With fan-cooled systems, the duty cycle of the spindle is approximately 60% to 70%. Another is the noise generated by the arbor fans which tends to be in the upper 70's to lower 80's decibel range; however, electric fan-cooled spindles do not have noise related issues. These spindle offerings usually do not exceed 24,000 rpm and are usually the most economical.

Compressed-air-cooled spindles allow for a 90% duty cycle. These require a constant stream of clean nonfluctuating source of compressed air. These spindles are typically used for greater then 24,000 rpm. These spindles usually incorporate ceramic bearings.

This method is the most efficient and allows for duty cycles of 100%. These spindles are ideal for very demanding applications such as production of very hard materials in a 24/7/365 day operation. The constant loads on the spindle generate a lot of heat, and the only method to remove this excess heat is through a separate liquid chiller unit.

Choosing a Spindle

There are many spindle options available for Techno CNC Routers. Correct selection of a spindle is imperative for optimum machine performance. The electric spindle is the heart of the machine. Many variables must be considered when selecting the correct spindle such as material to be cut, production volume, tooling, machine feed rate, and spindle rpm. Generally, each material and cut has an ideal tool profile and cutting speed. Larger diameter tools require slower speeds. Smaller diameter tools require higher speeds. Spindle speed and feed rate for a given cut must be balanced for best quality, tool life and spindle life. Incorrect spindle speed is a common error in CNC machining. Machine feed rate and spindle rpm are directly related to one another. The higher the rpm, the faster the machine must be cutting. The typical question is how fast should I be cutting? This can be determined by the chip load. Simple formulas can be used to predetermine feed rate.

Formula:

(chipload) x (# of cutting edges) x (rpm) = feed rate

Many cutter manufacturers supply this information with the specific cutter to be used for different types of material to be cut. This will get you close and the optimal feed rate can be fine-tuned at the machine. Typically, feed rates that are too slow will decrease tool life due to the increased friction. This increased friction will not only wear out the cutter, but will also heat up or burn the material being routed.

Other considerations that must be taken is how to enter the part. Ramping into the part is the preferred method. The ideal ramp should be between 0 and 20 degrees from the table surface. This angle will allow you to enter into the material at 100% of the feed rate. At any angle greater then 20 degrees, the feed rate should be reduced accordingly. Entering into the part on a ramp will greatly increase spindle bearing and tool life.

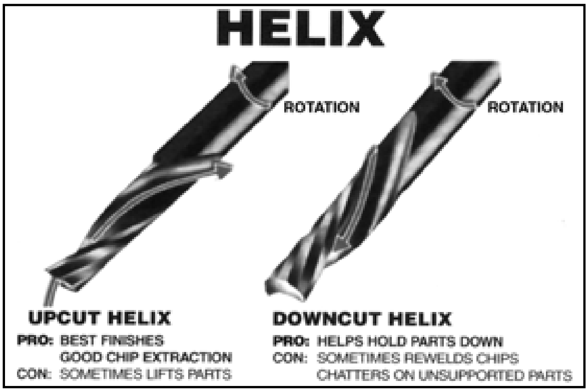
CNC Tooling

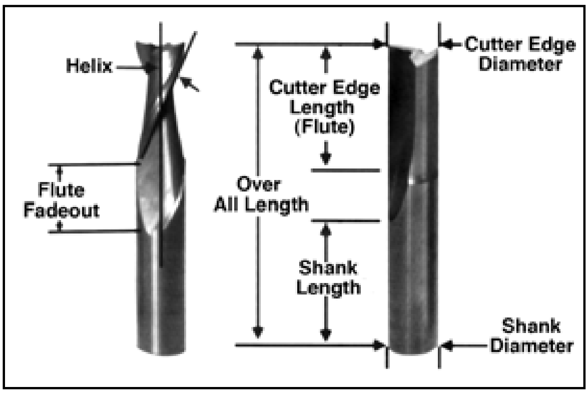
Cutting Tool Geometry

The tool geometry influences cut quality, cutting speed, chip removal, and the life expectancy of the tool in a specific material. Choosing the right tool for the specific material being routed can be critical. There are several different geometries to choose from such as upcut / downcut helix, straight flute, and ball nose, to name a few. The cutter selection of this catalog defines in more detail the ideal geometry for specific materials. Illustrations on the next page shows basic terms used to define a tool's geometry.

Router Bit Materials

There are four basic materials to choose from when selecting a router bit for a specific application: High-speed steel, carbide tipped, solid carbide and PCD diamond. The choice of a specific tool depends on the material being routed. High-speed steel and carbide tipped tools tend to be used more with hand held routers vs. CNC Routers. The steel body of the tools tends to withstand the vibration caused by the inconsistencies of hand feeding the router. Another reason they are used with hand routers is that they are available in many geometries for low cost, and available at most home centers. The ideal choice for a CNC Router is solid carbide or PCD Diamond. Solid carbide is the most common router bit used on CNC's. It provides long life due to the tougher body construction (carbide), delivers the best edge finish, largest variety of geometry profiles available, can achieve greater feed rates, is the best tool for plunging into the workpiece and has relatively low cost. PCD Diamond tools are ideal for very abrasive materials. These type of tools, however, are not cheap, perform very poorly on nonabrasive materials, cannot plunge, and are limited to how fast they can route.





Heat

Heat generation in cutting tools should be avoided. Heat can change the properties of the tool and melt or burn the material being routed. This can lead to very poor surface finish, causing the tool to fail/break, or material melting and wrapping onto the tool. There are many causes of heat buildup such as improper tool selection, incorrect machine feeds, incorrect spindle rpm and/or rotation, and improper toolpath programming. In order to minimize or eliminate the heat buildup, you must avoid the above causes. To determine if you have programmed the feeds and speeds correctly, you can follow these simple formulas:

Formula 1: Chipload = chipload % x cutter diameter
Formula 2: Feed Rate = [chipload] x [# of flutes] x [rpm]

A good starting point for determining chipload (thickness of the chip) is 2% of the cutter diameter (example 1/4" cutter = .005). For softer materials, it is recommended to take 4% of the cutter diameter.

2% **Hard material:**
Brass, hardwood, cast acrylic, solid surface

4% **Soft material:**
Aluminum, MDF, particle board, extruded acrylic, sintra, polycarbonate

Chipload = [2% or 4%] x [cutter dia]
Feed Rate = [chipload] x [# of flutes] x [rpm]

Example 1:
1/4" 2-flute endmill (Hard Material)
Chipload = .02 x .25 = .005
Feed Rate = .005 x 2 x 18000 = 180
Feed Rate = .005 x 2 x 12000 = 120
Feed Rate = .005 x 2 x 6000 = 60

Example 2:
1/4" 4-flute endmill (Hard Material)
Chipload = .02 x .25 = .005
Feed Rate = .005 x 4 x 18000 = 360
Feed Rate = .005 x 4 x 12000 = 240
Feed Rate = .005 x 4 x 6000 = 120

Note that for each cutter, there is a feed rate range that corresponds to the rpm range. The number of flutes determines the range at which a cutter should be fed. The number of flutes should be taken into consideration when cutting material. 3- and 4-flute cutters are best for high feed rates, while 1- and 2-flute cutters are best at lower feed rates. Also, keep in mind that spindles have peak power at higher rpm. It is better to use a 2-flute cutter at 18000 rpm than a 4-flute at 9000 rpm.

When dealing with material that melts, a thicker chip will reduce melting. When cutting deep into these materials, it becomes important to remove the chips from the cut. Cutters with fewer flutes are better capable of removing these chips. Also, larger cutters are better for dealing with melting.

Collet and Spindle Definitions

Pull Stud

Note: This is a maintenance item and should be replaced every 6 months.

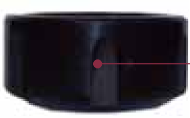
ISO Tool Holder

HSK Tool Holder

Note: Always make sure the tool holder and collet nut are rated for the appropriate spindle rpm. Improper selection will cause spindle bearing failure.

Collet

Note: Only a completed collet and collet nut assembly should be screwed onto a tool holder.



Collet Nut

Router Bit

Note: Always make sure the router bit is properly balanced for the appropriate spindle rpm. Improper selection will cause spindle failure.

Collets

There are two basic types of collets, half-grip and full-grip (see top right). Half-grip collets are identified by slits running from the bottom (or "mouth") of the collet toward the top for about 80% of the collet length. This type of collet grips the router bit at the mouth of the collet. Half-grip collet is the simpler of the two collet types, and is ideally used with shorter shank tools where the shank would not fill the entire length of the collet. Full-grip collets have slits running up from the bottom and down from the top, again for about 80% of the collet length. This type of collet grips the tool evenly over the entire length of the collet and tends to have more

Full-Grip Collet

Half-Grip Collet

flexibility, which results in "ranged" or "universal" collet sizes. This means that a specific collet size can hold a range of tool sizes (typically inch and close metric sizes).

Collets are made from spring steel, and regular usage causes loss of elasticity and the need to replace them more often. It is recommended to use collets designed for specific size tools, as requiring the collet to expand for too-large tools or compress for too-small tools will shorten the life expectancy of the collet.

Proper Tool Colleting

Installation of the tool in the collet and collet nut is equally important. To prevent unnecessary strain on the collet and to ensure a proper fit, only use a collet designed to fit your tool shank diameter. In addition, the tool's flute should not extend into the collet; doing so can score the inside of the surface, as well as force debris into the collet, putting the entire assembly off-balance and potentially damaging the spindle. The graphic below illustrates the correct and incorrect installation. Please note that these illustrations are designed to show proper insertion of the tool into the collet, and do not show the collet nut.