MACHINING METHODS

CLIMB & CONVENTIONAL MILLING

There are drastic differences between climb milling and conventional milling which produce dramatically different results. Understanding the differences is key to extending tool life, promoting quality and optimizing machine time utilization. Desired speed, finish, material, chip clearing, shear direction, and end mill construction are just a few things to consider when deciding on your choice of milling method. Regardless of your preferred method, your workpiece should be braced sufficiently in the direction you are milling.

CONVENTIONAL MILLING

Conventional milling requires lower forces and is preferred for roughing cuts. The cutter is revolving in the opposite direction as the table feed and the workpiece is fed into the rotation of the cutter. The width of the chip increases to a maximum at the end of the cut, advancing tool wear.

Characteristics of Conventional Milling:

- Conventional milling is preferred for rough, abrasive surfaces when removing or breaking through material scale, welded, work hardened or flame cut areas.
- Increased rubbing, harmonics, work hardening and premature tool wear
- The tooth meets the workpiece at the bottom of the cut
- Produces upward force on part, increasing part movement
- More torque is required to conventional mill than climb mill
- Surface finish is worse because chips are carried upward by teeth and dropped in front of cutter
- The width of the chip starts from zero and increases to the maximum width of the cut
- Tool deflection during Conventional milling will tend to be parallel to the cut

CLIMB MILLING

Climb milling produces excellent surface finishes and works best in most cases. The cutter is revolving in the same direction as the table feed, meeting the workpiece at maximum thickness, producing the largest chips first. When cutting in the direction of the table feed and rotation of the cutter combine, the mill will try to draw away from the work.

Characteristics of climb milling:

- Desired method for high performance solid carbide cutters
- Increased surface finish; decreased rubbing and work hardening; up to 50% increased tool life
- The tooth meets the workpiece at the top of the cut
- Produces downward force on part, decreasing part movement
- Less torque required to climb mill than conventional mill
- Higher initial spindle load and increased spindle load as end mill dulls
- Helps to prolong tool life, tools lasting up to 50% longer
- Chips are dropped behind the cutter (less re-cutting)
- The width of the chip starts at maximum at the maximum width of the cut and decreases to zero
- Tool deflection during climb milling will tend to be perpendicular to the cut, so it may increase or decrease the width of cut and affect accuracy

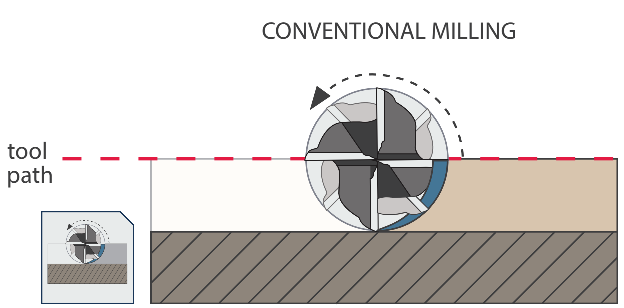


figure 1

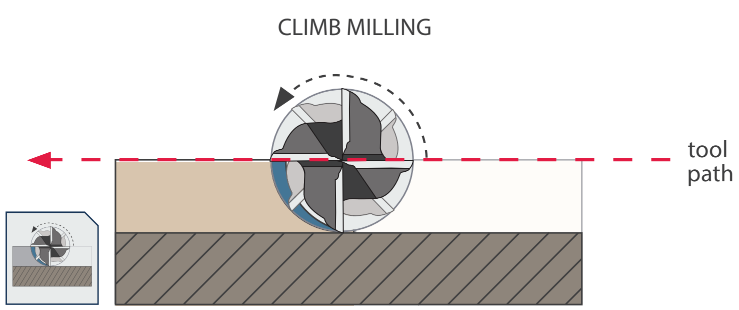


figure 2

ENGAGEMENT ANGLE

ENHANCING TOOL LIFE & MACHINE PERFORMANCE

The angular measurement of the cutter in which the contact between the tool and the workpiece occurs is referred to as the Tool Engagement Angle ("TEA"). Radial chip thickness is directly connected to the angle of engagement and increasing the axial depth of cut increases the tool engagement angle considerably.

When contouring (see figure 1), the tool engagement angle varies dramatically along a curved cut. As the tool approaches an inside corner (see page 51 for additional technical information), its engagement angle is increased dramatically and therefore its radial chip thickness is as well. This dramatic and quick increase in chip load per tooth can cause spikes in spindle load and horsepower requirements, a need to decrease the feed rate, increased tool deflection, lower tolerances, decreased surface finish and result in excess wear and tear on the cutter and machine.

As the tool engagement angle is decreased, either through a lower radial depth of cut or while cutting an outside corner, the stresses on the machine and tool are lessened. While decreased horsepower requirements, decreased tool deflection, tighter tolerances and improved finishes are all desirable, the programmed chip load per tooth may be too low and require an increase in feed rate (see page 50 for additional technical information) to avoid the tool from rubbing and prematurely wearing. This can present a perfect opportunity for high speed machining if the machine has high spindle speed capabilities.

CONTOURING AND CORNER ENGAGEMENT

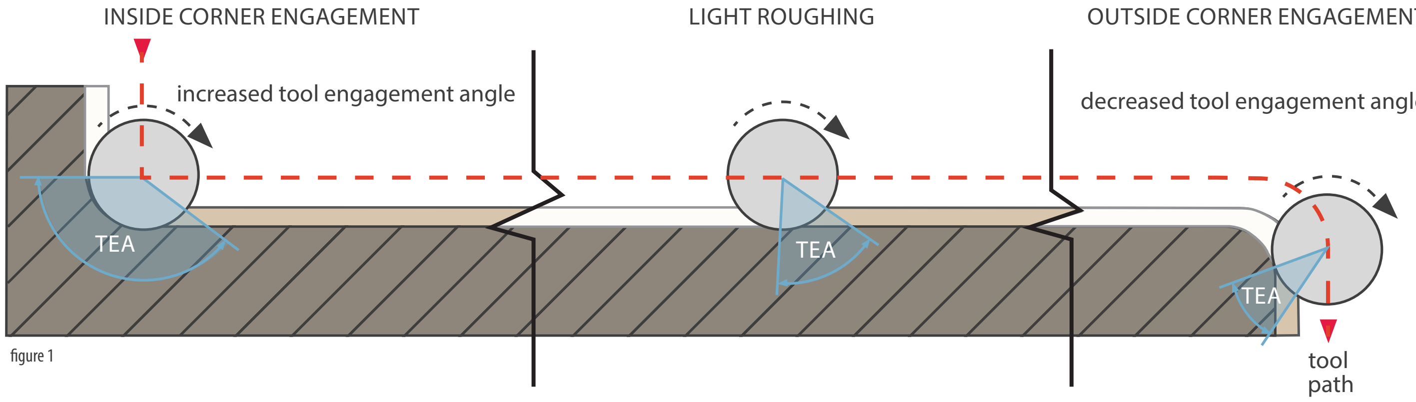


figure 1

ROUGHING AND FINISHING PASSES

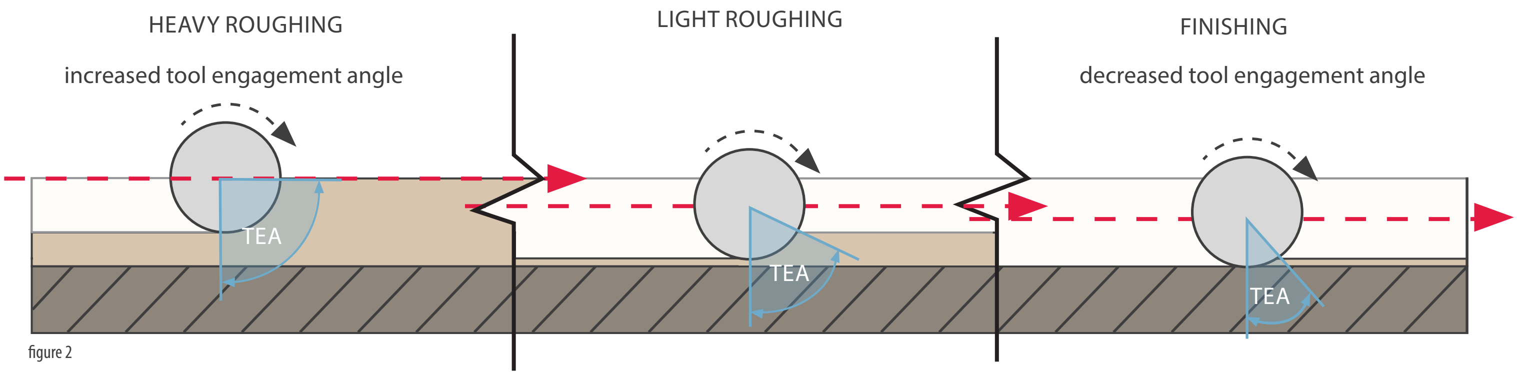


figure 2

CHIP THINNING

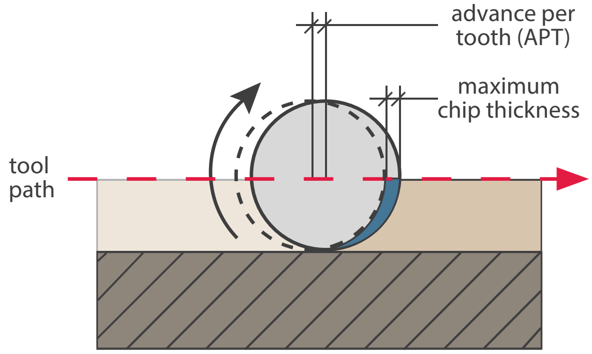
PROCEDURES AND CALCULATIONS FOR PROPER CHIP REMOVAL

A light radial depth of cut (less than half of the cutter diameter) causes the chip formation to be much thinner than the programmed feed rate. The end mill begins to rub, rather than cut, causing excessive tool wear by creating increased friction, work hardening and degrading the ability of cutting tool to transfer detrimental heat away from the tool and workpiece. This greatly diminishes and limits the cutting tool's performance in terms of chip load per tooth.

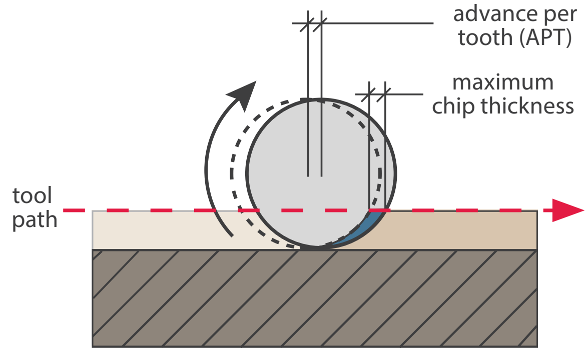
Many programs and speed and feed calculators show only the Advance Per Tooth (APT) and it is commonly used interchangeably with the Chip Load Per Tooth (CLPT). While taking a Radial Depth of Cut (RDOC) of 50% (see figure 1), the APT is the same as the CLPT which lends to the confusion. The APT is actually the measurement of forward feed that takes place in the time necessary for the cutter to rotate a single revolution, whereas the CLPT is the thickness of the chip produced. When the RDOC is equal to or greater than 50% of the diameter of the tool, the chip is thickest along the centerline of the tool, then decreases to zero as the cutting edge exits the material.

When programming a Radial Depth of Cut ("RDOC") less than half the tool diameter (see figure 2), use the calculation in Figure 5 to determine the Adjusted Chip Load Per Tooth ("ACLPT") to prolong tool life and lessen cycle time. If your width of cut is less than half the diameter of the cutter (unless otherwise listed on supplement feeds and speeds), your chip thickness is less than the programmed advance per tooth feed rate.

You also must consider the extent of the tool engagement when using this adjustment in feed rate. For instance, when milling into corners, the tool engagement angle ("TEA") increases dramatically and tool deflection and cutting forces are increased. Feed rate reductions in these areas may be required and will need consideration.



50% RADIAL DEPTH OF CUT (RDOC) figure 1



<50% RADIAL DEPTH OF CUT (RDOC) figure 2

figure 3

ADJUSTED CHIP LOAD PER TOOTH CALCULATION

SYMBOL	EQUATION
Adjusted CLPT =	$\frac{CLPT \times (D/2)}{\sqrt{(D*RDOC) - RDOC^2}}$

figure 5

ACTUAL CHIP LOAD PER TOOTH CALCULATION

SYMBOL	EQUATION
Actual CLPT =	$\left(\frac{(D/2)}{RDOC}\right)^2 \times CLPT$

figure 4

ADVANCE PER TOOTH CALCULATION

SYMBOL	EQUATION
APT =	$\frac{IPM}{\sqrt{RPM \times T}}$

KEY

SYMBOL	ELEMENT
APT =	Advance Per Tooth
IPM =	Inches Per Minute (Feed Rate)
RPM =	Revolutions Per Minute (Spindle Speed)
T =	Number of Teeth
CLPT =	Chip Load Per Tooth
D =	Diameter of Cutting Tool
RDOC =	Radial Depth of Cut

CONICAL™ CUTTING TOOLS

GLOBAL™ CUTTING TOOLS

CORNER ENGAGEMENT

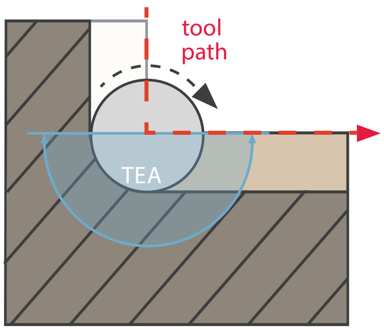
CREATING QUALITY CORNERS AND INCREASING PERFORMANCE

Improperly programmed tool paths can create a wide spectrum in spindle torque variations and result in uncontrolled parameters and premature tool wear. Traditional conservative programming results in lower productivity and simultaneously increases tool wear by causing chip thinning. Alternative programming may cause the tool's engagement angle to increase significantly, resulting in a spike in cutting forces which can weaken performance and lead to breakage. When milling inside corners, cutting forces are increased dramatically and unacceptable conditions may be apparent.

Indicators of a difficult to machine area:

- Chatter – Visible: finish level is noticeable worse
- Deflection – Measurable: taper increases along wall
- Sound – Audible: squawking or chirping when cutter is engaged
- Tool Breakage – Visible: chipping forms near the end of the tool,flutes are stripped or tool breaks

TRADITIONAL PROGRAMMING – NOT RECOMMENDED FOR MOST SCENARIOS



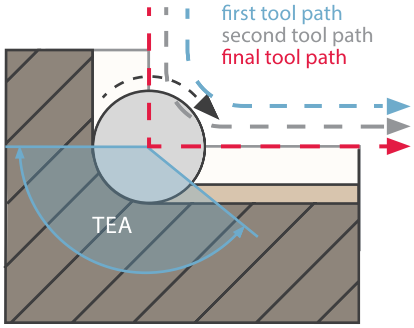
HEAVY ROUGHING
50% RADIAL DEPTH OF CUT

Match an end mill radius to that of the inside corner being machined and execute a 90º turn in cut direction. This increases the tool engagement angle to nearly 180º at a 50% RDOC, resulting in significant additional cutting forces, increased likelihood for chatter, tool deflection, breakage and ultimately poor surface finish.

Acceptable Scenario: This method should only be used when slotting or pocketing and clearance is an issue.

Programming Considerations: If employing a 90º turn in cut direction, feed rate will need to be lessened.

IMPROVED PROGRAMMING – GENERALLY ACCEPTABLE IN MANY SCENARIOS



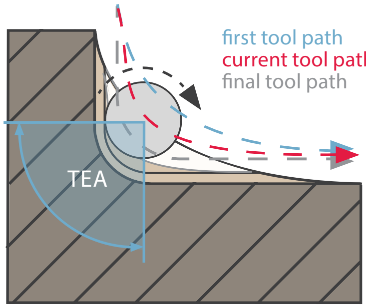
LIGHT ROUGHING
<50% RADIAL DEPTH OF CUT

Cutting in a sweeping direction that matches the radius of the tool reduces the tool engagement However, the final cut will have a drastic engagement angle which results in less than optimal machining. Again, chatter, deflection, poor surface finish and breakage can all occur. Utilizing this method will also require a reduction in feed rate on the final pass due to the increased tool engagement angle

Acceptable Scenario: When machining without tool changes and programming using the existing tool.

Programming Considerations: A smaller Radial Depth of Cut ("RDOC") will have to utilized and feed rate lessened on the each pass as the engagement angle increases to create the desirable surface finish.

OPTIMIZED PROGRAMMING – GENERALLY ACCEPTABLE IN MOST SCENARIOS



LIGHT ROUGHING
<50% RADIAL DEPTH OF CUT

Combining a smaller end mill and larger, sweeping radial tool path is the optimal condition for corner engagement. The tool engagement angle varies less and becomes much more manageable with smaller tool engagement, thus allowing for higher speeds and feeds. The engagement angle will still increase at the full depth of cut, but feed reduction will be minimized. Furthermore, surface finishes are improved and end mill life is prolonged.

Acceptable Scenario: In most scenarios where adequate room exists for the returning tool path.

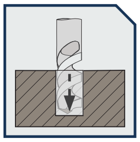
Programming Considerations: Feed rate may need to be heightened to eliminate chip thinning due to a less than 90º tool engagement angle.

TOOL ENTRY METHODS

APPROACHES & PROCESSES

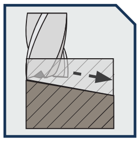
Tool entry is one of the most imperative operations to the performance of the tool and can have the most effect on a tool's life. Listed below are some conventional methods of tool entry, as well as tips on how to optimize performance.

TOP ENTRY



Pre- Drilling

Pre-drilling a hole slightly larger than the end mill diameter to full cutting depth is the best way of entering your end mill into a pocket. This creates the least amount of excessive end wear and reduces tool stress.

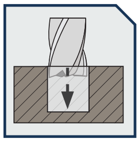


Ramping In

Ramping gradually increases the depth while moving the cutter in a linear path. There are multiple variations on ramping, some follow the contour of the pocket and not necessarily a straight line. In others, referred to as zig zag, the cutter moves back and forth in a straight line, at each pass increasing its depth.

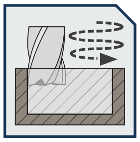
This method can be very advantageous but exerts various cutting forces that the tool must endure. Proper chip size, evacuation and core strength are crucial to minimizing wear and built up edge. Utilizing a corner radius will reduce corner wear on the most fragile part of the tool.

General guidelines for ramp angles:
Ferrous Materials 1 - 3˚
Non-Ferrous Materials 3 - 10˚



Straight Plunge

Plunging can easily break an end mill and requires a center cutting tool. Therefore, this is the least favorable method of tool entry. Feed rate is typically a fraction of a straight linear feed rate. Drills are intended for straight plunging and should be used instead of an end mill. End milling utilizes a flat or concave entry point creating natural chip packing and making evacuation difficult. Cutting forces on the tool are extremely high and the stresses make performance unpredictable when executing this operation.

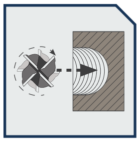


Helical Interpolation

Helical Interpolation is the process of using the end mill to define a helical motion, producing a circular hole, to the full cutting depth. End mills with a corner radius decrease tool wear and corner breakdown. Tool engagement angle is consistent and cutting forces are reduced by the end mill's own tool path. A programmed helix between 115-130% of the cutter size is suggested for optimal performance.

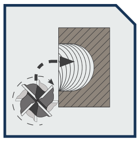
SIDE ENTRY

Use a corner radius for optimal performance



Straight Entry

A linear entry using the side of the end mill to enter the workpiece. This method is much harder on the end mill and makes it more susceptible to wear and shorter tool life. The feed rate during entry must be cut in at least half and speed reduced at a similar rate, until the tool is completely engaged at its operating RDOC.



Roll in Entry

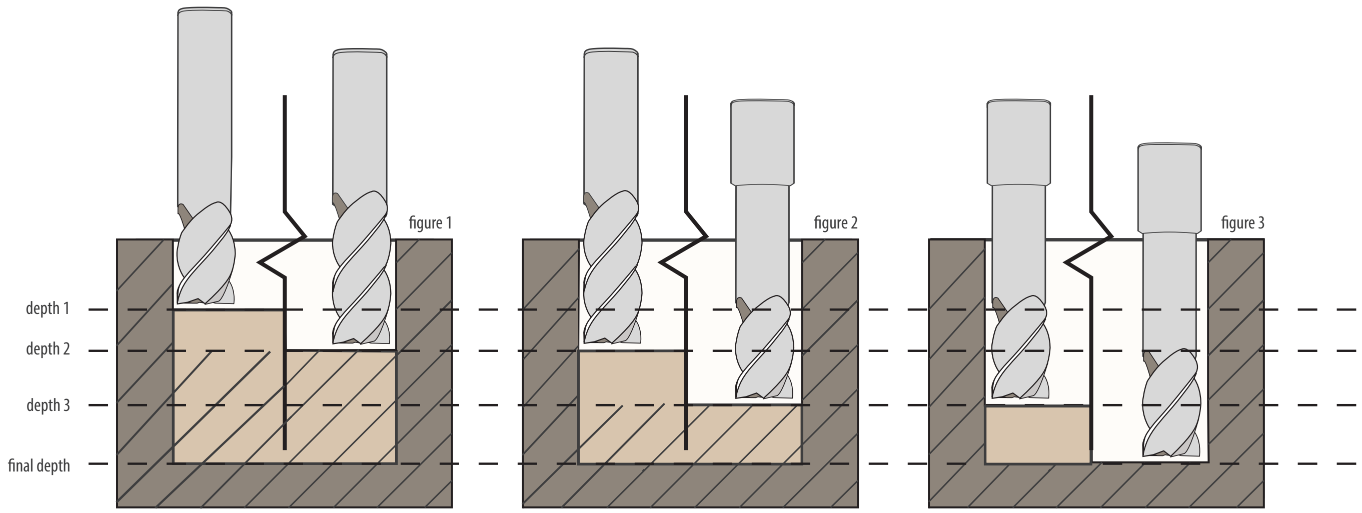
To execute a roll in entry, start the cutter out half the diameter to the right of the desired entry location. Then roll it along a path in an arched direction, with the same radius as the cutter. Rolling into the cut inherently generates proper chip thickness and yields complete engagement. The feed rate should be cut in half until the tool is fully engaged.

MATERIAL REMOVAL

DEEP POCKET MILLING

Removing material deep in a pocket is consistently one of the most challenging operations. Chip packing can occur due to poor chip evacuation, coolant flooding is not an option and air pressure may be inadequate to remove chips from the pocket. Without chip evacuation, the existing chips are recut. It may be required to periodically halt operations to clear chips and pooled coolant. To make matters worse, long flute length and overall length tools tend to deflect causing chatter, wall taper, reduced finishes, chip thinning and potential breakage.

In order to optimize speeds and feeds, employ a step down method to maintain a consistent axial depth, while using the largest diameter cutter possible. Utilize a stub length or regular length tool (figure 1) to get to at least 2 to 3 times the diameter of the cutting tool in depth. Using a stub or standard length tool will allow you to create a higher metal removal rate in the beginning steps of the pocket, reducing the overall machine time. Once this is achieved, change tools to a short flute length, reduced neck, extended reach tool. (figure 2)



BEGIN WITH REGULAR OR SHORT TOOL

EMPLOY MEDIUM LENGTH TOOL AND REDUCE NECK

EMPLOY REDUCED NECK TO AVOID WALL

Extended reach tools are much stronger than standard or length tools due to a shorter length of cut. They can maintain higher feeds and speeds without exposing the tool to the wear and deflection a standard tool would be subjected to. This is in part due the neck diameter being smaller than the cutting diameter, which allows for more clearance and a shorter flute length, strengthening and extending the core. If possible, a high speed machining technique should be used, increasing the spindle speed and feed rates while taking light cuts. Implementing this milling procedure will ensure maximum efficiency and the least tool wear while actually increasing the metal removal rate.

Resist the desire to reduce the feed per tooth and radial depth of cut to the point of generating thin chips. If less than half the tools diameter is engaged in cutting, the chips will be thinner than calculated and excess heat and pressure will be created in the cut. Use the Adjusted Chip Load Per Tooth calculation on page 35 to compensate.

Do not use conventional endmills with weldon flats and holders with setscrews. They pin the tool to a single side of the holder, pushing the tool between .0001 and .0005” off center. As the length of tool extended from the holder increases, the total indicated runout compounds, increasing chatter, deflection and poor surface finish.

MATERIAL REMOVAL

THIN WALL MILLING

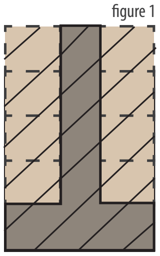
Creating thin walls while holding part tolerance and finish, requires careful programming and expertise. The force generated by metal removal along a thin wall's relatively weak structure, often creates a reverse taper along the wall, tolerance issues and surface finish problems.

Vibration and chatter must be controlled by harmoniously marrying the toolholder, cutter, material and tool path. Assuming the workpiece and table has been properly secured and is rigid enough for the operation, take care in selecting the proper shrink-fit collet holder and indicate the cutter to minimize any runout. Ensure the machine selected for the milling does not have excess spindle wear which will contribute to total indicated runout at the cutting edge.

Large core, rigid cutters work best for thin wall milling. Avoid tooling with a long overall length and a long length of cut when progressing into the pocket to minimize deflection, chatter and breakage. Just as with deep pocket milling, so long as adequate clearance exists, the largest diameter tool possible should be used. After reaching a depth of 2-3 times the diameter of the tool being used, the regular or stub length tooling should be replaced with a short flute length, necked down, extended reach tool. If the material allows, a flute count and a higher helix, extended reach tool is the optimal selection as more of the tool is engaged in the cut. It's shear plane pulls up on the workpiece material more than a traditional helix end mill, which tends to push either the cutter or the wall away from the tool.

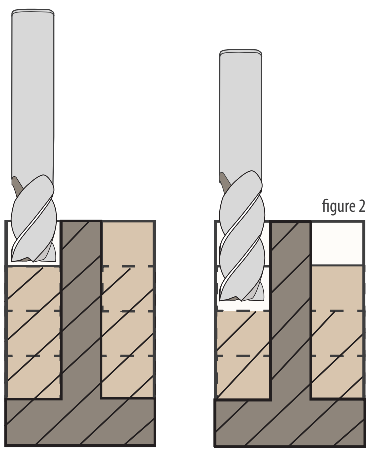
Climb milling will also assist in dampening vibration and eliminating chatter and should be used if possible. Because the rotational direction of the cutter is moving in the same direction as the part, it pulls the wall towards the cutter, rather than pushing it away from the cutter, using the cutter itself for stability in the cut.

figure 1



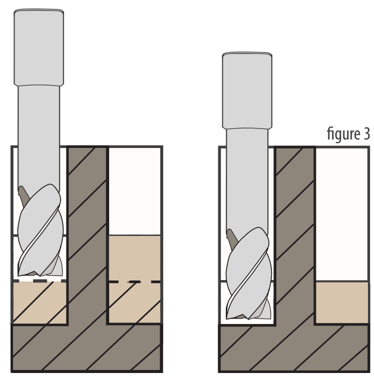
MATERIAL TO BE REMOVED

figure 2



EMPLOY REGULAR OR STUB LENGTH TOOL

figure 3

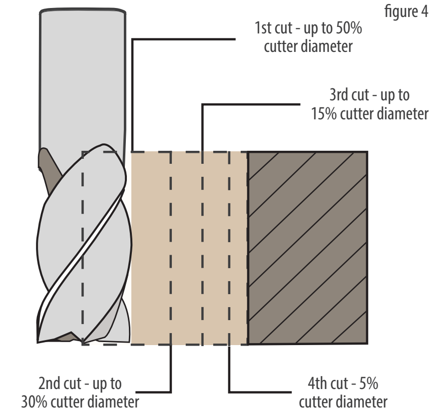


EMPLOY REDUCED NECK TO AVOID WALL

The cut should be segmented into equal segments (see figure 1) on both sides of the part, each with similar axial depths of cut. Beginning on one side of the wall, remove the material with a stub or standard length end mill, then alternate to the opposite side of the wall on each new pass. This leaves the wall supported from both sides throughout the cut and progresses in an incremental “stepped down” method. Upon reaching 2 times the cutter diameter, the tool should be changed to a reduced neck tool, as previously discussed, for the remainder of the cuts.

Depending upon the wall thickness and depth, a progressive radial depth of cut strategy may need to coincide with the above recommendations. This reduces the tool pressure against the wall after the opposite side’s support stock has been removed. After machining the opposing side, reduce the depth of cut as you approach the wall. Dependent upon the wall thickness and amount of stock to be removed adjacent to the wall, four to five passes should be implemented (see figure 2). The final pass may be an extremely light finishing pass, minimizing the vibration of the wall in its weakened form while maximizing surface finish.

figure 4



The objective in finishing is to eliminate or reduce final manual retouching and to achieve the desired dimensions, tolerances and surface finishes. There are many factors to consider when planning for finish passes. The material, workholding, toolholder, and cutter all contribute variables when programming an appropriate tool path.

Surface finish requirements vary from part to part. Finishing passes ensure accurate part measurement as well as create an aesthetically pleasing finish. Being aware of the many variables present and choosing the right procedures are vital to achieve the desired outcome.

Generally, using a cutting tool with a helix angle of 45 degrees or greater when the workpiece is aluminum and 38 degrees or higher for hardened or ferrous materials, will improve finish due to the greater shearing action of the cutting flutes. Simultaneously combining an increased helix and an increased number of flutes will improve tool engagement, minimize tool deflection, maintain dimensional accuracy and maximize the surface finish. Selecting a tool with an odd number of flutes staggers the entering and exiting of the flutes and contributes to smoother machining.

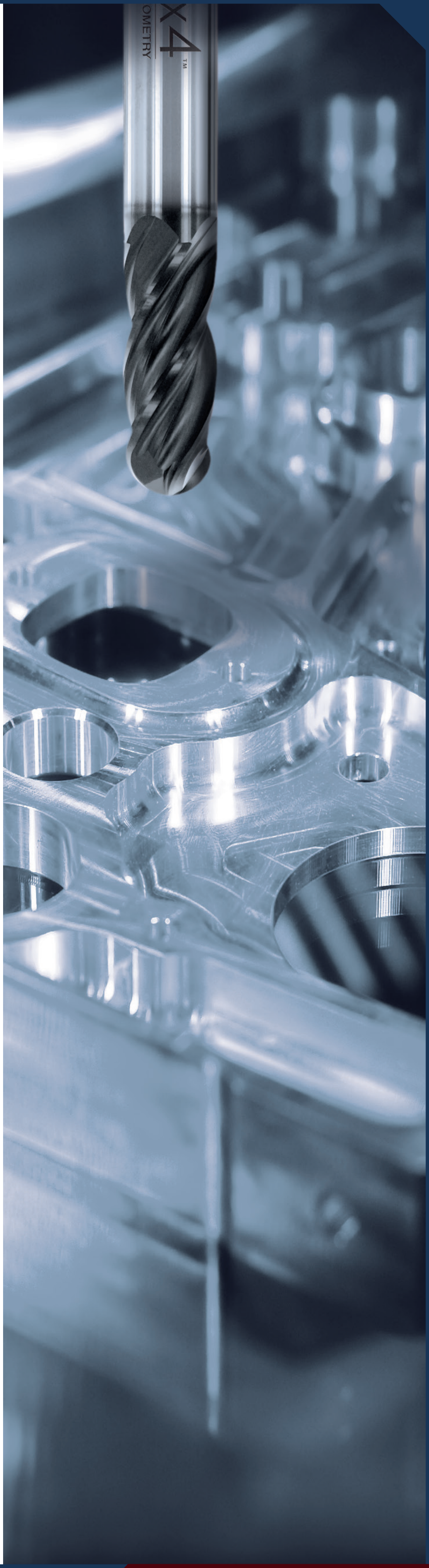
Be sure to use precision tool holders to minimize runout and cut with multiple progressively shallower radial depths of cut. A single pass maximizes cutter deflection and restricts chip evacuation, making surface finish harder to control.

Use climb milling whenever possible to create the best surface finish and dimensional accuracy. If the finishing depth is greater than two times the diameter of the tool, use reduced neck tooling to maintain stability in the cut while eliminating any rubbing that may occur from the shank. The Axial Depth of Cut (ADOC) should be approximately 75% of the tools length of cut, progressing at equal incremental passes to allow the top 25% of the tool's flutes to blend the radius at the bottom of the last cut with the top of the current cut. When finishing an existing hole, use an end mill with a slightly smaller diameter than the finished hole dimensions and circular interpolate the cutting path.

To maximize your cutters tool life, you may want to downgrade your visibly worn tools and use them in roughing operations only.

Further suggestions:

- The Radial Depth of Cut (RDOC) Should be between 1.5% and 5% of the cutter diameter
- Increasing the RPMs and decreasing the feed per tooth will improve surface finishes
- For walls greater than two times the diameter of the tool, use long reach end mills
- Advanced geometry cutting tools will dampen chatter and increase part finish



SURFACE ROUGHNESS

DEFINITIONS & CALCULATIONS

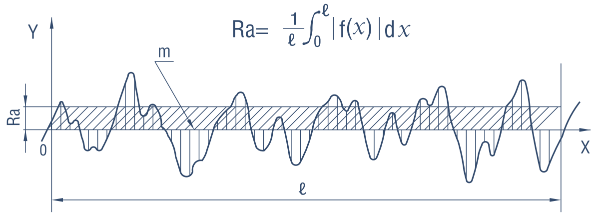
Achieving the required surface finish is generally the last step in production. Level of finish is specified for functional, dimensional and aesthetic reasons and has varying methods of measurement. The measurement of surface roughness is a mathematical equation, for a randomly sampled area, expressed as a constant or range.

TYPICAL WAYS FOR OBTAINING SURFACE ROUGHNESS

ARITHMETICAL MEAN ROUGHNESS (RA)

A section of standard length is sampled from the mean line on the roughness chart. The mean line is laid on a Cartesian coordinate system where in the mean line runs in the direction of the x-axis and magnification is the y-axis. The value obtained with the formula on the right is expressed in micrometer (µm) when y=f (x).

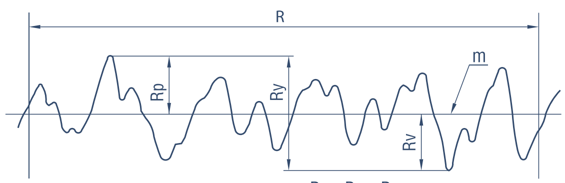
$$Ra = \frac{1}{\ell}\int_0^{\ell} |f(x)|\,dx$$



MAXIMUM PEAK (RY)

A section of standard length is sampled from the mean line on the roughness chart. The distance between the peaks and valleys of the sampled line is measured in the y direction. The value is expressed in micrometer (µm).

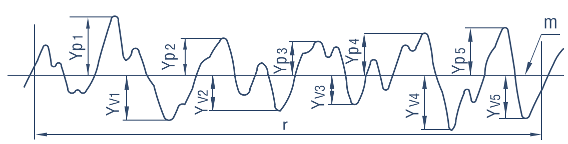
Note: To obtain Ry, sample only the standard length. The part, where peaks and valleys are wide enough to be interpreted as scratches, should be avoided.



$$Ry = Rp + Rv$$

TEN-POINT MEAN ROUGHNESS (RZ)

A section of standard length is sampled from the mean line on the roughness chart. The distance between the peaks and valleys of the sampled line is measured in the y direction. Then, the average peak is obtained among 5 tallest peaks (Yp), as is the average valley between 5 lowest valleys (Yv). The sum of these two values is expressed in micrometer (µm).



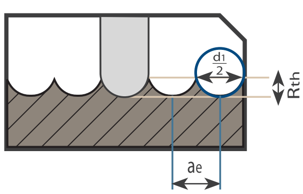
$$Rz = \frac{|Yp_1 + Yp_2 + Yp_3 + Yp_4 + Yp_5| + |Yv_1 + Yv_2 + Yv_3 + Yv_4 + Yv_5|}{5}$$

Yp_1 Yp_2 Yp_3 Yp_4 Yp_5: Tallest 5 peaks within sample

Yv_1 Yv_2 Yv_3 Yv_4 Yv_5: Lowest 5 peaks within sample

SPACING AND THEORETICAL SURFACE ROUGHNESS OF BALL NOSE END MILL:

The spacing "ae" of ball nose will be decided how the theoretical surface roughness you need, please use following information to decide "ae".



DESIGNATION			FORMULA
ae	Spacing	mm	$Rth = \frac{d_1}{2} - \sqrt{(d_1^2 - ae^2)/4}$
Rth	Theoretical surface roughness	mm	$ae = 2 \cdot \sqrt{Rth \cdot (d_1 - Rth)}$
d1	Ball nose diameter	mm	

RELATIONSHIP BETWEEN ARITHMETICAL MEAN ROUGHNESS(RA)AND CONVENTIONAL SYMBOLS

ARITHMETICAL MEAN ROUGHNESS (RA) – PREFERRED NUMBER SERIES	ARITHMETICAL MEAN ROUGHNESS (RA) – CUT-OFF VALUEC(MM)	ARITHMETICAL MEAN ROUGHNESS (RA) – INDICATION OF SURFACE TEXTURE ON DRAWINGS	MAX. HEIGHT (RY) – PREFERRED NUMBER SERIES	TEN POINT MEAN ROUGHNESS (RZ) – PREFERRED NUMBER SERIES	STANDARD LENGTH OF RY • RZ ℓ (MM)	TRIANGULAR INDICATION
0.012	0.08	0.012▽ ~ 0.2▽	0.05 s	0.05 s	0.08	▽▽▽▽
0.025	0.25		0.1 s	0.1 s		
0.05			0.2 s	0.2 s	0.25	
0.01	0.8		0.4 s	0.4 s		
0.2			0.8 s	0.8 s	0.8	
0.4		0.4▽ ~ 1.6▽	1.6 s	1.6 s		▽▽▽
0.8			3.2 s	3.2 s		
1.6			6.3 s	6.3 s		
3.2	2.5	3.2▽ ~ 6.3▽	12.5 s	12.5 s	2.5	▽▽
6.3			25 s	25 s		
12.5	8	12.5▽ ~ 25▽	50 s	50 s		▽
25			100 s	100 s	8	
50		50▽ ~ 100▽	200 s	200 s		~
100	-		400 s	400 s	-	

The above charts and graphs are excerpts from JIS B 0601 (1994) and JIS B 0031 (1994)

BALL NOSE APPLICATIONS

90º MACHINING TECHNIQUES AND SUGGESTIONS

BALL NOSE AT 90º INCLINE

Ball nose end mills are used to add a radius between perpendicular surfaces, reducing the concentration of stress. In addition, they are excellent for improved surface finishes and machining three dimensional contoured shapes, common in molds and dies. Follow the process below for optimum tool life and surface finishes when machining at 90° from the work piece.

Procedure for ball nose machining 90° (perpendicular) from the work piece

1. The effective cutting diameter (Deff)) should be calculated when using an Axial Depth of Cut (ADOC) that is less than half the diameter of ball nose end mill, or less than the full radius of the ball. Using the calculation in figure 4 will generate the effective cutting diameter of the ball end, when cutting at 90 degrees. If using a common axial depth of cut, you may be able to quickly determine the effective cutting diameter by using figure 3 of the chart below.

2. The machines RPMs will need to be adjusted to compensate for the smaller effective cutting diameter when using less than the full diameter of the tool. The velocity adjustment (Vadj) calculation in figure 5 will need the previously calculated effective cutting diameter (Deff) to determine the new RPMs.

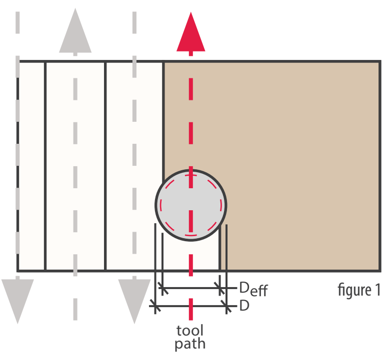


figure 1

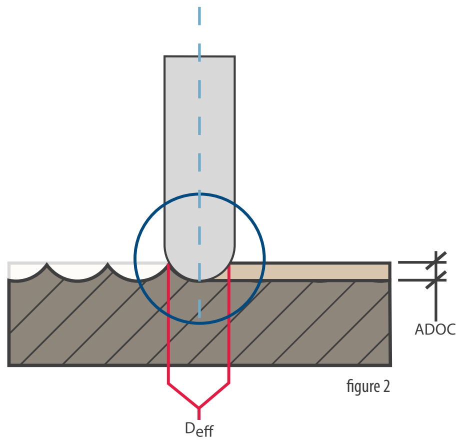


figure 2

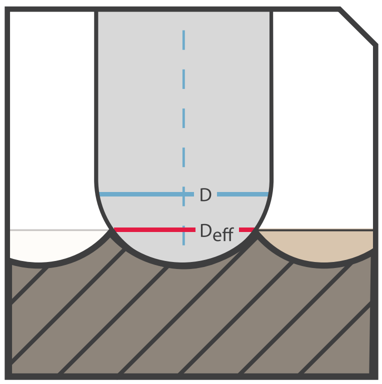


figure 2 detail

90° BALL NOSE EFFECTIVE CUTTING DIAMETER (D_{eff}) AT COMMON ADOC'S

CUTTER DIAMETER	AXIAL DEPTH OF CUT (ADOC)															
	0.010	0.020	0.030	0.050	0.070	0.090	0.100	0.125	0.150	0.175	0.210	0.250	0.300	0.375	0.400	0.500
1/8	0.068	0.092	0.107	0.122												
3/16	0.084	0.116	0.137	0.166	0.181	0.187										
1/4	0.098	0.136	0.162	0.200	0.224	0.240	0.245									
3/8	0.121	0.169	0.203	0.255	0.292	0.320	0.332	0.354	0..367	0.374						
1/2	0.140	0.196	0.237	0.300	0.347	0.384	0.400	0.433	0.458	0.477	0.494					
5/8	0.157	0.220	0.267	0.339	0.394	0.439	0.458	0.500	0.534	0.561	0.590	0.612	0.624			
3/4	0.172	0.242	0.294	0.374	0.436	0.487	0.510	0.559	0.600	0.634	0.673	0.707	0.735	0.750		
1	0.199	0.280	0.341	0.436	0.510	0.572	0.600	0.661	0.714	0.760	0.815	0.866	0.968	0.968	0.980	1.000

figure 3

KEY

SYMBOL	ELEMENT
ADOC	Axial Depth of Cut
D	Cutting Diameter
D_{eff}	Effective Cutting Diameter
R	Tool Radius (Dia. x 2)
SFM	Surface Feet per Minute
V_{adj}	Adjusted Revolutions per Minute

$$D_{eff} = 2 \times \sqrt{R^2 - (R - ADOC)^2}$$

figure 4

$$V_{adj} = \frac{SFM \times 3.82}{D_{eff}}$$

figure 5

BALL NOSE APPLICATIONS

15° INCLINE TECHNIQUES AND SUGGESTIONS

BALL NOSE AT 15° INCLINE

To avoid a zero surface feet per minute (SFM) at the center of the tool, ball nose tools should be used at a 15° incline. This strategy will increase tool life and surface finish. For maximum performance, it is highly recommended to use a climb milling technique and feed the tool in the direction of the incline. Follow the process below for optimum tool life and surface finishes when machining at a 15° incline from the work piece.

Procedure for ball nose machining at 15° from the work piece

1. Calculate the effective diameter using the calculation in figure 4 or if using a common axial depth of cut and diameter tool, by using figure 3. When using an angle other than 15°, you must use the calculation, rather than the chart and treat the angle of incline as a variable and substitute the programmed angle in its place.

2. The machines RPMs will need to be adjusted to compensate for the smaller effective cutting diameter when using less than the full diameter of the tool. The velocity adjustment (V_{adj}) calculation in figure 5 will need the previously calculated effective cutting diameter (D_{eff}) to determine the new RPMs.

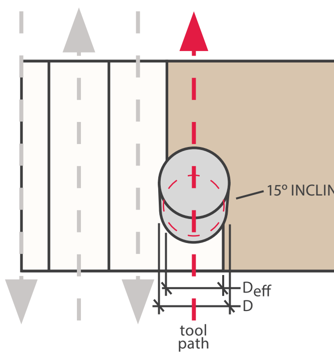


figure 1

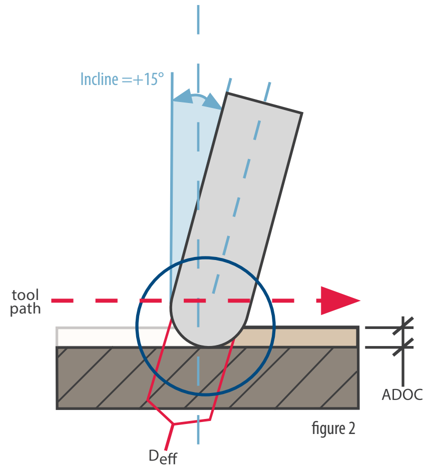


figure 2

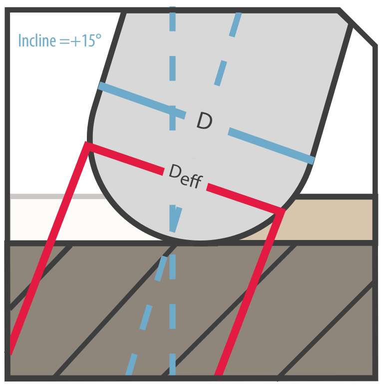


figure 2 detail

15° BALL NOSE EFFECTIVE CUTTING DIAMETER (D_{eff}) AT COMMON ADOC'S

CUTTER DIAMETER	AXIAL DEPTH OF CUT (ADOC)															
	0.010	0.020	0.030	0.050	0.070	0.090	0.100	0.125	0.150	0.175	0.210	0.250	0.300	0.375	0.400	0.500
1/8	0.093	0.111	0.120	0.125												
3/16	0.124	0.150	0.165	0.182	0.187											
1/4	0.154	0.185	0.206	0.232	0.245	0.250										
3/8	0.209	0.249	0.278	0.317	0.343	0.360	0.366	0.374								
1/2	0.259	0.308	0.343	0.393	0.428	0.454	0.464	0.483	0.494	0.500						
5/8	0.308	0.364	0.404	0.463	0.506	0.539	0.553	0.580	0.600	0.615	0.623	0.624				
3/4	0.355	0.417	0.463	0.530	0.579	0.618	0.635	0.669	0.696	0.720	0.736	0.748	0.749			
1	0.446	0.519	0.573	0.654	0.715	0.765	0.787	0.833	0.871	0.908	0.937	0.966	0.989	1.000		

figure 3

KEY

SYMBOL	ELEMENT
ADOC	Axial Depth of Cut
D	Cutting Diameter
D_{eff}	Effective Cutting Diameter
R	Tool Radius (Dia./2)
RDOC	Radial Depth of Cut
SFM	Surface Feet per Minute
V_{adj}	Adjusted Revolutions per Minute

$$D_{eff} = D \times \text{Sine}\left[I \pm \text{Arccos}\left(\frac{D - 2 \times ADOC}{D} \right) \right]$$

figure 4

$$V_{aj} = \frac{SFM \times 3.82}{D_{eff}}$$

figure 5

MACHINING PROBLEMS & SOLUTIONS

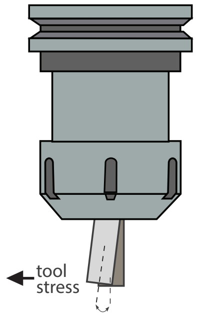
OUR HIGH-PERFOMANCE TOOLS ALLEVIATE MANY COMMON PROBLEMS

TOOL DEFLECTION

The most important factor in achieving tool performance and desired results is tool rigidity. Tool diameter increases rigidity and tool overhang decreases rigidity. Minimizing deflection is imperative for successful milling of your job.

Suggestions to minimize deflection:

- Use a more rigid tool (i.e. vibration dampening geometries, larger core design, etc)
- Maintain sharp tools
- Increase tool diameter
- Decrease depth of cut
- Decrease inches per minute (IPM)
- Use a climb milling approach
- Use shorter overall length tools and shorter flute lengths
- Use long reach end mills
- Increase the number of flutes
- Modify Surface Feet / Minute (SFM) parameters



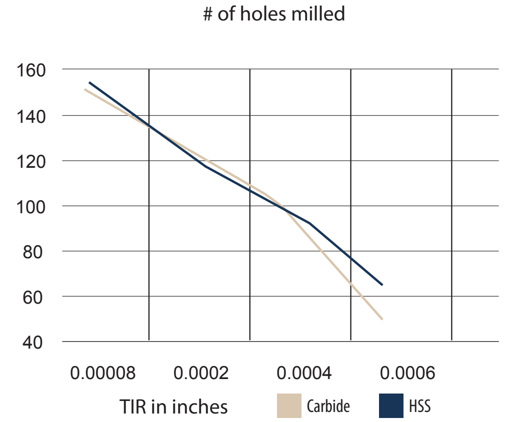
DEFLECTION RESULT

TOOL RUNOUT

To disperse heat quickly, running the spindle at high speeds is required. However, running at high speeds can also cause runout. More force is exerted if the tool does not run concentric to its centerlines, causing more wear on one side. Runout greatly affects accuracy and tool life. If the tools run-out are high, cutting edges become rough, which in turn can cause tool breakage, shorten tool life and decrease accuracy.

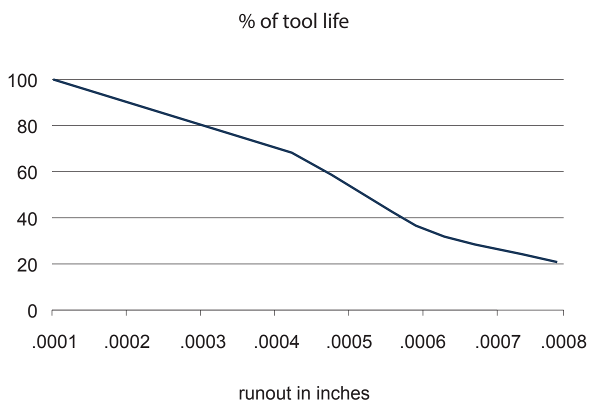
Furthermore, run-out increases the average chip thickness for the teeth engaged in the cut and increases the ratio of the maximum to average force. Run-out also shifts the frequency content of the force signal away from the tooth passing frequency to the spindle rotational frequency. The ratio of the run-out to the feed rate is identified as an important parameter which determines the effect of run-out on the cutting force.

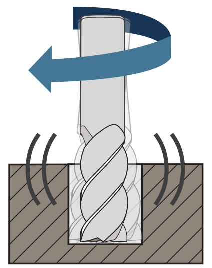
Controlling runout is imperative for maximum tool life and reducing costs. Improving run-out can be achieved by using correct tool holders and collets as well as choosing correct feeds and speeds.



EFFECT OF RUNOUT ON CARBIDE AND HSS

Tool size and material are important factors when calculating appropriate runout. In general, for 3⁄4" tools in diameter or larger, runout of 0.0005" is an acceptable measurement to control runout. However, smaller tools may require runout to be much better than 0.0005". Tool materials are also critical. The right run-out is relative not just to tool size, but also to tool material.
If run-out is controlled properly, carbide tools can last much longer than HSS. However, carbide is more affected due to run-out. Cutting forces that are evenly distributed on each flute (less run-out) stabilizes the cutting depth on each flute and produces a finer surface finish. Excessive force will be applied to only one flute with run-out of 0.0005" or higher.
Runout causes a tool's resonating edges to strike the side walls during the milling operation. This can result in uneven wall surface and poor finishes.





Runout causes a tool's resonating edges to strike the side walls during the milling operation. This can result in uneven wall surface and poor finishes.

RUNOUT CONSIDERATIONS

Although a higher-quality tool holder is more expensive, it can improve tool life dramatically and the savings can be measured in cost per hole. Allowing runout exceeding 0.0005" is equivalent to failing to cut milling costs by up to 65%.

Even the best collet cannot perform optimally in a worn spindle. Spindles should be checked regularly for run-out using a precision gage bar. Other influences on run-out include taper-to-taper contact, and the angle of the collet and corresponding clamping range. Basing tool holder purchase decisions solely on the price of the tool holder, or tool life and cost per hole, may sacrifice quality and accuracy. Other features to control run-out should be examined additionally, such as taper-to-taper contact, as well as collet angles and corresponding clamping ranges. More concentric clamping and increased clamping force can also improve run-out. A smaller range provides a more concentric clamping of the tool shank.

TOTAL INDICATED RUNOUT (TIR)

Rotary tools have two types of runout, static and dynamic. Static runout (static TIR), is the result of problems with the physical dimensions of, or arrangement of the components of the tool/collet/spindle system. Dynamic runout (dynamic TIR) might also result for dimensional inconsistencies, but can include other factors such as uneven material density, worn out spindle bearings, poor collet to spindle coupling, loose bits and spindle motor vibration.

Dynamic TIR

Dynamic TIR is usually more difficult to measure than Static TIR because it is normally smaller. Static TIR measurements can be reached by affixing a bit into the spindle to measure the concentricity via a test indicator. In most cases combining Angular and Radial TIR is the resulting Static TIR. At the spindle's operational speeds, runout can change as a result of heat, vibration and centrifugal force.

Angular TIR

Angular TIR is caused by an improper positioning between the rotational axis of the tool and the central axis of the collet/spindle system. Origins of the misalignment may include particles between the spindle bore taper and collet, misaligned central collet bore, deteriorated spindle taper, or improper setting of screws in the collet.

Radial Runout

Radial Run out results from a parallel offset of the central axis of the collet/spindle and the rotational axis of the tool. A common cause is a shank smaller than the minimum diameter of the collet gripping range. If a spindle assessment indicates that it can handle small runout on its own, then the determining factor to a low runout may very well be the tool holder itself.

By identifying, calculating and improving runout, significant increases in efficiency and savings can be seen almost immediately. Using the correct tool holder is crucial for any machine shop, large or small.