# Helical /////
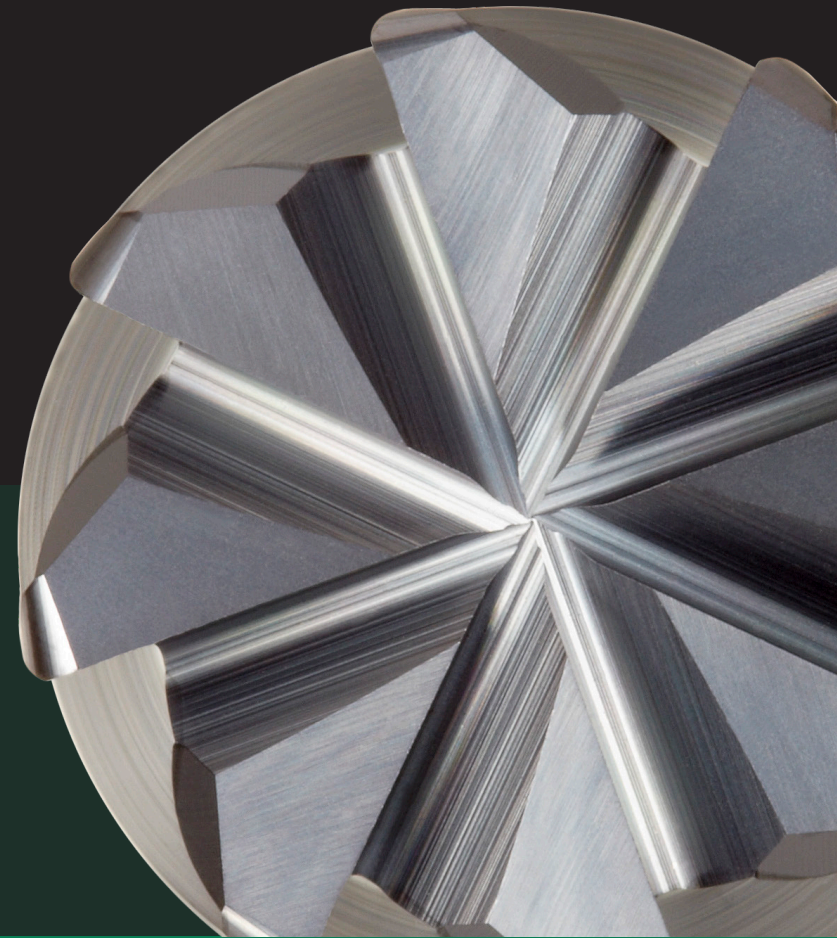
# MACHINING GUIDEBOOK

Quick Reference eBook for CNC Milling Practices & Techniques

# Contents

## Milling Techniques & Strategies

## Terminology & Common Calculations

## Troubleshooting

*Helical*

# 01
# Milling Techniques

*Helical*

# 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.**
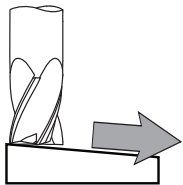
### Ramping-In

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.
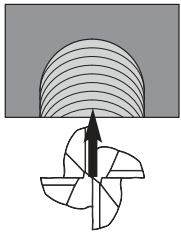
## Straight Entry

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%.
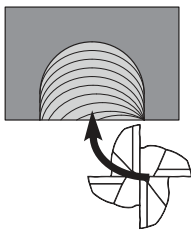
## Roll-In Entry

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.
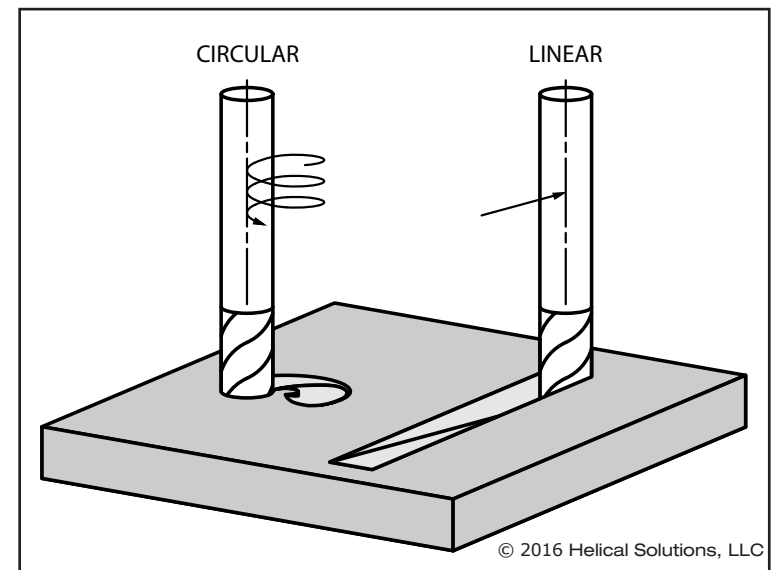
Figure 1: Circular & Linear Ramping



© 2016 Helical Solutions, LLC

## 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.

helical

# 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



© 2016 Helical Solutions, LLC

Helical

# 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.



Detail A

Start with stub/reg length tooling

Relieved neck will not rub on wall

This is the tool's LBS length
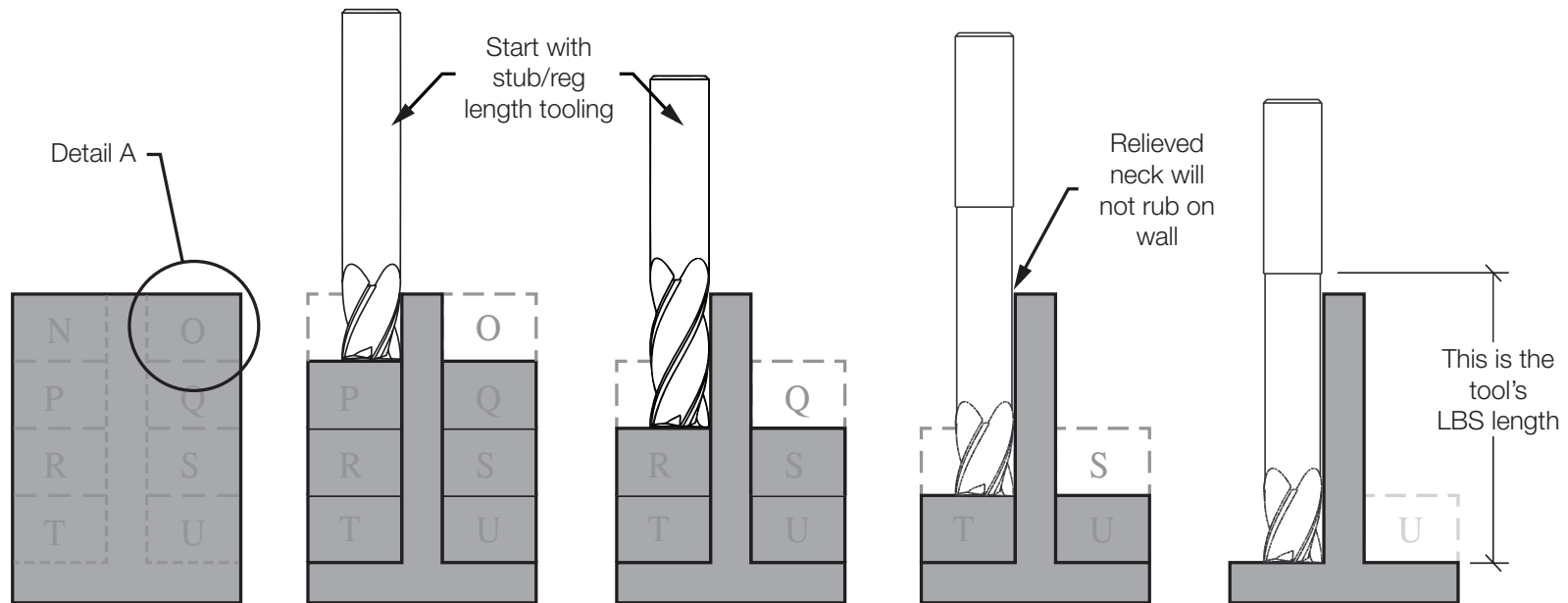
© 2016 Helical Solutions, LLC

# 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.

## 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.

**Detail A**

1st RDOC - Up to 50% cutter dia.

2nd  RDOC - 30% cutter dia.

3rd RDOC - 15% cutter dia.

4th RDOC - 5% cutter dia.
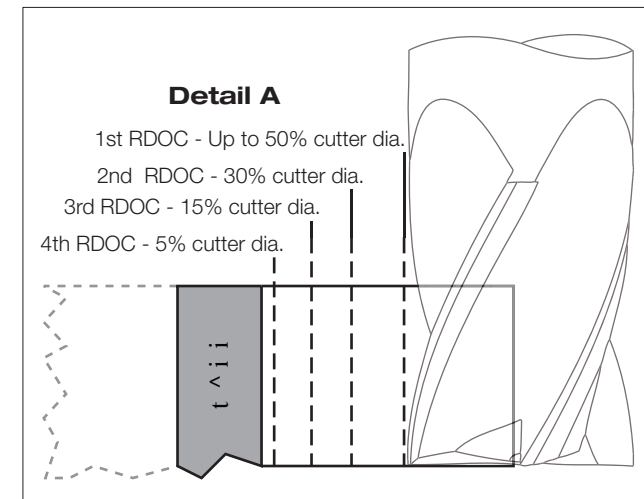
© 2016 Helical Solutions, LLC

# 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).
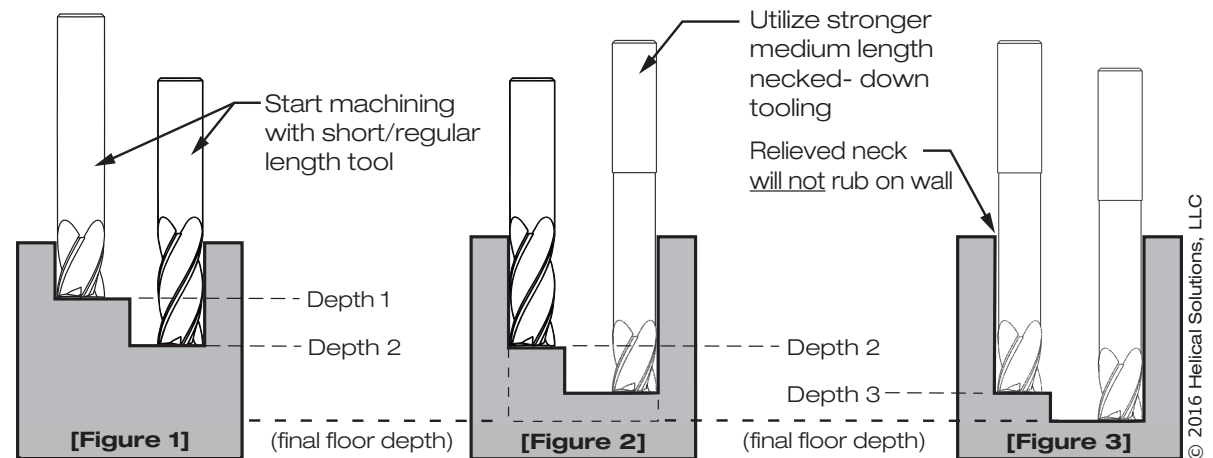
## Common Problems Experienced
- Chatter
- Wall taper
- Tool deflection
- Tool engagement violations
- Recutting chips
- Breakage

Start machining with short/regular length tool

Utilize stronger medium length necked- down tooling

Relieved neck will not rub on wall

Depth 1

Depth 2

Depth 2

Depth 3

[Figure 1]          (final floor depth)          [Figure 2]          (final floor depth)          [Figure 3]

© 2016 Helical Solutions, LLC

## 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.

Helical

# 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.
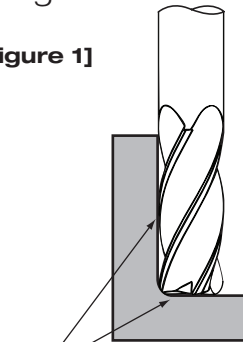
## Factors that Influence Finish

- 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

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

## 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.

© 2016 Helical Solutions, LLC

75% of LOC
75% of LOC
75% of LOC

[Figure 2]

Helical

# 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) = $R_a$ x 1.1
- $R_z$ = $R_a$ x 3.1

32

$R_a$ = µin = .000032 in.

part
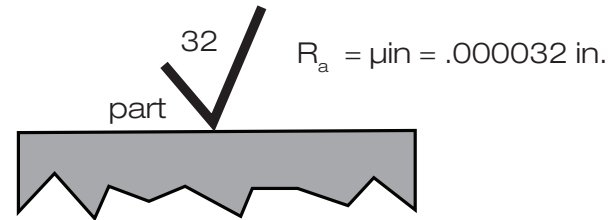
## QUICK REFERENCE GUIDE

| To improve surface finish | Increase RPM | Lower IPT | Climb Mill |

© 2016 Helical Solutions, LLC

*Helical*

# 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.
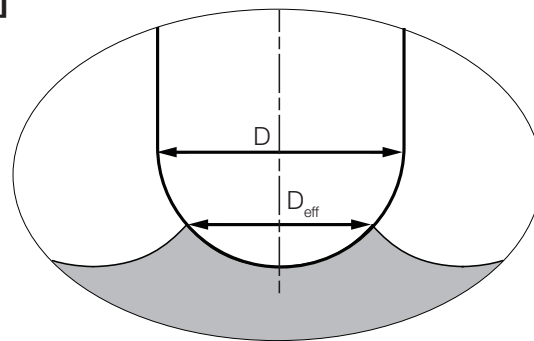
[Detail A]

[Figure 1]

D

$D_{eff}$

ADOC

$D_{eff}$

© 2016 Helical Solutions, LLC

*Helical*

## 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]**

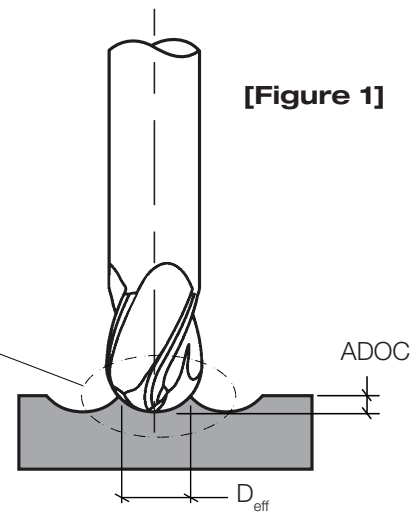| | | AXIAL DEPTH OF CUT (ADOC) | | | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | | .010 | .020 | .030 | .050 | .070 | .090 | .120 | .150 | .180 | .220 | .260 | .300 | .350 | .400 | .450 |
| TOOL DIAMETER | 1/8 | .068 | .092 | .107 | .122 | .124 | .112 | .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 |

© 2016 Helical Solutions, LLC

**[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]**

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

*Helical*

# 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]

$D_{eff}$    Tool Tilt
ß=+ 15°

© 2016 Helical Solutions, LLC

[Figure 1]

ß+

Feed Direction

Tool Tilt
ß=+ 15°

ADOC

$D_{eff}$

*Helical*
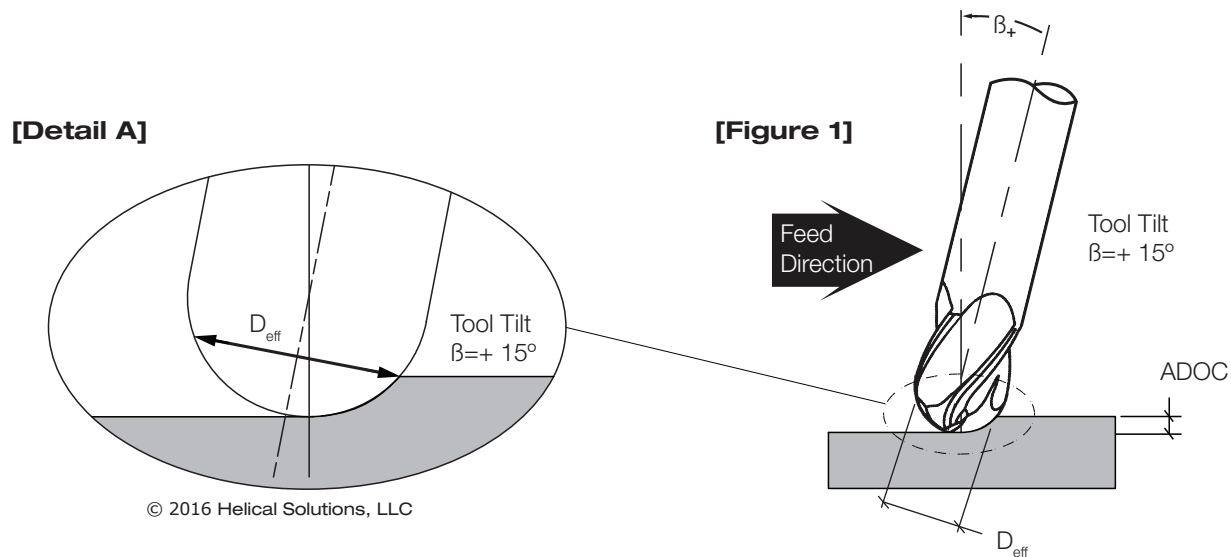
## Step One

*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 |

© 2016 Helical Solutions, LLC

**[Figure 3]**

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

## 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]**

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

Helical

# 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 - detertimental tool breakage or chipping, resulting in tool replacement.
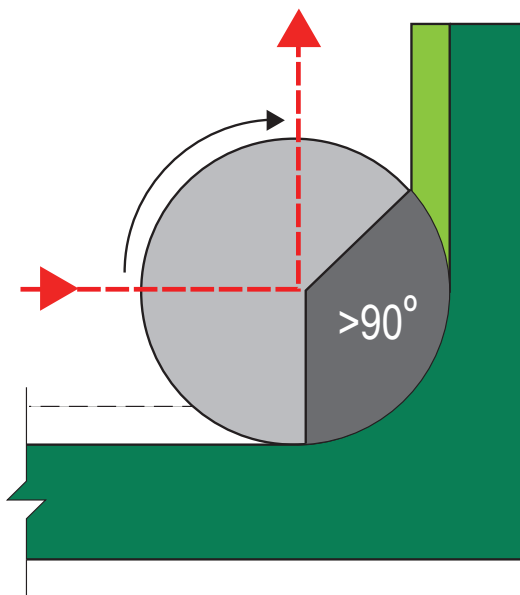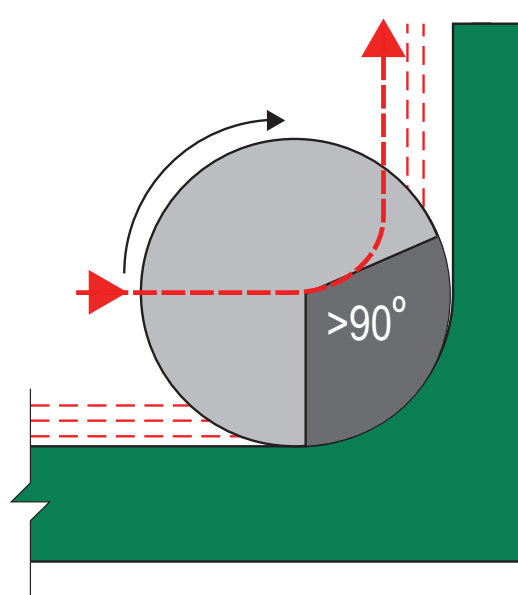


**Least Desirable**      **More Desirable**      **Most Desirable**

© 2016 Helical Solutions, LLC

Helical

# Corner Engagement (cont.)

## Least Desirable Condition

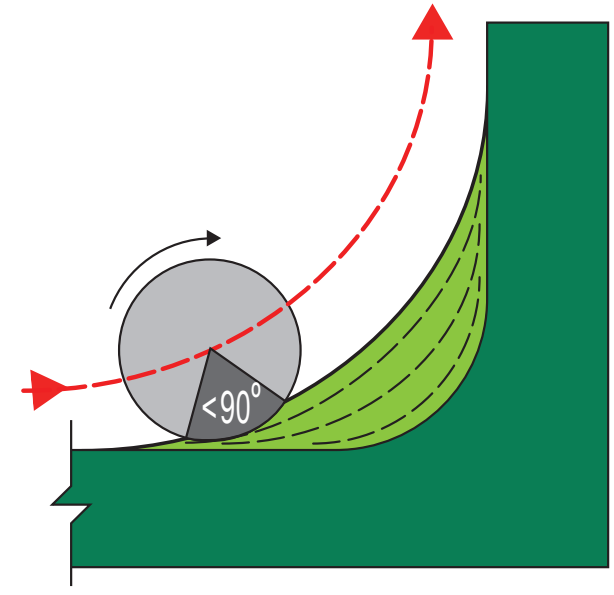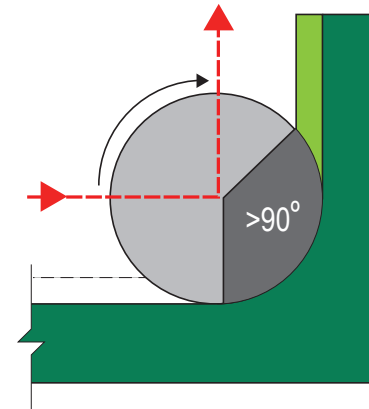Generating an inside part radius that matches the radius of the tool at a 90° direction change can make for a less than ideal machining condition. With the tool experiencing extra material to cut (light green), increased engagement angle and a direction change some of the common results will be chatter,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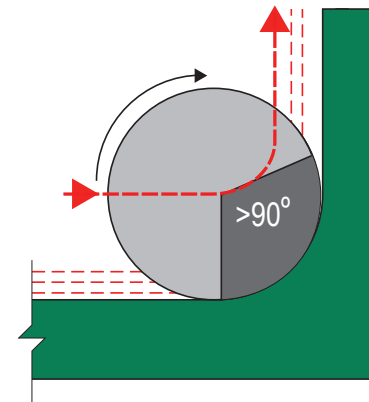
**Feed rate may need to be lessened depending on the "tool radius-to-part radius ratio."**


>90°

## 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."**


>90°

## 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.**


<90°
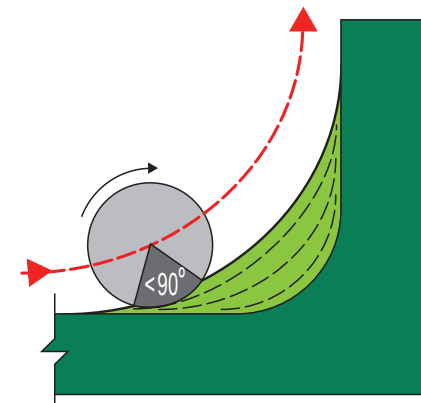
Helical

# 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)
- Increased peripheral feed

A decreasing $a_e$ can result in:

- Lower horsepower requirement
- Decreased tool deflection
- Lower spindle load (wear/tear)
- Decreased peripheral feed
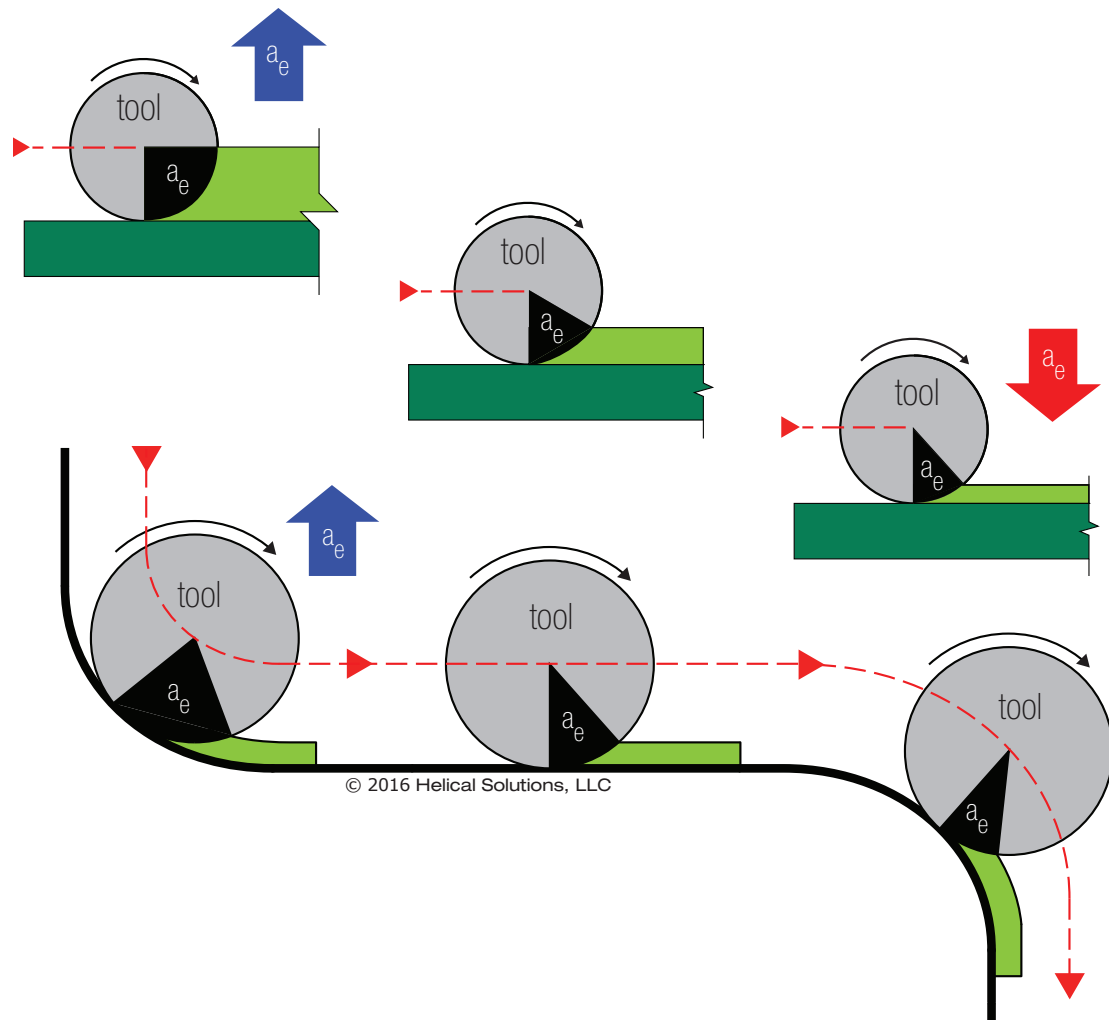
© 2016 Helical Solutions, LLC

Helical

# 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.

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

## 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



table feed

© 2016 Helical Solutions, LLC



table feed

© 2016 Helical Solutions, LLC

Helical
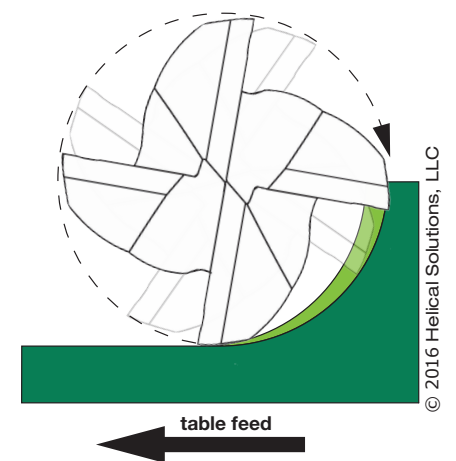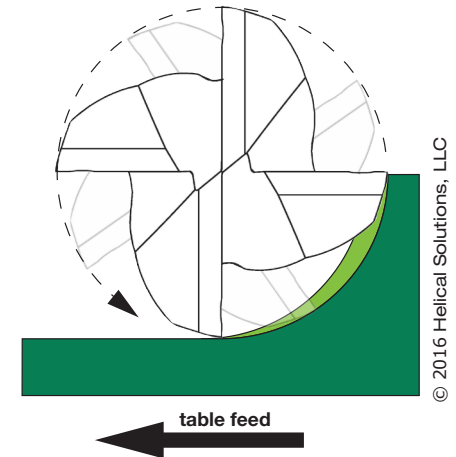
# Climb vs. Conventional Milling (cont.)

## 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.

*Helical*

# 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]**

Inches per tooth (IPT)

© 2016 Helical Solutions, LLC

Feed Direction

Radial depth of cut (RDOC)

Maximum chip thickness (CT)

**RDOC < 50% DIAMETER**

**[Figure 2]**

Inches per tooth (IPT) is equal to the maximum chip thickness when RDOC is 50% of the tool diameter

Maximum chip thickness (CT)

Radial depth of cut (RDOC)

**RDOC ≥ 50% DIAMETER**
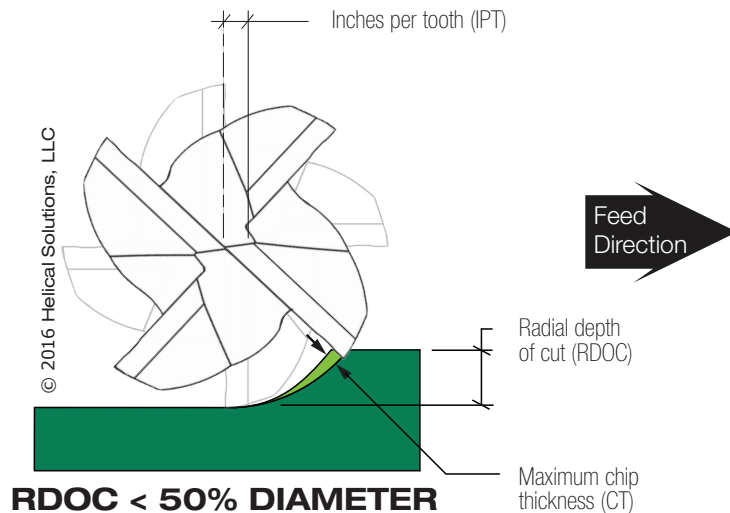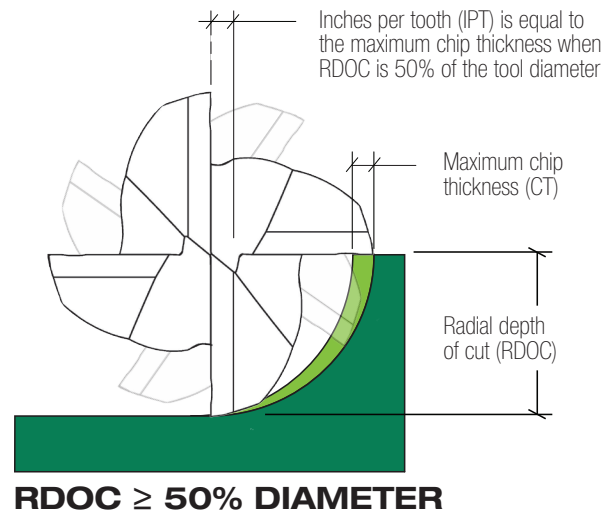
When programming a radial depth of cut (RDOC) less than 1/2 the tool diameter (Figure 1), employ the chip thinning calculation (Figure 3). A chip-thinning adjustment will prolong tool life and help reduce cycle time.

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]**

*Helical*

# 02
# High Efficiency Milling

*Helical*

# 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

**Work (and heat) is concentrated along a smaller portion of the cutting edge**

**Lower ADOC**

**Higher RDOC**

**More Axial Passes**

## High Efficiency Milling

**Work (and heat) is spread over entire cutting edge**

© 2016 Helical Solutions, LLC

**Higher ADOC**

**Lower RDOC**

**More Radial Passes**
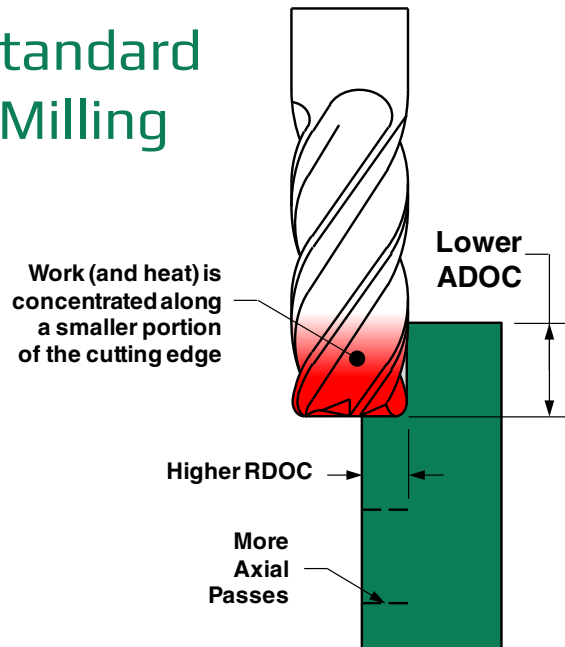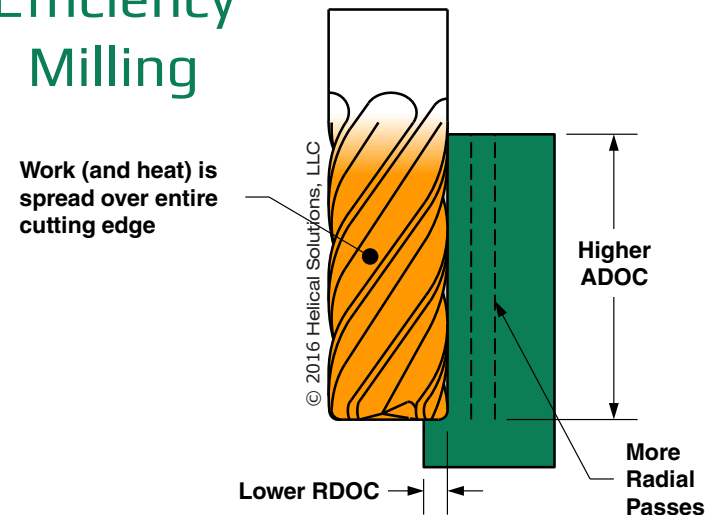
# HEM Tooling

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% |

*Helical* ⫸

# 04
# Depth of Cut

Helical

# 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
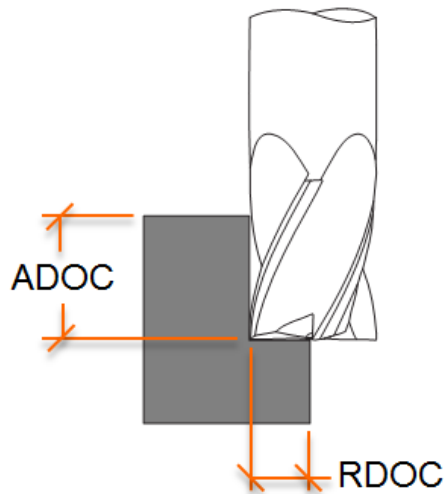


## Peripheral Milling

## Slotting

© 2016 Helical Solutions, LLC

*Helical* 〰〰〰
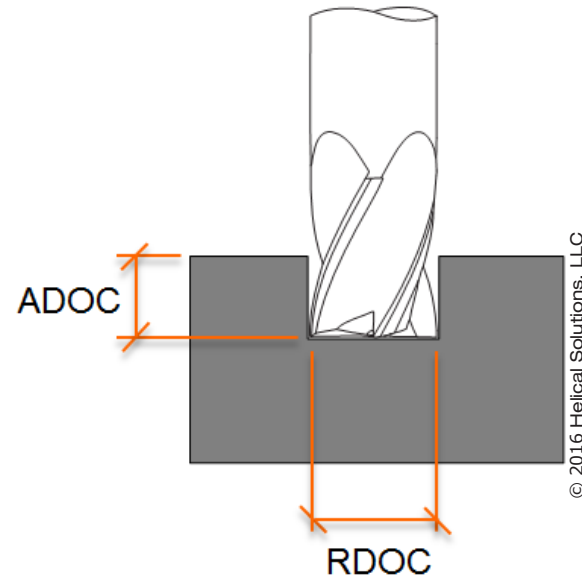
# Depth of Cut - Peripheral

Traditionally, it has been:

- Heavy RDOC
- Light ADOC
- Conservative IPM

New strategies include:

- Light RDOC
- Heavy ADOC
- Increased IPM



ADOC

RDOC (F)

RDOC (LR)

RDOC (HR)

© 2016 Helical Solutions, LLC

3-5% RDOC
*finishing*

10-25% RDOC
*Light roughing*

30-50% RDOC
*Heavy roughing*

Decreased..................................................................................................................Increased

Horsepower Req.    Tool Deflection
Chip Thickness

Horsepower Req.    Tool Deflection
Chip Thickness

Helical

# Depth of Cut - Peripheral (cont.)



**Necked down tool style** ▶

**Works with lower RDOC** ◀

**Too long of a LOC, leading to deflection and chatter** ◀

© 2016 Helical Solutions, LLC

ADOC

RDOC

ADOC

RDOC

Strongest Condition..................................................................Weakest Condition

*Helical*

# Depth of Cut - Peripheral (cont.)
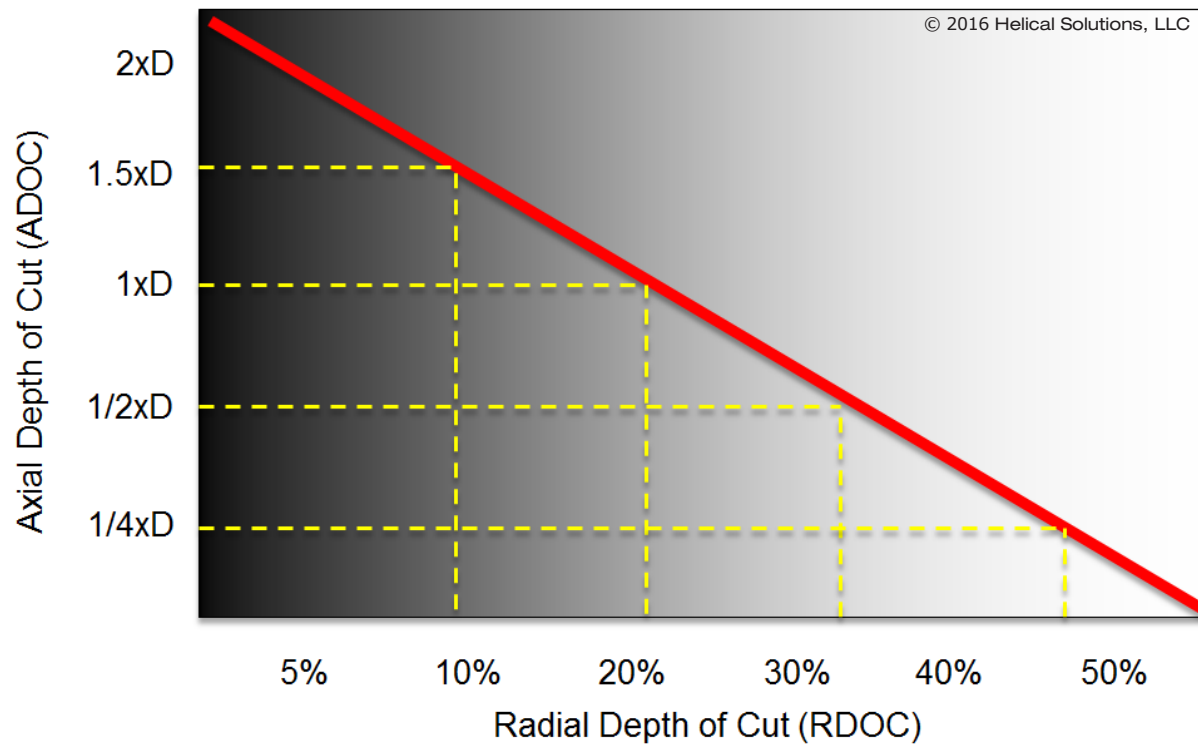
# Depth of Cut - Peripheral (cont.)

## 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!



© 2016 Helical Solutions, LLC

RDOC

*Helical*
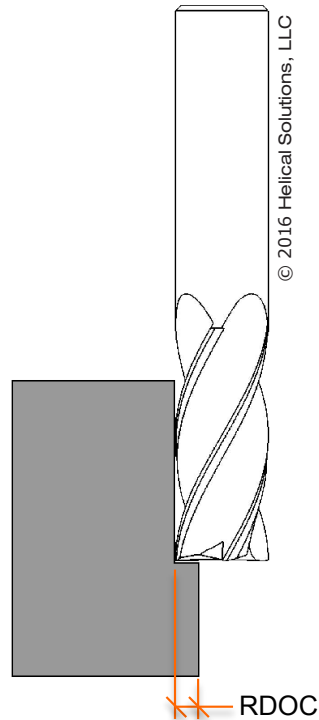
# Depth of Cut - Peripheral (cont.)

## 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!



© 2016 Helical Solutions, LLC

ADOC

Helical
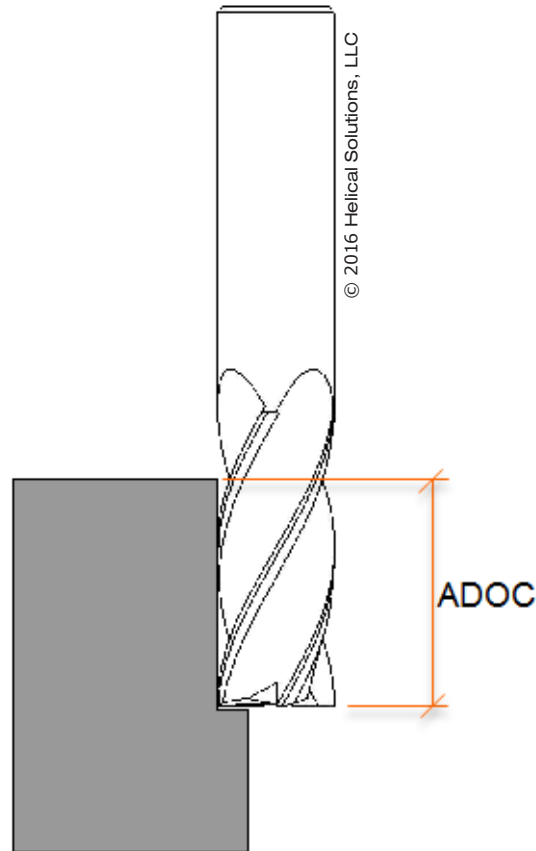
# Depth of Cut - Peripheral (cont.)

## 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



© 2016 Helical Solutions, LLC

↓ RDOC
↑ ADOC
↑ IPM
↑ MRR
↓ Tool Wear

ADOC

RDOC

# Depth of Cut - Slotting



Shallow
>0 to 1/4xD

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

Full Depth
>1/2D to 1xD

ADOC

© 2016 Helical Solutions, LLC

Decreased..........................................................................................Increased

Horsepower Req.    Tool Deflection
Chip Thickness

Horsepower Req.    Tool Deflection
Chip Thickness
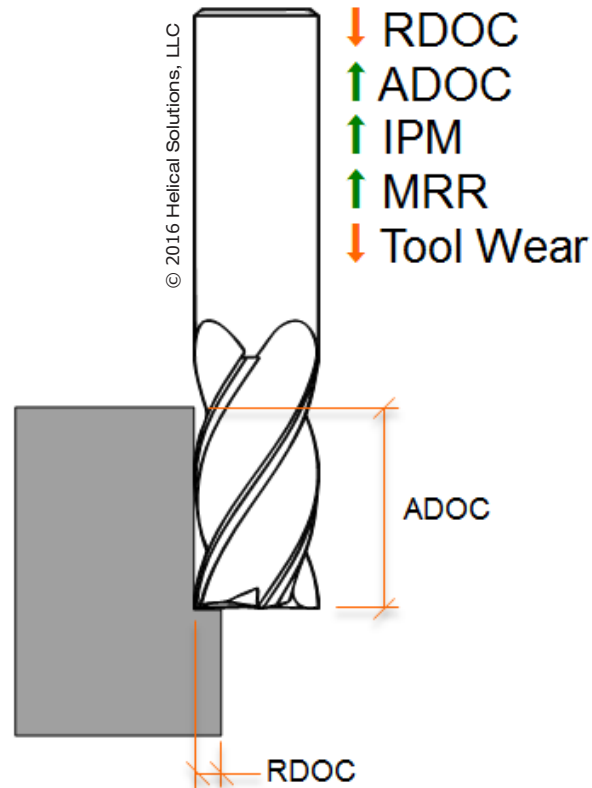
# Depth of Cut - Slotting (cont.)

## 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



© 2016 Helical Solutions, LLC
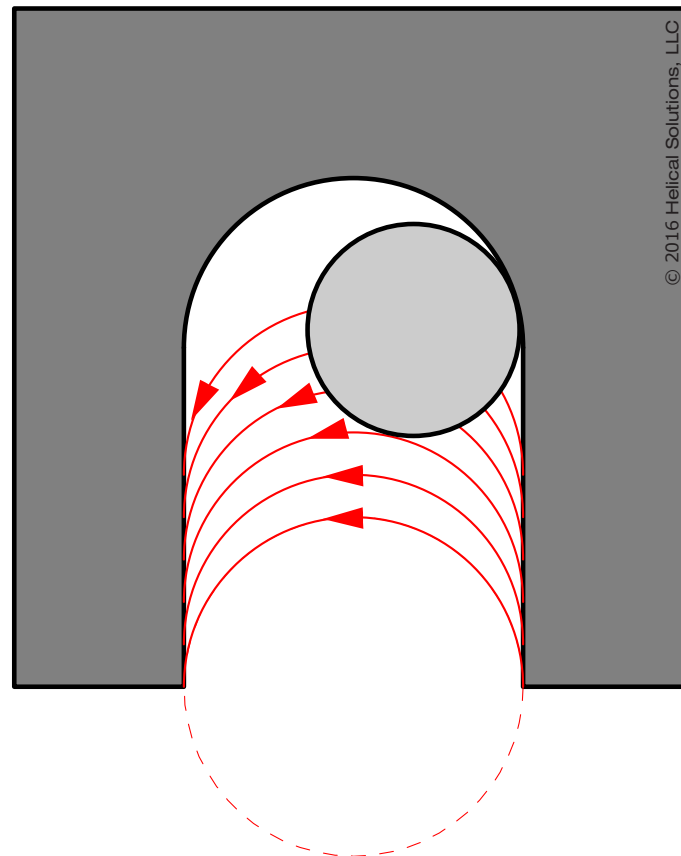
# Depth of Cut - Slotting (cont.)

## 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
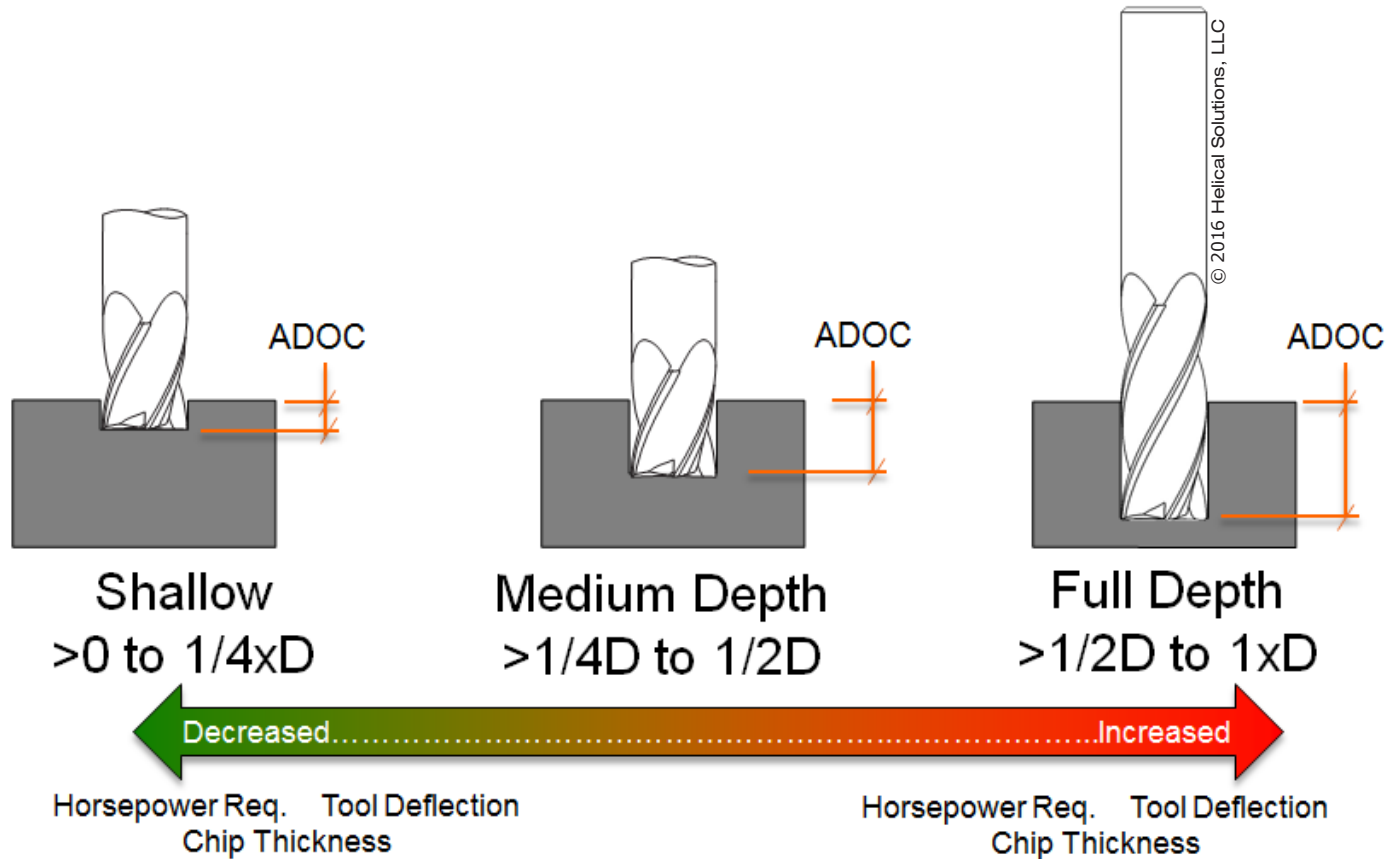- Utilize HE type of Tool Paths for best results!



© 2016 Helical Solutions, LLC

*Helical*

# 04
# End Mill Construction

Helical
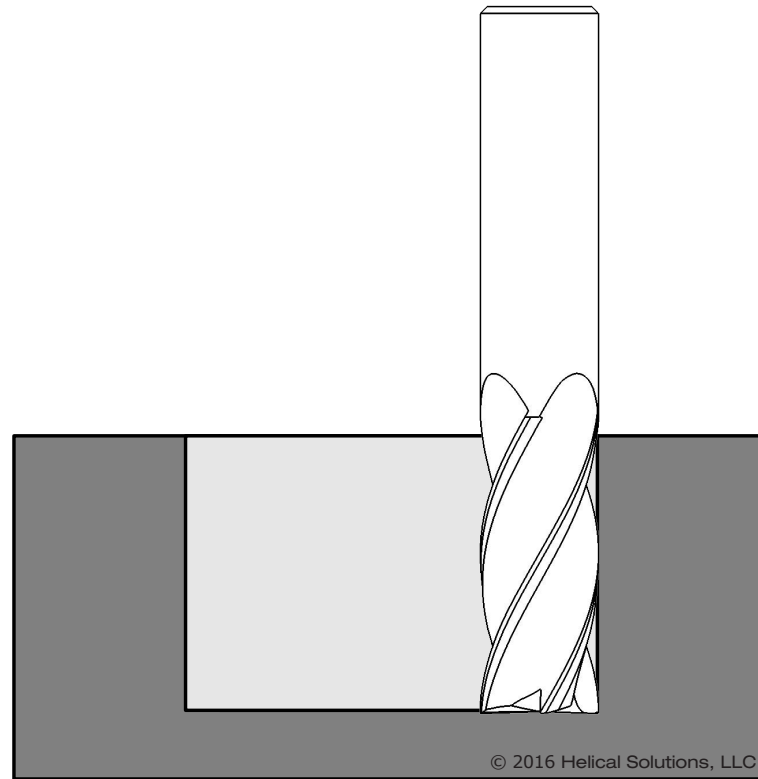
# Geometry Definitions

| | |
|---|---|
| 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. |

*Helical*

# 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.

Helical

# End Mill Construction



Gash Angle

Dish Angle

Cutting Diameter

₵L

**View A-A**

Radial Rake

Pitch

Pitch with unequal spacing is also known as "variable pitch"

Core Diameter

**View B-B**

Overall Length (OAL)

Length of Cut (LOC)

Helix Angle

Shank Diameter

₵L

© 2016 Helical Solutions, LLC

Flute Wash

A-A

B

B
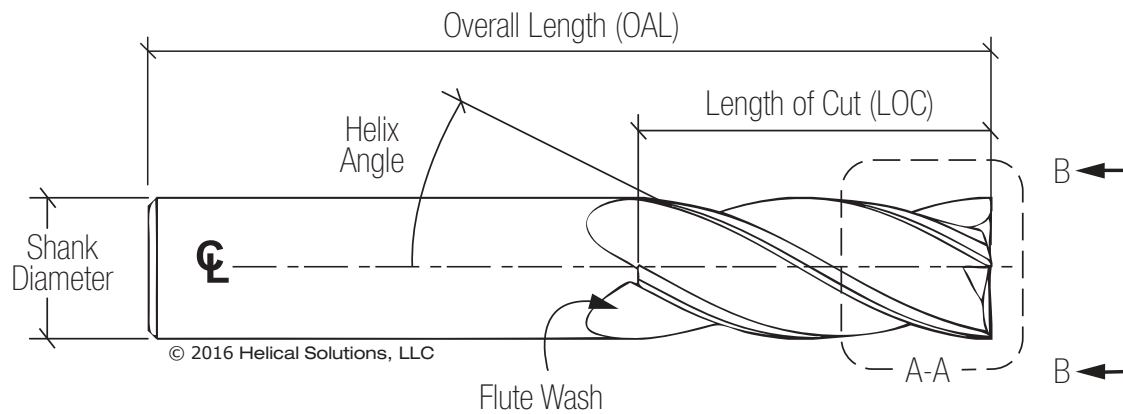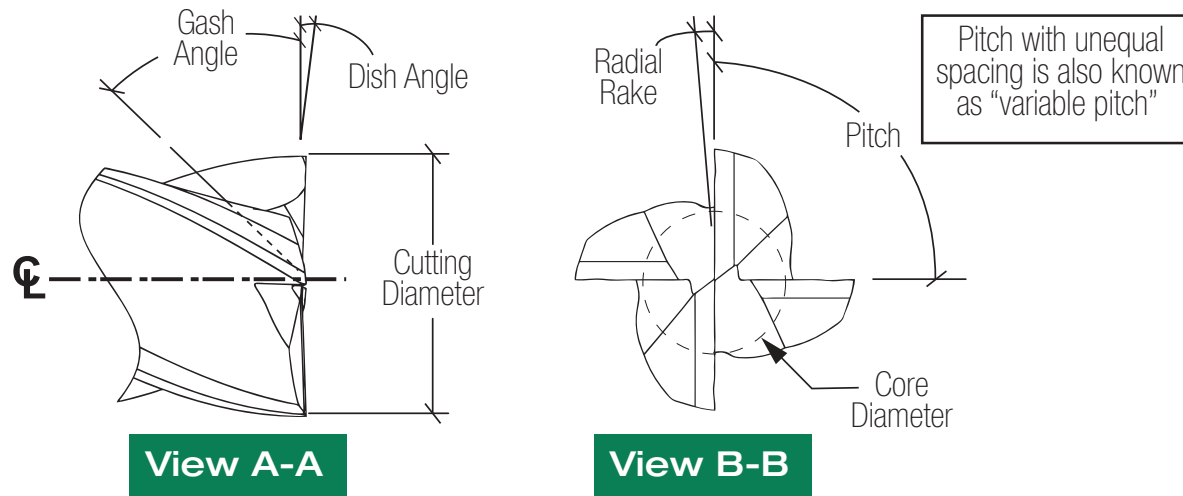
# End Mill Construction (cont.)

## Radial Relief Types

Margin

Rake Face

primary

secondary

**Cylindrical**

Rake Face

primary (measured radially)

**Eccentric**

Rake Face

primary

secondary

**Standard**

© 2016 Helical Solutions, LLC

Part

(OAL)

Length Below Shank (LBS)

(LOC)

*Helical*
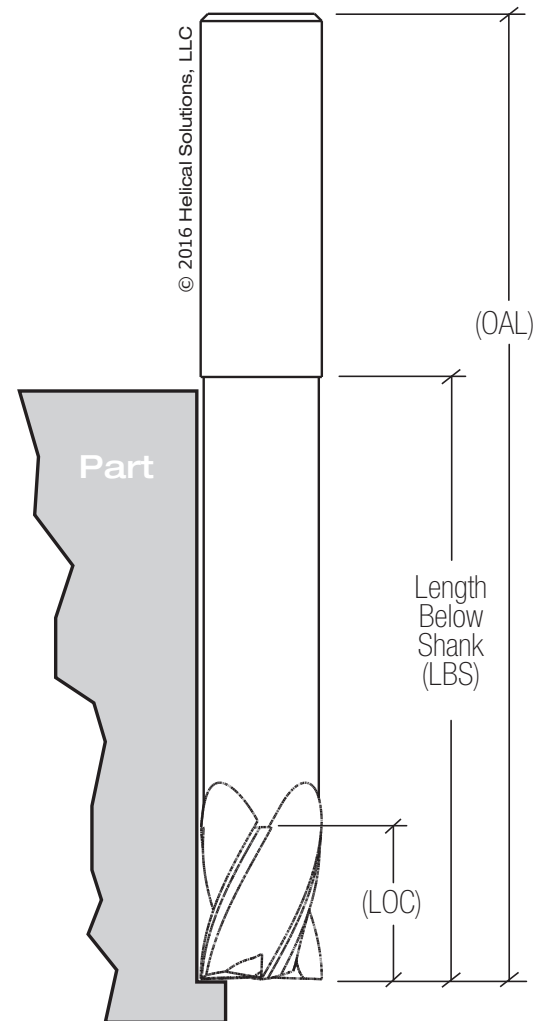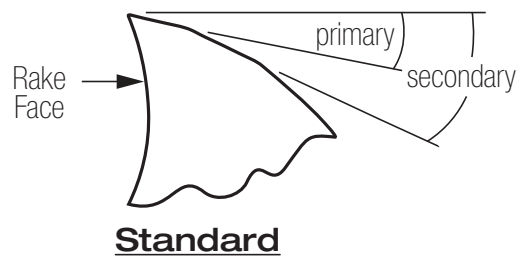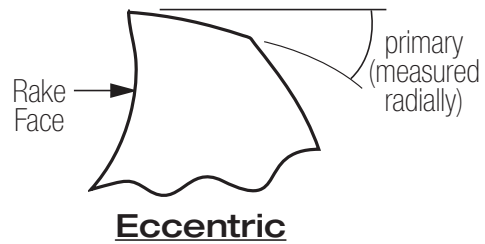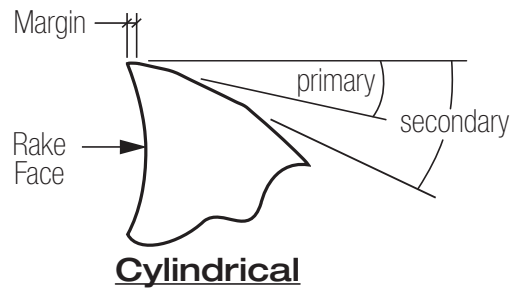
# End Mill Anatomy

## 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

Helical /////
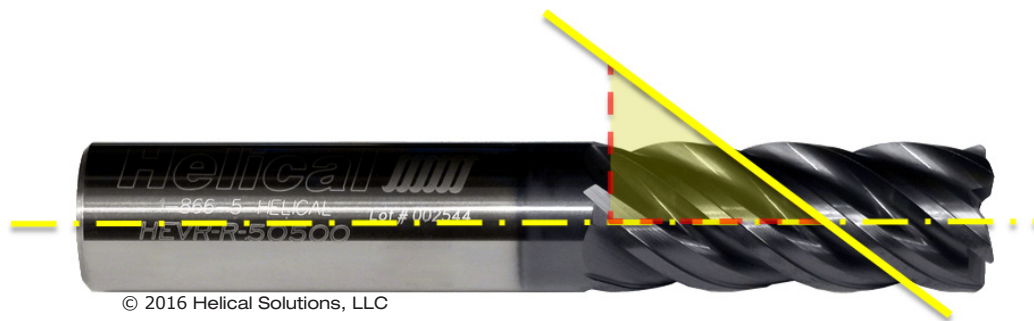
# End Mill Anatomy (cont.)

## 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

Helical
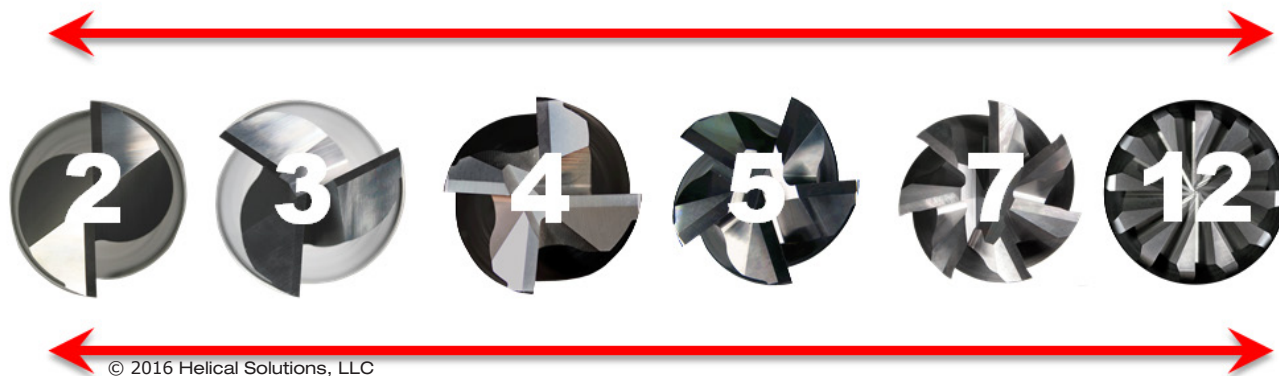
# End Mill Anatomy (cont.)

## 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.



90°
90.5°

90°
88.2°

90°
90.3°

90°
91°

© 2016 Helical Solutions, LLC
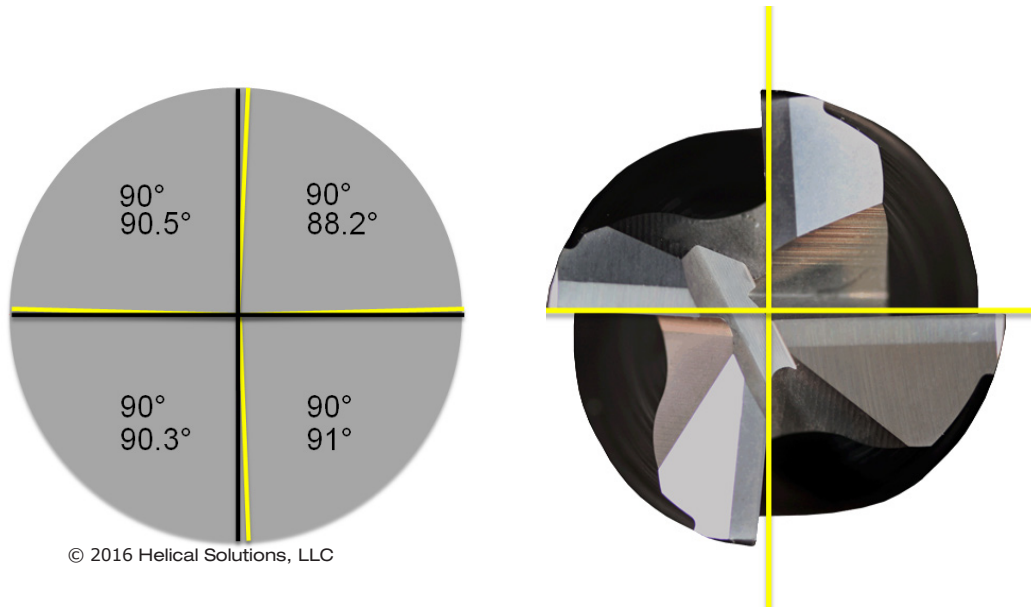
*Helical* 〰〰〰

# 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.



Reduced Neck (LBS)

LOC

Stronger Choice

4" OAL

Length Of Cut (LOC)

© 2016 Helical Solutions, LLC

# 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



3 flutes

Larger Core

8 flutes

*Helical* /////
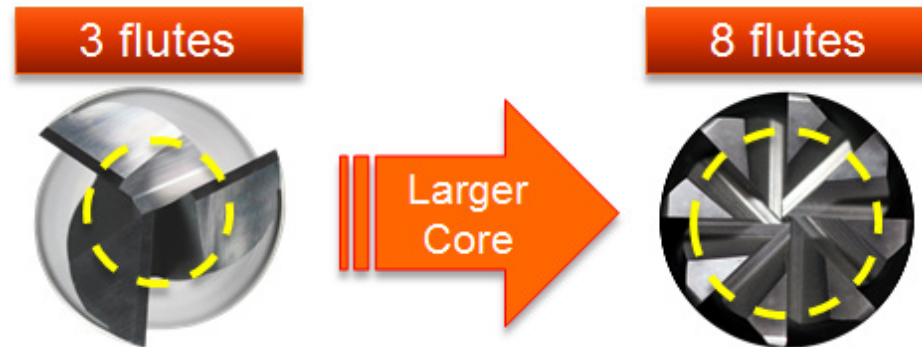
# End Mill Anatomy (cont.)

## Axial Relief

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



© 2016 Helical Solutions, LLC

Secondary "Axial" Relief Angle

Primary "Axial" Relief Angle

*Helical*
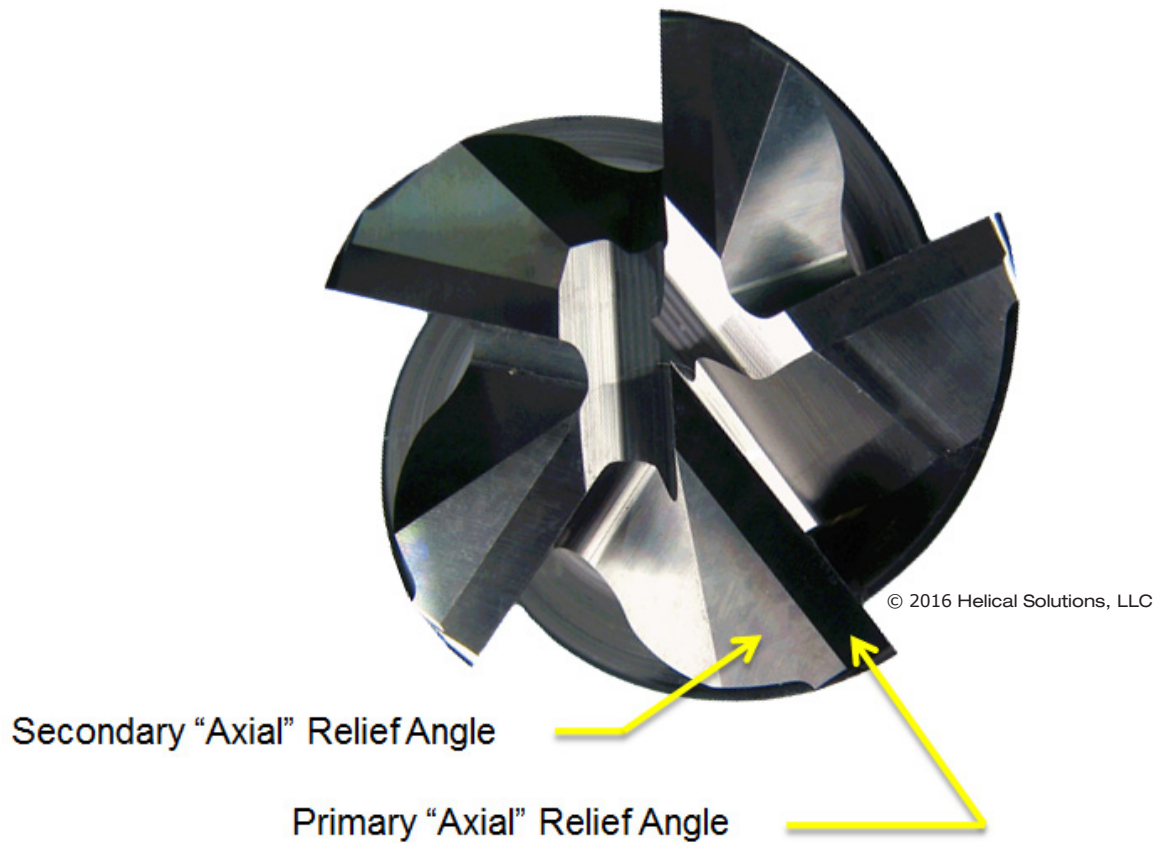
# End Mill Anatomy (cont.)

## 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

*Helical*

# 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.



"Radial" Relief
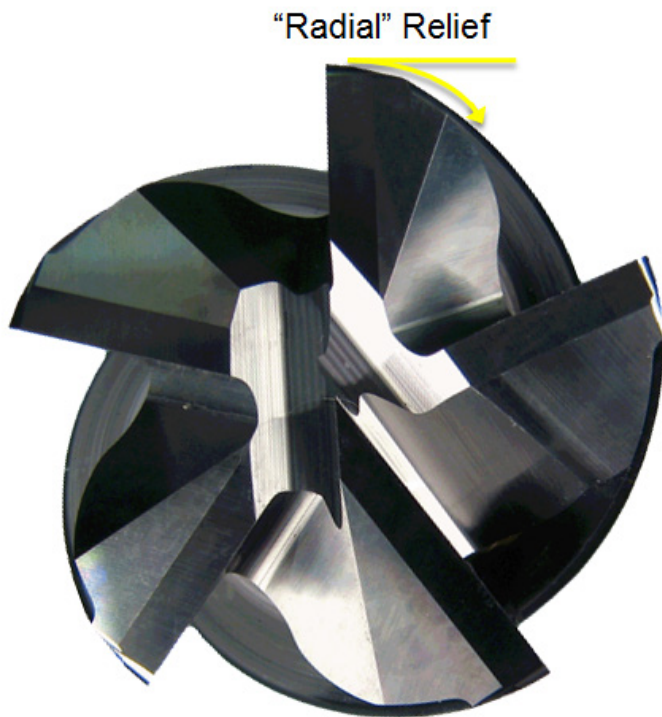
© 2016 Helical Solutions, LLC
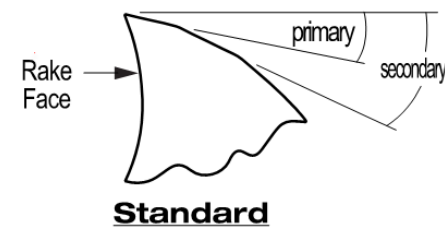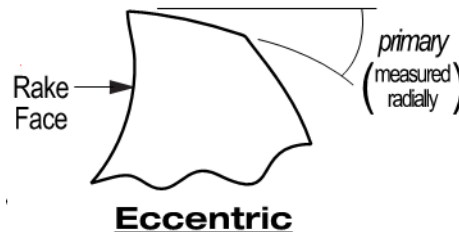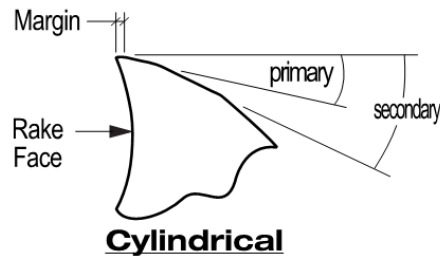
### Cylindrical Relief

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.



Margin
Rake Face
primary
secondary
**Cylindrical**

Rake Face
primary (measured radially)
**Eccentric**

Rake Face
primary
secondary
**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.
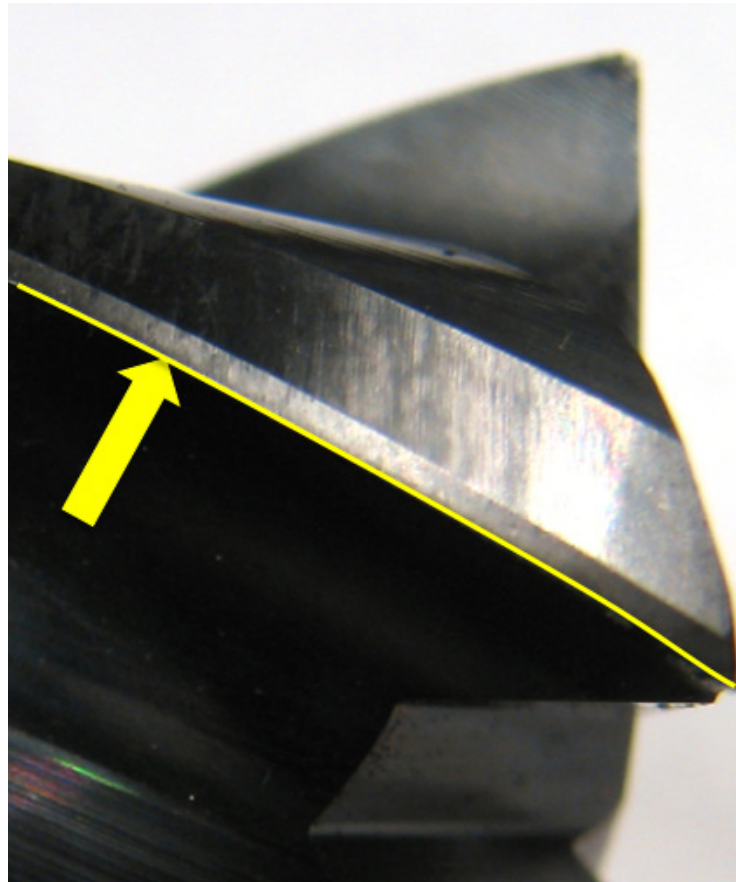


© 2016 Helical Solutions, LLC

*Helical*

# 05

# Common Calculations

**Helical**

# 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 |

| Wire | mm | Inch | Decimal |
|---|---|---|---|
|  |  | 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 |

| Wire | mm | Inch | Decimal |
|---|---|---|---|
| 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 |

| Wire | mm | Inch | Decimal |
|---|---|---|---|
|  |  | 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 |

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

## 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 |
| $\text{IPT}_{adj}$ | 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) |

helical

# 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.
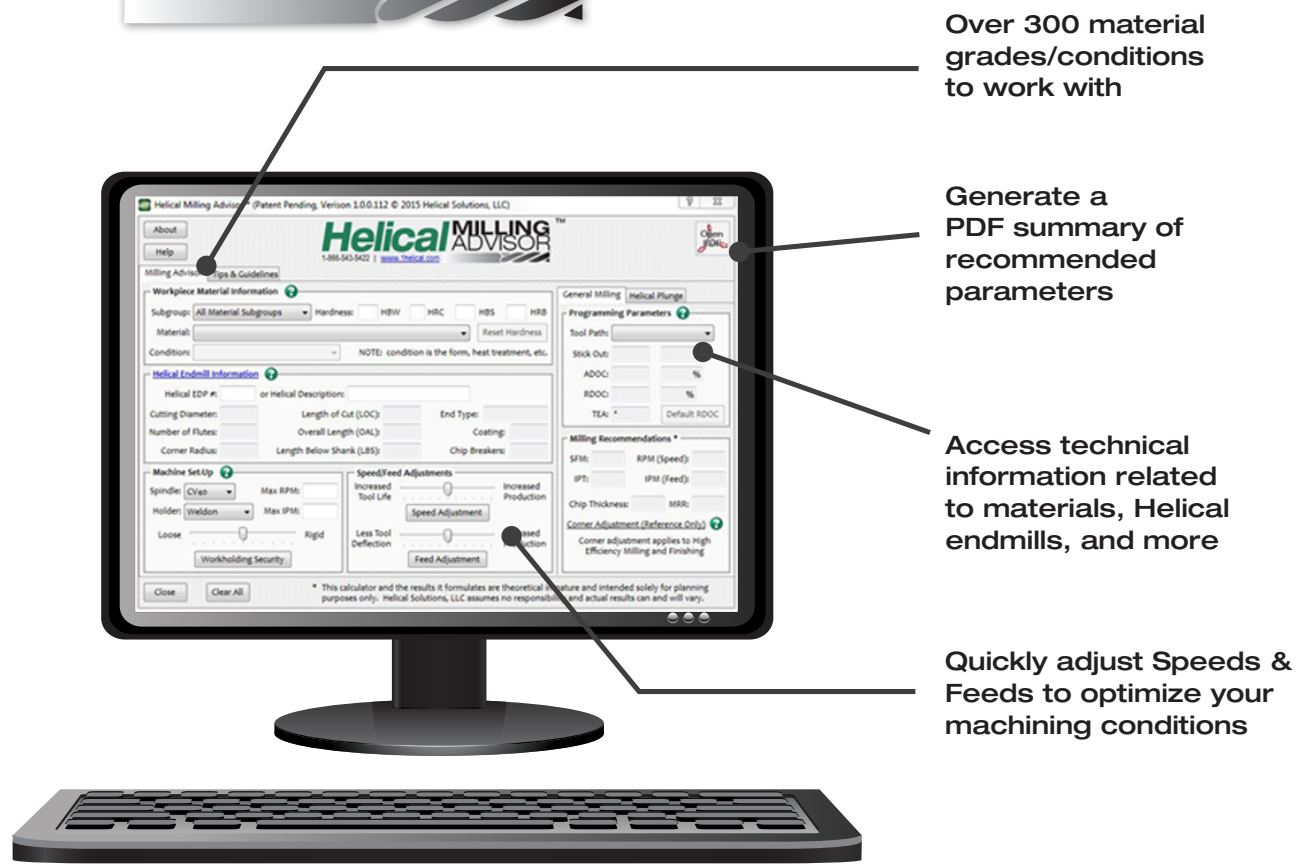


**Over 300 material grades/conditions to work with**

**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

*Helical*

# 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

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 Life Range

Concentricity as % of Feed-per-Tooth

© 2016 Helical Solutions, LLC
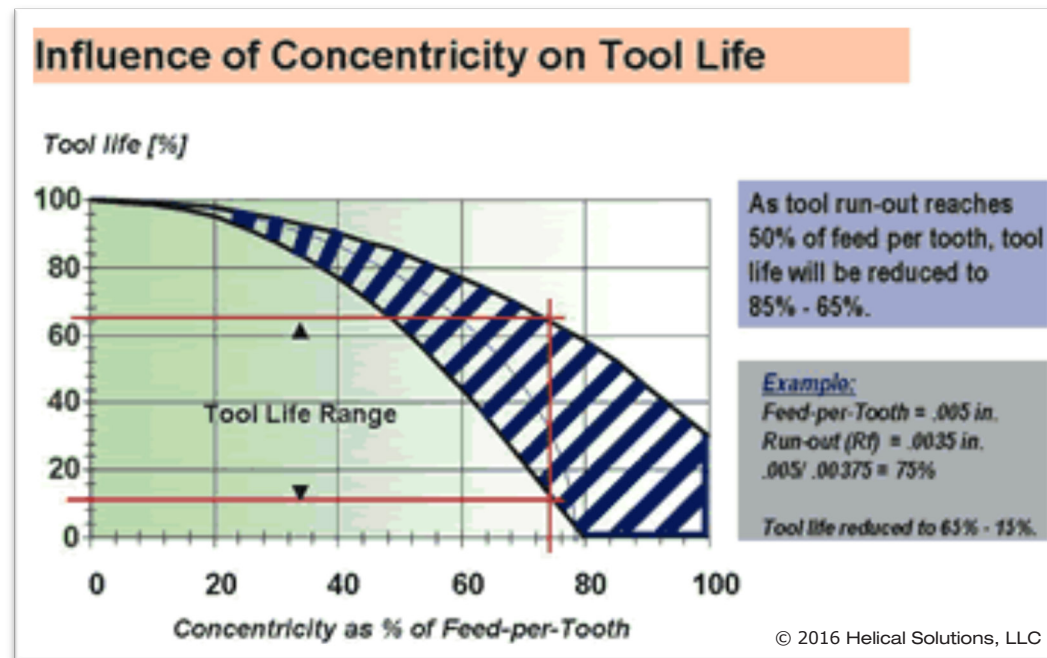
# Tool Holding (cont.)
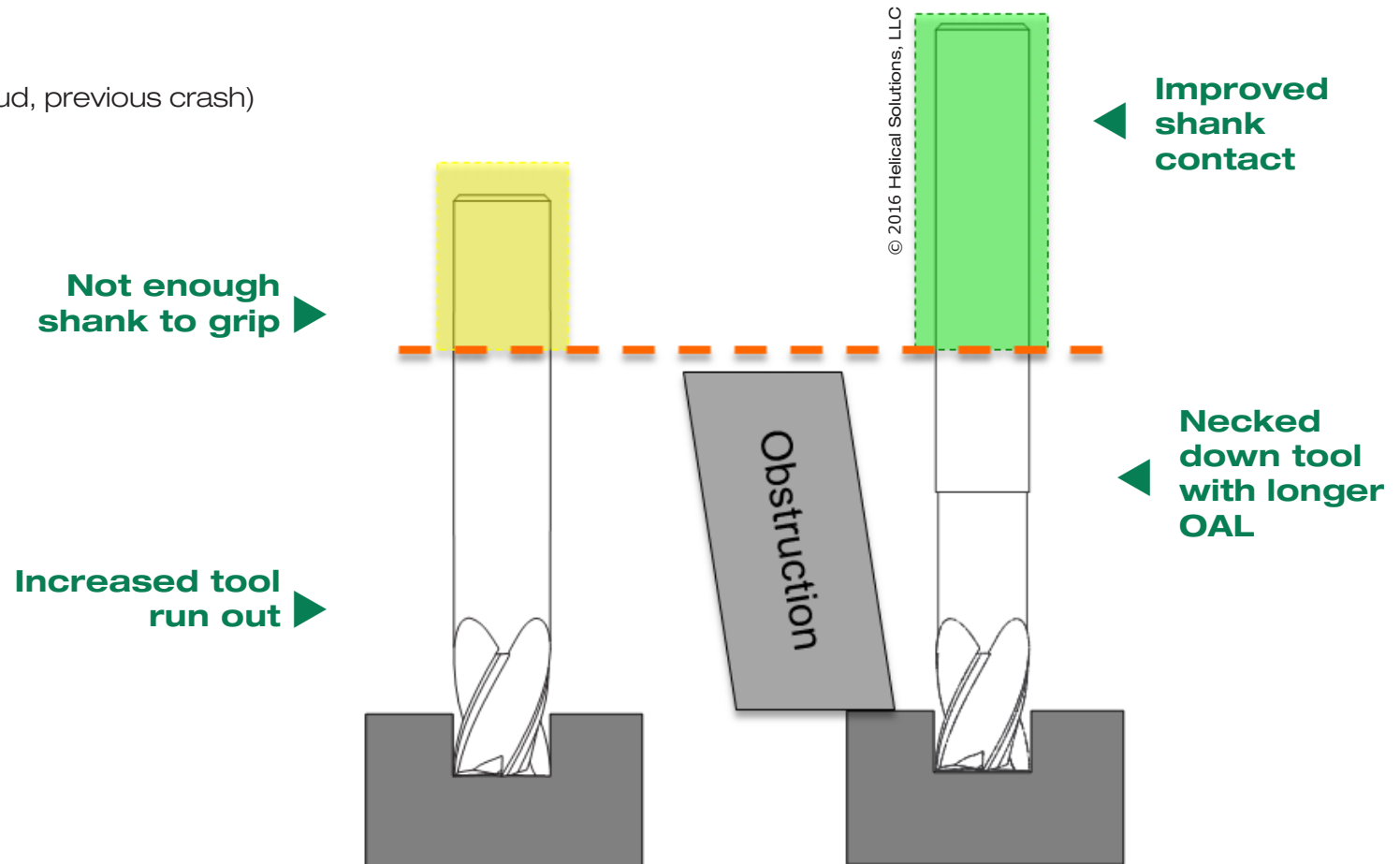
## Run Out

- 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



**Improved shank contact**

**Not enough shank to grip** ▶

**Necked down tool with longer OAL**

**Increased tool run out** ▶

Obstruction

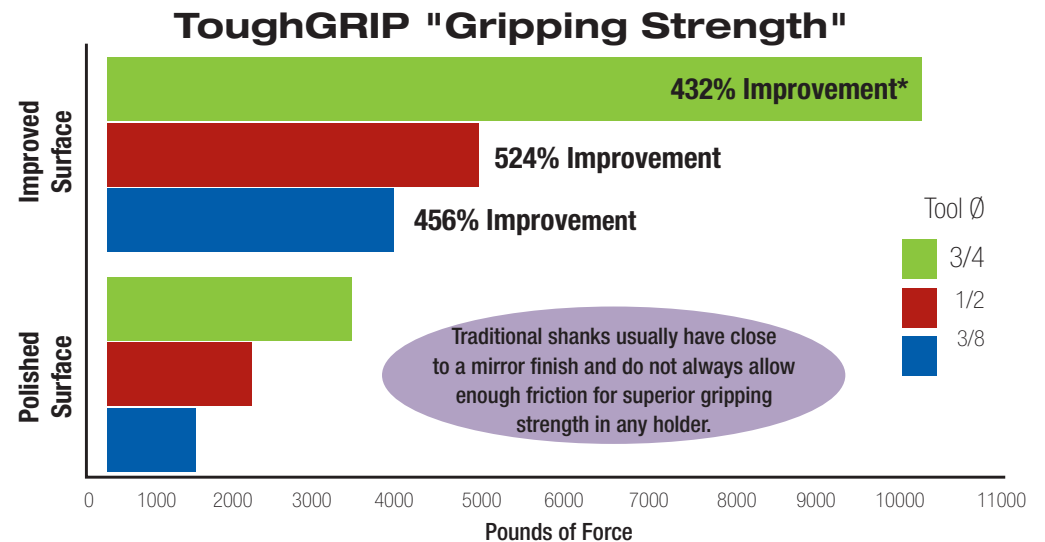© 2016 Helical Solutions, LLC

helical

# 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

### ToughGRIP "Gripping Strength"



Improved Surface
- 432% Improvement*
- 524% Improvement
- 456% Improvement

Polished Surface

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

Tool Ø
- 3/4
- 1/2
- 3/8

Pounds of Force (0 1000 2000 3000 4000 5000 6000 7000 8000 9000 10000 11000)

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
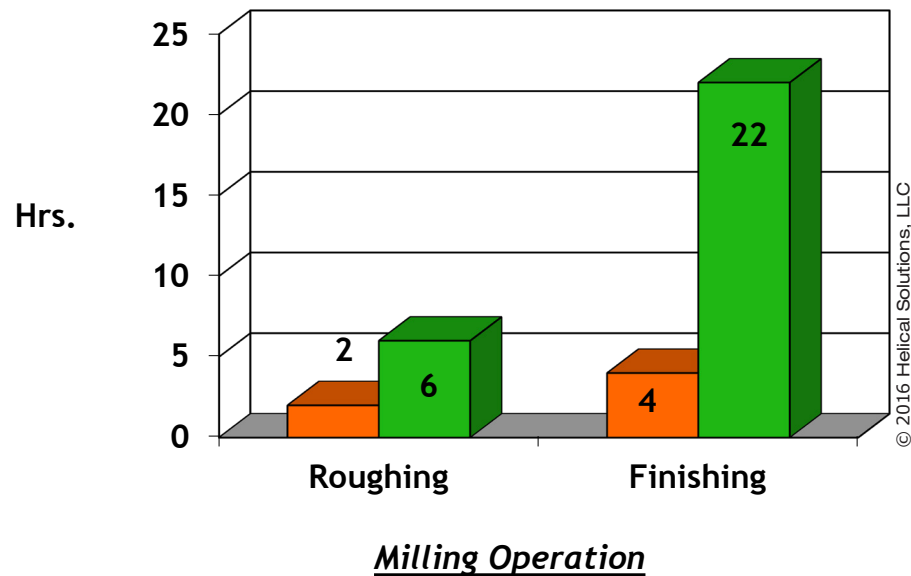

Helical™
1-866-5-HELICAL

Helical

# 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
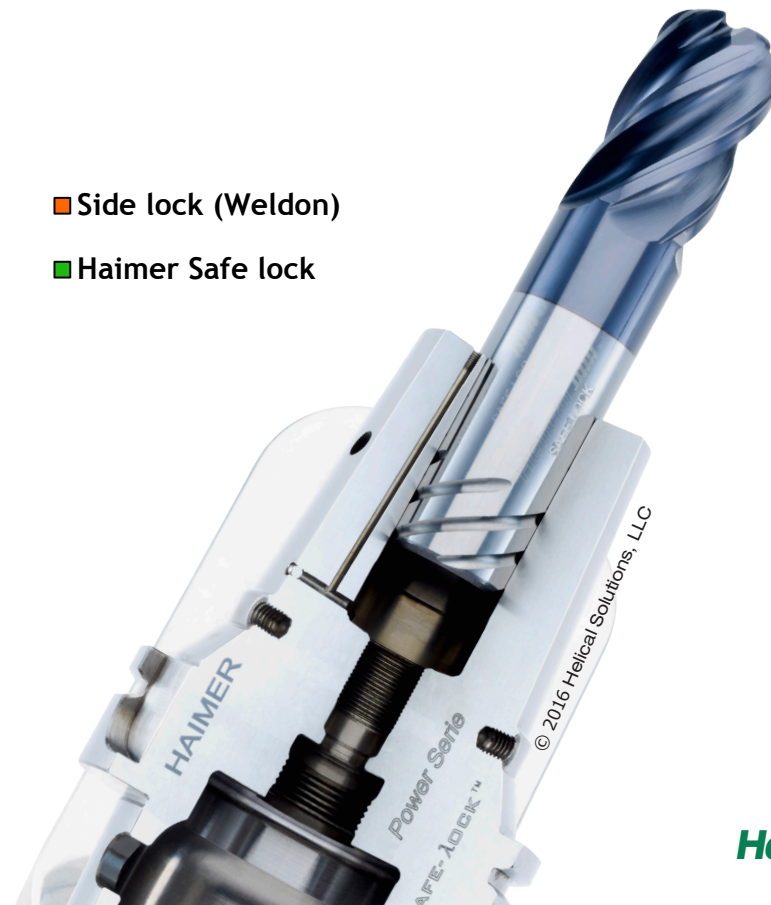- Can accommodate tools from 1/2" to 2" diameter

The below graph illustrates the machining hours before tool readjustment is required.

**HAIMER**®



Hrs.

Legend:
- Side lock (Weldon)
- Haimer Safe lock

Roughing: 2 (Side lock), 6 (Haimer Safe lock)
Finishing: 4 (Side lock), 22 (Haimer Safe lock)

© 2016 Helical Solutions, LLC

*Milling Operation*

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

*Helical*

# 07
# Troubleshooting

Helical

# 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 90º (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 +) |

**Helical**

# 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 90º (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. |

*Helical*

# 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. |

| 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 |

| The Problem | Possible Causes | Possible Solution |
|---|---|---|
| **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 |

| The Problem | Possible Causes | Possible Solution |
|---|---|---|
| **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.



© 2016 Helical Solutions, LLC

### 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.
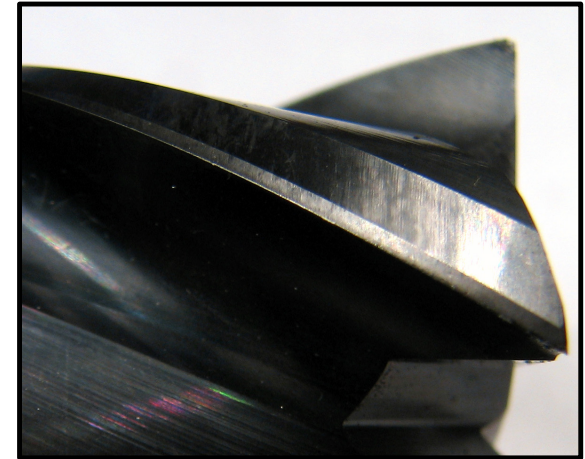


© 2016 Helical Solutions, LLC
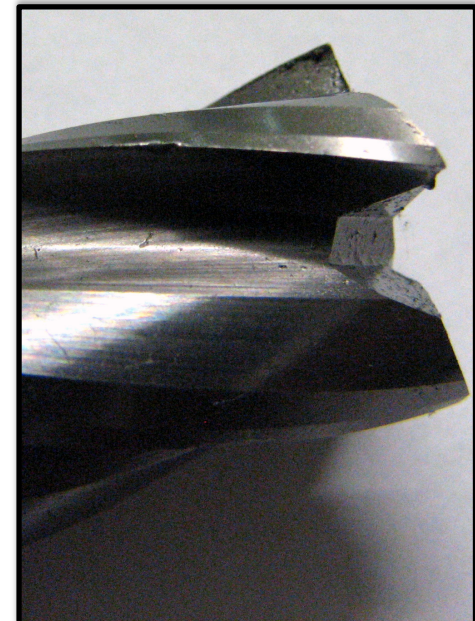
*Helical* /////

# 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.



© 2016 Helical Solutions, LLC

## 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.
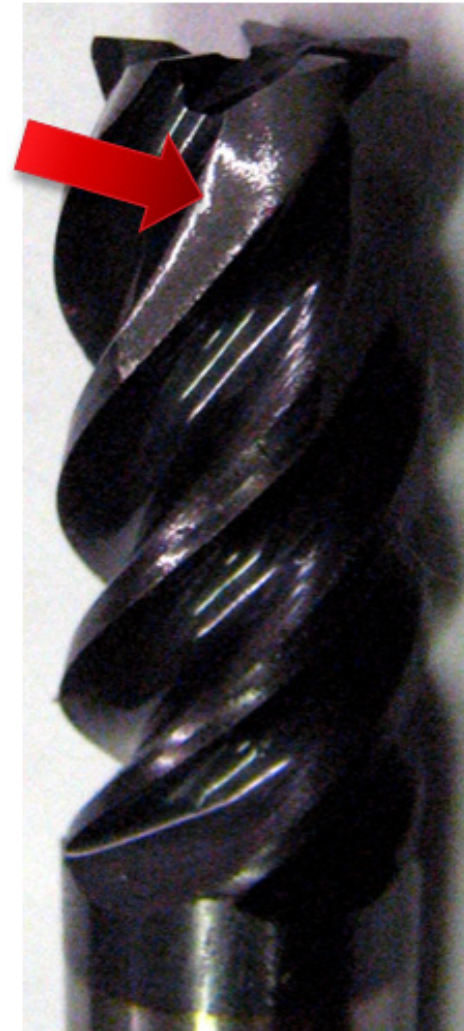


© 2016 Helical Solutions, LLC

*Helical*

# 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
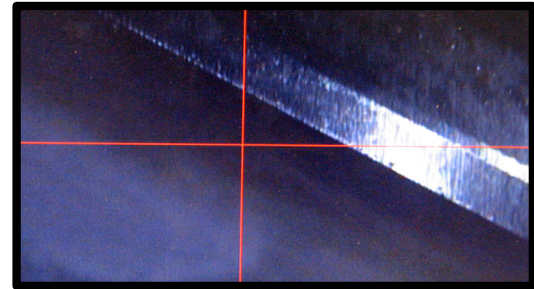- Unmanageable Metal Removal Rate (MRR)



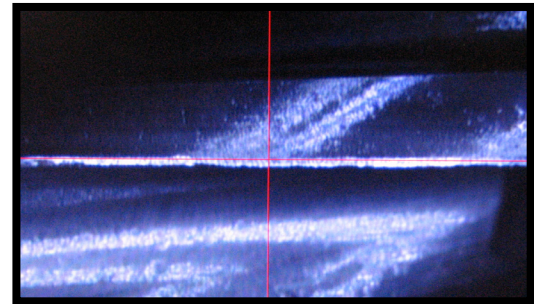© 2016 Helical Solutions, LLC

*Helical*

# 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.



Good

Bad
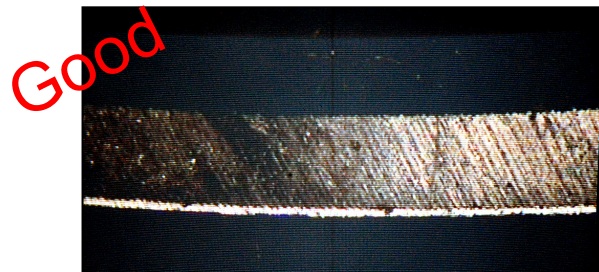
© 2016 Helical Solutions, LLC

Helical

# 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

© 2016 Helical Solutions, LLC
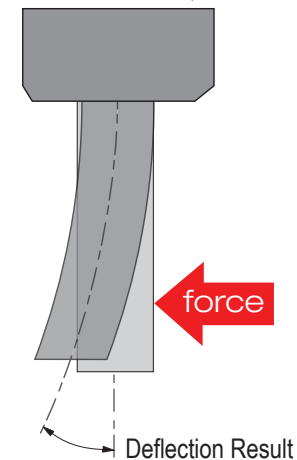
force

Deflection Result

*Helical*

## 2017 Helical Product Catalog

The 2017 catalog introduces over 600 new tools, including the brand new 6 Flute End Mills for Steels, High-Temp Alloys, and Titanium (HEV-6). 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