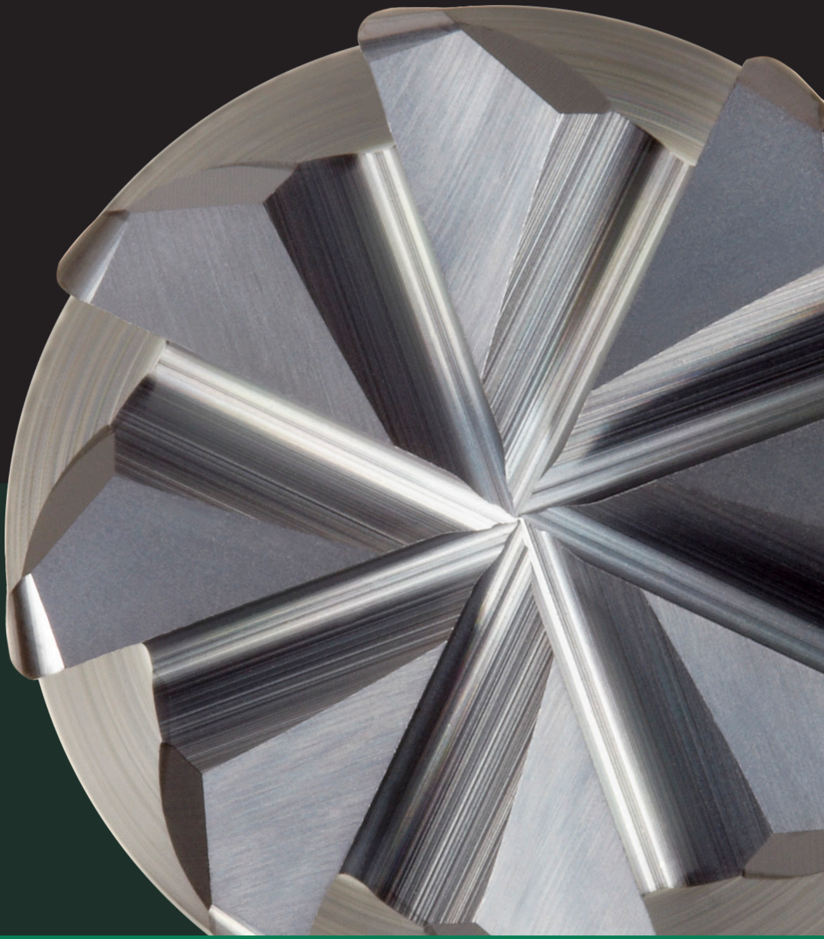
Helical

MACHINING GUIDEBOOK

Quick Reference eBook for CNC Milling Practices & Techniques



Contents

Milling Techniques & Strategies

Terminology & Common Calculations

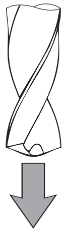
Troubleshooting

01

Milling Techniques

Types of Tool Entry

The type of part entry that is programmed has a lot of influence on the tool's success and is one of the most punishing operations for a cutter. Below we have listed some common part entry methods and suggestions on how to perform them successfully.



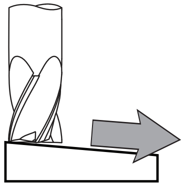
Pre-Drilled Hole

Pre-drilling a hole to full pocket depth (and 5-10% larger than the end mill diameter) is the safest practice of dropping your end mill into a pocket. This method ensures the least amount of end work abuse and premature tool wear.

Helical Interpolation

A very common and safe practice with ferrous materials. Employing corner radius end mills during this operation will decrease tool wear and lessen corner breakdown. We recommend a programmed helix diameter >110-120% of tool diameter.

Download the Helical Milling Advisor™ at www.helicaltool.com to get real-time Helical Interpolation information for your specific application.



Ramping-In

This type of operation can be very successful, but institutes many different torsional forces the cutter must withstand. Finding a tool with good core strength plus room for proper chip evacuation is key. Employing corner radius end mills during this operation will help immensely.

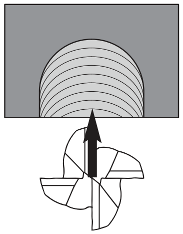
Below are some suggested starting ramp angles:

- Soft/Non-Ferrous Materials: 3° – 10°
- Hard/Ferrous Materials: 1° – 3°

Types of Tool Entry (cont.)

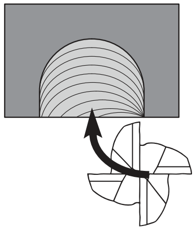
Straight Plunge

The least preferred method and one that can easily break a tool. The tool must be center cutting. End milling incorporates a flat entry point making chip evacuation tough, tool pressure very high and success random at best. Please note: Drill bits are intended for straight plunging and we highly recommend this type of tool for this operation.



Straight Entry

Straight entry into the part takes a toll on the cutter. Until the cutter is fully engaged, the feed rate upon entry is recommended to be reduced by at least 50%.



Roll-In Entry

Rolling into the cut ensures a cutter to work its way to full enagagement and naturally acquire proper chip thickness. The feed rate in this scenario should be reduced by 50%.

Side Entry

(use tools with a corner radius for best results)

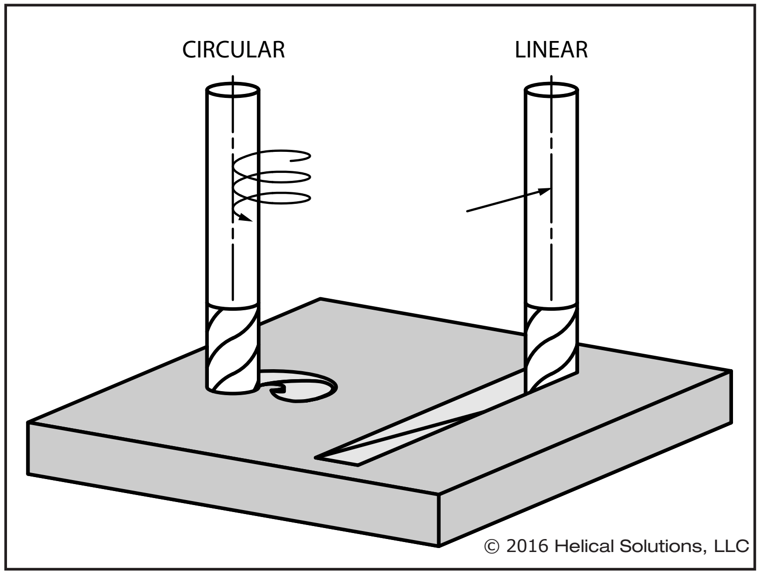
Ramping

Poor tool life and premature tool failure are concerns in every machining application. While there can be complicated answers, something as simple as tool path selection can make all the difference.

What Is Ramping?

Ramping refers to the simultaneous radial and axial motion of a cutting tool, making an angular tool path. Often times, this method is used to approach a part when there is a need to create closed forms such as pockets, cavities, engravings and holes. This eliminates the need to plunge with an end mill or drill to create a starting point. Ramping is particularly important in micromachining, where even the slightest imbalance in cutting forces will cause a tool to fail.

Figure 1: Circular & Linear Ramping



Ramping Tool Paths

There are two forms of ramping: Linear and Circular (see Figure 1). Linear ramping involves moving a cutting tool along two axes (the z-axis and one of the x,y axes). Circular ramping, or helical interpolation, has a spiral motion of the cutting tool that engages all three axes (x, y and z axes). End mills with either center cutting or non-center cutting geometry can be used for both forms of ramping, however the angle of descent will vary depending on end style.

Circular ramping typically has less radial engagement on the cutting tool, with the cutting forces distributed across the 3 different axes. Linear ramping has significantly more radial engagement with complementally increased cutting forces distributed across only 2 axes. However, both forms of ramping have a better distribution of forces than plunging, where all the cutting force is concentrated along the z-axis of the cutting tool. Consequently, circular ramping is recommended whenever possible, as it ensures the longest tool life.

Ramping (cont.)

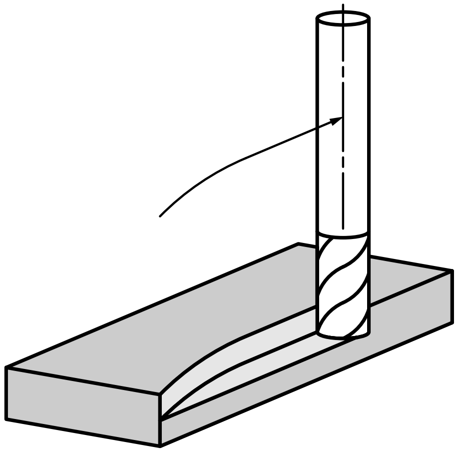
Benefits of Ramping

Ramping gradually increases in depth, preventing any shock loading on end mills, which reduces costs resulting from unnecessary tool breakage. Again, this is particularly helpful in fussy micromachining applications. Additionally, it produces smaller chips when compared to plunging, which makes chip evacuation faster and easier. As a result, cycle time can be decreased by running the end mill at faster speed and feed rates. Ramping also creates an extra space in the tool changer that would otherwise be occupied by a drill purposed with machining a starter hole.

Arcing

Similar to ramping in both method and benefit, arcing is another technique of approaching a workpiece (see Figure 2). While ramping enters the part from the top, arcing enters from the side. The end mill follows a curved tool path (or arc) when milling, thus gradually increasing the load on the tool as the tool enters the part, as well as gradually decreasing the load as the tool exits the part. In this way, shock loading and possible tool breakage are avoided.

Figure 2: Arcing



Thin Wall Milling

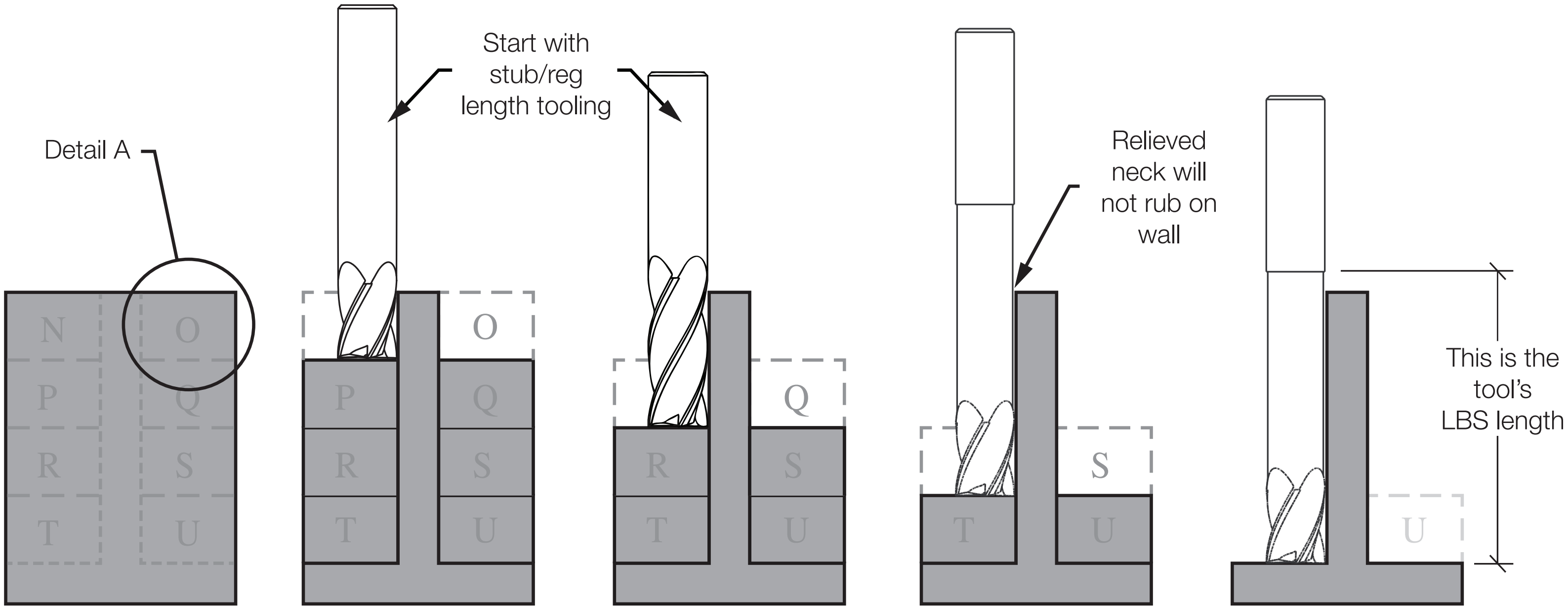
Milling part features with thin wall characteristics while maintaining dimensional accuracy and straightness can be difficult at best. Although multiple factors contribute, some key components are discussed below and can help turn these types of applications around.

Proper Tooling

A long length tool, combined with a long length of cut, can spell trouble in situations like this due to deflection, chatter and breakage. It is essential to keep the tool as strong as possible while maintaining the ability to reach to the desired depth. It is essential to look at necked-down tooling when reaching >3x dia. depths.

Axial Depth of Cut (ADOC)

Keeping a wide cross-section behind the wall for support on the way down is vital. Below, we recommend producing a “stepped down” approach dividing the total wall height to manageable depths while working each side of the wall. The ADOC dimension can/will vary depending on the material (and its hardness) being cut.

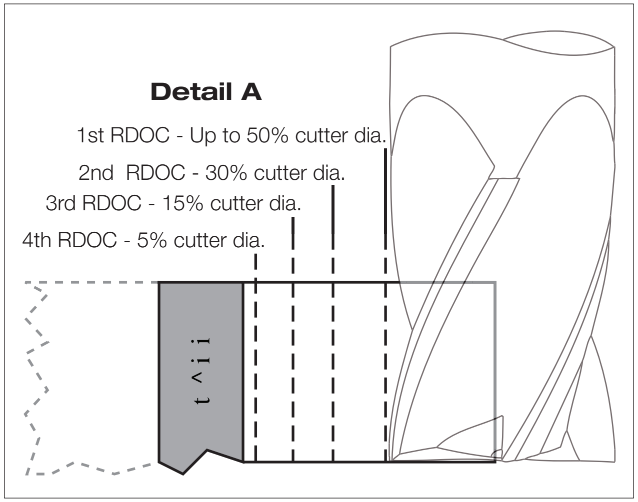


Thin Wall Milling (cont.)

RDOC

A progressive radial depth of cut (RDOC) strategy is of equal importance as wall height is being established. Reducing tool pressure while support stock is disappearing is equally important to keep wall stable.

- **Detail A** represents a 5-step progressive radial approach. The number of passes will depend upon your particular application, material/hardness & final wall thickness/height.
- This approach helps to keep the pressure off the wall as you make your way towards it. Additionally, it is recommended to alternate sides when using this depth of cut (RDOC) strategy.
- 4th/5th RDOC passes could turn out to be very light, keeping wall vibration to a minimum and part finish maximized.



© 2016 Helical Solutions, LLC

Other Ideas

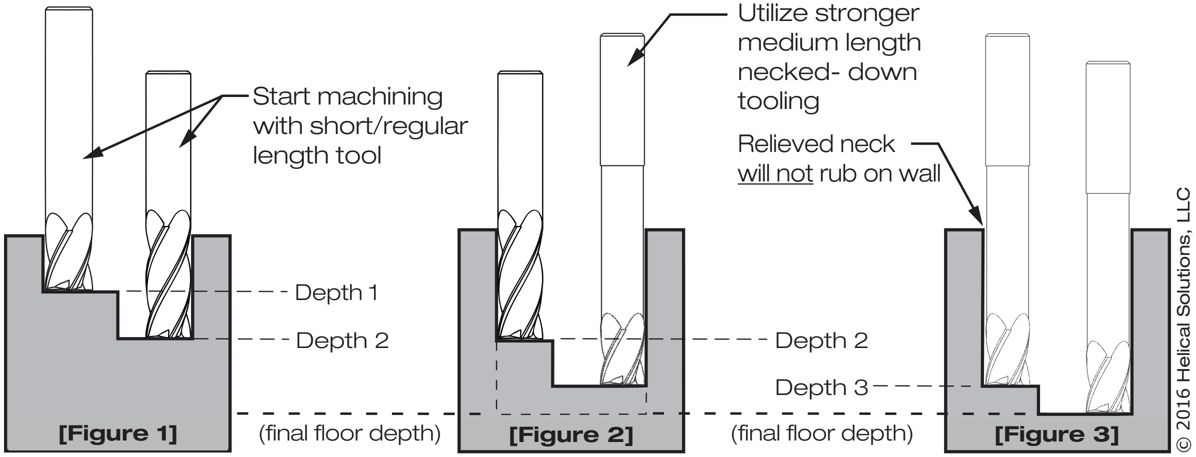
- Climb milling will help to keep tool pressure to a minimum.
- Vibration dampening/wall stabilization can be achieved in “hard to fixture thin wall situations” by using thermoplastic compounds or wax - which can be removed (thermally).
- The use of ultra-high performance tool paths can optimize tool performance, work with lighter depths of cut and offer less tool cutting pressure.

Deep Pocket Milling

Deep pocket milling continues to be one of the most demanding milling operations. Deep pocket milling routines usually involve long reach, poor chip evacuation, limited coolant delivery, deflection issues and serious tool engagement violations. We have illustrated some helpful techniques below (see Figures 1, 2, 3).

Common Problems Experienced

- Chatter
- Wall taper
- Tool deflection
- Tool engagement violations
- Recutting chips
- Breakage



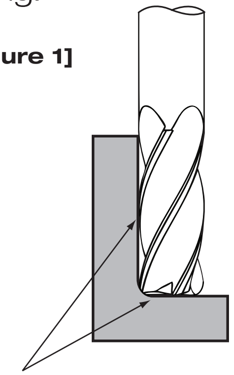
Some Things to Consider

- *Step down milling routine*: This procedure (shown in Figures 1, 2, 3) ensures that you are utilizing a controlled axial depth of cut (ADOC) at each level, thus optimizing speeds and feeds. It is imperative to start with a stub or standard length tool to get down to approximately 2-3 x dia. deep (depth 1-2), then employ necked down tooling.
- *Necked down tooling*: Once depths 2 and 3 are reached (Figure 2), use our stronger necked down tooling in order to maintain tool integrity and respectable feed rates. Necked tooling has a shorter length of cut (LOC), ensuring a much stronger tool and a neck diameter smaller than the cut diameter allowing for plenty of wall clearance.

Finishing

Finishing cuts are used to complete the part and achieve the final dimension, tolerances, and surface finish. The goal when finishing a component is to avoid or at least minimize the necessity for manual re-touching.

[Figure 1]



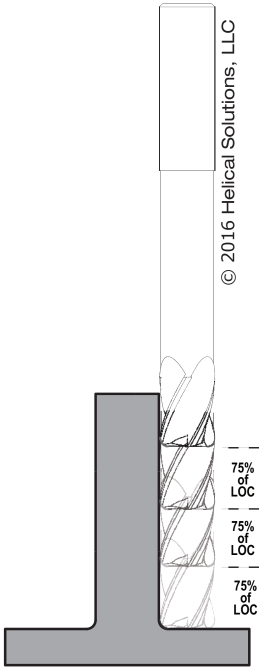
When contacting wall and floor, feed rate may need to be reduced.

Factors that Influence Finish

- Specific material and hardness
- Proper cutting tool speeds & feeds
- Tool holder accuracy
- Proper tool design and deployment
- Tool projection/deflection
- Tool-to-workpiece orientation
- Rigidity of work holding
- Coolant/lubricity

Tips for Successful Finishing

- Using an increased helix angle will help to improve surface finish.
  45° or higher for Aluminum
  38° or higher for hard metal machining
- Increasing the number of flutes will help to improve surface finish.
  3, 4 for Aluminum
  5, 7+ for hard metal
- Utilize tools with corner radii.
- Tool runout of .0003 or less.
- Using precision tool holders that are in good condition, undamaged and run true.
- Climb milling vs. conventional produces a better surface finish.
- Variable pitch tooling helps to reduce chatter and increase part finish.
- Proper radial depth of cut (RDOC) between 2-5% of tool diameter.
- For long reach walls, consider using “necked down” tools which allow less deflection, with LOC overlap a good step blending will occur (see Figure 2).
- Extreme contact finishing (> 3.0 x dia. deep) may require a 50% feed rate reduction.



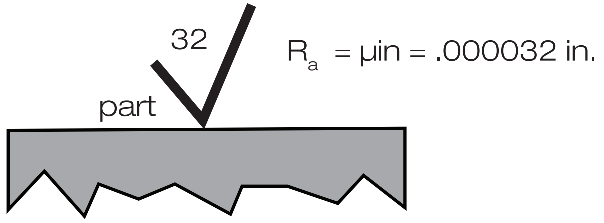
[Figure 2]

Finishing (cont.)

Surface finishing is an important step in the operations sequence for the production of any high quality part. Many times, this requirement is aesthetically driven but other times has to satisfy print specification.

Below is some common surface finish nomenclature:

- R_a = Roughness Average
- R_q = RMS (Root Mean Square) = $R_a \times 1.1$
- $R_z = R_a \times 3.1$



QUICK REFERENCE GUIDE

To improve surface finish	⬆ **Increase RPM**	⬇ **Lower IPT**	**Climb Mill**

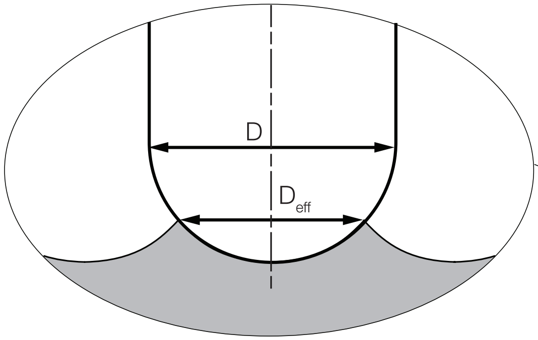
Ball Nose Milling Strategy

90°

Ball nose end mills are ideal for machining 3-dimensional contour shapes typically found in the die and mold industry, manufacturing of turbine blades and establishing general part radius requirements.

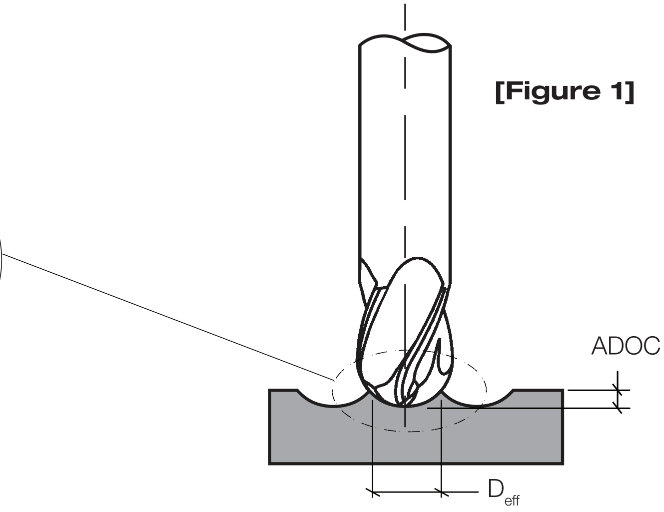
To properly employ a ball nose end mill (with no tilt angle) and gain the optimal tool life and part finish, is to follow the 2-step process on the following page.

[Detail A]



© 2016 Helical Solutions, LLC

[Figure 1]



Ball Nose Milling Strategy (cont.)

Step One

Calculate Your Effective Cutting Diameter (D_{eff}) – Implemented when using a ball nose end mill that is utilizing a ADOC that is less than the full radius of the ball. This can be done using the chart below (see Figure 2) that represents some common tool diameters and ADOC combinations or by using the traditional calculation (see Figure 3).

[Figure 2]

		AXIAL DEPTH OF CUT (ADOC)														
		.010	**.020**	**.030**	**.050**	**.070**	**.090**	**.120**	**.150**	**.180**	**.220**	**.260**	**.300**	**.350**	**.400**	**.450**
TOOL DIAMETER	1/8	.068	.092	.107	.122	.124	.112	.049								
	1/4	.098	.136	.162	.200	.224	.240	.250	.245	.224	.162					
	3/8	.121	.169	.203	.255	.292	.320	.350	.367	.375	.369	.346	.300	.187		
	1/2	.140	.196	.237	.300	.347	.384	.427	.458	.480	.496	.500	.490	.458	.400	.300
	5/8	.157	.220	.267	.339	.394	.438	.492	.533	.565	.596	.615	.624	.620	.600	.561
	3/4	.172	.242	.294	.374	.436	.487	.550	.600	.641	.683	.714	.735	.748	.748	.735
	1	.199	.280	.341	.436	.510	.572	.650	.714	.768	.828	.877	.917	.954	.980	.995

[Figure 3]

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

Step Two

Calculate Your New Velocity Adjustment (V_{adj}) - This new velocity adjustment will be calculated using the new effective cutting diameter (D_{eff}). If you are using less than the cutter diameter, then its likely your RPM's will need to be adjusted upward (see Figure 4).

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

[Figure 4]

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

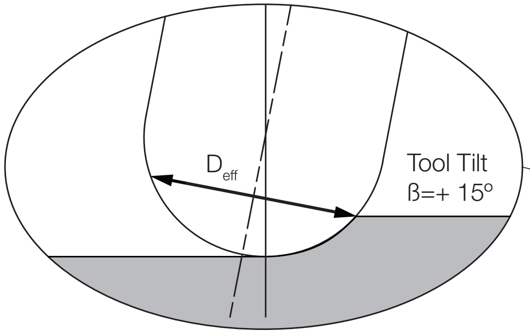
Ball Nose Milling Strategy (cont.)

At 15° Incline

It is highly recommended to use ball nose end mills on an incline (ß) to avoid a “0” SFM condition at the center of the tool, thus increasing tool life and part finish. For ball nose optimization (and in addition to tilting the tool), it is highly recommended to feed the tool in the direction of the incline and utilize a climb milling technique.

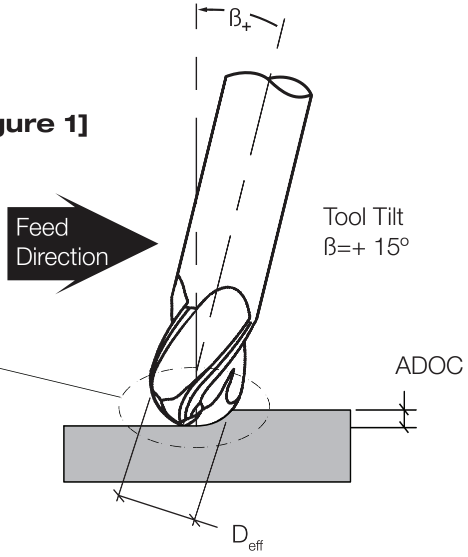
To properly employ a ball nose end mill (with a tool angle) and gain the most optimum tool life and part finish is to follow the 2-step process on the following page.

[Detail A]



© 2016 Helical Solutions, LLC

[Figure 1]



Ball Nose Milling Strategy (cont.)

Step One

Calculate Your Effective Cutting Diameter (D_{eff}) - To be implemented when using a ball nose end mill that is utilizing a ADOC that is less than the full radius of the ball. This can be done using the chart below (see Figure 2) that represents some common tool diameters & ADOC's at 15° tilt angle or by using the traditional calculation
(see Figure 3).

[Figure 2]

15° Tilt		AXIAL DEPTH OF CUT (ADOC)															
		.010	**.020**	**.030**	**.050**	**.070**	**.090**	**.120**	**.150**	**.180**	**.220**	**.260**	**.300**	**.350**	**.400**	**.450**	**.500**
TOOL DIAMETER	1/8	.093	.111	.120	.125	.116	.094	.018									
	1/4	.154	.185	.206	.232	.245	.250	.244	.224	.188	.108						
	3/8	.209	.249	.278	.317	.343	.360	.373	.374	.366	.340	.297	.232	.097			
	1/2	.259	.308	.343	.393	.428	.454	.480	.494	.500	.495	.477	.447	.391	.309	.186	
	5/8	.308	.364	.404	.463	.506	.539	.575	.600	.615	.625	.622	.610	.580	.534	.471	.386
	3/4	.355	.417	.463	.530	.579	.618	.663	.696	.720	.740	.749	.749	.736	710	.671	.618
	1	.446	.519	.573	.654	.715	.765	.824	.871	.908	.945	.972	.989	.999	.998	.987	.966

[Figure 3]

$$D_{eff} = D \times \text{sine}\left[\beta + \text{arccos}\left(\frac{D-2 \times ADOC}{D}\right)\right]$$

Step Two

Calculate Your New Velocity Adjustment (V_{adj}) - This new velocity adjustment will be calculated using the new effective cutting diameter (D_{eff}). If you are using less than the cutter diameter, then its likely your RPM's will need to be adjusted upward (see Figure 4).

D_{eff} = Effective Cutting Diameter
R = Tool Radius
ADOC = Axial Depth of Cut
D_{eff} = Effective Cutting Diameter (see Figure 2)
SFM = Mfg Recommended Surface Feet per Minute
V_{adj} = Adjusted RPM for lighter ADOC

[Figure 4]

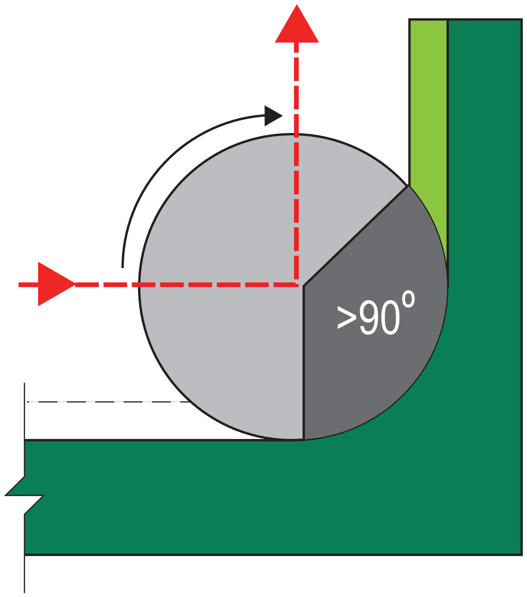
$$V_{adj} = \frac{SFM \times 3.82}{D_{eff}}$$

Corner Engagement

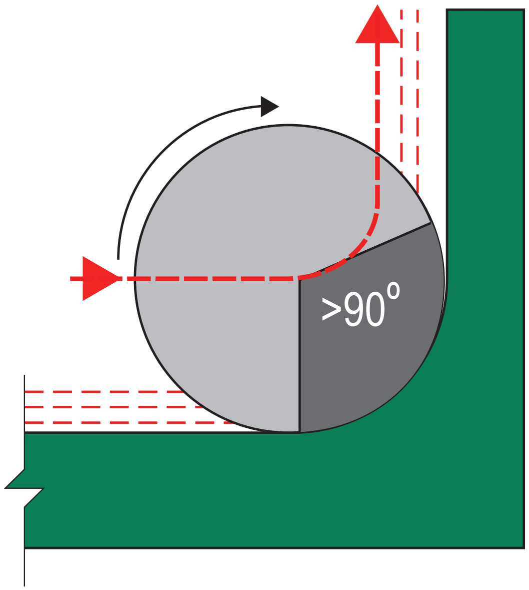
Milling involves significant variations in cutting forces, resulting in ultra-conservative tool running parameters and premature tool wear. One difficult (and often suspect) area of this type of machining is when the cutting tool experiences an “inside corner” condition. This is where the tool’s engagement angle significantly increases and poor performance may be experienced.

Evidence of this difficult to machine area is detected by:

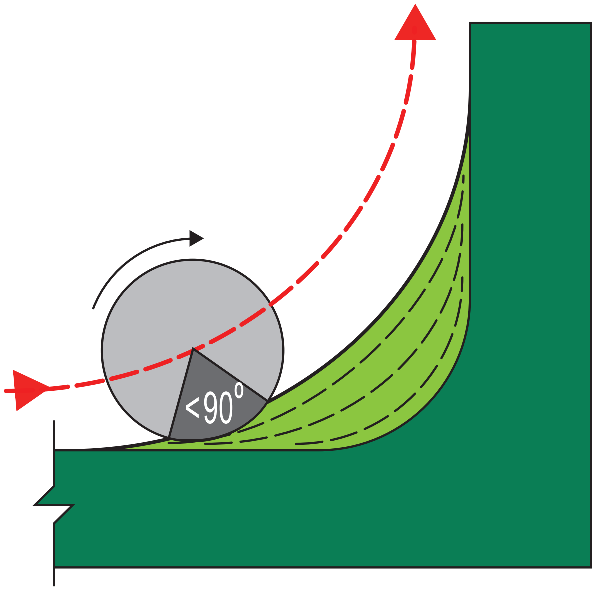
- Chatter - visible in “poor” corner finish.
- Deflection - detected by unwanted “measured” wall taper.
- Cutting sound - tool squawking or chirping in the corners.
- Tool breakage/chipping - detrimental tool breakage or chipping, resulting in tool replacement.



Least Desirable



More Desirable



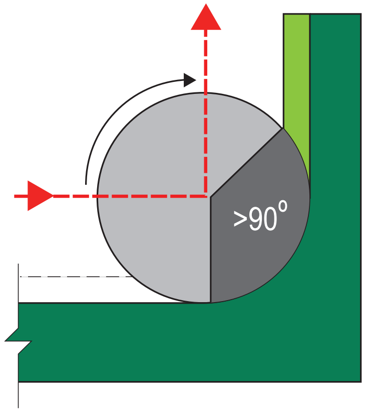
Most Desirable

Corner Engagement (cont.)

Least Desirable Condition

Generating an inside part radius that matches the radius of the tool at a 90° direction change can make for a less than ideal machining condition. With the tool experiencing extra material to cut (light green), increased engagement angle and a direction change some of the common results will be chatter, tool deflection/breakage and poor surface finish.

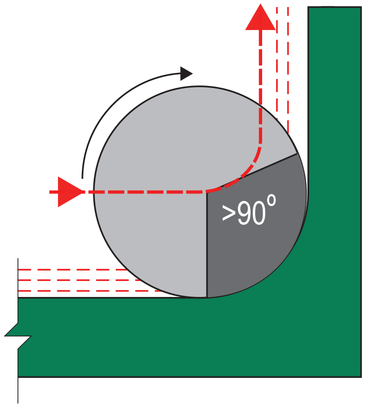
Feed rate may need to be lessened depending on the “tool radius-to-part radius ratio.”



More Desirable Condition

Generating an inside part radius that matches the radius of the tool with a sweeping direction change creates a more acceptable machining condition. The smaller radial depths of cut in this example help to manage the angle of engagement, but at the final pass the tool will experience a very high engagement angle and again, a less than desirable machining condition. Some of the common results will be chatter, tool deflection/breakage and poor surface finish.

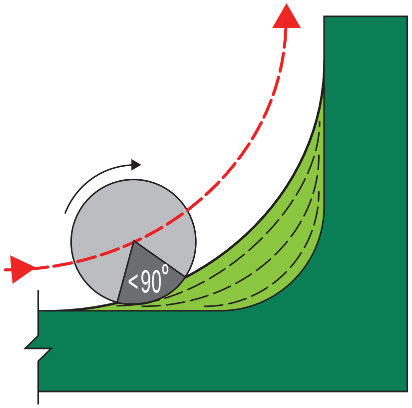
Feed rate may need to be reduced by 30-50% depending on the “tool radius-to-part radius ratio.”



Most Desirable Condition

Generating an inside part radius with a smaller tool and a sweeping action creates a very desireable machining condition. The manageable radial depths of cut and smaller tool diameter allow management of the tool engagement angle, higher feed rates and better surface finishes. As the cutter reaches full radial depth its engagement angle will increase, but feed reduction should be much less than the other conditions listed above.

Feed rate may need to be heightened depending on the “tool-to-part ratio.” Utilize tools that are smaller than the corner you are machining.



Angle of Engagement

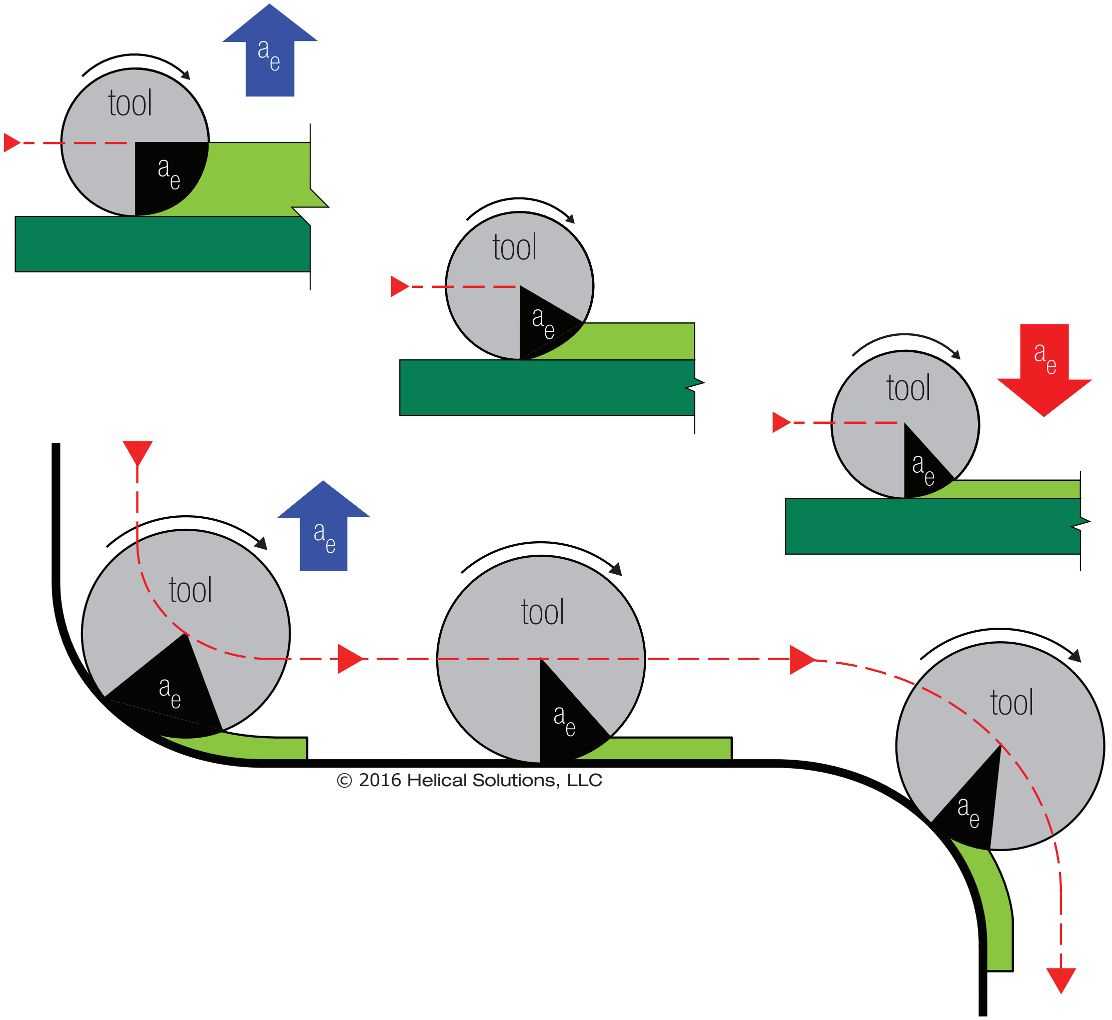
The Tool Engagement Angle (a_e) is an angular measurement about the periphery of the cutter that is in contact with the material being removed and directly related to the radial chip thickness.

An Increasing a_e can result in:

- Higher horsepower requirement
- Increased tool deflection
- Higher spindle load (wear/tear)
- Decreased feed rates

A decreasing a_e can result in:

- Lower horsepower requirement
- Decreased tool deflection
- Lower spindle load (wear/tear)
- Increased Feed Rates



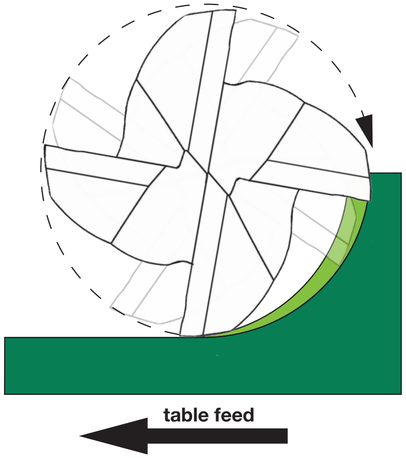
© 2016 Helical Solutions, LLC

Climb vs. Conventional Milling

There are two distinct ways to cut materials when milling, conventional (up) milling and climb (down) milling. The difference between these two techniques is the relationship of the rotation of the cutter to the direction of feed.

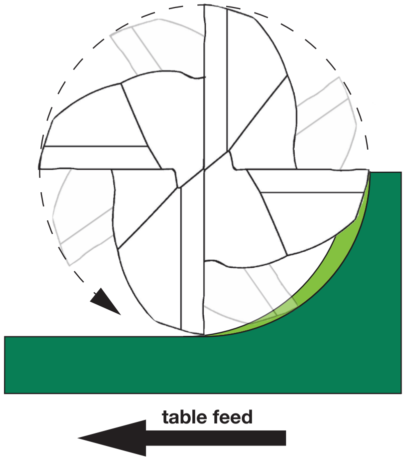
In conventional milling, the cutter rotates against the direction of the feed while during climb milling, the cutter rotates with the feed. Conventional milling is the traditional approach when cutting because the backlash, the play between the lead screw and the nut in the machine table, is eliminated.

Recently, climb milling has been recognized as the preferred way to approach a workpiece due to the fact that more and more machines compensate for backlash or have a backlash eliminator. Below are some key properties for both conventional and climb milling.



Conventional Milling

- Chip width starts from zero and increases which causes more heat to diffuse into the workpiece and produces work hardening
- Tool rubs more at the beginning of the cut causing faster tool wear and decreases tool life
- Chips are carried upward by the tooth and fall in front of cutter creating a marred finish and re-cutting of chips
- Upwards forces created in horizontal milling tend to lift the workpiece, more intricate and expansive work holdings are needed to lessen the lift created



Climb Milling

- Chip width starts from maximum and decreases so heat generated will more likely transfer to the chip
- Creates cleaner shear plane which causes the tool to rub less and increases tool life
- Chips are removed behind the cutter which reduces the chance of re-cutting
- Downwards forces in horizontal milling is created that helps hold the workpiece down, less complex work holdings are need when coupled with these forces

Climb vs. Conventional Milling (cont.)

When to Choose Conventional or Climb Milling

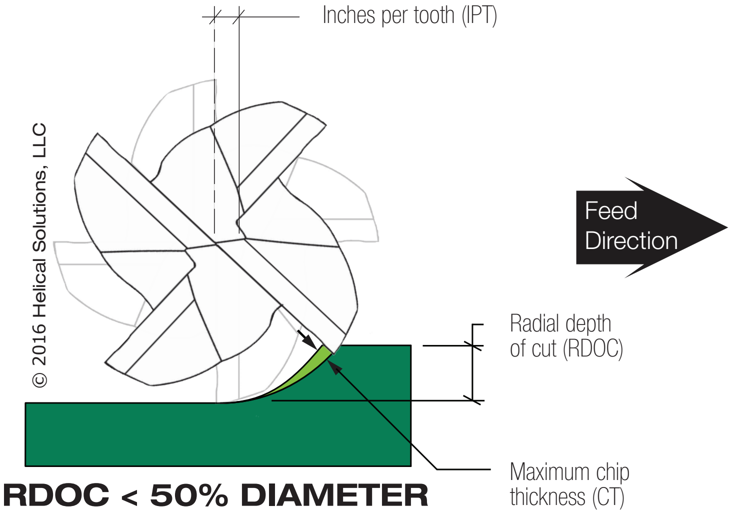
Climb milling is generally the best way to machine parts today since it reduces the load from the cutting edge, leaves a better surface finish, and improves tool life. During conventional milling, the cutter tends to dig into the workpiece and may cause the part to be cut out of tolerance.

Even though climb milling is the preferred way to machine parts, there are times when conventional milling is the recommended choice. Backlash, which is typically found in older and manual machines, is a huge concern with climb milling. If the machine does not counteract backlash, conventional milling should be implemented. Conventional milling is also suggested for use on casting or forgings or when the part is case hardened since the cut begins under the surface of the material.

Chip Thinning

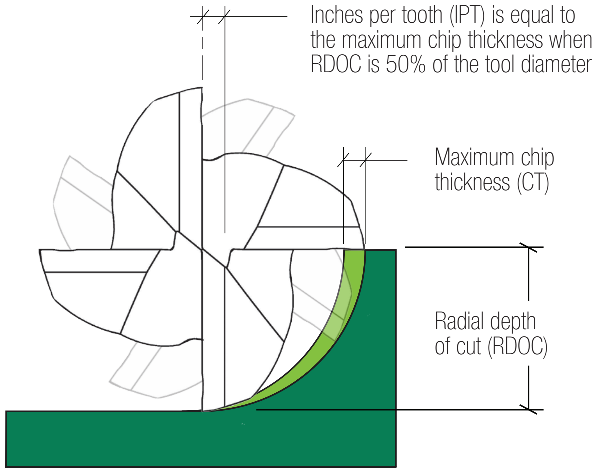
Milling with a light radial depth of cut (less than 50% of cutter diameter) causes the chip being formed to be much thinner than the programmed advance per tooth. This results in excessive tool "rubbing" and premature tool wear/life.

[Figure 1]



RDOC < 50% DIAMETER

[Figure 2]



RDOC ≥ 50% DIAMETER

When programming a radial depth of cut (RDOC) less than 1/2 the tool diameter (Figure 1), employ the chip thinning calculation (Figure 3). A chip-thinning adjustment will prolong tool life and help reduce cycle time.

This feed rate adjustment needs to consider drastic tool engagement and angle increases when milling into corners. Significant feed rate reductions in these areas still apply and will need attention.

$$IPT = \frac{CT \times D}{2 \times \sqrt{(D \times RDOC) - RDOC^2}}$$

[Figure 3]

02

High Efficiency Milling

High Efficiency Milling

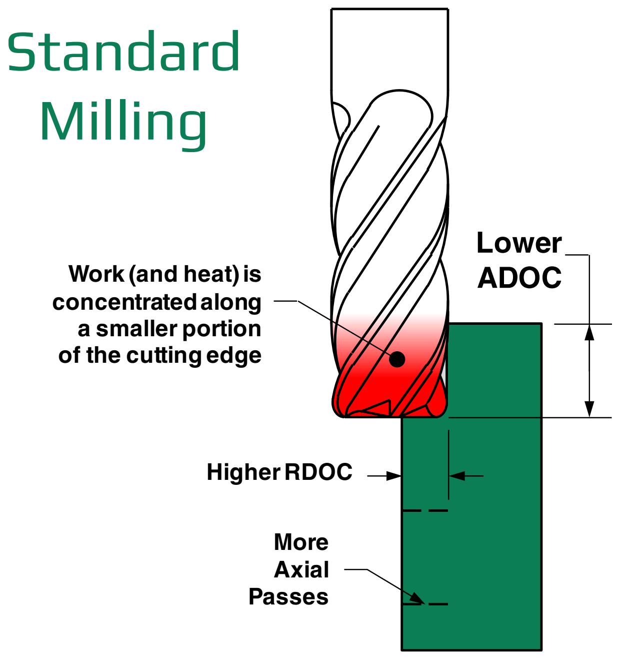
High Efficiency Milling (HEM) has become a common term in machine shops worldwide, but what does it mean? Simply, HEM is a milling technique for roughing that utilizes the entire flute length, spreading the wear evenly across the cutting length of the tool.

How It Works

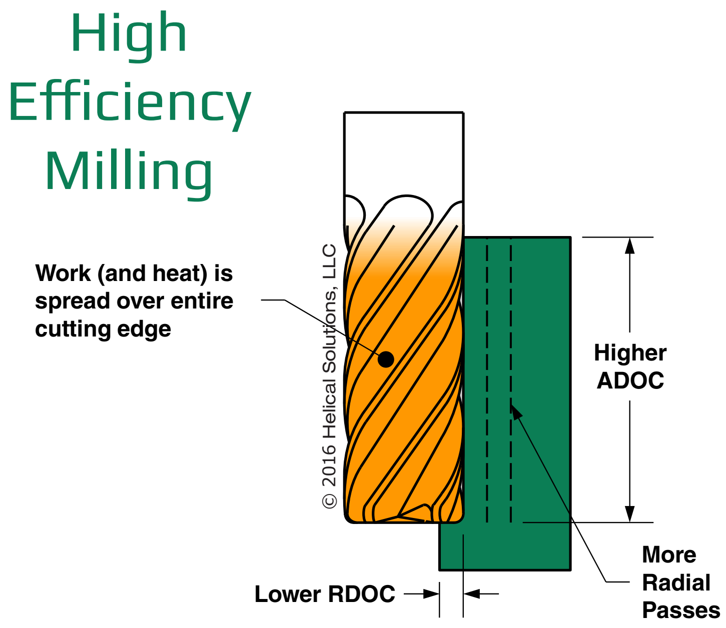
Machining technology has been advancing with the devlopment of faster, more powerful machines. In order to keep up, many CAM applications are generating more efficient HEM tool paths. These tool paths adjust parameters to maintain constant tool load throughout the entire roughing operation and allow more aggressive speeds and feeds.

Advantages

- Increased metal removal rates
- Reduced cycle times
- Increased tool life



Standard Milling



High Efficiency Milling

HEM Tooling

High efficiency milling can only go so far with general tooling. That's why Helical offers thousands of high performance tools specifically designed to withstand the rigors of HEM strategies. The following case studies illustrate the power of HEM using Helical end mills versus traditional roughing.

1/2" 5-Flute End Mill in 17-4ph (36 Rc) — (HEV-5)

	RPM	IPM	RDOC	ADOC	MRR	Cycle Time per Part	Parts per Tool	Tool Cost per Part
Traditional Roughing	2,200	20	.250 (50%)	.250 (50%)	1.25	11:20	15	14.66
HEM	6,000	80	.062 (12%)	.500 (100%)	2.50	7:00	40	9.22
Results	-	-	-	-	**+100%**	**-38.24%**	**+166.67%**	**-37.11%**

1/2" 3-Flute Rougher in 6061 Aluminum — (H45AL-C-3)

	RPM	IPM	RDOC	ADOC	MRR	Cycle Time per Part	Parts per Tool	Tool Cost per Part
Traditional Roughing	12,000	350	.250 (50%)	.500 (100%)	43.75	11:00	350	14.66
HEM	18,000	500	.200 (40%)	1.000 (200%)	100	3:00	900	3.33
Results	-	-	-	-	**+128.57%**	**-72.73%**	**+157.14%**	**-77.29%**

04

Depth of Cut

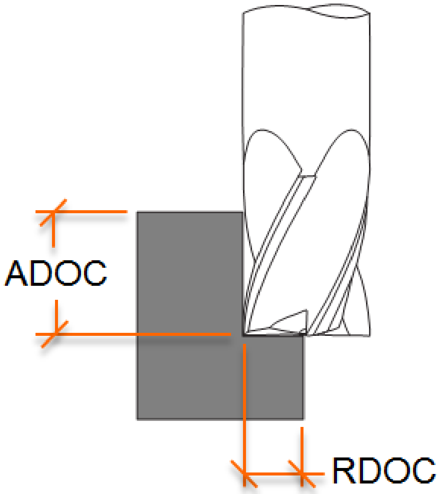
Depth of Cut

Radial Depth of Cut (RDOC)

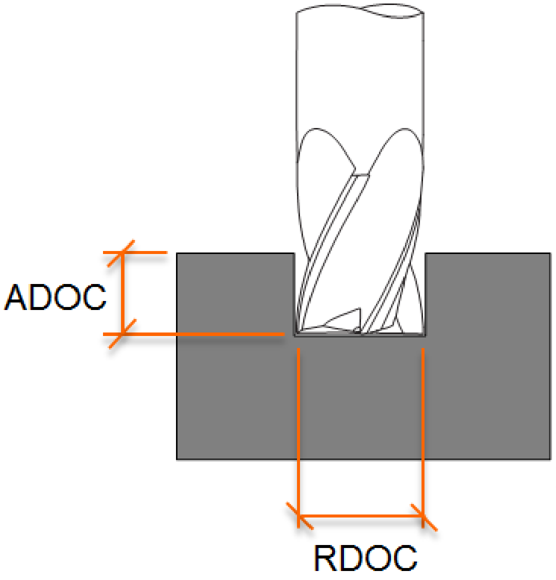
- The distance a tool is stepping over into the the material
- Stepover, Cut Width, XY

Axial Depth of Cut (ADOC)

- The distance a tool is being sent into the cut along it’s centerline.
- Directly related to MRR



Peripheral Milling



© 2016 Helical Solutions, LLC

Slotting

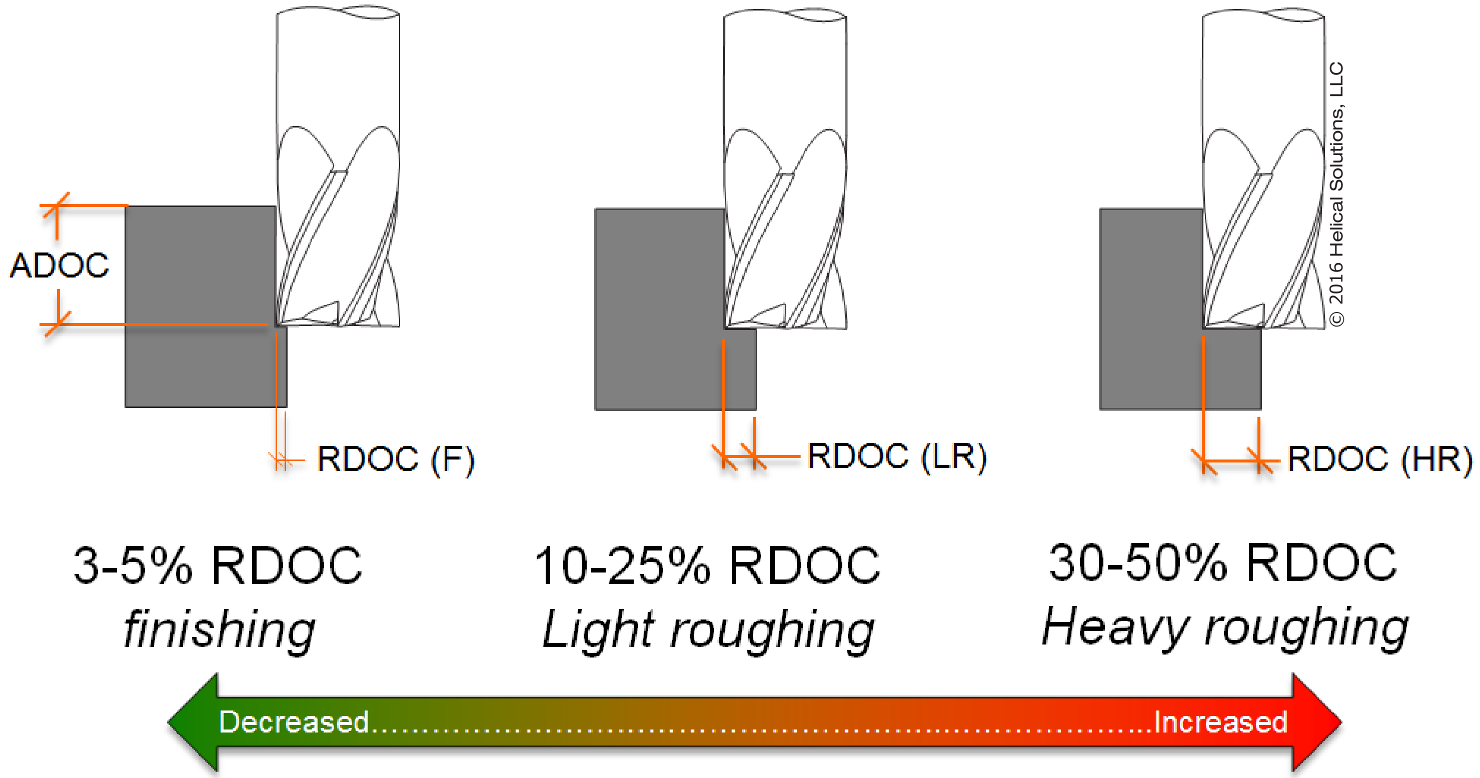
Depth of Cut - Peripheral

Traditionally, it has been:

- Heavy RDOC
- Light ADOC
- Conservative IPM

New strategies include:

- Light RDOC
- Heavy ADOC
- Increased IPM



3-5% RDOC
finishing

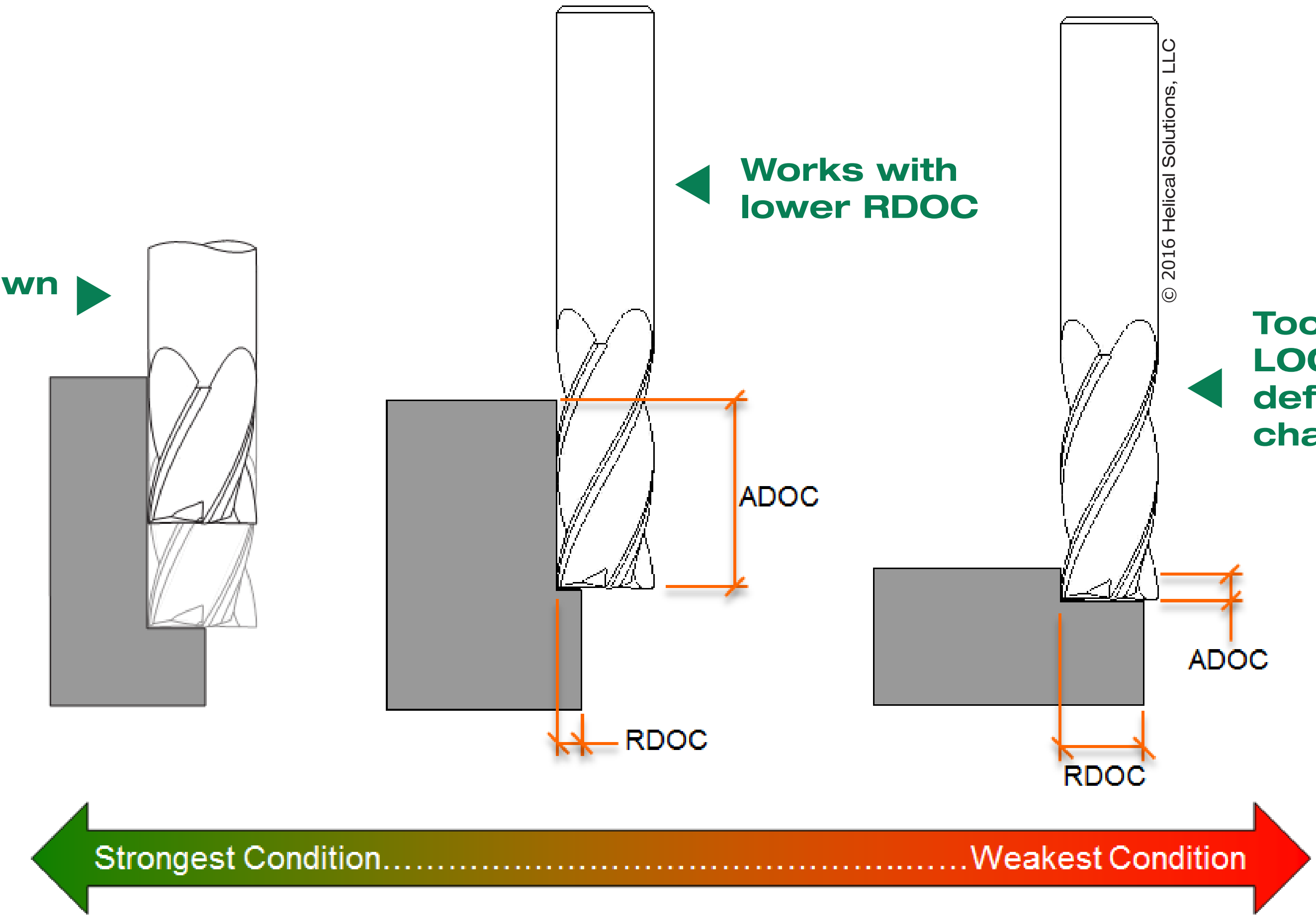
10-25% RDOC
Light roughing

30-50% RDOC
Heavy roughing

Horsepower Req. Tool Deflection
Chip Thickness

Horsepower Req. Tool Deflection
Chip Thickness

Depth of Cut - Peripheral (cont.)

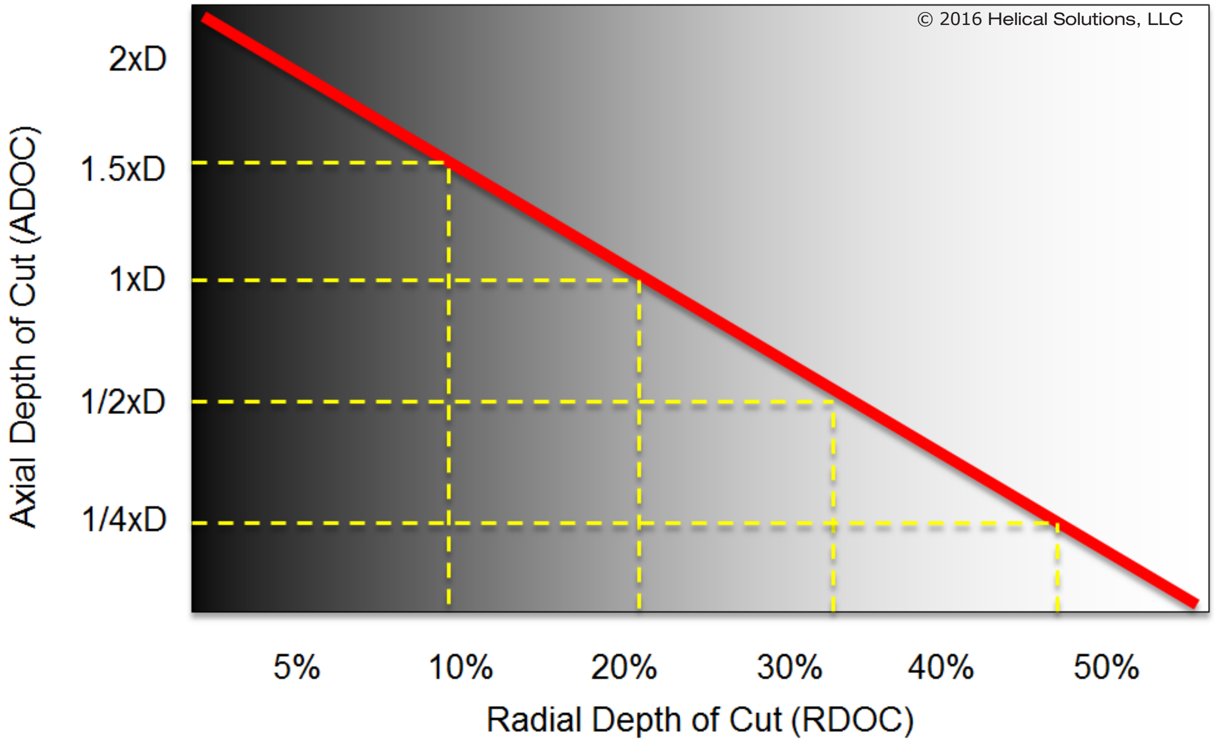


Necked down tool style

Works with lower RDOC

Too long of a LOC, leading to deflection and chatter

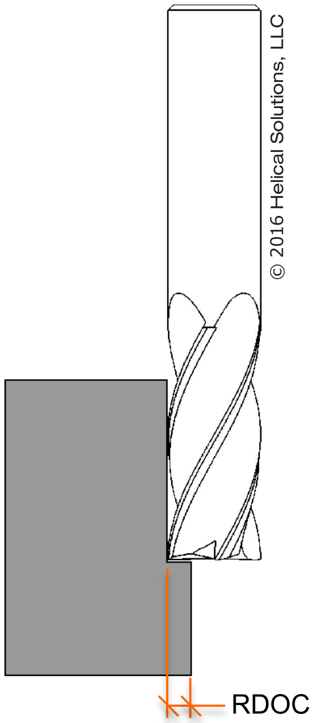
Depth of Cut - Peripheral (cont.)



Depth of Cut - Peripheral (cont.)

High Efficiency RDOC Strategy

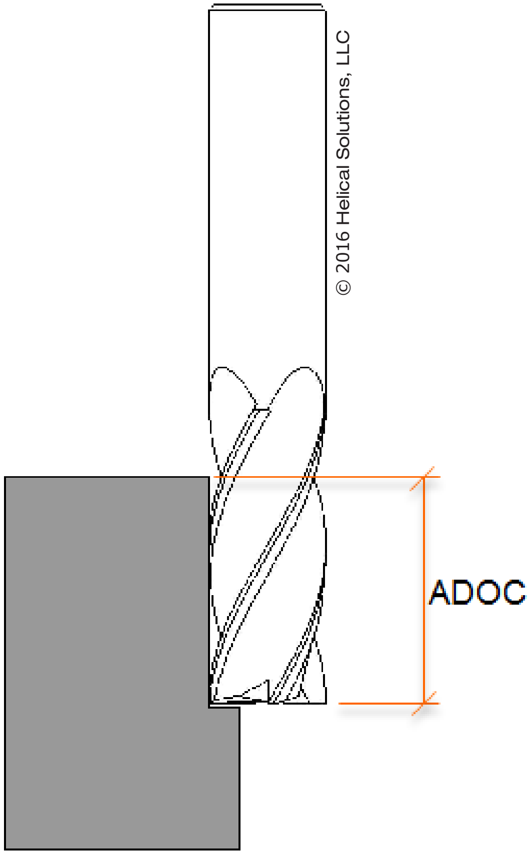
- Uniform cuts will increase tool life
- Climb Milling increases tool life
- Lowered RDOC = 7%-30% x D
- Increased IPM = chip thinning parameter must include “inside arc” feed reduction
- Multi-Fluted tools can be used
- Utilize HE type of tool paths for best results!



Depth of Cut - Peripheral (cont.)

High Efficiency ADOC Strategy

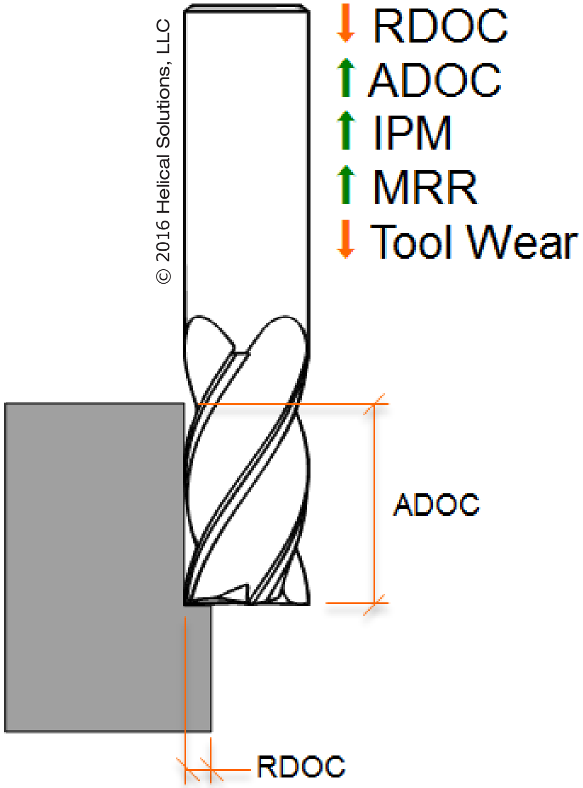
- Controlled slice milling
- Accommodates lower RDOC
- Up to 2 x dia. ADOC
- Utilizes majority of LOC
- Multi-fluted tooling allows larger core dia. = less deflection
- Tends to “stabilize cutter” by disbursing load for entire axial LOC length
- Increased ADOC = Increase in MRR
- Utilize HE type of Tool Paths for best results!



Depth of Cut - Peripheral (cont.)

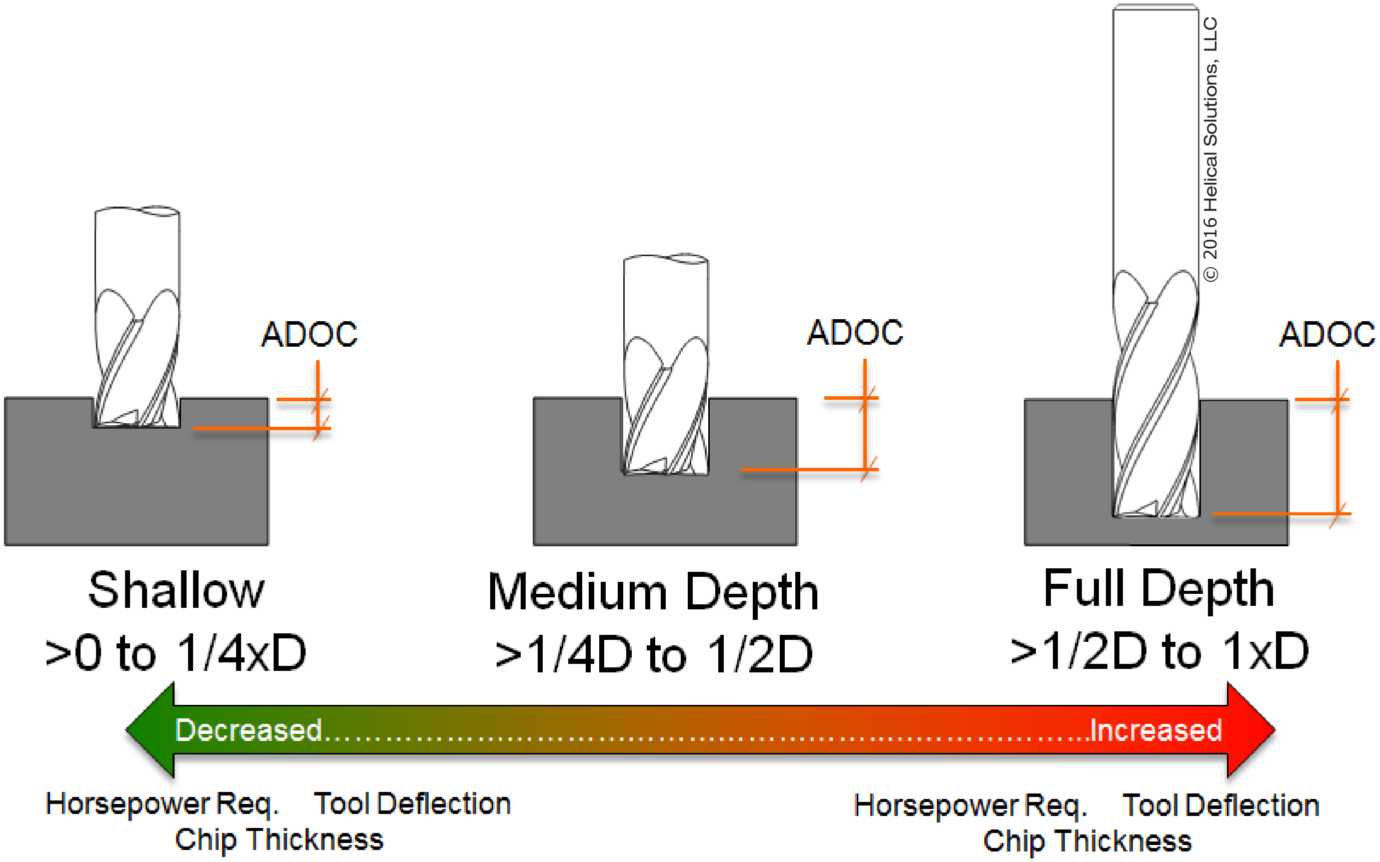
Light RDOC / Heavy ADOC Strategy

- Higher ADOC can help stabilize the cutter.
- Uniform cuts accommodate less “mechanical stress” and increase tool life
- Better heat/chip management
- Increased IPM (due to chip thinning)
- CAM applications with HP tool paths are good solutions for this type of strategy



↓ RDOC
↑ ADOC
↑ IPM
↑ MRR
↓ Tool Wear

Depth of Cut - Slotting



Shallow
>0 to 1/4xD

Medium Depth
>1/4D to 1/2D

Full Depth
>1/2D to 1xD

Decreased……………………………………………………………Increased

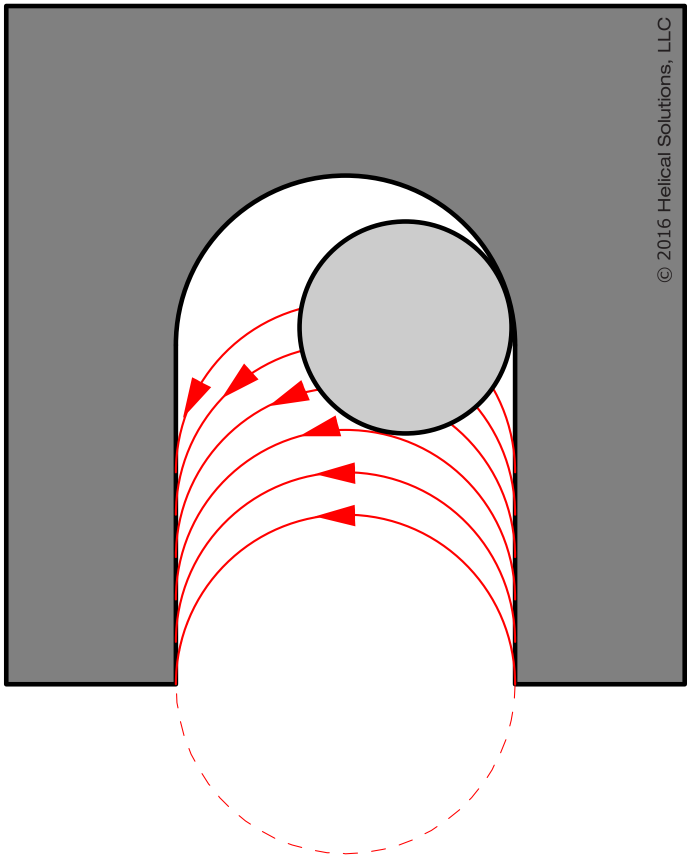
Horsepower Req. Tool Deflection
Chip Thickness

Horsepower Req. Tool Deflection
Chip Thickness

Depth of Cut - Slotting (cont.)

Controlled Slice Milling - RDOC Strategy

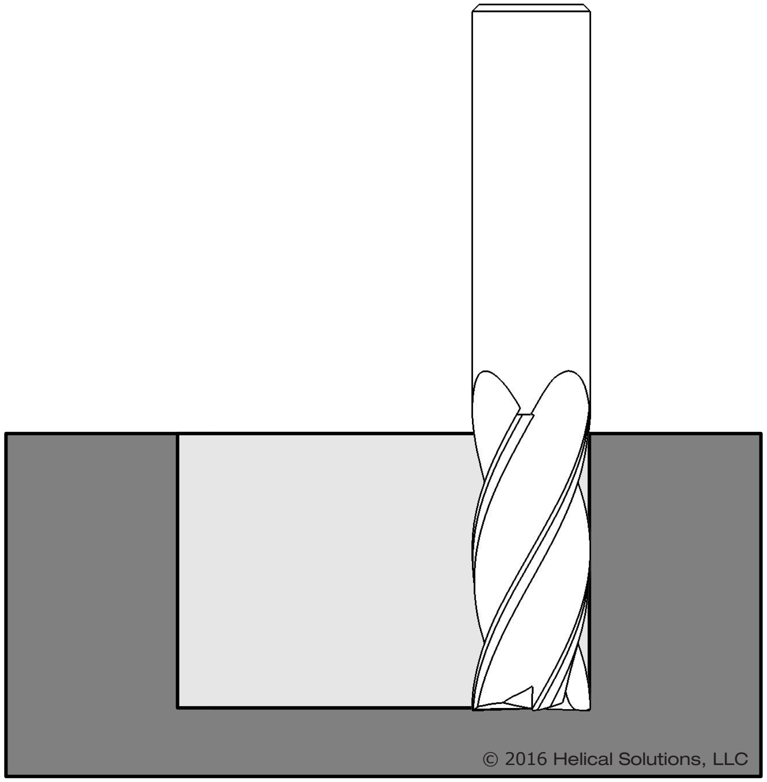
- Uniform cuts will increase tool life
- Lowered RDOC = 10-15% of tool diameter
- Increased IPM = chip thinning parameter must include “inside arc” feed reduction
- Max Cutter dia 55-65% of slot width
- Multi-Fluted tools can be used
- Must be able to evacuate the chip



Depth of Cut - Slotting (cont.)

Controlled Slice Milling - ADOC Strategy

- Allows for increased ADOC
- Up to 2 x dia. ADOC
- Utilizing entire LOC
- Multi-fluted tooling allows larger core dia. = less deflection
- Tends to “stabilize cutter” by disbursing load for entire axial LOC length
- Increased ADOC = Increase in MRR
- Utilize HE type of Tool Paths for best results!



04

End Mill Construction

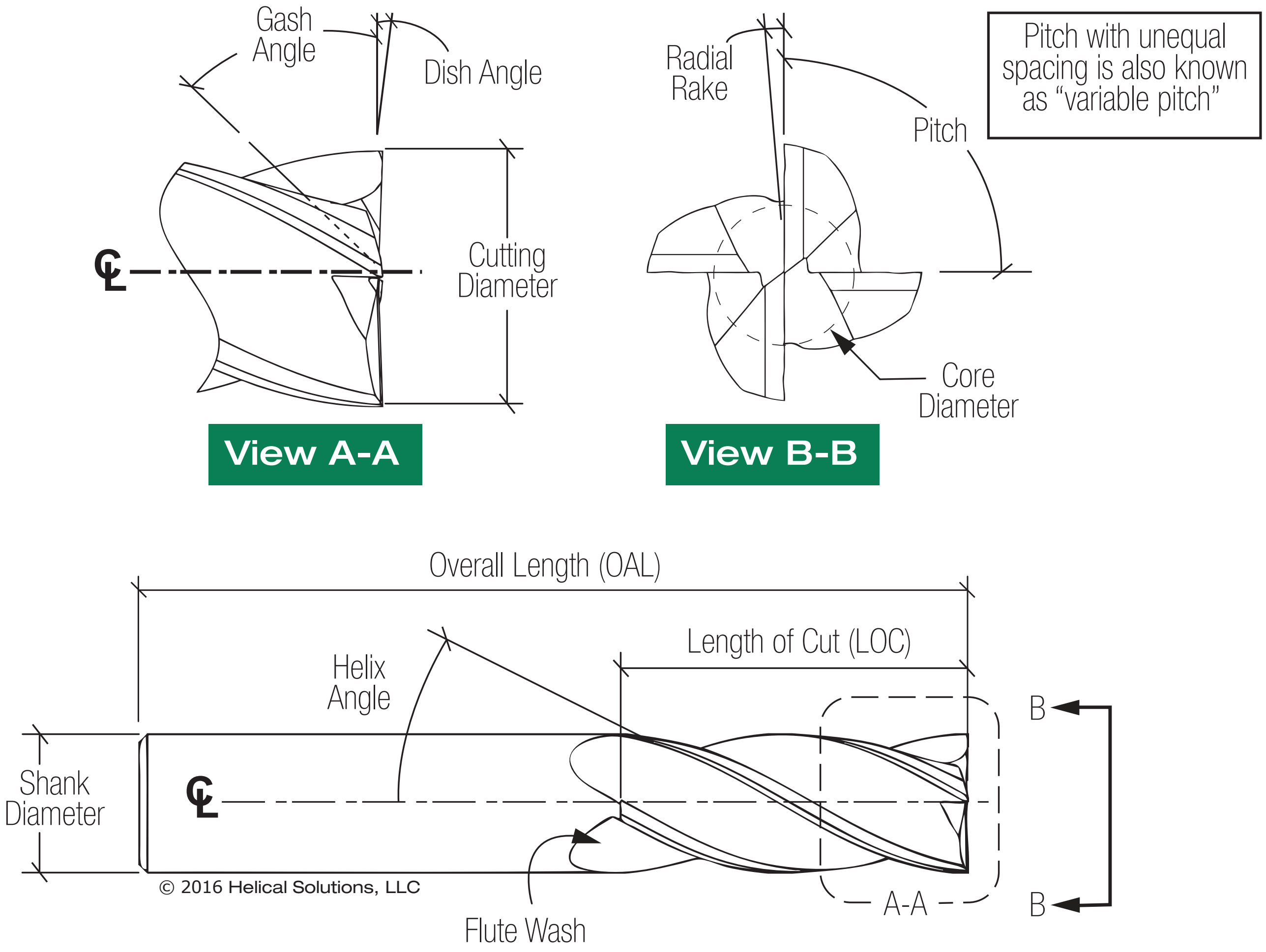
Geometry Definitions

Core Diameter	The diameter measured tangent from bottom of all flutes. This diameter dictates the strength of your end mill.
Cutting Diameter:	Measured from margin-to-margin on cutting end of tool. An even number of flutes can be measured 180° apart.
Dish Angle:	Angle perpendicular to centerline of tool and allows proper end cut characteristics - reduces full diameter contact.
Flute Wash:	Amount of non-cutting flute area past the length of cut.
Gash Angle:	The diameter measured tangent from bottom of all flutes. This diameter dictates the strength of your end mill.
Helix Angle:	This is the angle formed by a line tangent to the angle of the flute grind and parallel to the centerline of the tool.
Length Below Shank (LBS):	A length measured from front of tool back to the shank, allowing extra room for deep pocketing conditions.
Length of Cut (LOC):	This is the actual cutting depth of the tool in the axial direction.
Overall Length (OAL):	A measurement taken from end-to-end of the tool.
Cylindrical Margin:	Portion of the “uncleared” area on the peripheral area of the tool, allowing for a small area of contact with the work iece.
Pitch:	This is an equal angular measurement from flute-to-flute. If the tool is a variable pitch style then this spacing is unequal.

Geometry Definitions (cont.)

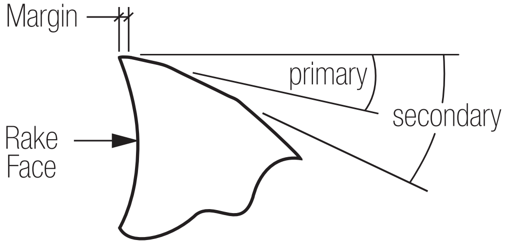
Radial Rake:	The angle of the rake face measured from center of the tool.
Radial Relief:	Area where cutting face is relieved behind the cutting edge in order to avoid rubbing, while maintaining maximum cutting tool strength.
Cylindrical:	A very effective relief for non-ferrous alloys. Includes a primary and secondary relief angle.
Eccentric:	A powerful edge design for ferrous and tough material cutting. This design includes a primary relief measured radially along its edge.
Standard:	A traditional grind allowing for moderate edge strength and high degree of primary and secondary radial relief.
Shank Diameter:	The end of the tool that is held in the holder and requires a high degree of accuracy and roundness.

End Mill Construction

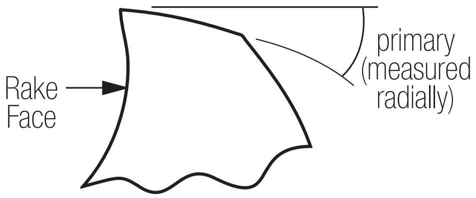


End Mill Construction (cont.)

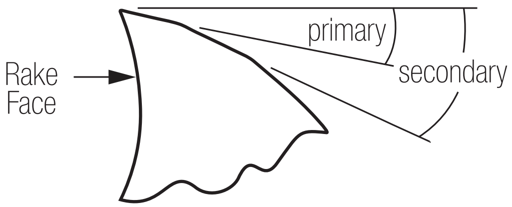
Radial Relief Types



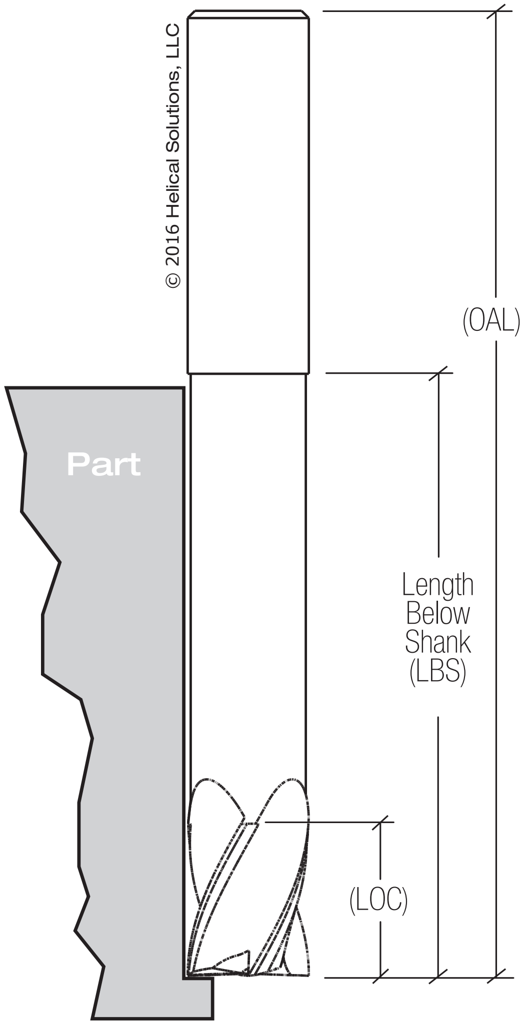
Cylindrical



Eccentric



Standard



End Mill Anatomy

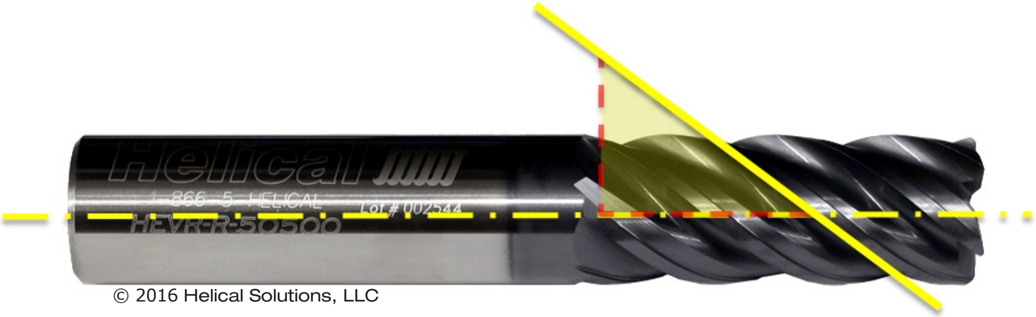
Cutting Diameter

Measurement taken from the “margin” on one flute – 180°- to the opposite margin/tooth in the first 10% of cutting diameter.

© 2016 Helical Solutions, LLC

Helix Angle

Angle formed between centerline of the tool and the edge of rake face.



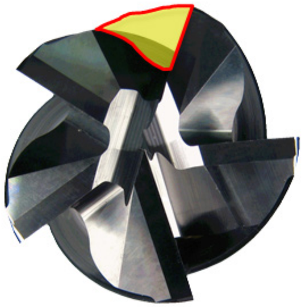
© 2016 Helical Solutions, LLC

End Mill Anatomy (cont.)

Flutes

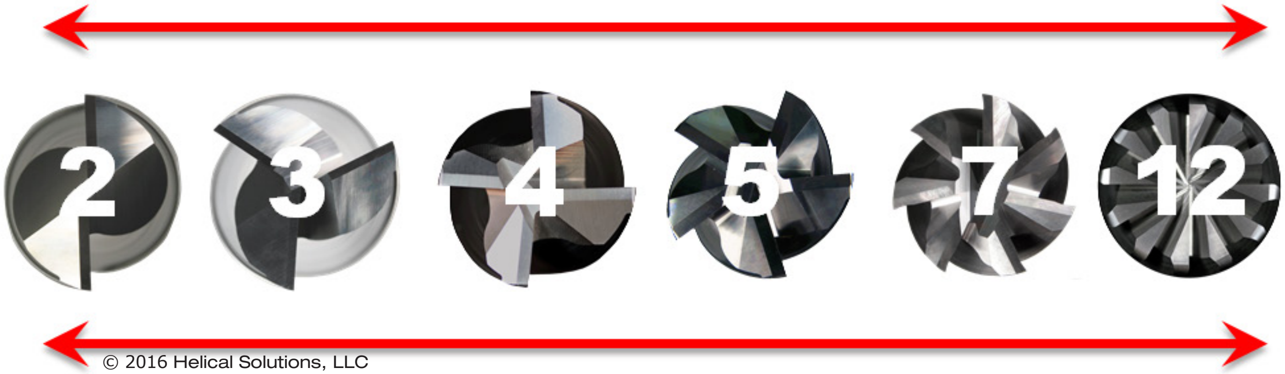
Flutes are the spiraled cutting edges of the tool that allow for chip evacuation.

© 2016 Helical Solutions, LLC



Flute Number

The number of flutes can dictate chip evacuation, tool strength, and part finish, among other things. Lower flute counts are ideal for gummy materials and heavy radial depths of cut, while higher flute counts are ideal for harder materials and light radial depths of cut.

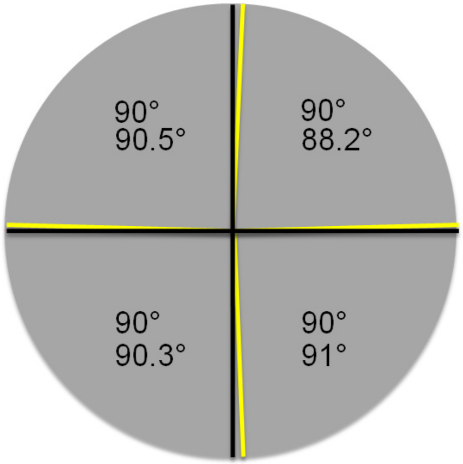


© 2016 Helical Solutions, LLC

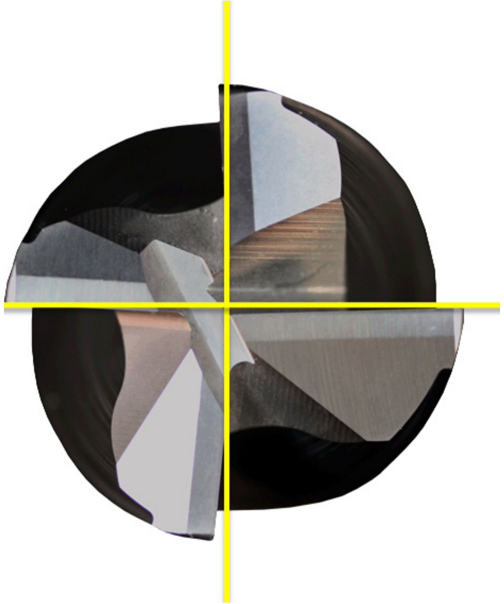
End Mill Anatomy (cont.)

Variable Pitch

While constant pitch tools are equally spaced tooling at a 90° angle, variable pitch designs have variable flute spacing. The helix angle is the same on all flutes, but the pitch varies to help break up harmonics and reduce chatter.



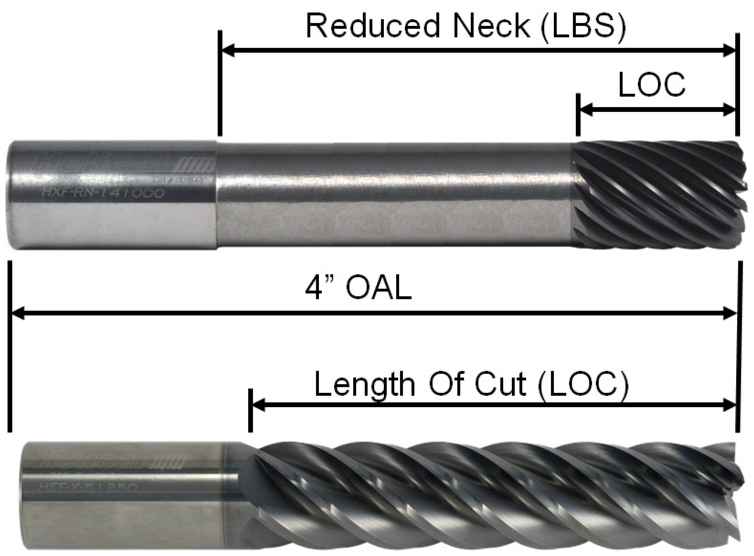
© 2016 Helical Solutions, LLC



End Mill Anatomy (cont.)

Neck Length

Measurement taken from the "margin" on one flute – 180°- to the opposite margin/tooth in the first 10% of cutting diameter.



© 2016 Helical Solutions, LLC

◀ **Stronger Choice**

End Mill Anatomy (cont.)

Core Diameter

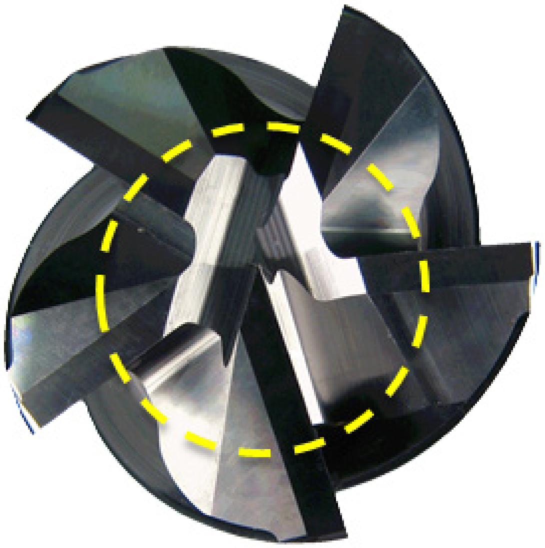
Measurement taken from bottom of one flute to bottom of opposite flute.

Smaller Core

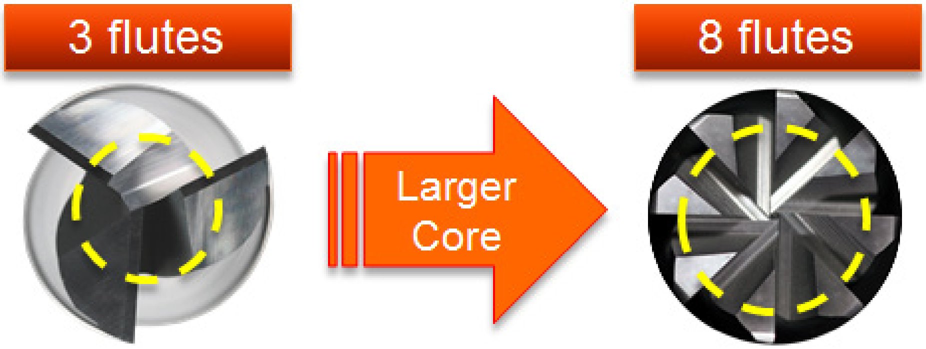
- Max. chip room
- Smaller # flutes
- Decreased tool strength

Larger Core

- Min. flute depth
- Larger # of flutes
- Increased tool strength



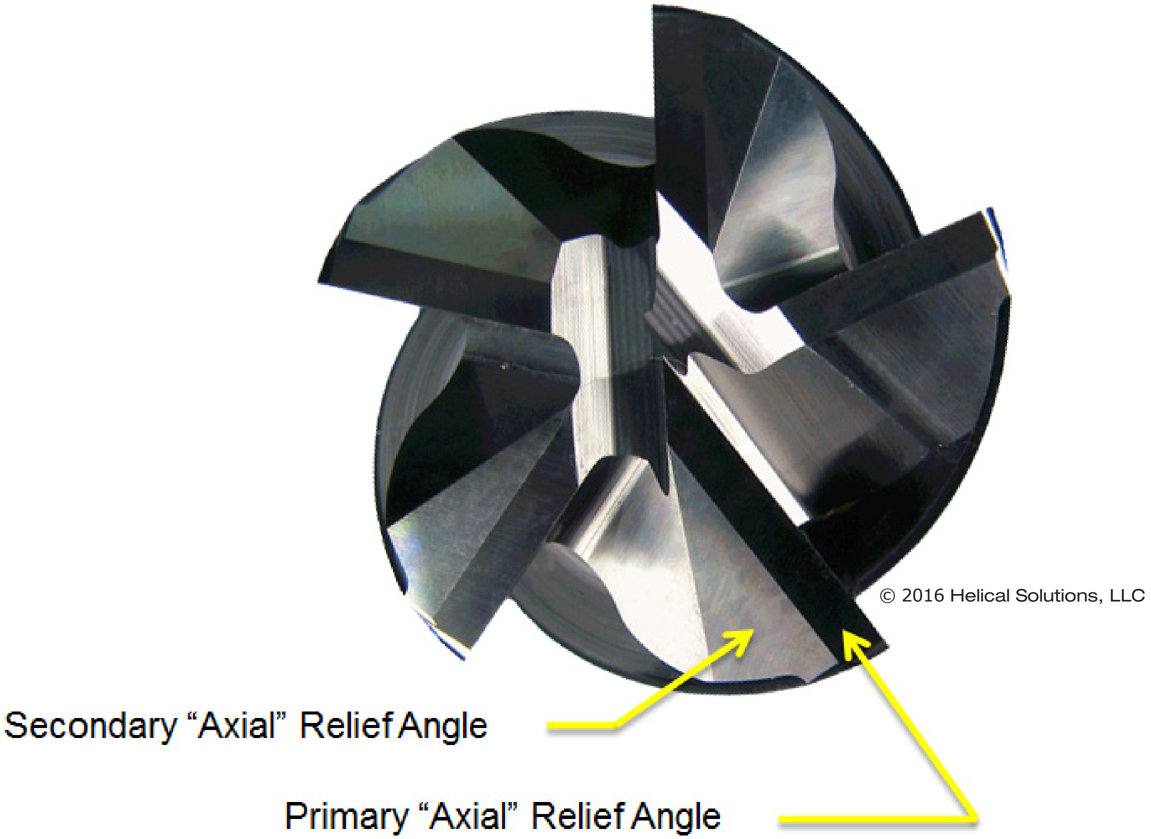
© 2016 Helical Solutions, LLC



End Mill Anatomy (cont.)

Axial Relief

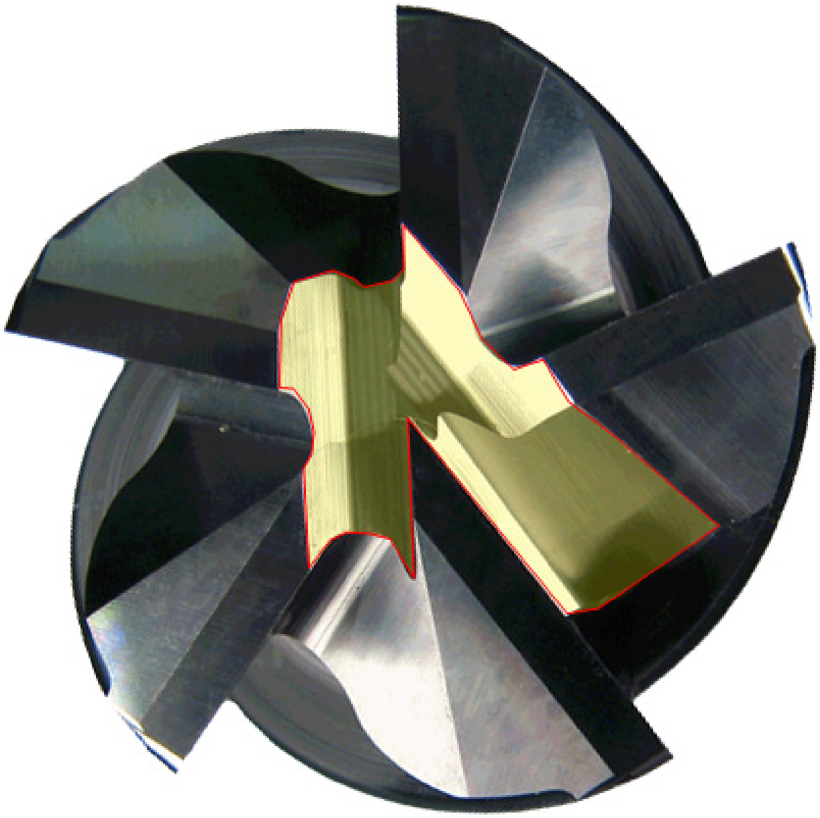
The axial relief is the relief behind the cutting teeth of the tool. They allow for the highest tooth strength without rubbing.



End Mill Anatomy (cont.)

End Gash

Establishes end clearance for center cutting operations and combines a specific width and angle for increased performance. The end gash is the traditional method of providing axial feed capabilities.

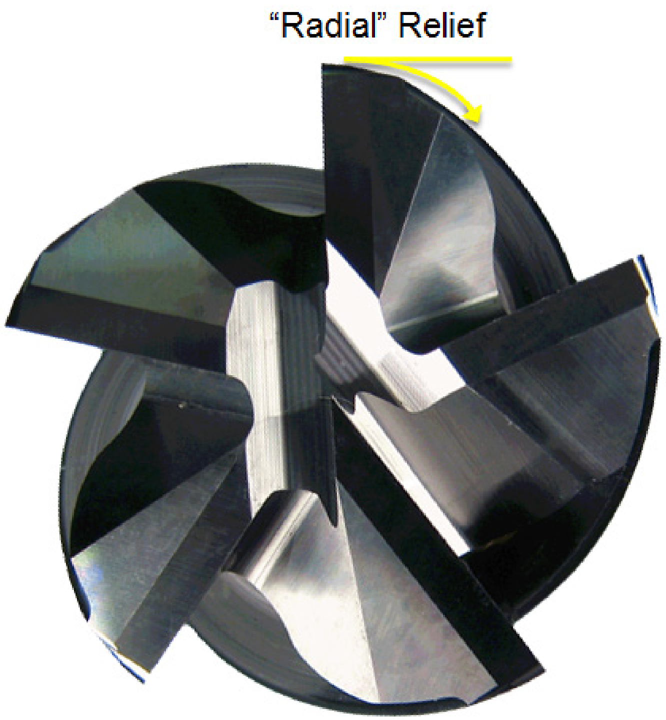


© 2016 Helical Solutions, LLC

End Mill Anatomy (cont.)

Radial Relief

Relief behind the cutting edge along the flute length. Common styles include cylindrical relief, eccentric relief, and standard relief.



© 2016 Helical Solutions, LLC

Cylindrical Relief

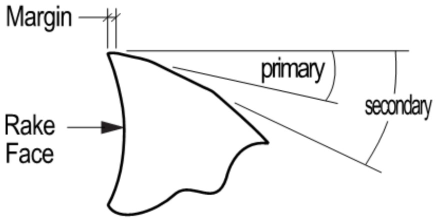
A very effective relief for non-ferrous alloys. Includes a primary and secondary relief angle.

Eccentric Relief

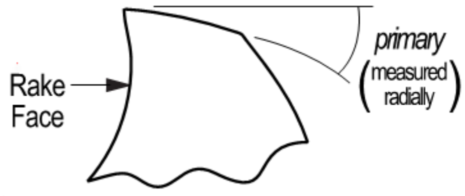
A powerful edge design for ferrous and tough material cutting. This design includes a primary relief measured radially along its edge.

Standard Relief

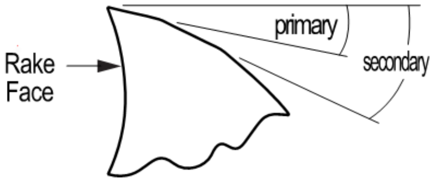
A powerful edge design for ferrous and tough material cutting. This design includes a primary relief measured radially along its edge.



Cylindrical



Eccentric

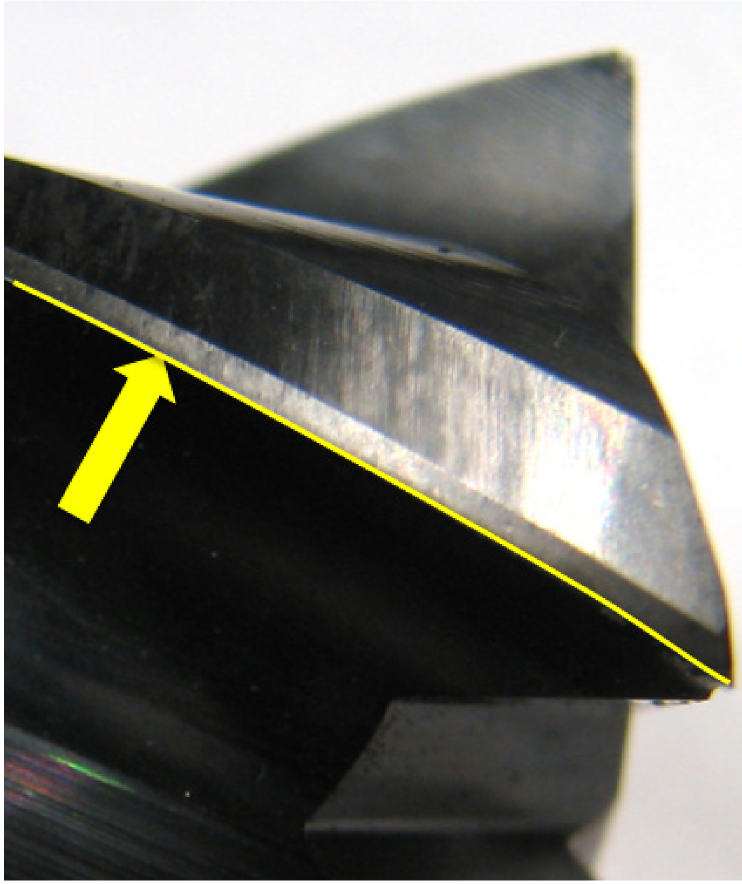


Standard

End Mill Anatomy (cont.)

Margin

The hairline area along the cutting edge at the intersection of the flute and the radial relief.The margin keeps cutting surface contact to a minimum and allows for superior edge strength and preparedness.



05

Common Calculations

Decimal Conversion Chart

Wire	mm	Inch	Decimal
	0.1		0.0039
	0.2		0.0079
	0.3		0.0118
80			0.0135
79			0.0145
		1/64	0.0156
	0.4		0.0157
78			0.0160
77			0.0180
	0.5		0.0197
76			0.0200
75			0.0210
74			0.0225
	0.6		0.0236
73			0.0240
72			0.0250
71			0.0260
	0.7		0.0276
70			0.0280
69			0.0292
68			0.0310
		1/32	0.0312
	0.8		0.0315
67			0.0320
66			0.0330
65			0.0350
	0.9		0.0354
64			0.0360
63			0.0370
62			0.0380
61			0.0390
	1.0		0.0394
60			0.0400
59			0.0410
58			0.0420
57			0.0430
56			0.0465
		3/64	0.0469
55			0.0520
54			0.0550
53			0.0595
		1/16	0.0625
52			0.0635
51			0.0670
50			0.0700
49			0.0730
48			0.0760
		5/64	0.0781
47			0.0785
	2.0		0.0787
46			0.0810
45			0.0820
44			0.0860
43			0.0890
42			0.0935
		3/32	0.0937
41			0.0960
40			0.0980
39			0.0995
38			0.1015
37			0.1040
36			0.1065
		7/64	0.1094
35			0.1100
34			0.1110
33			0.1130
32			0.1160
	3.0		0.1181
31			0.1200
		1/8	0.1250
30			0.1285
29			0.1360
28			0.1405
		9/64	0.1406
27			0.1440
26			0.1470
25			0.1495
24			0.1520
23			0.1540
		5/32	0.1562
22			0.1570
	4.0		0.1575
21			0.1590
20			0.1610
19			0.1660
18			0.1695
		11/64	0.1719
17			0.1730
16			0.1770
15			0.1800
14			0.1820
13			0.1850
		3/16	0.1875
12			0.1890
11			0.1910
10			0.1935
9			0.1960
	5.0		0.1968
8			0.1990
7			0.2010
		13/64	0.2031
6			0.2040
5			0.2055
4			0.2090
3			0.2130
		7/32	0.2187
2			0.2210
1			0.2280
A			0.2340
		15/64	0.2344
	6.0		0.2362
B			0.2380
C			0.2420
D			0.2460
		1/4	0.2500
F			0.2570
G			0.2610
		17/64	0.2656
H			0.2660
I			0.2720
	7.0		0.2756
J			0.2770
K			0.2810
		9/32	0.2812
L			0.2900
M			0.2950
		19/64	0.2969
N			0.3020
		5/16	0.3125
	8.0		0.3150
O			0.3160
P			0.3230
		21/64	0.3281
Q			0.3320
R			0.3390
		11/32	0.3437
S			0.3480
	9.0		0.3543
T			0.3580
		23/64	0.3594
U			0.3680
		3/8	0.3750
V			0.3770
W			0.3860
		25/64	0.3906
	10.0		0.3937
X			0.3970
Y			0.4040
		13/32	0.4062
Z			0.4130
		27/64	0.4219
	11.0		0.4331
		7/16	0.4375
		29/64	0.4531
		15/32	0.4687
	12.0		0.4724
		31/64	0.4844
		1/2	0.5000
	13.0		0.5118
		33/64	0.5156
		17/32	0.5312
		35/64	0.5469
	14.0		0.5512
		9/16	0.5625
		37/64	0.5781
	15.0		0.5906
		19/32	0.5937
		39/64	0.6094
		5/8	0.6250
	16.0		0.6299
		41/64	0.6406
		21/32	0.6562
	17.0		0.6693
		43/64	0.6719
		11/16	0.6875
		45/64	0.7031
	18.0		0.7087
		23/32	0.7187
		47/64	0.7344
	19.0		0.7480
		3/4	0.7500
		49/64	0.7656
		25/32	0.7812
	20.0		0.7874
		51/64	0.7969
		13/16	0.8125
	21.0		0.8268
		53/64	0.8281
		27/32	0.8437
		55/64	0.8594
	22.0		0.8661
		7/8	0.8750
		57/64	0.8906
	23.0		0.9055
		29/32	0.9062
		59/64	0.9219
		15/16	0.9375
	24.0		0.9449
		61/64	0.9531
		31/32	0.9687
	25.0		0.9842
		63/64	0.9844
	25.4	1	1.0000

Common Milling Calculations

Revolutions Per Minute	$RPM =$	$\frac{SFM \times 3.82}{D}$
Surface Feet Per Minute	$SFM =$	$RPM \times D \times .262$
Inches Per Minute	$IPM =$	$RPM \times IPT \times Z$
Inches Per Revolution	$IPR =$	$\frac{IPM}{RPM}$
Inches Per Tooth	$IPT =$	$\frac{IPR}{Z}$
Inches Per Tooth (Chip Thinning Adjustment)	$IPT_{adj} =$	$\frac{CT \times D}{2 \times \sqrt{(D \times RDOC) - RDOC^2}}$
Chip Thickness	$CT =$	$\frac{2 \times IPT \times \sqrt{(D \times RDOC) - RDOC^2}}{D}$
Metal Removal Rate (cu. in./min.)	$MRR =$	$RDOC \times ADOC \times IPM$
Feed Rate Adjustment - Outside Arc	$F_o =$	$\frac{IPM \times (r_o + R)}{r_o}$
Feed Rate Adjustment - Inside Arc	$F_i =$	$\frac{IPM \times (r_i + R)}{r_i}$
Ball Nose “Effective Diameter”	$D_{eff} =$	$2 \times \sqrt{ADOC \times (D - ADOC)}$
Ball Nose Velocity Adjustment	$V_{adj} =$	$\frac{SFM \times 3.82}{D_{eff}}$

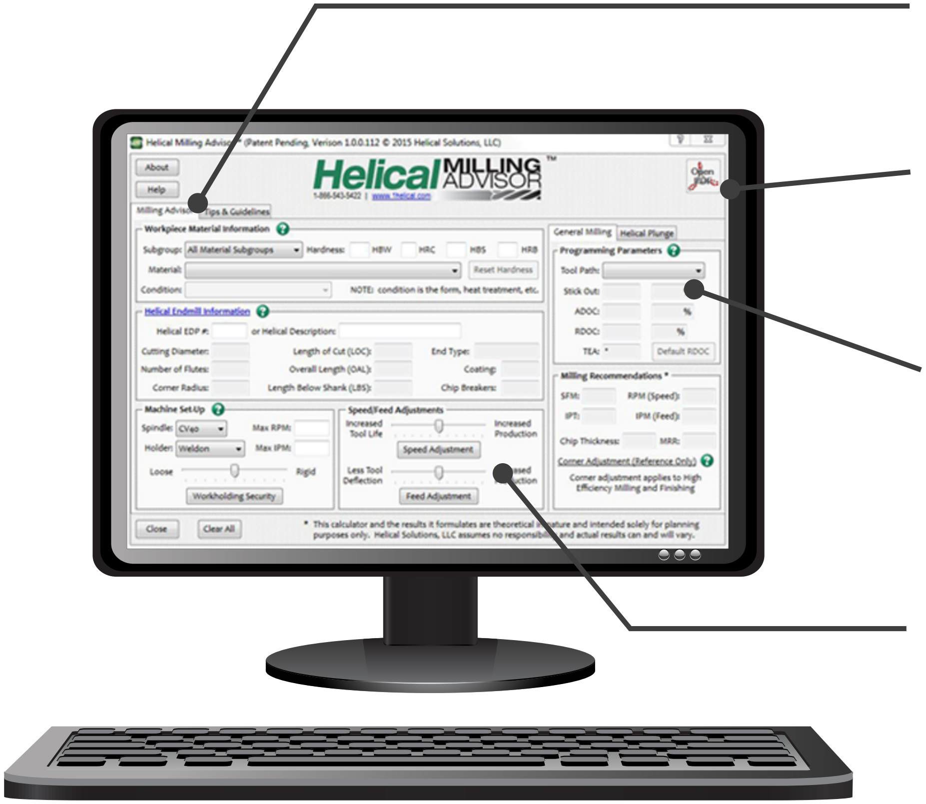
KEY

D	Tool Cutting Diameter
Z	Number of Flutes
RPM	Revolutions per Minute
SFM	Surface Feet per MInute
IPM	Inches per Minute
IPR	Inches per Revolution
IPT	Inches per Tooth
IPT_{adj}	Inches per Tooth (adjusted)
CT	Chip Thickness
RDOC	Radial Depth of Cut
ADOC	Axial Depth of Cut
MRR	Metal Removal Rate
r_i	Part Radius (inside arc)
r_o	Part Radius (outside arc)

Speeds & Feeds

Calculating Speeds & Feeds is easy with Helical Milling Advisor™. Designed to calculate optimal milling parameters, this free downloadable app helps users get the most out of Helical end mills. With over 300 material grades and conditions, you get the recommendations you need, all while working within the capability of your machine tool and set up. With easy to use feed and speed adjustment sliders and an extensive tips and guidelines section, optimal cutting parameters have never been easier to calculate.

Helical™
MILLING ADVISOR



Helical Milling Advisor™ (Patent Pending, Verison 1.0.0.112 © 2015 Helical Solutions, LLC)

Subgroup: All Material Subgroups

NOTE: condition is the form, heat treatment, etc.

Spindle: CV40

Holder: Weldon

Corner adjustment applies to High Efficiency Milling and Finishing

* This calculator and the results it formulates are theoretical in nature and intended solely for planning purposes only. Helical Solutions, LLC assumes no responsibility and actual results can and will vary.

Over 300 material grades/conditions to work with

Generate a PDF summary of recommended parameters

Access technical information related to materials, Helical endmills, and more

Quickly adjust Speeds & Feeds to optimize your machining conditions

© 2016 Helical Solutions, LLC

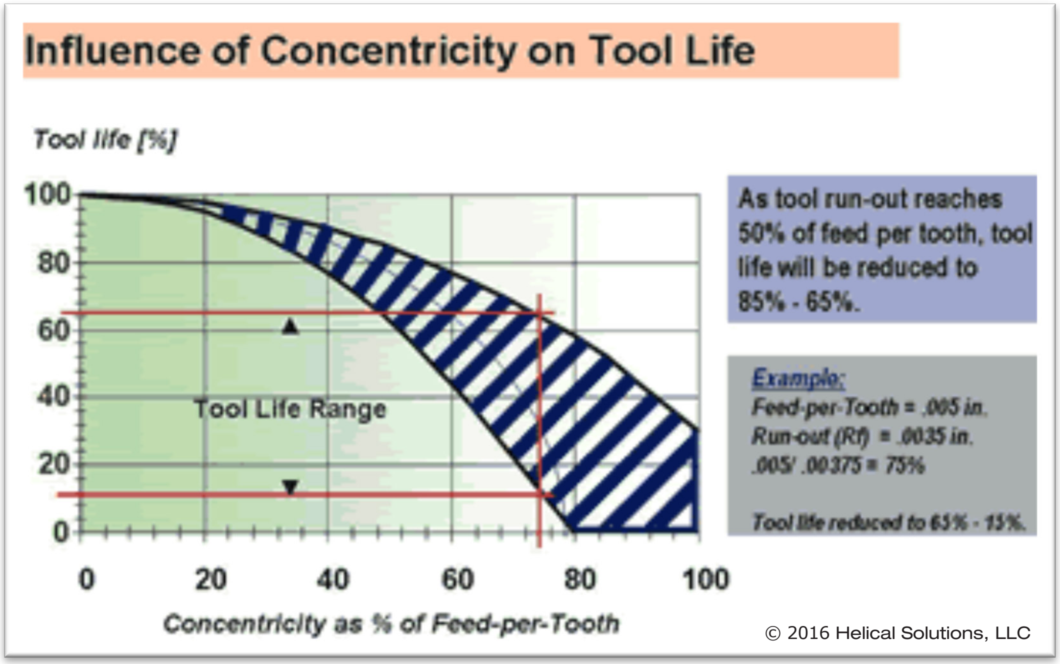
06

Tool Holding

Tool Holding

Tool Holders

- Use the right holder
- Keep holders clean and in good shape
- Don’t maximize ER collets.
- Check for cracked collet nuts
- Correct pull studs



Influence of Concentricity on Tool Life

As tool run-out reaches 50% of feed per tooth, tool life will be reduced to 85% - 65%.

Example:
Feed-per-Tooth = .005 in.
Run-out (Rf) = .0035 in.
.005/ .00375 = 75%

Tool life reduced to 65% - 15%.

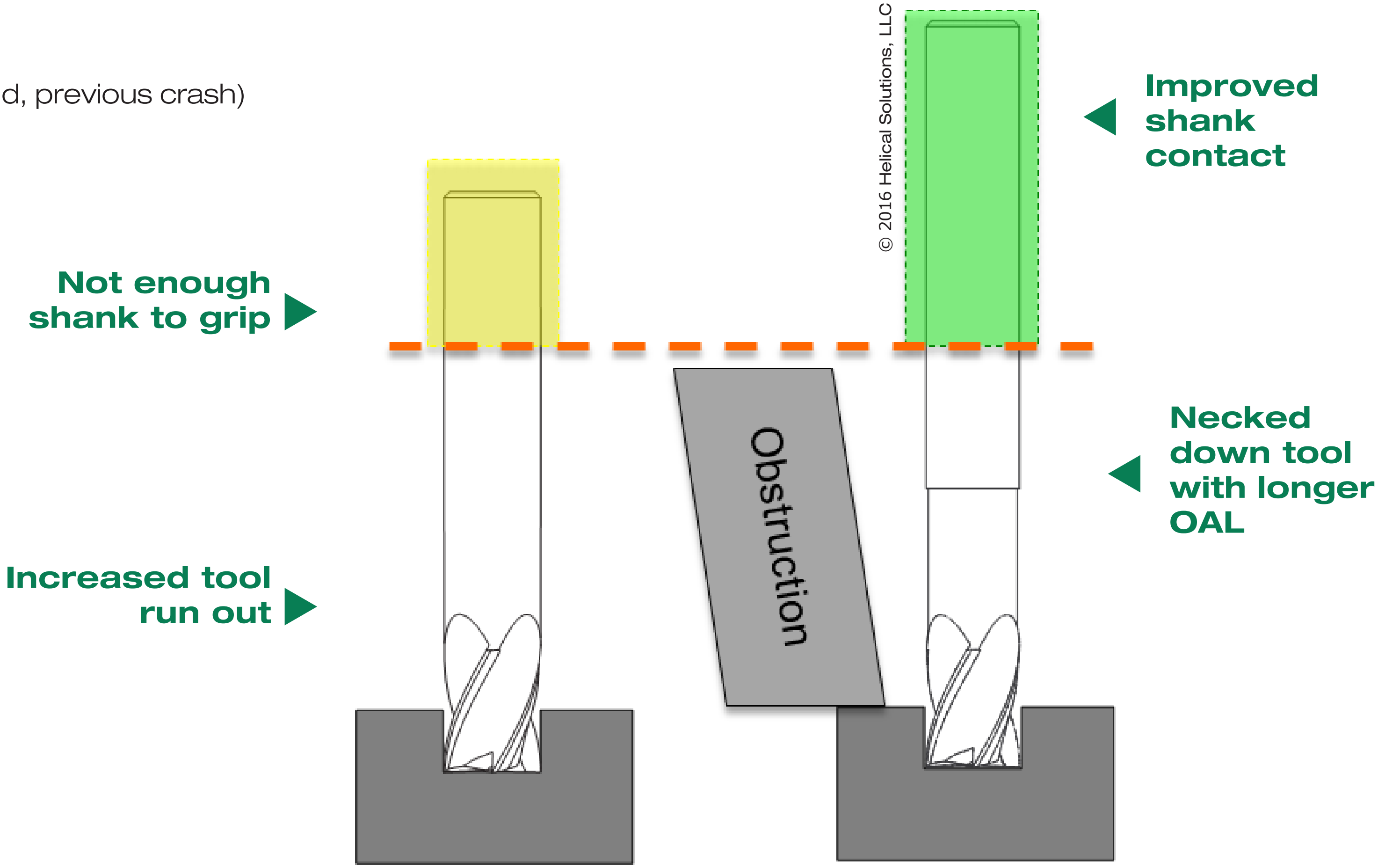
Tool Holding (cont.)

Run Out

- Lowers tool life
- Decreases part finishes
- Tool life decreases 10% for every .0001 TIR

Usual Suspects

- Spindle/taper condition
- Drawbar/backlash
- Tool Holder (wrong pull stud, previous crash)
- Short tool shank grip



Not enough shank to grip

Increased tool run out

Improved shank contact

Necked down tool with longer OAL

Preventing Tool Pull Out

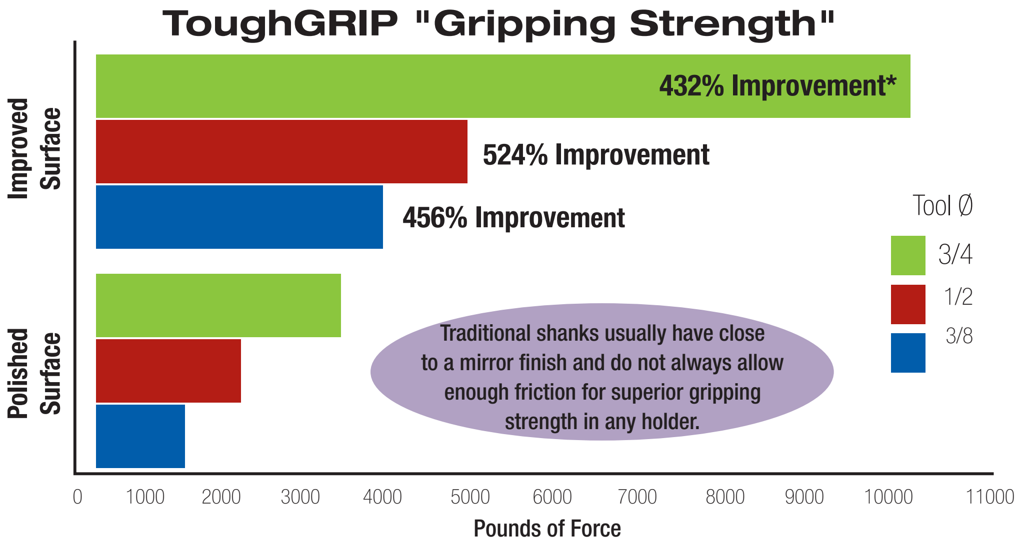
Tool Pull Out

- Experiencing slow micro-creeping and/or full tool pullout.
- Cutting parameters, traditionally, are reduced to counter this effect.

Helical's ToughGRIP Shank

Experience Helical Solutions's ToughGRIP shank and see for yourself the stronger tool-to-holder connection you may be looking for!

- Provides increased friction for superior gripping strength
- Maintains shank concentricity and h6 shrink-fit tolerance
- 30 +/- 3 Ra surface roughness, consistent within 2 Ra



Traditional shanks usually have close to a mirror finish and do not always allow enough friction for superior gripping strength in any holder.

Based on independent laboratory testing. All tests were completed utilizing Command Tooling Systems' line of HYDRO-GRIP® HD Hydraulic tool holders by ETP Transmission.

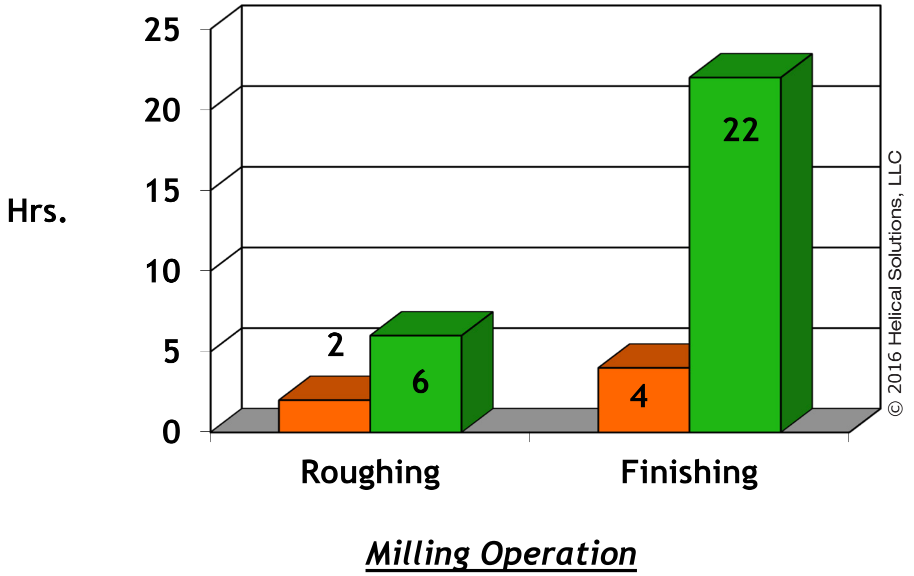
* Improved surface finish from the ToughGRIP exceeded laboratory testing equipment

Preventing Tool Pull Out (cont.)

Haimer Safe-Lock™ System

- Drive keys in the chuck and grooves in the tool shank prevent pull-out
- Accurate clamping due to shrink fit technology
- Can be applied to our standard tools or special tools
- Can accommodate tools from 1/2” to 2” diameter

HAIMER®



Helical Solutions will modify the shank of any of our tools to meet the specifications for the Haimer Safe-Lock™ System.

07

Troubleshooting

Troubleshooting Chart

The Problem	Possible Causes	Possible Solution
Breakage	Work piece rigidity	Ensure work piece is secure and supported - a common issue
	Speed too low	Increase the cutting speed (RPM's)
	Feed rate too high	Reduce IPT
	Chip compaction	Reduce MRR
	Heavy depth of cut	Reduce RDOC & ADOC
	Part Entry	Reduce IPT on entry - implement radius in or sweeping entrances - avoid 90° (perpendicular) entry
	Milling Strategy	Review tool path and ensure there are no arbitrary moves, extreme angle of engagement increases & undesirable situations for the tool.
	Tool Overhang	Use shortest OAL, shortest LOC & reduce overhang from tool holder. Consider necked down tooling for long reach.
	Tool Runout	Check tool runout in holder/spindle. Utilize collet, milling chuck or shrink fit holders if possible. Hand ground shank flats can be suspect and a common cause of run out. (<.0003 TIR desired)
	Built up edge (BUE)	See the BUE section to increase IPT, utilize tool coatings
	Reconditioning	Improper regrind/reconditioning – try factory service
Excessive Wear (Flank)	Speed too high	Reduce the cutting speed (RPM's)
	Feed rate too low	Increase feed rate (IPT)
	RDOC too high	Lessen RDOC as % of dia. - start with 10% reduction increments
	Chip Thinning	Utilize chip thinning adjustment
	Tool Runout	Check tool run out in holder/spindle. Utilize collet, milling chuck or shrink fit holders if possible. Hand ground shank flats can be suspect and a common cause of runout. (<.0003 TIR desired)
	Recutting Chips	Re-adjust coolant flow, air blast or "op stop" to clear chip build up
	Milling Strategy	Ensure you are climb milling unless the material has hard/abrasive outer skin then convention milling technique is preferred for breakthrough.
	Tool Coating	Ensure you have the appropriate coating for material being cut
	Hard Materials (> than 55Rc)	Try 90-100 SFM with multi-fluted tool (5 flutes +)

Troubleshooting Chart (cont.)

The Problem	Possible Causes	Possible Solution
Chipped Cutting Edge	Work piece rigidity	Check work piece is secure and supported - a common issue
	Tool holder rigidity	Use shortest holder possible and investigate for no tool slippage
	Feed rate too high	Reduce IPT
	Tool Heavy of a RDOC	Reduce RDOC
	Part Entry	Reduce IPT on entry – implement radius in or sweeping entrances - avoid 90° (perpendicular) entry
	Milling Strategy	Ensure you are climb milling unless the material has hard/abrasive outer skin - then conventional milling technique is preferred for breakthrough
	Tool Overhang	Use shortest OAL, shortest LOC & reduce overhang from tool holder. Consider necked down tooling for long reach.
	Tool Run out	Check tool run out in holder/spindle. Utilize collet, milling chuck or shrink fit holders if possible. Hand ground shank flats can be suspect and a common cause of run out. (<.0003 TIR desired)
	Tool Coating	Implement proper tool coating for material to be cut
	Edge prep	Ensure tool has proper edge prep
	Built Up Edge (BUE)	See BUE section to increase IPT, utilize tool coatings
Excessive Corner Wear	No Corner Radius	Implement corner radius on tool - adds strength & tool life
	Speed too high	Reduce RPM's
	Tool Run out	Check tool run out in holder/spindle. Utilize collet, milling chuck or shrink fit holders if possible. Hand ground shank flats can be suspect and a common cause of run out. (<.0003 TIR desired)
	Tool Overhang	Ensure you are using the shortest OAL/LOC possible. Utilize necked tooling for longer reach.

Troubleshooting Chart (cont.)

The Problem	Possible Causes	Possible Solution
Chip Compaction	Insufficient chip room	Reduce number of flutes
	Feed rate too high	Reduce IPT and increase RPM
	Heavy depth of cut	Reduce ADOC/RDOC in side milling & ADOC in Slotting
	Coolant flush	Re-adjust coolant flow, air blast or "op stop" to clear chip build up
	Heavy depth of cut	Reduce RDOC & ADOC
	Large chip size	Utilize chip breaker style tool to better manage chip size
Built up Edge (BUE)	Chip welding	Utilize proper tool coating for material being cut
	Feed rate too low	Increased IPT
	Speed too low	Increase RPM's
	Coolant Strategy	Re-adjust coolant flow & check coolant mixture percentage
Chatter/ Vibration	Work piece rigidity	Check work piece is secure and supported
	Tool holder rigidity	Use shortest holder possible and investigate for no tool slippage
	Tool Overhang	Use shortest length tool, shortest loc, and reduce overhang from tool holder. Consider necked down tooling for long reach.
	Tool Run out	Check tool run out in holder/spindle. Utilize collet, milling chuck, or shrink fit holders if possible. Hand ground shank flats can be suspect and a common cause of run out. (<.0003 TIR desired)
	Chip Thinning	Utilize chip thinning adjustment
	Speed too high	Lower the RPM's
	Feed rate too low	Increased IPT
	Angle of engagement violation	Use smaller tools generating corner radi in pockets - avoid tool diameters that match corner dia./radius.
	Too much surface contact	Try utilizing a lower flute count tool
	Milling Strategy	Ensure you are climb milling unless the material has hard/abrasive outer skin then convention milling technique is preferred for breakthrough.

Troubleshooting Chart (cont.)

The Problem	Possible Causes	Possible Solution
Poor Surface Finish	Feed rate too high	Reduce IPT
	Speed too low	Increase RPM's
	Too light of a RDOC	Increase RDOC to stabilize tool in cut.
	Tool Run out	Check tool run out in holder/spindle. Utilize collet, milling chuck, or shrink fit holders if possible. Hand ground shank flats can be suspect and a common cause of run out. (<.0003 TIR desired)
	Helix Angle	Change to tool with higher helix angle.
	Need more Flutes	Choose end mill with higher number of flutes
	Recutting Chips	Redirect/evaluate coolant flush – or use less number of flutes
	Built Up Edge	Increase IPT - Increase RPM - Utilize tool coatings
Deflection	Tool Overhang	Use shortest length tool, shortest loc, & reduce overhang from tool holder.
	Milling Strategy	Climb milling can help reduce the amount of deflection in some cases.
	Too heavy of a RDOC	Reduce ADOC/RDOC in side milling & ADOC in slotting
	Feed rate too high	Decrease IPT
	End Mill Diameter	Increase diameter of end mill for higher strength-to-length ratio
	Increase Number of Flutes	Higher number of flutes = larger core diameter = increased strength
Dimension Accuracy (Tapered Wall)	Coolant Strategy	Re-adjust coolant flow & check coolant mixture percentage
	Deflection	Refer to deflection section above
	Feed rate too high	Lower feed rate (IPT)
	RDOC too high	Reduce RDOC
	Tool Run out	Check tool run out in holder/spindle. Utilize collet, milling chuck, or shrink fit holders if possible. Hand ground shank flats can be suspect and a common cause of run out. (<.0003 TIR desired)

Tool Wear

4 Types of Tool Wear

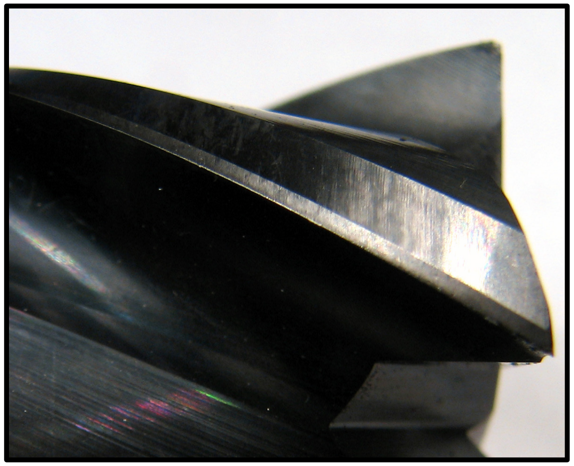
Tool wear describes the gradual failure of cutting tools due to regular operation. There are 4 different types: Wear Land, Chipping, Thermal Cracking, and Fractures.

Wear Land

Recognition: The wear land is a pattern of uniform abrasion on the cutting edge of the tool.

Causes: Mechanical abrasion and/or chemical wear from the work piece material rubbing against the cutter. Chemical wear dominates at higher speeds.

Remedies: If the wear land becomes excessive or causes a premature tool failure, reduce cutting speed and/or optimize coolant usage.

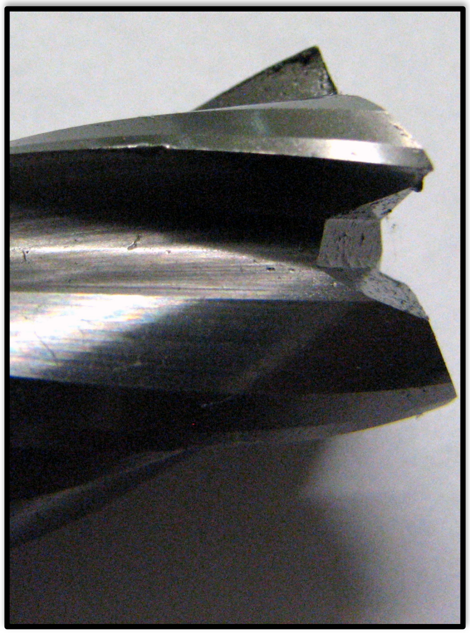


Chipping

Recognition: Chipping can be identified by a nicked or flaked edge on the cutting tool or by the surface finish of the part. Micro chipping promotes irregular wear and bad surface finish. Micro chipping can lead to catastrophic tool failure.

Causes: Chipping can be caused by excessive loads in the metal cutting operation or thermal cracking.

Remedies: Ensure operation is rigid and free of vibration or chatter, Decrease feed rates and/or increase speed to decrease mechanical load on cutting tool which reduces chip load.



Tool Wear (cont.)

Chipping

Recognition: In milling, cracks perpendicular to cutting edge are the most common types of thermal cracks. These cracks propagate slowly and lead to accelerated wear and chipping.

Causes: Ineffective coolant application exaggerates temperature fluctuations. Excessive feed rates aggravate thermal fluctuations.

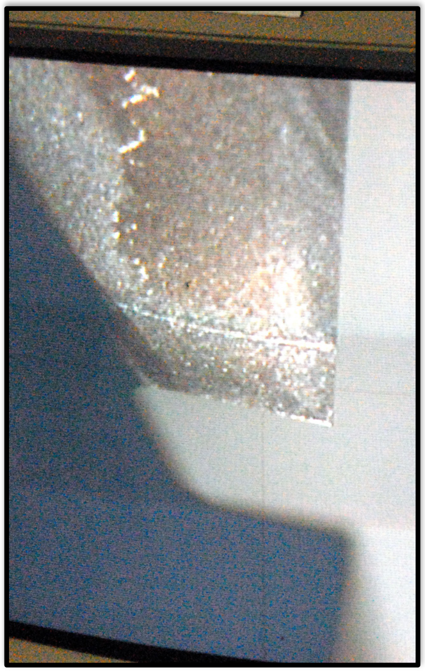
Remedies: Add coating, reduce feed rate, remove coolant – run dry.

Fracture

Recognition: Fracture is the loss of the cutting edge due to sudden breakage.

Causes: Improper selection of speed, feed, depth of cut, or coating. Loose work holding or tool holder issues (pull stud, dirty taper, high run out, tool spun in bore etc). Work material inconsistencies such as presence of hot spots, inclusions, or voids in castings.

Remedies: Reduce speed, feed, and/or depth of cut. Feed will be most influential. Check the setup. Make sure it is as rigid as possible & also check for source of chatter and vibration which can cause fracture. Optimize coolant usage. If coolant is used, be sure it is effectively reaching the entire cutting zone before and after the cut.



Tool Wear (cont.)

Thermal Stresses

- Workpiece inconsistencies, work hardening, heat resistant material advancements
- Rubbing, perhaps Under-Fed condition
- Work hardening changing SFM effectiveness

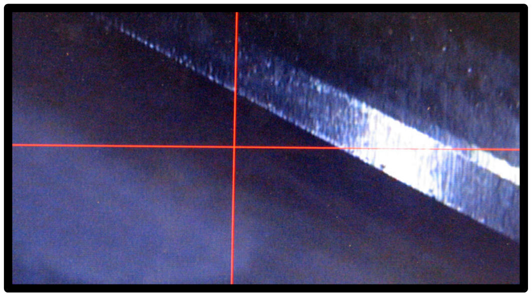
Mechanical Stresses

- Extreme tangential cutting forces
- Abrasion, Chip Congestion, Built up Edge
- Suspect Tool Holding or Work Holding
- Unmanageable Metal Removal Rate (MRR)

Tool Wear (cont.)

Initial Wear Period

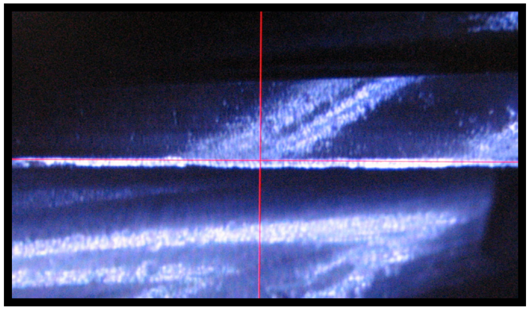
- Completely normal and can vary on length. Usually only the first few minutes of cutting and results in moderate and controlled wear rate, breaking in the cutting edge.



© 2016 Helical Solutions, LLC

Intermediary wear period

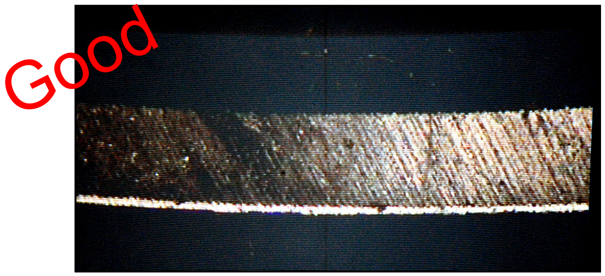
- Usually entails about 70-80% of the cutting life under normal conditions.
- Tool monitoring program is of utmost importance.
- Machine load meter monitored for any tool wear increase.

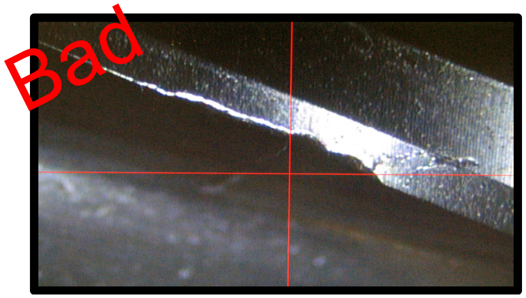


© 2016 Helical Solutions, LLC

Completion Period

- If using proper tool change program, tool will be salvageable.
- Commonly determined by catastrophic failure.





© 2016 Helical Solutions, LLC

Tool Deflection

Tool Deflection

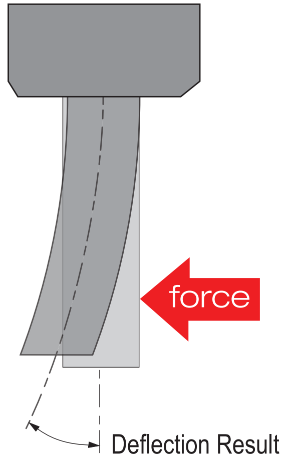
Rigidity during a milling operation is key for optimal tool performance and desired results. Keeping tool deflection to a minimum will help increase success on a deep reach application.

A Deflection "Rule of Thumb"

Tool overhang length decreases rigidity as a third power (L3), but even more importantly, tool diameter increases rigidity by the fourth power (D4).

Common Techniques to Combat Deflection

- Ensure tools are sharp
- Increase tool diameter
- Decrease depth of cut
- Climb mill in lieu of conventional milling
- Decrease IPM
- Use shorter tool and/or employ necked tooling
- Increase number of flutes
- Re-evaluate SFM parameter



Helical

Manufacturers of High Performance Carbide Cutting Tools

2016
PRODUCT
CATALOG

QUALITY

INNOVATION

PERFORMANCE

www.helicaltool.com • 866-543-5422

2016 Helical Product Catalog

The 2016 catalog introduces two brand new product innovations: High Performance Chamfer Mills offered in 3 & 5 flutes with 60°, 90°, and 120° options and the new 3 Flute Variable Pitch End Mill for Steels. The new catalog's sleek layout and product search features will make finding the right Helical tool for you easier than ever.

Download our catalog or request a hard copy today!

Download Catalog **Request Copy**

Contact Us

Our technical support representatives and application engineers are always available to walk you through tool selection, application support, and troubleshooting.

CALL 866-543-5422

EMAIL orders8-5@1helical.com

BROWSE www.helicaltool.com