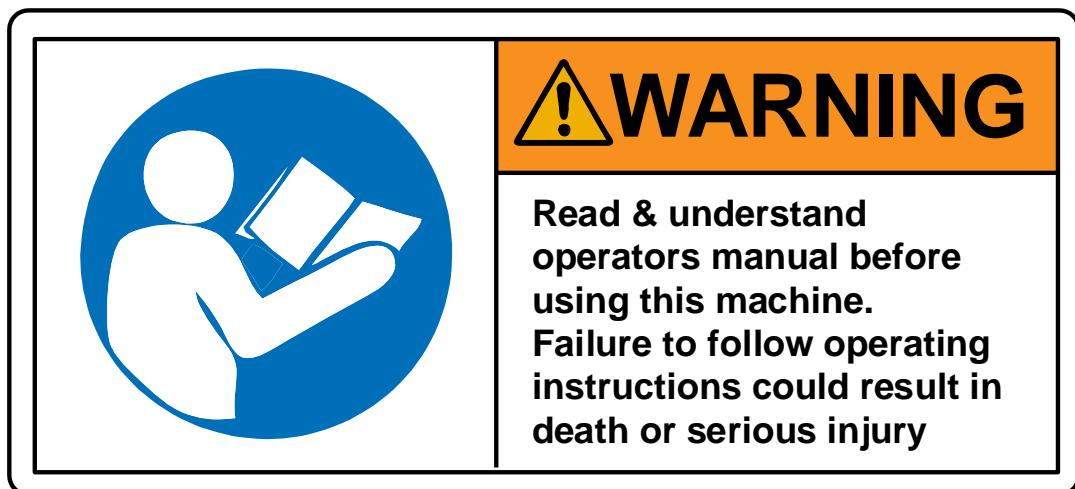
3198 VIC

Australia

Mob: +61 (0)405685515

Email: info@bgprecision.com.au

Website: www.bgprecision.com.au



Before using/turning on the machine, the device should be carefully checked to make sure all connections are secure and the device is technically sound.



**Ensure You understand
the safety considerations
of a machine provided
in the open configuration
without a safety
enclsoure**



**Do NOT Interfere
with the machine
when under CNC
control**



**NEVER LEAVE
THE MACHINE
WORKING
UNATTENDED**

TABLE OF CONTENTS

1	Introduction.....	5
1.1	Introduction and overview.....	6
1.2	Vectric Software installation.....	6
1.3	Job Setup - CAD.....	7
1.4	Job Setup – CAM.....	9
1.5	Post Processor for MTC.....	15
1.6	Post Processor for QTC	15
1.7	Help and Support for Vectric	15

1 INTRODUCTION

Thank you for purchasing your CNC system from BG Precision. This section of the manual is specific VECTRIC CAD/CAM Software. Supplementary material will be provided specific to the machine make and model of which you have purchased. We support UCCNC PC based control in conjunction with Vectric products. Cut2D and VCarve Pro are the two most popular CAM packages we support. The combination of Cut2D/VCarve Pro and the UCCNC is a very user friendly and powerful machine control package, whilst remaining easy to use. Visit www.vectric.com where you can download trial versions of Vectric CAD/CAM packages and run them in demo mode where you can test them out.

Please ensure you read all the operational manuals for this CNC machine prior to attempting to use the system. Through-out this manual there are references to "A Trained Operator" or "Trained and Experienced personnel". These are defined as follows:

All persons that uses, or comes into contact with, the CNC router system MUST:

- understand what a CNC router is and can do
- read and understood the content of this user manual prior to using the system
- be able to exercise control of the router system at all times
- follow all the guidelines presented including the use of appropriate PPE
- seek further instruction if anything is unclear
- be sure that you have understood these instructions completely

Responsibility of use or misuse belongs to the end user. BG Precision PTY LTD and its affiliates accept no responsibility for use or misuse by the user. If you may not be able to use this product properly, we recommend that you do not begin use or cease use immediately.

This manual was not intended to cover every facet of machine operation. This manual serves to provide the information needed to safely operate and maintain the CNC router system. This manual has been designed to be used as an instruction tool as well as a reference tool for everyday work. Step by step instructions are provided where possible to help all levels of users understand the machine.

NOTE: Important aspects of machine use and best practice are highlighted and should be adopted where possible to maximise the machine tool life and performance. It is VERY IMPORTANT that all personnel read and understand the safety chapter BEFORE operating the machine. All Warning and Caution notices must be noted before interacting with the machine. Please refer to STEP 1 – Introduction to CNC for all safety considerations.

If there are any further questions or if anything is not clear, please contact us at info@bgprecision.com.au

1.1 INTRODUCTION AND OVERVIEW

Cut2D is a very popular CAM package and is very easy to use. Cut2D is very similar to VCarve Pro, however VCarve Pro has a lot more functionality (which is why it is more expensive). The best way to learn how to use Cut2D is to open the software and try to use it. You can see the live preview that gives you an idea of what the part will look like. You can also see the machine tool path overlaid also. Cut2D gives you a preview so you can check what your part will look like and check for mistakes.

In this section we are going to talk through cutting a basic job.

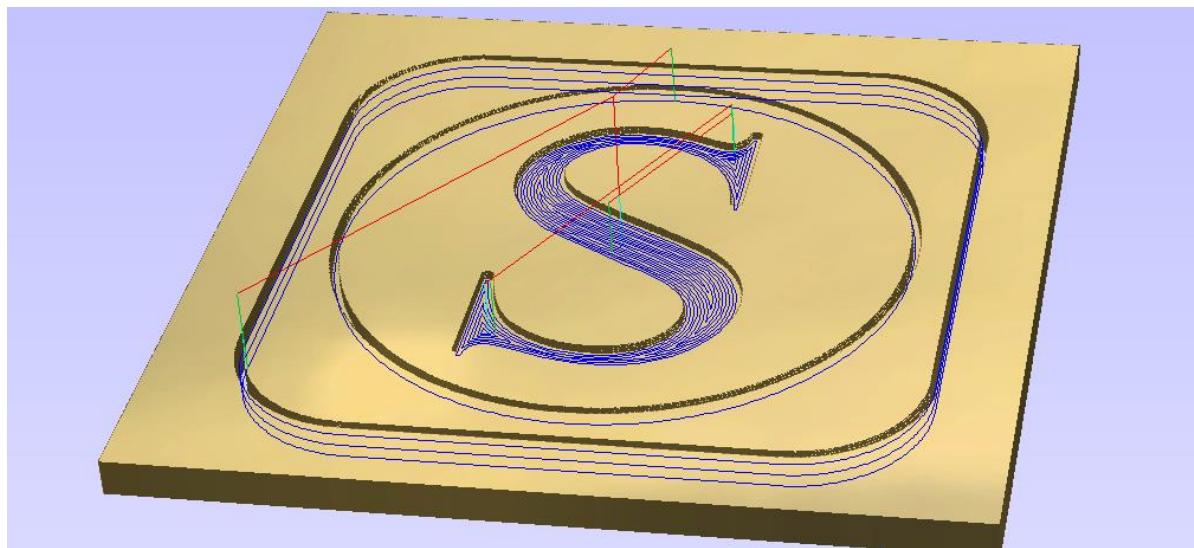


Figure 1: Vectric Screenshot

1.2 VECTRIC SOFTWARE INSTALLATION

When Vectric software is provided by BG Precision – the license number and user name specific to your machine should be on a sticker on the rear of the software case provided. If you purchased VCarve Pro or Cut2D from us and you do not have a case with a licence number – please let us know.

If you have purchased a digital copy of Vectric, BG Precision will provide you with an OEM license and a registration link. Please follow the link provided and you will be prompted to enter in your details associated with the purchase of this OEM license for updating in the Vectric Portal. Once complete and verified by Vectric (may take up to 24hrs on business days only) you will be set up with a V&CO account of which you will always have a digital copy of your Vectric Purchase, be prompted about updates and can use that profile to install your purchased Vectric software on other PCs.

If you purchase from Vectric directly, then Vectric will supply you with your software license key.

1.3 JOB SETUP - CAD

At this stage you need to create either a 2D design (Vectors), a 3D computer model (Components), or a combination of the two. This data will be used to calculate the paths the CNC will follow to cut the finished parts.

Depending on the reference material you have, you may start with a blank job and create everything within the software, or you may be importing information such as an image to trace or an existing 2D or 3D design supplied from another software package.

Once you have an idea of what you are trying to produce, the next step would be to start the software and setup a new project, followed by doing the actual CAD (design) work.

Next, we will talk about some of the important parameters in Vectric Job setup.

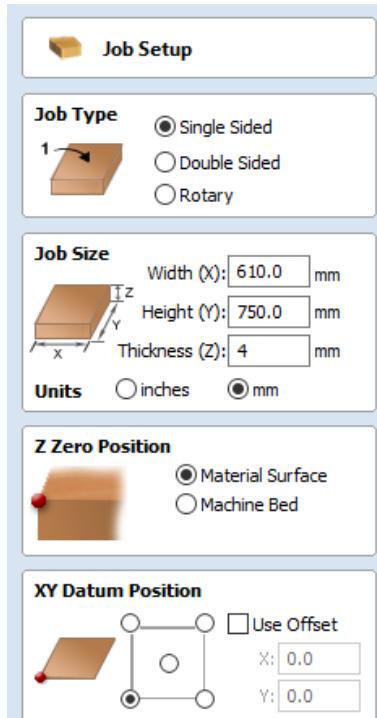


Figure 2: Vectric – Job Setup parameters on VCarve Pro

1.3.1 JOB SIZE

When setting Material Size, it can be the actual job size or just large enough to accommodate the job and space to cut it out. If you are going to be cutting your job from a much larger size of material than its actual size, then it is suggested for the design phase that you setup a material size just slightly larger than your job will be. You can either change this before calculating the toolpaths or simply position the XY Origin where you need it on the larger sheet of material.

Please ensure your units are set to “mm” as the post processor used and the CNC machine is calibrated in mm.

1.3.2 Z ORIGIN OR Z ZERO

The program gives you a choice between the top of the material or the bottom of the material as seen in Figure 2. It is easy to set your bit to the top of the material as the surface is fully exposed. For the bottom of the material if used, you will set the zero point to the bed of the machine, which is where the bottom of the material is sitting. For this reason, it is commonly just referred to as setting Z Zero to the machine bed which is the same as the bottom of the material.

If you zero on top of the material in Cut2D like in Figure 3 below – you need to zero the machine in the z-axis on top of the material also. This is where the tip of the cutting tool would touch the surface of the material.

If you have a sacrificial layer under your part to be machined (which you should) then cutting through deeper than the material thickness will ensure you cut all the way through around the entire perimeter and leave minimal finishing

1.3.3 X-Y ORIGIN OR XY DATUM POSITION

There are many considerations as you choose where to set the XY Origin. There is no right or wrong answer here as much of this is personal preference, but here are some considerations that may help for a specific project:

The programs give you 5 possible choices for this: The centre of the material or one of the 4 corners. Most projects tend to use either the centre or lower left corner, which agrees with how many of the CNC machines have been configured by the manufacturer. Again, this can be changed before the toolpaths are calculated. Many people like to design a project with the XY Origin at the centre but will then change it to the lower left before calculating the toolpaths.

Zeroing the job setup is important. You can see the faint cross hairs on the workpiece drawing as highlighted in Figure 3: Vectric - Job Set-up. The cross hairs appear in the centre of the work. This is because the X, Y origin is setup in the centre in the job setup in the left-hand side. You can setup the job origin to be the bottom left for example. It is very important that you are consistent with the setup here in Cut2D and also with the machine.

If you zero in the middle in Cut2D like here – you need to zero the machine in the middle of the work.

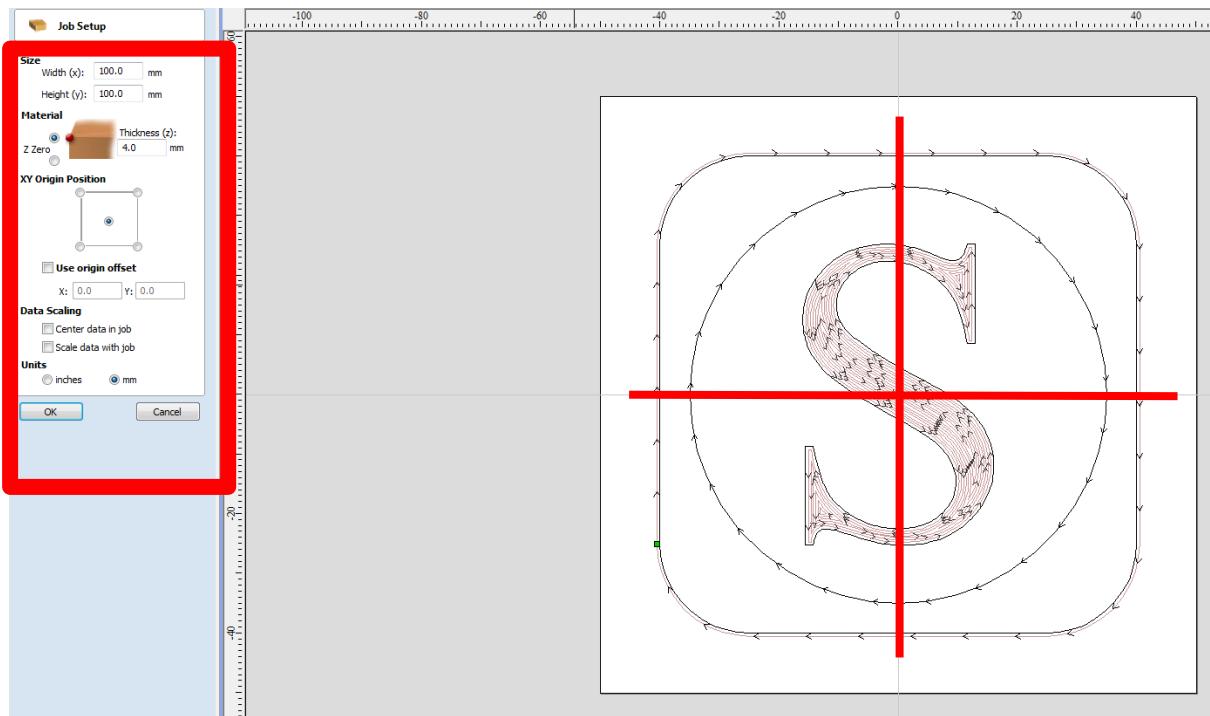


Figure 3: Vectric - Job Set-up

1.4 JOB SETUP – CAM

The first thing you need to do before you calculate any toolpaths is confirm your material setup and position, relative to how it will be set up on the machine itself. Although you will have previously made some choices on this when you setup the job for the design layout, you will need to double check they are still correct and potentially make changes based on things that may have evolved as you completed the design. For instance, if you set XY zero in the middle of the job for drawing, you may now decide to change this to one of the corners (typically lower left) for machining.

Once your material settings have been verified or edited, you are ready to start generating the Toolpaths. To do this you will run through this set of procedures.

1.4.1 TOOL SETUP

See Figure 4 below for the tool window that opens when you select the tool to use in cut2D/VcarvePro. Both Cut2D and VcarvePro allow you to build up a tool database. You can setup a tool for each materials/cutter combination. You can save tools for various jobs. As you learn you will find some settings work better than others. When you find the sweet spot you can save the settings in the tool library.

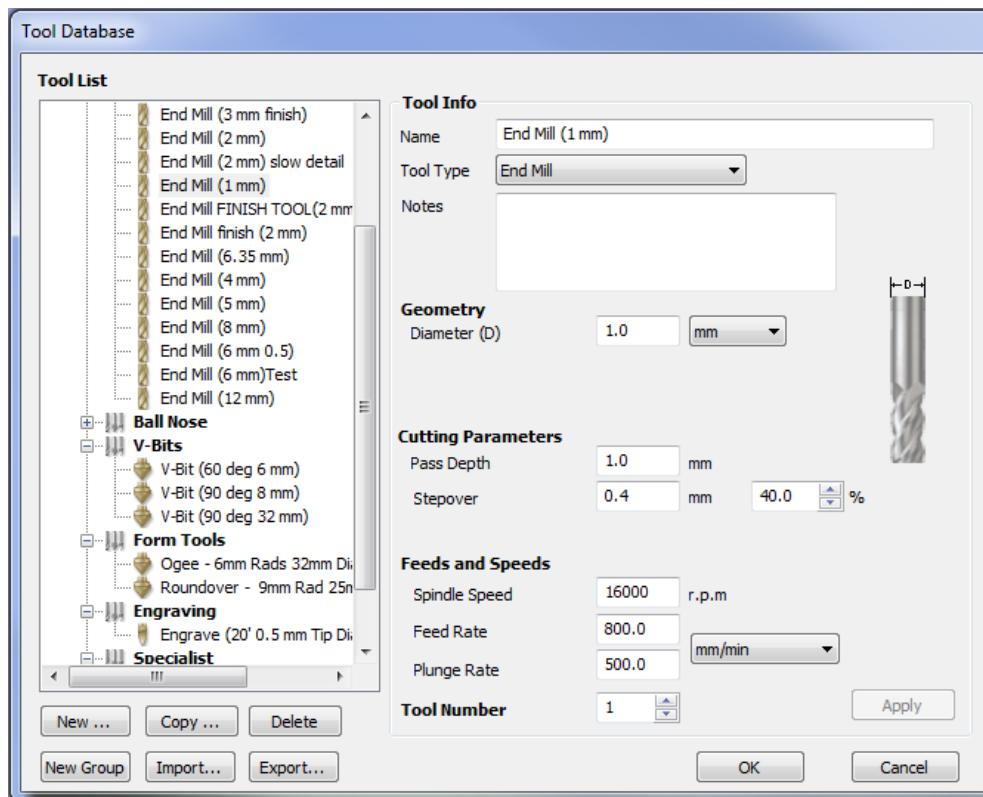


Figure 4: Vectric - Tool Database

NOTE: It is VERY important that you set your Feeds to be in mm/min

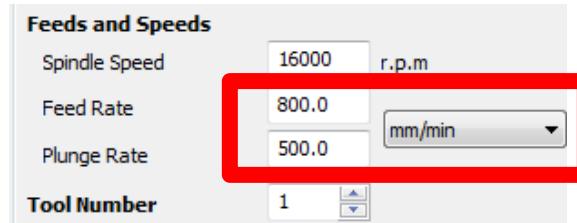


Figure 5: Vectric - Feeds and Speeds Settings

To help you with your settings they are explained as follows:

- **Name** – this is not important for the settings – but its good practice to name the tool to be descriptive of what is in the settings.
- **Tool Type** – this is important as it tells Vcarve what sort of tool you are using. This will affect the calculations and the live demo so always choose the same tool as you are using
- **Notes** – this field is useful for documenting specific outcomes or requirements needed when running with specific tools and settings. For example “make sure to clamp work piece well when running with these settings”
- **Diameter** – The diameter of the tool tells Cut2D how to approach tool radius compensation. If for examples you set the tool diameter to 4mm, Cut2D will then make the toolpath run 2mm

outside the path you specify (if compensating for tool radius on the outside) to account for the tool geometry.

- **Pass Depth** – is the maximum depth that tool can go at once. If your cut depth is set at 5mm for example. And you are using a 1mm cutter. A good strategy would be to machine the 5mm in 1mm passes. Cut2D will then program the machine to cut down 1mm at a time down to the 5mm cut depth.
- **Stepover** – this setting is related to the pocketing function where you need to machine out a pocket and not just an outline/profile toolpath. The stepover controls how much overlap there is as the cutter pocket out the work.
- **Spindle speed** – Your spindle speed will be set on the dial on the front of the machine. If you look at the front of the controller you will see a small screen. This is the VFD screen and you can adjust the spindle speed. 0 = ORPM. 400HZ - 24000RPM
- **Feed Rate** – this parameter is the speed of the machine when machining the work in the X and Y directions. The machine can rapid move at up to 5000mm/min. Typically machining speeds are at 700-2000mm/min. Stick to 1500 with a pass depth of 0.5mm or 1mm when you get started. You can build confidence and speed as you get used to using the machine. If your machine too fast the machine will stall and you will lose position. So tread carefully here.
- **Plunge rate** – is the same as the feed rate except it related to the Z direction. Typically milling cutters (like slot drills etc) prefer to mill in X and Y and are not as efficient in the Z / plunge direction. Keep this setting at 500mm/min or so as you get started.

1.4.2 TOOL RADIUS COMPENSATION

In order for Vcarve/Cut2D to carry out effective tool radius compensation (accounting for the fact that the tool has a diameter) Make sure that when you select your tool:

- You select the correct tool type – in this case an end mill. You will see an image of a tool sample on the right-hand side in the above below.
- Make sure you enter in the correct tool diameter. The tool diameter is the diameter of the “hole” or the width of the “channel” the tool will make in the work piece. One way to measure this (if not written on tool) is to slowly plunge the tool into a sample work piece – say MDF or plywood (jog only in Z and make sure the spindle is on). The circular hole the milling cutter makes in the work piece (ply or MDF) is the cutter diameter. You can now measure this with a ruler or calliper. NOTE it is best to specify the cutter diameter from the supplied cutter specifications.

- Take some notes for specific jobs. You might find the optimal settings for Perspex for example. It is a good idea to save a new tool with these settings so you always have this tool in your tool library for the next time.

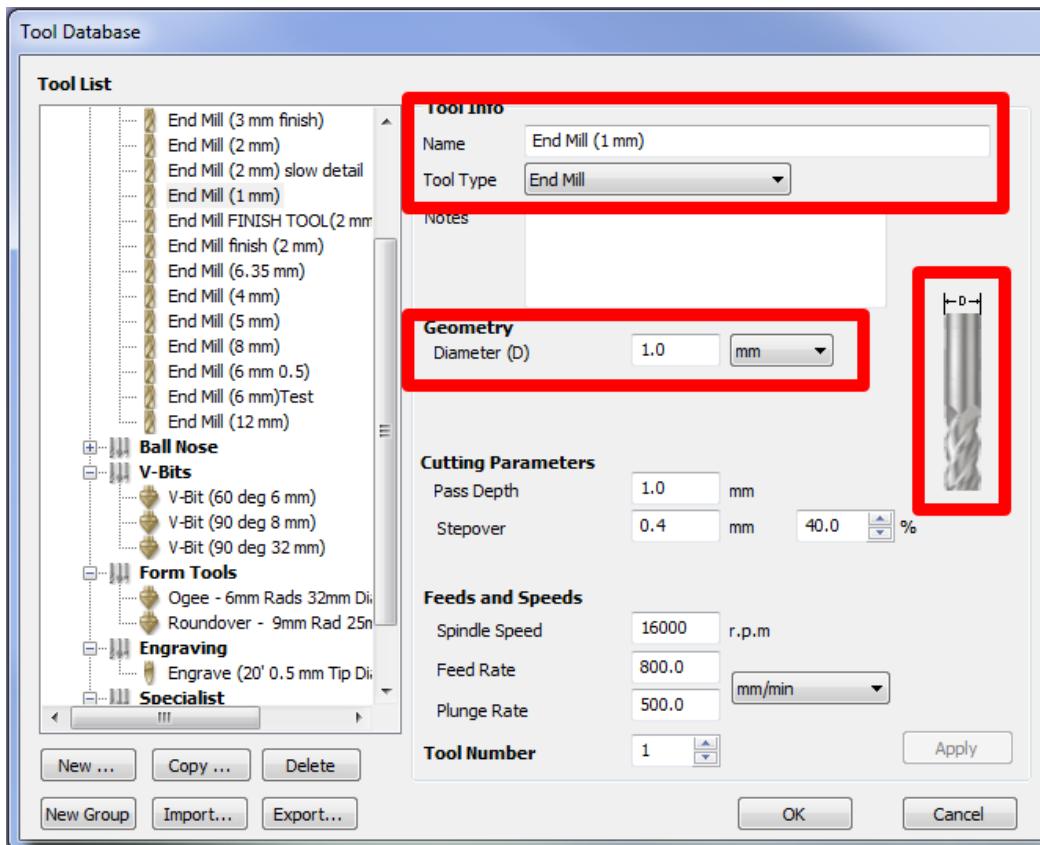


Figure 6: Vectric - Tool Database Parameters



1.4.3 SELECT A TOOLPATH OPERATION

Profile, VCarve, Pocket are all examples of toolpaths. For example, a “VCarving” strategy can be used to engrave incised vector letters or a “Profile” strategy may be chosen to cut-out a vector shape. You will find good information on the different Toolpath types in the Help document and Reference Manual on Vectric. Each one has different applications, there are many examples within the video tutorials that cover all the main uses and some specialized functions.

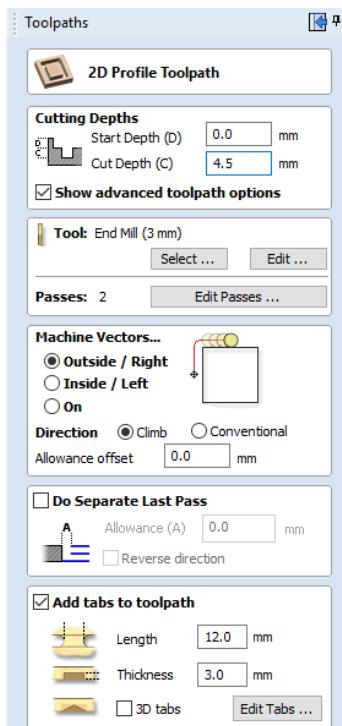


Figure 7: Vectric – Toolpath options

As we have set the thickness of the work to be 4mm (as per our plywood), and we set the cut depth of the final outline cut to be 4.5mm, Cut2D will give a warning. You can still proceed, it's simply telling you that you are going to machine deeper than your materials thickness.

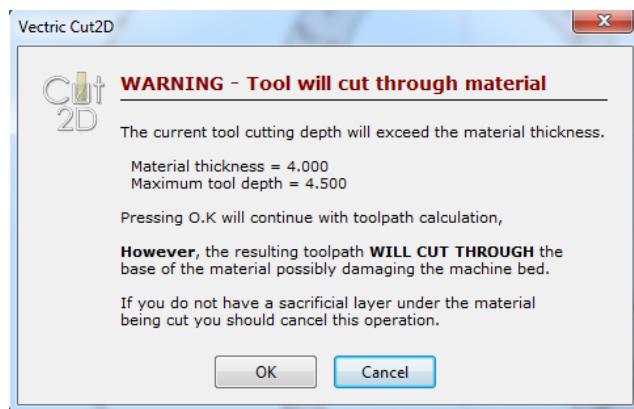


Figure 8: Vectric - Tool will cut through material warning

Each toolpath has options that can be set to customize it for a particular cut. These options vary and are dealt with in detail in the documentation and the tutorial videos. Some toolpaths have very few modifiers and some have a lot of additional choices depending on the type of cut and how much control the user may require over it. For instance, you may want to add "Tabs" to a toolpath where you are cutting out a job using a "Profile" toolpath strategy. That would let you specify the size and position of small pieces of material that leave the cut object still partially attached to the original stock to hold it in place while it is being machined. This is just one example of the way a toolpath can be adjusted. As a

CNC is flexible enough to use in many different applications and to cut a range of material it's important to have these options available when calculating different types of cut

1.4.4 SELECT A ROUTER BIT

Select a tool that will be best suited to machine the toolpath in the material you are using. Some toolpaths, by definition of what they do, are limited to the types of tools that can be assigned to them. For example, a VCarve toolpath can only use a V-Bit or Ball Nose tool. For each tool selected, you can 'Edit' the settings for that bit to exactly match the requirements for that project. A list of tools, their type, size and appropriate settings are remembered by the software and accessed from the "Tool Database". Tools can be added, edited and deleted to personalize this list to your specific needs. Over time this makes toolpath setup quicker and more accurate.

1.4.5 CALCULATE TOOLPATH AND PREVIEW

A powerful feature offered in all the Vectric programs is the Toolpath Preview; this allows you to accurately simulate the result of the toolpath on your computer screen in a virtual 3D piece of material.

Creating a simulation like this lets you check if the toolpath is correct based on the tool and settings you have chosen. If it does not look right in the preview, then it will not be right when you cut it. This feature helps you avoid costly mistakes, a few seconds at the computer can save hours at the machine, plus prevent damage, potential safety issues and a ruined project.

This function can also be used to generate realistic rendered images to show your customer exactly what the part will look like when its machined or even to use as images in promotional material to demonstrate the capabilities of what you make.

After you have created and calculated all the toolpaths and are happy with the Toolpath Preview, it is time to save them in the appropriate format for your CNC machine.

1.5 POST PROCESSOR FOR MTC

Part of the process requires you to export the toolpath from Cut2D/VcarvePro to a text file in order to import it into the machine. In order to do this, you must choose a post processor specific to your machine. In the case where you are using UCCNC you must select the “Mach2/3 Arcs (mm)(*.txt)” post processor. The drop-down menu in Vcarve/Cut2D has many post processor options as per Figure 9. Make sure choose the “Mach2/3 Arcs (mm)(*.txt)” in order to generate the toolpath to be compatible with the CNC machine and UCCNC.

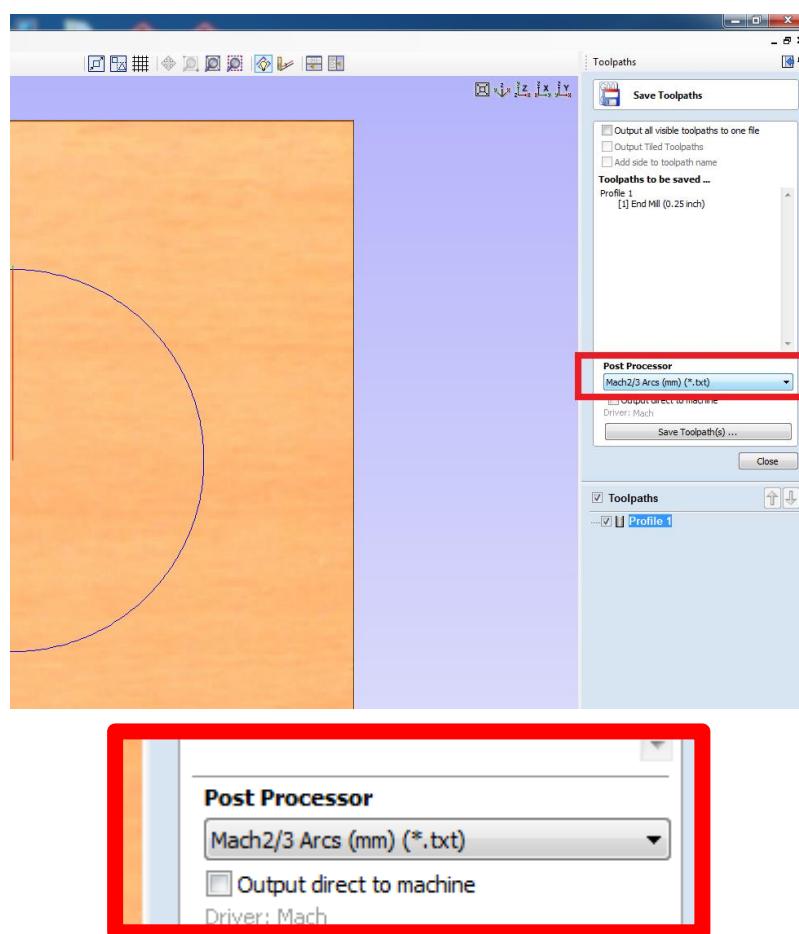


Figure 9: Vectric - Post Processor Selection

1.6 POST PROCESSOR FOR QTC

You must use the post processor provided by BG Precision

1.7 HELP AND SUPPORT FOR VECTRIC

Perhaps the most powerful tool for learning the Vectric products are the free video tutorials that are available for each program. You can find them on the Vectric Support Website: (www.support.vectric.com).