ROUTER BIT GEOMETRY

Terms

Helix Angle- Angle of the cutting flute, it is measured relative to the axis of the cutting toot.

Flute Fadeout- The length between the end of the cutting length and the begin of the shank length

CEL- Cutting edge length.

Shank Length- The length of the cutter shank that can be inserted into the collet.

OAL- Overall cutter length.

CED- Cutting edge diameter.

Shank Diameter- The diameter of the shank to be inserted into the collet.

Single Flute

Use for faster feed rates in softer materials. The single-flute cutter typically has lots of room for chips, but the feed rate or the hardness of the material to be cut is limited by the single cutting edge. A single-flute cutter is especially recommended for plastics.

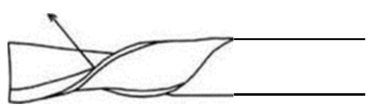
Double & Triple Flute

Use for better finish in harder materials. Double-flute bits provide a smoother cutting action because the chip load is smaller than a single-flute cutter for a given feed. With a Double-flute bit, harder materials can be handled.

Double & Triple Flute

Upcut Spiral

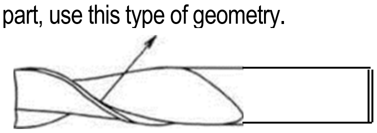
Use for grooving or slotting, for upward chip evacuation and best finish on bottom side of piece part. These bits allow for rapid cuts since the tool clears the chips away from the material. Because of the ragged finish that can result on the top surface, this type of tool is not recommended for softer materials such as MDF. Whenever the best finish is needed on the bottom side of a part, use this type of geometry.



Upcut Spiral

Downcut Spiral

Use for downward chip flow, better hold-down in fixture and best finish on the topside of cut part. Note: that the cutting speeds usually have to be reduced because the chips are pushed back into the material.



Downcut Spiral

Up/Down Spiral (Compression Spiral)

Use for double-laminated material and best finish on tap and bottom side of piece part. Because of the spirals, all the chips are forced back into the material. This will result in a very clean cut on the top and bottom edges, but the cutting speeds have to be reduced. Note that the center of the spirals should be approximately in the center of the material for best results. This usually means that a substantial scavenger or waste board would have to be used.

Compression Spiral

TOOLING MAINTENANCE

Tool Life

Tools should be changed at the first sign of edge deterioration causing finish degradation or increase in operator effort to maintain feed rates. Never allow the tools to dwell in a cut.

Feed the router bit in such a manner so that in moving through the work it has a chance to bite or cut its way freely. If the feedrate is too fast, strain and deflection will occur. If it is too slow, friction and burning will occur. Both too fast and too slow will decrease its life and cause breakage.

The router mechanism must be well maintained for any cutting tool to perform properly. Check the collet for wear regularly. Inspect tools for collet marks indicating slipping due to wear or dust build up. Check spindle on a dial indicator for run-out. Collet and run-out problems cause premature toot failure and associated production difficulties. Do not use adaptor bushings to reduce size of the collet on a routing or production basis. Tools will not perform properly in bushings over an extended period of time. Bushings are for prototype, experimentation, test and evaluation and not for production.

Wherever possible, use a coolant when routing. Heat caused by action between the tool and piece part is enemy #1 to toot life.

Heat is a function of surface footage per unit of time, thus, the more dense the material, the faster the feed rate to minimize heat. However a compromise must be reached between finish and heat.

Tool geometry dramatically affects the tool life. Rake and clearance angles, as well as cutting edge length should be examined.

Router bit breakage is most often caused by a misapplication of the router bit. Make sure to use the proper router bit.

Tool Breakage

In spite of the structural and metallurgical attributes, which are designed into industrial and professional router bits, breakage occurs. A detailed examination yields the following:

Application related breakage:

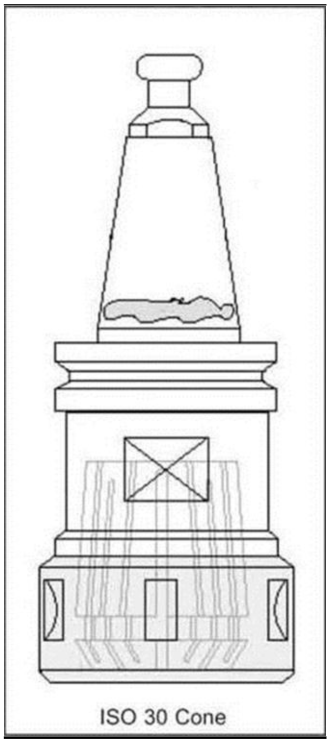
Cutting edge lengths should be as short as possible to accommodate length of cut required. Larger cuffing edge diameters require larger shank diameters. Spiral geometry can direct chip flow and expel chips to reduce heat. Changing the type of tool is the only solution when tool application becomes a problem.

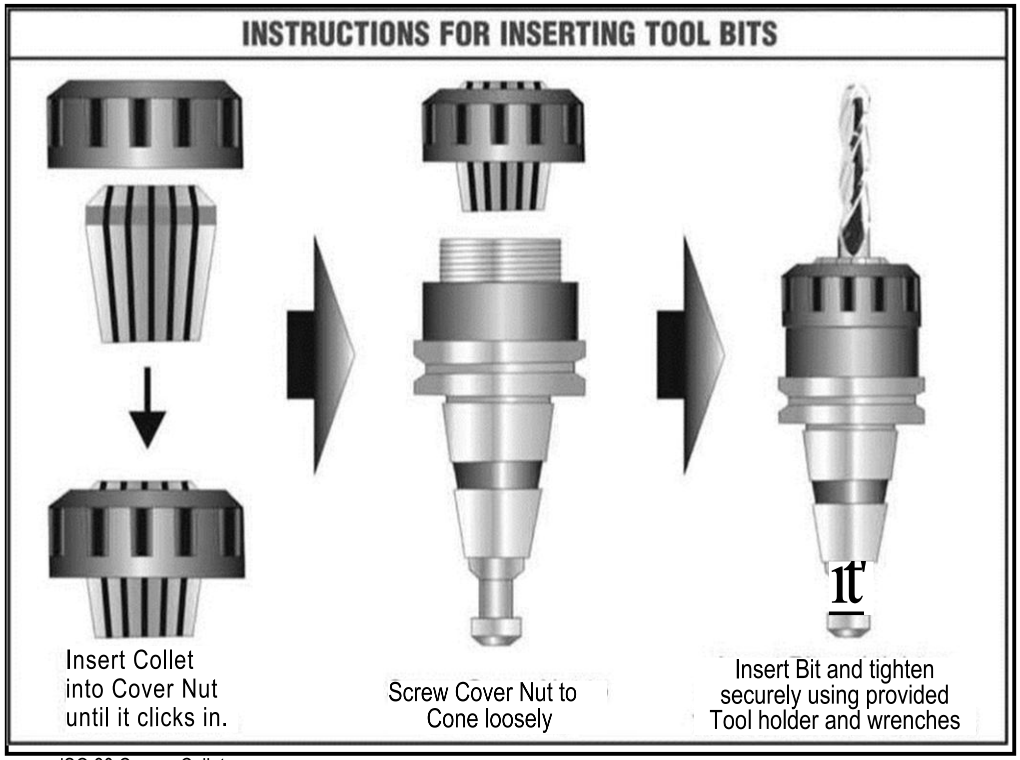
Tool quality shortcomings:

Upon investigation an internal flaw in the steel or carbide can cause failures. These failures are normally random, however, if the flaw was raw material batch based, an entire batch may be a problem. The same is true for heat-treating. Too high a hardness can lead to premature edge failure. This is generally confined to one batch of tools. Geometry induced fractures usually are related to improper rake and clearance angles as manifested in the bit riding the cut. The resulting heat generated by friction shortens edge life, tends to create burn marks and may cause the chips to accumulate behind each cutting edge and culminates in fracture. A shank out of round prohibits effective colleting and does not permit the tool to turn in a concentric manner. The whipping action generated is catastrophic to bit life and failure by fracture is imminent.

Router Collet Integrity

A router bit is rendered nearly useless if the mechanics of gripping and rotating the tool are not made to the same accuracy as the tool. Overt signals such as breakage and dark markings on the shank of the bit should be immediately investigated. Inspect the collet for out of round or bell mouthed conditions. Operators often overlook inspecting new collets based on the assumption that a new collet is geometrically correct. Tool manufacturers are aware and openly share the subtle nature of this problem. Dirt, dust, bonding agents and sap can also affect collet performance, which occupy space and accelerate wear.





iSO 30 Cone + Collet

Operator Breakage

If the router bit is within specification, tool breakage can still occur through incorrect routing techniques. Specifically feeding the router bit into the material must be accomplished in such a manner, that the router bit is permitted to "bite" or "cut" its way freely. If the router bit is fed too fast, excessive strain is put upon the tool, conversely if the router bit is fed too slow, excessive friction will be generated, causing destructive heat buildup. In both situations, tool life is significantly shortened or in sustained conditions, tool fracture is a definite. Any router bit can be broken if poor choices are made in the operating of the router.

Suggested Procedure

Should all of the above examinations be inconclusive, it is important to retain both new and expended sample tools (all pieces if broken). Please contact CAN.CAM and inform them of your issue. The following information should also be provided: router type, material being cut, spindle speed, feed rate and cutting conditions (mist oil) when breakage occurred. This data should provide enough clues for a solution to the tool breakage issue.

Collet Maintenance

Collet maintenance is one of the most common causes of inadequate tool life or breakage. There are up to six links in the chain that make up this critical tool holding system called a collet. As a chain is only as strong as the weakest link, a router bit can only be as good as the system that holds it properly. The increased productivity and reduction in overall costs is well worth your taking the time to regularly inspect and clean the collet system.

The six critical components are as follows:

1. Internal Collet Clamping Surfaces

The most important link in the tool holder chain is the inside of the collet. Resin migrates up through the slits in the collet and then deposits itself on the inside of the collet. This resin build up, if not removed, causes the collet to grip inconsistently on the tool shank. By not applying equal pressure throughout the entire gripping range of the collet, the tool holder allows the tool to resonate inside, causing slippage inside the collet.

Slippage can cause "fretting", a condition in which resins are deposited on the shank of the tool. This resin buildup can be easily removed from the inside of the collet with Rust Free and brass tube-type brushes. These brass brushes are non-destructive and in conjunction with Rust Free can adequately remove the deposits. Rust Free should be sprayed on and quickly brushed and wiped completely dry. Do not allow the liquid to sit and air dry.

2. Internal Spindle & Collet Taper

The inside taper of the spindle and tool holder is a critical surface which accumulates resin build up. To maintain best concentricity, they should be cleaned at each tool change. Felt brushes are available to fit most taper sizes and provide a quick means of removing short-term buildup.

3. External Collet & Tool Holder Taper

The outside taper of the spindle and tool holder require regular inspection and should be cleaned of all deposits each time the tool is changed. If the tapers are regularly maintained, felt cloths can be used. If not maintained, and the buildup is excessive, brass brushes will work well for cleaning.

4. Clamping Nut Surfaces

The inside taper of the nut should be clean and free of burrs on the surface. Any surface burrs or contamination will not only skew a collet but can also permanently ruin a new collet. Clean the clamping nut during every tool change with a brass brush. Special care should be taken to examine the clamping nut threads on a regular basis.

5. Thrust Bearings

Some collet nuts have an integrated thrust bearing connected to the inside taper. This bearing serves to reduce friction wear between the collet and nut as the nut is tightened. The bearing seating surface is the most critical feature and must be kept clean. The bearings should also be kept in smooth operating condition. Contamination or abuse will cause rough movement with the bearing. Either instance is indicative of runout and poor operation.

6. Tool Holders

Tool holders such as the ISO 30 have additional matching and mating tolerances beyond those of the older tapers. Because of their unique design, these tool-holding systems can be more prone to runout caused by resin buildup. “Fretting” or “'Bronzing” will cause inconsistent gripping in the taper and/ or the flat mating surface and reduce consistency of tool life. If ignored, these conditions can eventually produce premature spindle failure. The mating surfaces should be cleaned with Rust Free and hand dried immediately.

Manual Collet + Cover Nut

Note: To ensure trouble free operation, always insert the collet into the collet nut until it clicks in and thread it loosely onto the spindle prior to inserting router bit.

All six of these components are critical and should be regularly maintained. It is important to remember that collets should be replaced on a regular basis, approximately every 400 to 600 run time hours. This means inspection for metallic damage such as bell mouthing or burrs with every tool change. If metallic damage is visible; the collet should be discarded and replaced. Also consider that even if there is no damage present the collet can be worn out through metal fatigue. Heat is directly transferred from the tool to the collet. These heating/cooling cycles remove the original tempering of the steel. Collets are made from spring steel allowing them to have a certain amount of elasticity to grip the tool. As the heat cycle is repeated this elasticity diminishes. Over time, a collet requires increased tightening to maintain the tool in proper position. As over tightening increases, the collet is distorted, creating eccentricities in the tool holder. Therefore, instead of over tightening older collets and creating a number of other problems, the collet should be replaced. Often the cost of a new collet can be offset by the cost of needlessly broken tools in one shift alone.

Proper positioning of the tool in the collet is critical. The tool should only be gripped on the shank portion of the tool. At no time should any portion of the flute fade out be inside the collet.

Proper Collet Use & Maintenance

Many users select tools without regard to the importance of adequately holding them in the collet. Think of the spindle/collet system as a chain and just like a chain is only as strong as its weakest link, so too is the collets relation to the tool. A high performance tool can only perform if the collet is properly maintained each and every time the tool is changed.

FULL GRIP COLLETS

Full Grip Collets are identified by their slits that run from both ends, almost cutting the collet in pieces. This type of collet tends to have more flexibility and often comes in what is termed as **"Range Collets"**, which allow gripping in a range of shank sizes. This full grip type allows gripping over the entire length of the collet and to be properly used, the collet should be 75-80% full. The most important portion of the collet is the mouth, which is at the bottom. This area is important because all the lateral pressure taken by the tool must be evenly distributed on all ears of the collet for it to cut true or concentric. It is very critical that the 80% rule be followed when using a full grip collet due to the ability of the collet to flare at the back if not full. The collet can actually allow tool movement in even minute amounts often times resulting in tool breakage. There are times that the 80% rule is not possible due to the shank length available, so it is necessary to fill this void in the back of the collet with a life plug that is of the same size as the shank, thus to avoid the collapsing problem. Equally as important as filling the collet properly, it should also be understood that it is possible to over-collet as well. This is when the "Flute fadeout' portion of the tool is allowed to extend up inside the collet. This does not allow a firm equal grip by all ears of the collet at the mouth. This allows the tool to have uneven support at the most critical area often times with solid carbide, or high speed steel tools, the tool material is hard enough to actually scar the inside of the collet, causing permanent damage to the collet.

This is also a common reason for tool breakage when it occurs. Breakage often results in permanent damage to the collet due to intense pressure exerted often either "Burring" or “mushrooming” the mouth of the collet.

Heat is the biggest enemy of the tool, and the first place the heat goes from the tool is into the collet. It is also important to note that collets are made of spring steel that can, and will over a period of time lose its elasticity and harden, making it increasingly tougher to tighten adequately. As this hardening takes place, the steel does not fatigue evenly and often causes the collet to grip tighter on one side than the other, creating runout in the tool. It is important to understand that if they are overrun enough this over tightening will eventually damage the internal spindle taper resulting in costly repairs. Because it takes place over a period of time, it is very hard to notice. A safe recommendation for collet life is in the 400 -600 run time hours. This is about 3 months in a two-shift operation of normal run times. If collets are not changed, they will eventually become brittle enough to crack or break in half potentially causing permanent spindle damage that could have been avoided. Just like changing the oil in your car, it is good preventative maintenance that should be done regularly.

Cleaning the collets each and every time the tools are changed is just as important as replacement. Collets are in a brutally dirty environment and are expected to perform a very accurate task while undergoing some real extremes of heat and dirt. As material is routed, whether it be wood, plastic, aluminum or man-made board, the chips carry with them many resins that migrate up the slits in the collet and deposit themselves onto the inside of the collet ears, usually nearest the mouth of the collet. This miniscule migration is often the cause for tool breakage when seen in the actual shank area of the tool instead of down by the cutting edge. The resin acts like pressure points gripping the tool tighter at the mouth of the collet. These pressure points often distort the grip on the tool creating runout, this resin heats up as the tool does and actually ends up depositing itself onto the shank of the tool almost gluing the tool into the collet leaving brown marks at the mouth of the collet contact on the shank. These brown marks are sure sign of collet neglect. To prevent this problem the resin must be removed from all surfaces that it is prone to buildup using a non-abrasive brass tube brush for the inside of the collet and a mild solvent and rag for the external surfaces of the collet and inside spindle taper. It is important to point out that blowing out the collets does not get rid of the resin, nor does soaking them overnight in thinner. A brass brush is the best thing, along with some of the citrus-based cleaners available, allowing them to be safely used on the shop floor. **Do Not** use a petroleum based lubricant for cleaning, as it will only act as a magnet for all the dirt and dust by the residue it leaves behind.

Calculating Feeds and Speeds

There are certain parameters that must be considered, before setting up any file for cutting if you are to accomplish the finish and accuracy required. One of the most important of these factors is the Chipload (Cpt). Chipload can be defined as the size or thickness of the chip that is removed with each flute per revolution.

When material is machined the cutter must revolve at a specific RPM and feed at a specific feedrate to achieve the proper Chipload. There are also several factors to be considered when choosing the proper RPM and feedrate:

a. The power and rigidity of the machine
b. Depth and width of cut
c. Sharpness of the cutting tool
d. Design and type of cutter
e. Material being cut
f. Finish and accuracy required

The feed rate used depends upon a variety of factors, some of which are listed below:

a. Rigidity of part hold-down
b. Power and rigidity of the machine
c. Depth and width of cut
d. Sharpness of cutting tool
e. Length of cutter
f. Design and type of cutter
g. The material being cut
h. Finish and accuracy required
i. Spindle speed
j. Manufacturer of cutter
k. Number of cutting flutes
l. Diameter of the bit
m. Material the bit is made of
n. Spindle used
o. Coolant used

What we must do is sort through this list of variables and given the equipment and material we have to work with optimize each of these variables when choosing the optimum feed rates and RPM to attain the optimum Chipload. One thing to remember is to make chips and not dust. Chips will help by removing the heat produced in the cutting process thus increasing tool life and edge quality.

Feed is calculated using the following equation:

Feed = N x x RPM

N- the number of cutting edges
cpt - chip load *(chip per tooth)* is the amount of material, which should be removed by each tooth of the cutter as it revolved and advances into the work. *(Feet inch per tooth)*
RPM - the speed at which the cutter revolves in the spindle. *(Revolutions per minute)*

We will now break down the relationship between the Feed rates, number of cutting edges, chip load and RPM. For most materials there is a recommended chip load.

If you are running at 18000 RPM using a 1" endmill with two flutes, and a recommended chip load of 0.004 ft/tooth:

Feed = 2 x 0.004 x 18000 = 144 inches per min

If the RPM were increased to 24000 RPM the new feed rate would work out to be:

Feed = 2 *x* 0.004 x 24000 = 192 inches per min

Based on this mathematical equation as RPM increases, feed rate will also increase if all other settings remain the same. If the number of cutting edges changes, however the feed rate will either increase or decrease depending on the whether the number goes up or down. The same applies to chip load. If the recommended chip load is 0.004 ft. /tooth the RPM, feed or number of cutting edges may go up or down to maintain the required chip load. Therefore if chip load remains the same, and feed rate increases, either the RPM and/or number of cutting edges must increase to maintain the recommended chip load.

When calculating the feed rate for any material the chip load is therefore one of the most important factors to be taken into account, because the chip load determines the amount of material that each tooth will remove, plus the load that each tooth will have to take.

		Feed Rate (inches per minute)						
N	cpt	18000	19000	20000	21000	22000	23000	24000
1	0.004	72	76	80	84	88	92	96
2	0.004	144	152	160	168	176	184	192
3	0.004	216	228	240	252	264	276	288
1	0.016	288	304	320	336	352	368	384
2	0.016	576	608	640	672	704	736	768
3	0.016	864	912	960	1008	1056	1104	1152

Another factor that affects chip load is the diameter of the cutter. A larger cutter will be able to handle a larger chip load.

Therefore depending on the diameter of the tool, if the RPM and number of cutter edges stay the same, chip load will increase with a larger diameter cutter, thus the feed rate will also increase. When machining softer materials or using a stubby endmill the chip load can be increased. If an extra-long end mill is being used, the chip load should be decreased.

For most material that will be cut on a router table you will typically use the RPM between 18000 to 24000, and adjust your feed rate to obtain the required results.

The speeds and feeds chosen can be affected by the horsepower of the spindle being used (horsepower varies from 3Hp to 10 Hp). At higher horsepower you will produce more torque thus allowing the machine to run at a variety of RPM s (torque drops off as the RPM is reduced). For most application we typically work in the 18000 to 22000 ranges.

Even though there are formulas for calculating feed rates you will find that optimum feed rate will be determined from experience. You will typically start off with the calculated feed rate, under ideal conditions it is usually suggested that the calculated be set to approximately one-half the calculated amount and gradually increase to the capacity of the machine and the finish that you desire.

Once you have determined what feed and speed to be start with. there are other factors to be taken into consideration. The first thing to be considered is the direction of cut, which is the direction the cutter is fed into the material. Conventional milling or cutting forward is the most commonly used method.

With this method the work is fed against the rotation direction of the cutter. The other method is climb milling or cutting reverse for this method milling; the works and the machine must be rigid. The CAN.CAM router machine is such a machine. When machining non-ferrous materials, climb cutting should be used to get a good finish. Another factor is depth of cut. Depth of cut will effect edge finish as well as tool life, so depending on the type of material and size of cutter you will have to adjust your depth to achieve the desired results. Usually a depth of cut that equals the radius of the cutter is a good starting point when cutting non-ferrous metals. There are other factors that can affect your results. These factors are defined below as they relate to the cutter.

Type of End Mill

There are a staggering number of bit manufacturers and designs on the market. Out there somewhere is likely the best bit but the probability of finding it is just about nil. In addition, the best bit for one type, despite all these complications, it is imperative that the proper bits are found and used. We have found that many bits will often not work at all and others can make a job simple.

Bit Material Composition

Bits can be made of various grades of steel, various grades of carbide, various types of ceramics, and gemstones. For most practical purposes carbide bits are what most Router tables use. Steel bits wear out too fast and the ceramic and gem stone bits cost too much.

Even within the carbide category of bits the material will vary in strength and hardness. The stronger the material is, the less likely it is to break. The harder the material the longer the bit will wear. Unfortunately in carbide bits these two features cannot be found in a single bit….strong bits will not break easily but will get dull quickly, hard ones stay sharp but tend to break quickly.

Number of Flutes

The flutes are the bits cutting edges. It is possible to purchase bits with between 1 and 4 flutes. The number of flutes that you choose depends on the application and cuter design. It will also affect how well the chips are evacuated from the cut. The more flutes used, will reduce the space between flutes thus reducing the tools ability to expel the chips.

Cutting Length

This is the cutting length of the end mill. Generally a shorter cutting length is better as they are less likely to break. As a rule, use bits whose cutting lengths are no longer than 3 times the diameter. When using small diameters it is sometimes advisable to go to a "stub" length bit, which has flute lengths only 2 times the diameter of the bit.

Cutting Diameter

This is the cutting diameter of the end mill. Always use the largest diameter allowable.

Cutter Failure

Heat is one of the main causes of cutter edge failure. It is present in all milling operations and is caused by the friction of the cutter and the material coming into contact. Heat cannot be eliminated totally but by using the correct and sharp cutting tools, proper feeds, and speeds for the material being machined. And proper application of coolant it can be minimized.

Friction and heat are interrelated so when dealing with friction you would apply the criteria as heat.

Chipping or crumbling of cutter edges occur when cutting forces impose a greater load on cutting edges that their strength can withstand, small fractures occur and small areas of the cutting edges chip out. Possible causes of chipping and crumbling:

i. Excessive feed per tooth
ii. Poor cutter design
iii. Running cutter backwards
iv. Chatter due to a non-rigid condition
v. Inefficient chip washout
vi. Built-edge break away

Built-up edge occurs when particles of the material being cold-weld or otherwise adhere to the faces of teeth adjacent to the cutting edges. When this occurs the tool can no longer cut cleanly.

Two other factors to be taken into consideration are rigidity of the table and if coolant will be use. When machining aluminum, coolant should always be applied, if you are to have any success. When it comes to rigidity, if your machine is loose, you will not be able to machine at fast feed rates.

Router Table Rigidity

Basically any vibration in the Router tables arm will be transmitted to the tip of the end mill. The faster you go the greater this vibration is amplified and the more likely that you will break the bit because of it. As a general rule the heavier the gantry the better. As you increase cutting speed the gantry at some point will begin to vibrate. At this point you are running too fast for the design of the Router table. Slow down.

In larger tables, this vibration should only become a factor in larger diameter bits, i.e. greater than 1 /4 inch. With smaller bits the speed limitation imposed by other variables will prevent you from reaching the point where it will vibrate. In smaller tables with lighter gantries this vibration will be a limiting factor.

Misting

When cutting metals, a mister is of critical importance. Without it the end mill and/or the chips heat up to the point that the bit will break or the material chips will melt. If you are doing a lot of metal cutting, dual misters are required. There are two types of misters, positive feed and non-positive feed. The positive feed type actually pumps a measured amount of misting fluid out of the nozzle.

Feed and Speed Calculations

Parameter	Formulas
Cutting speed, (srm)	**SFM = .262 x x RPM**
Revolutions per minute (rpm)	**RPM = 3.82 x SFM/0**
Freed rate, in/min	**Fr = Cpi x N x RPM**
Feed per tooth, in	**CPI Fr/N x RPM**
Cutting time, min	**t L / Fr**
Rate of metal removal, cu in/min	**mrr=Wx0xFr**

Symbol Definitions

D=	**Diameter of milling cutter, inches**
RPM=	**Revolutions per minute of cutter**
SFM=	**Cutting speed, feet per minute**
Fr=	**Feed rate, inches per minute**
Cpt=	**Feed, inches per tooth**
N=	**number of teeth in a cutter**
Mrr=	**Rate of metal removed, cubic inches per minute**
t=	**Cutting time, minutes**
L=	**Length of cut, inches**
D=	**Depth of cut, inches**
W=	**Width of cut, inches**

SOLID SURFACE ROUTER BITS

Solid Carbide Double Edge Upcut Spiral

Application: Designed for perfect balance and ultra smooth finish over a wide speed range.

CED	CEL	SHK DIA	OAL
1/4	3/8	1/4	2 1/2
1/4	3/4	1/4	2 1/2
3/8	1	3/8	3
1/2	1 1/8	1/2	3 1/2

Solid Carbide Three Edge Finisher Slow Helix

Application: Designed for perfect balance and ultra smooth finish over a wide speed range.

CED	CEL	SHK DIA	OAL
3/8	5/8	3/8	3
3/8	1 1/8	3/8	3
1/2	1 1/8	1/2	3 1/2
1/2	1 5/8	1/2	4
1/2	2 1/8	1/2	4 1/2
3/4	1 5/8	3/4	4
3/4	2 1/8	3/4	5

CED- Tolerance + .000 - .0004"

Carbide V Bottom

Application: Decorative and specialty two flute tool designed for grooving and beveling.

CED	CEL	SHK DIA	OAL
3/16	5/8	1/4	2
1/4	3/4	1/4	2
3/8	3/4	3/8	2 1/2

CED Tolerance +.000 - .005”

Solid Carbide Single Edge 0 Flute Metric & Imperial

Application: Reinforced design, suitable for a variety of solid surface materials.

CED	GEL	SHK DIA	OAL
1/2	1 1/4	1/2	3

Solid Carbide Single Edge Upcut Spiral 0 Flute

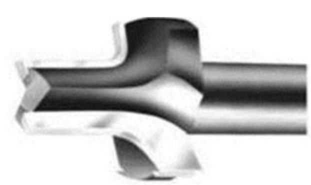
Application: Provides a smooth finish in solid surface materials with upward chip removal.

CED	GEL	SHK DIA	OAL
1/16	1/4	1/4	2
1/16	1/4	1/8	2
1/8	1/4	1/4	2
1/8	1/4	1/8	2
1/8	1/2	1/4	2
1/8	1/2	1/8	2
3/16	3/8	3/16	2
3/16	3/8	1/4	2
3/16	5/8	1/4	2
3/16	5/8	3/16	2
1/4	3/8	1/4	2
1/4	3/4	1/4	2 1/2
1/4	1 1/4	1/4	3
3/8	1 1/8	3/8	3

CED Tolerance + .000-/0004”

Carbide Tipped Double Edge Round & Rout

Application: Two flute cutter designed to put a radius on the edge and dress the stock.



CED	SM CED	CEL	SHK DIA	OAL	CE RAD
1	1/2	1 3/16	1/2	3 3/16	3/16
1	1/2	111/16	1/2	3 11/16	3/16
1 1/8	1/2	1 3/16	1/2	3 3/16	1/4
CED	**SM CED**	**CEL**	**SHK DIA**	**OAL**	**CE RAD**
1 1/8	1/2	1 11/16	1/2	3 11/16	1/4
1 3/8	1/2	1 3/16	1/2	3 3/16	3/8
1 3/8	1/2	1 11/16	1/2	3 3/16	3/8

ALUMINUM ROUTER BITS

Solid Carbide Single Edge Upcut 0 Flute

Application: Achieve premium finishes in aluminum with fast feed rates and excellent chip extraction

CED Tolerance + .000 - 004

CED	CEL	SHK DIA	OAL
1/16	1/4	1/8	1 1/2
1/8	1/4	1/8	1 1/2
1/8	1/4	1/4	2
1/8	1/2	1/4	2
3/16	3/8	3/16	1 1/2
3/16	3/8	1/4	2
3/16	5/8	1/4	2
1/4	3/8	1/4	2
1/4	3/4	1/4	1 1/2
114	1 1/4	1/4	3
3/8	3/4	3/8	3
3/8	1 1/8	3/8	3

Solid Carbide Single Edge Upcut 0 Flute

Application: Achieve premium finishes in aluminum with fast feed rates and excellent chip

CED	CEL	SHK DIA	OAL
1/8	5116	1/4	1 1/2
1/8	5/16	1/8	1 1/2
3/16	1/2	1/4	2
3/16	1/2	3/16	2
1/4	5/8	1/4	2
3/8	1	3/8	3
1/2	1 1/8	1/2	3 1/2

2 Flute Stub Lengths Carbide End Mills

Application: General purpose tool

CED	CEL	SHK DIA	OAL
118	1/4	1/4	2
3/16	3/8	1/4	2
1/4	1/2	1/4	2

2 Flute Standard Lengths Carbide End Mills

Application: General purpose tool

CED	CEL	SHK DIA	OAL
1/8	1/2	1/4	2 1/2
3/16	5/8	1/4	2 1/2
1/4	3/4	1/4	2 1/2

2 Flute Solid Carbide Spiral Extrusion Router Bits

Application: Machining aluminum extrusion

CED	CEL	SHK DIA	OAL
5/16	3/4	1/2	3
3/8	13/16	1/2	3
1/2	1	1/2	3 1/2

WOOD PRODUCT ROUTER BITS

Solid Carbide Single Edge Compression Up/Down Spiral

Application: Upcut/downcut design for fast feed rates and optimum edge finish on both sides of laminated

CED Tolerance + .000 - .005"

CED	CEL	SHK DIA	OAL
1/4	7/8	1/4	2 1/2
3/8	1 1/8	3/8	3
1/2	1	1/2	3

Solid Carbide Mortise Single Edge & Double Edge Compression Spiral

Application: Superior material removal and optimum finishes in natural woods and wood composites. Short upcut cutting length for use on mortises cuts.

CED	CEL	SHK DIA	OAL
1/4	7/8	1/4	2 1/2
1/2	1 5/8	1/2	3 1/2
1/4	7/8	1/4	2 1/2
1/2	1 3/8	1/2	3 1/2
1/4	5/8	1/4	2 1/2

Solid Carbide Double Edge & Three Edge Chipbreaker Finisher

Application: This tool allows for high feed rates and smooth edge finish.

CED	CEL	SHK DIA	OAL
3/8	1 118	3/8	3
112	1 5/8	1/2	3 1/2
3/8	1 118	3/8	3
1/2	1 5/8	1/2	3112

Solid Carbide Double Edge Straight Wood Rout

Application: The double edge wood rout in straight flute configuration provides a superior finish & provides optimum cutter life.

CED	CEL	SHK DIA	OAL
3/16	3/4	1/4	2
1/4	7/8	1/4	2 1/2
3/8	1 1/8	3/8	3
1/2	1 1/8	1/2	3
1/8	1/2	1/4	2
1/4	3/4	1/4	2 1/2
3/8	7/8	3/8	2 1/2
1/2	1	1/2	3

CED Tolerance + .000 - .004"

Solid Carbide Double Edge Compression Up/Down

Application: Upcut/downcut design for fast feed rates and optimum edge finish on both sides of laminated materials.

CED	CEL	SHK DIA	OAL
3/8	1 1/8	3/8	3
1/2	1 1/8	1/2	3
1/2	1 3/8	1/2	3 1/2
1/4	7/8	1/4	2 1/2
1/2	1 3/8	1/2	3 1/2
1/2	1 3/4	1/2	3 1/2
5/8	2	5/8	4
3/4	2	3/4	4
3/4	2 1/2	3/4	5
1/2	1	1/2	3
1/2	1 1/8	3/8	3
3/8	3/4	1/4	2 1/2

CED Tolerance + .000 - .004"

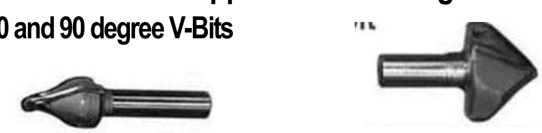
2 Flute Standard Lengths Carbide End Mills

Application: General purpose tool

CED	CEL	SHK DIA	OAL
1/8	1/2	1/4	1 1/2
3 1/16	5/8	1/4	2 1/2
1/4	3/4	1/4	1 1/2

2 Flute Carbide Tipped "V" Grooving Bit

60 and 90 degree V-Bits

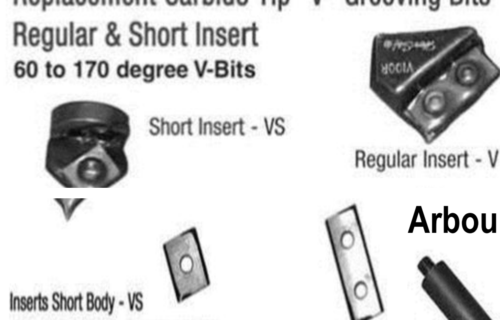


CED	CEL	SHK DIA	OAL
1/2	1/2	1/4	1 3/4
1	3/4	1/2	2 3/8
1/2	1/2	1/4	1 3/4

Replacement Carbide Tip "V" Grooving Bits

Regular & Short Insert

60 to 170 degree V-Bits



Arbours

SHK DIA	OAL
1/4	1 1/7
1/2	1 1/2
1/2	2 3/8
1/2	3 1/4

PLASTIC ROUTER BITS

Solid Carbide Single Edge Upcut 0 Flute

Application: Provides smooth finish in plastics, with adequate speed and chip extraction

CED	CEL	SHK DIA	OAL
1/8	1/2	1/4	1 1/2
1/8	1/2	1/4	2
3/16	5/8	1/4	2
3/16	5/8	3/16	2
3/16	1 1/4	1/4	3
1/4	3/4	1/4	2
1/4	1 1/2	1/4	3
1/2	1 1/4	1/2	3

Solid Carbide Single Edge Upcut 0 Flute

Application: Soft, flexible, non-abrasive plastics (polycarbonate, polyethylene, polypropylene, soft ABS & PVC and PETG) the single edge spiral 0 flute provides a smooth finish in soft plastics with upward chip removal

CED	CEL	SHK DIA	OAL
1/16	1/4	1/4	2
1/16	1/4	1/8	2
1/8	1/4	1/4	2
1/8	1/4	1/8	2
1/8	1/2	1/4	2
1/8	1/2	1/8	2
3/16	3/8	3/16	2
3/16	3/8	1/4	2
3/16	5/8	1/4	2
3/16	5/8	3/16	2
1/4	3/8	1/4	2
1/4	3/4	1/4	2 1/2
1/4	1 1/4	1/4	3
3/8	1 1/8	3/8	3

Solid Carbide Single Edge Upcut 0 Flute

Application: Hard & rigid plastics (acrylic, phenolic, rigid ABS & PVC) the single edge spiral 0 flute provides a smooth finish in rigid plastics with upward chip removal.

CED	CEL	SHK DIA	OAL
1/16	1/4	1/4	2
1/16	1/4	1/8	2
1/8	1/4	1/4	2
1/8	1/4	1/8	2
1/8	1/2	1/4	2
1/8	1/2	1/8	2
5/32	9/16	1/4	2
3/16	3/8	3/16	2
3/16	3/6	1/4	2
3/16	5/8	1/4	2
3/16	5/8	3/16	2
7/32	3/4	1/4	2 1/2
1/4	3/8	1/4	2
1/4	3/4	1/4	2 1/2
1/4	1 1/4	1/4	3
3/8	1 1/8	3/8	3

CED Tolerance + .000 - .004

2 Flutes Miniature End Mills

Application: Plastic sign making.

CED	CEL	SHK DIA	OAL
.031	.0930	1/8	1 1/2
.032	.0960	1/8	1 1/2

CED Tolerance + /- .0005"

ACM ROUTER BITS

Replacement Carbide Tip "V" Grooving Bits

90 to 170 degree V-Bits

Inserts Regular Body -

Solid Carbide Single Edge Upcut 0 Flute

Application: Achieve premium finishes with fast feed rates and excellent chip extraction.

CED	CEL	SHK DIA	OAL
3/16	1/2	1/4	2
3/16	1/2	3/16	2
1/4	5/8	1/4	2

2 Flutes Carbide Tipped "V" Grooving Bits

90 and 135 degree V-Bits

CED	CEL	SHK DIA	OAL	DEGREES
3/4	1/8	3/8	2 3/8	90
3/4	5/64	3/8	2 3/8	135

Arbours

SHK DIA	OAL
1/4	1 7/8
1/2	1 1/2
1/2	2 3/8
1/2	3 1/4

Solid Carbide Single Edge Upcut 0 Flute

Application: Achieve premium finishes with fast feed rates and excellent chip extraction.

CED	CEL	SHK DIA	OAL
3/16	3/8	3/16	1 1/2
3/16	3/8	1/4	2
3/16	5/8	1/4	2
1/4	3/8	1/4	2
1/4	3/4	1/4	1 1/2
1/4	1 1/4	1/4	3

CED Tolerance + .000 - .004.