CNC ROUTER BIT FEEDS & SPEEDS

IMPERIAL (INCH) & METRIC

Provided by **IDC WOODCRAFT**

www.idcwoodcraft.com

SUBSCRIBE TO IDC WOODCRAFT ON YOUTUBE

CNC ROUTER BIT FEEDS & SPEEDS

PLEASE READ

Hello CNC’er!

The following table will guide you through feeds & speeds for most CNC router bits you will use, as well as less common ones.

The feeds & speeds provided are average accepted values for benchtop CNC routers. These will work with soft, medium and moderately hard wood.

For extremely hard wood (Ex. Brazilian Ebony, Snakewood), please do some research before you run projects.

You may see values on this table that are not in listed in your software. Some programs require more information than usual. Don’t worry, if you don’t see it in yours. It won’t be needed.

CNC ROUTER BIT FEEDS & SPEEDS

This table has been built to make ordering bits easy for you. All bits listed have a ***BUY NOW*** link to save you from searching when you need to get bits.

If the IDC Woodcraft store does not carry a bit listed in this table, you will be directed to the best source to get it.

Make sure to put this document on your desktop so you always have immediate access to it.

Some router bits will be highlighted like this example → 1/4” Down Cut

When you see this, it means the bit is used a lot.

If you do not have backups for those bits, you’ll want to consider getting them, because no one wants to wait on a project because of a broken or worn bit.

***NOTE**: Only high-quality bits are listed. For lower-quality, or practice bits, please search Amazon.*

CNC ROUTER BIT FEEDS & SPEEDS

SPECIAL NOTE

Feeds & Speeds tables always refer to speed in *rpm* (revolutions per minute). This is fine when the router uses a spindle controlled by the CNC.

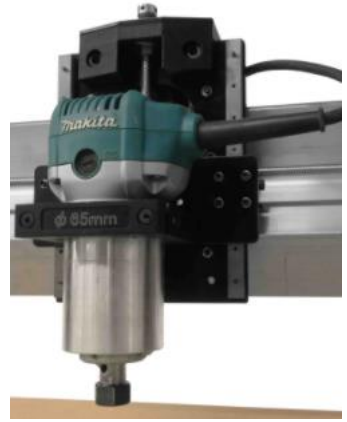
However, for those that have manual routers, the rpm number is not very helpful.

To help with this, you will see an unusual field titled “Router Dial”. This is for users with manual routers.

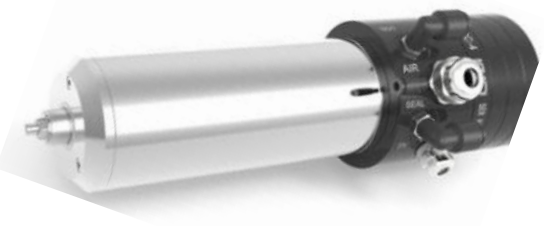
Note: The “Router Dial” number refers to Makita model RT0701C that commonly comes with benchtop CNC routers (shown at right top).

Makita RT0701C Dial Settings	
1	10,000
2	12,000
3	17,000
4	22,000
5	27,000
6	30,000

Manual CNC Router



Spindle



CNC ROUTER BIT FEEDS & SPEEDS

CAUTION

Different CNC routers have different levels of rigidity. The rigidity of a machine will determine the feeds & speeds limits a machine can handle. A rigid machine will work well with these settings. One with flex in the gantry will not.

If your machine has flex, you will want to back the feeds off by about 20%.

How to tell if you need to tone down the feeds.

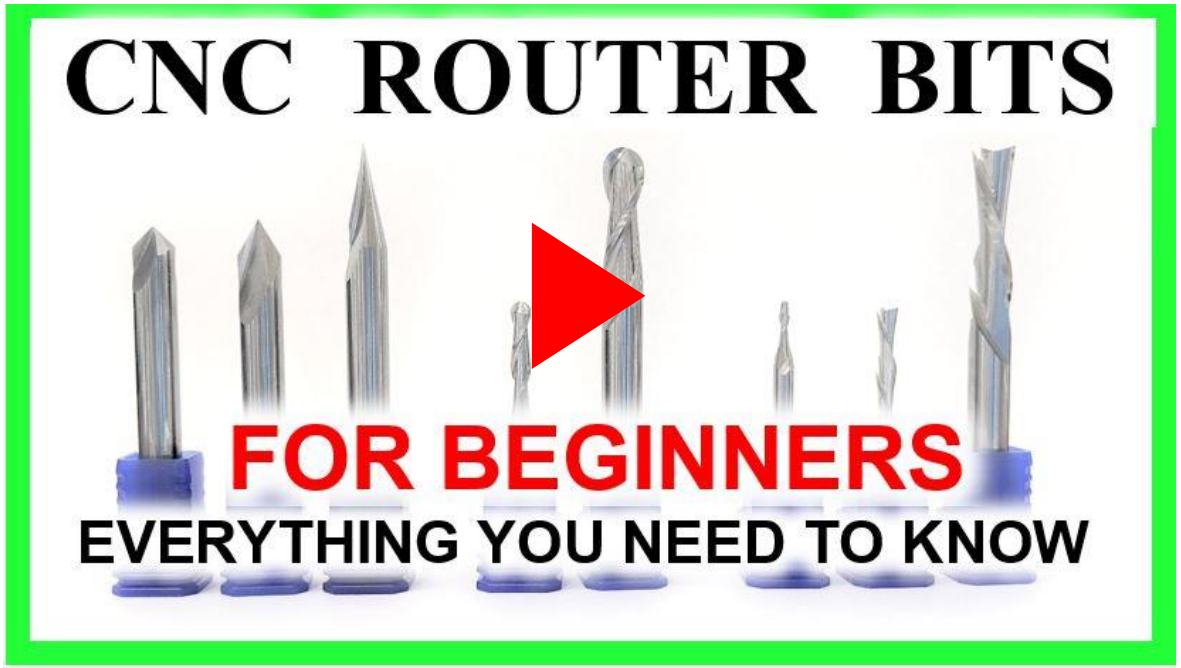
Grab your router and push back and forth. If it moves easily, reduce the feed rates in this table.

CNC ROUTER BIT FEEDS & SPEEDS

Learning About Router Bits

If you are **brand new** to CNC routers and router bits, this tutorial video will teach you everything you need to know about router bits.

Click the image to watch →



BE SURE TO SUBSCRIBE

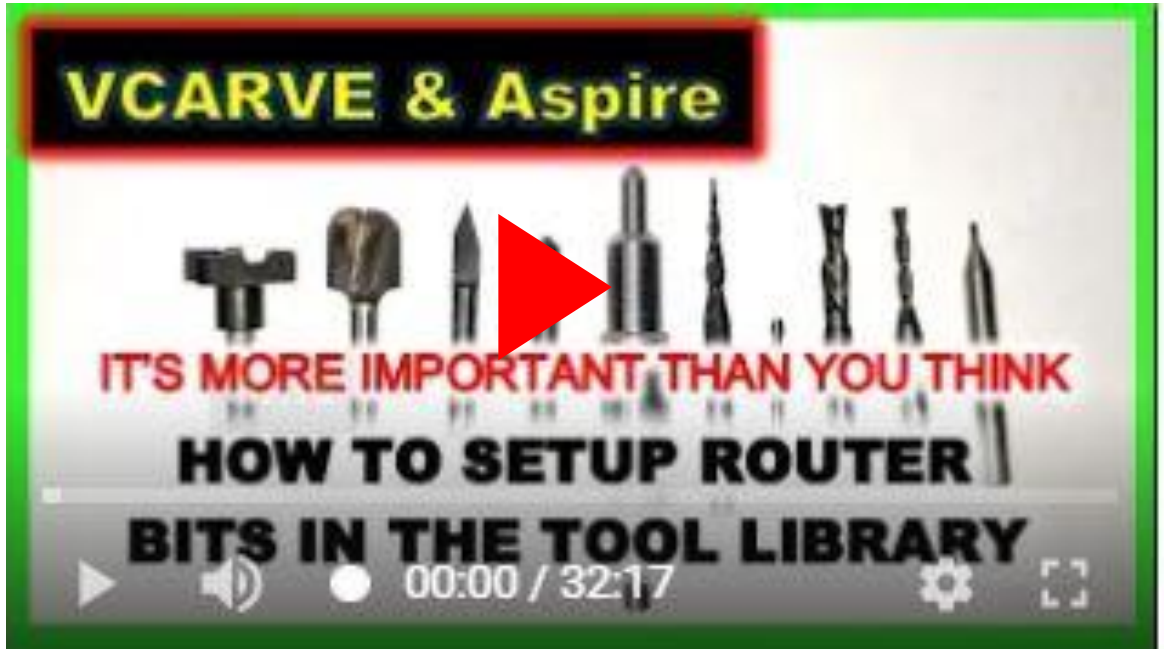
CNC ROUTER BIT FEEDS & SPEEDS

Learn How To Set Up Your Router Bits In Vectric

Setting up your bits properly in the Vectric software can feel a little intimidating at first.

This video will walk you through the process, so you get it right the first time.

Click the image to watch →



BE SURE TO SUBSCRIBE

CNC ROUTER BIT FEEDS & SPEEDS

FOR METRIC USERS

The first section of this *Feeds & Speeds* table is for imperial (inch) units.

Section 2 is identical, except all units are metric.

Look for the page titled “METRIC FEEDS AND SPEEDS”. Your information is after that.

Please be sure to read the header information in that section.

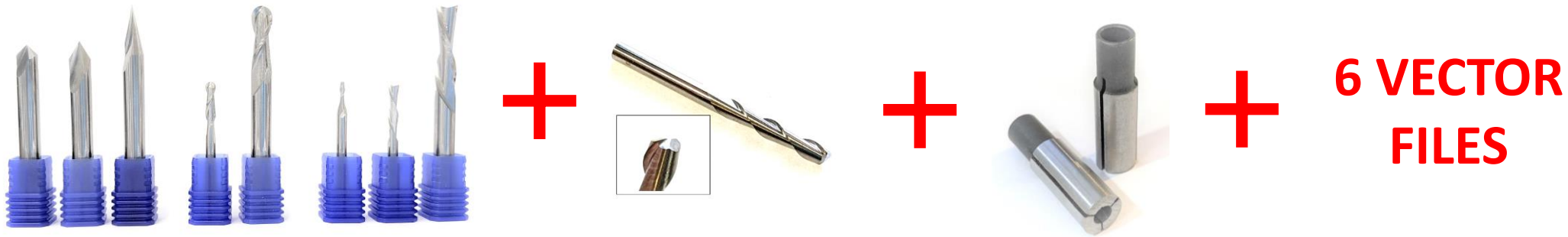
CNC ROUTER BIT FEEDS & SPEEDS

This table is set up in 2 sections…

The next page is for the 8-piece starter pack provided by IDC Woodcraft (see image below).

The rest are broken down by bit type, with feeds & speeds for each size of that type.

ROUTER BIT STARTER SET

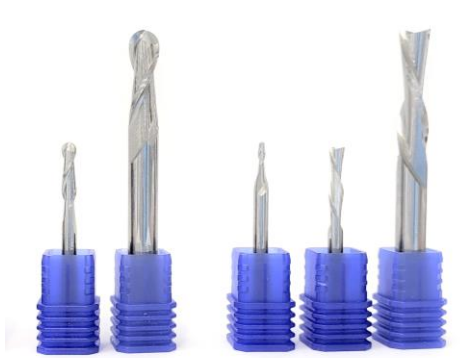


6 VECTOR FILES

LEARN MORE ABOUT THIS SET NOW

FEEDS & SPEEDS for the Complete CNC Router Bit Starter Set

ENDMILLS												
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Feed (in/min)	Plunge (in/min)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
1/16" Up Cut	0.0625	2	0.188	1.50	0.125	20	10	0.032	40%	27,000	5	BUY NOW
1/8" Down Cut	0.125	2	0.750	1.75	0.125	35	15	0.125	40%	22,000	4	BUY NOW
1/4" Down Cut	0.25	2	1.000	2.50	0.25	80	30	0.25	40%	19,000	3.5	BUY NOW
1/8" Ballnose	0.125	2	0.500	1.50	0.125	35	15	0.05	40%*	22,000	4	BUY NOW
1/4" Ballnose	0.25	2	0.750	2.50	0.25	70	30	0.12	40%	19,000	3.5	BUY NOW



ENDMILLS

V-BIT														
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Side Angle	Feed (in/min)	Plunge (in/min)	Depth Per Pass	Clear Pass Stepover	Final Pass Stepover	Spindle (rpm)	Router Dial		
30° V-bit	0.25	1	0.466	2.50	0.25	15	35	20	0.025	20%	0.005	27,000	5	BUY NOW	BUY THE SET (save $$)
60° V-bit	0.25	2	0.216	2.00	0.25	30	40	20	0.05	20%	0.005	22,000	4	BUY NOW	
90° V-bit	0.25	2	0.125	2.00	0.25	45	50	25	0.1	20%	0.005	17,000	3	BUY NOW	

V-BITS

1/8" DRILLING ENDMILL *See notes below												
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Feed (in/min)	Plunge (in/min)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
Drilling	0.125	2	0.750	1.50	0.125	60	60*	0.2	40%	22,000	4	BUY NOW
Conventional	0.125	2	0.750	1.50	0.125	60	20	0.2	40%	22,000	4	
* The plunge value is for using the spiral drilling technique. Watch this video to learn how									Watch Video			

SPIRAL DRILLING UP-CUT ENDMILL

CNC ROUTER BIT FEEDS & SPEEDS

SECTION 2

ALL COMMON CNC ROUTER BITS FEEDS & SPEEDS

IMPERIAL (INCH)

FEEDS & SPEEDS
FOR CNC ROUTER BITS

PLEASE READ

All *Feeds & Speeds* information in this section are in inches (in).

NOTE: All units use the CNC industry standard ***inches per minute (ipm)***

FEEDS & SPEEDS – DOWN CUTTING ENDMILLS

Down cutting endmills, also known as ‘down bits’, are the most common bit you will use for material removal and getting squared edges on your projects.

Despite the benefits of creating clean sharp corners and edge, they do have a drawback. This *MIST WATCH* video will explain what it is and how to overcome it.

Items with GREEN ‘→’ are bits you want to have backups of. You will use these most often.

DOWN CUT ENDMILL												
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Feed (in/min)	Plunge (in/min)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
1/32"	0.031	2	0.032	1.50	0.125	10	7	0.015	40%	27,000	5	BUY NOW
1/16"	0.0625	2	0.250	1.50	0.125	20	10	0.032	40%	27,000	5	BUY NOW
→ 1/8"	0.125	2	0.750	1.75	0.125	35	15	0.125	40%	22,000	4	BUY NOW
3/16"	0.1875	2	1.000	2.38	0.25	70	30	0.187	40%	19,000	3.5	BUY NOW
→ 1/4"	0.25	2	1.000	2.50	0.25	80	30	0.25	40%	19,000	3.5	BUY NOW

FEEDS & SPEEDS – UP CUTTING ENDMILLS

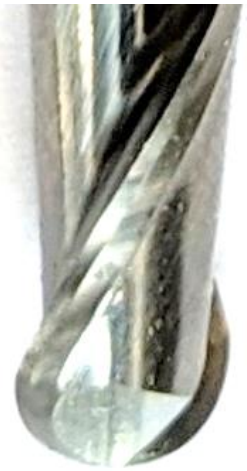
Up cutting endmills, also known as ‘up bits’, are best used to remove lots of material quickly while leaving a very good finish along the bottom surface.

They can be used in place of down cutters with proper feeds and speeds. A ggod time to use this bit is when a V-bit is used to accent an edge of a large area that is recessed.

The bit with the GREEN ‘→’ is the one you want to have. If you plan on using this often, get a backup bit.

UP CUT ENDMILL												
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Feed (in/min)	Plunge (in/min)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
1/32"	0.031	2	0.125	2.00	0.125	10	7	0.015	40%	27,000	5	BUY NOW
1/16"	0.0625	2	0.813	2.50	0.125	20	10	0.032	40%	27,000	5	BUY NOW
→ 1/8"	0.125	2	0.750	1.50	0.125	50	20	0.2	40%	22,000	4	BUY NOW
3/16"	0.1875	2	0.750	2.00	0.25	80	40	0.28	40%	19,000	3.5	BUY NOW
→ 1/4"	0.25	2	0.750	2.50	0.25	80	40	0.375	40%	19.000	3.5	BUY NOW

FEEDS & SPEEDS – BALL NOSE ENDMILL

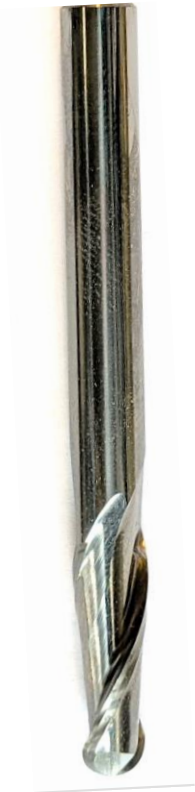


The ball nose endmill is categorized as an endmill. The difference is the rounded end.

It is used to accent your projects, such as putting a rounded edge on a sign. You can also use them to do 2.5D or 3D relief carves when extremely high detail is not a concern, but you still want a good-looking project.

To learn about this application, watch this video.

It is standard to have both in your set of bits. You do not need backups of these unless you find you use them a lot.



BALLNOSE												
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Feed (in/min)	Plunge (in/min)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
1/8" Ballnose	0.125	2	0.500	1.50	0.125	35	15	0.05	20%*	22,000	4	BUY NOW
1/4" Ballnose	0.25	2	0.750	2.50	0.25	70	30	0.12	20%	19,000	3.5	BUY NOW

* When using 1/8“ ballnose for 2.5D & 3D relief carves, set stepover to 5-8%

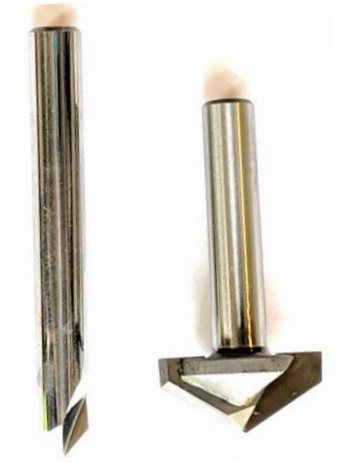
FEEDS & SPEEDS – V-BITS

You will notice more fields in this section for V-bits than for the previous bits. That's because more information is required for your software to create toolpaths.

You might find some fields aren't even in your software. Don't worry., you won't need them. They are for more advanced engineering type software programs.

The bit with the **GREEN '→'** are typical backups. However, it is suggested to get the 120 V-bit since it easily does the job of the 90-degree bit. And it can do much larger projects efficiently.

V-BIT														
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Side Angle	Feed (in/min)	Plunge (in/min)	Depth Per Pass	Step Over	Final Pass Stepover	Spindle (rpm)	Router Dial	
30° V-bit	0.25	1	0.466	2.50	0.25	15	35	20	0.025	20%	0.005	27,000	5	BUY NOW
→ 60° V-bit	0.25	2	0.216	2.00	0.25	30	40	20	0.05	20%	0.005	22,000	4	BUY NOW
→ 90° V-bit	0.25	2	0.125	2.00	0.25	45	50	25	0.1	20%	0.005	17,000	3	BUY NOW
120° V-bit	0.75	2	0.180	1.50	0.25	60	80	60	0.19	30%	0.01	17,000	3	BUY NOW



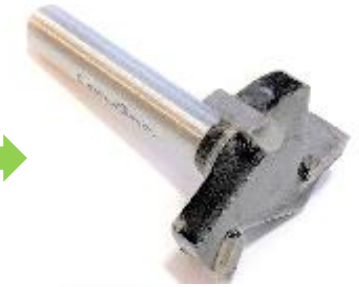
FEEDS & SPEEDS – SURFACING & BOWL BITS

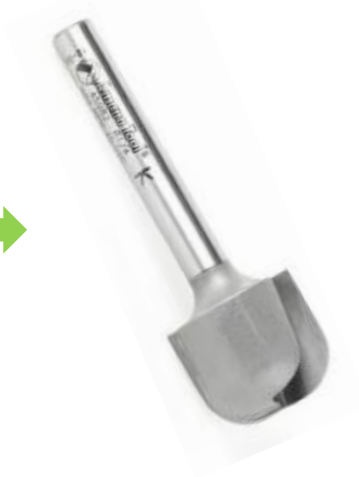
Here, you will find 2 types of bits. The surfacing bit and bowl bit. Each serve a different purpose.

The surfacing bit is an **ABSOLUTE MUST HAVE**! It is used to surface your spoilboard and smooth warped material or botched projects. Watch this video to learn more about surfacing a spoilboard.

The bowl bit is used to remove large amounts of material without a lot of force applied to your machine. This is commonly used to remove material from deep pockets. It is an extremely handy bit to have.

SURFACING BIT - FOR SURFACING SPOILBOARD (Category: Endmill)												
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Feed (in/min)	Plunge (in/min)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
1"	1.0	3	0.250	1.50	0.25	80-100	7	0.125	70%	17,000	3	BUY NOW
BOWL BIT - HOGGING (Category: Endmill)												
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Feed (in/min)	Plunge (in/min)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
Bowl	0.75	2	.75	2.5	0.25	80-100	15	0.375	40%	17,000	3	BUY NOW





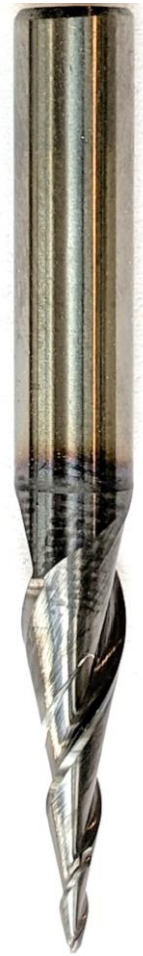
FEEDS & SPEEDS – TAPER BALL NOSE CARVING BIT

The taper ball nose bit is use for fine detail carving. This is the bit to use for exquisite projects.

You may notice the image is a bit ‘stumpy’ for this type of bit. Most carving bits tend to be longer. The major drawback to long, narrow bits in this family is ‘tool deflection’.

A long narrow bit will have side bounce which will show up as lines in your project.

To learn more about taper ball nose bits and how they are used, watch this video.



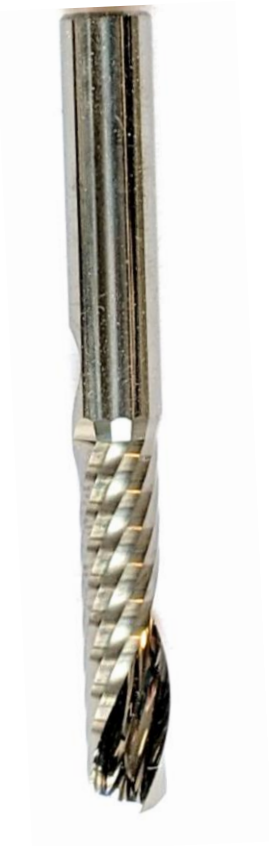
TAPER BALLNOSE CARVING BIT													
Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Tip Radius	Angle	Feed (in/min)	Plunge (in/min)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
0.3 dia	2	0.750	2.00	0.25	0.015	6	70	25	0.25	5-8%	19,000	3.5	BUY NOW

FEEDS & SPEEDS – 0-FLUTE

The 0-flute bit is used for acrylic and aluminum, depending on the design of the bit. The bits and settings listed here are for acrylic only.

Always use CAST ACRYLIC when creating these types of projects. Click these links to see it on Amazon.

1/8” thick clear – 1/4” thick clear



0-FLUTE BIT (For Acrylic)												
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Feed (in/min)	Plunge (in/min)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
1/8"	0.125	1	0.500	1.50	0.125	60	15	0.125	70%	17,000	3	BUY NOW
1/4"	0.25	1	1.000	2.00	0.25	60	15	0.25	70%	17,000	3	BUY NOW

METRIC

FEEDS & SPEEDS
FOR CNC ROUTER BITS

PLEASE READ

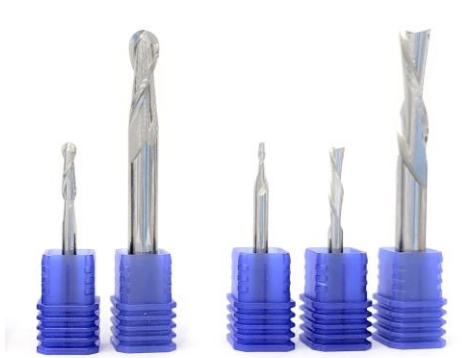
All *Feeds & Speeds* from this point forward are in millimeters (mm).

The ‘Bit’ column refers to the inch size the bits are normally labeled as.

NOTE: All feed units use the CNC industry standard of ***mm/second***

METRIC FEEDS & SPEEDS for Complete CNC Router Bit Starter Set

ENDMILLS												
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Feed (mm/sec)	Plunge (mm/sec)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
1/16" Up Cut	1.587	2	4.8	38	3.175	8	4	3.175	40%	27,000	5	BUY NOW
1/8" Down Cut	3.175	2	19.0	44	3.175	14	6	3.175	40%	22,000	4	BUY NOW
1/4" Down Cut	6.35	2	25.4	63	6.35	33	13	6.35	40%	19,000	3.5	BUY NOW
1/8" Ballnose	3.175	2	12.7	44	3.175	14	6	6.35	40%*	22,000	4	BUY NOW
1/4" Ballnose	6.35	2	19.0	63	6.35	29	13	3.0	40%	19,000	3.5	BUY NOW



ENDMILLS

V-BIT															
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Side Angle	Feed (mm/sec)	Plunge (mm/sec)	Depth Per Pass	Clear Pass Stepover	Final Pass Stepover	Spindle (rpm)	Router Dial		
30° V-bit	6.35	1	11.8	63	6.35	15°	15	8	0.64	20%	0.13	27,000	5	BUY NOW	BUY THE SET (save $$)
60° V-bit	6.35	2	5.5	51	6.35	30°	17	8	1.3	20%	0.13	22,000	4	BUY NOW	
90° V-bit	6.35	2	3.175	63	6.35	45°	21	10	2.5	20%	0.13	17,000	3	BUY NOW	

V-BITS

1/8" DRILLING ENDMILL *See notes below												
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Feed (mm/sec)	Plunge (mm/sec)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
Drilling	3.175	2	19	38	3.175	25	25*	5	40%	22,000	4	BUY NOW
Conventional	3.175	2	19	38	3.175	25	8	5	40%	22,000	4	
* The plunge value is for using the spiral drilling technique. Watch this video to learn how									Watch Video			

SPIRAL DRILLING UP-CUT ENDMILL

METRIC CNC ROUTER BIT FEEDS & SPEEDS

SECTION 2

ALL COMMON CNC ROUTER BITS FEEDS & SPEEDS

METRIC FEEDS & SPEEDS – DOWN CUTTING ENDMILLS

Down cutting endmills, also known as 'down bits', are the most common bit you will use for material removal and getting squared edges on your projects.

Despite the benefits of creating clean sharp corners and edge, they do have a drawback. This *MIST WATCH* video will explain what it is and how to overcome it.

Items with GREEN '→' are bits you want to have backups of. You will use these most often.

DOWN CUT ENDMILL												
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Feed (mm/sec)	Plunge (mm/sec)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
1/32"	0.79	2	0.82	38	3.175	4	3	0.381	40%	27,000	5	BUY NOW
1/16"	1.587	2	6.35	38	3.175	8	4	0.81	40%	27,000	5	BUY NOW
→ 1/8"	3.175	2	19	44	3.175	15	6	3.175	40%	22,000	4	BUY NOW
3/16"	4.763	2	25	60	6.35	29	13	4.75	40%	19,000	3.5	BUY NOW
→ 1/4"	6.35	2	25	63	6.35	34	13	6.35	40%	19,000	3.5	BUY NOW

METRIC FEEDS & SPEEDS – UP CUTTING ENDMILLS

Up cutting endmills, also known as ‘up bits’, are best used to remove lots of material quickly while leaving a very good finish along the bottom surface.

They can be used in place of down cutters with proper feeds and speeds. A ggod time to use this bit is when a V-bit is used to accent an edge of a large area that is recessed.

The bit with the GREEN ‘→’ is the one you want to have. If you plan on using this often, get a backup bit.

UP CUT ENDMILL												
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Feed (mm/sec)	Plunge (mm/sec)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
1/32"	0.79	2	3.175	51	3.175	4	3	0.4	40%	15,000	5	BUY NOW
1/16"	1.587	2	60	63	3.175	8	4	0.8	40%	15,000	5	BUY NOW
→ 1/8"	3.175	2	19	38	3.175	21	8	4	40%	15,000	4	BUY NOW
3/16"	4.763	2	19	51	6.35	34	17	7	40%	15,000	3-3.5	BUY NOW
→ 1/4"	6.35	2	19	63	6.35	37	17	9	40%	15,000	3-3.5	BUY NOW

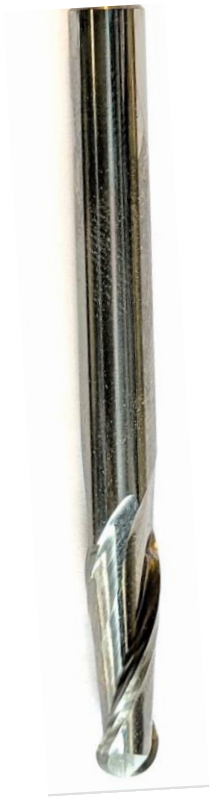
METRIC FEEDS & SPEEDS – BALL NOSE ENDMILL

The ball nose endmill is categorized as an endmill. The difference is the rounded end.

It is used to accent your projects, such as putting a rounded edge on a sign. You can also use them to do 2.5D or 3D relief carves when extremely high detail is not a concern, but you still want a good-looking project.

To learn about this application, watch this video.

It is standard to have both in your set of bits. You do not need backups of these unless you find you use them a lot.



BALLNOSE												
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Feed (mm/sec)	Plunge (mm/sec)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
1/8" Ballnose	3.175	2	12	38	3.175	15	6	1.3	20%*	14,000	4	BUY NOW
1/4" Ballnose	6.35	2	19	63	6.35	29	13	3.0	20%	13,000	3-3.5	BUY NOW

* When using 1/8“ ballnose for 2.5D & 3D relief carves, set stepover to 5-8%

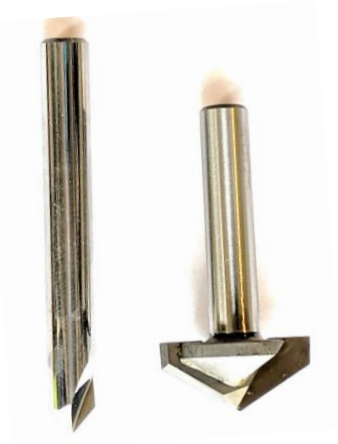
METRIC FEEDS & SPEEDS – V-BITS

You will notice more fields in this section for V-bits than for the previous bits. That's because more information is required for your software to create toolpaths.

You might find some fields aren't even in your software. Don't worry., you won't need them. They are for more advanced engineering type software programs.

The bit with the **GREEN '→'** are typical backups. However, it is suggested to get the 120 V-bit since it easily does the job of the 90-degree bit. And it can do much larger projects efficiently.

V-BIT														
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Side Angle	Feed (mm/sec)	Plunge (mm/sec)	Depth Per Pass	Step Over	Final Pass Stepover	Spindle (rpm)	Router Dial	
30° V-bit	6.35	1	11	63	6.35	15	15	8	0.6	20%	0.13	27,000	5	BUY NOW
→ 60° V-bit	6.35	2	5.5	51	6.35	30	17	8	1.3	20%	0.13	22,000	4	BUY NOW
→ 90° V-bit	6.35	2	3.2	51	6.35	45	21	10	2.5	20%	0.13	17,000	3	BUY NOW
120° V-bit	6.35	2	4.5	38	6.35	60	34	25	4.8	30%	0.25	17,000	3	BUY NOW



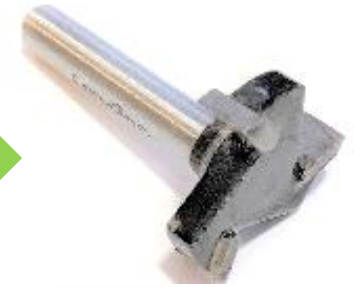
METRIC FEEDS & SPEEDS – SURFACING & BOWL BITS

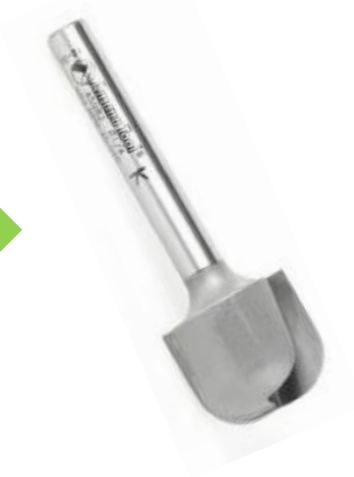
Here, you will find 2 types of bits. The surfacing bit and bowl bit. Each serve a different purpose.

The surfacing bit is an **ABSOLUTE MUST HAVE**! It is used to surface your spoilboard and smooth warped material or botched projects. Watch this video to learn more about surfacing a spoilboard.

The bowl bit is used to remove large amounts of material without a lot of force applied to your machine. This is commonly used to remove material from deep pockets. It is an extremely handy bit to have.

SURFACING BIT - FOR SURFACING SPOILBOARD (Category: Endmill)												
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Feed (mm/sec)	Plunge (mm/sec)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
1"	25.4	3	6.35	38	6.35	34-42	3	3.175	70%	17,000	3	BUY NOW
BOWL BIT - HOGGING (Category: Endmill)												
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Feed (mm/sec)	Plunge (mm/sec)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
3/4” Bowl	19	2	.19	63	6.35	34-42	6	9.5	40%	17,000	3	BUY NOW





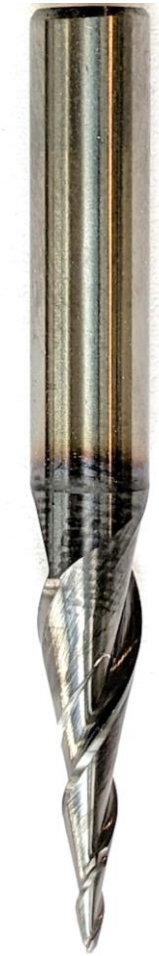
METRIC FEEDS & SPEEDS – TAPER BALL NOSE CARVING BIT

The taper ball nose bit is use for fine detail carving. This is the bit to use for exquisite projects.

You may notice the image is a bit ‘stumpy’ for this type of bit. Most carving bits tend to be longer. The major drawback to long, narrow bits in this family is ‘tool deflection’.

A long narrow bit will have side bounce which will show up as lines in your project.

To learn more about taper ball nose bits and how they are used, watch this video.



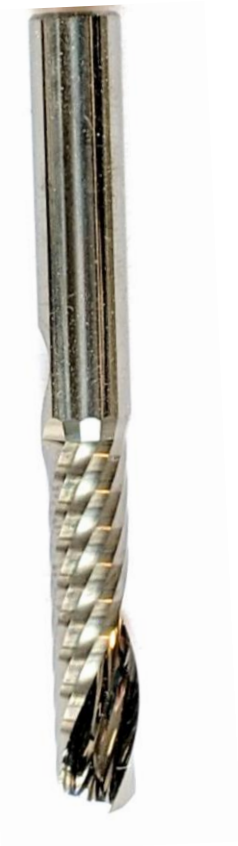
TAPER BALLNOSE CARVING BIT													
Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Tip Radius	Angle / Side Angle	Feed (mm/sec)	Plunge (mm/sec)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
1mm rad.	2	19	51	6.35	.38	12/6	29	10	6.35	5-8%	19,000	3.5	BUY NOW

METRIC FEEDS & SPEEDS - 0-FLUTE

The 0-flute bit is used for acrylic and aluminum, depending on the design of the bit. The bits and settings listed here are for acrylic only.

Always use CAST ACRYLIC when creating these types of projects. Click these links to see it on Amazon.

1/8” thick clear – 1/4” thick clear



0-FLUTE BIT (For Acrylic)												
Bit	Cut Dia.	# Flutes	Flute Length	Overall Length	Shank Dia.	Feed (mm/sec)	Plunge (mm/sec)	Depth Per Pass	Step Over	Spindle (rpm)	Router Dial	
1/8"	3.175	1	12	38	3.175	25	6	3.175	70%	17,000	3	BUY NOW
1/4"	6.35	1	25	51	3.175	25	6	6.35	70%	17,000	3	BUY NOW