

800 Liberty Drive Libertyville, IL 60048
800-234-1560 PH 847-362-5028 FAX

CNC Production Routing Guide

TABLE OF CONTENTS

	<i>PAGE</i>
INTRODUCTION.....	4
CUTTING TOOL MATERIALS & DESIGN.....	5
ROUTER BIT ANATOMY	6
TOOL WEAR	10
WORKPIECE MATERIAL CHARACTERISTICS	15
FEEDS & SPEEDS	17
FEEDS & SPEEDS FORMULAS	18
PROGRAMMING TECHNIQUES.....	20
PROPER COLLETING & MAINTENANCE.....	22
PROPER SPOILBOARD TECHNIQUES	28
APPLICATION ENGINEERING	30
WHICH ROUTER BIT SHOULD I USE?.....	31
GETTING MORE FROM YOUR CNC ROUTER	34
TROUBLE SHOOTING.....	39
ROUTING ALUMINUM	44
ROUTING PLASTIC	46

CUTTING TOOL MATERIALS

I. **Steel Router Bit Materials**

Carbon Steel

HSS-High Speed Steel - M2 / M7

Cast Alloys -TanTung, Stellite

II. **CARBIDE Router Bit Materials**

A. **TIPPED BITS**

1. **STEEL BODY**

a. **FATIGUE PROOF- Good**

b. **4140 Heat Treated - Better**

c. **H13 - Heat Treated - Toughest Best**

2. **BRAZED CARBIDE TIP**

a. **SILVER ALLOY BRAZE -Best Uniform Bond**

b. **Micrograin Carbide Tip**

B. **SOLID CARBIDE**

1. **Micrograin Carbide**

III. **CERAMIC**

IV. **DIAMOND**

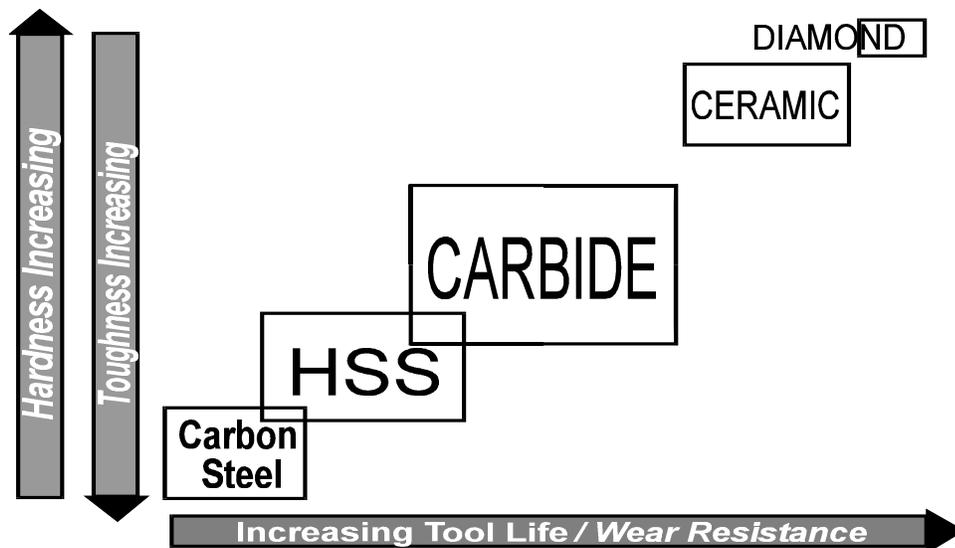
A. **PCD (Poly Crystalline Diamond)**

B. **CVD (Chemical Vapor Deposition)**

V. **SURFACE COATINGS**

A. **TIN**

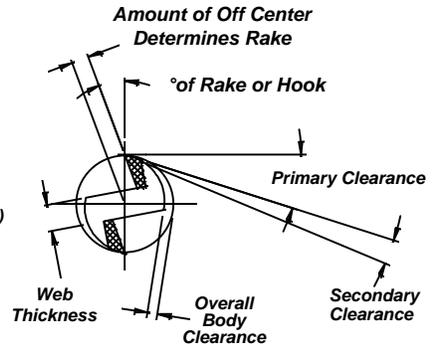
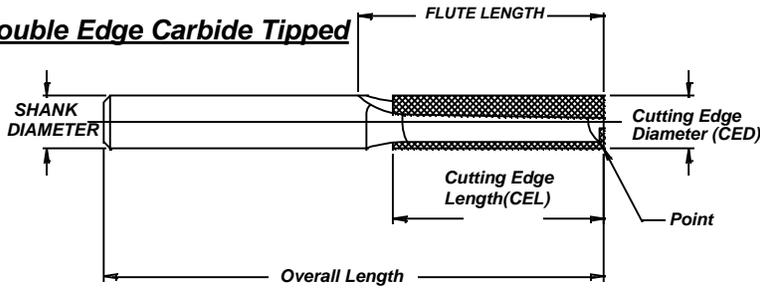
B. **ZRN**



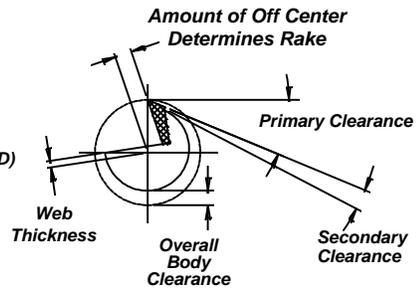
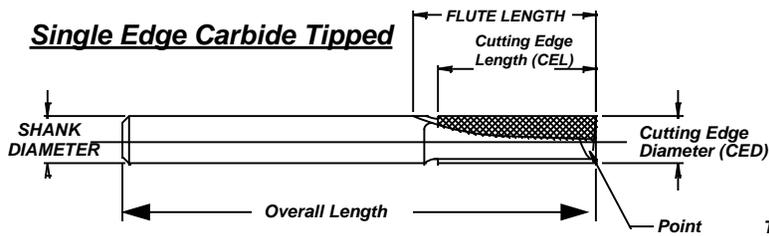
ROUTER BIT ANATOMY

Router Bit Terminologies - Double Edge Straight Bits

Double Edge Carbide Tipped

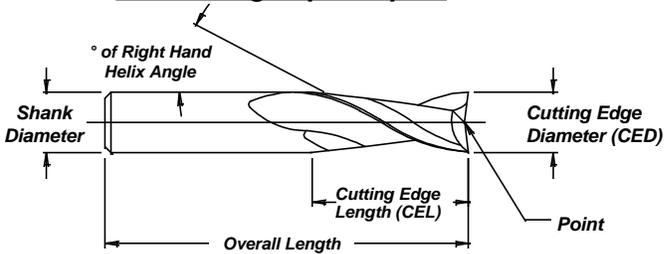


Single Edge Carbide Tipped

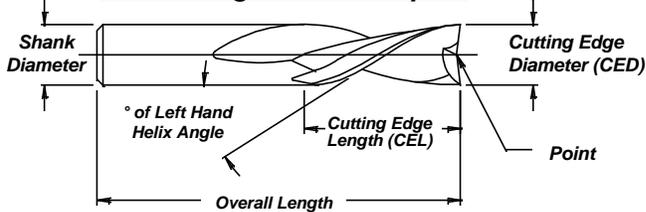


Double Edge Spiral Bits

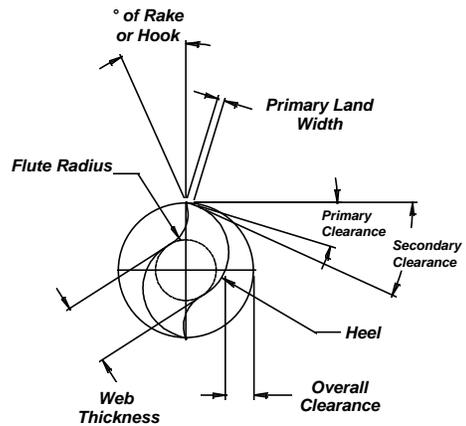
Double Edge Upcut Spiral



Double Edge Downcut Spiral

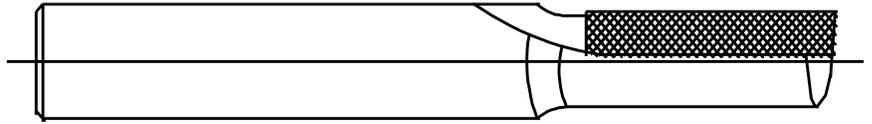


End View

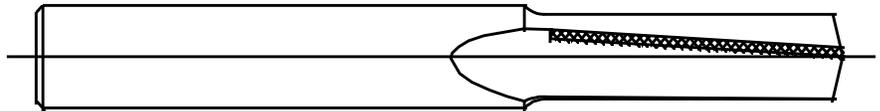


Cutting Tool Geometry

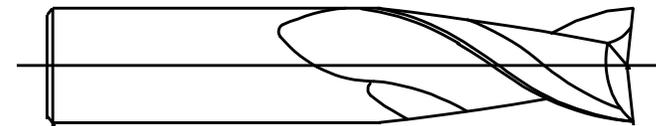
SE Carbide
Tipped Straight



SE Carbide Tipped
Shear Faced



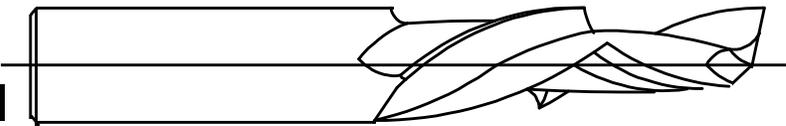
Solid Carbide DE
Upcut Spiral



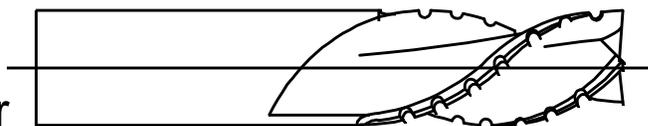
Carbide Tipped
Stagger Tooth



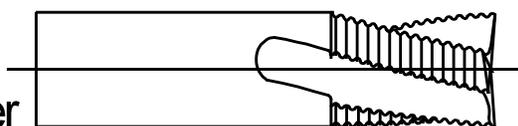
Solid Carbide
Up / Down
Compression Spiral



Solid Carbide
Three Edge
Chipbreaker / Finisher



Solid Carbide
Three Edge
Chipbreaker / Hogger



Router Bit Geometry Styles -

STRAIGHT

"O" Flutes
"V" Flutes

STRAIGHT WITH SHEAR

OnSheer
(Up to 15° is common)

SPIRALS

Upcuts
Downcuts
Helix Angles Fast and Slow

COMBINATIONS

Opposite Shear Staggertooth
Up/Downs
Roughers
Chipbreaker-Finishers

A GOOD TOOL DESIGN MUST HAVE A PURPOSE OR GOAL BY WHICH TO MEASURE ITS SUCCESS OR PERFORMANCE.

ROUTER BIT DESIGN IS A BALANCE OF VARIABLES ASSOCIATED WITH:

- *Tool Strengths & Weaknesses*
- *Workpiece Material Cutting Characteristics*
- *Machine Characteristics*
- *Methods To Be Used*

TOOL MATERIAL STRENGTHS & WEAKNESSES

- Tool wear strengths & weaknesses.
- Structural limitations relative to the geometry required on the cutting edge.(See chart of cutting wedge angles for different tool materials.)
- General size or geometry limitations of a material?
- *Standard rule of thumb* - No cutting edge length should exceed 3 times the cutting edge diameter. (We often make tools with 4:1 ratio or larger, i.e. 1/2" x 2 1/2"CEL)
- Manufacturing limitations making the tool?
- What are the economics of the tool for the application?
- What is the reliability of the material for the application, i.e., HSS versus carbide tools?

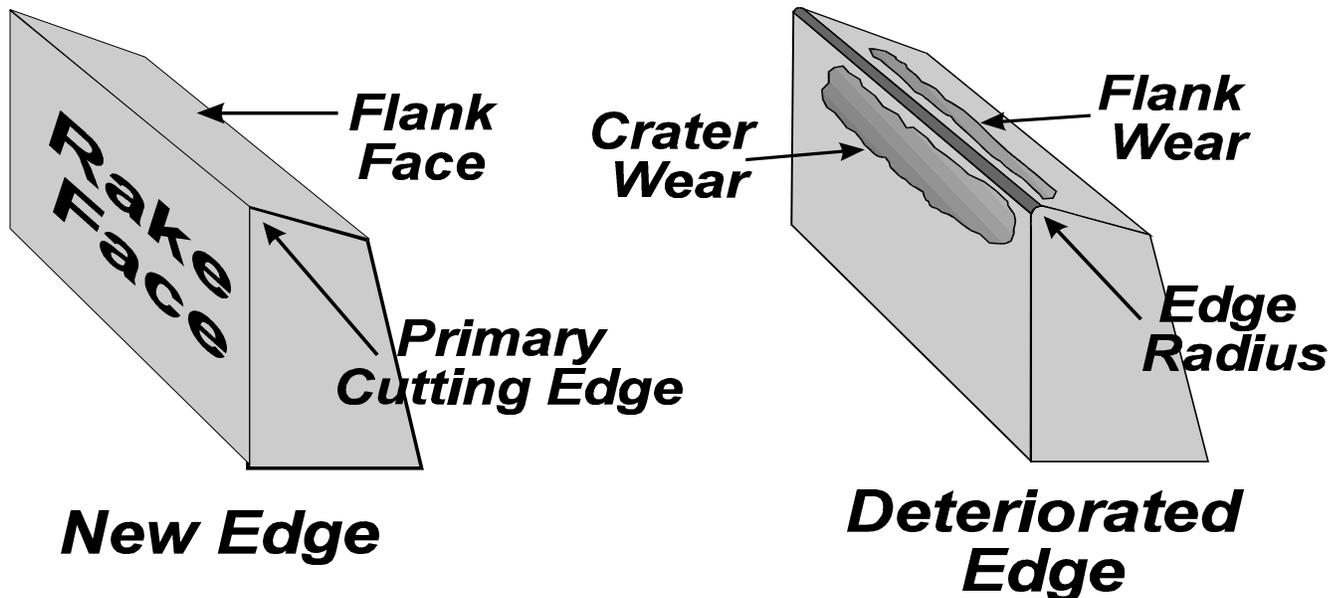
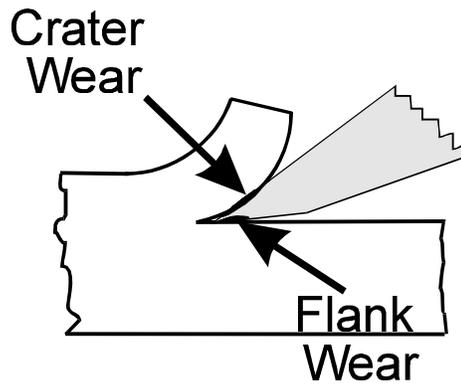
FACTORS INFLUENCING TOOL WEAR

MICRO-VIEW OF WHAT IS HAPPENING -

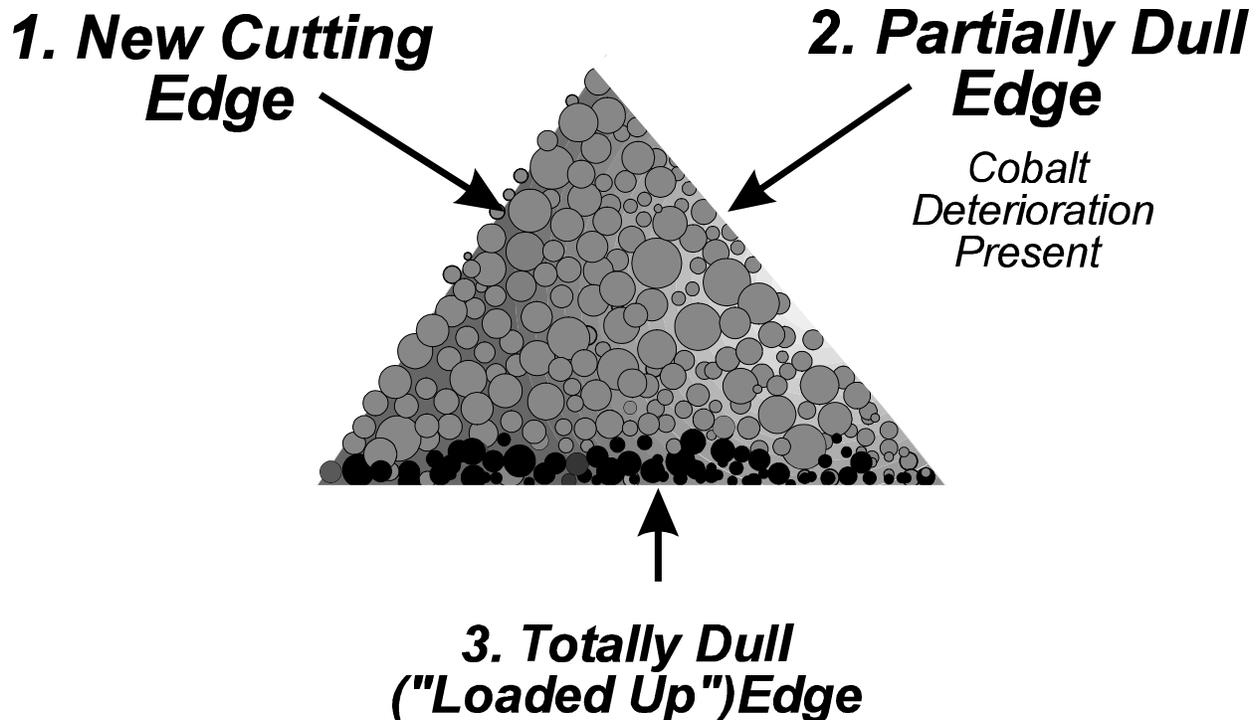
CAUSE	EFFECT
Friction Related <ul style="list-style-type: none">● Abrasion● Adhesion● Diffusion	<i>-Localized wear & build up</i>
Fatigue Related <ul style="list-style-type: none">● Cyclic Pressure● Cyclic Heat● Cyclic Stress	<i>-Micro chips of edge</i>
Atomic Transfer Related <ul style="list-style-type: none">● Hot Corrosion● Chemical Reactions● Mechanical Diffusion● Electrical Reactions	<i>-Corrosion of edges & tool diffusion into chips & cut material</i>
Structural Related <ul style="list-style-type: none">● Mechanical Limits	<i>-Edge chipping</i>

GENERAL TOOL WEAR DEFINITIONS

- **FLANK WEAR** - Clearance or relief side of the edge is wearing or rubbing. This is how most router bits wear. Clearance side of the cutting edge should rub a little for controlling feed and vibrations during a cut.
- **CRATER WEAR** - Rake face side of the tool's edge is wearing. Normally this is due to a very abrasive material flowing over the surface of the rake face. This is very rare in routing. However, with greater and greater chip loads this may become more of a factor.
- **EDGE RADIUS** - Localized wear of the very tip of the tool's edge. This is really a combination of both of above but concentrated on the very edge.
- **CATASTROPHIC FAILURE** - Chipping or breakage due to hitting something harder than the edge material or the tool and/or edge was too weak for the geometry or application of the tool.



The Dulling Process of Carbide



Edge Deterioration due to "Hot Corrosion" of the cobalt binder at the surface of the cutting edge caused by heat generated during the machining process. This "Hot Corrosion" process cannot be stopped!

It can however, be slowed substantially by dissipating the heat in the chip instead of in the tool. The larger the chips cut, the more heat they carry away from the tool!

REAL WORLD CRITICAL VARIABLES AFFECTING TOOL LIFE

I. TOOL EDGE MATERIALS

- A. High Alloy Tool Steels And Cast Alloys(Tan Tung)
- B. Carbide
- C. Ceramics
- D. Super-Hard Materials Diamond And CBN
Grain Size And Element Contents

II. MACHINING (CUTTING) RATES

- A. Part Configuration And Size
- B. Feed Rate (Chip Load)
- C. Spindle Speed (Rpm)
- D. Continuity Of Chip Load In The Cut

III. WORK PIECE MATERIAL CONDITIONS

- A. Moisture Content
- B. Hardness Surface And Core
- C. Grain Direction

IV. TOOL GEOMETRY

- A. Rake (Balance With Properties Of The Work Piece)
- B. Clearance (Behind Edge And Body)
- C. Rigidity Of Body
- D. Edge Surface Finish
- E. Special
 - a)Chip Flow - Upcut & Downcut Spirals, Shear
 - b)Chipbreakers Reduce Lateral Pressures And Heat

V. MACHINE CONDITIONS

- A. Rigidity Leading To Vibration
- B. Spindle Power And Feed Speed Capability
- C. Part Holding Ability, Vacuum Pressure CFM
- D. Programing Limitations
 - A. Point To Point
 - B. Circlular Curves
 - C. Sharp Corners

VI. MISCELLANEOUS FACTORS

- A. Special Tool Size, Shape, Profile
- B. Resharpener Considerations

HOW TO ADJUST REAL WORLD CRITICAL VARIABLES AFFECTING TOOL LIFE

TOOL MATERIALS

- Select corrosion resistant hard materials for the cutting edge that can withstand the abuse.
- Select a body material that is both tough and stiff to minimize vibration.

MACHINE CUTTING METHODS

- Maximize the chip load by increasing the feed rate as high as possible while achieving an acceptable cut.
- If you still want faster feed rates, look at increasing the RPM or the number of flutes.
- If you can't run fast due to part configuration, *slow* the RPM down to increase the chip size.
- Never stop or dwell in a cut if you don't have to.
- Cutting bigger chips will keep the tool cool.

TOOL GEOMETRY

- Add rake for speed and continuous chip styles and use only enough clearance to keep from rubbing.
- Design for rigidity.
- Grind for sharpness and keen edges.
- Keep the flutes open for chip flow at high speeds.

MACHINE PROGRAMMING TECHNIQUES

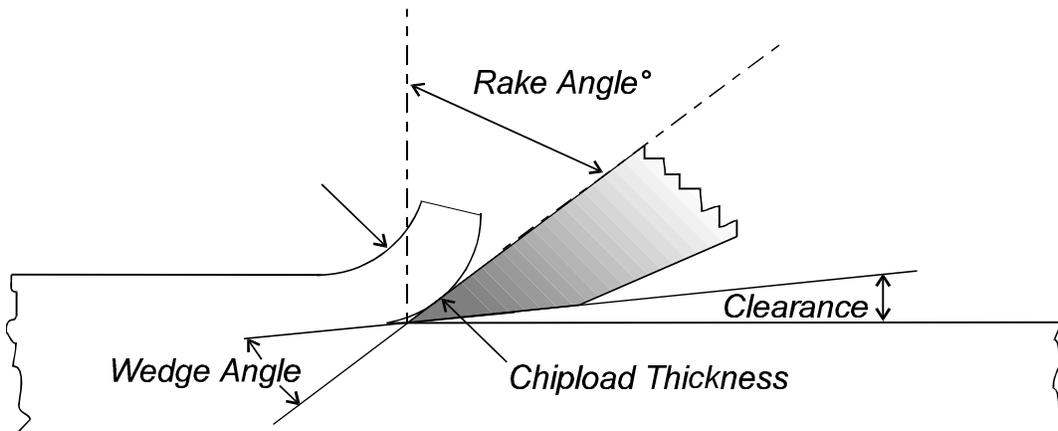
- Use a stout heavy machine to minimize vibration.
- Rigid work piece holding capability
- Chew up small diameter plugs generated from interrelation of holes to minimize tool damage or personal injury caused when these plugs kick out.
- Use programming techniques to your advantage such as oscillating the tool to prevent wear lands at the surface of the board or glue line.

- Machining Properties Of The Work Piece Materials
- Edge Finish Required On The Cut Surfaces
- Directional Grain In The Materials
- Spring Back Of The Materials During The Cut
- How Do The Work Piece Materials Wear The Tool Materials?

Rake, Clearance, And Flute Design Are The Main Geometry Adjustments To Match The Above Requirements.

CHART OF WOOD CUTTING ANGLES

Wood Species	Kiln Dried	Wood Species	Kiln Dried
Ash	15°	Maple(Soft)	15°
Basswood	20°	Maple(Hard)	5°
Beech	15°	Red Oak	15°
Birch(Plain)	15°	White Oak	10°
Cedar	20°	Yellow Pine	15°
Cypress	20°	White Pine	20°
Elm	20°	Poplar	15°
Fir	15°	Redwood	20°
Hemlock	20°	Spruce	20°
Hickory	10°	Walnut	15°
Mahogany	15°		



Regardless of the type of routing machine used, there are basic principles that insure the effectiveness of the tooling and the quality of the part being routed. They include the following:

RIGIDITY

The part being machined must be held firmly by mechanical or vacuum holddown devices to insure a quality cut, maximize tool life, and minimize premature breakage problems.

MAINTENANCE

Collets, collet nuts and spindle should be part of an on-going maintenance program. These areas should be properly cleaned with each tool change and collets should be replaced on a schedule relative to the production environment. Spindles should be checked for concentricity with a dial indicator as indicated in the back section of the Onsrud Cutter catalog to insure the router tool is running in a true circle.

DUST COLLECTION

Adequate chip removal is necessary to carry the wood chips away from the work piece and the tool. Feeds and speeds can be adjusted to increase the chip load and take more heat away from the cutter, but the dust collection system must be powerful enough to carry the chip away from the work area or they will crowd the tool.

TOOL SELECTION

Choose the right tool for the job and follow proper procedures for colletting, running and reconditioning the tool.

FEEDS & SPEEDS

To understand the concept of speeds and feeds, it is necessary to visualize what is occurring at the cutting edge of the tool. A chip of material is being removed from the base part. The size and thickness of the chip is controlled by the speed of the rotation speed of the spindle and the forward movement cause by feeding the tool into the material. If there is one flute, then the chip load is equal to the amount of travel in one revolution of the spindle. If there are two flutes, then there are two chips equal to one-half of the amount of travel in one revolution. If there are three flutes, then the chip load is one-third of the amount of travel in one revolution.

Most of the energy expended during these actions is released as heat. Heat is one of the major factors in tool wear. The most effective way of getting rid of the heat is by having it carried away with the chip. This can be accomplished by cutting larger chips which both dissipate heat as well as yield a high quality part edge finish due to minimization of re-cut chips. This is possible if you have a tool that possesses a geometry that allows for both speed and finish characteristics. The tool selection guide on page 2 of the Onsrud Cutter catalog will help you determine the proper tool type. The feed and speed chart in the back will give you the value for that tool vs. specific materials.

After running a program you can determine the actual feed value by timing one part, or a total cycle time for a complete table of parts. The formula will be provided later in the book.

There is another indication of proper speed and feeds, and that is the tool temperature. After a run of parts, and after the spindle stops, check the temperature of the tool. If it is hot, or warm to the touch, then the feed is too slow or the spindle speed is too high. If a proper speed and feed is used the tool should be at or near room temperature. Remember *heat* is what breaks down the cutting edge of the tool.

The first change to make is to the feed speed. This is *the* controlling factor in *productivity!* If the feed rate is at it's maximum due to part configuration, hold down capabilities, software limits, or machine limitations, then the spindle speed should be lowered. This does two things; 1) It increases the chip thickness and 2) It lowers the number of times the cutting edge is presented to the material. This second factor can be a major factor in increased tool life if this tool in this material only has a limited number of cuts per sharpening. This could increase tool life by 15 to 20%. It also reduces the spindle bearing temperatures by reducing heat transmitted into the spindle.

NOTE: If the machine is a shuttle or twin table model, and the reason the operator does not want to turn up the feed rate is because the machine would have to wait on the operators to clear the other table of finished parts.....let it wait or get another operator to help remove parts. ***Time is Money!*** The increased in tool life and reduction of wear and tear on the spindles far outweighs the appearance of wasting machine time.

CNC FEEDS & SPEEDS FORMULAS

$$\text{Chip Load} = \frac{\text{Feed Rate (IPM)}}{\text{RPM X No. Of Flutes}}$$

(Inches)

$$\text{Feed (IPM)} = \text{RPM X Number Of Flutes X Chip Load}$$

$$\text{Spindle Speed} = \frac{\text{Feed Rate}}{\text{Number Of Flutes X Chip Load}}$$

(RPM)

FOR TIME STUDIES AND TRUE AVERAGE CHIP LOADS USE THE FOLLOWING:

$$\text{ACTUAL Feed Rate (IPM)} =$$

$$\text{CIRCUMFERENCE OF THE PART (INCHES) X 60 / RUN TIME (SECONDS)}$$

- **SET-UPS SHOULD BE VERY RIGID**
- **SPINDLES AND SLIDES - TIGHT AND TRUE**

Feeds and Speeds Chart

TOOL SERIE S	SOFT WOOD	HARD WOOD	PARTICLE BOARD	PLYWOOD	MDF	MDF w/ LAMINATE	FLEXIBLE PLASTIC	RIGID PLASTIC
10-00	200						150	
11-00							150	125
12-00	250	200					150	
12-80		200	250	350	250			
40-000								
48-000		250	350	350	300		150	150
48-600							350	250
48-700			500	450	450	400		
52-000								
52-200	250	350	400	400	400	350	250	
56-000	200						200	250
56-200	200	300	400	400	400	350		
57-200	200	300	400	400	400	350		
60-000	400	500	500	650	600			
60-100	350	350	500	500	500	500		
60-200	250	300						150
60-300	400	350	400	400	450			
60-350	350	400	450	450	500			
61-000	200						250	200
61-200	200	250	300	300	300			200
62-200	200	250	300	300	300			
63-200	200	250	300	300	300			

Above information is based on 1/2" tool diameter and cutting 3/4" thick material @ 18,000rpm.
These rate are only guidelines, actual rates may vary with the many variables that exist in the field.

BIT DIAMETER ADJUSTMENTS:

1/4" CED = CHART FEED X .6

5/8" CED = CHART FEED X 1.2

3/8" CED = CHART FEED X .8

3/4" CED = CHART FEED X 1.4

DEPTH OF CUT ADJUSTMENTS BASED ON CUTTING EDGE DIAMETER

3/8" AND BELOW SIZES:

Normal Depth Of Cut = 2 X Cutting Edge Diameter

Feed Rate = .75 X Value Found In Bit Diameter Adjustments

1/2" AND ABOVE SIZES:

Normal Depth Of Cut = 3 X Cutting Edge Diameter

Feed Rate = Full Chart Values

Feed Single Flutes Slightly Faster X 1.1

Programming Techniques

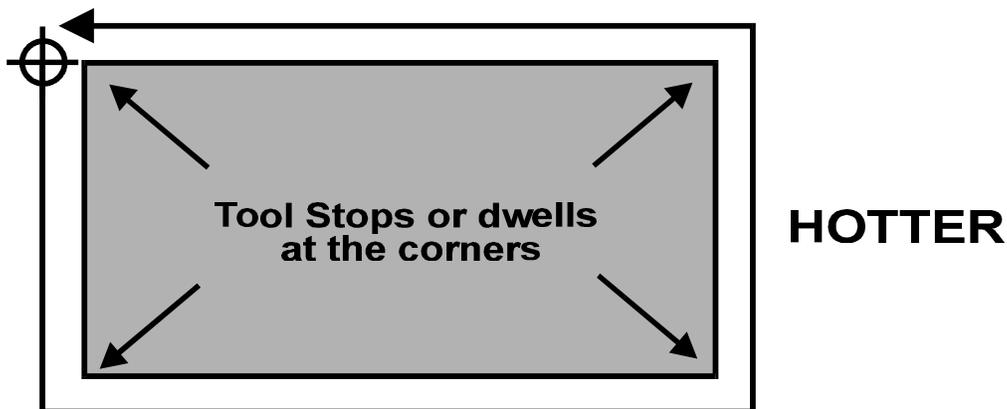
Parts programming can be a major factor in speeds and feeds. All software and machine controllers have acceleration, deceleration, and curve speed limitations. To maximize these items several things can be done.

Avoid Dead Stops

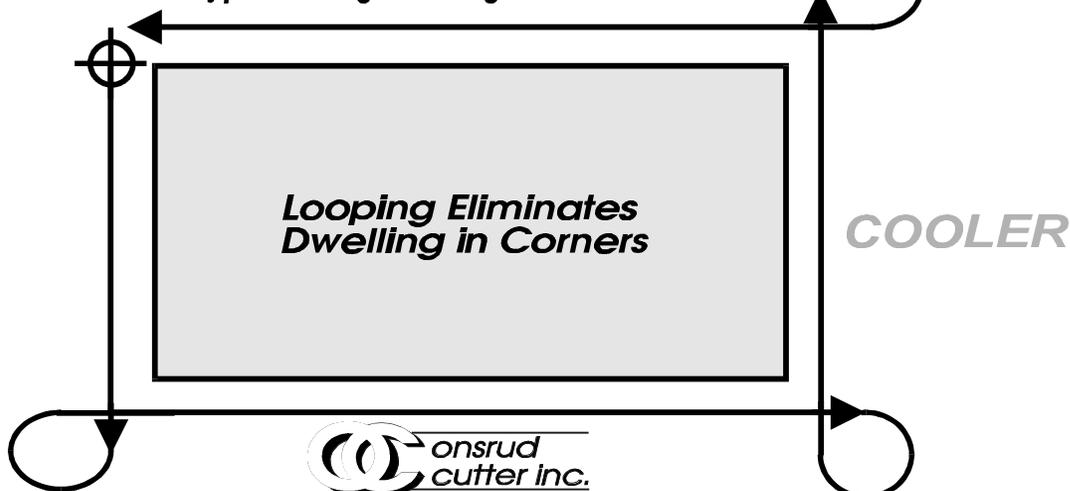
At 18,000 RPM, the tool rotates at 300 times per second. If the tool is in the work, it is rubbing the edge and creating heat. At corners the tool stops when it changes direction. If an exit-ramp loop is used, the tool does not stop moving, and the intersections of the two straight lines make a square corner. Another way is by putting a very small radius of .001", on the corners and this will also prevent these stops. These techniques can, if employed, be easier on both tools and machines.

"Exit Ramp" Programming

Traditional Programming For Outside Corners



Interstate Type of Programming for Outside Corners



Parts That Are Mirrored Or Nested

If possible, try to make the tool run the edges of both parts in one pass. This is a shared edge cut which means one edge is climb cut and the other is conventional cut. This works well for parts that do not require high part edge quality finishes. Example: Plywood panels for the boating industry covered with upholstery or laminated in fiberglass, raw MDF that will be covered with laminate, stair risers, etc. Be careful when using mirroring techniques in programming that you don't mirror circles or often you will climb cut one circle while conventional cutting the other.

Conventional Cutting

Typically conventional cutting provides you with the best edge provided you have picked the right tool geometry to cut the specific material. This applies mainly to manmade board products. If you are cutting solid wood where multidirectional grain patterns have to be considered, it is most often necessary to employ the climb cutting method thereby limiting the chip the tool can remove at one time and reducing splintering. That is the main application for climb cutting. Use conventional cutting wherever possible utilizing combination geometry tools to reduce fuzzing and tearouts. This allows the tool to clear the chip from the work instead of pushing into the work. Conventional cutting allows the chips to be thrown behind the tool yet climb cutting requires the chips be thrown in front and then run over, thus creating more pushing of the work piece due to preloading of the flute. If rough and finish passes are utilized, use a climb cut on the finish pass.

Oscillating The Tool For Better Tool Life

When cutting laminated materials such as plywood or laminated MDF or particleboard with a decorative surface such as melamine, glue lines represent the biggest threat to tool life. These glue lines create extreme amounts of heat focused as a beam of light through a magnifying glass, thereby breaking down one or more areas of the edge prematurely. One thing that can be done, provided you are using either dedicated spoilboards or a pod system, is to ramp the tool up and down through each tangent of the part so that the glue line is never focused on one spot of the tool for any length of time. This will increase the life of the tool substantially while not actually reducing cycle times at all. If you currently employ the method of dropping the "Z" axis when the tool starts to get dull, this is highly ineffective, because by the time you realize the edge is chipped, it is already too late to manually drop the "Z" axis, and the tool will then leave lines in subsequent parts.

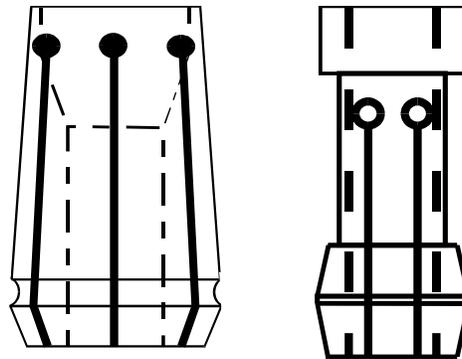
PROPER COLLET USE AND MAINTENANCE

Many users select tools without regard to the importance of adequately holding them in the collets on their router. This is often the case in routers ranging from simple air routers, to the most complex CNC machines in the marketplace. These collets come in two basic types, and their attributes and peculiarities are important in the way in which they secure the tool in the machine. Think of the spindle/collet system as a chain, and just like a chain is only as strong as it's weakest link, so too is the collet's 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*.

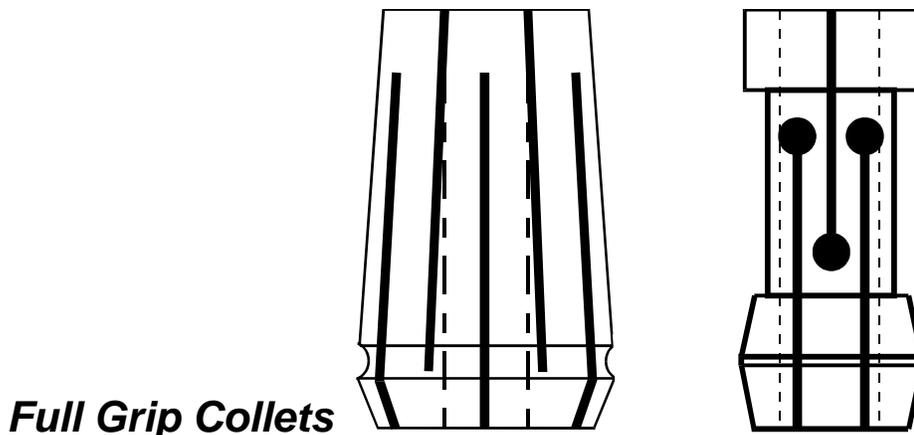
There are a number of things that are important to the system, the first of which we'll discuss are the two basic types of collets themselves. Both types are not always available depending on the spindle type itself.

- 1) Half Grip Collets are identified by their slits that run from the bottom or mouth to about 80% of the way to the top. This allows them to squeeze the tool with a force primarily directed at the mouth or bottom of the collet. These collets are often counterbored at the top so as to not require the tool to fill the entire length of the collet. This type of collet is the simplest of the two, and is ideal where tools sometimes don't allow a long enough shank to fill the entire collet length.

Half Grip Collets

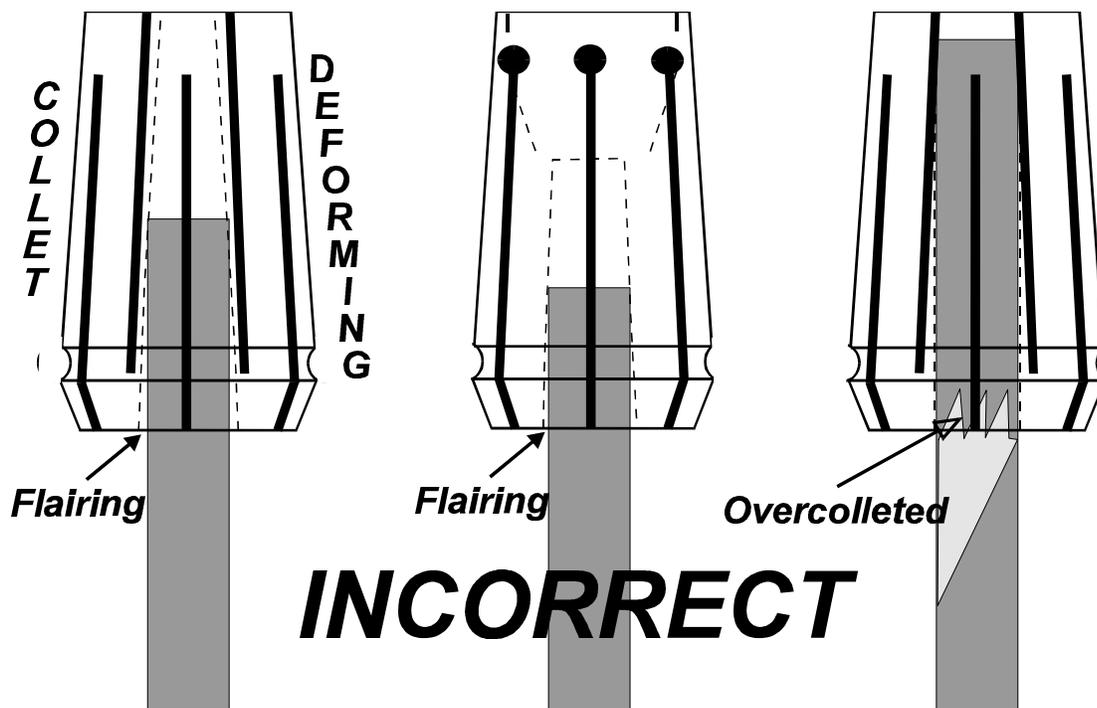
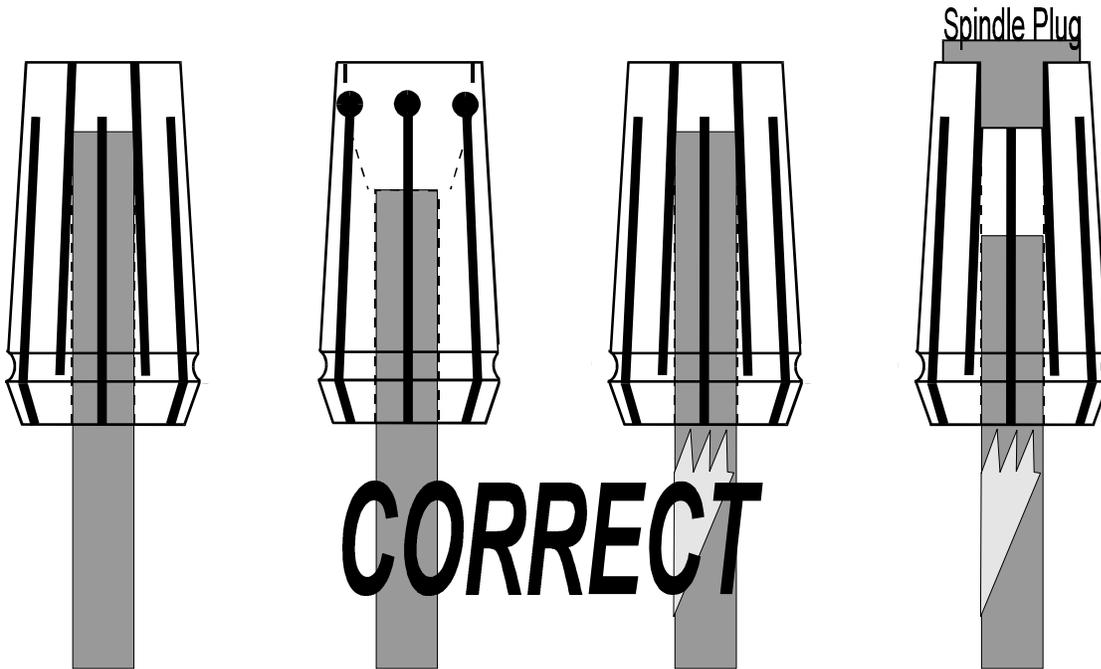


2) Full Grip Collets are identified by their slits that run from both ends, almost cutting the collet in pieces. This type of collet tends to have more flexibility and often comes in what is termed as "Range Collets", which allow gripping a range of shank sizes. Example: 12-13mm is used for 1/2" shank tools. This full grip type allows gripping over the entire length of the collet and requires that to be properly used, be filled 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 very minute amounts often times resulting in tool breakage. There are times that this 80% is not possible due to shank lengths available, so it is necessary to fill this void in the back of the collet with a filler plug that is of the same size as the shank to avoid the collapsing problem. Equally as important as filling the collet properly, it should also be understood that it is possible to overcollet 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 and, 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 can also be a common cause for tool breakage. It is also necessary to visually inspect the collets each time for wear from tool breakage when it occurs. Breakage often results in permanent damage to the collet due to intense pressures exerted often either "burring" or "mushrooming" the mouth of the collet.

PROPER TOOL COLLETING

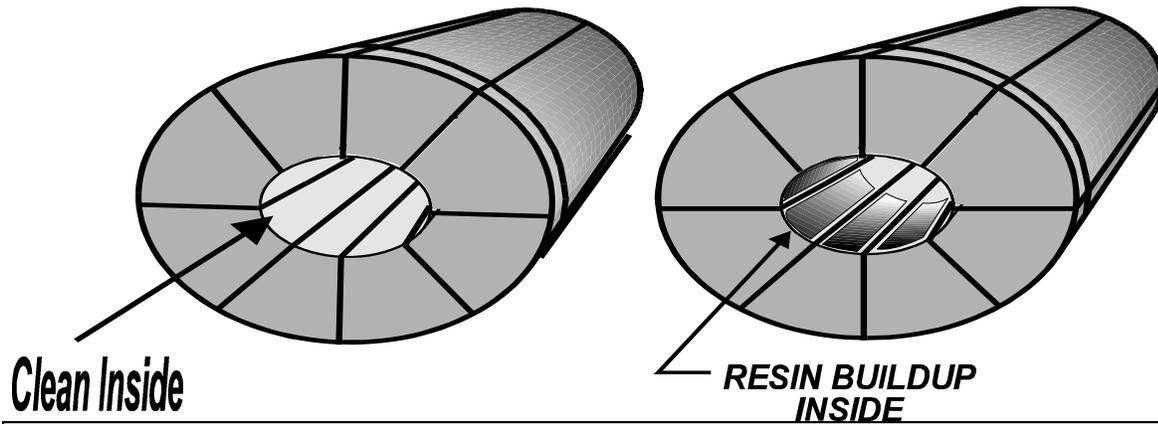


Heat is the biggest enemy of the tool, and the first place the heat goes from the tool is into the collet. It is also important to note that collets are made of spring steel that can, and will over a period of time lose its elasticity and harden, making it increasingly tougher to tighten adequately. 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.

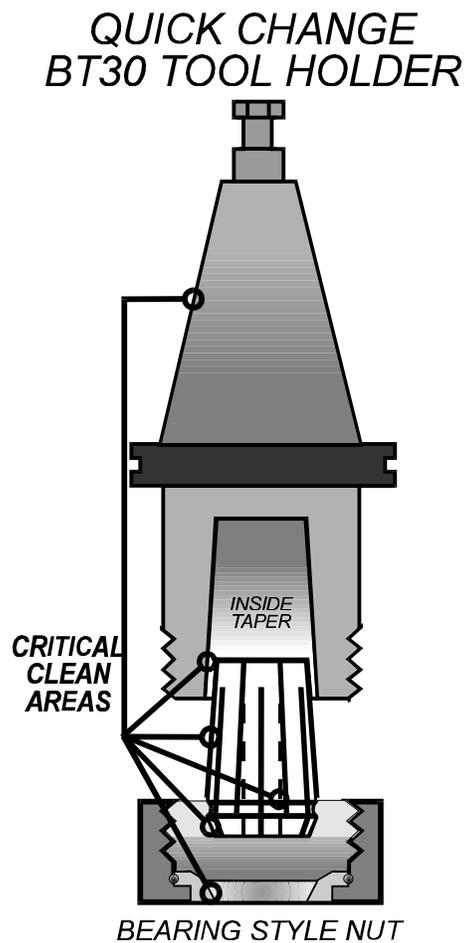
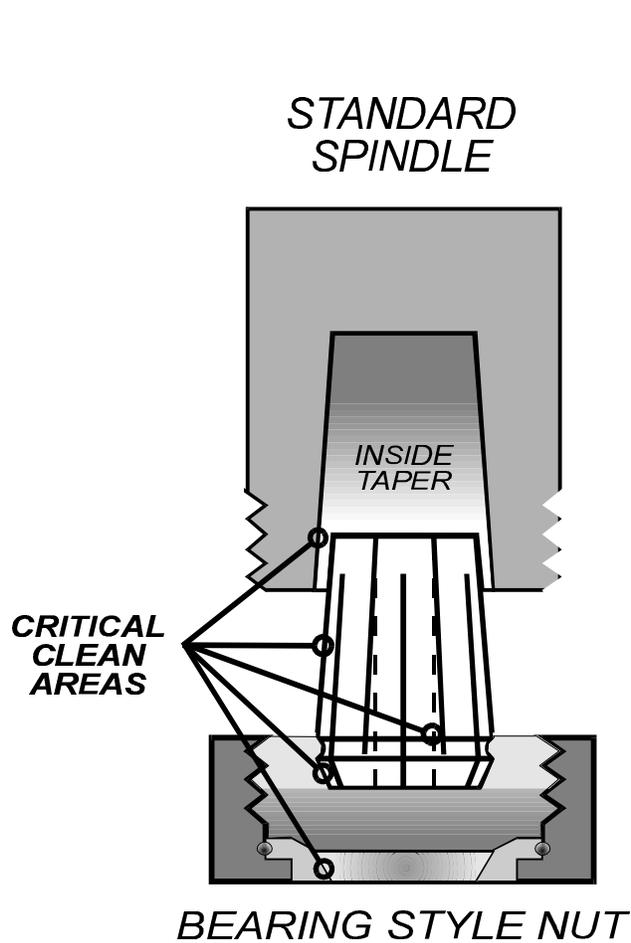
It is important to understand that if they are overrun enough, this overtightening will eventually damage the internal spindle taper resulting in costly repairs. Because it takes place over a period of time, it is very hard to notice but, a safe recommendation for collet life is in the 600-700 run time hours. This is about 3 months in a two shift operation of normal run time averages, probably much more frequent than one might expect. This is not the absolute, but it stands to reason if you're running a \$75 tool in the machine and you're only getting half the true tool life due to runout caused by bad collets, it doesn't take many tools to justify the added expense of replacing the collet. If collets are not changed, they will eventually become brittle enough to crack or break in half potentially 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!

Just as replacement as important, equally as important is cleaning the collets *each and every time the tools are changed*. 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 vibration 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, often almost gluing the tool into the collet leaving brown marks at the mouth of the collet contact on the shank. These brown marks are a 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. The Onsrud Cutter catalog provides collet maintenance information in the back and brass collet brushes are also contained in the catalog.

COLLET EXAMPLES



SPINDLE TYPES



As is the case with collet types, there are two spindle nut types as well. Both types are not always available depending on the spindle type itself.

- Bearing Type Nuts contain a thrust bearing, as illustrated at left, in the bottom of them allowing the collet to be tightened without the friction that normally is present as the spindle nut is tightened onto the collet.
- Solid Nuts are just that, and often require greater tightening pressure than their counterpart to attain equal grip on the tool, depending on collet nut design, diameter and taper of course. The force required to tighten them can sometimes be reduced with use of dry graphite film, but petroleum based lubricants should be avoided due to their ability to act as a magnet for any dirt or dust.

Variations on the above are “Quick Change” tool holders, also illustrated, come in a variety of sizes and styles and allow the convenience of changing the tools off of the machine allowing excellent opportunity to clean them properly. The most common types are BT30, BT35 and BT40 tapers. They differ from the standard spindle arrangement only in that there is one more set of matching and mating surfaces to keep clean.

Proper Gasketing of Spoilboards



■ = Gasket Tape



Gasket is Allowed to Compress into Groove and Allows Part to Seat on Spoilboard Like a Platen, thus Deadening Vibration of the Part



In addition to these concerns, the actual spoilboard is often significantly more effective if constructed of single or double sided melamine faced board to minimize vacuum leakage as is allowed with raw board. This way the only area that vacuum is allowed to draw through is the vacuum port holes drilled through the surface. If single sided board is used it should be face down. A tremendous amount of vacuum can be lost through the board itself which becomes very critical when holding small parts.

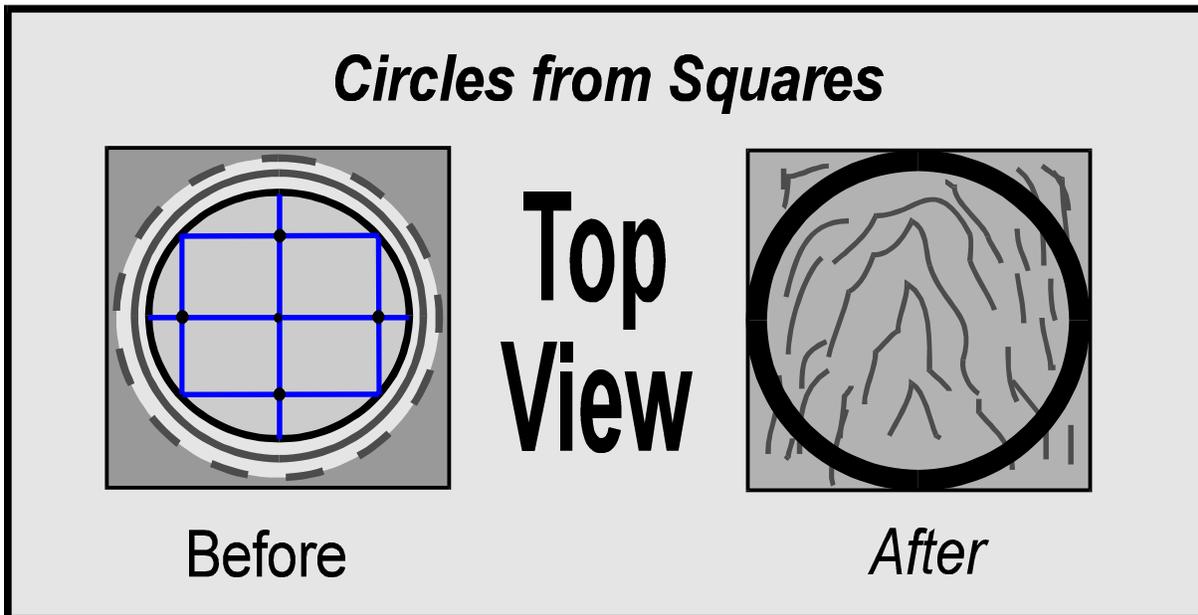
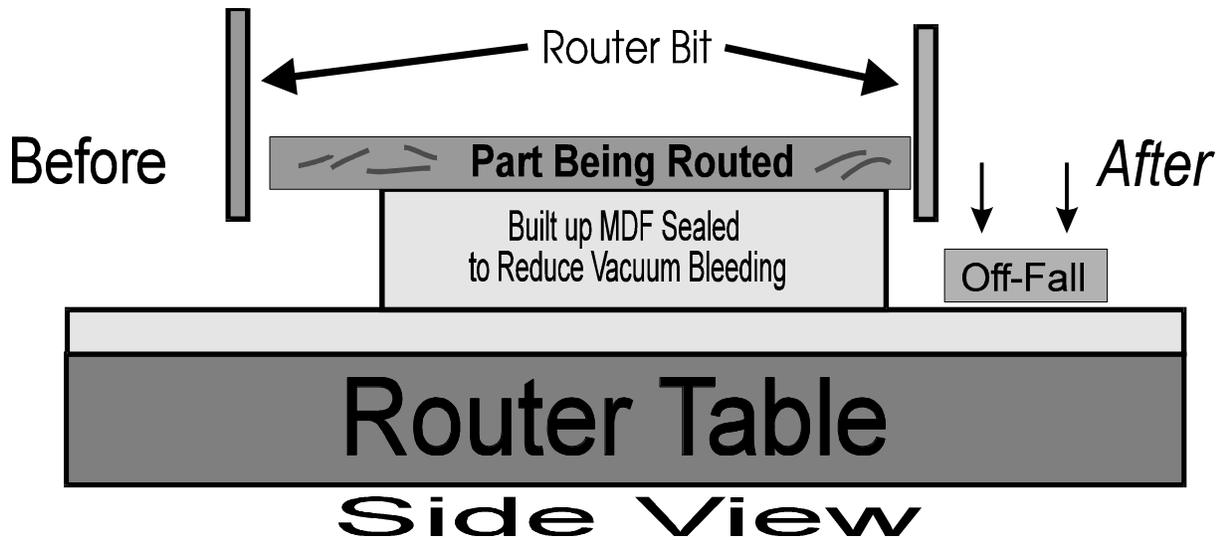
"Suck-Through" is another technique gaining in popularity due to it's minimized set up time. This method relies on particleboard being porous enough to allow a large vacuum pump of 20-30HP to actually draw through the board. This method can be successfully employed but it can have it's quirks in small slippery parts such as plastic.

One must definitely keep in mind tool selection is very dependent on the techniques employed to hold the parts on the table and that the choices must be relative.

Raised spoilboards are another technique that works quite well for routing parts like circles from squares and other parts which have off-fall of significant size to be of harm to both tool and operator when it kicks. This is often encountered in parts where a two pass operation is performed. Such is the case with a bar stool top where we would first want to rough out the part with a chipbreaker tool, and then come back with a final pass with a 3 edge finisher or shape tool if a shaped edge is required. We want to make sure that the off-fall doesn't interfere with the first or the second tool from doing it's job properly so we must insure that the off fall is free and clear of the tool path.

Such is the case in the example as follows. These particular raised pods are built of MDF layers and then sealed to prevent leakage or vacuum bleeding. The same scenario is true when working with pod systems available in the marketplace.

Raised Spoilboard or Pod Systems



*Allows Waste Parts Off Fall to Drop Clear of Tool
Common Examples : Lids, Seat Bottoms, Chair Parts*

APPLICATION ENGINEERING

Application engineering in the area of router tooling, like any other product, is the art of taking what you have learned and applying it to a specific situation. This process for router tooling seems to fall into three categories which include; needs assessment, problem solving, and tool selection.

Needs assessment is the initial process of deciding by asking some of the questions below in determining how to pick the tools and the peculiarities of the application at hand. Appropriate questions may include the following:

- What type of router is being utilized?
- What type of materials are being cut?
- What is the material thickness?
- What is the part configuration?
- Are there any inside radius requirements that require a small diameter tool?
- How is the part being held or fixtured, Vacuum, clamps, pods, etc.?
- How will the tools influence the part, (upspirals lift and downspirals press down)?
- Which tools produce the least amount of lateral pressure on the part?
- Have you encountered problems with similar parts before?
- If CNC router to be utilized, what are the feeds and speeds?
- Should the parts be “skin cut” or “tabbed” due to minimal vacuum area?
- What is the *most* important consideration, *speed* or *finish*?
- Which tools will give the best finish?
- Which tools will allow the fastest feed rate?
- Which tools will last the longest?
- What tool will accomplish the best combination of all of these goals?

These are all steps that must be properly analyzed before making any fast decisions. Each item must be carefully weighed.

If we in fact are dealing with an application that is ongoing we must first do some detective work which includes not only asking ourselves the preceding questions but also looking for the root of the problem. Just changing tools may only put a “Band-Aid” on the problem and we would much rather solve the problem once and for all. As follows is a guideline of the various problem solving opportunities and possible solutions to these problems.

WHAT STYLE ROUTER BIT SHOULD I USE?

- **Look Up The Material In The Router Bit Selection Guide**
- **Examine The Alternatives**
- **Ask Yourself A Few Quick Questions:**
 1. What direction do I want the chips to go relative to the tool?
 2. Do I want a better finish or a faster cutting tool?
 3. What tool material is required to cut the work piece?
 4. What is the production requirement for the tool - high volume or low?
 5. Which tool will give me the lower overall cost per part?

A Good Tool Design Or Selection Must Have A Purpose Or Goal By Which To Measure Its Success Or Performance!

Your #1 Goal With Is The Combined Goal Of:

- *Best Finish*
- *Fastest Feed Rate*
- *Longest Tool Life*
- *Lowest Cost*
- *Fastest Delivery*

Has This Goal Been Achieved?

**NORMALLY THERE ARE TOOL DESIGN TRADE-OFFS:
FINISH -**

Multiple Cutting Edges Yield Better Finishes
Slower Feed Rates Yield Better Finishes
Larger Diameters Yield Better Finishes Due to Less Vibrations

Best Finish	Acceptable Finish
Slow Feed	Fastest Feed
Short Life	Longest Life Without Break

<i>Lowest Cost</i>	<i>Highest Cost</i>
Steel	Diamond

HARDWOODS

<i>STRAIGHT</i>	12-00 <i>DOUBLE FLUTE</i> 12-80 <i>DOUBLE FLUTE</i>
<i>SPIRAL</i>	40-000 HSS SE & DE DOWNCUT SPIRALS 40-200 HSS SE WOOD ROUT SPIRALS 40-000 HSS SE & DE UPCUT SPIRALS 40-300 HSS DE WOOD ROUT SPIRALS 40-400 HSS DE CNC SPIRALS
<i>STRAIGHT</i>	56-200 SC DE WOOD ROUT STRAIGHT
<i>SPIRAL</i>	52-200 SC DE WOOD ROUT UPCUT SPIRAL 57-200 SC DE WOOD ROUT DOWNCUT
<i>SPIRALS</i> <i>(SPECIALTY)</i>	60-000 SC 3 EDGE CHIPBREAKERS (ROUGHING) 60-168(HW) SC UP/DOWN COMPRESSION SPIRAL 60-300 SC DE CHIPBREAKER / FINISHER 60-350 SC 3 EDGE CHIPBREAKER / FINISHER 60-200 SC 3 EDGE FINISHER

SOFTWOODS

<i>STRAIGHT</i>	10-10 HSS SE STRAIGHT 10-80 HSS SE ON/SHEER 12-00 HSS DE STRAIGHT 12-80 HSS DE ON/SHEER
<i>SPIRAL</i>	40-000 HSS SE & DE DOWNCUT SPIRALS 40-200 HSS SE WOOD ROUT SPIRALS 40-000 HSS SE & DE UPCUT SPIRALS 40-300 HSS DE WOOD ROUT SPIRALS 40-400 HSS DE CNC SPIRALS
<i>STRAIGHT</i>	61-200 SC SE WOOD ROUT STRAIGHT 56-200 SC DE WOOD ROUT STRAIGHT
<i>SPIRAL</i>	63-200 SC SE WOOD ROUT UPCUT SPIRAL 62-200 SC SE WOOD ROUT DOWNCUT 52-200 SC DE WOOD ROUT UPCUT SPIRAL 57-200 SC DE WOOD ROUT DOWNCUT SPIRAL
<i>SPIRALS</i> <i>(SPECIALTY)</i>	60-000 SC 3 EDGE CHIPBREAKERS (ROUGHING) 60-168(HW) SC UP/DOWN COMPRESSION SPIRAL 60-300 SC DE CHIPBREAKER / FINISHER 60-350 SC 3 EDGE CHIPBREAKER / FINISHER 60-200 SC 3 EDGE FINISHER

Raw Particleboard or MDF

STRAIGHT	48-700 DE CARBIDE TIPPED MDF/CNC 56-200 SC DE WOOD ROUT STRAIGHT
SPIRAL	52-200 SC DE WOOD ROUT UPCUT SPIRAL 57-200 SC DE WOOD ROUT DOWNCUT SPIRAL

Laminated Particleboard or MDF (Vinyl, Veneer, Melamine, High Pressure Laminate, Paper, Etc.)

STRAIGHT	48-000 SE & DE CARBIDE TIPPED 48-700 DE CARBIDE TIPPED MDF/CNC 56-200 SC DE WOOD ROUT STRAIGHT
SPIRALS (SPECIALTY)	60-100 SE & DE SC UP/DOWN COMPRESSION SPIRALS (For Single sided laminated board) 52-200 SC DE WOOD ROUT UPCUT SPIRAL (Face Down) 57-200 SC DE WOOD ROUT DOWNCUT SPIRAL (Face Up)

Plywood All Types

STRAIGHT	56-200 SC DE WOOD ROUT STRAIGHT
SPIRALS (SPECIALTY)	(For excellent finish on both sides) 60-100 SE & DE SC UP/DOWN COMPRESSION SPIRALS (For good finish on one side) 52-200 SC DE WOOD ROUT UPCUT SPIRAL (Face Down) 57-200 SC DE WOOD ROUT DOWNCUT SPIRAL (Face Up)

Plastic- Flexible or "Gummy"

Polyethylene, Polypropylene, Polystyrene, ABS, PVC, Extruded Acrylic

STRAIGHT	11-00 SE & DE HSS "O" STRAIGHT 61-000P SE SC "O" FLUTE STRAIGHT 56-003P DE SC "O" FLUTE STRAIGHT(Found In 56-000 Series)
----------	--

Plastic- Rigid or "Breakable"

Acrylic (Cast & Some Extruded), Melamine, Phenolics, PVC, Nylon

STRAIGHT	56-000P SC DE STRAIGHT 56-003P DE SC "O" FLUTE STRAIGHT(Found In 56-000 Series)
SPIRALS (SPECIALTY)	60-000 SC 3 EDGE CHIPBREAKERS (ROUGHING) 60-200 SC 3 EDGE FINISHER(EXCELLENT FINISH)

GETTING MORE FROM YOUR CNC ROUTERS

EVALUATIONS REQUIRED

1. CONDITION OF THE MACHINE

- Spindle runout
- Collet, collet nuts, inside spindle taper
- Dust collection ability
- Ability of machine for higher feed speeds & still maintain pattern amp meters to objectively measure tool life

2. FIXTURING

- Ability to hold parts rock solid
- Properly made spoilboards allowing above
- Good pod systems if utilized that allow waste off fall to drop below tool path and clear subsequent tool pass to be made

3. MATERIALS CONSISTENCY

- Solid wood with consistent moisture content
- Board product free of foreign matter
- Laminated products with good integrity & stability

4. OPERATOR SKILLS & ABILITY

- Willingness to change methods of procedure
- Ability to adequately measure tool life
- Responsible for recording control data
- Ability to recognize when tool change required to eliminate over running the tool reducing total tool life
- Willingness to feed more parts across the machine to gain output

This concept of looking at tool choices is much more important than calling the supplier to get the cheapest bits to your plant when you need them. Many times the bit that looks like the most economical bit upfront, is not really that at all, but instead, the most expensive, based on cost per part. Many endusers buy router bits, but a few customers know the productive reasons to buy parts instead. They make tooling choices based on parts costs, and these costs often times are the difference between profit and loss. In order to do this they must know a number of cost factors in order to properly calculate what their actual tooling cost will be on a given part. Many dedicated manufacturers do not know why some of these associated costs are due to inability to monitor tool life and performance. Many job shops know machine time costs, but do not know tool life expectancies due to short runs. Both are aware that these hidden costs are present. In order for the customer to see the "forest for the trees" he or she must be aware of these associated costs and their effect on the bottom line. These are the customer's who have the abilities to adjust feed rates, track tool life, monitor results and project accurate costs.

The following report is designed to enable the end user to determine the actual costs associated with tooling on a particular application. Given that you can provide the information required on the report, you will have a much more accurate evaluation of actual tooling costs. **Note:** *The most important parameters that influence the results are total machining time required to run the job (O), and total router time costs (P).* This reinforces the “need for speed” in routing in order to get the most throughput on the machine. What’s surprising about this chart is that the actual tool price often has little effect on the bottom line!

Value Analysis Report

	DATA	Onsrud Cutter	Competitor	Customer	Sample	
A*	Number of Parts to be Produced	500	500	Contact		
B*	Machining Rate Cost @ Hour \$	\$ 80.00	\$ 80.00	Phone		
C*	Number of Passes x Length of cut(Inches) Per Pass	180	180	FAX		
D*	Spindle (RPM)	18,000	18,000			
E*	Feed Rate (Inches Per Minute)	600	200			
F	Chip Load $\frac{\text{Feed Rate}}{\text{RPM} \times \# \text{ of Edges}}$.033"	.022"			
G*	Tool Price	\$ 75.00	\$ 15.00			
	<i>Indexable Tools Only:</i>			Tool Tested	<i>Onsrud</i>	Competition
AA	Insert Cost Each			Tool Part #	60-165	DE CT Tool
BB	# Edges Per Insert			Diameter	1/2"	1/2"
CC	# of Inserts used Per Tool <i>Substitute 1 if No Inserts are Used</i>	1	1	Depth of Cut	1 1/4"	1 1/4"
H*	Time to Index or Change Tools(Minutes)	5	5	Climb/Conv.Cut	Conventian	Conventian
I*	Cost To Resharp the tool	\$ 24.50	\$ 6.00	CNC Router		
J*	Total # Parts Produced per resharp	116	50	Material Cut		
K	# of resharps to produce total number of parts required A(A/J)x CC	4.3	10.0	High Pressure Laminated Chipcore Desk top parts w/ a Particleboard Core		
L*	# of resharpenings available per tool	3	5			
M	# of Parts Produced Per Tool (JxL)	348	250			
N	# of Tools used to Produce A (A/M)xCC	1.4	2.0			
O	$(C \times A) / E \times 60 \text{ Minutes} = \text{Hours Runtime}$ <i>Total Machining Time in hours Required</i>	12.9	30.0			
P	Total Router time Costs (O x B)	\$ 1,034.48	\$ 2,400.00			
Q	Total Tool Costs = (N x G)	\$ 107.76	\$ 30.00			
Q	Insertables Costs = (N x AA) + I H x (K - 1) x B / 60 Minutes	NA	NA			
R	<i>Tool Changeover- Downtime Costs</i>	\$ 22.07	\$ 60.00			
S	Regrind Cost I x (K - 1)Omit for Indexables	\$ 81.10	\$ 54.00			
T	Total Costs (P + Q + R + S)	\$ 1,245.41	\$ 2,544.00	Directions: All Lines with an * beside the letter must be completed in order to successfully use this formula for costing. Once completed , Perform the functions that each line requests and your analysis will be complete!		
U	Cost Each (Per Part) T/A	\$ 2.49	\$ 5.09			
	Total Savings with OC Tools vs. Comp.	\$ 1,298.59				
	Savings Per Part	\$ 2.60				
	% of Savings $\frac{\text{T Cost vs. OC}}{\text{T Competitor}} \times 100 = \%$	49.0%				

This preceding form is provided as an example, you will find a blank version of this formula in a worksheet form at the end of this book that you can copy and use for your own formulations. We hope it will be of profit in it's use!

Now that you understand the importance of why it is necessary to select the "right tool for the job", you will no doubt want to know more about how to do just that based on the application you are using the tools on. There are many parameters which effect tool selection, some of which are listed below:

TOOL SELECTION

- Match the tool with the material to be routed with tooling as specifically matched with the application as possible to ensure productivity.
- Choose the tool with the shortest cutting length available that allows the part to be cut as well as the shortest overall length to maintain tool rigidity.
- Choose the tool for the speed of cut and finish desired, i.e., single flute for speed and larger chiploads and double flute for finish.
- Keep cutting diameters as close as possible to shank size to maintain better tool strength, i.e., 1.2" diameter cutting edge on a 1/2" shank.
- Do NOT make tooling choices based heavily on initial tool cost. Often times the more productive tools with higher costs more than pay for themselves in reducing overall machining time offering a better payback than the inexpensive tools.

In addition to these factors, you will also want to make note that there are other influential factors which can make tool selection even trickier that all have to be taken into account. They are as follows:

TOOL SELECTION VARIABLES

Parts Fixturing Concerns

- **Problem:** Up Spiral tools lifting parts off vacuum hold down system
- **Solution:** Use a straight tool for neutral pressure and a down spiral tool to increase downward pressure. Use an up/down Compression Spiral for neutral pressure as well as reduction of lateral pressure due to breaking up of the cut into two parts.
- **Problem:** Poor chip extraction using a straight Carbide Tipped tool
- **Solution:** Use up spiral Wood Rout tools to extract them from the slot. If too much lifting pressure is created with up spirals use OnSheer tools or Chipbreaker Finisher series tools which further break up the chips and reduce lifting pressure by taking smaller bites of material at a time.

Small Parts Dilemmas

- **Problem:** Inability to maintain vacuum on a part consistently.
- **Solution:** Check spoilboard building techniques and make sure you are using a gasket tape with some memory, and not weather-stripping from the local hardware store! Combine this with use of possibly smaller diameter tools if application allows thus reducing the lateral pressure exerted on the part, i.e., 1/2" instead of 3/4" or 3/8" instead of 1/2" diameters.
- **Problem:** Parts seem to lose vacuum no matter what when they are through cut regardless of spoilboard design and tooling considerations.
- **Solution:** Skin cut the parts leaving a .020" skin on the bottom of the part, and then sand them free with a widebelt sander. (Check with your Widebelt Sander manufacturer for compatibility of your machine to do this.) Another

solution can be to leave “Tabs” on the part similar to skin cutting except on the majority of the cut you cut all the way through but at regular intervals you leave tabs that can either be broken away at the end or sanded away. This works well on extremely small parts that have a tendency to kick and get damaged when the cut is completed.

- Problem: Inability of the machine to run the part with any significant speed thus burning the tools up prematurely.
- *Solution:* Slow the spindle speed RPM to increase the chip size thereby reducing heat in the tool.

Tool Hazards

- Problem: Routing out inside pockets such as grommet holes where plugs are generated allowing them to kick when cut free causing possible injury to both operator and tool.
- *Solution:* Re program machine to generate the circle or shape in a corkscrew manner thereby chewing up the plug into dust. This doesn't add much to the cycle time but will add much to the longevity of the tool and operator!
- Problem: Routing out parts such as circles from squares where the corners are kicked across the room when cut free, causing possible injury to both operator and tool!
- *Solution:* Redesign the spoilboard to either allow the part to be raised off the table so that the off-fall pieces drop free and clear of the tool or, if the pieces are big enough, create a vacuum area and hold them down! Either alternative will increase speeds run as well as reduce rejects caused when the tool deflects into the part and making a bad part.

These factors all need to be combined into a rational decision making process which must also be utilized with a little discretion and trial and error with tool testing in the actual material to be cut. *There is no substitute for actually making chips on the machine!* This goes not only for CNC routers, but as well as for hand-fed routers as well due to differences in operator preferences concerning feedability. The aggressiveness of the tool must be married with a tool that allows a quality part edge finish to be achieved. Otherwise, the operator is allowed to run a tool that quality control may not accept. Some operators don't want a tool to pull itself through the material quickly, yet others insist on something that's easy to feed and often reduces incidence of carpal tunnel syndrome. This is becoming a very big factor as well as in tool selection consideration, particularly in hand routing applications.

REMEMBER, TO MAXIMIZE TOOL LIFE, ALWAYS PUSH THE TOOLS AS FAST AS YOU CAN TO THE FOLLOWING DEFAULTS:

- 1. The edge finish deteriorates.**
- 2. The machine defaults to a slower speed to execute the part accurately.**
- 3. The part is pushed off the vacuum.**
- 4. The tool breaks.**

Note: If item 2 occurs before item 1 then reduce the RPM's by 10% increments until items 1, 3, or 4 happen.

If the tools are run to the above defaults, you will not only get more parts from the tool, but you will have achieved more productivity from your routing operations!

TROUBLE SHOOTING TOOL BREAKAGE PROBLEMS

In the unlikely event that you encounter a breakage problem, here are some things that may help you identify the problem. The tool is the weakest part of the routing system. When it breaks, it is the first thing that is suspect, which not always true. 80% of all router breakage can be attributed to three factors; improper tool selection, poorly maintained or worn equipment, and or poor fixturing.

Poorly maintained equipment includes, collets, collet nuts, spindle bearings, spindles, spindle housings, slides and head control items.

The first thing that should be done is to change the batch of the tool *if* you have a different batch number on hand. Onsrud Cutter batch numbers every tool we make. If your tooling doesn't have this information on it, you have no way of isolating a bad batch of tools. If the problem goes away then the tool is suspect. Return a sample of these, preferably a broken one, *and* a new one from that batch via your distributor. If the problem still exists after changing the batch, then something else is suspect.

Examine the shank of the tools for poor collet contact or "collet burn". This will show as brownish, burned marks on HSS and Carbide tipped tools. The same will appear to be shiny marks on Solid Carbide tools.

The weakest part of the tool is where the primary clearance angle stops. If a tool is a smaller diameter cutting edge than the shank diameter, i.e., 5/16" on a 1/2" shank, and the shank is broken, then the collet is definitely the main suspect. Try a tool with a straight through shank I.E. 3/8" on a 3/8" shank.

The following items require that the machine be off and the operating items shutdown for safety reasons. Please be careful of all machinery.

A quick check of the slides, bearings and head mountings for play or for looseness can be made by grabbing the spindle by hand and pushing and pulling it. This will show most unacceptable conditions. The full procedure for spindle runout is in

the back of the Onsrud Cutter catalog. This will reduce the variables in trouble shooting the problems.

POOR TOOL LIFE

The first factor to check is making sure you have the proper tool for the application. The second item is that the speeds and feeds must be correct in order to maintain proper chiploads. The third item is the condition of the equipment. The last item would be the fixturing or spoilboard designs allowing for rigidity and the ability to clear chip from the work.

POOR PART FINISH

All of the same factors in the above section need to be checked plus these additional ones. Check the feed direction, it should be conventional cutting for most applications. Most times if the proper tool geometry is employed, climb cutting is not recommended, except in solid wood where grain allowances must be considered. Check for dull tools, sometimes tool life is short in some materials. Check for burrs on the cutting edge. Some materials will run faster and cleaner by using a roughing and finishing pass. One example is expanded PVC. Another is solid wood, where chipbreaker tools can be complimented by a finishing tool pass producing superior quality parts finish in about the same amount of cycle time as with one tool alone, provided the spindle stations are available to do so.

ROUTER BIT DESIGNS ARE A BALANCE OF VARIABLES ASSOCIATED WITH THE FOLLOWING

- Material Cutting Characteristics
- Machine Characteristics
- Methods To Be Used
- Tool Strengths & Weaknesses

Work Piece Material Cutting Characteristics

- Machining properties of the work piece materials
- Surface quality required on the cut surfaces
- Grain directions in the materials
- Spring back of the materials during the cut
- How do the materials react (wear) with the tool materials?

RAKE, CLEARANCE, AND FLUTE DESIGN ARE THE MAIN GEOMETRY ADJUSTMENTS TO MATCH THE ABOVE REQUIREMENTS.

Machine Characteristics

- Brand name & style of machine - hand electric router, hand air router, pin router, CNC router, other?
- Horsepower - limitations?
- Rigidity of system - machine and clamped part?
- Clamping System - Vacuum or physical clamps?

Methods To Be Used

- Pattern of the cut - sharp corners? Long straight sections? Small radii?
- Grain directions in the material affecting finish
- Treatment of the material - i.e., annealed versus heat treated
- Climb, conventional, or both cutting directions
- Speed & Feed of cut

Tool Strengths & Weaknesses

- Limitations of the tool material relative to the geometry of the cutting edge, general size or pattern of the bit?
- Standard rule of thumb is that no cutting edge length should exceed 3 times the cutting edge diameter. We often make tools with 4:1 ratios 1/2" Diameter X 2" Cutting Edge Length
- Manufacturing limitations making the tool?
- What are the economics of the tool for the application?
- What is the reliability of the material for the application?

Trouble Shooting Pointers

<u>PROBLEM</u>	<u>CAUSE</u>	<u>SOLUTION</u>
<i>TOOL BREAKAGE</i>	Excessive Cutting Edge Length	<i>Use shortest CEL to achieve depth of cut</i>
	Improper Colleting	<i>Collet on smooth surface of shank at the end of the flute fade out</i>
	Poor Maintenance of Collet, Nut & Spindle	<i>Identify proper procedure for cleaning collet, nut & spindle</i> <i>Change collets on a regular basis</i>
	Cutting Edge Diameter Less Than Shank Diameter	<i>Go to straight through tool with CED & shank the same</i>
	Use of Adapter Bushings	<i>Utilize proper collet size</i>
	Machine Problems	<i>Check collet & spindle runout</i> <i>Check for play in slides, bearings or head mountings</i>
	Part Movement	<i>Check vacuum hold down & clamping devices to insure part rigidity</i>
	New or Inexperienced Operator	<i>Part should be fed smoothly to allow router tool to cut freely</i>
	Poor Tool Selection	<i>Match tool with material being machined</i>
	Manufacturer Problem	<i>Have manufacturer do quality check</i>
<i>TOOL LIFE</i>	Poor Tool Selection	<i>Match tool with material being machined</i>

	Speeds & Feeds	<i>Make proper changes to improve chipload & heat removal</i>
TOOL LIFE	Poor Dust Collection	<i>Suggest improvements in dust collection capability to remove chips & heat</i>
	Part Movement	<i>Check vacuum hold down & clamping devices</i> <i>Make chips whenever possible</i>
	Machine Problems	<i>Check condition of collets, collet nuts, spindles, slides, bearings and head mountings</i>
	Material Being Machined	<i>Maximize conditions for best results</i>
PART FINISH	Dull Tools	<i>Check for edge deterioration & replace with new or resharpened tools</i>
	Tool Selection	<i>Double edge or multi-edge tools provide better finish</i> <i>Use shortest CEL available to make necessary depth of cut</i> <i>Use keener edged HSS tools on natural wood & some plastics to improve finish</i> <i>Use up/down compression spirals to improve top & bottom finishes on veneered or laminated materials</i>
	Feed Direction	<i>Should be conventional for most applications</i>

ROUTING ALUMINUM

TOOL SELECTION

Most aluminum can be cut with the high speed spiral tools. Hand operator air routers use 40-000 series downcuts and 49-001 series tools. In some rare occasions, a solid carbide multiple flute tool, such as 59 or 54-000 series.

In the sign industry, they usually have electric hand routers, in these the 40-000 series upcuts are used. This will pull the material up to the router base plate making a stable cut.

Pin Routers have a path for the chips to flow to and need a down force to hold the part to the pattern. It is suggested to use a 40-000 series single edge downcut in these applications.

Broken arm routers need single edge upcut high speed steel 40-000 series to clear the chips from the part and patterns.

CNC Routers are used in three basic ways; single sheet, stacked sheets with screw hold down, and stacked sheets with headless rivets. In the single sheet application, the aluminum sheet is held via vacuum through a particle board or MDF material. *This base material is too abrasive to use high speed steel tools.* First choice is an 81-000 series solid carbide double edge Lo-Helix upcut tools. Second choice is a 57-200 series solid carbide double edge downcut tool.

Stacked sheet, screw hold down should use 40-000 series HSS single edge upcut. Second choice 40-000 series HSS double edge upcut. Third choice is 81-000 series HSS double edge Helix upcut.

Stacked sheets with headless rivet hold down should use 81-000 series, second 81-000, third 40-000 series double edge upcut.

Hand fed routing should always be done in a continuous cut direction for operator safety. In some cases, two passes should be made on parts to yield the best edge finish. In using a two pass (rough/finish) it is necessary to leave .060" to .100" excess to clean up on the finish pass. Use a 40-000 series to rough cut and a 54-000 series to finish cut.

CNC routing techniques are unique to each machine. It is suggested that all parts be cut in a conventional manner, however, some machines will give better part finishes going in a climb cut. An examination of both the part and its scrap will show which works best on the machine. If two passes are used to correct tool deflection, guarantee part size, etc., then use climb cuts to finish the parts. Leaving less than half of the cutter diameter to cut.

COOLANTS

On all CNC routers, coolants should be used. Synthetic or soluble spray mists should be mixed at high water/coolant levels. The main objective is cooling, not lubricating. Flood coolants run the best in deep cuts, such as stacked sheet routing.

In hand router applications, where it is not feasible to use either mist or flood coolant, use bees' wax or bar soap. It can be applied to the part and/or the tool can be "dipped" into it before cutting.

PROBLEMS

Soft Aluminum-Marked as "O" Condition

Use single flute tools and the combination or "O" geometry CNC tools. In some cases an "SO" grind version of the standard 40-000 series tool is required as a special. Contact manager/sales for quotes.

Chip Welding

This can be caused by several factors, such as dull tools, no coolant, wrong tool type, wrong feed direction on CNC routers, poor chip loads, spindle run-out, etc.

Tool Deflection

High speed tools under 1/2" diameter will have some deflection in cuts deeper than 1/8". Double passes or reduced feed rates will control most of this problem. A change to a solid carbide tool, such as 81-100 series will eliminate most of this in a single pass, but do not perform well in soft or "O" condition.

Routing Plastic

Routing plastics material often presents a problem to many CNC router users. Plastic comes in many varieties much more diverse than any other routed medium. This is due to the different processes in formulations as well as colorants and how they can adversely effect the cutting tool material. Some materials such as PVC actually emit a chemical gas when cut, which chemically attacks the cobalt binder in the carbide tools. Such unique characteristics and the flexible nature of the materials always poses unusual problems.

Frequently, we recommend straight cutting tools for plastics so as not to adversely influence the part with a lifting or repelling action as with a spiral tool. By using straight tools we tend to have a neutral cutting action which doesn't cause problems with parts rattling while being cut. This is particularly true when cutting many thermoformed parts on 5 Axis machines where parts are extremely complex in shape and often present problems in fixturing. Plastics also have a unique characteristic, similar to soft aluminum, in that they have the ability to "reweld" or have the chips stick back together after the parts are cut presenting a real problem. This can often be eliminated the same way we eliminate it in aluminum applications and that is to use a single edge "O" fluted tool which has a lot of rake so the chips get thrown far enough away from the cut, that they don't reweld. The finish produced is usually much better than a multifluted tool as well. "O" fluted tools have a unique ability to "peel" a chip rather than snapping a chip like the violent action taking place with a "V" or "Z" fluted tool which abruptly snaps the chip. This non-violent cutting action can produce a much better edge in softer plastics than any other style often resulting in significant noise reduction as well. "O" fluted tools come in both single and double edge varieties is HSS 11-00 series or Solid Carbide 56-003P and 61-000P series.

When cutting the harder material it is better to use a "V" or "Z" fluted design to break the chips clean and there again expel them from the cut before they reweld. Rewelding is not as much of a problem here but with any plastic the sooner you rid the chips the better the finish will be. Recently we have found that our 3 Edge Finisher Series 60-200 series have achieved outstanding edge finishes cutting all types of acrylic. Although these tools are a spiral design they are mainly used in three axis applications where unidirectional chipflow is not a problem.

When routing plastics the number one thing to keep in mind is to *cut chips and not dust. Dust gets rewelded, chips don't.* Dust, is chips cut twice. Always run the router at a feed speed as fast as possible. Often times it is not necessary, in fact, it is detrimental to run the spindle speeds at high rpm's. Often the spindle can be slowed to 12,000-14,000rpm at good feed rates with excellent results. With some materials such as polypropylene the rpm should be slowed to 8,000-10,000 to get the proper chip size required to prevent rewelding.

Should you have any questions that you would like to discuss with our field sales, or engineering department please do not hesitate to contact Onsrud Cutter 800 Liberty Drive, Libertyville, IL 60048 at 800-234-1560 and we will be more than happy to assist your in further accessing your router tooling requirements.