Introduction to CNC for a Total Novice

Part of a series by Graham Bland

Bits

Contents

Changes from V1.0

Greatly expanded the ER11 collet description and include details on how to fit and change a collet.

Introduction

I have been playing around for a while but decided it was time for me to learn about Bits and finally understand what the terms actually mean and how they relate to each other.

This won't tell you what speed and feed settings to use to cut, but will describe and explain all the information your CAM software asks for and hopefully explains what it all means.

This guide is meant for users of small CNC routers typically 3018's but may be of use to others.

The maximum no load spindle speed for my 3018 PRO is ~8,250 RPM, for the 3018 PROVer it is ~9,250 RPM, using the updated GS 775M motor will give ~20,000 RPM. All are low power motors running at 24V. In any examples I am assuming a maximum spindle speed of 10,000, just for the sake of round numbers. Your router may have different speeds depending on the quality and specific type of the spindle motors.

Just for reference the power of the stock spindle motor is less than 100Watts or 1/8HP. The machines for which the 'normal' CNC bit guides are written will range from, well let's just say 1HP or 800W is a low powered machine.

Also there are other considerations, I am limiting this guide to what I would call affordable bits. My router was not expensive so I am not really interested in using 'professional' bits at more than £50 each, even if I could fit them into my ER11 collet holder. *(Actually I am not really interested at £25 each, that was just a random figure, and I **definitely do not** have a poster of a 6 flute 25mm long end mill using internal coolant on my bedroom wall)*

I am using free to use software here wherever possible (Apart from Windows of course). *NOTE: I mean free to use for me as a non commercial hobbyist, for example I am using Autodesk Fusion360 and have a free licence as long as I don't try and make any money out of it, that's fine as a full licence would be hundreds of pounds a year and it's a hobby.* Any software which has a free version and a subscription version, well I am referring to the free one unless specified.

I am producing this as a document, mainly because It is an edited series of notes I made, partly because I am tired of watching instructional videos on YouTube which play at Mach 2 and so I am constantly stopping, rewinding and re-watching. *Also I have been told I have a body made for Radio, a voice most suited to the Deaf and I still haven't worked out how to use the video camera on my phone properly.*

My Background

- I am a retired IT guy, started as a trainee programmer using an IBM 360/40 a long, long time ago *(the /40 designation meant it had ~ 40k memory, massive at that time)*.
- Until recently if I wanted to design something I would use pencils, paper, compass and ruler.

- I did get 98% on my last woodwork exam and the table I made is still in use, but that was an even longer, long time ago.
- Always been keen on making things and trying to understand how things work.

Disclaimer

Yeah nowadays it has to be included, and it makes sense as I accept NO liability for anything I may say, write or think.

1. I am a total novice (or was when I started out). **I MAKE NO CLAIM THAT ANYTHING I SAY IS CORRECT!** All this is based on searching the Internet, forum comments.... and my own experiences.
2. I am not a professional machinist, just a hobbyist trying to make sense of the world around me and make nice things. If you disagree with anything in here for any reason, please let me know.
3. While I do not work for, or am associated with SainSmart in any way they did provide some samples of bits for me to review and update this series of guides.
4. These are power tools, sharp pieces of metal designed to cut things while rotating at high speed and throwing small bits away also at high speed. **Keep everybody's fingers out!** A wood chip flying up your nose is nothing to be sneezed at, never mind in one of your eyes.
5. Take account of the materials you are using and take adequate precautions, dust and ashes can be harmful.
6. I work in mm. Why anyone still uses inches for design work is a mystery. *But then I still think of temperatures in Fahrenheit, anything beyond 500m in miles and property plot sizes in acres!*
7. I have been told by lots of people that my sense of humour is at the least a bit strange. I do not apologise for it.
8. Any corrections, clarifications or discussion welcome. You can find me on the Facebook SainSmart Genmitsu CNC Routers Group as Graham Bland. https://www.facebook.com/groups/SainSmart.GenmitsuCNC/

Accompanying files

NOTE: Facebook has a problem, it will not allow compressed files (.zip, .gz, ...) to be uploaded to the Files section of any group, but it bases this decision solely on the file extension, not the contents.

It is easy to bypass this by changing the file extension before uploading, I use .zip files, and rename them to .zipp This means that after downloading you will have to reverse this by renaming any xx.zipp file to xx.zip, windows will issue warnings that you are probably going to make the file unusable, just ignore it and rename anyway. Then you can open and extract the files as you wish.

So you there may be 2 versions in the files section. One an xx.pdf which is the basic words, and if there are any accompanying files with it an xx.zipp with the .pdf and everything else.

Accompanying files for this guide are:

Tool library for Fusion 360 with all SainSmart bits.	SainSmart Master V1.tools
Tool library for Vectric V-Carve Desktop with all SainSmart bits.	SainSmart Master V1.vtdb
The raw bit data for the tool libraries	SainSmart Master Bit Data V1.0.xlsx
Basic tool feed and speed calculators and V-Bit and Ball nose bit Width vs Depth calculations.	Simple Bit Calculators V1.0.xlsx

Other files in this series

All these are in the files section of this group (https://www.facebook.com/groups/SainSmart.GenmitsuCNC/), there may be more or less as I am not going to update everything if I add a new one or remove one.

They can be found in the files section of the Facebook group by searching for 'Introduction to CNC for a Total Novice -' Or 'Introduction to CNC for the Total Novice -' *Consistency has never been one of my strong points, just search for' Introduction'!!*

Title	Contents	FaceBook link
- Getting Started	Introduction to the process and how to check and test your 3018. READ THIS FIRST!	https://www.facebook.com/groups/SainSmart.GenmitsuCNC/permalink/2352850398359114/
- Making a Spoilboard	Example of how to cut, fix, face and engrave a 3018 Spoilboard with specific instructions for 3018-Pro and PROVer.	https://www.facebook.com/groups/SainSmart.GenmitsuCNC/permalink/2451607738483379/
- Setting up a Laser	Fitting, focussing and starting out with a Blue Diode Laser on your 3018.	https://www.facebook.com/groups/SainSmart.GenmitsuCNC/permalink/2356427874668033/
- Tuning Grbl Settings	Exploring the optimized settings for your 3018 to make it faster and better.	https://www.facebook.com/groups/SainSmart.GenmitsuCNC/permalink/2373672706276883/
- Spindle Speed and Laser Power	The relationship between the Spindle Speed and Laser power of your 3018 and how to set the values.	https://www.facebook.com/groups/SainSmart.GenmitsuCNC/permalink/2531618443815641/
- Basic GCode for Grbl	Introduction to GCode on Grbl based routers plus non GCode control commands.	https://www.facebook.com/groups/SainSmart.GenmitsuCNC/permalink/2705472216430262

Versions of a lot of these along with more information and resources, yes there are other people who do this sort of thing, can be found at https://docs.sainsmart.com

Units

I work in mm, rates of movements on routers are usually expressed as x per minute so that's what I use here *(anyone coming from a 3D printing world, well they normally use x per second). Other units do exist, are readily available and can be easily converted.*

The Basics

How does a router bit work? Well it has sharp cutty bits mounted on a shaft which when rotated and pushed into the stock cuts bits out of it which are carried away by the spaces on the cutter! *Pretty obvious but please read on, this does get a little more interesting!*

Bits come in a bewildering variety of shapes and sizes and an even greater variety of names and descriptions, often there is more than one name for basically the same thing. So let's start with the more common shapes or types of bits used on hobby machines. Your CAM software will have names and definitions for the types of bits it knows how to use. These vary between CAM software suppliers.

Common shapes of bits

These are the most common and here I am only describing the shape of the cutting surfaces and so the shape they will carve out of the stock. Each of these can have lots of variances but the basics remain the same.

V (AKA V Groove, Chamfer Mill, Engraving)

It will cut a V shaped groove into the stock the angle of the V is normally referred to as the angle between each side of the groove. Also you may see a tip radius or diameter specified, this would be a small flat spot at the tip of the bit, often these are called Engraving bits.

End Mill (AKA Flat End Mill, Straight Mill, Slot Drill)

Basically a straight rod, it will leave a flat bottomed square groove as it cuts. Lots of variations in shape such as fishtail which describes the shape of the bottom face.

Ball End (AKA Radiused Slot Drill, Ball Nose, Lollipop)

A round ended bit where the circular cutting edges extend past the vertical. *Some software doesn't allow this so the closest specification would be a Ball Nose which is mostly close enough.*

Ball Nose (AKA Round Nose)

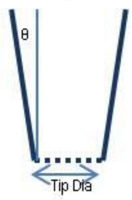
An end mill with a hemispherical end. *OK an end mill with half a ball stuck on the end.*

Bull Nose (AKA Radiused End Mill)

An end mill with a cutting edge radiused at the edges.

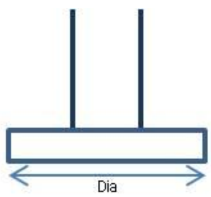
Tapered Mill (AKA Dovetail Mill)

Where the edges of the bit are tapered, not vertical. This will have a shape on the tip such as a tapered ball nose, flat nose, bull nose.... The advantage of a tapered bit is that they tend to be more robust, but they are not going to cut a vertically edged pocket. *A dovetail Mill normally has a negative taper angle, the wider portion being at the tip.*



Surfacing Mill (AKA Face Mill)

Really these are just big end mills, the cutting edges are designed to leave a flat surface and the larger diameter reduces the ridges which can be left by a smaller diameter bit.



Others

Lots of these exist for many reasons, but especially when starting out you are unlikely to need a Radiused Square End 23 degree Shallow Ogee bit.

There is one however which is worth mentioning here is a Form Mill. Only normally supported by higher end CAM software this lets you define a line or form which describes the shape of the bit.

You might have noticed the beginnings of potential confusion already. Well hopefully not confusion but a blurring of the boundaries. For example what is the difference between a V Bit and a tapered mill with a very small tip diameter, or an end mill and a surfacing mill........

The simple answer is not a lot; just pick the closest match in your CAM software. For example if it has a V bit definition well a 0.1mm tip diameter is not going to make a lot of difference for 99.99% of the things you are doing but you may find an Engraving bit allows you to specify the tip diameter.

Like many things the descriptions and terms used to describe CNC bits and their properties are diverse and many words which basically mean the same are used. This is not a thesaurus but I have tried to mention all the common terms I have come across.

ER 11 Tool holder and collet

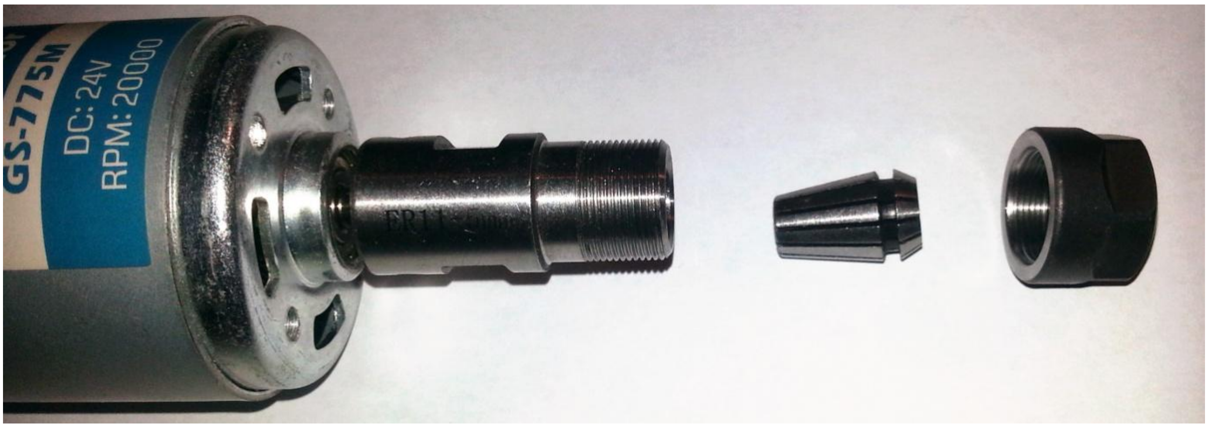
Not strictly part of the bit but very important. What the ER 11 collet system is on your router and how to use it. This is very important to ensure your bit is mounted correctly. This is the 'standard' tool holder for small routers. ER is the 'name' 11 means that it has an 11 mm diameter at the bottom of the taper in the holder.

Why is a collet system is used, why not just mount a chuck which would be more versatile and easier to adjust? A Collet system will:

- Clamp the bit more firmly.
- Give a superior alignment of the bit within the holder.

The drawback is that the size of the collet used must be matched to the diameter of the bit shank, but different sized collets are relatively cheap.

It has 3 parts, Holder, Collet and Clamping nut:



Holder

Normally factory fitted onto the motor shaft as it is a very tight or interference fit onto the shaft in order to keep it exactly aligned. It is further held by two grub screws tightened onto the motor shaft. It can be removed and replaced but will probably require heating to slacken the fit and the very careful application of force to avoid damaging the motor.

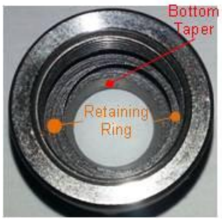
It has a tapered interior which matches the angle of the collet so that as the collet is forced into the holder the collet is compressed and clamps round the shank of the bit. A threaded area at the bottom to match the clamping nut and a couple of flats allowing a spanner to be used to tighten and release the collet.

Collet

The Collet fits round the shank of the bit and slides into the tapered interior of the holder. Near the bottom is a groove which fits into the clamping nut and at the bottom another taper which matches the clamping nut. The slots cut through the collet allow the segments to be forced inwards round the bit as it is tightened into the holder. There is obviously a limit to how much it can be compressed meaning the hole through the collet must match the diameter of the shank of the bit.

Nut

Not just a simple flanged nut, the design of the interior is important.



At the base there is a bottom taper, this matches the taper angle of the bottom of the collet and ensures as it is tightened the collet is compressed evenly.

Above this is a retaining ring, this is very deliberately made wider at one side than the other, it's not supposed to be concentric to the nut. This retaining ring fits into the groove of the collet. This holds the collet into the nut and also when you wish to remove a bit it forces the collet down out of the taper of the holder breaking any 'cold welding' which may have occurred and so releases the clamping round the bit allowing it to be removed easily.

The simple reason that the retaining ring is offset within the nut is to allow a collet to be inserted and removed.

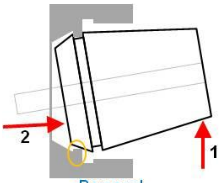
Collet Sizes

ER11 Collets are described by the internal diameter. So a 3mm collet is designed for a bit with a 3mm shank diameter. There is a bit of tolerance built in due to the clamping action so a 3mm collet will also take a 1/8 " or 3.175mm shank. A 1/4" shank is 6.35mm so it's closer to the tolerances but a 6mm or 7mm collet *should* work. Depending on the bits you use get the collets to fit them, if you frequently use 6mm bits then get a 6mm collet, if you often use 1/4" bits get a 1/4" collet, if you use both, get both.

Collets are also available in sets which can be cheaper and very useful if you also use drills.
https://www.sainsmart.com/products/genmitsu-15pcs-er11-precision-spring-collet-set

Changing a collet

Removing a collet



Removal

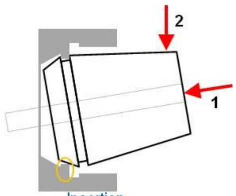
First remove any bit and unscrew the clamping nut completely from the holder. The collet now needs to be popped out of the retaining ring by pressing the collet over to the side (1) and then applying a push to pop it out (2).

It will depend on where the direction of the sideways push is. It will be easiest to push out when you are lifting the collet slot over the narrowest point of the retaining ring on the nut as shown by the orange circle. As you cannot see where this is if it doesn't pop out at the first attempt rotate it a bit and try again.

Inserting a new collet

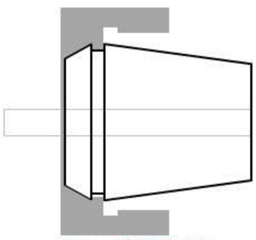
Before inserting a new collet make sure;

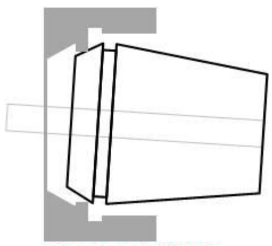
- The inside of the tool holder is clean and free from dust and debris.
- The collet itself is clean, on the outside of the tapers, in the spaces between the collet segments and in the bore which will take the bit.
- The nut is clean and free from debris around the retaining ring and on the bottom taper.
- Do not insert a collet into the nut with a bit inserted into the collet! This will limit the amount the segments of the collet can move and will make it nearly impossible to seat the collet into the nut correctly.



Insertion

Locate the widest point of the retaining ring in the nut and engage the slot of the collet into it, while pushing it in to the nut (1) press it straight (2), it will 'clip' in. After insertion check that the collet is straight and is fully seated into the nut.


Correctly Fitted


Incorrectly Fitted

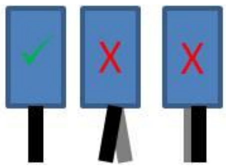
The left photo shows a correct alignment with the collet aligned with the nut and the bottom taper sitting on the corresponding part of the nut. On the right the collet is not aligned with the nut, the bottom taper is resting on the retaining ring rather than it being in the slot of the collet. When tightened from this position the collet will be forced out of alignment and damage to the nut, collet and tool holder could result.

For ease of use I have purchased an ER-11A clamping nut for each collet. This means I can just leave the collet permanantly in the nut. https://www.sainsmart.com/products/genmitsu-4pcs-er11-a-collet-clamping-nut

Inserting a bit

- Make sure the nut is slack on the tool holder.
- Make sure the shank of the bit is clean.
- Slide the shank into the collet, the maximum depth of the bit the collet will clamp is 18mm (some collets may have a shorter clamping distance). There is a gap between the top of the collet in the tool holder so a bit can be inserted further if required.
- Using the spanners firmly tighten the nut onto the holder, compressing the collet around the bit. This does need to be tight.

Checking everything



After mounting a bit, especially after changing the collet, manually rotate it and ensure the rotation of the tip of the bit is straight and true.

If it is not check:

- The collet is seated correctly in the nut.
- A dirty bit shank or collet interior is not throwing things out of true
- The tool holder taper and collet exterior are clean
- The collet is the correct size for the bit shank
- The collet has been tightened enough and holds the bit securely
- The bit is not bent or damaged.

If excessive vibration occurs when the spindle is turning a process of elimination can identify the cause. Remove the nut, collet and bit from the holder and try again. If the vibration persists check that the holder is straight and secure on the motor shaft, if it still persists then the motor itself is the suspect.

Remove the bit from the collet and reattach the nut and collet into the holder and try again. If the vibration persists check the collet mounting into the nut. If it goes away check that the collet is the correct size for the bit shank or try a different bit.

NOTE: These checks should be made without performing any cutting. If excessive vibration occurs when cutting it is often due to over stressing the router with the depth of cut and/or feed rate being too high.

Bit Dimensions and definitions

Why is this important? Well for any CNC machine your CAM software needs to know some things to generate a toolpath, which are instructions on how to move the **centre and tip** of the bit around to achieve the desired result. Things like the diameter of the bit are very important so it can work out how to cut out the shapes and contours you want. The shape of the bit is also important so it can calculate how deep it needs to be to achieve the desired contours, can it cut a vertical edge....

Things like how deep is your bit capable of cutting before, for example, a thicker shaft starts rubbing against things it should not touch so it can show an error rather than just proceeding, well that depends on the complexity of your CAM software. It's still important but a lot of the simpler 2.5D CAM software just ignores the problem which means it is down to you to prevent it from happening.

Some of the speeds and feeds you will use depend a lot on the bit you are using and its physical characteristics, you will break a lot more bits if they are long and thin than if they are short and fat! So size is important.

So let's look at some of the makeup and dimensions of a typical bit.

Shank or Shaft

The round piece at the top of the bit that goes into the collet, the shaft diameter must match the collet in use (See above).

Overall Length

Length of the bit from top to tip.

Stickout (aka length below holder)

How far the bit sticks out from the end of the tool holder. The maximum length that the collet can clamp is 18mm, the length that can go into the tool holder is ~25mm. The Stickout and the positioning of the Spindle motor in the holder needs to be long enough to reach the bottom of the cut with the Z axis at its lowest and still to clear the top of the stock with the Z axis at its highest.

Keeping the Stickout as short as is necessary helps reduce flexing of the bit increasing accuracy, reducing wear and the chances of bit failure.

Shoulder length

Especially in the case of small diameter bits the shaft diameter will be larger than the cutting diameter, the shoulder length is the lowest of how far from the tip this change starts. Effectively this is the maximum depth a bit can go vertically into the stock material, not as far as it can cut effectively though.

Flute Length

This is the maximum height of the working range of the flutes from the tip of the bit. *This is not the same as the length of the spirals from the tip as these normally continue up a bit more than they can effectively work.*

Maximum Depth of Cut (DOC)

Simply how far down the bit can effectively cut, this is normally the smallest of the flute length and shoulder length for a straight edged bit. If you are using a Vbit for example this would normally be the maximum depth at which it will cut a V before just starting to leave a round hole. DOC is also used to describe the distance the bit is actually instructed to cut in a single pass but for a bit definition it's the maximum depth it can cut.

Flutes (AKA Teeth)

The number of cutting surfaces on the bit, this is usually 1 to 4 but can be higher, Burrs can get well into the teens. Used in speed and calculations to determine things like how big the chips are going to be and it will affect surface finish, speeds.....

Diameter

Simple the diameter of the cut or slot the bit will make, well not so simple in practice, what is the diameter of a V Bit? It changes from the tip to the top! So this depends for some bits on what CAM software you are using.

Corner Radius

Normally applicable to bull nose bits, the radius of the corner between the tip and the side of the bit.

Taper and other angles

Just angles but depending on the type of bit and CAM software. **BUT** the angles can be expressed in different ways! Normally a V bit angle is measured by the angle of the groove it will make from edge to edge, often referred to as the Included Angle. But for a tapered bit it is normally measured between one edge and the vertical or the half angle.

Spirals

The spirals on a bit have a simple job, to remove the chips away from where it is cutting so the bit does not get clogged. Something like a V bit doesn't have spirals as the chips can just move away sideways or upwards by themselves. Well that's the theory anyway.

But as they will also come into contact with the side of the stock so while they are not exactly cutting surfaces they still rub and so have an effect on the stock.

Corncob

Where the spirals are shaped to provide extra cutting edges or surfaces to break up the ships more, or both. Think of a cob of corn but with sharp kernels arranged in spirals rather than straight lines.

Upcut, Downcut and Compression

This is all about the direction of the spirals on the edges of the bit. The edges of a straight bit will create friction from the rotating spirals and the movement of the chips can cause splintering on the edges of the cut, especially on laminated materials such as plywood or veneers and others.. It can also leave a lip on softer materials.

Upcut

The standard bit type, just like a drill the spirals move the chips UP away from the cutting edge. If a bit description doesn't mention any cut type it's probably an Upcut. Can cause a bad finish at the top of the stock.

Downcut

The spirals move the chips DOWN towards the cutting edge, this means that the action of the spirals on the edge of the material is downwards which can help prevent splintering or the creation of a lip at the top, but if you are cutting all the way through a material can transfer the splintering/lip to the bottom surface.

There is a major consideration when using these in that the chips still need to be moved away from the cutting edges to prevent clogging. Having them leave at a side is the only option so don't use them to plunge or drill into the stock!

Compression

Well you have a cut that using an Upcut bit leaves the top ragged but when using a Downcut bit leaves the bottom ragged. Use a compression bit, Upcut at the bottom and Downcut at the top!

That is a compression bit in a nutshell, disadvantages are that all the chips meet somewhere in the middle hence the compression name so make sure there is some way for them to get out.

Normally these will have a short Upcut section at the bottom and a longer Downcut section at the top rather than a meet in the middle design as this allows much more flexibility in machining different thicknesses of stock.

Material

What the bit is made of not what you are cutting. A basic rule of thumb is that the bit must be a lot tougher than the material you are cutting but there are a lot of other factors such as rigidity, wear resistance and don't forget cost.

HSS

Called High Speed Steel ~~*because it can be made very rapidly*~~ This is pretty much a generic description for a modern steel blend designed mainly for high speed tools.

Carbide

Carbide tools are formed from a tungsten (or other metals) and carbon powder. This has a very high wear resistance and so keeps sharp for longer but it is also expensive. Often a tool will be made from HSS with just the cutting surfaces made from Carbide. Because of the cost pure carbide is not normally used by hobbyists.

Tungsten Carbide Steel

A High Speed Steel with carbide blended in to make it more wear resistant.

Coatings

A coating can be applied to the bit to enhance mainly to enhance its resistance to wear, reduce friction heat transfer and finish quality.

Nano Blue Coat

A high performance Nanograin (very small particles) carbide and silicon coating giving 2-3 times better tool life than a micrograin carbide coating.

Titanium

A titanium nitride coating to enhance cutting performance and wear. This has a gold colour.

Others

There are a myriad of other options including no coating, the differences between them are beyond my comprehension and best left to materials scientists and professional machinists!

Milling and routing terms

Some questions that need answering to get good (*or bad*) results are how fast is it turning, how big is it, how fast is it being pushed into the stock and in which direction, how many flutes, what do the spaces or spirals do with the waste material, how deep is it cutting, what is it cutting......?

Splintering

Mainly for laminated materials this refers to the action of the bit as it cuts causing the top (or bottom) surfaces to separate slightly and results in splinters on the surface. Other materials can do this as well, normally hardwoods, softwoods can have 'strings' left on the edges of a cut, which is basically the same.

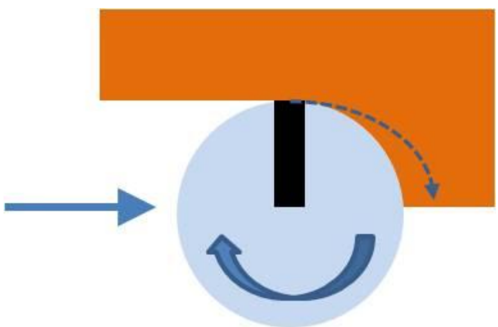
Climb vs conventional milling

Lets start with a router bit, flat end, straight edges on the shaft, one flute at the bottom, spirals round the edges to remove the chips and 3mm in diameter, or in other words a 3mm single flute flat end mill.

It will be turning clockwise, that is traditional and small router controllers will only turn the motor in a single direction which is clockwise. In very simple terms the bit is being moved into the stock at a constant speed while rotating clockwise.

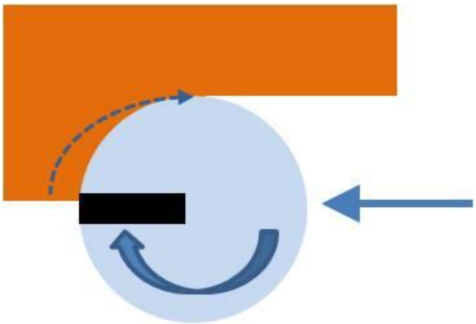
Climb vs Conventional is a term used to describe which direction the bit is being fed into the stock.

Conventional milling



Each flute will carve out a chip from the stock as shown by the dotted line due to the movement and rotation of the flute as it turns and moves. The chip will start off small and get thicker as the bit moves and rotates.

Climb milling



As for conventional milling but the chip will start out thicker and end up small.

As far as I can tell climb milling gets the name from when using a handheld router it will try and climb out of the stock unless firmly held, but please don't quote me on that, especially as it is sometimes referred to as down cut, but I suppose if it is cutting down then...!!!

Climb milling will normally leave a cleaner finish with less splintering at the top than conventional milling.

In both cases the thickness of the chip depends on the rotation speed of the bit and the feed rate of movement as well as the diameter of the bit and how far it is cutting into the stock. The chip must then be removed by the spirals on the bit ready for the next cut.

As to what happens during the rest of the rotation while the bit is still moving but not cutting – well it's turning quite fast so will get round to the next cut before it becomes a serious problem, if it is turning too slowly then it will bend and possibly break.

Spindle speed

A simple one, the rotational speed of the bit in Revolutions Per Minute, often just referred to as speed.

Feed Rate

The speed at which the spindle is being moved into or across the stock, often just referred to as feed. The faster you move the bit through the material the quicker it will cut but it will also put more stress on the bit and the machine. If you exceed the federate that your router is capable of for a cut then the stepper motor will skip or lose steps and your work will be ruined.

Depth of Cut (DOC)

Mentioned as a bit parameter but when cutting it is how deep each cutting pass will be into the stock. The larger the Depth of Cut then more material will be removed on each pass meaning fewer passes are needed, but again larger values will put more stress on the bit and machine.

Max Feed

The maximum achievable feed rate for the router. This is normally set as the feed rate the router is capable of moving the bit, not the rate at which it will cut! For a Grbl based machine this is held in the settings as $110-112 for the X, Y and Z axis respectively, often no more than 1,000mm/min. If you use a value which exceeds the setting then Grbl will just use the setting value instead. *Just because it can move at a certain rate does not mean it can cut at that rate!*

Feed per Tooth

Calculated value dependant on the number of flutes and Feed. It's just Feed / number of flutes. *(Please don't ask why it's called feed per tooth when the teeth are normally called flutes, just don't go there, OK?)*

Feed per Revolution

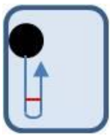
Calculated value dependant on the Speed(RPM) and Feed. It's how far the bit moves along the cutting path in a single revolution. It's Feed/RPM and is expressed in mm.

HorsePower or Kw

The power of the spindle motor, measured in HorsePower or Kw. The 'standard' power of the 'standard' 775 spindle motor on a 3018 is around 60W, or 0.06Kw, or 0.08HP. *Think more of Hamster Power!*

Stepover

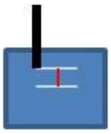
The distance the bit is moved sideways between each cut or line. This can be expressed as either an absolute distance or a percentage of the bit diameter depending on your CAM software. 0 would mean that the bit would just retrace the previous cut, anything over the bit diameter or 100% would mean that ridges would be left between each cut.



A good rule of thumb is 40-50% of the bit diameter, lower values increase the machining time but give a better finish quality.

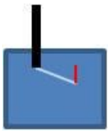
Stepdown

See Depth of Cut! The distance the bit is moved down after each layer has been cut. This depends on the material and bit, the greater the Stepdown the faster a pocket will be cut as it will take fewer passes but will increase the stresses on the bit and the machine. These are small machines so use small values to start, 1mm is quite massive on these small machines but it depends on what you are doing to what.



Ramp

Where the printer gradually applies the Stepdown as a ramp rather than plunging and then cutting horizontally.



Lead in and out

The way the bit is moved into and out of the cut, the options you have depend on your CAM software. Often a separate spindle speed and feed can be specified for lead in and lead out operations.

Plunge

Moving straight down or plunging into the stock. Normally used as a lead in/out method, though it could apply to drilling. This is the most aggressive lead in method. *NOTE: Do not use with compression or downcut bits! They move the chips down and so when plunging leaves them with no way to leave the cut and will cause the bit to clog, overheat...!*

Spiral and helical

Other lead in methods, once will lower the bit in a spiral ramp, the other will use a helical ramp. These terms can also be used to describe the toolpath used, normally for roughing operations.

Chatter and Sawdust

In an ideal world the bit moves through the stock with each flute carving out a distinct chip removing the stock cleanly and the chips carry away the heat generated by the bit action without overstressing the machine or wearing out bit too much.

Sawdust

What you get when the chips are too fine, basically they stop being recognisable chips and become sawdust. You can reduce this by slowing the RPM and/or increasing the feed rate so that the bit is cutting larger chips.

Cutting too small chips can greatly increase the heat generated by friction between the bit and the stock as the chips are supposed to carry a lot of the heat away.

This can be especially important when cutting materials such as plastics with a low melting point where it's not so much sawdust as the bit looking like the centre of a candyfloss machine due to the material melting.

This can be reduced by decreasing the RPM, increasing the feed rate, possibly increase the depth of cut. It is advised to only change one at a time and see what happens.

Chatter

What you get when the chips are too large. Called this as the machine is overstressed trying to push the bit it starts to make chattering noises.

This can be reduced by increasing the RPM, decreasing the feed rate, possibly decrease the depth of cut. It is advised to only change one at a time and see what happens.

Roughing and finishing

Imagine you are trying to make letters in relief, so you have a lot of material to remove around them, but also want fine detail around and in the lettering.

If you try and make a large carving just using a 0.5 mm bit it may come out well, but there is a chance you will expire from old age before it is complete.

So the first pass is with a bigger and coarser bit (roughing), this removes the material much quicker but doesn't give as good a finish. So you tell your CAM software to leave a small margin around all the surfaces.

This is followed by a second pass with a smaller or different shaped bit (finishing), it will fit into all the nooks and crannies, go round the edges, smooth the surfaces.... but although it will be slower it doesn't have to remove large swathes of material!

The major drawback is that you have to change the bit between the passes. If you carefully change the tool without manually moving anything on the XY then the XY axes 0 position should not to be changed. If you have limit switches it's a good idea to write down the machine coordinates after positioning the XY axes on the first pass so that you can always home, then jog back to the same coordinates and reset the XY work origin.

The new tool will need a reset of the Z axis zero so it's wise to set the original Z zero somewhere where it can be reset after the roughing pass has completed. Remember the Z zero does not have to be set in the same XY coordinates of the work origin.

How to generate roughing and smoothing passes is totally dependent on the CAM software you are using. Some will support a number of passes not just 2!

Surface speed (AKA Cutting speed)

This is just the speed that a single flute at the circumference of the bit travels across the surface of the stock. It is calculated by RPM * bit circumference so a 3mm dia bit has a circumference of π*Bit Diameter or in this case 9.42mm so at 100 RPM the surface speed is 942 mm/min, the bigger the bit the faster the surface speed will be (at the same RPM of course).

Surface speed also generates heat by friction which can be especially annoying when cutting materials with a low melting point like plastics.

Chip Load

This is a measure of the thickness of the chip that each flute will carve out of the stock. It is calculated by Chip Load = Feed Rate / (Spindle Speed * Number of flutes). As you can see Chip Load does not depend on the diameter of the bit BUT larger bits can normally take a larger Chip Load simply due to the fact that the flutes will be larger and the bit more rigid. If the Chip Load is bigger than the size of the flutes you will have problems.

If you have a recommended Chip Load for a bit then you can determine a range of speeds and feeds to supposedly give the same results.

NOTE: The size of the chips or Chip Load is especially important for materials such as plastics which have a low melting point. This is because cutting generates heat by friction between the bit and the stock. The chips help to carry away this heat from the stock and so prevent it from melting so a larger Chip Load is beneficial for these materials.

Runout

Runout is the difference between the diameter of the bit and the distance it will cut when rotated. A professional machinist will probably measure this using an, often expensive (cheap ones are available), dial guage by comparing the distance measured to the bit as it rotates. *Well I don't have a dial guage and can practically ignore this on my 3018's as they are not designed for precision machining,* ***well unless I can see a wobble at the end of the bit****.*

Setting up your CNC software

This is important as your CAM software needs to know what bit you are using before it can generate a toolpath, Mainly it needs to know the shape as well as what speed and feed to use. At the simplest level it will, quite rightly, not even try and fit a 3mm diameter bit into a 2.5mm gap. It may also use the bit dimensions for things like checking the maximum cutting depth, is the shank of the bit going to hit anything.......

All CAM software has a slightly different way of defining the bit, some are more expansive, some more limited in what bits they can understand.

Generating the movements in a toolpath is all about the shape and size of the bit in relation to the pattern you are cutting, this makes no difference if the bit is made from steel, carbide, or even cardboard, it's just moving it around so that the stock is cut in the right places to produce the correct result. Imagine you wish to cut a round pocket 10mm in diameter. To work out the correct tool path the CAM software has to know the diameter of the bit to leave the correct sized hole. If you set up a tool path saying you are using a 3mm bit and fit instead a 6mm bit the pocket is not going to be 10mm in diameter!

Then there is the adding of speed and feeds, depths, passes etc. All these depend on not just the bit but the material, machine capabilities.... the software may contain some hints, default settings for a particular material but remember that not only does the USDA Wood Handbook lists 49 separate hardwoods, and that's just hardwoods native to the USA, add softwoods and species not native to the USA and the number goes up and up! Never mind about kiln dried, naturally dried and if the tree grew on a north facing or south facing slope.

There are nearly as many 'species' of aluminium and I am not even going to think about plastics.

Defining the bit in your CAM Software

Nearly all CAM software has a tool library where details of the bits you have can be stored to make life easier. Most CAM software will allow you export from and import to the tool library in bulk which is a simple way of saving your bits and adding new ones.

I would like to say that all the measurements are accurate, they rely both on the manufacturers specifications and my own measurements of the bits I have, but they are small little..... and counting the flutes on the burrs is very tricky especially with my eyesight while trying to hold 2 magnifying glasses and the bit at the same time, even without the varifocals! But I hope any errors I may have introduced will make little difference to the results.

If you find any errors or disagree with anything I have said or have better ideas then please let me know via the Facebook Sainsmart CNC users group.

Common CAM software

Lots of other options are available than those given here but some of the more common ones are with some details of the tools they can handle.

Feature	Carveco Maker	Fusion 360	Vectric V-Carve	Easel
Export Tools	Y	Y	Y	N
Import Tools	Y	Y	Y	N
Form Tool Support	Y	Y	Y	N
Milling Tool Types	13	13	11	3
Use user defined tools	Y	Y	Y	*Pro mode only*
Define tool Holder	N	Y	N	N
Save material default Speeds/feeds		Y	Y	
SainSmart Tool library accompanying		Y	Y	N/A

- This relates solely to their handling of different bits.
- Which CAM software you use is a personal choice and support for bits may be a minor factor.
- Some of the above relate to use on small hobby machines like the 3018's.
- SainSmart Tool library refers to a file accompanying this document, not included in the software.

Telling your software about the bits

This should be straightforward, but, and there is always a but!

Let's take 2 bits

- 20 degree V bit
- 3mm ball nose bit.

All straightforward, but:

- The types and shapes of bits supported in the each product varies. For Example Fusion 360 doesn't have a V bit type, it does have for milling Tapered mills and chamfer mills and Countersink bits for hole making....
- For a ball nose bit what is the difference between a 5mm bull nose with a 2.5mm corner radius and a 5mm ball nose bit??
- V-Carve allows the definition of V bits and engraving bits. Should the V bit be defined as a V bit or an engraving bit? It can be used for both purposes so which is correct?

There is a very simple answer – **I don't know!** So I have tried to pick what makes most sense to me, I am often wrong!!

Tool Libraries

At the moment tool libraries are defined for Fusion 360 and Vectric V-Carve. I also include the source spreadsheet of tool dimensions to make it easier for you to define the bits in your own CAM software.

Tools which would have to be defined as Form tools are not included, Bit types which cannot be defined are ignored.

The Tool Library is named 'SainSmart Master' and can be imported. I then suggest you create another group in your tool library called something like My Bits and copy the tools you possess into that to create a list of the tools you have, organise them how you like so you can select the one you want easily, depending on the software you are using of course.

Fusion 360 Notes

- For a milling operation you must use a milling bit type, a V bit would have to be specified as a chamfer bit or taper mill which are of a milling type, not a countersink bit which is a type for drilling operations.
- An ER11 tool holder is defined in the tool library and used for ALL tools. The shape and dimensions of the tool holder are used by Fusion 360 for collision detection.
- In the Post Processor the tool number and turret are unused – no tool changer on a small router. Manual Tool change and Live tool are always selected.
- Break control and coolant are not supported and so are turned off.
- The default Cutting Data is set, I hope, on the conservative side. Modify this as needed, these are not recommendations!
- Presets for different materials can be added to your tool library later if you want to save your working values.
- The tool name is generated by Fusion 360. It adds the tool cutting diameter and Tool length below the holder to the front of the individual tool description. You can select tools you see by using filters.

Easel Notes

- There is no facility to export or import tools so no tool library can be provided.

Vectric V-Carve Desktop Notes

- The tool name allocated depends on the type and includes minimal information tool type and basic dimensions so there will appear to be duplicates, for example a End Mill (3.175mm) will appear twice being the 1 flute and 2 flute variants. Full details can be seen by clicking on the tool and examining the detailed settings. I have tried to add meaningful names and notes for all the tools.
- The library is organised by the SainSmart bit set Product ID on the assumption that you will copy the tools you have to a different group and organise them as you see fit.
- A V bit does not allow a tip diameter to be specified, an Engraving bit however does but note that the angle defining a V bit is specified as being between the outer edges of the V but for an engraving bit it is specified as the angle between one edge and the vertical, so a 20˚ V bit becomes a 10˚ Engraving bit.

The real world

Everything works perfectly, apart from in the real world, so remember.

- Cutting edges don’t remain sharp forever.
- The power available to drive the spindle is finite, a professional router may have 2 or 3 horses harnessed to turn the spindle, most hobby routers have a single, quite elderly, hamster running inside a wheel to turn the spindle.
- If you try and force a bit too deep and too fast through the stock either your stepper motors will stall loosing steps and the routers position or maybe the bit will break.

- Nothing is perfectly rigid, bits will distort and flex, especially if you are using small diameter bits, not a lot but perhaps enough to affect even these low powered machines.
- Bits can and will generate heat by both cutting and friction against the stock. If you have ever left a drill bit running in a hole until you see smoke coming out then you will appreciate that this can start a fire (If you haven't seen it then take my word for it) so don't leave your router unattended.

Some Simple Formulae

Here are the formulae used for the calculations. There is an excel spreadsheet in the accompanying files which contains the calculators.

Values:

Bit Diameter	The diameter of the cutting edge of the bit
Spindle Speed	Rotation speed of the bit in RPM
Flutes	The number of cutting edges on the bit
Feed Rate	The rate the bit is moved into the stock in a straight line
V Bit Angle	For V bits only the angle between the edges of the V it will leave. Only used in the V and ball depth and width calculators.
Depth of Cut	How deep the finished cut will be. Only used in the V and ball depth and width calculators.
Cut Width	How wide the top of a finished cut will be. Only used in the V and ball depth and width calculators.

Formulae:

Bit Circumference	Bit Diameter * π
Surface Speed	Bit Circumference * Spindle Speed
Cutting Speed	Surface Speed * Flutes
Chip Load	Feed Rate / (Spindle Speed * Flutes)
Feed per Cut	Feed Rate / Flutes
V Width	tan(V Bit Angle/2) * 2 * Depth of V + Tip Diameter
V Depth	(V Width – Tip Diameter) / (Tan(V Bit Angle/2) * 2)

As you can see all these calculations can rapidly become circular, they are all using the same few basic parameters and are often only slightly different ways of expressing basically the same thing.

Depth and Width Calculators

These calculators can be useful of you are using a V aka engraving bit or a ball nose bit if you want to know how deep to cut to get a specific width of cut or vice versa.

I have included some easy reference tables for width and depth of cut.

V Width vs Depth for Tip diameter: ***0.0*** mm								
V Angle	15°	20°	30°	45°	60°	90°	120°	150°
Width mm	**Depth mm**							
0.5	1.90	1.42	0.93	0.60	0.43	0.25	0.14	0.07
1.0	3.80	2.84	1.87	1.21	0.87	0.50	0.29	0.13
1.5	5.70	4.25	2.80	1.81	1.30	0.75	0.43	0.20
2.0	7.60	5.67	3.73	2.41	1.73	1.00	0.58	0.27
2.5	9.49	7.09	4.67	3.02	2.17	1.25	0.72	0.33
3.0	11.39	8.51	5.60	3.62	2.60	1.50	0.87	0.40
3.5	13.29	9.92	6.53	4.22	3.03	1.75	1.01	0.47
4.0	15.19	11.34	7.46	4.83	3.46	2.00	1.15	0.54
4.5	17.09	12.76	8.40	5.43	3.90	2.25	1.30	0.60
5.0	18.99	14.18	9.33	6.04	4.33	2.50	1.44	0.67
5.5	20.89	15.60	10.26	6.64	4.76	2.75	1.59	0.74
6.0	22.79	17.01	11.20	7.24	5.20	3.00	1.73	0.80
6.5	24.69	18.43	12.13	7.85	5.63	3.25	1.88	0.87
7.0	26.59	19.85	13.06	8.45	6.06	3.50	2.02	0.94
7.5	28.48	21.27	14.00	9.05	6.50	3.75	2.17	1.00
8.0	30.38	22.69	14.93	9.66	6.93	4.00	2.31	1.07
8.5	32.28	24.10	15.86	10.26	7.36	4.25	2.45	1.14
9.0	34.18	25.52	16.79	10.86	7.79	4.50	2.60	1.21
9.5	36.08	26.94	17.73	11.47	8.23	4.75	2.74	1.27
10.0	37.98	28.36	18.66	12.07	8.66	5.00	2.89	1.34

V Width vs Depth for Tip diameter: ***0.1*** mm								
V Angle	15°	20°	30°	45°	60°	90°	120°	150°
Width mm	**Depth mm**							
0.5	1.52	1.13	0.75	0.48	0.35	0.20	0.12	0.05
1.0	3.42	2.55	1.68	1.09	0.78	0.45	0.26	0.12
1.5	5.32	3.97	2.61	1.69	1.21	0.70	0.40	0.19
2.0	7.22	5.39	3.55	2.29	1.65	0.95	0.55	0.25
2.5	9.11	6.81	4.48	2.90	2.08	1.20	0.69	0.32
3.0	11.01	8.22	5.41	3.50	2.51	1.45	0.84	0.39
3.5	12.91	9.64	6.34	4.10	2.94	1.70	0.98	0.46
4.0	14.81	11.06	7.28	4.71	3.38	1.95	1.13	0.52
4.5	16.71	12.48	8.21	5.31	3.81	2.20	1.27	0.59
5.0	18.61	13.89	9.14	5.91	4.24	2.45	1.41	0.66
5.5	20.51	15.31	10.08	6.52	4.68	2.70	1.56	0.72
6.0	22.41	16.73	11.01	7.12	5.11	2.95	1.70	0.79
6.5	24.31	18.15	11.94	7.73	5.54	3.20	1.85	0.86
7.0	26.21	19.57	12.88	8.33	5.98	3.45	1.99	0.92
7.5	28.10	20.98	13.81	8.93	6.41	3.70	2.14	0.99
8.0	30.00	22.40	14.74	9.54	6.84	3.95	2.28	1.06
8.5	31.90	23.82	15.67	10.14	7.27	4.20	2.42	1.13
9.0	33.80	25.24	16.61	10.74	7.71	4.45	2.57	1.19
9.5	35.70	26.66	17.54	11.35	8.14	4.70	2.71	1.26
10.0	37.60	28.07	18.47	11.95	8.57	4.95	2.86	1.33

V Width vs Depth for Tip diameter: ***0.2*** mm								
V Angle	15°	20°	30°	45°	60°	90°	120°	150°
Width mm	**Depth mm**							
0.5	1.14	0.85	0.56	0.36	0.26	0.15	0.09	0.04
1.0	3.04	2.27	1.49	0.97	0.69	0.40	0.23	0.11
1.5	4.94	3.69	2.43	1.57	1.13	0.65	0.38	0.17
2.0	6.84	5.10	3.36	2.17	1.56	0.90	0.52	0.24
2.5	8.74	6.52	4.29	2.78	1.99	1.15	0.66	0.31
3.0	10.63	7.94	5.22	3.38	2.42	1.40	0.81	0.38
3.5	12.53	9.36	6.16	3.98	2.86	1.65	0.95	0.44
4.0	14.43	10.78	7.09	4.59	3.29	1.90	1.10	0.51
4.5	16.33	12.19	8.02	5.19	3.72	2.15	1.24	0.58
5.0	18.23	13.61	8.96	5.79	4.16	2.40	1.39	0.64
5.5	20.13	15.03	9.89	6.40	4.59	2.65	1.53	0.71
6.0	22.03	16.45	10.82	7.00	5.02	2.90	1.67	0.78
6.5	23.93	17.86	11.76	7.60	5.46	3.15	1.82	0.84
7.0	25.83	19.28	12.69	8.21	5.89	3.40	1.96	0.91
7.5	27.72	20.70	13.62	8.81	6.32	3.65	2.11	0.98
8.0	29.62	22.12	14.55	9.42	6.75	3.90	2.25	1.05
8.5	31.52	23.54	15.49	10.02	7.19	4.15	2.40	1.11
9.0	33.42	24.95	16.42	10.62	7.62	4.40	2.54	1.18
9.5	35.32	26.37	17.35	11.23	8.05	4.65	2.68	1.25
10.0	37.22	27.79	18.29	11.83	8.49	4.90	2.83	1.31

Width vs Depth for Ball Nose Bits																
Tip Dia	0.25	0.5	0.75	0.8	1	1.5	2	3	3.175	3.5	4	4.5	5	6	6.35	6.5
Width mm	Depth mm															
0.1	0.01	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-
0.2	0.05	0.02	0.01	0.01	0.01	-	-	-	-	-	-	-	-	-	-	-
0.3	-	0.05	0.03	0.03	0.02	0.02	0.01	-	-	-	-	-	-	-	-	-
0.4	-	0.10	0.06	0.05	0.04	0.03	0.02	0.01	0.01	0.01	0.01	-	-	-	-	-
0.5	-	0.25	0.10	0.09	0.07	0.04	0.03	0.02	0.02	0.02	0.02	0.01	0.01	0.01	-	-
0.6	-	-	0.15	0.14	0.10	0.06	0.05	0.03	0.03	0.03	0.02	0.02	0.02	0.02	0.01	0.01
0.7	-	-	0.24	0.21	0.14	0.09	0.06	0.04	0.04	0.04	0.03	0.03	0.02	0.02	0.02	0.02
0.8	-	-	-	0.40	0.20	0.12	0.08	0.05	0.05	0.05	0.04	0.04	0.03	0.03	0.03	0.02
0.9	-	-	-	-	0.28	0.15	0.11	0.07	0.07	0.06	0.05	0.05	0.04	0.03	0.03	0.03
1.0	-	-	-	-	0.50	0.19	0.13	0.09	0.08	0.07	0.06	0.06	0.05	0.04	0.04	0.04
1.5	-	-	-	-	-	0.75	0.34	0.20	0.19	0.17	0.15	0.13	0.12	0.10	0.09	0.09
2.0	-	-	-	-	-	-	1.00	0.38	0.35	0.31	0.27	0.23	0.21	0.17	0.16	0.16
2.5	-	-	-	-	-	-	-	0.67	0.61	0.53	0.44	0.38	0.33	0.27	0.26	0.25
3.0	-	-	-	-	-	-	-	1.50	1.07	0.85	0.68	0.57	0.50	0.40	0.38	0.37
3.5	-	-	-	-	-	-	-	-	-	1.75	1.03	0.84	0.71	0.56	0.53	0.51
4.0	-	-	-	-	-	-	-	-	-	-	2.00	1.22	1.00	0.76	0.71	0.69
4.5	-	-	-	-	-	-	-	-	-	-	-	2.25	1.41	1.02	0.93	0.90
5.0	-	-	-	-	-	-	-	-	-	-	-	-	2.50	1.34	1.22	1.17
6.0	-	-	-	-	-	-	-	-	-	-	-	-	-	3.00	2.14	2.00
6.5	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	3.25

NOTE: The tolerance for the depth is 0.01mm which is about the maximum you can achieve on a 3018 router.

How to select the right bit

This has nothing to do with speeds feeds etc. but is a little more practical before you try and determine what speed and feed to run it at.

What do you have?

A very important consideration, if you don't have it you can't use it! *OK you could always buy some more!*

Bit Size

What is the width of the smallest detail you are leaving? This will vary if you are using both finishing and roughing passes.

Imagine you are carving letters in relief, the finishing bit diameter must be able to fit in the smallest gap between the letters and into the letters themselves, an A may be 1mm away from other letters but what is the size of the hole you need to make in the top half of the A or the bottom part of an a?

Another thing to consider is internal radii, if you want an internal corner to look square then you are going to need a very small diameter bit, it's never going to be precise as you cannot cut a square internal angle with a round bit or a vertical surface with a V or tapered bit.

Bit Shape

Sometimes obvious, if you want a V shaped groove, well use a V bit! It's sometimes more subtle though. For finishing a 3D carving then a round nose or ball end is a good choice, they are good at finishing smooth curves, but not very good, in fact rubbish, at giving a good definition to straight angles between surfaces.

What are you doing?

Seemingly obvious but still has to be thought about. Even if V carving with a V Bit there are still the questions of what V Angle do you want the finished work to be, how deep will the cut be....

The material you are using

This makes a lot of difference! Trying to cut and engrave aluminium is very different from Pine.

Enjoy!