CNC Tooling Guide
Quick Start Parameters and Creating New Tools in the Bit Library

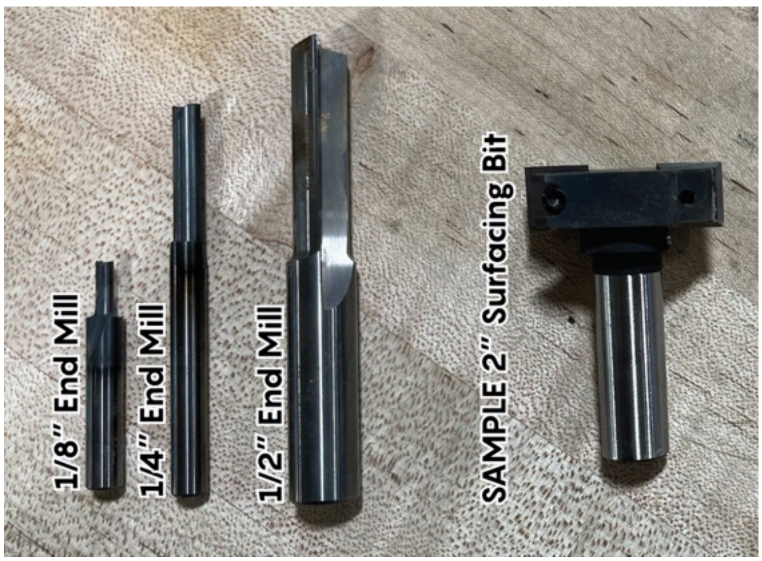
What is a router bit?

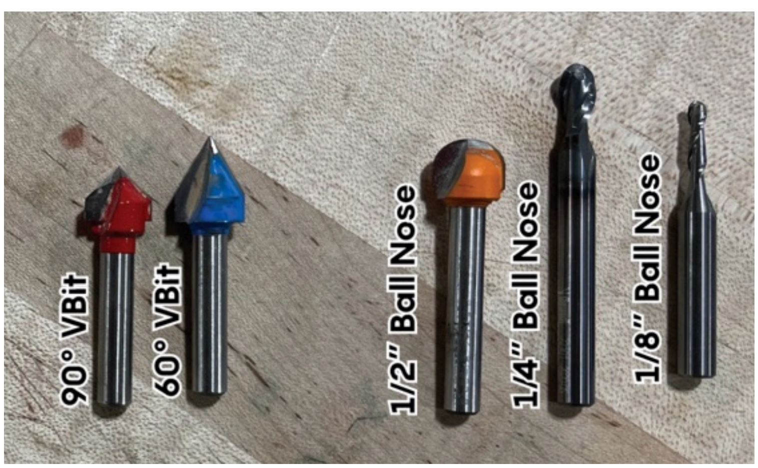
A router bit is the business end of a router or spindle. Its purpose in life is to remove material.

We have a selection of bits available for use at Maker Playground. If you are looking for a more specific router bit, you may need to bring in your own. If you are surfacing slabs or river tables, you will need to provide your own surfacing bit.

Plug-n-Play Bit Parameters

Quick, general-use parameters to enter for commonly used bits we have available in the shop. Take note that **all feed units are in “inches/sec”.**





*Some bits may look different from these simply because of a different shank size or bit length but still represent the same cut parameters.

End Mills

End Mill (1/8")

Notes				
Tool Type	End Mill			
Geometry				
Units	inches			
Diameter (D)	0.125	inches		
No. Flutes	2			
Cutting Parameters				
Pass Depth	0.0625	inches		
Stepover	0.05	inches	40	%
Feeds and Speeds				
Spindle Speed	18000	r.p.m		
Feed Units	inches/sec	Chip Load	0.0033	inches
Feed Rate	2	inches/sec		
Plunge Rate	0.25	inches/sec		
Tool Number	3			

End Mill (1/2")

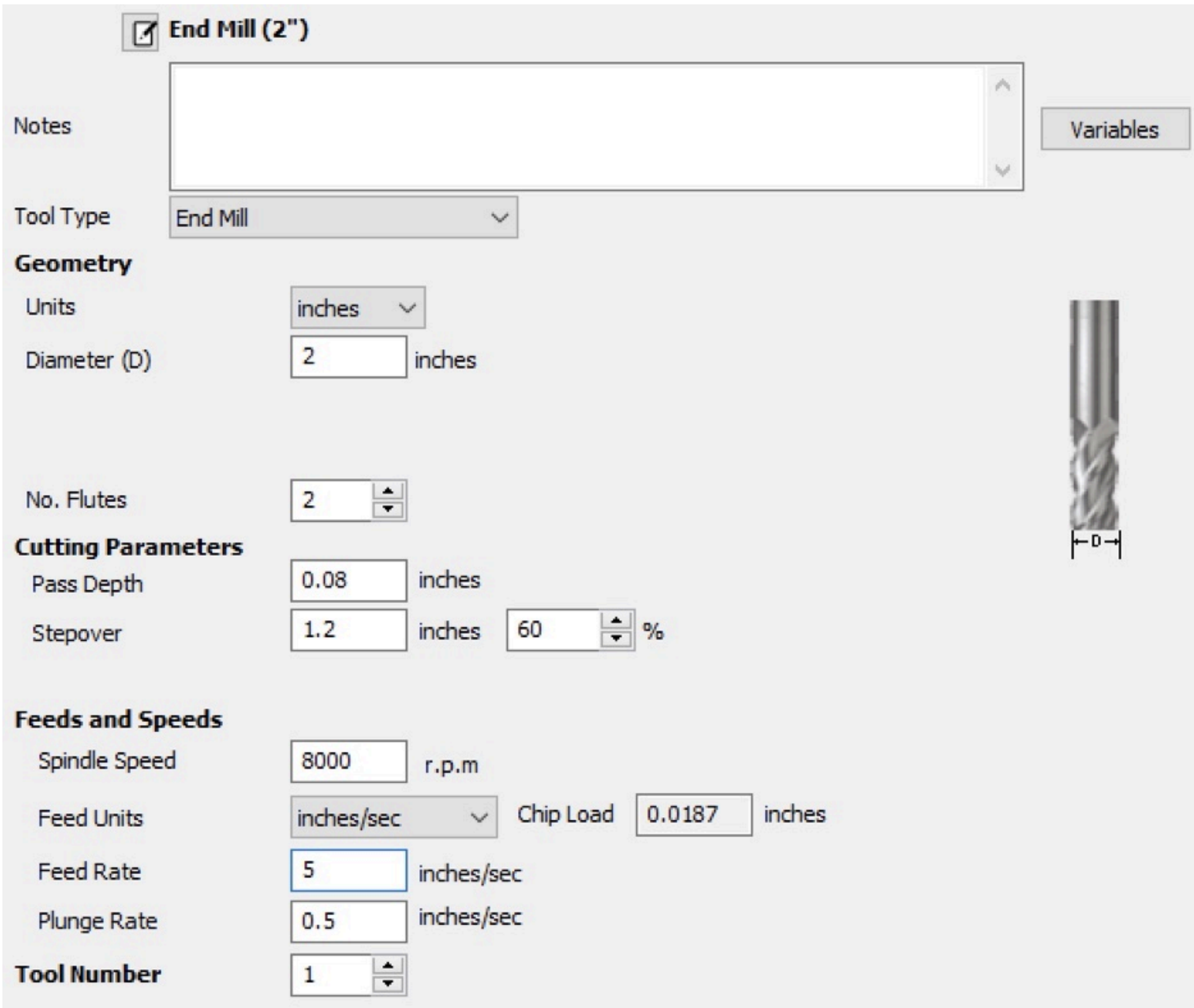
Notes				
Tool Type	End Mill			
Geometry				
Units	inches			
Diameter (D)	0.5	inches		
No. Flutes	2			
Cutting Parameters				
Pass Depth	0.25	inches		
Stepover	0.2	inches	40	%
Feeds and Speeds				
Spindle Speed	12000	r.p.m		
Feed Units	inches/sec	Chip Load	0.01	inches
Feed Rate	4	inches/sec		
Plunge Rate	.5	inches/sec		
Tool Number	4			

End Mill (1/4")

Notes				
Tool Type	End Mill			
Geometry				
Units	inches			
Diameter (D)	0.25	inches		
No. Flutes	2			
Cutting Parameters				
Pass Depth	0.125	inches		
Stepover	0.1	inches	40	%
Feeds and Speeds				
Spindle Speed	14000	r.p.m		
Feed Units	inches/sec	Chip Load	0.0086	inches
Feed Rate	4	inches/sec		
Plunge Rate	0.5	inches/sec		
Tool Number	1			

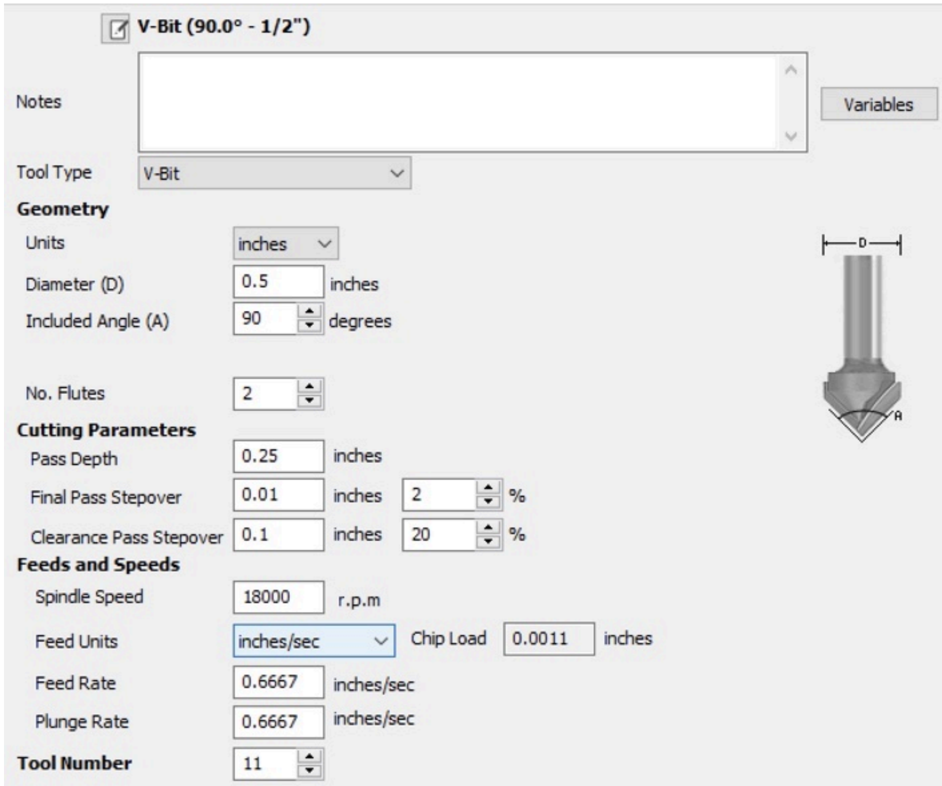
Surfacing Bits*

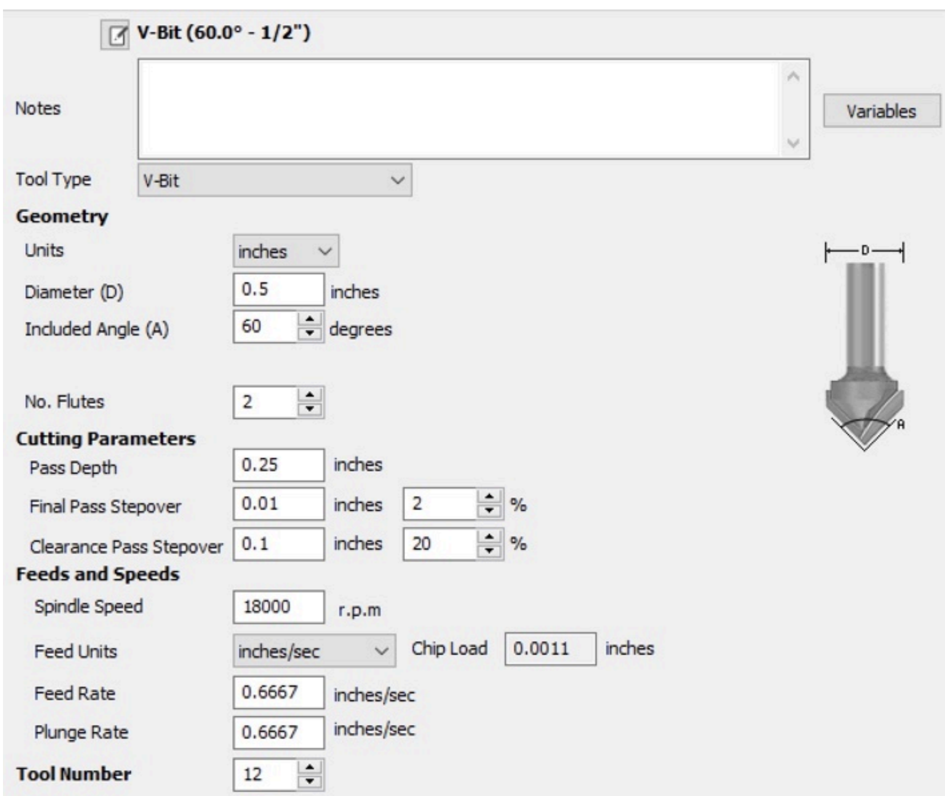
Maker Playground will not provide surfacing bits, so these are sample numbers for similar surfacing bits.



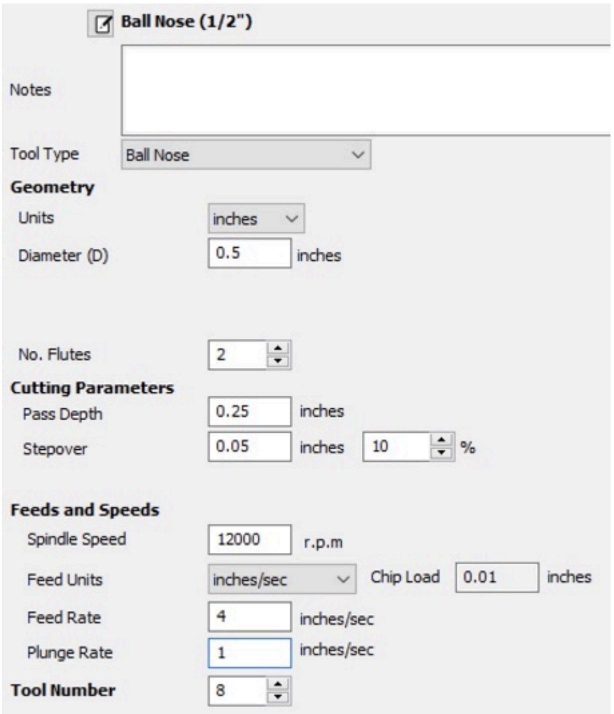
Surfacing Bits are NOT made for plunging, so the bit must ALWAYS plunge while off of the material before feeding in. Larger bits will want to go faster - aim for a chip load between .015 and .025, the higher being for 4” bits. Do not exceed ⅛” per pass.

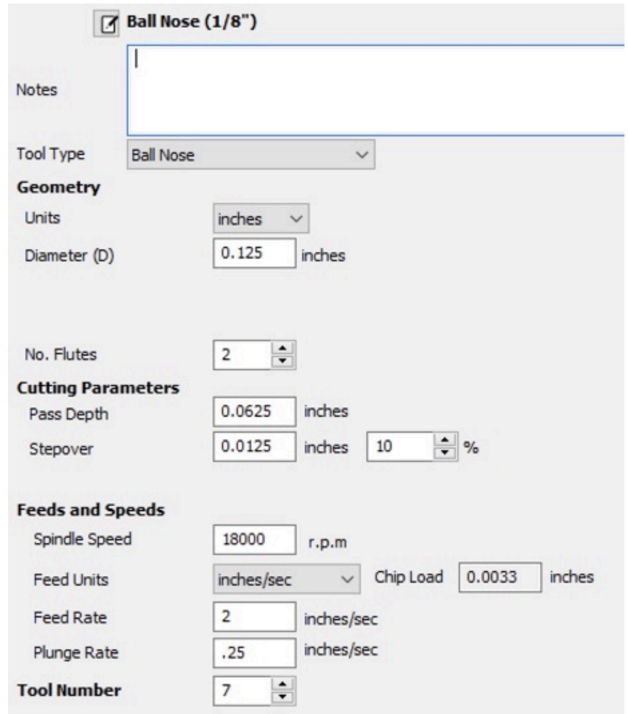
V-Bits

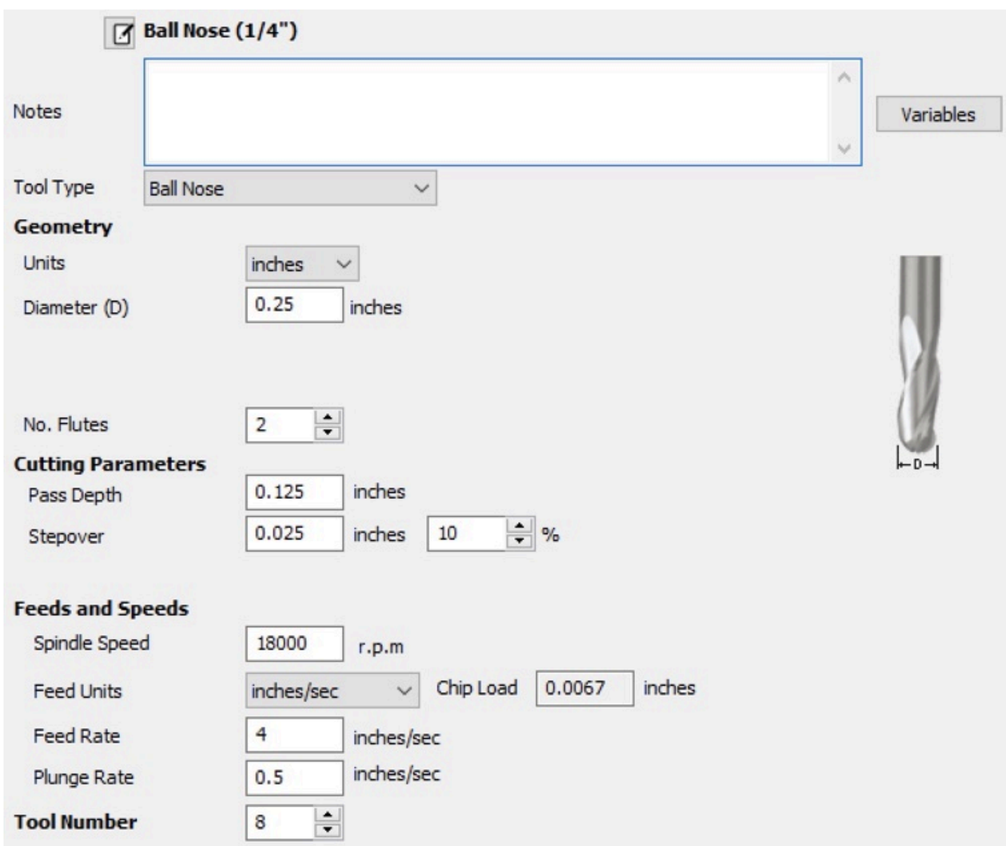

V-Bit (90.0° - 1/2")
Notes
Variables
Tool Type V-Bit
Geometry
Units inches
Diameter (D) 0.5 inches
Included Angle (A) 90 degrees
No. Flutes 2
Cutting Parameters
Pass Depth 0.25 inches
Final Pass Stepover 0.01 inches 2 %
Clearance Pass Stepover 0.1 inches 20 %
Feeds and Speeds
Spindle Speed 18000 r.p.m
Feed Units inches/sec
Chip Load 0.0011 inches
Feed Rate 0.6667 inches/sec
Plunge Rate 0.6667 inches/sec
Tool Number 11
D
A


V-Bit (60.0° - 1/2")
Notes
Variables
Tool Type V-Bit
Geometry
Units inches
Diameter (D) 0.5 inches
Included Angle (A) 60 degrees
No. Flutes 2
Cutting Parameters
Pass Depth 0.25 inches
Final Pass Stepover 0.01 inches 2 %
Clearance Pass Stepover 0.1 inches 20 %
Feeds and Speeds
Spindle Speed 18000 r.p.m
Feed Units inches/sec
Chip Load 0.0011 inches
Feed Rate 0.6667 inches/sec
Plunge Rate 0.6667 inches/sec
Tool Number 12
D
A

Ball Nose Bits



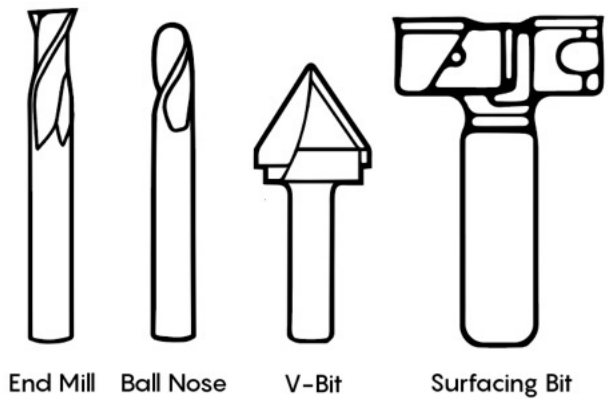




Shopping for your own Bits

If you're looking for something more specific than what we can provide at Maker Playground, here's some basics you'll need to know to figure out what you'll need.

Basic Tool Types



End Mills - End mills are often used for simple but versatile cutting paths such as cutting profiles to create shapes, pockets with square bottoms, dados, etc.

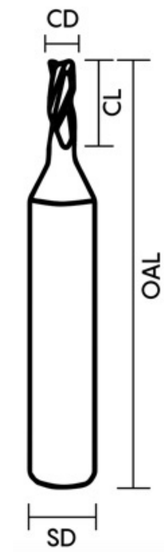
Ball Nose Bits - Larger ball nose bits (½" or more) are often used for pocketing out bowls and dishes or profile cutting juice grooves. Smaller ball nose bits (¼" or less) are often used for finish passes when 3D milling. Smaller ball nose bits will be able to create more detailed 3D milled models, but will take much longer to carve.

V-Bits - V-bits are often used for VCarve toolpaths, and may also be used for shallow profiles to achieve decorative lines or chamfers along existing cuts. The larger the angle, the shallower VCarve toolpaths will cut, but it will be more critical for the surface to be level to make sure your cuts are all carved at the perfect height.

Surfacing Bits - Surfacing bits are used for surfacing hardwood slabs wider than will fit in our planer (20") or end grain cookies. They can also be used to flatten composite material to ensure a completely flat bed when cutting a very precise project. The larger the bit, the faster your project will be surfaced. However, our machine has maximum speed parameters that will limit this, so 2 inches is perfectly sufficient for most surfacing jobs.

Some brands we recommend include Freud, Onsrud, CMT, Amana, and Whiteside.

Bit Sizes



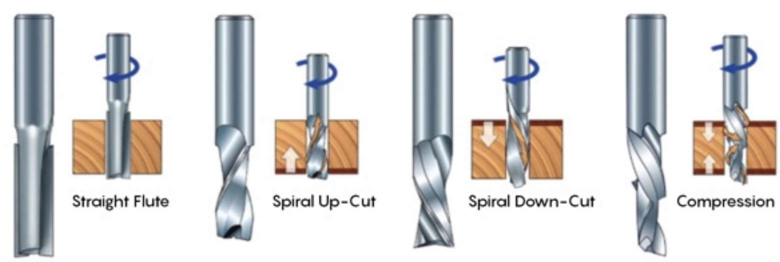
Cutting Diameter (CD) - This is the width of the cut the tool will make through the workpiece. Smaller diameters will cut a smaller kerf and have a smaller radius on inside corners, but will cut much slower than large cutting diameters. They may also have more tool deflection (covered more later in this guide).

Cutting Length (CL) - Cutting length is the length of cutting flutes on the end of the bit. This is the absolute maximum depth that can be cut at once (though typically not recommended in most applications at MPG). If the cutting diameter and shank diameter are the same, you may be able to plunge the bit deeper than the cutting length to reach a desired depth, but never while doing stepover cuts deeper than the cutting length.

Shank Diameter (SD) - This is the diameter of the body of the bit where it goes into the collet. Shank and cutting diameter may sometimes be the same, but not always, as is pictured in the bit to the left. A larger diameter bit will deflect less and give a smoother, more accurate cut.

Overall Length (OAL) - This is the total length of the tool from end to end. A bit will typically go into the collet 1" or more. Consider this length when cutting thicker materials or needing the bit to reach deep into a pocket or 3D milling project. Also, remember that the length that the bit is out of the collet reduces your usable max Z height that amount. The longer the OAL, however, the more the tool may deflect (covered more later in this guide).

Flute Direction



Straight Flute - Good, general purpose bit with good edge quality on most materials.

Spiral Up-Cut - May chip the top face of the material, but will have a good quality cut on the bottom. Much better at chip clearing due to the chips being directed upwards and out of the cut. Can create pulling force on the material, lifting unsecured sections upwards. These bits are ideal for plastics, aluminum, or any material where it is necessary to reduce heat buildup.

Spiral Down-Cut - Will leave a good quality cut on the top, but may chip the bottom face while cutting through. Chips may be compressed downwards into the cut. The downwards force assists with cutting very thin sheets.

Compression - Has flutes going upwards on bottom and downwards on top. Clean edge on both the top and bottom face when cutting all the way through materials. However, the first pass must go deep enough to reach the up-cut flutes of the bit, or it will act as an up-cut bit and chip out the top face.

Setting up a New Bit in the Tool Library

If you bring in your own bit, or want to use one that is not listed above, follow this guide to understand how to create a new bit and what to enter for its parameters.

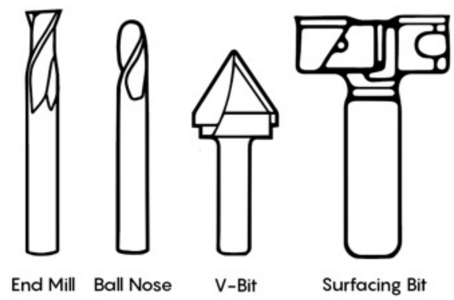
The tool library will automatically name your bit based on the entered “Tool Type” and “Diameter.”

Notes

Any notes about the tool can be added here.

Tool Type

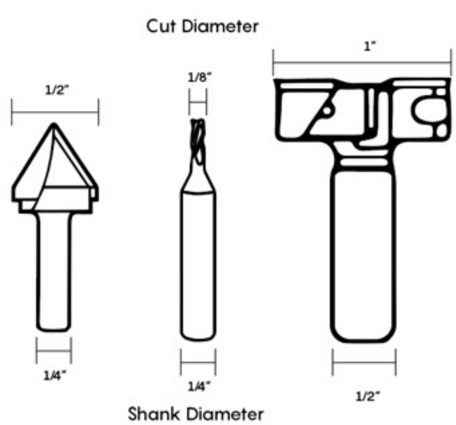
Select your tool type. When building out surfacing bits, just select “End Mill” and be sure to enter the correct diameter for the bit.



Geometry - These parameters all relate to the size and shape of the selected bit.

Diameter

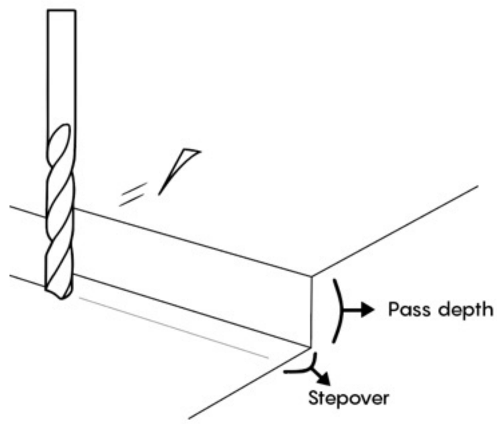
The diameter refers to the widest point of the **cutting portion** of the bit. See below. Sometimes, the shank and cutting diameter are the same measurement. In all the examples below, they are all different sizes.



Number of Flutes

The number of cutting flutes for the bit. Make sure to make this exact to your particular bit to use the Chip Load calculator, covered later on.

Cutting Parameters - These parameters all relate to how much material is being removed.



Pass Depth

The maximum depth of cut the tool can cut. The Pass Depth controls the number of z level passes that are calculated for a toolpath. This amount should generally be about half of the tool diameter. For example, ¼" tools will have a pass depth of ⅛". For surfacing bits, we usually use a conservative pass depth (no more than ⅛") and use a higher stepover (60-80%), explained further below.

Stepover

The distance the cutter moves over in the X or Y direction when cutting pockets or peripheral milling, as referred to previously. The greater the stepover, the faster the job will be machined, but this must be balanced with other parameters to make sure the tool can cool efficiently. 40% is a good amount to start with.

When doing a VCarve toolpath, you will instead see the following:

Final Pass Stepover

The distance the v-bit moves over when finish machining a flat depth and is usually set to be a relatively small distance to produce a smooth surface finish.

Clearance Pass Stepover

Only used when the v-bit is being used to rough machine at multiple Z levels. This stepover can be much larger than the Final Pass Stepover because the tool is only rough machining material away.

Feeds & Speeds - These parameters all relate to how fast the CNC moves the bit through the material. Also refer to the next section for help with finding a good Spindle Speed and Feed Rate based on Chip Load.

Spindle Speed (The "Speeds" in "Feeds & Speeds")

Speed of tool rotation, specified in revolutions per minute (RPM). The speed must not be too fast relative to the feed rate in order to prevent the bit overheating and becoming dull.

Feed Rate (The "Feeds" in "Feeds & Speeds")

The rate at which the bit is moving and cutting through the material in the X or Y directions. The feed must not be too fast relative to the speed in order to prevent the bit from becoming overloaded and breaking.

Plunge Rate

The cutting rate at which the cutter is moved vertically into the material or during ramping moves. This should be slower if you are not using ramps, since many CNC tools are not meant for plunging straight down.

Chip Load (The Magic Number)

This is the calculated chip load based on the entered values for the **Number of Flutes**, **Spindle Speed,** and **Feed Rate**. This is displayed to conveniently be able to compare it with manufacturer-recommended chip load values.

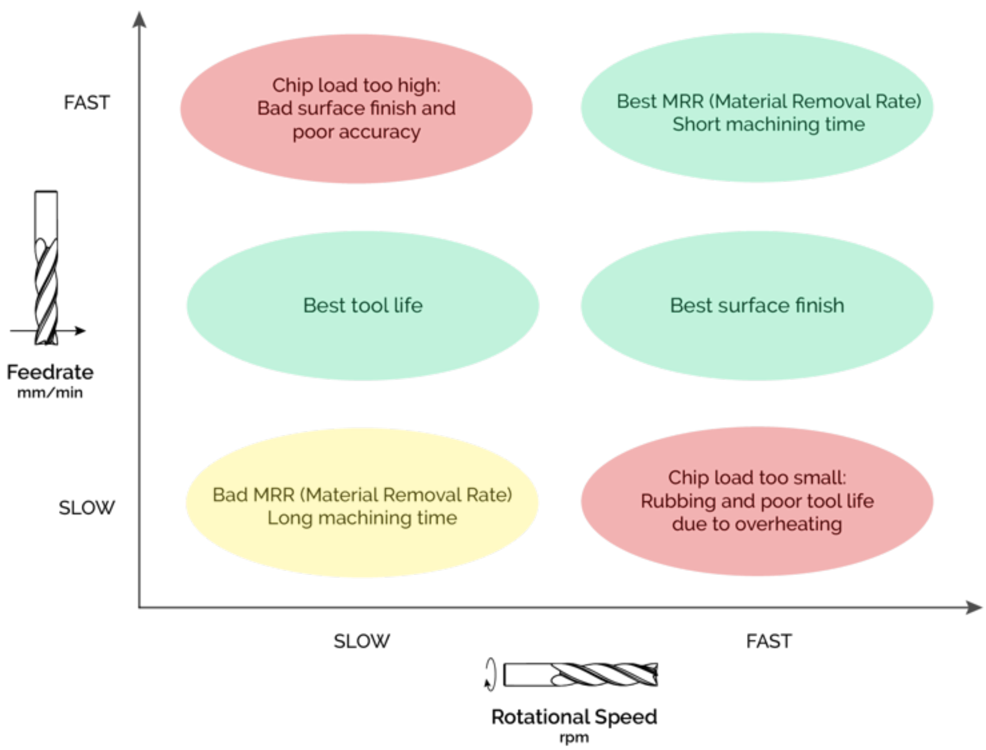
Tool Number

Since we do not have an Automatic Tool Changer on our ShopBot, this number is arbitrary. However, **make sure this number is less than 20**, or the machine thinks you have a second spindle and will not start up correctly.

Calculating “Feeds & Speeds” Using Chip Load

Getting your feeds & speeds right simply means finding the sweet spot where your tool is spinning at the perfect speed relative to its moving speed inside the material.

These concepts can be visually summarized on a graphic, where the feed rate is plotted against the spindle speed and helps us to identify 6 different zones.



As illustrated above, there are mainly two bad spots that you want to avoid. The first one happens when you reduce your spindle speed too much relative to the feed rate. Doing so, you’re forcing the flutes of your end mill to cut off too much material, which can lead to unwanted vibration or worse, a broken tool.

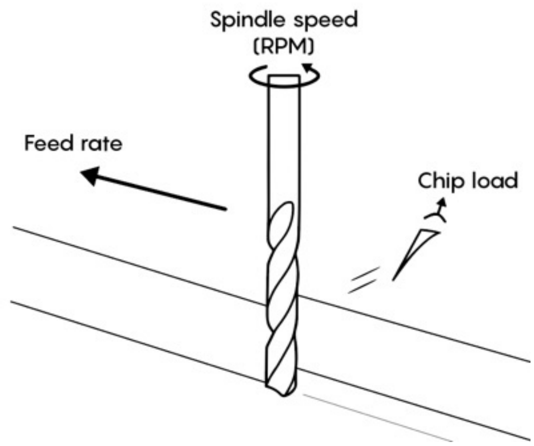
On the other side of the graphic, if you reduce the feed rate too much relative to spindle speed, the flutes of your end mill will start rubbing the material instead of cutting nice chips. This action will make your tool overheat, and thus soften. Its sharp edges will become dull, and if you keep cutting with dull edges, you will start to see a very deteriorated surface finish on your material.

A good rule of thumb is to always remember that you need to make chips, not dust. The question is now: how do we find the sweet spots for any given material? This is where chip load comes in!

Chip Load

The parameter that links Feeds & Speeds and is widely used as a standard metric to determine optimal rates is called chip load.

Chip load, also called “feed per tooth”, is the thickness of material that is fed into each cutting edge as it moves through the work material. A chip load of .01 inches per tooth means that bit is taking away approximately that amount per tooth while rotating, and the chips will be approximately .01 inches in thickness.



Chip load is expressed in inches/tooth and can be found using the following equation:

Chip Load = Feed Rate (inches per minute) / (RPM x number of flutes)

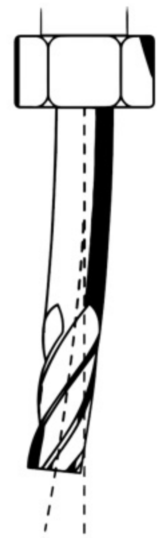
This calculation is done automatically by our tool library using what is entered in the **number of flutes**, **spindle speed**, and **feed rate** sections. This allows you to tweak your feed and speed while keeping an eye on the chip load. **Take note that these parameters have a direct relationship - the faster your spindle speed, the faster your bit will need to travel in order to maintain the same chip load.**

Here is a basic chip load chart as a good baseline for wood (note, these are fairly conservative numbers compared to what you may find online):

Bit Diameter	Chip Load
1/8”	.001-.004”
1/4”	.005-.008”
1/2”	.009-.011”
3/4”	.012-.014”
1” +	.015-.020

Larger bits will typically need a slower RPM and a faster feed rate to achieve the larger recommended chip load. Smaller bits will typically need a faster RPM and slower feed rate to achieve the smaller recommended chip load. Look at the "Plug-n-Play Bit Parameters" in the beginning of this guide to see examples of this.

Once you feel more comfortable understanding feeds and speeds, you may also want to adjust your chip load based on the following:



Hardness of the material

The harder the material, the more your end mill will deflect, as shown to the right. This will cause chatter and vibrations. Be patient when milling denser material and use a lower chip load.

Sharpness of the bit

If necessary to use a more dull end mill, reduce the chip load to keep a good surface finish.

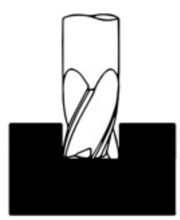
Depth of cut

A general rule of thumb is to take passes that are around half the diameter of your tool diameter. If you want to cut deeper, you need to lower your chip load.

Tool engagement

During some toolpaths, the tool's entire circumference "touches" or engages the material during the cut. As a result, the end mill can't cool down as easily and is likely to overheat. Slotting will need to use a lower chip load or a more reduced depth of cut than Peripheral Milling.

Peripheral Milling
(pocket toolpaths)



Slotting
(profile toolpaths)

Finding the perfect feeds and speeds relative to bit type, material, and a particular machine comes with lots of experimentation and experience (and sometimes a few broken bits!) Additional resources for feeds, speeds, and cutting different materials include Onsrud Cutting Data Recommendations and Shopbot or Vectric forums.

If you need assistance, as always, feel free to ask Maker Playground staff.