Router Bit Basics For CNC

A Compilation of the Wisdom and Knowledge of ShopBot Forum Members

Ver. 3.5 May 2012

Router Bit Basics

The challenge to to create an article on router bits was precipitated by questions asked on the ShopBot forum by newbie and experienced users alike.   What do I do with the different bits that came in the ShopBot starter kit? Can I use the Bits I found in a garage sale down the street? They were really cheap! Why is my bit smoking? It feels (felt) sharp? I don't understand what you guys are talking about… Compression, chip load, down cut, onion skin and more. Where can I learn about all this? I'm to embarrassed to ask…

It is my hope that this can be a resource for educators, hobbyists, and professionals alike. I intend that no previous knowledge is assumed. And If a topic is not thoroughly covered, links to more information will be given. The hardest part about writing this article was knowing when to stop… It seems that new topics never end! This article was made possible by the contribution of ShopBot forum members practical knowledge and wisdom.

The terms below are just a quick glimpse of what's to come!

Also: Links to valuable resources.

Router motor vs. Spindle	Spiral	Plunge end
Collets	Flutes	FEM
CNC	X,Y,Z Axis	Ball Nose
HP	Spoil Board	Ball End
RPM	Chip Load	Core Box
ATC	HSS	"O" Flute
Adapter Sleeve	SC Tipped	V-Bit
Shank	Shear	V-Carve
CED	Up cut, Down Cut	Taper
SHK	Compression	Climb Cut
OAL	Mortise	AND MORE…

A router bit is the business end of a router or spindle. Its purpose in life is to remove material. Traditionally, routers were a hand held wood shop power tool that was guided by templates or ball bearing pilots and moved manually around or across the project.

Router
Bit
Collet

Traditional hand held router
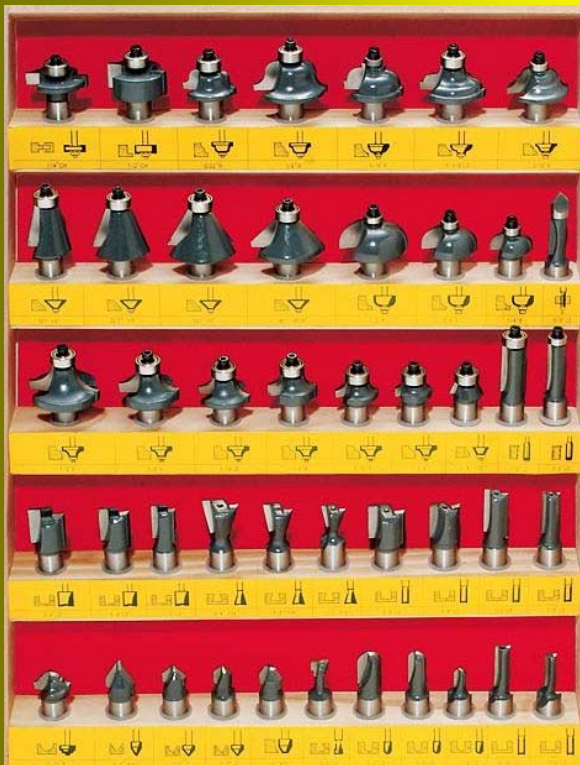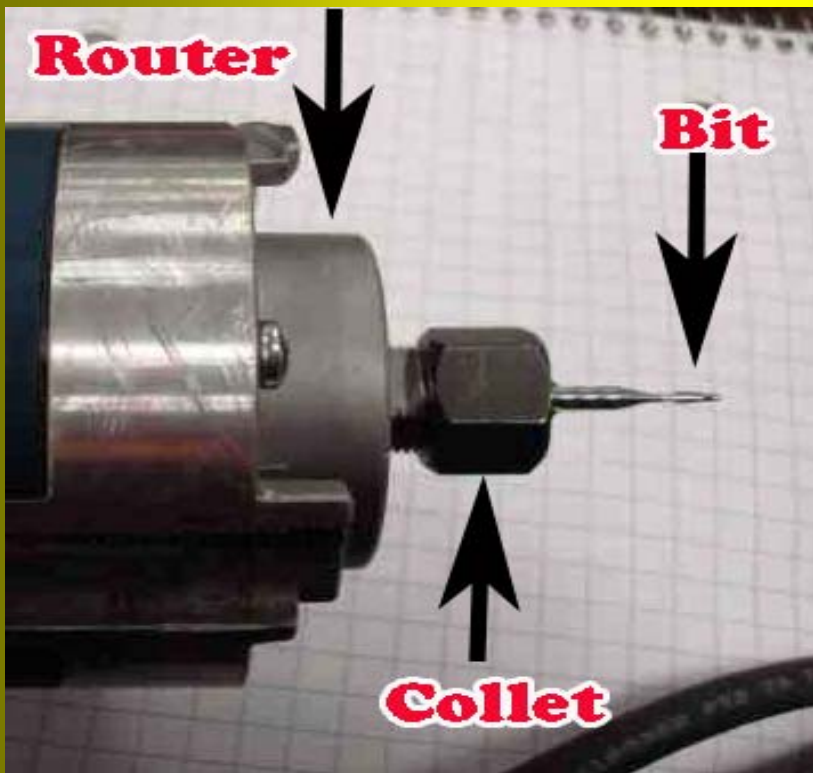
Typical Non-CNC Router Bit Set

Typically, these bits were shaped to give a rounded, coved, beveled, beaded, ogee, bull nose (and more) shape to a edge. Some brave souls even used them to do dados and rabbets, a job usually left to the table saw. Today, in the 21st century... we see routers guided by **CNC** (Computer Numeric Control) machines. Many of the procedures formerly done on the table saw are now being done by routers, plus a far wider range of materials are cut on CNC routers than ever saw the inside of a traditional woodshop. When the router is computer controlled... very complex cuts are possible...even 3D carvings, formally the province of the wood carver or artisan. The range of work and materials cut by router bits has grown exponentially and so have the types of router bits. We old timers used to go to *Sears* and get THE router bit we needed. Now we have so many options that the choice isn't always clear. Hopefully this article will help.

A CNC Router uses a motor to rotate the Router Bit. Here are two common types of motors…

The Spindle

- Quieter!
- More horse power
- Precision bearings (smoother cuts)
- Commercial quality (long life)
- Variable speed, controllable within software

The hand held **Router Motor**

- Less Expensive
- You can get replacements at your local tool store…
- Upgradeable

Seldom seen: the air router motor

The speed at which a Router spins the bit is called its **RPM** (revolutions per minute). Every bit has an ideal range of speed or RPMs it needs to operate within. The ability to change the routers' RPM is essential to good bit life and a good finish. The #75182 Porter Cable router motor has 5 speeds ranging from 10,000 - 21,000. A Spindle has infinitely variable speeds ranging from around 5,000 – 25,000. In either case the speeds available are not slow enough for machining of metals, however with proper bits, alloys and cooling methods some people experience success with non ferrous metals.

The strength or power of a router is rated in **HP** (horse power) Generally more is better… The harder you plan to work the machine (commercial environment like a cabinet shop) the more HP you will want. Lighter work loads (hobby, prototype, or 3D) call for a smaller motor. **NOTE: The method of determining HP is different for spindles and for router motors, the two are not directly comparable**.

ShopBot Tools manufactures a full line of CNC routers. Their product line includes a full range of bed sizes, accessories and motor offerings to compliment every need and pocketbook. www.shopbottools.com

Collet: A router bit is held in the router with a collet. A collet is designed to hold a bit with much more force than a typical drill chuck. To do this its grip range is limited to just a few thousandths of an inch. The shank of a router bit must be accurately manufactured to fit a standard collet and keep it from slipping, or damaging the collet.

Simple collets that come with most routers are operated with two hand held wrenches. It is necessary to have a collet for every bit shank diameter you use, 1/4" and 1/2" and less common 1/8". These collets do wear out (about 500 hrs) and are in the $25 range. Collets for high end tools (spindles) are designed to be switched (changed) rapidly, some even with **ATC** (automatic tool changers). You need a collet for every bit in your ATC holder. These collet/holder combos can run to the $100 - $ 200 range. When they wear out just the internal jaws are replaced.

| Typical hand held router collet/nut	Typical Spindle collet and nut	Typical Spindle ATC tool holder

Caution: Keep collets clean, a piece of debris or dust between the collet and bit can cause the bit to spin elliptically, harming bit, part or even operator.

At least one manufacture makes an after market precision collet set for the Porter Cable router motor. The nut and collets of this set snap together.

Collets Cont.

Bushings that adapt a collet from one size to an intermediate one received mixed reviews by ShopBot Forum members. Several members reported bits slipping in the adapter under use and very awkward moments while trying to assemble adapter and bit while using two wrenches
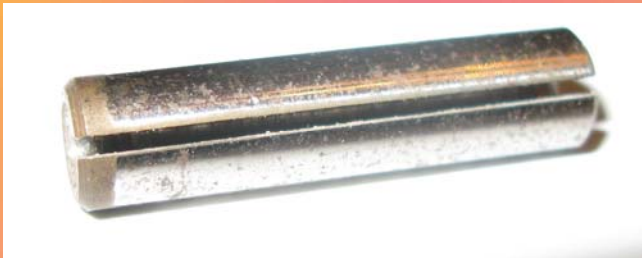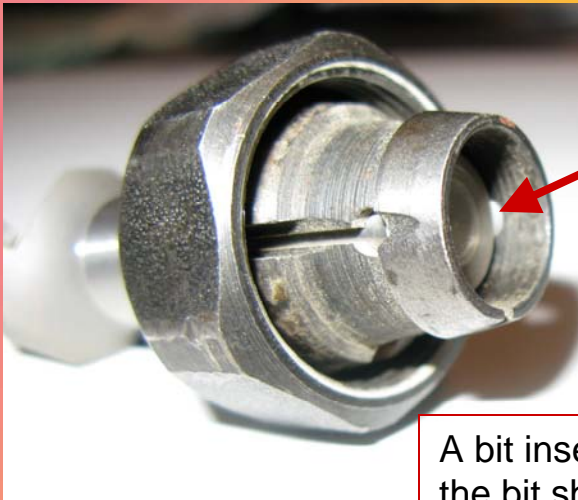

1/4" X 1/8" adapter sleeve

The best solution is to find a collet the right size.

Router bits must be inserted full depth into the collet, failing to do this could result in the bit slipping in the collet causing damage to the bit, collet, product or even the operator.
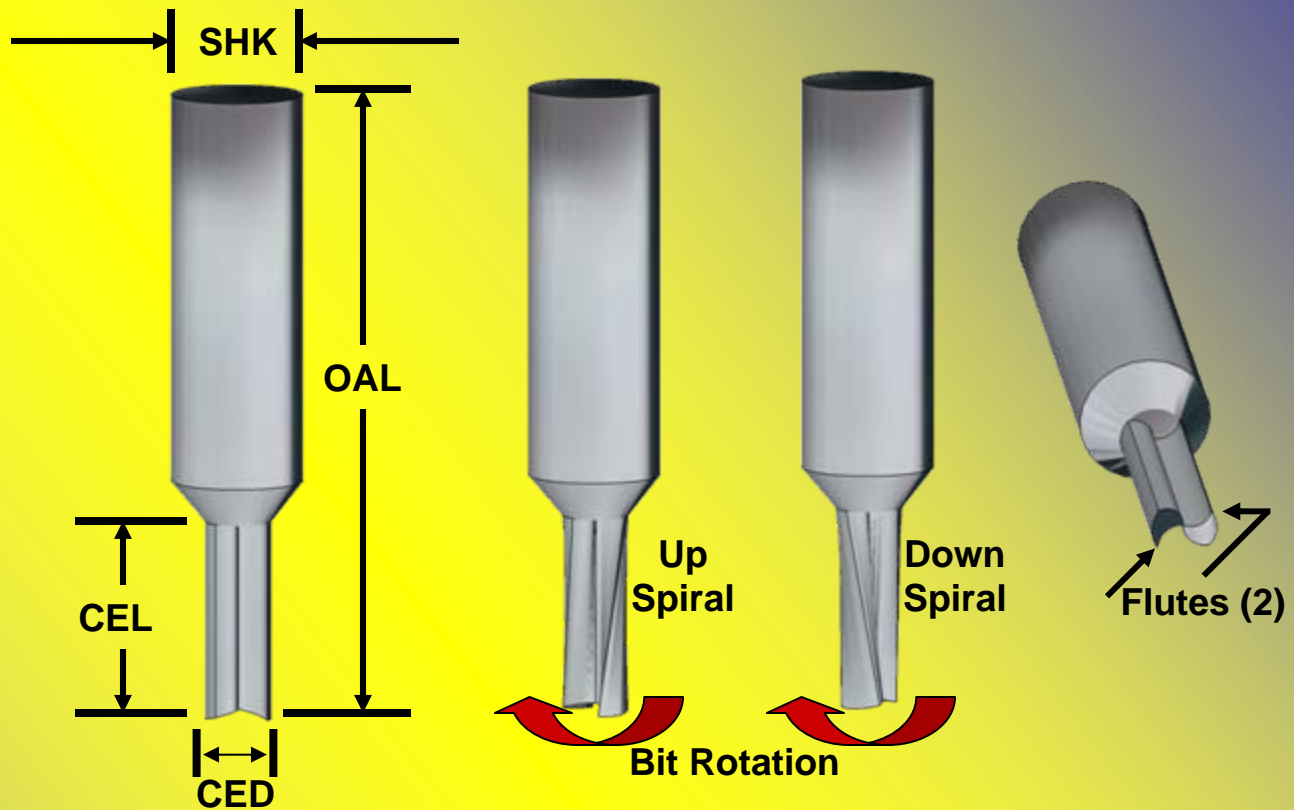

Improper

A bit inserted full depth allows the collet to grip the bit shank properly and develop its full designed holding force

If your bit is to short… Not inserting it fully into the collet is a bad decision!

Typical Router Bit

CED – *cutting edge diameter* or the width of the cut the tool will make through the work piece. This dimension can be in inches or MM. common inch sizes for straight bits are 1/16",1/8" 3/16", 1/4", 3/8", 1/2" .

CEL – *cutting edge length* is the maximum thickness of material it can cut. The CEL has nothing to do with the length of the bit, a 6" long bit might only be sharpened the last 1/2" giving it a 1/2" CEL. A bit with a CEL of 1/2" *MAY* be able to step cut deeper than 1/2" but it is still a CEL of 1/2"

SHK – *shank diameter* This is the diameter of the body of the bit where it goes into the collet. SHK and CED my be the same but not always. A 1/8" bit may have a SHK of 1/4" This would mean that the CED will be 1/8" for the CEL length and then step up to the SHK diameter. Common fractional SHK sizes are 1/4" and 1/2". Collets and adapters are readily available for 1/8" SHK bits. Not as common, spindle collets are available for 3/4" bits. **A larger diameter bit will deflect less and give a smoother, more accurate cut**, but may limit how closely the parts can be nested. (arranged on material)

OAL – *overall length* This is the total length of the tool from end to end. A bit will typically go into the collet 1" or more...so a bit with a OAL of 2¼" will only extend a little more than 1" out of the collet. This would be a poor choice for cutting 1¼" material! Long OAL bits (4-6") are available, but are typically used for cutting foam where the cutting loads are light. Also, remember that the length that the bit is out of the collet reduces your usable max Z height that amount.

Router →

Collet →

Bit →

Z Axis

Product

Y Axis

Spoil board or Bed

X Axis
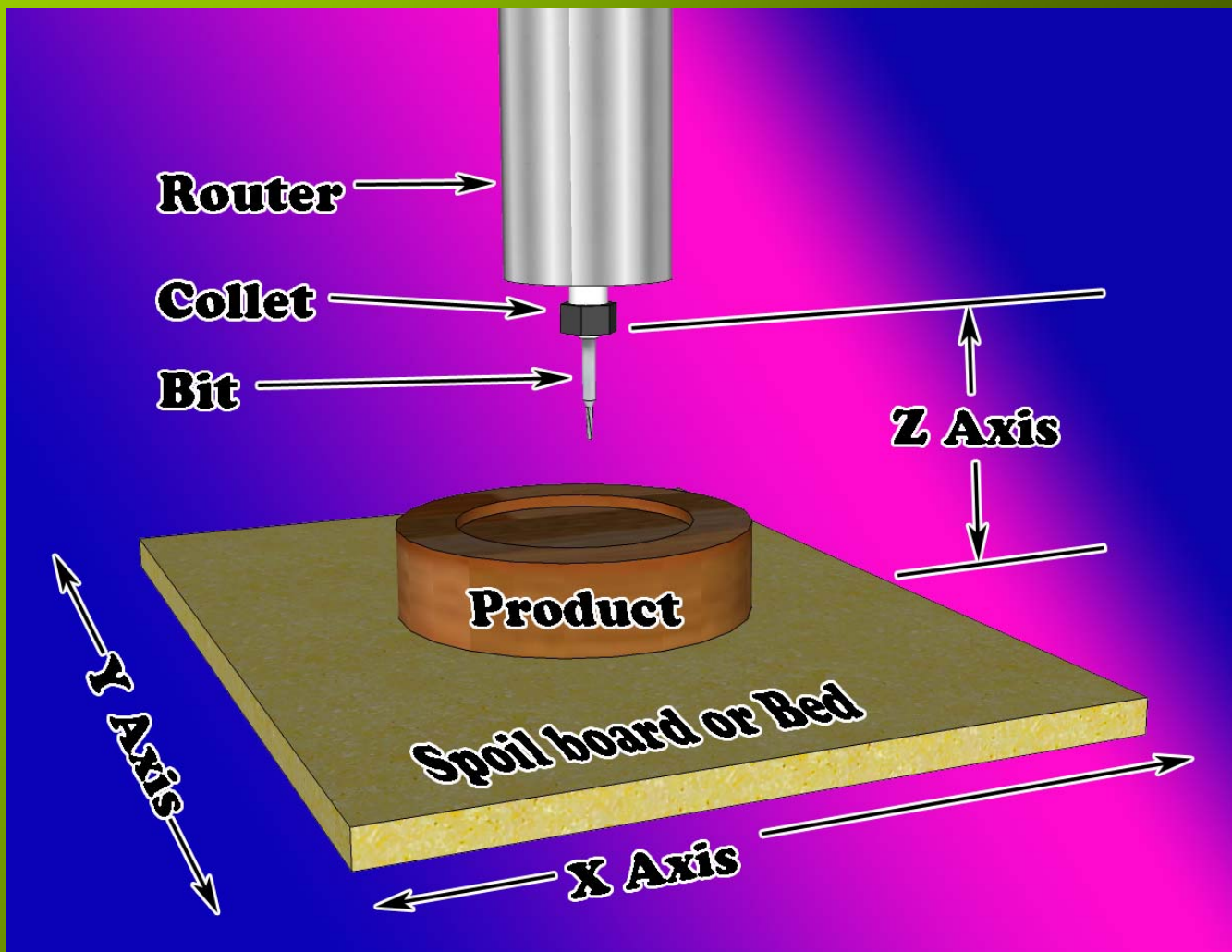
The distance the router bit is out of the collet reduces the usable Z axis that amount

Chipload:
This is simply the thickness of a chip which is formed during the machining of material. Chipload is important because the proper size chip will carry away heat, promoting long tool life. When the chip is too small, heat is transferred to the cutting tool causing premature bit failure. Too high of a chipload will cause poor edge finish, and transfer cutting load or thrust to the part, possibly causing it to move. **A bit in good condition and running at recommended loads will be at room temperature when a cut is finished.**
The ShopBot software has a chipload calculator to help you determine spindle rpm and move speeds for best bit life and finish. Bit manufactures also have specific formulas for each of their types of bits.

Some industries have different definitions for chipload...

Chip Load Calculations

CHIPLOAD	¼ inch diameter	½ inch diameter
		Values in inches/flute
Softwood	.002-.020	.003-.030
Hardwood	.005-.020	.005-.030
MDF	.005-.020	.005-.030
Plywood	.005-.020	.005-.030
Laminated Chipboard	.008-.020	.010-.030
Soft Plastic	.004-.012	.006-.014
Hard Plastic	.006-.012	.008-.016
Aluminum	.003-.010	.004-.012

Note: These Chip Loads are for a tool path depth 1X the bit diameter. For 2X Dia. depth of Cut **(DOC),** reduce the load by 25%, 50% for 3XDia.

Chip Load Calculator

Proper chip load is essential for good cut quality and cutter longevity. Set the indicator below to the parameter you are solving for and input all other values before clicking Calculate.

Units
- ⦿ Inches   ◯ mm

Chip Load Help

- ⦿ 2.933 — Cutting Speed (feed rate) in Inches/Sec
- ◯ 11000 — Spindle/Router RPM
- ◯ .008 — **CHIP LOAD in Inches/Flute**
- 2 ▼ — # of Edges (flutes) on Bit

Calculate

The above chip-load ranges from the SB3 help file are very general. Specific loads can be obtained from your router bit manufacturer.

The Chip Load Calculator on the left is located in the SB3 control software under [Tools] … [Chip Load Calculator].

To use… simply choose the units you want and the value you're trying to find. Supply the other 2 values and the third is calculated for you.

Since this is a Basic article… I'm not going to recommend that you go to www.onsrud.com/xdoc/feedspeeds where you'll find 13+ pages of specific chip loads for specific situations.
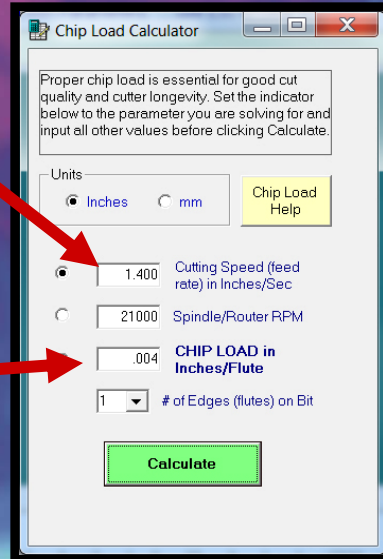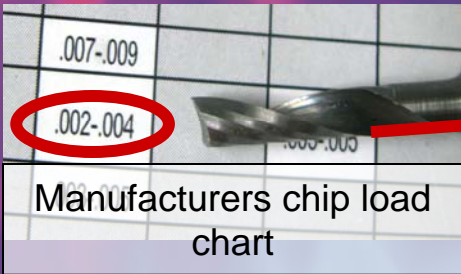www.onsrud.com/xlink/xtdLkTechArticles is a source of interesting tech articles, many dealing with routing plastics

An Example…

Here's an example of how chip load calculating works… I have a 1/8" single "O" flute bit that I want to cut a hardwood profile with. I look it up in the bit manufacturers charts and see that the chip load per flute is .002" -.004". I go with the high side .004"… and use the routers top speed of 21,000 and calculate a result of 1.4" per second.
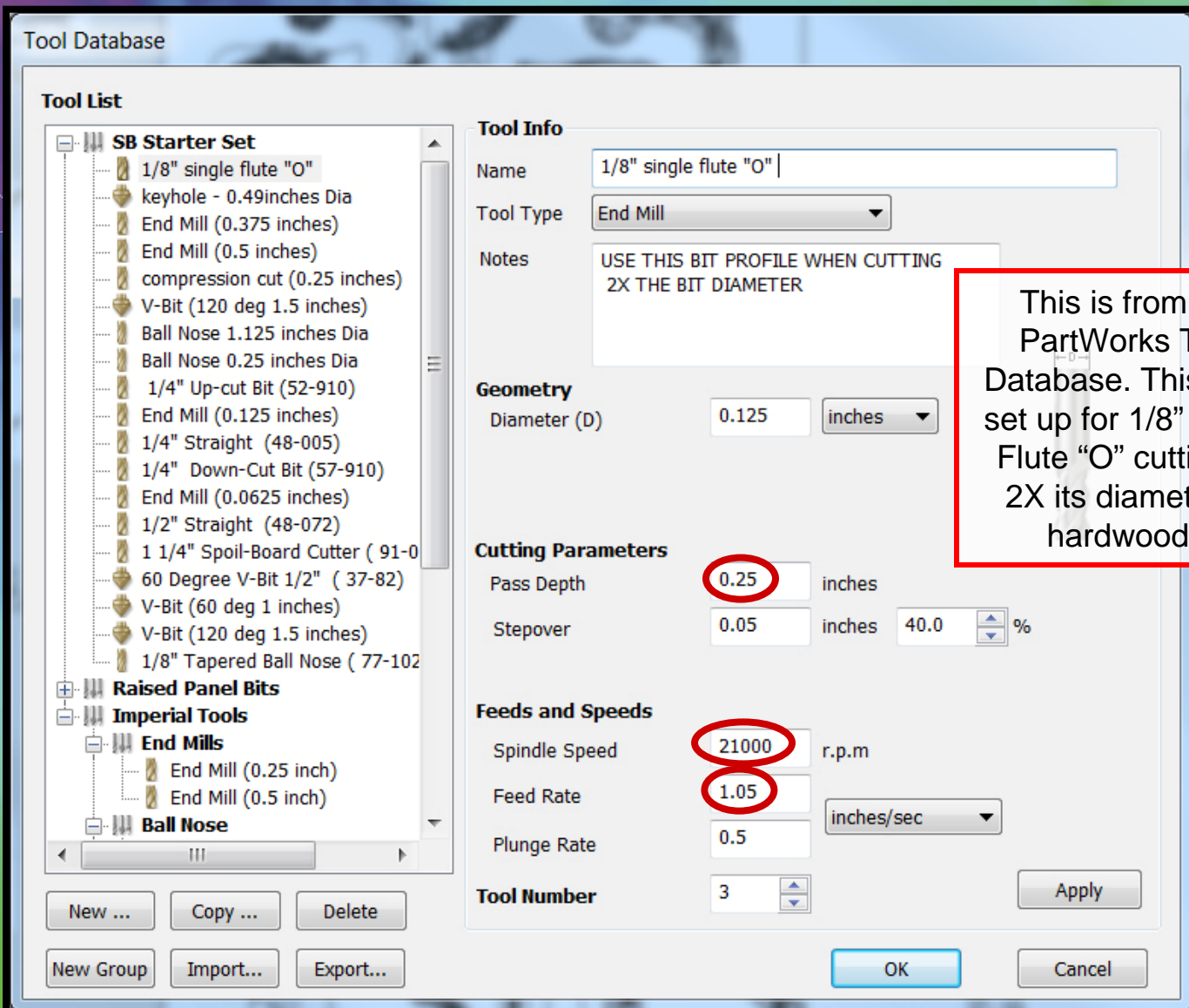
1.400 Cutting Speed (feed rate) in Inches/Sec

Result of calculation

.007-.009
.002-.004
.005-.005

Manufacturers chip load chart

Chip Load Calculator

Proper chip load is essential for good cut quality and cutter longevity. Set the indicator below to the parameter you are solving for and input all other values before clicking Calculate.

Units
● Inches   ○ mm

Chip Load Help

1.400   Cutting Speed (feed rate) in Inches/Sec
21000   Spindle/Router RPM
.004   CHIP LOAD in Inches/Flute
1 ▼   # of Edges (flutes) on Bit

Calculate

Because I want to cut at .25" (or 2X bit **CED**.) per pass I have to reduce the speed by 25%, or to 1.05in/sec.

If you don't have a bit in your tool database with these parameters, you create one. You may have several tools defined for the same bit but defined for different materials or pass depths.

Tool Database

Tool List

SB Starter Set
- 1/8" single flute "O"
- keyhole - 0.49inches Dia
- End Mill (0.375 inches)
- End Mill (0.5 inches)
- compression cut (0.25 inches)
- V-Bit (120 deg 1.5 inches)
- Ball Nose 1.125 inches Dia
- Ball Nose 0.25 inches Dia
- 1/4" Up-cut Bit (52-910)
- End Mill (0.125 inches)
- 1/4" Straight (48-005)
- 1/4" Down-Cut Bit (57-910)
- End Mill (0.0625 inches)
- 1/2" Straight (48-072)
- 1 1/4" Spoil-Board Cutter (91-0
- 60 Degree V-Bit 1/2" (37-82)
- V-Bit (60 deg 1 inches)
- V-Bit (120 deg 1.5 inches)
- 1/8" Tapered Ball Nose (77-102
Raised Panel Bits
Imperial Tools
- End Mills
  - End Mill (0.25 inch)
  - End Mill (0.5 inch)
- Ball Nose

Tool Info

Name   1/8" single flute "O"

Tool Type   End Mill

Notes   USE THIS BIT PROFILE WHEN CUTTING 2X THE BIT DIAMETER

Geometry
Diameter (D)   0.125   inches

Cutting Parameters
Pass Depth   0.25   inches
Stepover   0.05   inches   40.0   %

Feeds and Speeds
Spindle Speed   21000   r.p.m
Feed Rate   1.05   inches/sec
Plunge Rate   0.5

Tool Number   3

New …   Copy …   Delete
New Group   Import…   Export…
OK   Cancel

Apply

This is from the PartWorks Tool Database. This bit is set up for 1/8" single Flute "O" cutting at 2X its diameter in hardwoods.
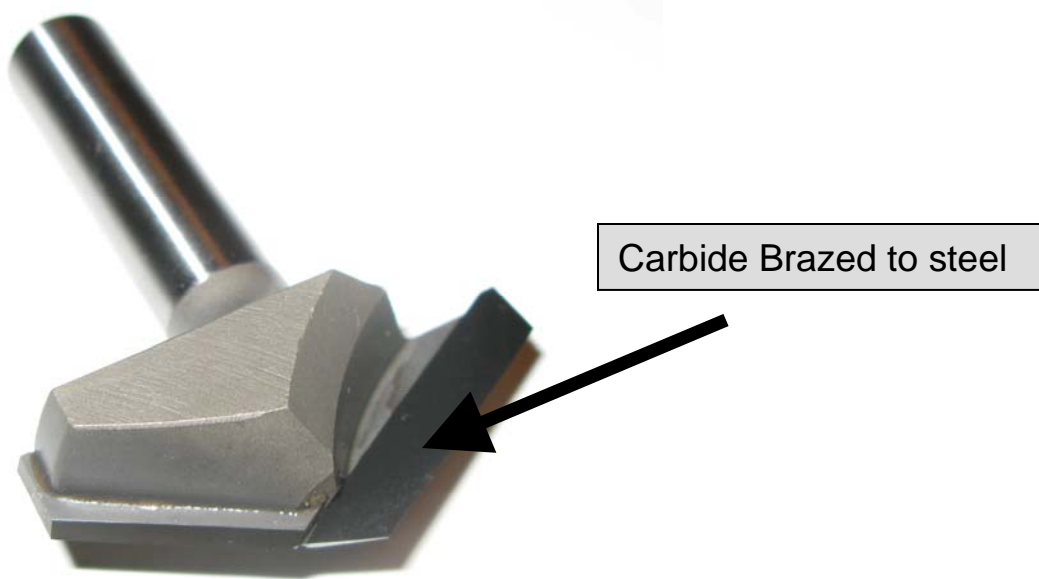
Router Bit Materials

HSS – *High Speed Steel* A tool steel far superior to carbon steel when used for drill bits and cutting tools. A HSS drill bit can drill a hole much faster than a carbon steel bit, hence its name. Quite the innovation in the 1940's! We've moved past this type of tool steel today, HSS bits are only suitable for cutting foams. (some disagreement here)

SC – Generally just called *Solid Carbide.* A very tough wear resistant steel. Many forms of Carbide exist (silicon carbide, tungsten carbide, Beryllium carbide, etc.) as do forms with different grain size. Solid carbide router bits can vary from worthless to extreme high quality. Mass marketers can honestly say "solid carbide" and sell you total junk. If it's cheap... there's a reason. Develop a relationship with a supplier you trust! (one note of caution.. SC bits are brittle, I've had them shatter when dropped!)

Carbide Tipped – Carbide is expensive... larger diameter bits are available with chunks of carbide brazed (welded) to a steel body. This reduces the total amount of carbide used to make the bit and hopefully the price. Every thing above applies here. Generally, bits with .1" or more carbide thickness is considered a sharpen-able bit. **Note: a sharpened bit may no longer be its full design diameter**, and will need to be compensated for in the CNC software.


Carbide Brazed to steel

OTHER – There are many other exotic router bit materials available... Diamond, Diamond like, poly crystalline, and more. These bit materials tend to be used in specialty, high volume, and niche markets. I hope your business grows to the point where you need to research and use these types of bits!

Bit Geometry

Not all bits are created equal or for the same purpose or to cut the same materials. Bit geometry can affect its usefulness for a specific purpose. Lets try to sort through some major differences!

Flutes –The Number of flutes refers to the number of cutting edges that are cut into the body of the tool. More flutes increases the strength of the tool, but reduces space for chip flow. You can cut fast with a single flute tool, but the finish of the three flute will be smoother. CNC router bits are commonly available in 1, 2 and less commonly 3 flute configurations. (there are exceptions to every statement)
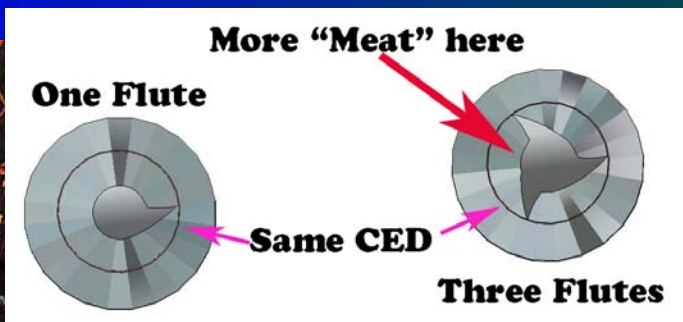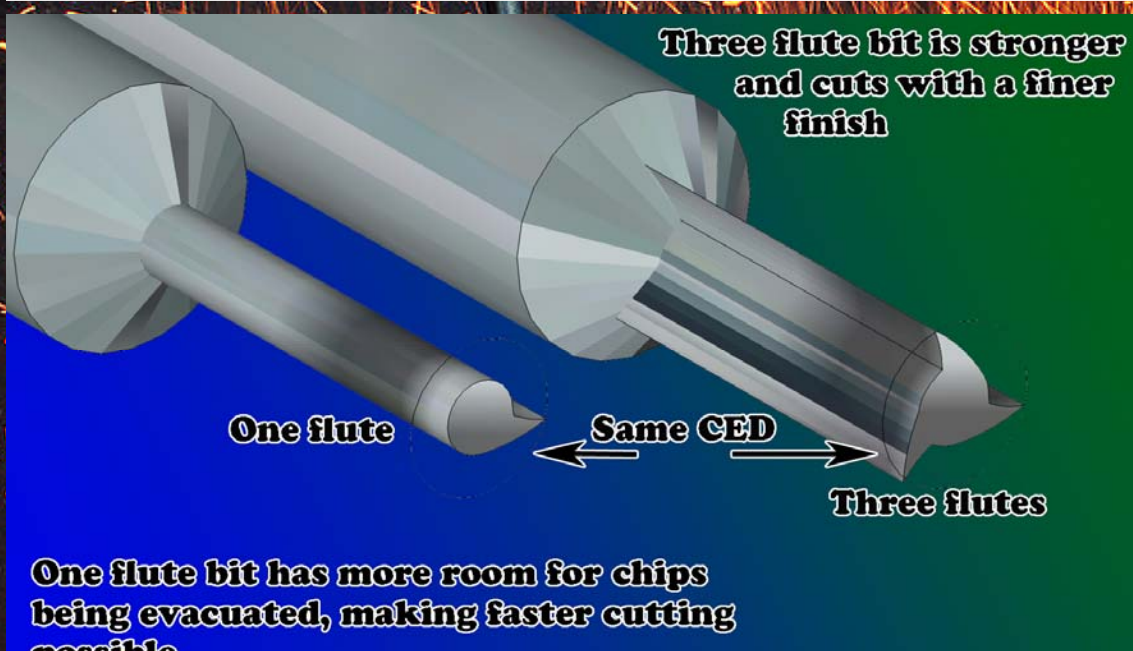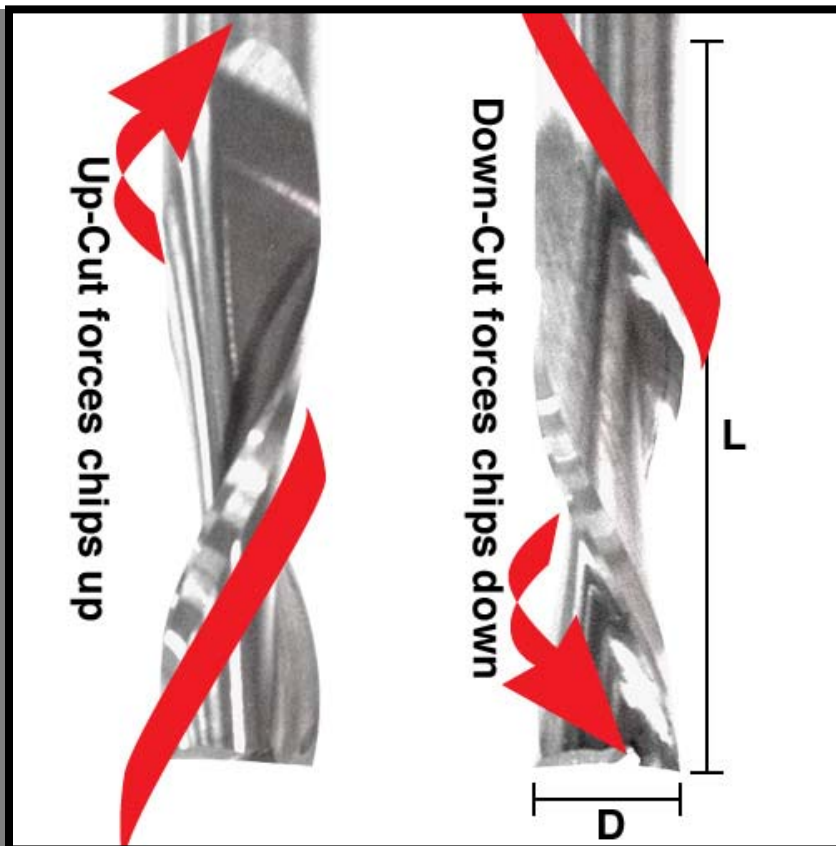

Shear

The flutes of a bit can be shaped (or ground) to obtain different advantages when cutting different materials.

Straight – Straight bits have their cutting edges parallel to the body or shank of the bit. The only advantage I can come up with is cost. It's cheaper to manufacture bits this way. Exotic, expensive CNC grinding equipment is not required to make this old fashioned standby bit. I'm not aware of any brazed or carbide tipped bits that are not straight geometry bits, including shaped bits we'll talk about later.

Spiral – The flutes of this bit are ground (cut) in a helix around the shank or body of the bit. They cut with a shearing action that is smoother and with less vibration than a straight bit. A straight bit has the entire cutting edge of a flute in contact with the material being cut at the same instance in time, while a spiral flute edge has a very small contact area continuously moving or shearing during the rotation of the bit. A spiral bit always has a part of one or more flutes in contact with the material being cut, while a straight bit is only in contact while that particular flute passes by.
There are three major types of spiral bits, each has their advantages and disadvantages for particular uses.


Up Cut – The flutes on this type of spiral bit shear from the bottom up, pulling chips up, This clearing or cleaning action allows for deep cuts with less stress on the tool. The major disadvantage of this geometry is that the up-cut action can lift the part up and off whatever is holding the part on the router table. Also, up cutting action can splinter the top face of veneered or other susceptible materials

Down Cut – The "down-cut" spiral bit flutes are designed to cut from the top surface down, leaving a smooth edge at the surface. It pushes down on the material being cut, and helps hold the material in place on the table surface. Some operators like this bit as it packs the sawdust in the groove being cut, this helps hold parts in position and helps maintain vacuum when that type of hold down system is used. Disadvantages are the packed sawdust is not removed by the dust collector and must be scraped or brushed off manually. Also this bit is not appropriate for thermoplastics as the "packing" of the dust re-welds the parts together. When making "through cuts" the down cutting force can splinter the bottom face of veneered or other susceptible materials

Compression Bit
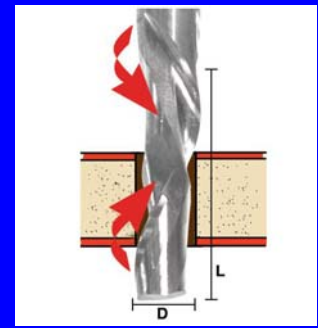
Compression – The third type of spiral bit is a hybrid design of the up and down cut bit. This bit has a up cut portion on the lower part of the bit and down cut on the remainder of its CEL. Material being cut is augured downward from the top of the material and upward from the bottom. This compression of cutting forces results in a clean, splinter free top and bottom face. A special type of compression bit is the **Mortise** compression. This style has a much shorter up-cut section than the standard compression bit and is used for grooves and dados as shallow as ¼" while still having a splinter free top surface.

There are other variations of these three types, one involves the angle of the helix, fine tuning the bit for particular materials.

Bit Ends

Some router bits are designed for plunge cutting and some are not.

Standard End – bits will plunge cut nicely, these bits can be identified by the swallow tail style profile of the end of the bit. This bit is the right choice when cutting profiles. The disadvantage of this style bit is it leaves marks or swirls when pocket cutting.

FEM – (flat end mill) this style of end geometry is designed to cut a smooth bottom surface when pocketing. The disadvantage of this style is it need to be ramped into its cut. Forgetting to do this will result in a burned bit. The end profile of this bit is flat.

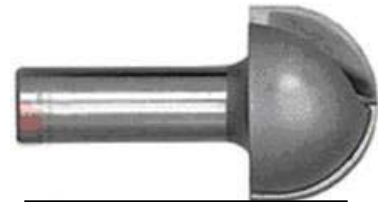FEM end bit

Standard plunge end bit

Ball – Ball bits are used to give a radius to the inside corners of pocket cuts. When used for 3D projects with low percentage stepovers, the radii help blend each path together, resulting in less finish work. A ball nose bit is a poor choice for profile cuts as the bit will have to go past the bottom of the material the radius of the bit into the spoilboard to get a clean profile.

Ball-nose bit	Ball-end bit	Core Box bit

Other Geometry

"O" flute: This is a relatively new wrinkle in bit geometry. The gullets of the O flute bit are designed to "roll" the shavings into a ball. The purpose of this property is to compact the shavings and make them easier to throw clear of the bit. This property is especially important when cutting materials that create lightweight shavings... like plastics.

Single flute "O" Bit

"O" Flute Curling Action

**Heavier chips are easier to throw from the cutting area**
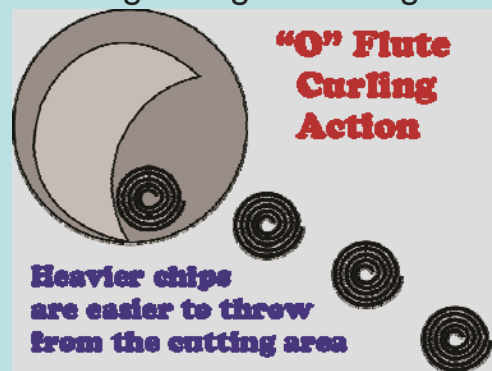
Shaped bits

This category of bits will include any bit that is not intended to cut a straight profile. The most common for CNC users will be the V-bits, next the plunge shape cutters for faux raised panel doors followed by any number of specialty bits (like toy train track bits). Somewhere in this discussion it needs to be said... **NEVER use a ball bearing or piloted bit in a CNC router.** The computer/CNC equipment is "piloting" the bit, using these will result in interference between the bit and the equipment and yield lost steps, damaged parts or worse!

V-bits – Come in many flavors, and are typically used for signs and V-carving (a topic worthy of its own write-up) some things that affect how they are used are:

Angle: measured as the included angle, v-bits are commonly available in 30°, 45°, 60°, 90°, and 120°. the tip of a V-bit can come to a true point, a small radius or a flat.

Diameter: measured perpendicular to the shank, this is the width of a full depth cut at the surface of the material.

V-carving… A technique that uses V-bits

V-Bit

Vector on surface of material

The Path a V-bit tool tip takes when V-Carving a closed vector
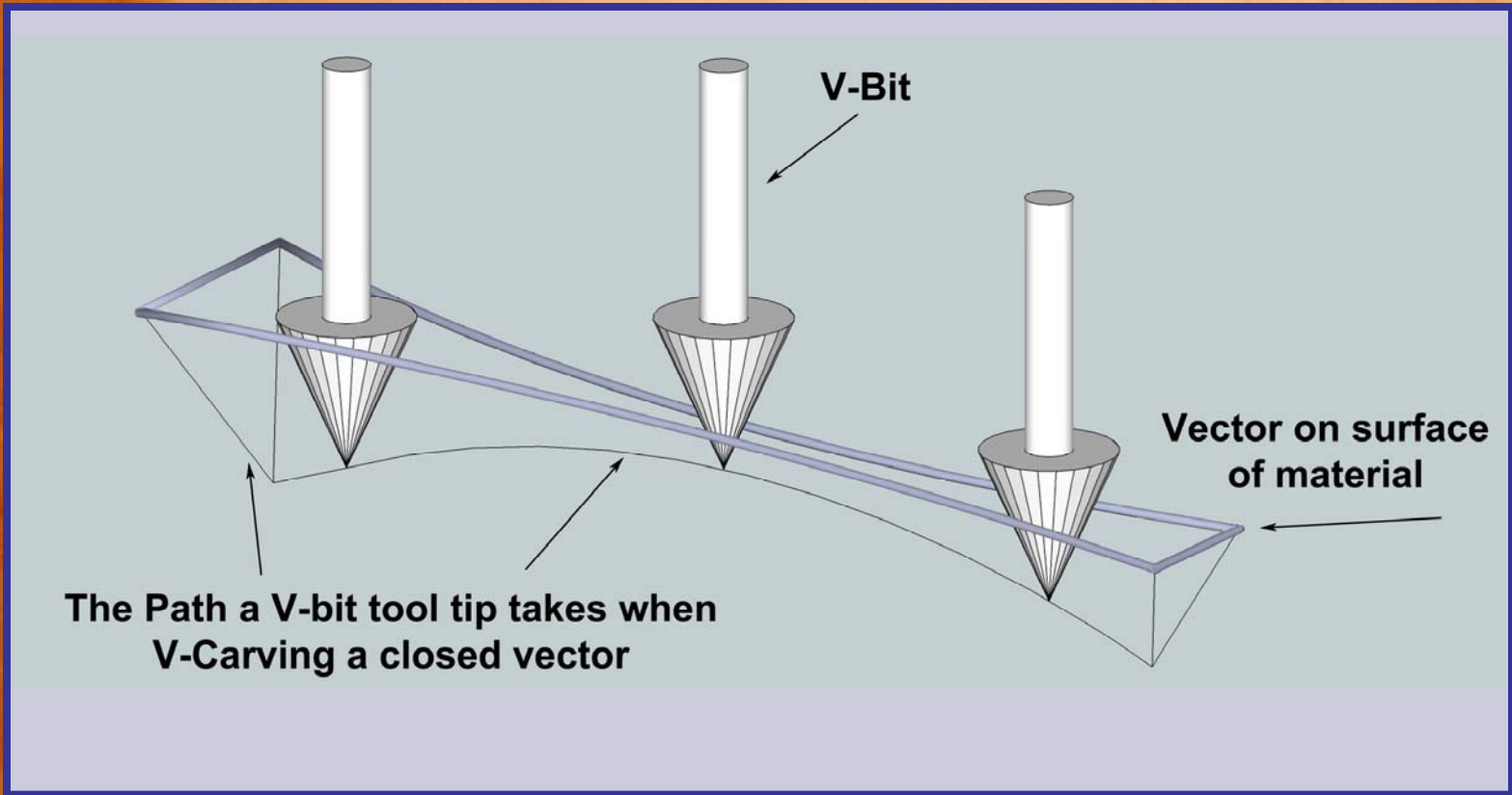
V carving is typically known as a method used by sign makers to create sharp corners while using round, cone shaped router bits (known as V-bits). This is accomplished by causing the bit to rise out of the material being cut as it approaches a corner, effectively continuously reducing the bit diameter until it is a single point. The visual effect is that of a square corner, making lettering crisp, clear and easy to read.

60° Sign Carving Bit

30° Engraving Bit
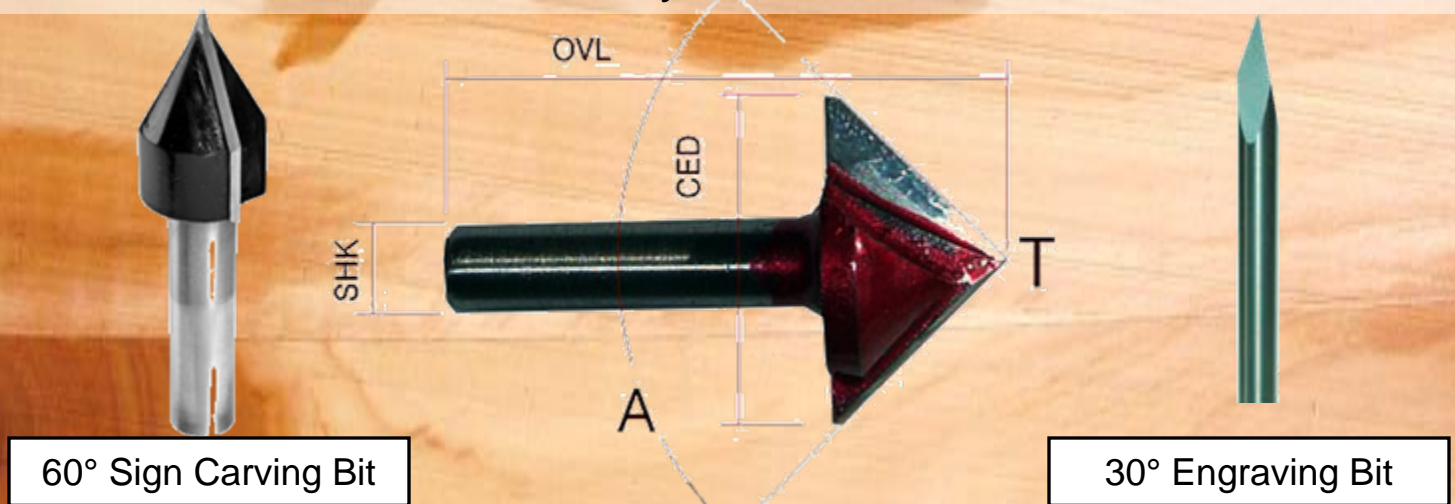
Form Bits – This is a wide open category... the only restriction I'm aware of is it must be able to plunge. Some of these bits remind you of the ogee and radius combo bits used with a bearing pilot... only there's a flat or radius or point where you might expect a pilot. often these bits are used for decorative grooves and in combinations to create faux raised panel doors in MDF.
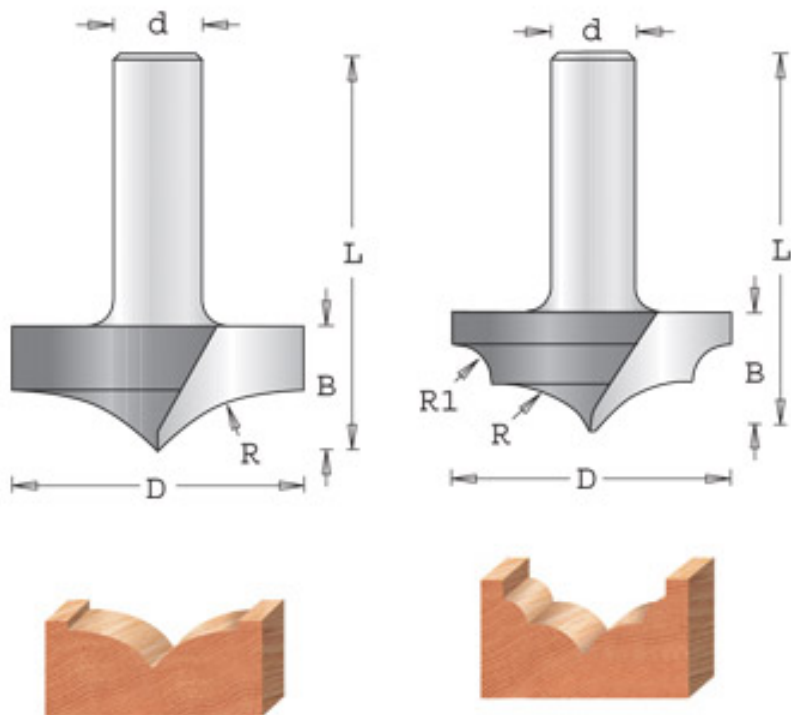
Some bits, particularly form bits can be found that use replaceable inserts.

Spoil Board Cutters – A specialty bit used to re-surface a spoil board. Not a plunge style bit, but designed for a smooth bottom cut, generally limited to 1/16" **DOC.** They generally have a small radius to help smooth the transition between step-overs. Also available with replaceable insert tooling.

Ball nose Taper –Typically these bits have a 1° taper on each side. The feature of these bits is that a 1/8" bit can have the reach and strength of a 1/4" bit. They are generally used for 3D detail carving. When used with about a 7-10% step over, the ball end gives a very smooth finish, blending the cutter paths together.

Practical suggestions

Climb vs. Conventional cutting

Climb cutting

CW

CCW
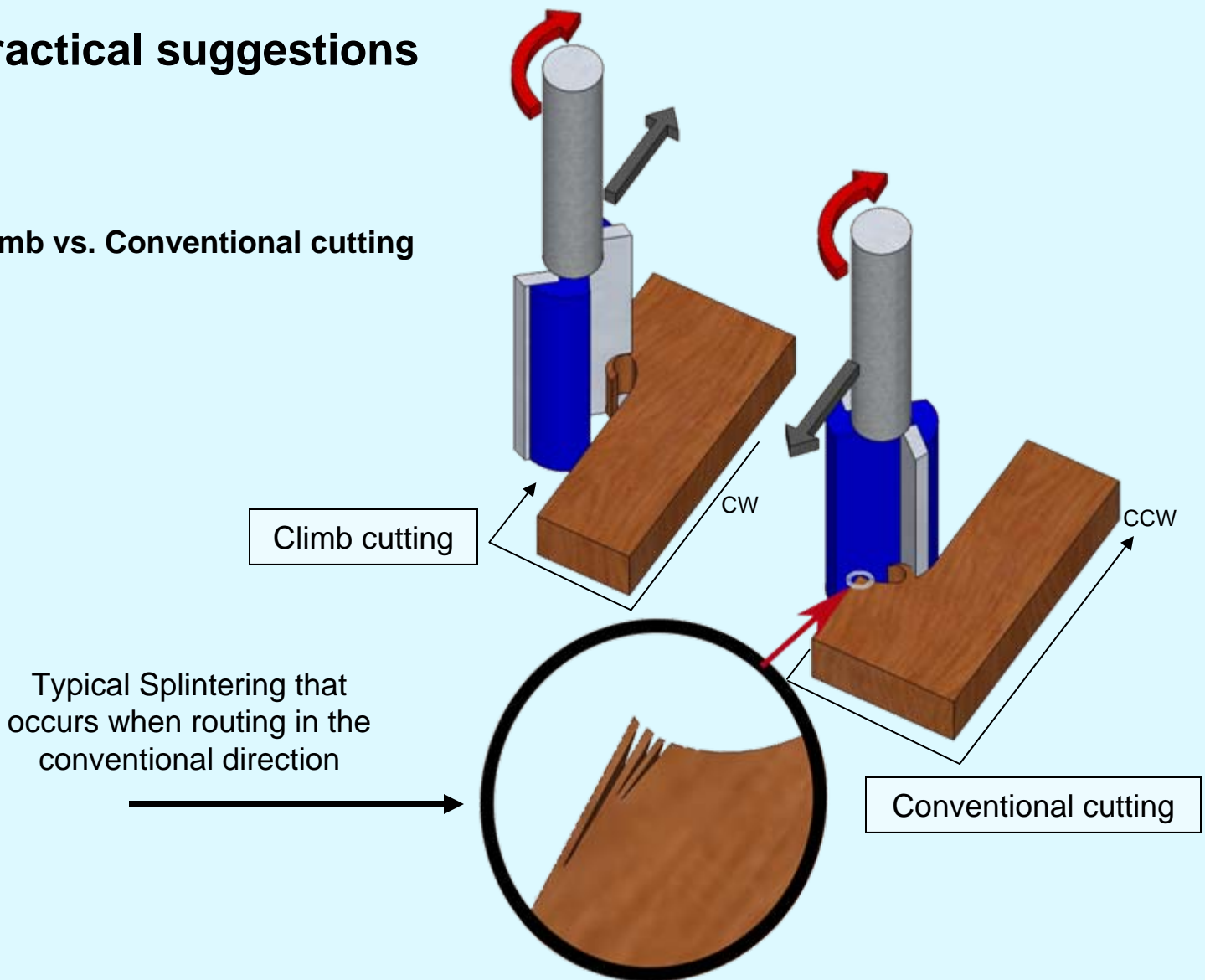
Typical Splintering that occurs when routing in the conventional direction

Conventional cutting

When cutting the perimeter of an object … say a rectangle, you can choose to cut it in a clockwise or counter clockwise direction. The **CCW** direction is known as conventional, while the **CW** direction is known as a climb cut. In an effort to make something simple confusing… these definitions reverse when cutting the interior of an object!

The illustration above shows the splintering that's typical when cutting conventional. This end grain splintering can be avoided by climb cutting…HOWEVER…when climb cutting the chip is formed from max size tapering off to minimum size. This and the forces created by the router bit trying to "climb" out of the cut result in a finish that's not as smooth as a conventional cut.

The solution to this dilemma is to cut to each methods strength. Climb cut an object allowing .01" - .015" then make a final pass, full depth without allowance in a conventional direction. This method will often result in the best possible finish on a part.

NOTE: When the finish of a part is poor, check the scrap. If the scrap finish is better, change the direction of the tool path.

Practical suggestions...
Putting It All Together…

Now that you know the basics about router bits, It's time to put that knowledge to use! After you have chosen the material for the job and the correct bit and chip loads, some people feel the hardest part of CNC work is next… Holding the material in place. There are several techniques for holding material, Vacuum, double stick tapes, cams, screws, tabs, and plastic nails to name a few. Hold down methods are outside the scope of this article, but there is a cutting strategy that many of us use to help whatever method of hold down you use. It's called the "**Onion Skin Strategy**".

When using this method you make an initial cut in the climb direction cutting away from the finished product about .005 --.010 and about .015--.020 less than the total material thickness. You then make a finish pass at full depth in the conventional direction and on the product cut line. This method leaves a continuous "tab" in place while the heavy cutting forces are being realized and then makes the final cut full depth eliminating the "water" or step marks of multi pass cuts. The Example below assumes .5" material.

Rough or first path of an Onion skin Strategy   **Finish or Second path of an Onion skin Strategy**

Practical suggestions cont.

In the battle between router bits and clamps … the clamp wins every time!

Don't forget to turn the router on!

Use the shortest bit that will do the job…Less deflection, fewer broken bits

Cutting too slow will burn the bit or worse…

Using a 1/8" downshear bit to drill holes in 3/4" plywood leads to bad, bad things

I chose the wrong tool for a job and set fire to the table on my ShopBot. It was a heavy smoke, flameless and very hot fire. Think blast furnace. Fortunately, I caught it in time. Had I left the building for even a minute, I could have lost the whole shop. The damage shown only took about 10 seconds to happen. If you have a ShopBot or other CNC router, don't EVER leave the machine alone while it's running!

http://www.flickr.com/photos/33471283@N05/5340963973/
Gene Buckle

"I'm left handed… I thought maybe those LH bits were for me to use"

Practical suggestions cont.

Just like a baby… Feed a screaming bit. Usually a bit that's "screaming" wants a greater chip load.

Just like you…. A router bit shouldn't smoke, it's bad for its health. A smokin' bit wants a greater chip load.

Keep it clean… A bit with built up tree resins, tars and sap are susceptible to pitting form the acids in those compounds. You can use solvents and a fine brass brush on them. Forum members also suggest a toothbrush.

A published chipload isn't gospel… Start out with the recommended chipload and increase the feedrate until finish starts to deteriorate, then back off 10%. Now decrease the RPM and repeat the same steps. This process also works when you don't know the manufacturers recommended chipload.

Breakin' em left and right… Even a properly used tool will occasionally break. If you are experiencing multiple tool failures, STOP and ask yourself, Is this the right tool geometry for this job? Is the collet clean and is the bit properly installed? Recheck your feed speeds (those pesky decimal points!) Is the bit too hot to touch? Is the part held securely or is it moving, even a little bit? Are you using the largest diameter, shortest length possible bit for the job? Rethink your DOC (depth of cut).

Resources

Onsrud www.onsrud.com   an endless source of bits and information about bits

Beckwith Decor Products www.beckwithdecor.com an Onsrud rep and fellow ShopBotter

Centurion Tools www.centuriontools.com a bit source forum members trust

Magnate www.magnate.net/index.cfm?event=showCategory&theID=24
        Large "V"-bits and a good selection of shapes

Ekstrom Carlson www.ekstromcarlson.com Spindle repair, router bits, tooling

Cutter-shop www.cutter-shop.com/main.asp?sitepages=feeds_and_speeds
        Generic chip load tables

Edited select posts about router bits from the ShopBot forum

Q So how do you Z-Zero a **diamond coated bit**? Apparently the coating is a very poor conductor.

A Make a brass sleeve with a bottom (like a cup). The sleeve will be pulled up on the bit with a strong rubber band. The Z-Zero clip will be clipped onto the sleeve and Z-Zero will operate as normal. Knowing the thickness of the bottom of the sleeve, you can compensate for the sleeve, or add a "sleeve Z-Zero" routine so the compensation will be automatic.

Q We are cutting small parts out of **6061 aluminum** and have both 1/4 an 1/8 "O" flutes. Both can complete the job. Is there one that you would recommend over the other?

A Use the largest diameter bit that will do the job... less flex, less breakage

Q I need to cut some 1" and thicker **cork** boards. Does anyone have any advice?

A Cork cuts very easily. I use a Onsrud 65-025.

Q I have a partition project in the near future that will need to be cut from 1/2 inch **cherry plywood**. It will need to have a very clean cut since both sides and the cut edges will be visible.

A Thanks everyone, I get it now. I'll get a compression bit.

Q Has anyone tried cutting **brass** in thicknesses up to 1/4" with their ShopBot?

A I cut brass all the time. I regularly cut .090" but I have done up to 3/8". Just get bits designed for non ferrous machining. I run an 1/8" 2 flute bit about .3 -.5 in/sec @ 12k rpm, .02" passes. I find a tapping lubricant to help the cut immensely, and be sure to ramp into every (and I mean every) cut.

Q Is anybody cutting **Kydex** and can share what bit you are using, cut speed and RPMs that you are running?

A I use a single, double or triple flute Onsrud or Behlin spiral-O. Speed will depend on design and bit diameter. RPM in the 13,000-15,000 range

Q I would like to try my hand as some simple **3d carvings**. What's the best general purpose bit to get my feet wet with?

A http://www.beckwithdecor.com/index_files/Cutters.htm

Edited select posts about router bits from the ShopBot forum. Continued…

Q Any advice when cutting **corian**?
A Onsrud has a series of 1/4" single flute O-flute straight bits that are ideal for cutting Corian. You want to use a 10% ramp into your part, cut it in two passes plus a skin for the final pass, rotate your spindle around 8K, cut at 1.5 or so IPS and step down at .25ips. This is a good place to start

Q Where can I find out more info about **cutting plastics**?
A http://www.plasticrouting.com/

Q Can I use a 5mm **line boring bit** in the ShopBot
A This works well. Assuming that you have a spindle brand that does not recommend against numerous plunge actions. The HSD that is most sold with the ShopBot works well for this purpose.
Install a 10mm collet, and making sure that the flat on the shank is lined up well with the collet segments, set your RPM to around 5K RPM. Depending on your material density, try a plunge speeds starting around 1 IPS and increase till you get nice chips and a clean hole. Due to the hard surface of Melamine, it requires a slower plunge than Plywood or MDF.
I have not found peck drilling to be needed, and I have drilled thousands of holes thru 3/4 material using 3mm, 5mm and 10mm boring bits

Q I am trying to make a piece out of **MDF** that has allot of small details. I keep having trouble breaking off some of that detail. Any suggestions for things that I can do to help prevent this?
A If you are buying your MDF at the big box stores you're not getting good enough quality material to do the job.

Q Can I get opinions on feed speed for cutting **MDF**?
A the chipload calculator on the SB3 tools menu shows that you need fast feed speed to keep from burning up your cutters. I cut MDF at 12,000 RPM at a feed rate of 8-ips with a 2-flute downcut spiral 3/8-inch cutter. That gives me a chipload of 0.020, which is about what I want with MDF. Using a 1-flute cutter, I normally run at 5-ips at 15,000 RPM to get the same chipload. My vacuum system keeps me from cutting all of the way through the MDF, but I normally cut -0.730" on 0.750" material.

Q. What's the best way to cut Styrofoam…

Cutting Styrene Foam Board

All Cuts made at 18000 rpm

Conventional cut direction

Climb cut direction

2 ips 2 flute down cut

6 ips 4 flute up cut

6 ips 4 flute up cut

6 ips 2 flute down cut

2 ips 4 flute up cut

Thanks to:

steve knight
Senior Member

A. So the conclusion is 18k and 2ips up cut or down cut and a normal cut, not a climb cut.