Feeds and Speeds Charts

Table of Contents

ShopBot Contact Info
ShopBot Tools, Inc.
3333B Industrial Dr.
Durham, NC 27704
919-680-4800 or 888-680-4466
www.shopbottools.com

Introduction

One of the challenges in getting good CNC cuts 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. One of the primary concerns 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, will chip or break your bit. Chip load is a function of three different parameters: feed rate, RPM, and number of flutes on the tool.

Selection can be facilitated using the guidance provided by manufacturers regarding the best chip load for a particular cutter. Chip load is the thickness of the chunk of material taken by a tooth of the cutter. This parameter has a relationship between the combination of 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 relationship shown in the formula gives you starting point speed values for testing to determine the most suitable parameters for any cutting situation. The ShopBot “Chip Load Calculator” (pg. 5) simply provides a quick way to explore these values, and is most useful if you know the approximate desired chip load for a particular cutter and material.

Your machine will have either a router or a spindle. The advantage of a spindle is that speed can be very precisely controlled. This allows for optimization of your 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 it is cutting at. For routers these numbers will be a starting point, but will have to be adjusted for real life conditions. This is a necessity because routers tend to run a speed lower than the setting on the router indicates; settings should be recorded as they will work for similar cuts in similar material. We recommend that you start with our low numbers to ensure it is within an acceptable range for your tool.

Note that there is considerable overlap in the range of parameters; you will need to test in this range for the best speed selection for a particular cutting or machining operation. Below is the strategy that 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 you are cutting (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 so that you are taking the largest possible chips.

This will be the best feed and speed for that particular cut. In order to reduce the amount of work needed we supply a range of feeds and speeds to start at (step 1 above). To increase performance you will want to optimize feeds and speeds as detailed above. If you run into any surface finish problems you can adjust them until performance is acceptable. If you run into a lot of work holding issues you may need to revisit your work holding method to ensure it is adequate.

Manufacturer settings for bits

Luckily, most manufacturers have done the work of finding the optimal speeds and feeds for you. Your first resource for proper feeds and speeds should be the manufacturer of your bit. There are numerous calculators available online that will give you rough numbers. These should work fine, but may not be optimized for your particular bit. The manufacturer of the bit will still be the best resource; most reputable companies have information and live support available to help users of their tools utilize them properly. We recommend Onsrud tools and will be passing on their recommendations for our basic bit kit.

If you are going to do production work in a certain type of material, you would be best served by purchasing a bit made for that type of cutting. If you are going to do a number of different materials, then a more general bit could be used. We will try to give you a general overview, but bit manufacturers will be happy to work with you to help you determine the bit you will need for a particular use.

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

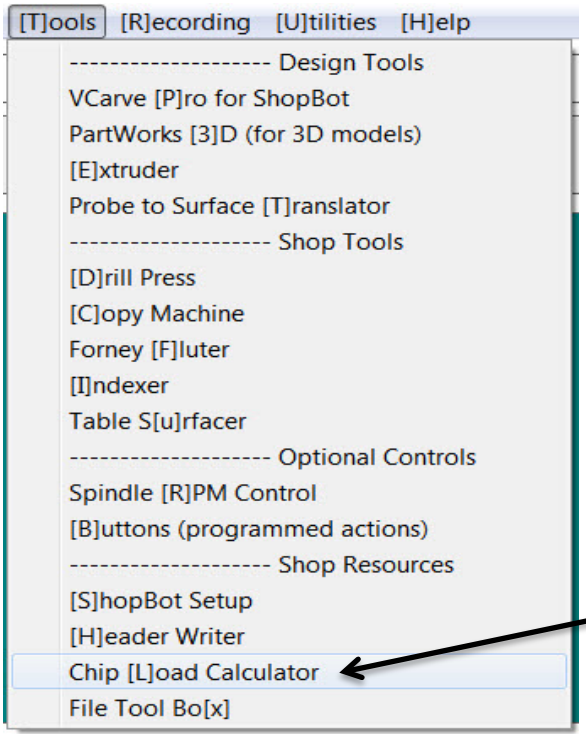
Depth of cut: A function of cutting edge diameter set by manufacturer.

Onsrud bits are typically allowed a cut depth per pass equal to the cutting edge diameter unless otherwise specified for a particular material. If you wish to cut deeper you will need to reduce the chip load. For twice the depth of cut you will have to reduce the chip load per tooth by 25% and for triple the depth of cut you will have to reduce the chip load by 50%. Please note that if you do this you will have to calculate the feed rate and speed yourself instead of using our chart.

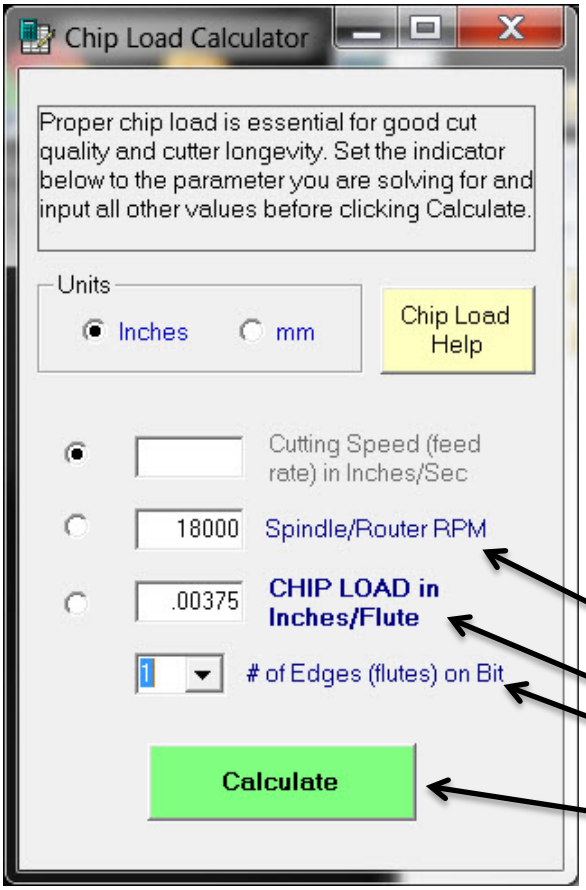
Example using a 1/4” or 0.125” bit – Straight V Carbide Tipped Endmill SB# 13642:

You decide to use this bit for soft wood, and decide to use this to a depth of cut of 1/2” (2 times the diameter) which means you can’t use our chart. You decide to start with the middle of the range of recommended chip load provided on the chart ((.006+.004)/2=.005). You then reduce that by 25% to allow for the deeper cutting depth (0.75x0.005=0.00375). You decide you are going to cut at a RPM of 18,000 and this tool only has one flute as shown in the chart provided below. Now that you have the information needed to find the feed speed, we will use the ShopBot 3 “Chip Load Calculator” to decide what value to start with.

Chip load calculator



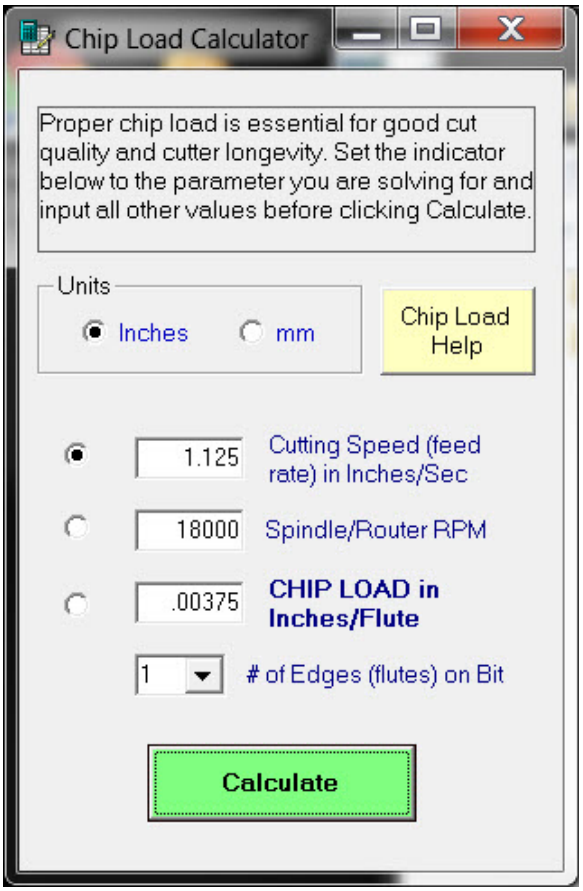
The "Chip Load Calculator" comes included in the ShopBot 3 software. It can be accessed by going to Tools > Chip Load Calculator (TL).



In our previous example we found some values for a tool in order to calculate the feed rate we will use.

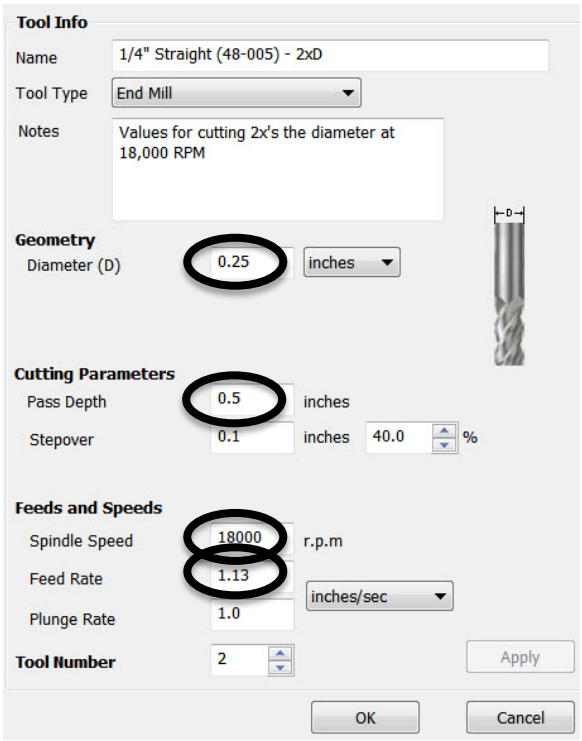
RPM: 18,000
Depth of cut: 2 x Diameter = 1/2"
Chip load: 0.00375
Flutes: 1

These will be input into the calculator and then we will click "Calculate."



We now have all of the values needed to create an entry into our tool database in VCarve Pro.

Diameter: 0.25”
Pass depth: 0.5”
Spindle speed: 18000 RPM
Feed rate: 1.125 inches/sec



Now you can enter the data into your CAM software. The tool info sheet for VCarve Pro is shown.

Feeds and speeds charts

The values in these charts have been taken from Onsrud's recommendations and calculated for you in order to save time for the bits provided in our bit kit. For further information, Onsrud series numbers are provided. We also provide the ShopBot numbers for an easier, streamlined ordering experience for you. 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 you contact them about your specific use they may provide you with 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"

MDF (Medium Density Fiberboard)

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"