



Introduction to CNC

A detailed beginners guide to the
basics of CNC technology, process
workflow and terminology

Vectric Ltd.

Document V.1.0 – June 2013

Contents

Introduction	3
CNC Overview.....	3
Key Areas of Knowledge	5
1. Computer skills	5
2. Design & Toolpath Software.....	5
3. Operating and Maintaining your CNC Machine	5
4. Knowledge of Materials and Tooling	6
Workflow Overview of a typical CNC project	7
Detailed Workflow of a typical CNC project.....	11
Step 1: Concept.....	11
Step 2: Design (CAD)	12
Step 3: Toolpaths (CAM)	15
Step 4: Machining.....	18
Step 5: Finishing and Assembly.....	21
Summary.....	22
Help for Vectric Programs	22
Help for your CNC machine and its control software.....	24
Important CNC Concepts and Terminology (Glossary).....	25

Introduction

This guide is intended to be an introduction for people who are just entering the exciting and rewarding world of CNC (Computer Numerically Controlled) machining. Its purpose is to familiarize you with the typical steps required to create a CNC project from start to finish and also to provide a glossary of terminology to help you better understand some of the jargon involved. Much of the information would apply to all CNC machines but the focus here is on what are typically referred to as CNC Routers.

If you're a novice, then whether you're looking at a CNC for a new hobby, to start your own business, or are adding the technology to an existing company, then this guide will be a good starting point.

If you don't have a machine yet but are considering a purchase, this document will also help you understand more about how they work and provide some important considerations to aid your decision. For the purposes of this guide, when we discuss the Design and Machining software it will be with regard to Vectric's software products – for more information on the range of software please visit <http://www.vectric.com>

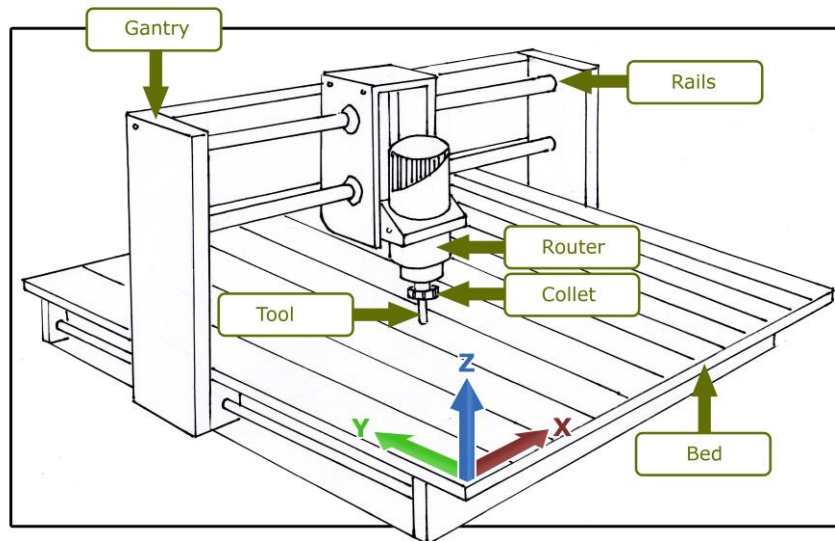


CNC Overview

Broadly speaking a “CNC Router” is a computer controlled machine that has a router or spindle mounted on it that holds a cutting tool (router bit). It is typically set up with 3 directions of movement referred to as the X, Y and Z axis. The position of the router is determined by a computer telling the motors mounted on each axis how much to move in each direction.

Using this method of positioning, any location within the machines work area (envelope) can be defined and the router can be moved within that space. As the machine is driven by a computer telling it where to move, the operator uses a software program to draw the shapes they want to cut and create the path that the machine will follow.

For the purposes of this document we'll just refer to the machines and technology as “CNC”. As with any technical subject there are many variations and nuances in the terminology but at this stage we'll try to keep it as simple as possible. The image below shows a typical layout for a CNC (there are many variations though) and a number of the key components along with an indicator of the possible directions of movement (X, Y and Z).



While CNC is not a simple subject it also does not require you to be a genius to master it. For many, CNC is an extension of an existing skill such as woodworking and your knowledge in these related subjects will be helpful as you progress along the learning curve. To start with though, the best way to approach CNC is as a completely new skill, begin with the basics, read, learn and practice with the software as much as you can and use the many support mechanisms to ask questions and get help. Over time you'll be able to follow the same route to success as the many 1000's of other CNC users (addicts!) who have gone before you.

Key Areas of Knowledge

As with any subject, the more time you invest in learning about CNC and the related technologies, the more you will get from it. To achieve the best results, there are a few key areas which you should concentrate on:

1. Computer skills

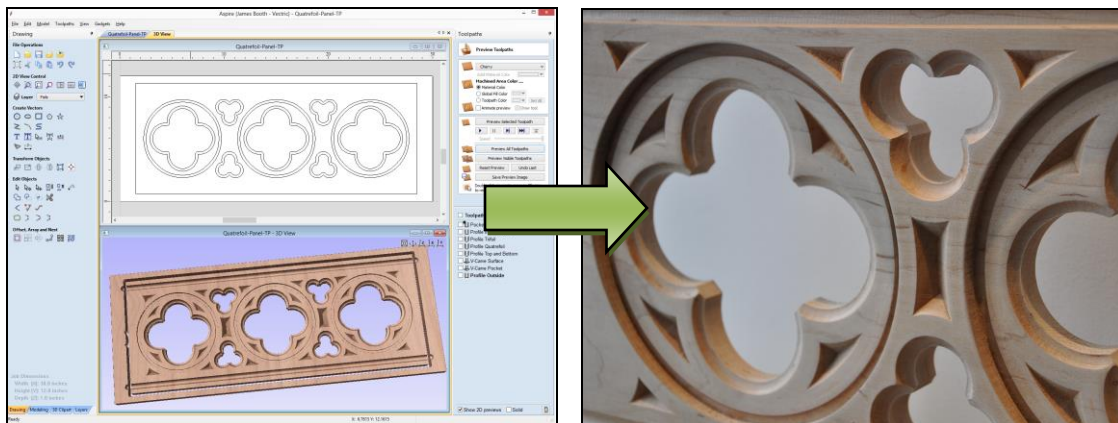
One requirement common to all aspects of CNC work is how to use a computer to perform basic tasks. You will be working with computers and computer programs during almost all the steps of the process as you design your parts and need to understand basic operations such as starting and stopping programs, saving, copying and deleting files, finding files stored on your computer and installing programs and updates.

Your CNC machine is also run by a computer, this may be a standalone PC or a dedicated Control Box.

This guide will assume a basic knowledge of computers and the Windows operating system, if you don't feel comfortable with your current computer skills or are new to running a PC then it would be well worth taking a basic course or buying a general guide to working with your PC.

2. Design & Toolpath Software

Before you can cut anything with a CNC, you need to first create the design layout that the machine is going to follow to cut the parts. The software you choose will play a significant role in successfully creating projects with your CNC. Simply put, the design software will allow you to transform “pencil and paper” ideas to a set of instructions used to run the machine. When done correctly, the end result will be a physical product you can touch and hold that has value and purpose and a great sense of achievement.



3. Operating and Maintaining your CNC Machine

If you currently own or use a CNC machine, you already know how important it is to keep it properly maintained and adjusted, to know and understand its limitations and how to set it up correctly to run a job.

If you don't own a machine yet, then it's important to spend time thinking about what you want your machine to be able to produce, this can eliminate a lot of future frustration. Cost will always be an important factor, but realize that you need to balance that with capabilities, because nothing can be more expensive than a machine that cannot do what you need. For example, if you want to cut large sheet goods then a desktop model will probably not be your best choice. However, if you only have room for a small machine this may be your only option and you need to understand its limitations on how large a part it can cut. Only you can determine what this balance will be for your situation and budget.

Some important considerations when researching the purchase of a machine or when looking at building one yourself include size, speed and accuracy and the technical support offered both before and after the purchase. As with software, the importance of a company's reputation, support, and an active website and/or forum cannot be understated.

Every CNC machine needs software to directly drive its movement; this is commonly referred to as the 'Control Software'. Some common generic third-party packages that do this include "Mach3" and "WINCNC". Many manufactures create and use their own proprietary systems specific to their own models and these may be installed on an external PC or be loaded onto a dedicated Control Box attached to the machine.

Most control systems offer settings that can significantly improve the smoothness and accuracy of your machine when correctly set. While this goes beyond the scope of this guide, it is something worth investigating for your particular CNC. Remember, the best designed project will not cut well on an incorrectly "tuned" machine.

4. Knowledge of Materials and Tooling

When it comes to obtaining the best possible results, you cannot forget the material you are working with or the tool you are using to cut it. The type of material will factor into every stage of the Project – from initial concept through final finishing.

The common materials people using CNC Routers work with include; wood, plastics, dense foam board and softer (non-ferrous) metals (brass, aluminum, etc.). If you are not already familiar with the type of material you want to use, there are many sources of information that can help you.

Typical questions you must answer for the type of material include proper tool (bit) selection, how fast you can move that tool through that material (Feed Rate and Plunge Rate), how much material you can remove at one time (Pass Depth and Cut Depth) and how fast the bit should be rotating (Spindle or Router speed). Typically suppliers of tooling offer information on the correct settings for the router bits they sell.

You will also find many sources of information on other users experiences with different tools, materials and settings on websites and user forums such as Vectric's – <http://www.vectric.com/forum>



Workflow Overview of a typical CNC project

When you step back and look at a complete project from start to finish, you can identify a series of major steps that will form the “Workflow” to complete it. Having a good understanding of this process will help you start to appreciate where the different software packages and setup procedures fit into the overall creation of parts with your CNC

In this section is a list of the basic process stages with a brief description of what is involved for each one, in the following section is a more detailed description of each process stage.

1: Concept

This is the idea for what you are going to make. This may range from a specific customer requirement, something you have sketched on a napkin or a ready to go file that someone has already prepared. At this stage you need to try and think through the other processes in the job to help to get the best approach to achieving it. You should also assemble any reference material you will use to help design the part such as photos, data from the customer, design sketches etc.



2: Design (CAD – Computer Aided Design)

For the design you need create the computer data that will define either the 2D or 3D forms you want to cut on your CNC. This is done in what is typically called “CAD software” and you may also hear this type of software referred to as a drafting, drawing or design program.



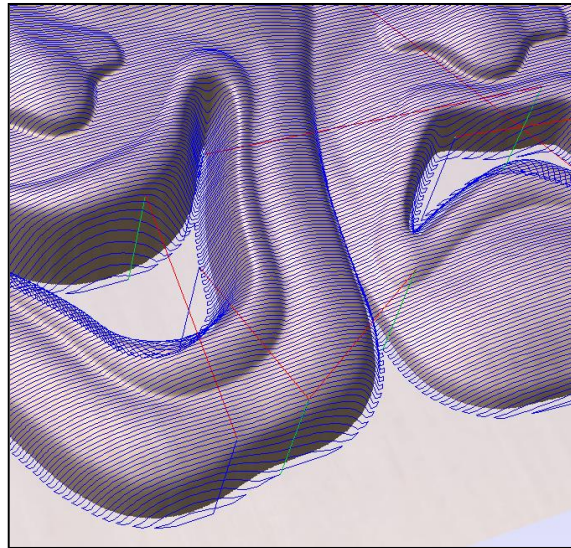
The finish point of the Design stage is to have prepared all the 2D data (Vectors) or 3D data (Components) you require to start calculating the specific movements the CNC machine will follow, these moves are typically referred to as the “Toolpaths”.

Most of our customers use one of the Vectric products (VCarve Pro or Aspire) to do their design although there are many other design (CAD) programs available for either 2D drawing or 3D modeling and depending on the file format export options available, this data can be saved and imported into the Vectric programs for Toolpath creation.



3: Toolpaths (CAM – Computer Aided Manufacturing)

Once the design is complete, you will start to calculate the actual paths that will drive where the tool will move on the machine, as previously stated these are called “Toolpaths”. Creating your Toolpaths is the key stage in going from the virtual world of a computer design to the reality of the physical world. At this point you will start to take into account the shape and size of the tool, the type of movement you want the tool to make (the shape you want it to leave in the material) and appropriate settings for how fast the tool can be moved and how much material can be removed safely.



Once the Toolpaths have been calculated the software will let you Preview how they will look in a virtual piece of material. This lets you check that they are doing what you expected. Once you are happy the Toolpaths are correct then they can be saved in a format that is appropriate for your particular CNC.

All Vectric software can be used to calculate toolpaths to drive a CNC. Each product is designed to work with different types of data depending what you plan to make. Visit www.vectric.com for information on the different products and their specific functionality.

4: Machining

Once your toolpaths have been saved then you transfer them over to the CNC. At this stage you need to set the CNC to match the job setup you specified in the Design/Machining software. This will involve setting up your material in the right orientation, and making sure it will be secure while you're cutting it. Then you need to load the correct tool and tell the machine where the X, Y and Z reference position is for the tool tip (normally this is the zero position for each axis), again this will be to replicate how it was set in the software so all the positions and sizes you specified in the software will be replicated at the machine.



Once the machine is setup correctly, the toolpath can be loaded and then executed from the CNC's control software. The machine will feed the co-ordinates of the toolpath to the machine to continually move its position and create the cuts you setup in your toolpaths. Running a toolpath may take less than a minute or potentially many hours depending on what type of operation it is. Once it is complete, you can run additional toolpaths and if required change the tool and reset the Z zero datum position for the new tool. Once you have run all your toolpaths you remove the material from the machine.

5: Finish and Assembly

Finish and assembly are obviously going to vary dramatically depending on the type of job you are doing and the material you are cutting. We will not cover this in detail in this document but it is important to be aware of the finish you plan to use and where applicable use appropriate options in the software or on the machine to help minimize or aid with your finishing process.

Tip: As you understand more about the different stages of the process, then it's good to get into the habit of thinking forward through all the steps as you plan and work through your project. For example, while in the design phase, think about the assembly and ask yourself if a minor change now can save time or make that stage easier to achieve.



Detailed Workflow of a typical CNC project

Step 1: Concept

A project can have many potential starting points that will determine how much preparation (time) is involved to get it ready for the machine. In some cases you may start with a completely new idea or customer requirement and complete all the steps from the initial idea and drawing to successfully end up with a product that fulfills its intended purpose. At the other end of the spectrum, you may download a part which has already been fully designed. This may be a 3D model (www.vectorart3d.com) or something like the free Project of the Month offered by Vectric (<http://www.vectric.com/WebSite/Vectric/projects/projects.htm>) that would be ready to setup for your machine and cut. The middle ground may be a job where the customer has the design as a computer drawing (CAD) and you need to adjust and prepare the data for machining.



For any project, you will need to analyze and determine a few important factors early on. One is the size of the project versus the size capabilities of your CNC machine. Generally you will cut parts that fit into the work area of your machine/material. There are techniques that can be used if the project is larger than your machine's capabilities, these are advanced topics and beyond the purpose of this guide but you can find out more by watching the tutorial videos on Tiling on the Vectric website.

Another important decision involves what material you are going to be using. Considerations include where the project will be used (interior or exterior), likely wear and tear for its intended use, material availability, machine capabilities and finishing options for that material etc. Knowing what material you plan to use at the concept stage can be essential for the design layout, for example having the

exact thickness measurement may be essential to allow you to draw accurate shapes for creating joints that would slot together.

At the concept stage it can also be helpful to know what tooling you plan to use or have available, as that may also factor into how the part is drawn. For example, you may need to allow for the tool radius when creating joints or cutting out intricate shapes.

Step 2: Design (CAD)

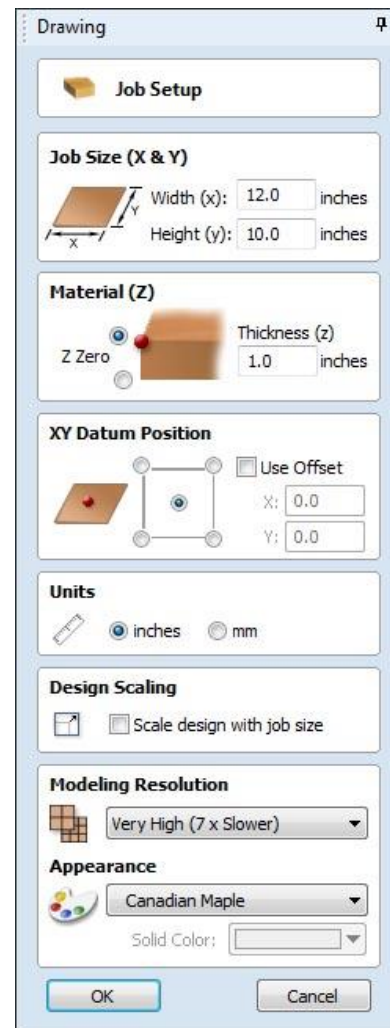
At this stage you need to create either a 2D design (Vectors), a 3D computer model (Components), or a combination of the two. This data will be used to calculate the paths the CNC will follow to actually cut the finished parts.

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.

Tip: Even though we have powerful computers available, don't hesitate to first sit with pencil and paper and create a few sketches, with dimensions and notes. You will find this can be time well spent before jumping into working on the computer.

The part is designed in a virtual area which is setup to represent the size of the piece of material you plan to machine it into (see image to right). This allows you to reference the correct size and position that the part will be cut in as you complete the layout. This includes the material size in Height, Width and Thickness (each of which relates to the X, Y and Z axis), where on this material will you set your machine's X and Y axes to zero (referred to as the XY Origin or Datum) and where you want to set the Z axis zero position for the tip of your router bit (tool). The size and location of all these points will be replicated when you set up the CNC, and this is how you relate the virtual positions in your software to the real positions on your CNC. At



The 'Drawing' dialog box is shown with the following settings:

- Job Setup**
- Job Size (X & Y)**: Width (x): 12.0 inches, Height (y): 10.0 inches
- Material (Z)**: Thickness (z): 1.0 inches
- XY Datum Position**: X: 0.0, Y: 0.0, Use Offset: ☐
- Units**: ☒ inches, ☐ mm
- Design Scaling**: ☐ Scale design with job size
- Modeling Resolution**: Very High (7 x Slower)
- Appearance**: Canadian Maple, Solid Color:
- Buttons: OK, Cancel

this setup stage you will also pick the measurement units, either metric or imperial than you intend to use to do your design.

As the design evolves, these values can be changed if required and only needs to be finally set before you calculate the actual toolpaths. It will be important to make any final adjustments before calculating the toolpaths.

There are many considerations as you choose where to set the XY Origin and Z Zero points. 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:

XY Origin: The programs give you 5 possible choices for this: The center of the material or one of the 4 corners. Most projects tend to use either the center 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 center, but will then change it to the lower left before calculating the toolpaths.

Z Zero: The program gives you a choice between the top of the material or the bottom of the material. 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, you will actually 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.

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.

After you have finished making the initial settings for the new project setup, there are three general ways to approach the rest of the design work:

First approach: Complete all the design work using the tools and features contained in the program you are using. If you are new to the design process and don't know how to use any other CAD programs, this approach offers the benefits of only needing to purchase and learn one program plus being able to complete the entire project using only one program. For the Vectric Programs, the tutorials and User Forum can help you learn this process and start to apply it to your own concepts.

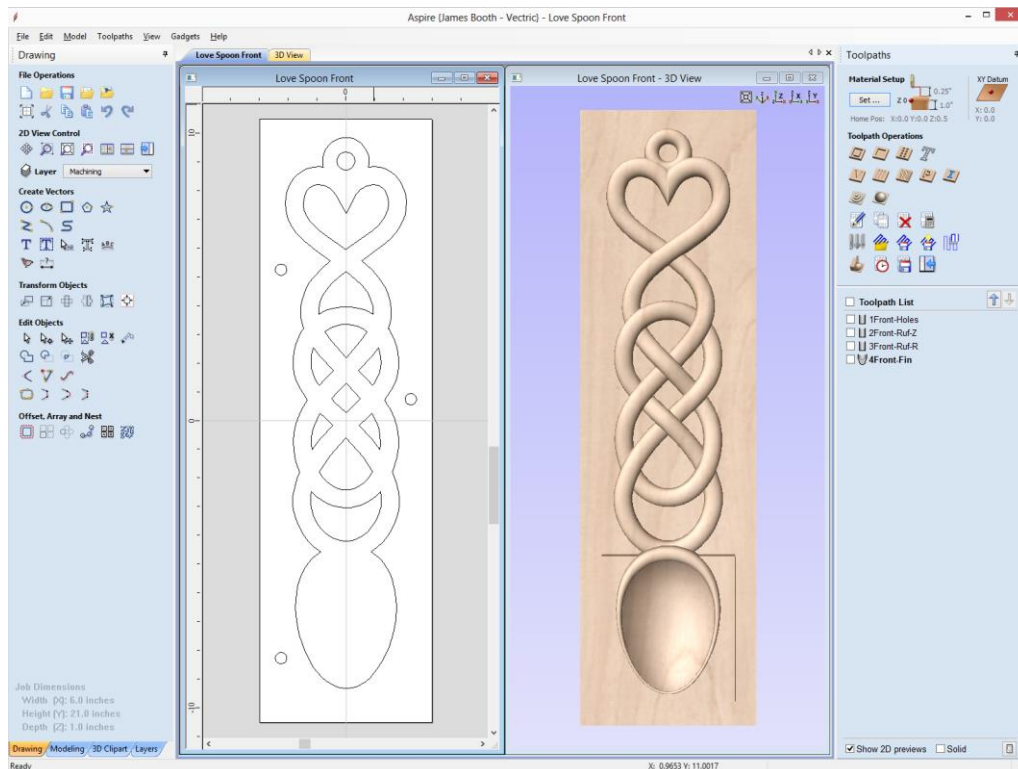
Second approach: Complete all the design work in another design (CAD) program that you are familiar with and import the design into your

software in preparation for the toolpath (CAM) process. The Vectric programs offer different options to open many industry standard 2D and 3D file formats depending on what the specific software is designed to do (example: dxf, dwg, eps, ai, stl, obj and many more).

Third approach: This approach is simply a combination of the above two. Here are a couple of examples:

- You may start with a design created in another CAD program, but want to make changes or additions. Instead of needing to return to the other program, make the changes and perform the export/import operation again, simply make the changes using the tools included in the Vectric program you are using.
- You may purchase a 2D or 3D design or piece of clipart from a third party vendor and want to include that in your project. You can simply open this or import the data to add it to your project, resize and position it and then create text, borders, etc., to complete your design.

Tip: While designing a new project, it is a good practice to look ahead to the remaining steps and ensure what you are designing now can easily be machined and assembled later on. For example, you may want to add screw holes or simply “dimple” hole locations with a small diameter bit for later finishing with a hand drill or drill press.

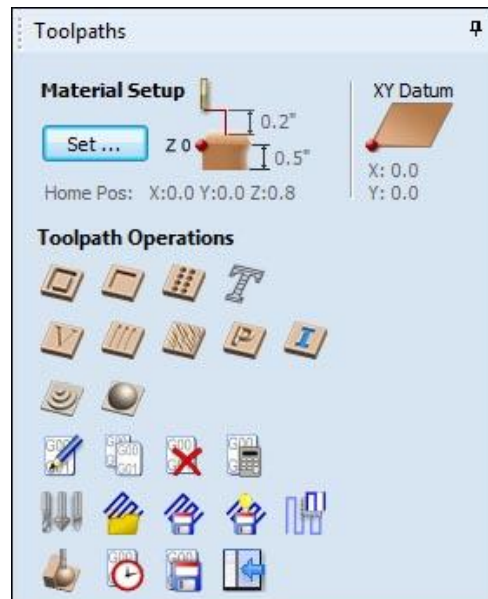


Step 3: Toolpaths (CAM)

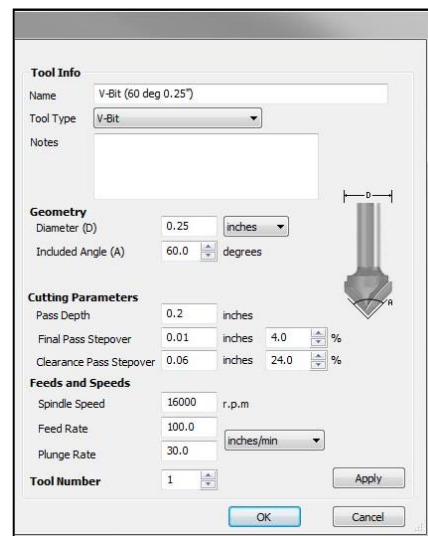
The first thing you need to do before you calculate any toolpaths is confirm your material setup and position, relative to how it will be set up on the machine itself. Although you will have previously made some choices on this when you setup the job for the design layout, you will need to double check 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. **Select the type of toolpath:** Profile, VCarve, Pocket, etc. 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. Each one has different applications, there are many examples within the video tutorials that cover all the main uses and some specialized functions.



2. **Select the correct 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

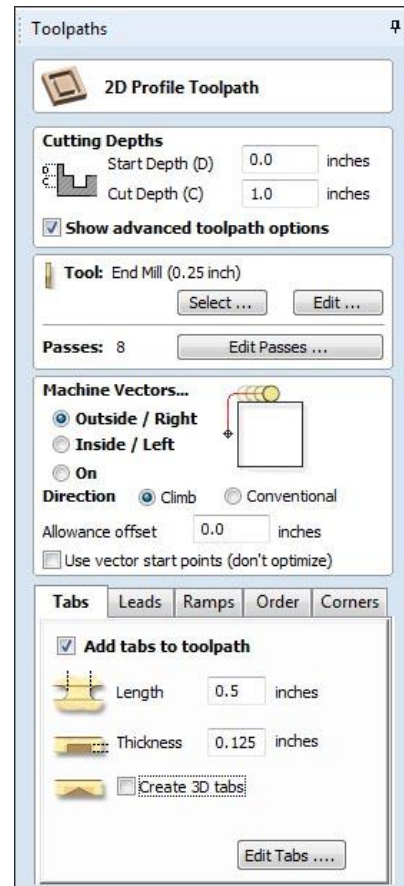


and deleted to personalize this list to your specific needs. Over time this makes toolpath setup quicker and more accurate.

3. **Choose any modifying options for that specific toolpath:** 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.



4. **Calculate the toolpath and then 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. See over for an example of a colored toolpath preview generates using Vectric’s VCarve Pro software.



Tip: You can learn a lot by reviewing the free Monthly Projects offered by Vectric, even if you don't intend to make one. Look at the design and the type of toolpath and bit selection made to complete it.

(<http://www.vectric.com/WebSite/Vectric/projects/projects.htm>)

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.

When you save your Toolpaths, you will be required to select what is referred to as a Post Processor from a list and this will determine the format that the Toolpath data will be saved in. You need to select the appropriate Post Processor from the list that matches your specific machine or the control software you use to drive your machine. This will create a file that has all the setup information and XY and Z coordinates for the toolpath formatted in a way that your CNC can understand and execute them as instructions to move. Typically the machine tool vendor can tell you which Post Processor you need to use or you can contact Vectric Support to ask. Once you have selected the right Post Processor, the software will remember your choice so you only need to do this once.

Next you work through the list of toolpaths and save them as separate files, one for each toolpath. You may also choose to append multiple toolpaths together that use the same tool shape and size and save them as one file. A further option if your CNC has an Automatic Tool Changer (ATC) is to setup numbers for each tool and save all the toolpaths for the whole job as a single file. However it should be noted that an ATC is rare and is normally a relatively high-cost option for most CNC Routers.

Once the toolpath files have been saved to your hard-drive, you can copy them onto a portable disk, memory stick or copy them across a network to load them into your CNC's control PC.

Note: Some brands of CNC machines can use different processes to load toolpaths. Contact your machine tool vendor if you're not sure how to move the saved files to your CNC to run them.

Step 4: Machining

There are a few critical steps that need to be completed at your machine before the actual cutting can take place. How you complete these steps depends on the CNC machine and its control system and you will need to know the specific steps for your system.

In general, the main steps you will complete at your machine include:

1. Transfer the toolpaths onto the computer or controller that operates your CNC machine.

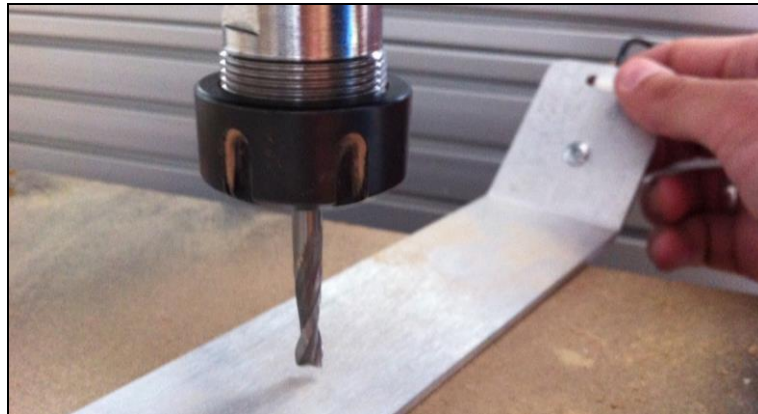
2. Position the material on the machine bed and fasten it securely in place using a variety of possible methods including vacuum, double-sided tape, clamps, screws, cams or wedges and so on. Machines with a dedicated vacuum table will probably always use vacuum except for very unique cuts.

Machines without a vacuum table will require one or more of the other methods. Remember, to get the best possible results, the material must not move while running the toolpaths, and any parts that are being cut-out need to be held in place along with the original stock material.

Install the router bit correctly in the router or spindle collet. This is a critical step and 'correctly' implies the proper amount of shank in the collet, proper tightening of the collet nut, using a clean and well maintained collet, etc. Also, it is very important for the router bit to be high quality (meaning it will endure the rigors of CNC operation) and sharp. More information on best practice for this will be available from quality tooling suppliers.



3. After installing the router bit, it must be correctly positioned to agree with the setup as you defined it in the CAM program or else the toolpaths will not cut in the right location on your material. There are two distinct settings to be made and exactly how will depend on your CNC machine and its control system:
 - a. The center of the tool set to the correct location for X zero and Y zero (XY Origin/Datum). This can vary, but is normally the lower left corner or the center of the material. Typically this is done by manually moving the bit to the position required and then telling the control software that the bit is now at the position you want to be $X = \text{zero}$ and $Y = \text{zero}$.
 - b. The bottom of the bit set to Z Zero at either the Machine Bed or Top of the Material. This may need to be done manually by moving the tool and telling the control software that is $Z = \text{zero}$. Some CNC machines have an automated Z setup such as the one shown in the image below that uses a metal plate to automatically set the zero position for the machine.



Important: For a project that will require multiple toolpaths, once you set the XY Origin point, you will NOT change that again for that job. This ensures all separate toolpaths are referenced to the same XY position and all cut relative to the same point on the material.

After each bit change you DO need to re-set the Z-zero for the tool so it is adjusted to match the new tool tip position(different tools will protrude different amounts from the collet).

Tip: It is always good to pick one point on your material or table and set Z Zero each time at the same location. This is especially important when you have designed the project with Z Zero set to the top of the material. Be sure to pick a point that will not be machined away as you run through the toolpaths, so you can continue to set each router bit to the identical Z-height.

4. If your router or spindle has a manual speed setting on it (typically specified in Revolutions per Minute or RPM), set it to an appropriate speed. The Router/Spindle may be switched on and the speed controlled automatically in which case this step would not be necessary.
5. Once you have everything set, it is very good practice to train yourself to pause and double check everything. The few seconds this takes is a small price to pay for not breaking a bit, ruining your project or otherwise damaging your machine. At this point, having a few hand-made notes about the job and its setup or a print-out from the design software can be very helpful to avoid you not remembering everything about the job setup.
6. When you have mentally double checked your setup, that the material is secure, you have loaded the right tool etc. then it is time to start your router or spindle, switch on the dust collection system, and select the toolpath you want to run within your control software or on the machine controller and start the cut.



Important Safety Information: CNC work has the potential for both short term and long term safety issues.

The potential short term issues are obvious: A powerful machine not currently under your direct control moving a sharp cutting edge spinning at 1000's of revolutions per minute – if something goes wrong it can happen fast. Always protect yourself by wearing appropriate safety glasses and following accepted rules for working around rotating equipment.

The long term safety issues may not be as obvious and the two big ones are hearing loss and respiratory issues. Always wear appropriate hearing

protection and protect your lungs from the large amounts of chips and dust that are typically produced. It is highly recommended you use some method of dust control and collection, whether it is a “point of cut” type system or enclosure with adequate ventilation and filtering. ***Remember, it is the airborne particles that your eye cannot see which are the most dangerous.***

Step 5: Finishing and Assembly

For completeness, there is normally more to do once all the toolpaths have been run in the form of finishing and assembly. While not the purpose of this guide, projects made from wood can possibly need sanding, staining or painting, and a final finish.

There are many good sources of information regarding how to finish wood, including books, magazines, websites and user forums. At this point it really doesn't matter how the wood has been cut and machined so don't think you need a source of finishing information that is related to the CNC process.



The same would apply to other applications from Sign Making to cutting part for Radio Control Models and all else in between. As has been previously stated, understanding the finishing process and thinking about it at the concept stage of your job is a really useful way to help you make design and machining decisions that may help you to reduce how much work you are left to do once the part has come off the machine.

Summary

As a beginner, this process may seem very complex. Much of it takes longer to describe than to actually do it. For instance the setup of your machine will be repeating essentially the same process each time you want to run a toolpath. As such you will become quicker as you gain more experience with this side of the operation and will develop methods and equipment to help make it more efficient.

Typically the most time consuming part of running a CNC (other than letting a toolpath run – which does not require user input) is the design and toolpath calculation software. At Vectric this is our specialty. All our software is designed to offer a full featured solution to different requirements and applications. We focus on making it as easy to use as possible and providing you with the backup to help you succeed (more on that below). We want all our customers to succeed but also know that is not just a case of education but also having a product which has suitable functionality for what you want to do with it. As such, all our programs can be downloaded as a FREE Trial version – this is not time limited and has all the design and toolpath functionality (including the 3D preview) available to use within each program to learn and practice with. The trial version will not let you save toolpaths for your own designs, but Vectric do provide sample files for which you can save the toolpaths and drive your CNC. So if you're looking to learn more about the process, evaluate how easy our software is to use and see what it will let you do then the next logical step would be to download the Trial as part of the next stage of your CNC education. In addition to allowing you to learn the design and machining part of the process if you do decide to purchase the full version then any files you have created in the Trial version can be opened and the toolpaths saved with a licensed version installed on the same PC.

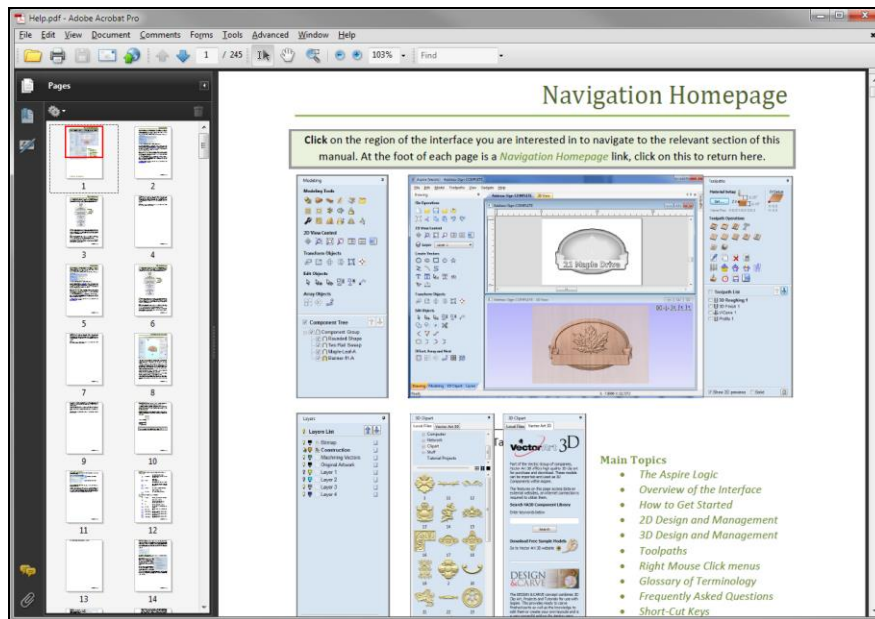


In the next section you will find information on getting Help with the software and furthering your CNC learning experience. In the last section of this document is a Glossary of useful terms and concepts related to CNC and the software which will help as you learn more about this exciting new world!

Help for Vectric Programs

All Vectric programs, including the free Trial Versions, have Help available which can be quickly accessed by pressing the F1 key or using the Main Menu and selecting Help > Help Contents.

The Help document will open to the Navigation Homepage where you will find brief instructions on how to use it, and graphics showing the computer screens for that program. To find information about a particular command, you would click on the graphics area of interest which will take you to that area in the document. For some commands, you may have to click again on the specific icon image to get to the exact information you are looking for. In addition, using Ctrl-F while the help file is open will open a search box where you can enter search terms.



In addition to the Help Contents, a complete Reference Manual is also provided with Aspire, VCarve Pro and Cut2D. It contains the same information as the Help Contents described above, but is organized more to be used as a reference document or printed if desired. This can be accessed from your PC's hard-disk.

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).

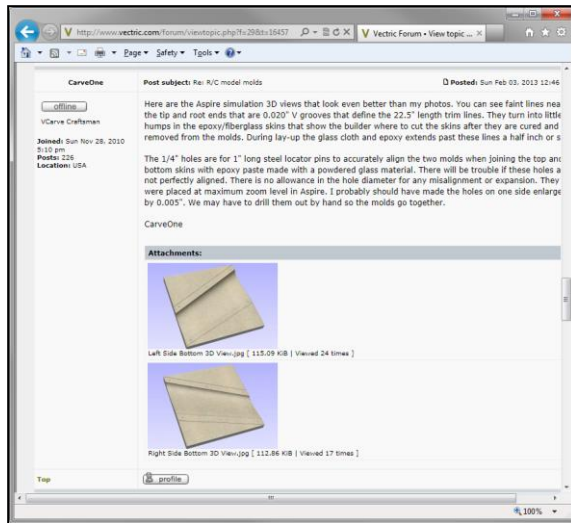
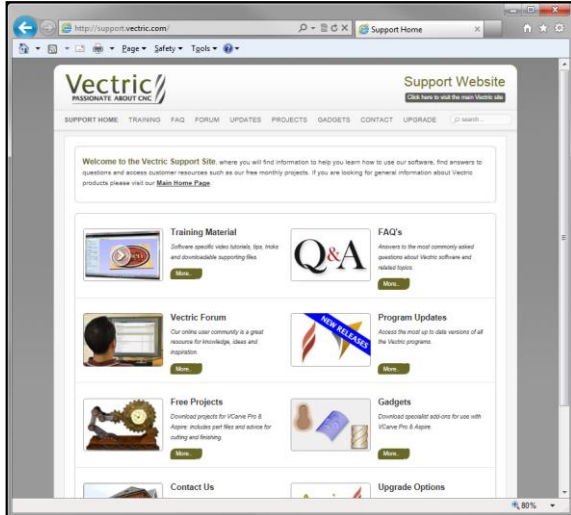
	1 - 2D View Overview Introduces objects found in the 2D view and how they are manipulated (8 minutes, 7Mb) DOWNLOAD iPad / iPhone
	2 - 3D View Overview Introduces objects found in the 3D view and how they are manipulated (8 minutes, 9Mb) DOWNLOAD iPad / iPhone

In addition to the website tutorials, Aspire owners will receive a DVD with a significant number of additional videos tutorials and 3D clip-art files.

Besides the tutorials, the Support Website (<http://support.vec tric.com>) also provides you with a number of other important resources including links to the Monthly Projects, FAQs (Frequently Asked Questions), Program Upgrade information and the User Forum. It is a good website to bookmark for quick reference.

Some of the best help can come from other users and the Vec tric User Forum is excellent for this. It is a good place to learn and ask questions that haven't been answered by the Help documents or tutorials, or to seek advice. If you have not visited it yet, you are encouraged to do so.

If you have questions or concerns that need to be addressed directly by Vec tric, the best approach is to use the support@vec tric.com email service. When received, the Vec tric support staff will respond and you can typically expect a response no later than the next business day.



Help for your CNC machine and its control software

If you need help with your machine or the software system that controls it, it is best to start with the machine's manufacturer or supplier. How you would do this depends on your machine and control system and can include direct contact (phone or email), manufacturer's or distributor's website or a User Forum. You can also find websites and CNC related forums that discuss many different subjects and manufacturers.

Important CNC Concepts and Terminology (Glossary)

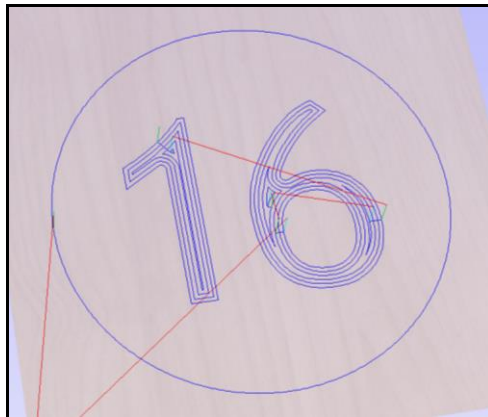
As with most new ventures into another profession or hobby, there is the need to learn a “new language” and its common abbreviations. We have introduced some of these in the description of the workflow process above. Here we’ll look at some in more detail. These are listed in alphabetical order.

2D/2.5D/3D Toolpaths

Depending on the information source, the definitions of 2D, 2.5D and 3D can be quite vague and appear to overlap. With regard to the Vectric we would typically use these to describe types of toolpath within the software as follows:

2D is simple horizontal cuts at a constant Z depth. For example, you cut out letter or a plaque shape to hang on the wall by moving the tool down to a specific depth and tracing the shape of the part you are cutting in 2D.

Examples of 2D toolpaths are; Profile, Pocket, Drill, Quick Engrave and Inlay. The image below right shows a pocket toolpath (the 16) and a profile toolpaths (circle). Below right shows the preview result of the part that these would cut.



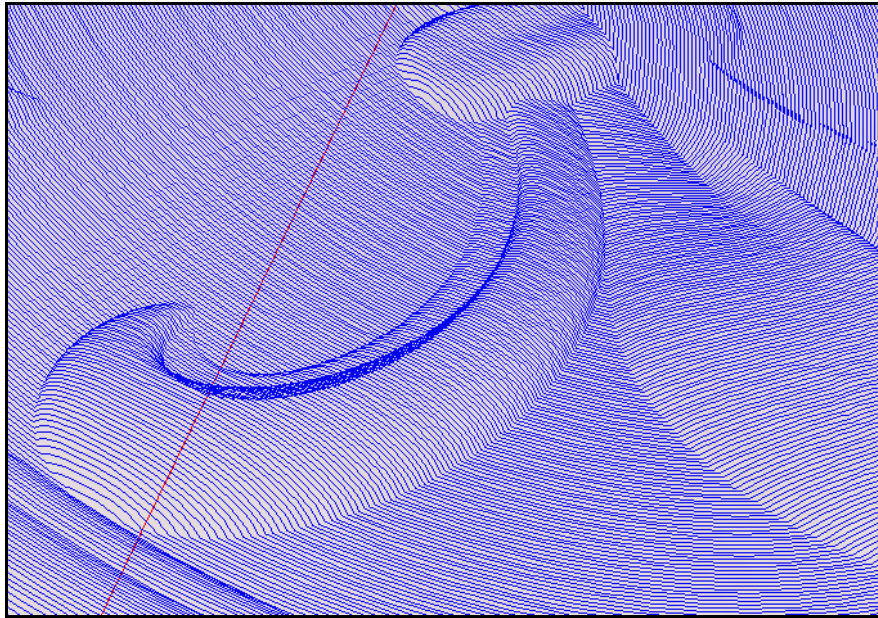
2.5D would be a little more complex than 2D as while the tool is cutting, the Z depth would be changing instead of staying constant. Typically this still uses 2D vector data to define the path but the way the tool moves in relation to the 2D data and its form will determine the shape that is cut. The most common example of 2.5D is VCarving (see image to right). Although this cuts 2D vector shapes, the V-shaped tool “rides” on the edges of the vectors, so as it gets wider it cuts deeper and as the vector get closer together it will lift up in Z.

Examples of 2.5D toolpaths are; VCarve, Fluting, Texture and Prism Carving.



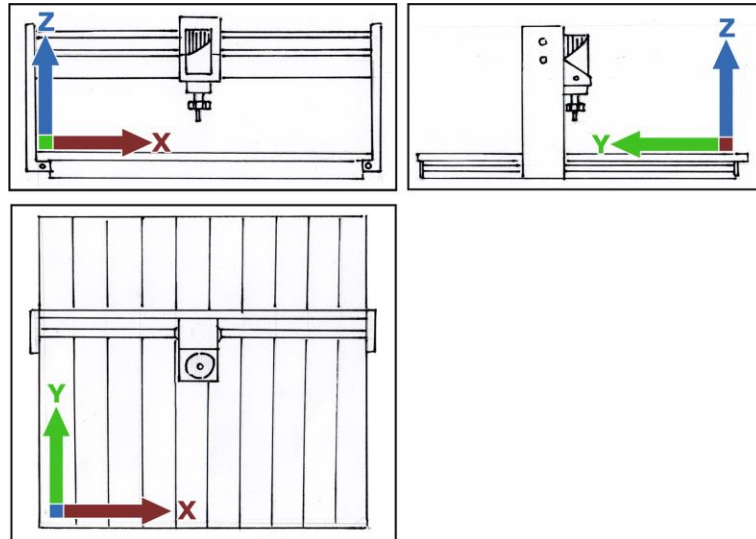
3D These are toolpaths that like 2.5D toolpaths can move simultaneously in all 3 (X, Y and Z) axes however the shape they cut is described by following a 3D computer model and not vectors. They are typically cut with a Ball-Nose (round ended) tool. The tool will go back and forth across the 3D model following the contours of the part similar to an old fashioned copying or digitizing machine. The smoothness of the finished part depends on the Stepover setting in the toolpath as that determines how many times the tool will pass over the model. The size of the tool is also important as the software will only let it cut the detail that the end of the tool can fit into as it traces over the virtual surface.

Examples of 3D toolpaths are; 3D Rouging and 3D Finishing



3-axis

A 3-axis CNC machine is one that can move a router bit in the 3 primary directions, X, Y and Z. The image below show 3 views of a typical CNC setup and how the axes are referenced from each viewing direction.

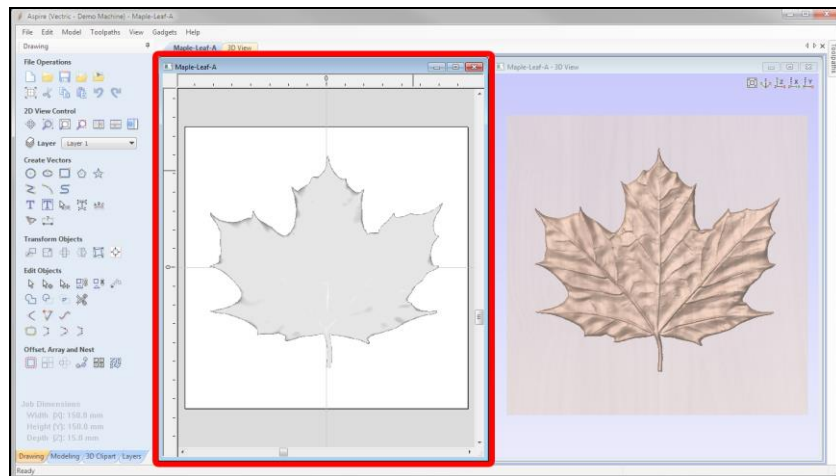


4-axis

See Rotary Axis

2D View

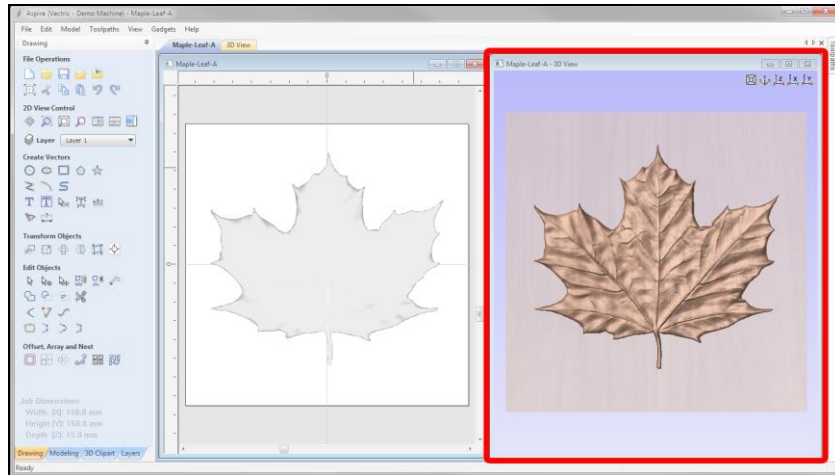
A window in Cut2D, VCarve Pro and Aspire used to display the Vectors, Bitmaps, Component Grayscales and 2D Toolpath Previews. Predominantly used for 2D design and selection of object to create toolpaths (or 3D models in Aspire).



2D View shown highlighted in red on the left of screenshot above

3D View

The Window in Cut2D, VCarve Pro and Aspire used to display the toolpaths and to view the 3D Toolpath Preview virtual block. In Aspire it is also used for visualizing the 3D model/Components.

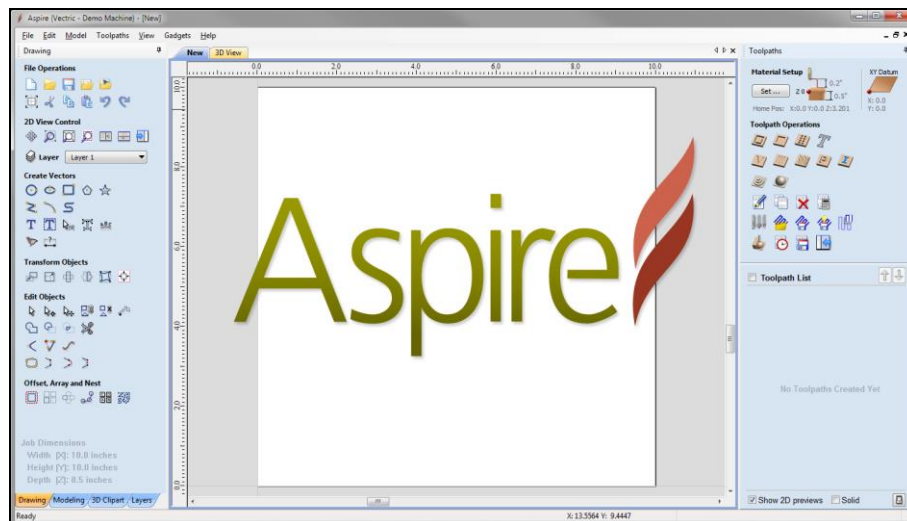


3D View shown highlighted in red on the right of screenshot above

Aspire

Aspire provides all the functionality and ease of use of VCarve Pro but adds the ability to create 3D relief models from 2D data. Shapes can be constructed, edited and joined to build your final design as well as having the ability to import and modify existing 3d data and clipart. Aspire is an elegant solution to the complex problem of design and machining 2D, 2.5D and 3D parts with a CNC.

See **VCarve Pro** for more information on that software.



ATC/MTC

ATC stands for “Automatic Tool Changer” while MTC stands for “Manual Tool Change” and both describe techniques where the process of changing a router bit can be fully or partially automated.

An ATC system is fully automated in that a number of router bits are prepared and placed in a location where the CNC machine can move the spindle to access them. When one of those bits are needed for the next toolpath, the CNC machine will automatically return the previously used bit to its location and then select the next bit (as indicated by the post-processed file) and run the next toolpath. There are many configurations for this type of setup that involve mounting the tools on a carousel near the spindle or having them in a separate location at one end or the side of the machine bed. All the tools on an ATC machine have to have their Z-zero positions set by the operator before the machine is used to cut a part.

With a typical MTC system, the actual router bit change must be done by an operator but the software can automate the process by moving to a predetermined area, stopping the spindle and waiting for a command from the operator to continue. After receiving the command to continue, it will set the Z Zero reference point for the router bit and then continue to run the next toolpath.

Bas-Relief

This is type of sculpture or carving where the subject of the model is proportionally much lower than the actual item but still gives the impression of the objects contours and shape. A good example of this style of modeling is the design on the back of most coins where a complex scene is usually rendered with very shallow depths or a sculpture such as the one shown in the image. “Relief” is a good description for the typical type of parts that the 3D modeling and 3D machining part of Aspire is used to create.



Vector Art 3D (www.vectorart3d.com) is a good source of bas-relief models that can be used with a number of the Vectric software programs to carve 3D parts with a CNC.



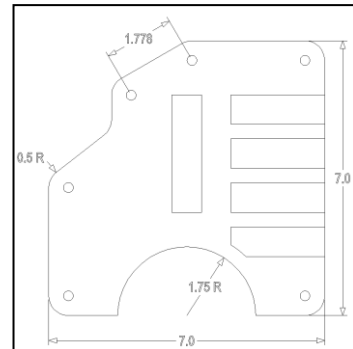
Bitmap

A bitmap is an image file format made up of many small squares called 'pixels', where each pixel can be assigned a color. A good example of a bitmap is the picture from a digital camera and saved in a common file format such as .jpg. Some Vectric software can import a "Bitmap" and allows the user to automatically trace colors from the image with a vector (VCarve Pro and Aspire). This does generally require very high quality image to get good results. A bitmap can also be used as a template to manually trace a vector over within the same programs. Aspire can use a bitmap to generate a 3D texture automatically assigning proportional heights to the light and dark areas of the image. The lighter the color the higher that part of the model is and vice-versa. PhotoVCarve is a specialist product which is designed to do this on a standalone basis. There are detailed tutorials covering all uses of image files within the appropriate programs.



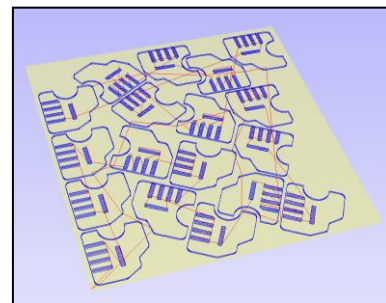
CAD

CAD has been used as an abbreviation for both "Computer Aided Design" and "Computer Aided Drafting". Now it has become a generic term in its own right that refers to a software program that can be used to create a 2D or 3D design.



CAM

When you discuss designing and preparing projects for your CNC machine, the follow-on side of CAD is CAM, which stands for 'Computer Aided Manufacturing' or 'Computer Aided Machining'. This is the part of the process where the geometry developed during the CAD design process are now used to create the instructions that will move your CNC machine to 'manufacture' or machine the part.



CAD/CAM

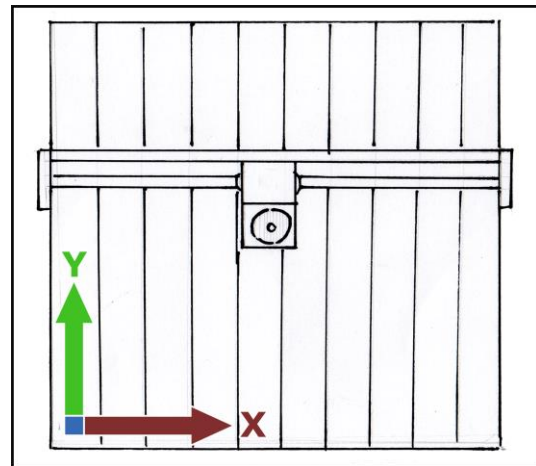
It is important to understand that some programs exist to do only CAD or CAM, while other programs have the ability to perform both functions and these are referred to as CAD/CAM programs. Aspire and VCarve Pro are both considered CAD/CAM programs since you can both design and create toolpaths in them. On the other hand, Cut2D, Cut3D and PhotoVCarve are considered CAM programs because the CAD (design) part has been completed in another program (or process) and these programs only create the toolpaths so the design can be machined. However, it should be noted that Cut2D does contain some rudimentary editing tools for doing some basic CAD operations to help fix data or add simple text to an imported design.

Cartesian Coordinates

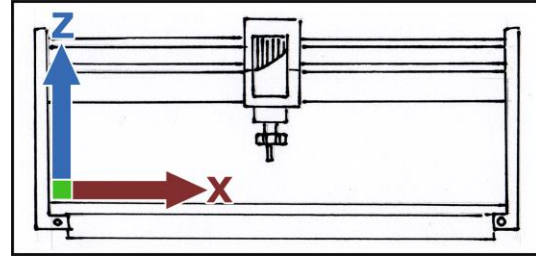
In order to design a project in a virtual space in one program and run it from another, there has to be a common method to accurately describe a point or position on your CNC machine. The method chosen for our 3-axis machines is based on the “Cartesian” coordinate system and just like many of the other terms, an internet search will find additional information if you are interested in learning more about this term.

For now, let’s look at it from a simple point of view: In 3-axis machines, two coordinates are used to position the router bit horizontally anywhere on the machine bed and the third coordinate is used to position the router bit vertically.

The two horizontal coordinates are referred to as X and Y and your machine is designed with motorized X and Y axes to physically move the router bit to these coordinates when instructed. It depends on your machine which direction the manufacturer has picked for this, but they will always be at a right angle (90 degrees) from each other. Typically, movements from left to right or bottom to top are referred to as movement in the positive direction. Conversely, movements from right to left or top to bottom are referred to as movement in the negative direction. Don’t let this confuse you; it is just the way directions are referred to in the CNC world. Try to relate it to reading a road map, where one direction is East (positive) and its opposite direction is West (negative). Likewise, the same is true for North (positive) and South (negative).



The third coordinate, Z, is the vertical coordinate and your machine will have a motorized Z axis to provide this vertical movement and positioning. It is this coordinate and axis that determines the vertical position or height of the router bit within the workspace of the CNC.



One very important point to understand is every project will have a point where the X, Y and Z values are set to zero. This provides the reference points necessary for all the toolpaths to work together to properly machine the part. Once these zero reference points are set, coordinates from one direction from the zero point will be a positive number (+) while the opposite direction will be a negative number (-).

CNC

The abbreviation stands for “Computer Numerical Control”. This is used as a generic term to describe machines which are controlled by computer generated co-ordinates. It is a very wide topic with lots of sub-genres and specialization. If you are interested in learning more, then a quick internet search will find volumes of additional information.

For the purposes of this document when you see the term ‘CNC’ simply picture a system where an electronic device (computer) controls a mechanical device (CNC machine) to move a tool (router bit) to cut material to make parts.

Cut2D

Cut2D is an entry level CAM software package from Vectric which has some limited CAD ability. It allows Hobby CNC machine users to convert CAD DXF and graphic designs into CNC Toolpaths for machining parts and components. It has a very simple and intuitive workflow to let you get from the design to part as quickly as possible and is one of the most affordable and accessible ways to start cutting-out parts with your CNC from 2D data.



Cut3D

Cut3D lets you very quickly and easily create toolpaths to carve free-form 3D parts modeled in another 3D design program. It utilizes a simple wizard-based interface but includes important options to help hold the part, allow you to cut it in slices or machine from multiple sides. Cut3D is a great solution for cutting 3D on your CNC if you do not require Aspire's modeling and layout tools.



Cut3D can also be used with Cut2D or VCarve Pro, to combine the 3D with 2D or 2.5D toolpaths.

Cut Depth

See **Pass Depth**

Feed Rate and Plunge Rate

For each router bit you set up in the Tool Database, you will need to select a Feed Rate and a Plunge Rate. Feed rate is the maximum speed your router bit will move through the material when cutting in a horizontal direction. Plunge rate is the maximum speed your router bit will move through the material when cutting in a vertical or angled direction.

Note: We use the term 'maximum' here because a machine's control system will typically slow the speeds during a toolpath when reversing direction, cutting a sharp corner or edge, etc. As an example, picture safely driving your car around a sharp corner: You slow down, make the turn, and speed back up. To make a sharp direction change while cutting, your CNC machine must follow the same approach.

A screenshot of the 'Tool Info' dialog box in the software. The dialog is divided into several sections: 'Tool Info' (Name: V-Bit (60 deg 0.25"), Tool Type: V-Bit, Notes), 'Geometry' (Diameter (D): 0.25 inches, Included Angle (A): 60.0 degrees), 'Cutting Parameters' (Pass Depth: 0.2 inches, Final Pass Stepover: 0.01 inches 4.0%, Clearance Pass Stepover: 0.06 inches 24.0%), 'Feeds and Speeds' (Spindle Speed: 16000 r.p.m., Feed Rate: 100.0 inches/min, Plunge Rate: 30.0 inches/min), and 'Tool Number' (1). The 'Feed Rate' and 'Plunge Rate' fields are highlighted with a red rectangle. An 'Apply' button is at the bottom right.

Both speeds are set by the ‘distance traveled per second or minute’ and you can use either metric (mm) or imperial (inches) values for the distance amount. There is no right or wrong choice here as they can all be converted from one to another, but it is suggested to pick one and use it consistently. This will help you get used to how the numbers relate to reality and with time you can better optimize your “speeds and feeds” as your experience with your machine and the material and tooling increases.

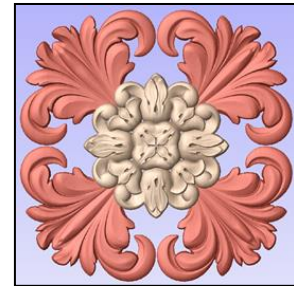
Collet Assembly

The collet assembly is the component on a router or spindle that holds the router bit securely and consists of a precision collet and a collet nut. The collet is precisely sized to hold the router bit shank and the collet nut applies a compressive, friction fit as it is properly tightened onto the router or spindle shaft. It is very important to keep the collets clean and well maintained and depending on use, they must be periodically replaced.



Components

Component is the term we use to describe a 3D object in Vectric’s Aspire software. A 3D model can consist of a single component or be made of many different Components in combination to create the complete 3D form. Components are created either by using the modeling tools within Aspire to generate different 3D shapes from Vectors or by importing an existing design saved as Clip-Art or created in another 3D design program. They can also be generated as a texture from an imported image (Bitmap).



Gadgets

A gadget is an external add-in program which performs a specific task and is a feature of Aspire and VCarve Pro. The gadget concept is intended to allow Vectric to produce simple add-ons which address specialist requirements without cluttering up the main interface. You can read more about Gadgets and see what is available for the full versions of VCarve Pro and Aspire using the link below.

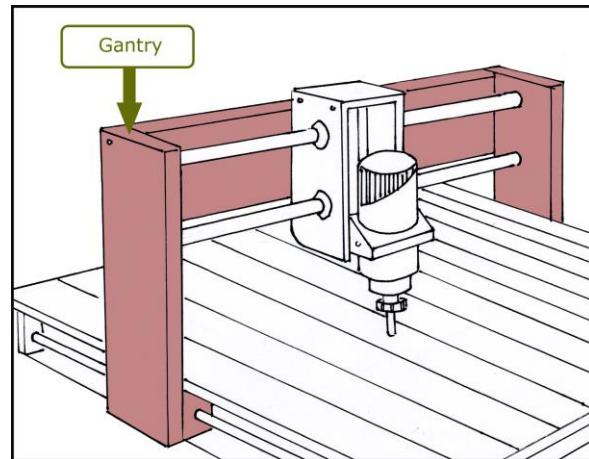
<http://www.vectric.com/WebSite/Vectric/support/gadgets.htm>



Gantry

The gantry is the part of a CNC machine that typically has one of the horizontal axes running along it and also moves along the other horizontal axis to create the X and Y movement for the tool. You can see one example of a configuration for this in the image to the right.

The Z axis is also typically mounted on the gantry.



G-Code

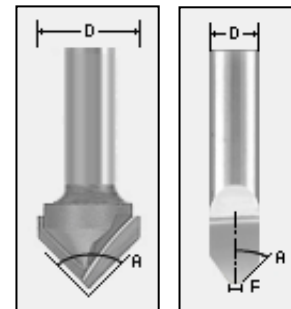
This is a generic term for the code which is sent to the machine with the instructions telling it how to move. Or, to put it in our context, the file that is saved from the software using the Post Processor to format it for the CNC machine. In reality it's a particular type of data that a lot of CNC's use a variant of – so G-code is not necessarily interchangeable to different machines. The name comes from the fact that many of the codes within in start with a G. For example G01 specifies a Feed Rate move and G00 specifies a Rapid move etc.

```
G0Z0.2000
G0X3.0250Y20.9740
G1Z-0.1250F30.0
G1X3.0249Y21.2266F100.0
X2.9730Y21.2603
X2.9182Y21.2982
X2.8646Y21.3376
```

An example of some “G-Code” from a program is shown in the image above right.

Included Angle and Side Angle

To program toolpaths using router bits with V-shaped cutting edges, such as a v-bit or engraving bit, the CAM program must accurately know the angle of the cutting edge. In the Vectric programs, this is specified using either the angle on one side of the bit or the total angle formed by the bit. For a V-bit, you would specify the Included Angle which is the total cutting angle formed by the cutting edges. An Engraving bit will be set up with a Side Angle, which is the angle formed on one side



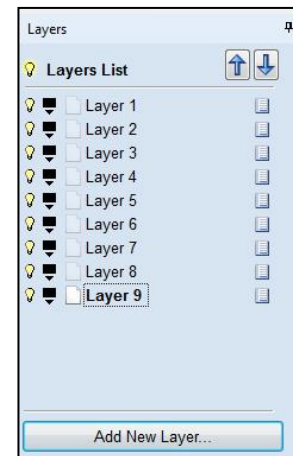
referenced to the center of the bit. Fortunately, the programs provide a graphic of the bit and show you exactly how to enter the angle measurement so you do not need to remember the details, just look at the picture.

Indexer

See **Rotary Axis**

Layers

Layers are a feature of many CAD programs and provide a method to help organize your different types of data (Vectors, Components and Bitmaps). A simple use of Layers is to use them to manage vectors as you draw a part. For instance you can assign different colors to a layer and make layers visible/invisible to simplify what can be seen in the 2D view. Layers can also be used when setting up toolpaths in VCarve Pro and Aspire to help select vectors on specific layers and this is a very powerful tool when combined with the Toolpath Template feature.



Node

See **Vector**

Pass

When you have a toolpath that is too deep, too wide or a combination of both for the router bit to remove all the material at one time, the toolpath is broken up into multiple 'Passes' with each pass removing a part of the material.

When the toolpath is too deep, a **Pass Depth** (see entry below) value for the bit will be set and used to indicate the maximum **Z** depth the tool should cut on each Pass. The amount of material a tool can remove will be determined by the tooltype, size, material being cut and many other parameters. Tooling manufacturers typically offer recommendations for these values.



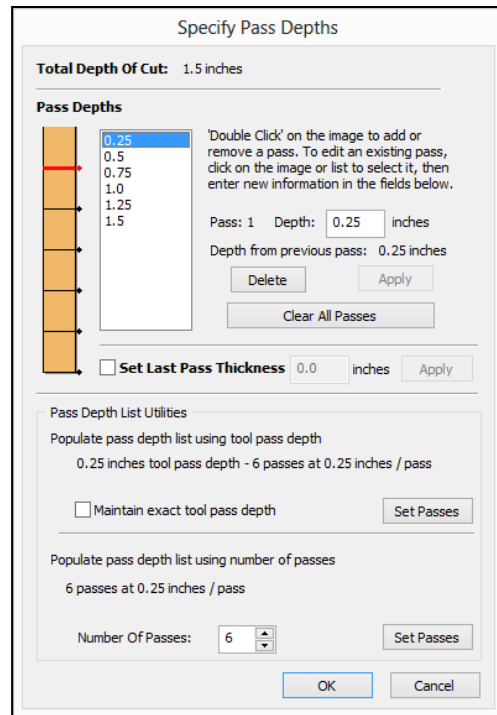
The image above shows a "Roughing" toolpath with steps created by the Pass Depth setting for the tool used.

When the area being cut is too wide to machine in a single pass then the amount the tool cuts on each pass is controlled by a setting called the "Stepover". See the **Stepover** entry in this document for information on how you set the amount of material that will be cut horizontally (in **X** and **Y**) by the tool.

Pass Depth and Cut Depth

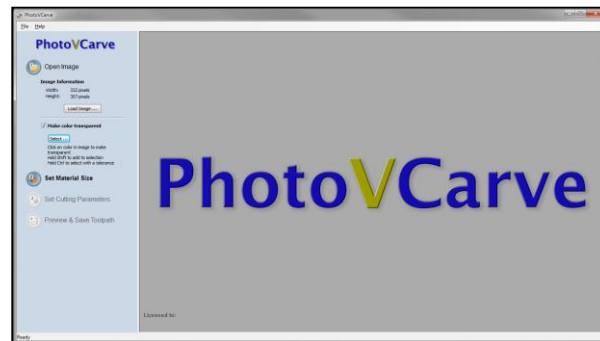
Two very important toolpath settings are the Cut Depth and Pass Depth. The Cut Depth sets how deep the toolpath will cut into the material and the Pass Depth sets how deep the router bit will go on each pass. Sometimes you can complete a toolpath with a single pass, but often you must break it up into multiple passes because of the material, the bit or your machine capabilities.

For example, you want a toolpath to cut 18mm deep, but need to limit the depth for a single pass to 6mm. So you would set your Cut Depth to 18mm and your Pass Depth to 6mm and the program will calculate 3 passes with each pass cutting 6mm deep. The first pass would cut from the material surface down to 6mm, at the end of that pass the bit would move down another 6mm and remove the material between 6 and 12mm and finally, the last pass would remove the material between 12mm and 18mm. At the completion of the toolpath, you would end up with your desired 18mm deep cut and you completed this without asking more of the bit or machine than it was capable of doing.



PhotoVCarve

PhotoVCarve is a specialist program designed to convert photographs and images into toolpaths that can be run on a CNC Router or Engraver. There are two main ways to use the program, either to create a line-etched image or a back-lit permanent picture known as a lithophane. The software can also be used to process grey-scale height maps into simple 3D toolpaths.



Post Processor

There are hundreds of different brands and models of CNC machines currently in use. When it comes to controlling them, many have a lot in common, but there

can and will be differences. So, if you look at this as each machine speaks a slightly different language, there has to be a way for a single CAD/CAM program to speak to each machine and this is accomplished by using an “interpreter” called a Post Processor, also sometimes simply referred to as ‘pp’ or ‘post’.

For example, you design a project to cut an exact size circle out of a piece of material and calculate a toolpath to do that. When you actually save that toolpath so it can be used at your CNC machine, the correct Post Processor will ensure the instructions for cutting that circle are prepared correctly so your CNC machine can read and understand them and accurately cut the circle as you intended.

Router

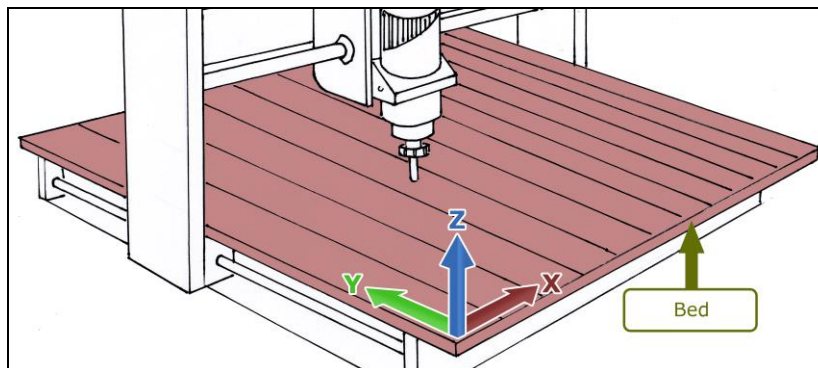
A router is a commercially available woodworking tool specifically designed for hand held or router table use. Many models are used in CNC machines as a lower cost alternative to spindles, with possible trade-offs of increased noise, less precision and increased bearing and brush maintenance. Typically, routers are more than adequate when used for hobby and light production applications.

The image to the right shows a typical configuration for a router mounted onto a CNC setup.



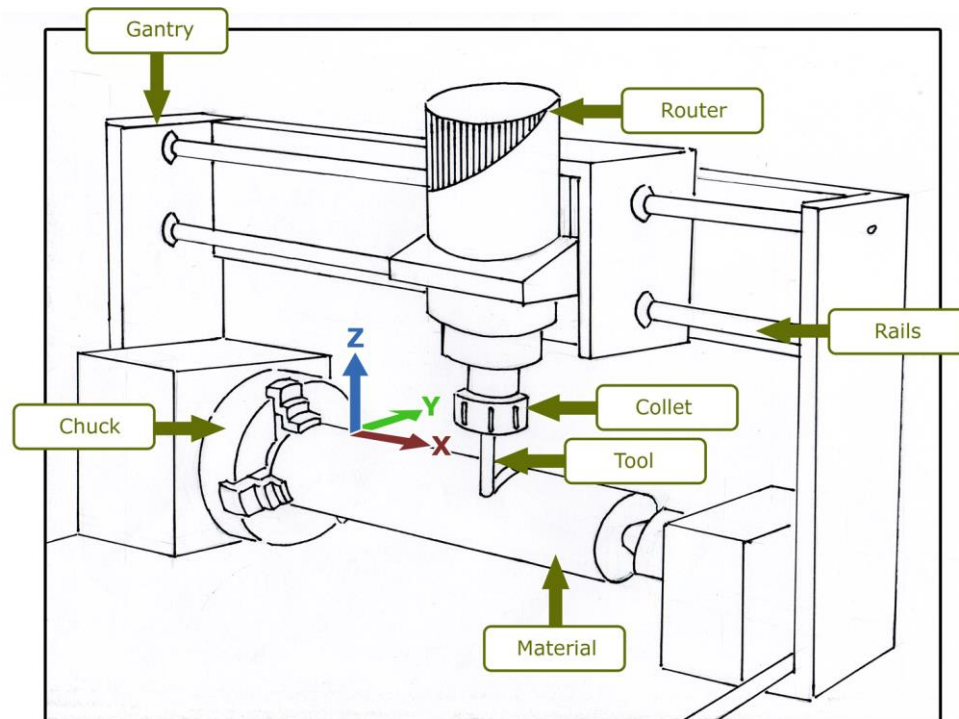
Router Bed or Machine Bed

The ‘bed’ identifies the part of a CNC machine where the material being machined is placed and held securely in position. For best results, the bed needs to be flat and sturdy. In most designs the bed is stationary; however there are exceptions to this where the bed moves to provide either the X or Y axis movement.



Rotary-Axis

Many CNC machines can be equipped with a rotating device that simulates a lathe. The machine is still a 3-axis machine, but through control software one of the horizontal axes, X or Y, is converted to rotating the stock. The other horizontal axis is now used to move the router or spindle along the centerline of the rotary attachment and Z is still used to control depth of cut.



The image above shows the key components in one possible configuration of a Rotary axis setup.

The actual name may vary between manufactures, but these can be described as rotary axis machines or Indexers. The CAD/CAM software would need to provide the functions necessary to create and toolpath a rotary project to be able to use one of these (VCarve Pro and Aspire have this capability)

Often Rotary Axis machines are misidentified as being “4-Axis”. Technically a 4-Axis machine is able to cut X, Y and Z and also rotate a cylinder as well. Whereas most Rotary machines substitute a linear axis so it becomes rotary so are still really 3-Axis machines. Vectric software cannot run a full 4-Axis machine but some of our programs can drive two linear axes and the rotary as indicated above.

Sacrifice Sheet

See **Spoilboard**

Spindle

A spindle is an industrial duty motor used to turn a router bit or cutter. Compared to a router they are typically quieter, more precise, produce more useable power over their speed range and have better bearings and longer life, but at a significantly higher cost. Many machines designed for production work come with spindles, and almost all machines can be upgraded with one if desired.

The image to the right shows a spindle mounted onto the Z axis of a CNC Router.



Spoilboard


The spoilboard is normally a sacrificial sheet of material placed on top of the machine bed and is intended to be replaced as necessary. It protects the machine bed and allows frequent surfacing which will help to keep a truly flat surface to mount material and help maintain accuracy.

The image above right shows a sheet of plywood mounted onto a sacrificial sheet of MDF allowing the stock material to be screwed down to it and the parts drilled and cut all the way through the material thickness.



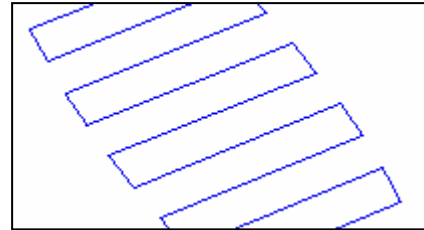
Start Depth

This is a setting available in most toolpaths and provides the ability to specify a distance below the surface of the material where you want a toolpath to start cutting. Toolpaths with a non-zero Start Depth set typically follow another toolpath that has already removed material. A good example would be the case where you have created a pocket using a pocket toolpath and now want to v-carve some text at the bottom of the pocket. By setting Start Depth to the same value as the pocket depth, the text will machine correctly. The proper use of Start Depth can also reduce machining time as less time is spent “air cutting” where material has already been removed by another toolpath.

Cutting Depths		
	Start Depth (D)	0.0 inches
	Cut Depth (C)	0.25 inches
<input type="checkbox"/> Show advanced toolpath options		

Stepover

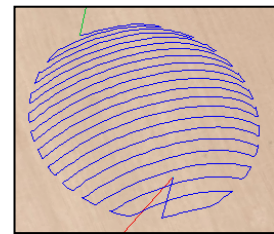
The Stepover setting found for most router bits allows you to control horizontal movement when machining areas wider than the router bit. Physically, stepover is the distance the center of the bit moves horizontally for each pass. The image (right) shows a region of toolpath and how the stepover effects how the area is filled.



The Stepover is shown in the programs as both a physical distance (0.025" or 0.5mm for example) and as a percentage of bit diameter (6%, 8%, 50%, etc.). You can enter either value and the software will automatically calculate the other one for you. Typically most users refer to Stepover by its percentage value as it is a simple whole number and can easily be translated to other tool sizes for a particular task, for example when finishing setting the Stepover to be 10% of the ballnose tools diameter normally creates a reasonably finish surface.

Toolpath

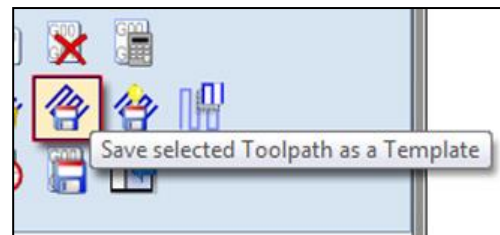
A toolpath is the path created during the CAM part of your project design and represents the instructions your CNC machine will use to make the router bit move to duplicate what you have designed. In the software it is typically displayed as a set of lines which show a virtual picture of the path the tool tip will move along when the toolpath is run on the machine.



Toolpath Templates

This is a method of saving the settings for a single or set of toolpaths defined in VCarve Pro or Aspire. These settings can then be imported into another set of data and applied to a different design. They are a good way to help automate toolpath creating for similar parts (Example:

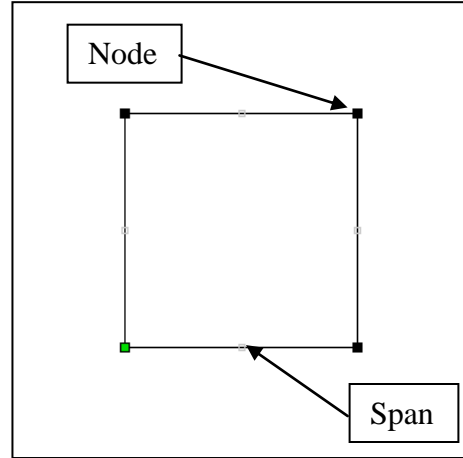
Nested Cabinets). They are particularly powerful when combined with the ability to associate a toolpath with vectors on a specific Layer.



Vector

Vector is the word used to describe a 2D line, arc or curve in the Vectric software. For Cut2D, VCarve Pro and Aspire it is the fundamental building block used for creating a 2D design and as a basis for creating 3D models that are then both used to create toolpaths that actually drive the CNC machines movement.

For example if you want to cut out a square this would be described in the software by a single vector made up of four straight equal length vector “Spans” which would be joined between 4 “Nodes”, one at each corner. The Spans are the actual shape a vector follows and the Nodes control the points that these spans will pass through. Once you have a Vector drawing, this can be used to calculate a Profile Toolpath that can run the tool on the “Outside” or “Inside” of the shape or run the center of the tool “On” the line. Cutting outside the vector would cut the vector shape out, cutting inside would leave a square hole in your material the size of the vector or cutting on would let you scribe the shape, perhaps with a pointed cutter.

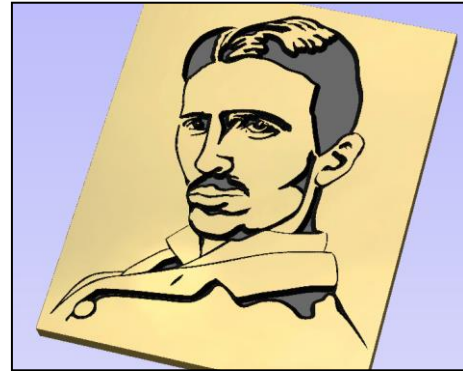


A vector can be used to describe almost any 2D shape. As well as straight vector spans as described in the square example above, in the Vectric products there are two other types of vector span: the arc and a special type of curve called the Bezier curve. Each of these has additional special types of nodes to help you control their shapes, these are typically referred to as control handles. Each of these lets you draw a different type of shape and a vector may be made of one or more spans joined together.



The image above left shows vectors representing a more complex shape, above right is a close up which shows the nodes and spans of one of the vectors.

As described above all toolpaths are either derived from a vector directly (in the case of 2D or 2.5D toolpaths) or are based on 3D shapes that are themselves created from vectors. So it cannot be overstated how important it is to understand either how to create vector data in another software program to import into the Vectric software or to learn how to draw and edit vectors within VCarve Pro or Aspire yourself.



Vectric Ltd.

Vectric was founded in 2005 by Brian Moran and Tony McKenzie; they had previously managed the specialist creative modeling and CNC machining software product range of a large UK CAD/CAM company. In forming Vectric, they applied more than 30 years of combined experience to create an entirely new range of software products that are quick and easy to learn, precise, self-supporting and, importantly, affordably priced. Vectric has seen its products become very popular with growth in both sales and personnel every year since its inception. Website: <http://www.vectric.com>



VCarve Pro

Built on the same interface as Vectric's Cut2D software, VCarve Pro is a complete solution for advanced 2D and 2.5D CNC routing. It has a comprehensive set of 2D vector design tools and large variety of toolpath strategies including profiling, pocketing, drilling, v-carving, texturing and many more. VCarve Pro's combination of affordable power and simplicity makes it our most popular program.

