888-680-4466 • ShopBotTools.com

Selecting the Right Bit/ Feeds and Speeds Charts

Table of Contents

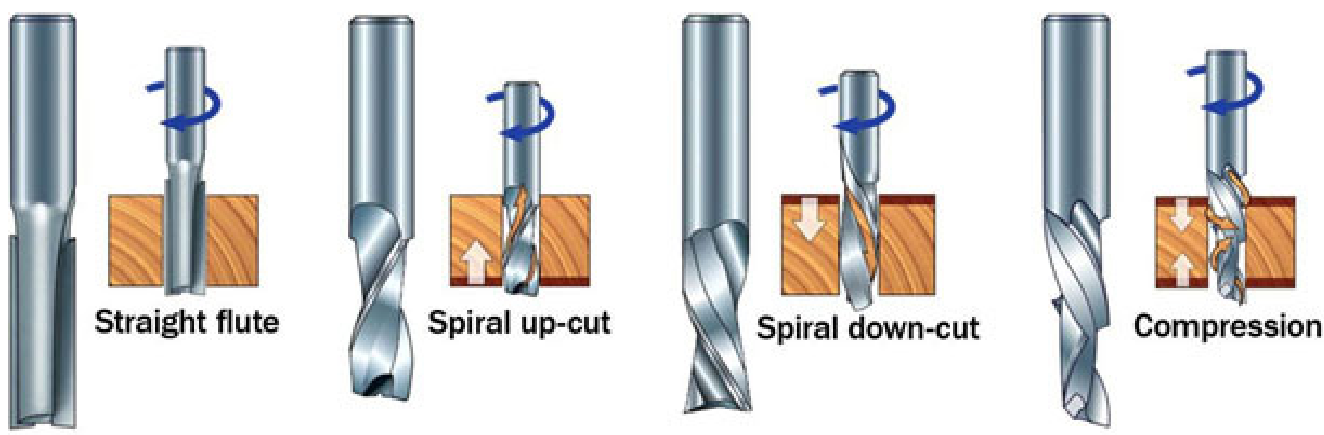
Introduction

A challenge of getting a good CNC cut is in selecting the best bit, best cutting speed (feed rate) and router/spindle RPM (speed of rotation). Bits, feeds, and speeds are a critical part of machining and should be fully understood before deviating from recommended settings. Bits choice is important in chip load, which is a representation of the size of the chips produced during cutting. The goal is to get the maximum chip load possible to increase productivity, reduce heat, and prevent premature dulling. When chip load is too small, bits will get too hot and dull quicker. When chip load is too high, the tool will deflect creating a bad surface finish and, in extreme cases, chip or break the bit.

Selecting the Right Bit for the Job

Bit material: Router bits are made from a variety of materials. The most common are solid carbide, carbide-tipped steel, and high-speed steel. Both solid carbide and carbide- tipped are good choices. We do not recommend using high-speed steel bits as they dull quickly and must be re-sharpened.

Flute type: There are four basic flute types: Straight, spiral up-cut, spiral down-cut, and compression. Each type has its own advantages and disadvantages, which are outlined in the chart below.

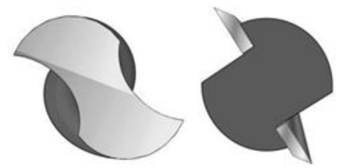


Good edge quality on most materials	May chip or fray top face, good quality on bottom face when through-cutting	Best edge quality on top face, may chip or fray bottom face when through-cutting	Clean edge on both top and bottom face
Moderate chip clearing abilities	Excels at clearing chips and dissipating heat, es-pecially with “o-flute” bits	May compact chips in a groove	Designed to cut veneered or laminated materials at full depth in one pass
	Upwards force may cause part lifting	Downwards force may help with cutting thin sheets	
Ideal for: general-purpose cutting	Ideal for: plastics, alu-minum, or any material where heat buildup is a concern	Ideal for: plywood and laminates (pocketing)	Ideal for: plywood and laminates (profile cutting)

Number of flutes: The number of flutes on a bit is essential to calculating proper feed and speed rates. For most applications a bit with 1, 2, or 3 flutes can be used, but the feed rate and RPM must be adjusted accordingly to maintain proper chip load.

End shape: Straight and up-spiral bits come in a variety of end shapes. Square ends are most common, and are a good choice for creating pockets and grooves, profile cutting, simple lettering, and drilling operations. Ball (or rounded) ends are best for 3D carving. V-carve bits are often used to create complex letters for sign making. They can also be used to chamfer edges and create countersinks for screw holes.

What's the difference between a square-end bit and an end mill?



An **end mill** has cutting flutes that extend across the bottom (end) of the bit. It is designed for plunge- cutting as well as lateral cutting.

"Square-end" is simply a description of end shape. Square-end bits are not always end mills, and end mills do not always have square ends.

The first image at right shows a true end mill, and the second shows a straight-fluted, square-end bit that is not an end mill. A ramp-in must be applied to a toolpath when using this type of bit.

Calculating feeds/speed with Chip Load:

Chip load refers to the actual thickness of the chip cut by each revolution of the cutter. It is the measurement that all feed/speed calculations are based on.

A spinning bit generates friction and heat as it moves through the material, and part of this heat is pulled away by the flying chips. A larger chip load pulls away

more heat, but also puts more stress on the cutter. Each material has its own ideal chip load range that balances heat dissipation with cutter stress.

A basic chart for common materials is available in the SB3 software. Click on Tools > Chip Load Calculator, then click on Chip Load Help. You can use this chart along with the Chip Load Calculator to determine a good starting speed for each toolpath. More detailed chip load charts are available online at Onsrud's website (www.onsrud.com).

When calculating feeds/speeds for a toolpath, do not rely on the defaults in your tool database. Those values are only placeholders and are not intended for any particular material.

Feeds and Speeds

A challenge of getting a good CNC cut is in selecting the best cutting speed (feed rate) and router/spindle RPM (speed of rotation). Feeds and speeds are a critical part of machining and should be fully understood before deviating from recommended settings. A primary concern of machining is chip load, which is a representation of the size of the chips produced during cutting. The goal is to get the maximum chip load possible to increase productivity, reduce heat, and prevent premature dulling. When chip load is too small, bits will get too hot and dull quicker. When chip load is too high, the tool will deflect creating a bad surface finish and, in extreme cases, chip or break the bit.

Chip load is a function of three different parameters: feed rate, RPM, and number of flutes on the tool.

Chip load is the thickness of the chunk of material taken by a tooth of the cutter. This is determined by how fast the cutter is moving forward into the material and how fast it is turning (Chip load = Feed

Rate / [RPM x number of flutes]). This formula provides a starting point for determining the most suitable parameters for any cutting situation.

A ShopBot tool has either a router or a spindle. The advantage of a spindle is that speed can be very precisely controlled. This allows for optimization of cuts. A router does not have a precise speed controller, so the actual speed of the router is directly correlated to the depth of cut, material, and feed rate. For routers, these numbers will be a starting point, but will have to be adjusted for real life conditions. This is necessary because routers tend to run lower than the setting on the router indicates; settings should be recorded as they will work for similar cuts in similar material. It is recommended to start with low numbers to ensure it is within an acceptable range for the tool.

There is considerable overlap in the range of parameters; it will be necessary to test in this range for the best speed selection for a particular cutting or machining operation. Below is the strategy that bit manufacturer Onsrud suggests. This works particularly well for routers.

Optimizing feed rates and speeds:

1. Start off using an RPM derived for the chip load for the material being cut (see charts).
2. Increase the cutting speed (feed rate) until the quality of the part's finish starts to decrease or the part is starting to move from hold downs. Then decrease speed by 10%.
3. Decrease RPM until finish deteriorates, then bring RPM back up until finish is acceptable.
4. This optimizes RPM and speed to remove the largest possible chips.

To reduce the amount of work needed, a range of feeds and speeds is provided to start at (step 1 above). To increase performance, optimize feeds and speeds as detailed above. If there are any surface finish problems, adjust until performance is acceptable. If there are a lot of work holding issues, it may be necessary to revisit the hold down method to ensure it is adequate.

Manufacturer settings for bits

Most manufacturers provide optimal speeds and feeds for their bits. There are numerous calculators available online that can provide rough numbers. These calculators should work fine, but may not be optimized for each particular bit. The manufacturer will still be the best resource, as most reputable companies have information and live support available to help users of their tools utilize them properly. ShopBot recommends Onsrud tools.

If doing production work in a certain type of material, it would be best to purchase a bit made for that type of cutting. If performing a number of different materials, then a more general bit could be used.

http://www.onsrud.com/xdoc/FeedSpeeds

Chip Load = per cutting edge.

IPS = Inches Per Second.

IPM = Inches Per Minute.

RPM = Revolutions Per Minute.

# of cutting edges = # of flutes.*

*typically true unless stated otherwise by manufacturer.

Chip Load = Feed Rate (IPM) / (RPM x # of cutting edges)

Feed Rate (IPM) = RPM x # of cutting edges x Chip Load

Speed (RPM) = Feed Rate (IPM) / (# of cutting edges x Chip Load)

IPM = IPS x 60

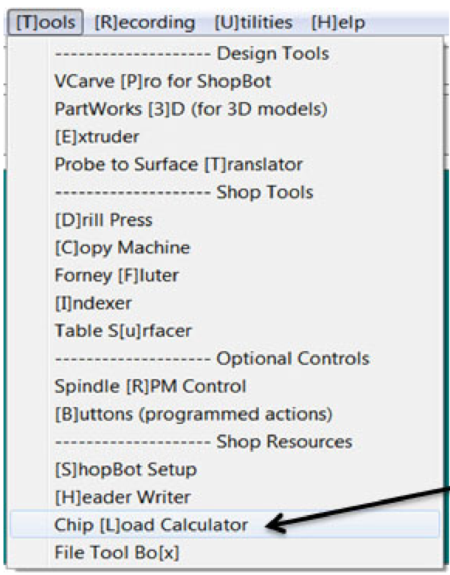
Depth of cut: A function of cutting edge diameter set by manufacturer.

Onsrud bits are typically allowed a cut depth per pass equal to the cutting edge diameter unless otherwise specified. For a deeper cut, it will be necessary to reduce the chip load. For twice the depth of cut, reduce the chip load per tooth by 25% and for triple the depth of cut, reduce the chip load by 50%. If doing this, it will be necessary to calculate the feed rate and speed instead of using the chart.

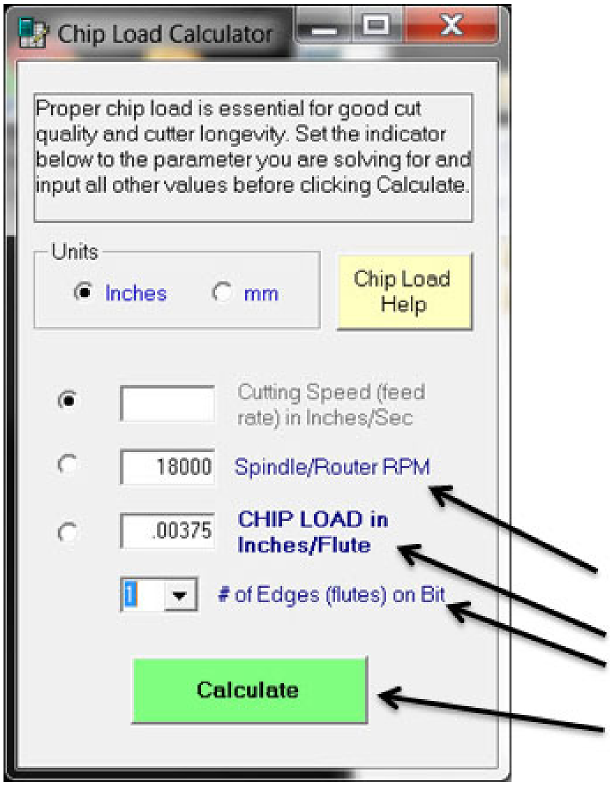
Example using a 1/4" or 0.125" bit – Straight V Carbide Tipped Endmill SB# 13642:

This bit is used for soft wood, at a depth of 1/2" (two times the diameter). The chart cannot be used for this calculation. Start with the middle of the range of recommended chip load provided on the chart ((.006+.004)/2=.005). Reduce that by 25% to allow for the deeper cutting depth (0.75x0.005=0.00375). Assuming that cutting will be performed at an RPM of 18,000 and that this tool only has one flute. Let's use the ShopBot 3 "Chip Load Calculator" to decide what value to start with.

Chip load calculator



The "Chip Load Calculator" comes included in the ShopBot 3 software. It can be accessed by going to Tools > Chip Load Calculator (TL).

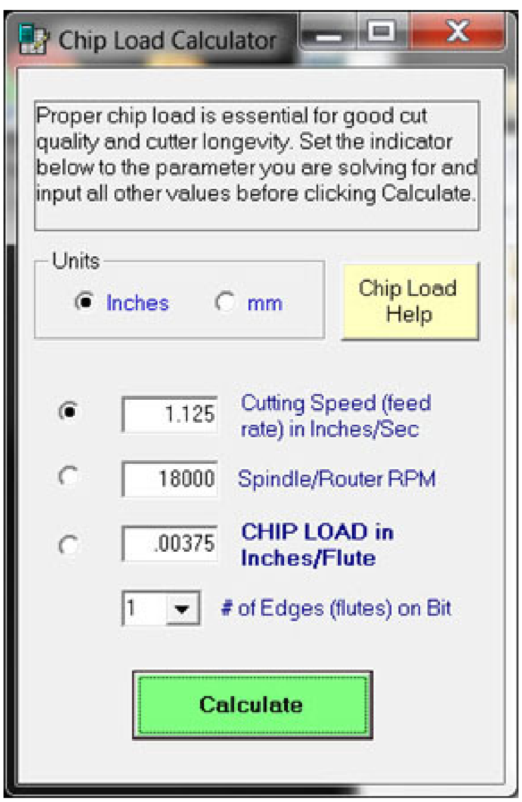


In the previous example values were provided to calculate the feed rate.

RPM: 18,000

Chip load: 0.00375, Depth of cut: 2 x Diameter = 1/2”
Flutes: 1

Enter these into the calculator and click “Calculate.”



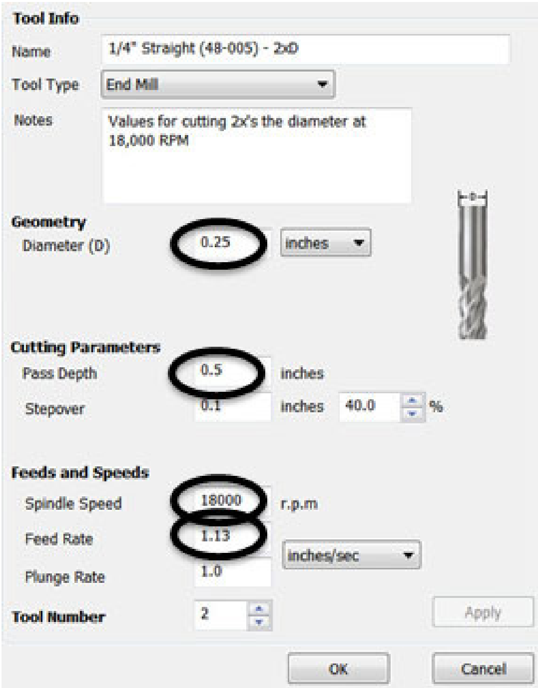
This now provides the values needed to create an entry into the tool database in VCarve Pro.

Diameter: 0.25”

Pass depth: 0.5”

Spindle speed: 18000 RPM

Feed rate: 1.125 inches/sec



The data is now available to input into the CAM software. The tool info sheet for VCarve Pro is shown.

Feeds and speeds charts

These charts have been taken from Onsrud's recommendations and calculated in order to save time for the bits provided in the ShopBot bit kit. For further information, Onsrud series numbers are provided. ShopBot numbers are also provided for an easier, streamlined ordering experience. Onsrud provided numbers are cut depths, chip loads, and flutes. The exception to this rule is the "Carbide Tipped Surfacing Cutter" which has all values provided by Onsrud. Any tools with no values provided are not listed by Onsrud, but if they are contacted about a specific use, they may provide some base numbers from which to calculate.

The "Chip Load Calculator" in ShopBot 3 was utilized with a starting RPM of 18,000 on all tools to find the calculated feed rate. Any deviations from provided numbers should be verified and adjusted through recalculation in ShopBot 3 "Chip Load Calculator."

Soft wood

Name	SB#	Onsrud Series	Cut	Chip Load per leading edge	Flutes	Feed Rate (ips)	Feed rate (ipm)	RPM	Max Cut
1" 60 degree Carbide V cutter	13648	37-82	1 x D	.004-.006	2	2.4-3.6		18,000	
1/4" Straight V Carbide Tipped End Mill	13642	48-005	n/a	n/a	1	n/a	n/a	n/a	
1/2" Straight V Carbide Tipped End Mill	13564	48-072	n/a	n/a	2	n/a	n/a	n/a	
1/4" Upcut Carbide End Mill	13528	52-910	1 x D	.007-.009	2	4.2-5.4		18,000	
1/4" Downcut Carbide End Mill	13507	57-910	1 x D	.007-.009	2	4.2-5.4		18,000	
1/4" Upcut Carbide End Mill	1108	65-025	1 x D	.004-.006	1	1.2-1.8		18,000	
1/8" Tapered Carbide Upcut Ball End Mill	13636	77-102	1 x D	.003-.005	2	1.8-3.0		18,000	
1-1/4" Carbide Tipped Surfacing Cutter	13555	91-000	1/2-3/4 x D		2		200-600	12,000-16,000	1/8"

Hard wood

Name	SB#	Onsrud Series	Cut	Chip Load per leading edge	Flutes	Feed Rate (ips)	Feed rate (ipm)	RPM	Max Cut
1” 60 degree Carbide V cutter	13648	37-82	1 x D	.004-.006	2	2.4-3.6		18,000	
1/4” Straight V Carbide Tipped End Mill	13642	48-005	1 x D	.005-.007	1	1.5-2.1	90-126	18,000	
1/2” Straight V Carbide Tipped End Mill	13564	48-072	1 x D	.006-.008	2	3.6-4.8		18,000	
1/4” Upcut Carbide End Mill	13528	52-910	1 x D	.006-.008	2	3.6-4.8		18,000	
1/4” Downcut Carbide End Mill	13507	57-910	1 x D	.005-.007	2	3.0-4.2		18,000	
1/4” Upcut Carbide End Mill	1108	65-025	1 x D	.004-.006	1	1.2-1.8		18,000	
1/8” Tapered Carbide Upcut Ball End Mill	13636	77-102	1 x D	.003-.005	2	1.8-3.0		18,000	
1-1/4” Carbide Tipped Surfacing Cutter	13555	91-000	1/2-3/4 x D		2		200-600	12,000-16,000	1/8"

Medium Density Fiberboard (MDF)

Name	SB#	Onsrud Series	Cut	Chip Load per leading edge	Flutes	Feed Rate (ips)	Feed rate (ipm)	RPM	Max Cut
1” 60 degree Carbide V cutter	13648	37-82	1 x D	.004-.006	2	2.4-3.6		18,000	
1/4” Straight V Carbide Tipped End Mill	13642	48-005	1 x D	.005-.007	1	1.5-2.1	90-126	18,000	
1/2” Straight V Carbide Tipped End Mill	13564	48-072	1 x D	.005-.007	2	3.0-4.2	**180-252**	18,000	
1/4” Upcut Carbide End Mill	13528	52-910	1 x D	.006-.008	2	3.6-4.8		18,000	
1/4” Downcut Carbide End Mill	13507	57-910	1 x D	.006-.008	2	3.6-4.8		18,000	
1/4” Upcut Carbide End Mill	1108	65-025	1 x D	.004-.006	1	1.2-1.8		18,000	
1/8” Tapered Carbide Upcut Ball End Mill	13636	77-102	1 x D	.003-.005	2	1.8-3.0		18,000	
1-1/4” Carbide Tipped Surfacing Cutter	13555	91-000	1/2-3/4 x D		2		200-600	12,000-16,000	1/8"

Soft Plywood

Name	SB#	Onsrud Series	Cut	Chip Load per leading edge	Flutes	Feed Rate (ips)	Feed rate (ipm)	RPM	Max Cut
1” 60 degree Carbide V cutter	13648	37-82	1 x D	.004-.006	2	2.4-3.6		18,000	
1/4” Straight V Carbide Tipped End Mill	13642	48-005	1 x D	.005-.007	1	1.5-2.1	90-126	18,000	
1/2” Straight V Carbide Tipped End Mill	13564	48-072	1 x D	.007-.009	2	4.2-5.4		18,000	
1/4” Upcut Carbide End Mill	13528	52-910	n/a	n/a	2	n/a	n/a	n/a	n/a
1/4” Downcut Carbide End Mill	13507	57-910	n/a	n/a	2	n/a	n/a	n/a	n/a
1/4” Upcut Carbide End Mill	1108	65-025	1 x D	.004-.006	1	1.2-1.8		18,000	
1/8” Tapered Carbide Upcut Ball End Mill	13636	77-102	n/a	n/a	2	n/a	n/a	n/a	n/a
1-1/4” Carbide Tipped Surfacing Cutter	13555	91-000	1/2-3/4 x D		2		200-600	12,000-16,000	1/8"

Laminated Chipboard

Name	SB#	Onsrud Series	Cut	Chip Load per leading edge	Flutes	Feed Rate (ips)	Feed rate (ipm)	RPM	Max Cut
1” 60 degree Carbide V cutter	13648	37-82	1 x D	.004-.006	2	2.4-3.6		18,000	
1/4” Straight V Carbide Tipped End Mill	13642	48-005	1 x D	.006-.008	1	1.8-2.4		18,000	
1/2” Straight V Carbide Tipped End Mill	13564	48-072	1 x D	.008-.010	2	4.8-6.0		18,000	
1/4” Upcut Carbide End Mill	13528	52-910	n/a	n/a	2	n/a	n/a	n/a	n/a
1/4” Downcut Carbide End Mill	13507	57-910	n/a	n/a	2	n/a	n/a	n/a	n/a
1/4” Upcut Carbide End Mill	1108	65-025	n/a	n/a	1	n/a	n/a	n/a	n/a
1/8” Tapered Carbide Upcut Ball End Mill	13636	77-102	n/a	n/a	2	n/a	n/a	n/a	n/a
1-1/4” Carbide Tipped Surfacing Cutter	13555	91-000	1/2-3/4 x D		2		200-600	12,000-16,000	1/8"

Laminated Plywood

Name	SB#	Onsrud Series	Cut	Chip Load per leading edge	Flutes	Feed Rate (ips)	Feed rate (ipm)	RPM	Max Cut
1” 60 degree Carbide V cutter	13648	37-82	1 x D	.004-.006	2	2.4-3.6		18,000	
1/4” Straight V Carbide Tipped End Mill	13642	48-005	1 x D	.005-.007	1	1.5-2.1		18,000	
1/2” Straight V Carbide Tipped End Mill	13564	48-072	1 x D	.007-.009	2			18,000	
1/4” Upcut Carbide End Mill	13528	52-910	n/a	n/a	2	n/a	n/a	n/a	n/a
1/4” Downcut Carbide End Mill	13507	57-910	n/a	n/a	2	n/a	n/a	n/a	n/a
1/4” Upcut Carbide End Mill	1108	65-025	n/a	n/a	1	n/a	n/a	n/a	n/a
1/8” Tapered Carbide Upcut Ball End Mill	13636	77-102	1 x D	.003-.005	2	1.8-3.0		18,000	
1-1/4” Carbide Tipped Surfacing Cutter	13555	91-000	1/2-3/4 x D		2		200-600	12,000-16,000	1/8"