

## **How To Choose The Milling Bit That Is Right For You**

End mills, sometimes called milling bits, are the most commonly-used bits used with CNC machines. They are utilized to cut horizontally (or laterally) across a surface. (Drill bits that are cut through an object.) End mills come in a variety of styles and are specifically designed to help you create the look you want for the particular project you are working on.

If you're new to CNC machines and carving, the information below provides the different characteristics of different bits. Knowing what each bit does and how they operate can help you choose the best one for your project.

Keep in mind that the kind of material you're using with your tool is the primary aspect in determining the type of piece of equipment you'll need. Different materials have different qualities that impact your carving and your bit's behavior. You can find out more about which bits to make use of with your particular material by [clicking here](#).

### **Shank Diameter as compared to. Cutting Diameter**

The shank's diameter is the diameter of the non-cutting part of the bit. This end of the bit is then inserted into your router. Most routers are made to support a certain diameter bit. For example, the DeWalt 611 router is able to hold bits that are 1/4" in shank diameter. If you want to use bits that have smaller shanks, you'll need to buy the collet and nut adapter to your router. The 1/8" collet for the DeWalt 611 is an everyday upgrade many of our customers purchase as they get into more intricate and detailed work.

The cutting diameter is the overall cut diameter of the bit. Cutting diameters can range anywhere from just a few inches (for leveling surfaces) to 1/64" or 1/32" (for very fine details). When choosing the appropriate bit for your project, aim to select one with the biggest possible cutting diameter for the task at hand. If the project you're working on contains intricate components that are not cut with your chosen bit, Easel will recommend that you pick a smaller piece.

Shank diameters and cutting diameters aren't always the same. It is essential to ensure that your tool fits in your spindle/router (shank diameter), and then choose the appropriate bit for the job.

### **Cut-Up, Cut-Down and Compression**

The cutting method of a milling tool determines how the material chips are removed from the project and also which side gets smoothed.

The pieces that are up-cut push the material upwards and smoothen the bottom of the cut passes. They draw heat away from the piece and is an excellent choice for metals, plastics and other materials at risk of burning or melting. But due to the bit's design and upward-pulling motion, it can interfere with the clamping system you use with your material, especially if your material is thin.

Down-cut pieces push the chips downwards (towards the surface). This method of cutting can heat or ignite your material, particularly if you have to make many passes. The benefit to this style of tool is that, by pressing down against the surface, it will aid in keeping thin materials firmly to the cutting surface. Also, you will notice a smoother edge if you choose to use an upward-cutting bit.

Compression bits are neutral in their cutting action. They smooth the top- and bottom edges of the cut. This is accomplished by using the up-cut style for the lower half of the bit and down-cut for the upper part. These bits are best suited to thicker materials that need to be fully cut. It might not be the best option if your project requires very shallow cuts. The bottom (up-cut), and not its top (downcut) sections are utilized.

## **Flutes**

The flutes are the cutting edges of the bit. The name of a fluted bit is based on the amount of flutes present on the bottom of the bit (for instance, our 2-flute bits are all equipped with two cutting edges).

In general, having more flutes on an end mill will give you a more smooth edge finish. An increased surface area is a great way to get rid of chip pieces from stock materials. Additionally, there are more flutes in an [end mill](#). This means that you'll need to choose a slower feed rate, to allow the bit the time to eliminate the cut materials. You can remove material faster, but your cuts will be less smooth when you have fewer flutes.

Certain materials need to be removed faster than others. If the chips aren't taken off the piece fast enough, HDPE will melt. For a material such as this, you'll need one or two flutes for your endmill. For tougher materials that are less prone to melt, you could make use of more flutes. Just be mindful of the depth of your cut, to ensure you don't break the bit when you try to cut too deeply into the material.

## **V-Bits**

V-bits, also known as v-carve bits, create a pass shaped like an "v" and are commonly employed to create intricate engraving. V-bits can be used in Easel Pro for creating intricate V-carved designs. Easel Pro users are able to make projects for free as long as they have an Easel account. For carving, you'll need to sign to a Pro account.

## **Fishtail Bits, Ballnose Bits, and special Bits**

The tip of a milling tool helps determine its purpose. Fishtail bits produce an even surface that is located at the bottom of the cut. Ballnose bits make the appearance of a rounded bottom cut. Since the tip is rounded, these bits can be used to create contours or step layers.

We also offer a selection of bits designed for specific applications. We also offer bits that are designed for fine-details, engraving and working using PCB.