Understanding CNC Bits

{or a Guild members treatment for insomnia}

One of the first questions often asked by individuals starting with the CNC is "What bits should I purchase?" Choosing a CNC bit can be confusing. There are many variables the user must consider when picking the perfect bit for their project.

The following are some factors that should be considered when choosing a CNC bit to be used at the Guild.

This article is limited to bits used to machine solid wood and wood products.

Can Standard Router Bits be used on the CNC?

The short answer to the question above is Yes and No. For a bit to safely be used on the CNC the bit should be able to plunge into your material, meaning it must have a cutting edge all the way across the bottom (see photo to the right). Many standard router bits will have a portion of the bottom end of the bit that has no cutting surface (see photo at left). Improper use of one of these bits' places undue stress on the CNC machine and can cause burning of your workpiece.

If using a bit that does not have a cutting surface across the bottom of the bit, it is recommended that you slowly ramp your bit into the toolpath over a distance 4 to 8 times the bit diameter and that you make shallow cuts. Alternatively, you can use a "lead" into your material where the bit starts its rotation outside of the material being machined and then proceeds into the material to cut the toolpath.

Some projects do not lend themselves to using a lead and for this reason it recommended to use only bits that are designed to plunge directly into the material.

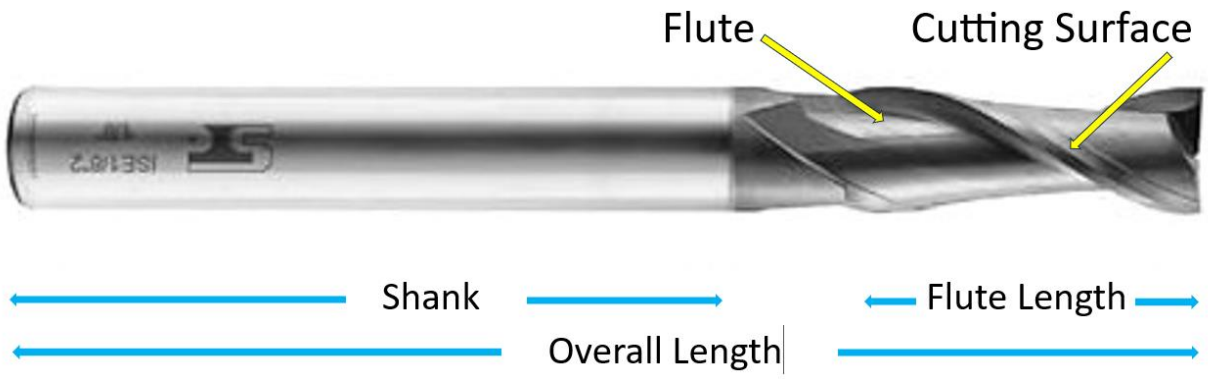
To summarize, it is recommended when purchasing CNC bits that you buy only bits that are designed for use on the CNC. When purchasing bits choose those that state in their description that they are "plunge" or "center cutting" bits. If

you are in doubt about a bit contact the manufacturer or retailer before making your purchase.

Bits with a bearing on the lower end of the shank should not be used on the CNC machine.

Anatomy of a CNC **router** bit.

For the purpose of this discussion the “top” of the bit will refer to the shank end and the “bottom” will refer to cutting end of the bit.



All CNC bits have the same basic structure. The shape and length of the Cutting Surface will determine the capabilities of the bit.

- Shank – the portion of the bit that is inserted into the collet on the spindle of the CNC. You should purchase only bits that have a shank diameter that matches the collets on the CNC. The Guild currently has collets that accept 0.125”, 0.25” and 0.5” shanks.
- Cutting Surface – the knife edge portion of the bit that slices the chips from the material being machined. Caution should be exercised when handling CNC bits as the Cutting Surface is extremely sharp and can cause laceration of the skin if touched.
- Flutes – the groove behind the cutting surface. This groove allows for chip evacuation from the toolpath. In general, the greater the number of flutes the smoother the cut, however, this is balanced against increased heat production from less efficient chip removal with an increase in the number of flutes. It is recommended to use bits with two or three flutes for most projects at the Guild.
  The terms “Cutting Surface” and “Flute” are often used interchangeably since they are always in a 1:1 ratio.

Flutes can have any of four geometries:

- Straight – The flute runs parallel to the axis of the bit.
- Spiral – The flute is coiled around the axis of the bit. Spiral flute bits can be either of the three types listed below:
  - Up-cut spiral – the cutting surface is on the top side of the flute. These bits are excellent for removing chips from the material being machined. Because these bits pull upward on the material being machined, they may produce tear out on the top surface of the project. When machining all the way through the material these bits tend to produce a smooth surface on the bottom side of the material.
  - Down-cut spiral - the cutting surface is on the bottom side of the flute. Because these bits push downward on the material being machined, they tend to produce a smooth cut on the top surface. When machining all the way through the material these bits tend to produce tear out on the bottom side of the material. Caution: When machining a profile toolpath these bits will pack chips into the toolpath which can result in increased heat generation and subsequent shortening of tool life as well as the potential for burning of the material being machined.
    Caution: Down-cut bits should never be used to drill down into a project as the friction from the packed chips against the bit may result in a fire.
  - Compression Bits – These bits combine the best of both worlds of spiral bits with an up-cut flute on the bottom portion of the bit and a down-cut spiral for the remainder of the cutting length. Since these bits have a down-cut spiral for most of their flute length they can pack chips into the toolpath resulting in excessive heat and shortened tool life, however, because of their design they will produce a smooth cut on the top and bottom of your material when making a full depth cut.
    These bits work well for any veneer finished material such as plywood as well as MDF.

- Flute Length – This is the length of the bit designed to slice chips from your workpiece. This is sometimes referred to as Depth of Cut (DOC). The DOC is important because it limits the depth you can cut into your material without the body of the bit coming into contact with the

unmachined edge of the toolpath. If machining deeper than the DOC for the bit you are using, the risk of a poor finish increases and the risk of bit breakage is increased.
In general, it is best to use the shortest flute length possible for your project, excessive length increases the possibility of bit breakage, bit deflection, bit vibration and also increases the chance of a poor cut in your workpiece.

Cutting Surface Materials

Not all router bits contain the same material. A search of available bits suitable for use on the Guild CNC returns a variety of types of material and each has its own advantages and disadvantages. In general, the harder the cutting surface the more prone it is to chipping and breakage.

High Speed Steel (HSS)

A hardened alloy of steel

- Advantages
  - ❖ Usually less expensive than other types of CNC bits
  - ❖ Less prone to chipping of the cutting surface
  - ❖ Many cutting edge geometries available
- Disadvantage
  - ❖ Usually shorter life than other types of CNC bits

Solid Carbide (SC)

Another alloy of steel, but one with increased hardness and resistance to wear. The core and the cutting edge of the bit are made of this alloy

- Advantages
  - ❖ Increased life compared to HSS
  - ❖ Produces smooth finish secondary to increased rigidity
  - ❖ Many cutting edge geometries available
- Disadvantages
  - ❖ Bits can be brittle and chip easily if mishandled
  - ❖ More expensive than HSS

Carbide Tipped (CT)

Solid Carbide bits can be very expensive. The advantages of Solid Carbide bits are maintained by constructing the Carbide Tipped bit with only the

cutting surface made of Carbide which is welded to a core made of high-speed steel.

- Advantages
  - ❖ Increased life compared to HSS
  - ❖ Usually less expensive than SC
- Disadvantage
  - ❖ Cutting edge can chip easily if mishandled
  - ❖ Limited cutting edge geometries available
  - ❖ May not be able to plunge

Solid Tungsten Carbide (STC)

Tungsten and other metals are added to Carbide to produce a bit that has many advantages over other materials.

- Advantages
  - ❖ Harder than Carbide bits therefore remain sharp longer
  - ❖ Exceptionally strong
  - ❖ Less brittle than standard Carbide
- Disadvantage
  - ❖ Can be costly

Coatings

Many manufacturers offer a special coating that can be applied to their bits. Some common coatings are:

- Zirconium Nitride (ZrN)
- Diamond Like Carbon (DLC)
- Tetra Amorphous Carbon (Ta-C)
- Spektra (nACo)

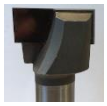
These coatings are advertised as producing extended tool life and a smoother cut. However, it should be noted that coated bits are usually more expensive than comparable uncoated bits. Coated bits are used extensively in industrial applications, but for the hobbyist the increased cost may not be justified.

Bit Geometry

The shape of the cutting surface of a CNC bit will determine the profile of the cut the tool makes as it follows the toolpath. The geometry of the cutting surface can be generalized into six shapes.

- Cylindrical (Flat Bottom)
- Round Nose
- V Bits
- Drill Bits
- Form Bits
- Specialty Tools and Cutters

Cylindrical Bits – The cutting surface is a cylinder usually with a flat bottom. [Photo below left] This type of bit can be used to create a profile toolpath or used to pocket out a portion of your workpiece. This bit can also be used to cut out a part from your workpiece leaving a straight edge. These bits come with a flute that is straight, spiral up-cut, spiral down-cut or compression. Spiral up-cut bits can also be used to drill down into your material creating a hole the diameter of the bit.
These bits come in a wide variety of diameters which allows the user versatility in programing their toolpaths.



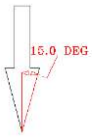
A subcategory of cylindrical bits is the Surfacing bits. [Photo above right] These bits have a wide flat bottom cutting edge and are useful for creating large flat surfaces in your workpiece.

Round Nose Bits –These bits are sometime referred to as Core Box bits. {The difference is that Core Box bits are round nose bits with the cutting surface extended parallel to the axis of the bit.} [Photo at right]. These bits create a round bottom grove. These bits find use in creating flutes in solid material and cutting channels with a concave bottom and can also be used to create a cove. This type of bit comes with many variations including:

- Bowl Bits – have a radiused end with a flat central cutting surface between the sides (see photo at left). As the name implies these bits are good for creating pockets with a curved transition (bowl shape) between the bottom and the sides. They will also create a flat bottom groove with curved sides.

- Tapered Ball Nose bits – have a hemispherical cutting end with a diameter that is less than the shank diameter. The transition (taper) between the shank and the cutting end has a fluted cutting surface. These bits are used extensively in 3-D carving due to their small diameter cutting surface and tapered sides.

V Bits – As the name implies the cutting surface of these bits are V shaped and they cut a beveled groove or chamfered edge. These bits are commonly of a two or three straight flute design, although at least one manufacturer produces a V bit with down-cut flutes. V shaped bits are sold as either V-Bits or as Engraving Bits. Care should be taken when purchasing V shaped bits because the bits may be described using two different conventions.
V-Bits will be described by the "included angle" (sometimes called "total angle"), which measures the angle between the two cutting surfaces, as in the diagram to the right.



Engraving Bits are described by the "side angle" which is the angle of the cutting surface to the centerline of the bit, as in the diagram to the left. Engraving bits will also specify a tip diameter which is a small flat area at the tip of the bit. For an engraving bit the included angle is twice the side angle.
Common V-bit included angles are 30, 60, 90 and 120 degrees.

Drill Bits – As the name implies these bits are used to drill holes into your workpiece. These bits should never be used to machine any other toolpath. If purchasing a drill bit for use on the CNC the shank must be the same diameter as the collet you are using.

Form Bits – These bits have compound geometries which allow them to cut shapes such as round overs, ogees, beads, coves, flutes and other shapes. These bits allow the user to add decorative profiles to their projects.

Specialty Tools and Cutters – This group would include Dovetail bits, T Slot cutters, side v cutters, diamond drag bit and the Donek drag knife. All of these require special consideration when programing your toolpaths, and in some cases require that the spindle be set to zero RPM.