Your first E-series / Evolution / Quantum bits

A little information and advice for new CNC owners

What bits should I buy?

I probably see this question every week from new and prospective CNC buyers (note: I was one of the people asking that question back in 2017), and I see lots of responses from lots of helpful folks, but I worry that the myriad responses are causing more confusion than clarity. Even worse, I worry that the new folks are spending money on bits that they won't use (you don't have to look too hard to find some mint condition bits in my drawer … heavy sigh).

This document provides my opinions, as well as a brief overview of the most common types of bits. I hope you find it useful.

> **Important Note:** If your router included a spiral bit, please do note use it with your CNC. Trim router bits are designed for cutting out drywall and OSB, and you will have difficulty getting reasonable results using it in your CNC.

Spoiler Alert! For those who don't want to read the whole document (hey, it's only 6 pages and has lots of pictures … but whatever), here is my answer to the above question: Before you spend a bunch of money on bits, go to your local DIY store and buy a 60° v-groove bit with a 1/4" shank, similar to the one pictured at right. It will cost you about $12—$15 and will give you a good general-purpose bit that you can use as you learn to program and operate your new machine.

If you're already past that point, or just want to learn more, read on! At the end of this document I have a list of bits that would make a good starter kit for the beginner.

But before we jump in, here are a few general terms that I think will be useful:

Engraving vs Carving. In this document I use the term Engraving to mean constant depth cutting, and Carving (or VCarving) to mean variable depth cutting. I'm not sure those are exact definitions in the industry, but that's how I think of them so you're stuck with them, at least for this document.

Shank Diameter. The shank is the end of the bit that you will insert into the router collet.

The DW660 router that was included with the E3 and E4 comes with two collets, one each for 1/8" and 1/4" shanks (left pic).

The DW611 and Makita RT701 routers that could/can be purchased with the Evolution and Quantum CNCs only include a 1/4" collet, so you'll need to purchase a 1/8" collet (or an adapter) if you want to use bits with smaller shanks. (right pic, Makita collets, showing a purchased 1/8" collet[1] along with the stock parts)

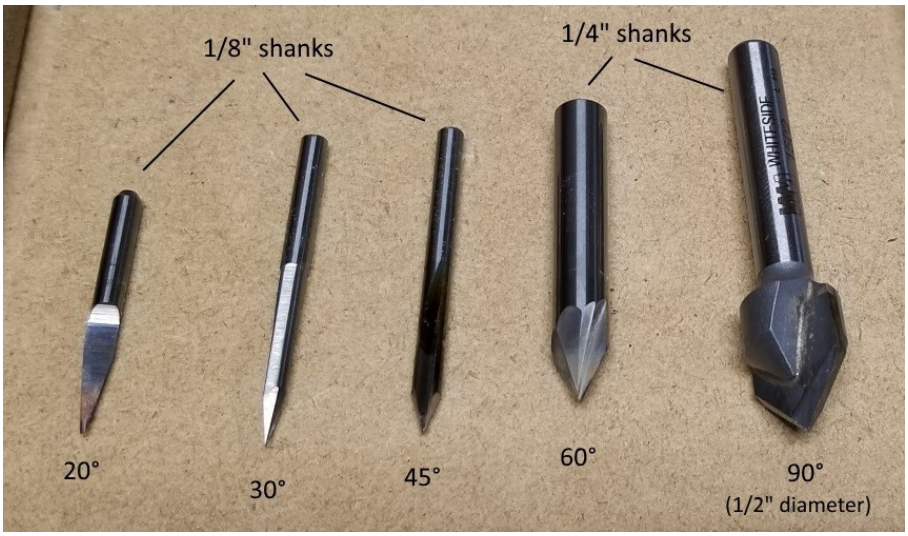
You'll want to pay attention to the shank diameter when you buy bits to ensure they will fit into the collets that you have.

Vbits. You're almost certainly going to want to own at least one or two vbits, which can be used for engraving, but are more typically used for carving (which then is usually referred to as "vcarving" for the obvious reasons).

The cool part about vbits is that they can make variable-width cuts simply by changing depths … going deeper makes a wider cut, and shallower makes a narrower cut. See example at right, which was done with a single bit in a single carve, just at varying depths.

Vbits are primarily used for:

- Lettering
- Graphics with variable-width lines



As you can see here, vbits come in all shapes and sizes.

The key parameters when talking about a vbit are:

1. the cutting angle, and
2. The cutting diameter.

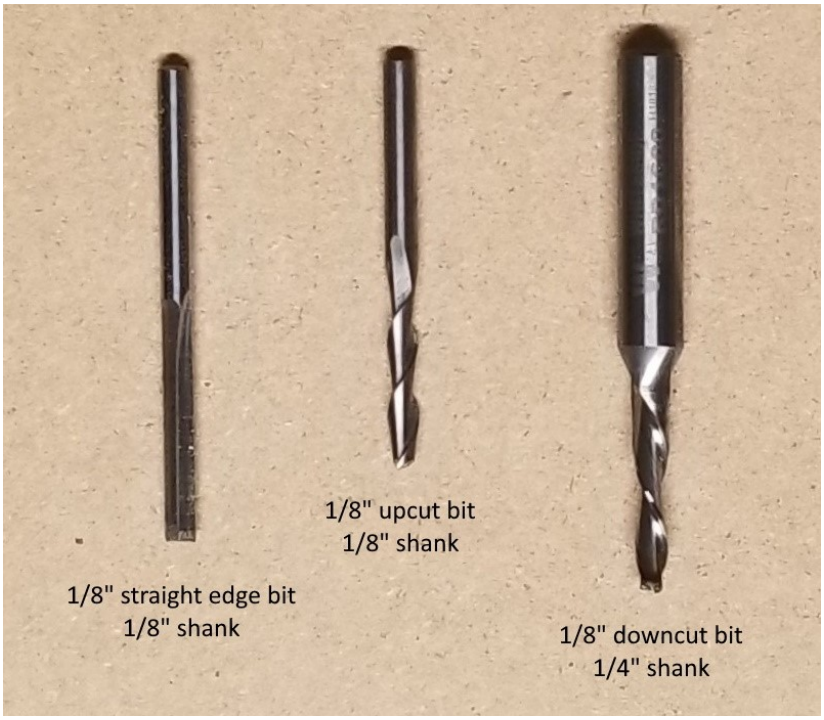
Together, these two parameters determine how deep the bit can carve.

For example, the 60°, 1/4" diameter vbit (second from the right in the photo) can cut up to about 7/32" deep … the 1/2" diameter 90° bit at far right, despite being a much larger bit, can only cut slightly deeper … about 1/4" … because of the larger angle.

There are also vbits with flat tips that can be used when you don't want a sharp "V" cut, and I imagine they would work well when you're carving the outline of something that you're going to clear out with flat bit (see below), but I haven't used them so I am only mentioning them in passing.

General rule of thumb: the more small details in your carve, the smaller angle vbit you should use. A 90° vbit will work wonderfully for a carving someone's name on an 8" x 18" sign, but you'll want something more like a 30° or 45° to carve the wording into that business card holder you're making for your boss (which I'm sure he / she will love!)

Flat End Mills. These bits have straight sides and flat ends (thus the name!).



Primary used for:

- Removing material while leaving a flat surface
- Cutting out shapes

There are a number of parameters that can be selected for flat end mills, including:

- Cutting diameter
  - All of the bits at right are 1/8”
  - Common sizes also include 1/16” and 1/4”
- Shank diameter (the bit on the far right has a 1/4” shank, while the other two have 1/8” shanks).
- For spiral end mills it’s also important to note the direction of cut, which are commonly referred to as upcut or downcut:
  - The bit on the left, having straight edges, is neither upcut nor downcut, of course.
  - The bit in the middle is an upcut bit, which means that as it rotates the cutting edges are pulling up on the wood, which is good for clearing chips. Upcut is the most common type of spiral bit.
  - The bit on the right is a downcut bit, which is good for leaving a cleaner edge on the top surface of the wood because the fibers are being cut downward instead of upward. Note the opposite direction of the spiral compared to the upcut bit. However, because of this motion, a downcut bit is a very bad choice for drilling or cutting in close quarters because it will not be able to clear the chips as it cuts.
  - Another type of bit that is sometimes mentioned is the compression bit, which is upcut on the bottom and downcut on the top, thereby trying to use the best of both types, but they are an iffy choice for hobby CNCs because they require a first pass that is deep enough so that the upcut portion is completely under the surface of the wood.
- Number of flutes (cutting edges) All of the bits shown have two flutes, which will be the most common type for wood, but you can find bits with other quantities of flutes as well. The reason for adding flutes is to achive more cuts per revolution of the spindle, which isn’t a concern for CNCs that use routers, so sticking with 2 flutes for starters will be a good option.

Ball Nose End Mills. These are similar to their flat-ended cousins, but with a radiused ('ball nose") end.

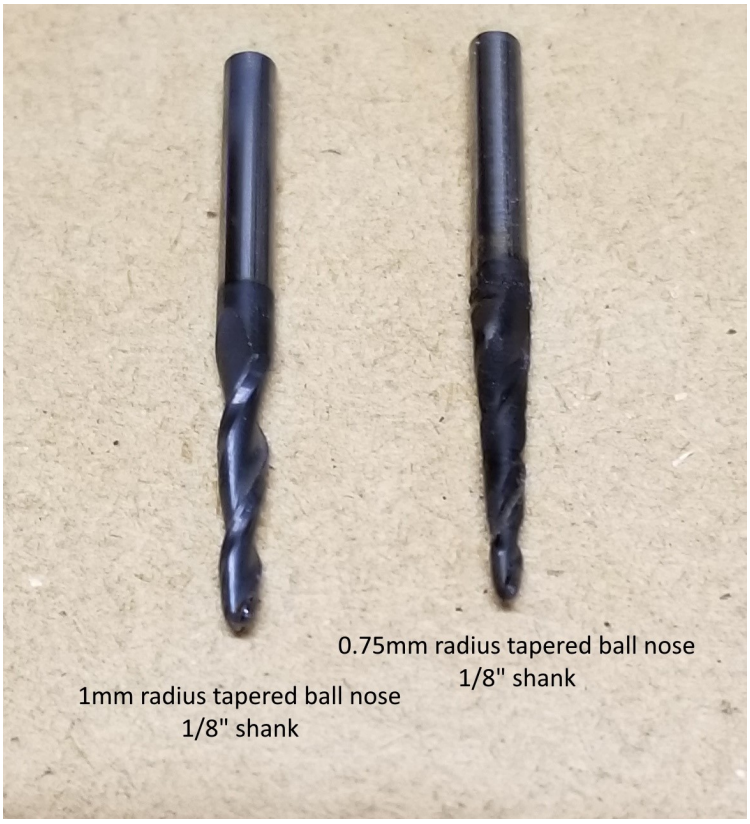
Primary uses:

- Clearing away material, especially detailed areas of 3D carvings.
- Engraving
- Drilling (see below)

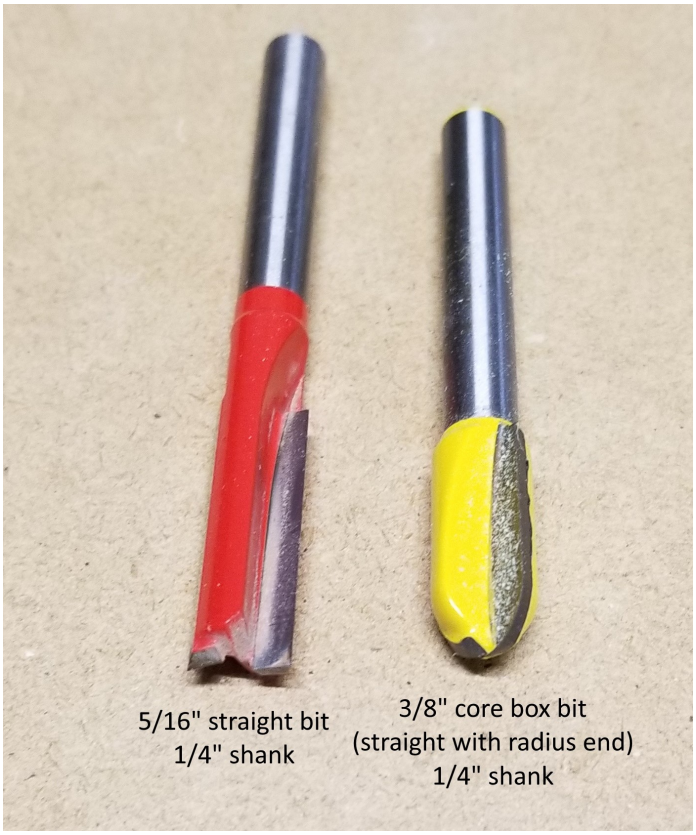
You can find ball nose bits with straight or tapered sides.

The bits shown at right are tapered ball nose end mills (often abbreviated TBN, for Tapered Ball Nose), which I initially purchased for drilling cribbage board peg holes (and they rock in that application!), but I have also used them for various engraving and 3D uses with very good results.

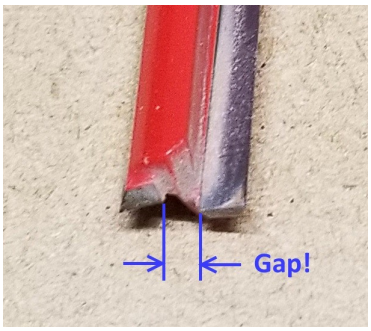
(I don't own a straight ball nose so I don't have a pic, but I'll bet you can visualize them.)



Straight Router Bits. I'm not sure how many folks use these in their CNCs, but I'm going to mention them because I find them quite handy. In my experience, straight router bits work wonderfully for quickly and cleanly clearing out pockets and other large areas. In fact, the red bit is the one I use to surface my spoil board! All of the straight bits that I own have two flutes (cutting edges), but I assume there are other configurations out there. You may want to check the max spindle rate on the bit before using it in the DW660, which spins nominally at 30,000 rpm. (The DW611 and Makita have integrated speed control, so you can adjust them to fit the bit requirements.)



Important Note: When using straight bits like those shown here, be sure to note whether there is a gap in the cutters on the bottom. If there is, then you should *not* plunge straight into the workpiece with that bit because the cutting edges will not be able to clear all of the material as the bit plunges. Instead, you should set up the toolpath to ramp in, and it will work very well.



Surfacing Bits. I don't own a surfacing bit and have never used one, but they come up as a topic on the Bobscnc FB page from time to time so I thought I would at least mention them here.

With a surfacing bit, you can use a CNC to "plane" or "surface" a piece of wood (e.g., the spoil board) to ensure that the surface of the piece is parallel with the CNC's Z plane. In other words, after surfacing, you can be confident that your carve will have an even depth everywhere on the piece.

The bit at right is one example (I swiped the pic off Amazon) … it has a 1/4" shank and 1" cutting diameter.

(Note: The bit has a gap between the cutters on the end, so as noted on the previous page you wouldn't want to plunge it straight in very far, but since surfacing bits are generally used for removing only a small amount of material … typically 0.01" or so … I'm guessing that isn't typically an issue.)

The bit shown here costs $28 (and that seems sort of typical), which is why I don't own one. I have a thickness planer that I can use for work pieces, and as I stated earlier, I use my 5/16" straight bit for surfacing my spoil board. It takes longer, but I only have to surface it once unless something changes on the machine or I decide to replace it.

However, if you don't have a planer, or if you for whatever reason want / need to regularly surface pieces of wood, then a bit like this will save you a bunch of time!

Important notes:

⇒ Please be sure to note the max rated operating speed for these bits, and if you don't have a speed control on your router you may want to consider getting one.

⇒ But here is perhaps the biggest caveat with using these bits: they will make it very obvious if your machine is not perfectly trammed … i.e., the larger the cutting diameter of flat end bit, the more apparent small deviations in tram will be (you will create ridges in your surface).

Conclusion. And now for the answer to the original question: What bits should a new E / Evo / Quantum owner buy? Well, of course, the answer depends on what he/she wants to do with their CNC, but as I stated in my "spoiler alert" on the first page, the best thing to do might be to avoid spending bunches of money on bits while you're still learning to use the machine.

After that, however, it's time to get busy! And in my humble opinion you can't go wrong with the following bits for starters, and then work from there:

⇒ At least one vbit for lettering and general carving … 60° for medium stuff, and/or 90° for larger stuff[2]

⇒ 1/8" flat end mill (straight-edged, upcut, or downcut, as you prefer) for cutting out shapes and cleaning up vcarved pieces[3] … tip: these are often available in sets of 10, which is great because you're likely to break one or two along the way (not that I would know … heard it from a friend).

⇒ If you're going to do some engraving or 3D relief carving, then you'll want at least one ball end mill as well … if you only buy one, I'd recommend one with a 0.75 mm radius (about 0.06" diameter) on the tip.

One important additional note:

- If you're doing work that requires some precision, please do yourself a favor and accurately measure the actual size of your bit, then record those values in the software that is creating the toolpath. Just because a bit is advertised as, say, 1/8" doesn't mean that it will be exactly 0.125".

Well, that's all I have for this edition. I hope it is helpful. Feel free to ping me in the BobsCNC Routers FB group if you have questions or suggestions for improving this document.

Now get in there and make some sawdust!

Greg Grote

Alternate reference: Search YouTube for "Mark Lindsay bits" and you'll find an excellent video on bits.

Footnotes

[1] I purchased 1/8" collets for my Makita routers from Elaire and have been very pleased with them. There are some inexpensive sleeve-type adapters available that some folks use, but I don't like that concept and have never tried them.

[2] For Vbits, I can recommend Whiteside as good bits that don't cost a fortune, but I know a lot of folks have had luck with other brands as well.

[3] My favorite bits for cutting out shapes and milling small pockets can be found by searching Amazon for "HQMaster Straight CNC Router Bit"