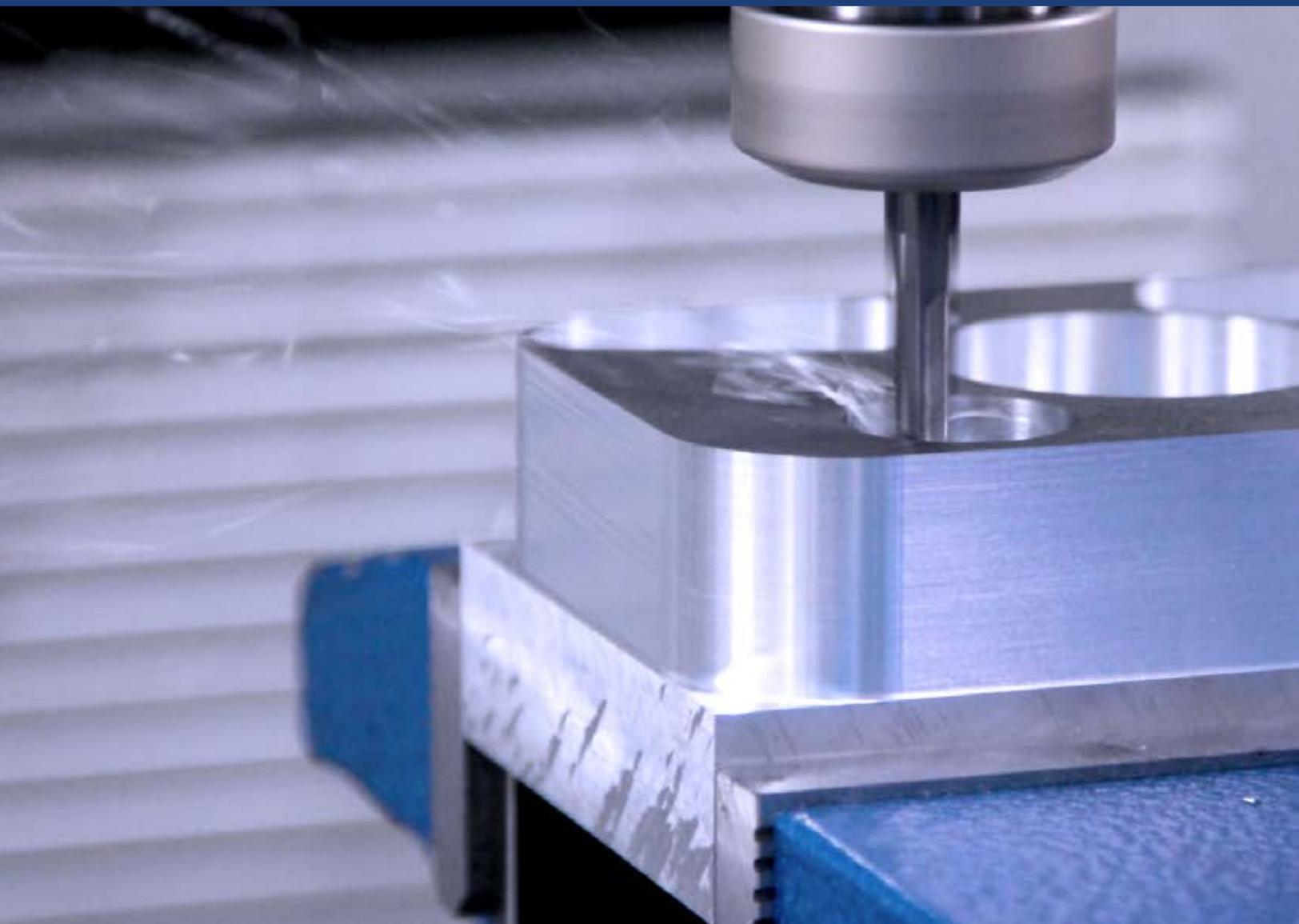


FREE EBOOK

HEM GUIDEBOOK

A Machinist's Guide to Increasing Shop Productivity with High Efficiency Milling





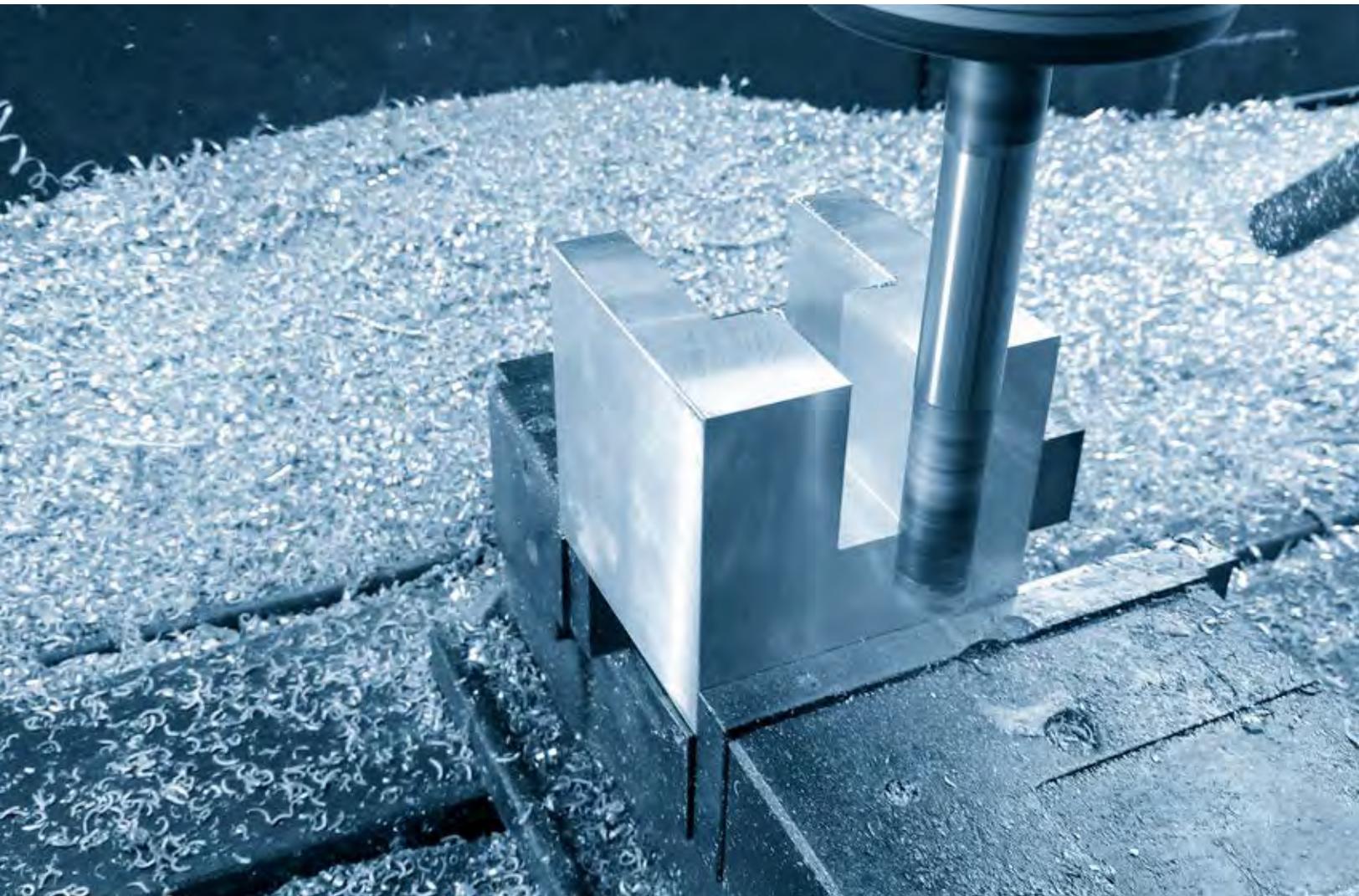
CONTENTS

| | |
|---|----|
| 1 State of High Efficiency Milling | 3 |
| 2 Introduction to High Efficiency Milling | 7 |
| 3 High Speed Machining vs. HEM | 13 |
| 4 Combat Chip Thinning & Boost Tool Potential | 17 |
| 5 Diving Into Depth of Cut | 21 |
| 6 Preventing Tool Wear | 26 |
| 7 Applying HEM to Micromachining | 32 |
| 8 Best Practices for Trochoidal Milling | 36 |
| 9 HEM FAQs | 42 |
| 10 Pro Tips | 45 |

Using HEM can result in profound shop efficiencies, extended tool life, greater performance, and cost savings.

1

STATE OF HIGH EFFICIENCY MILLING





THE STATE OF HIGH EFFICIENCY MILLING

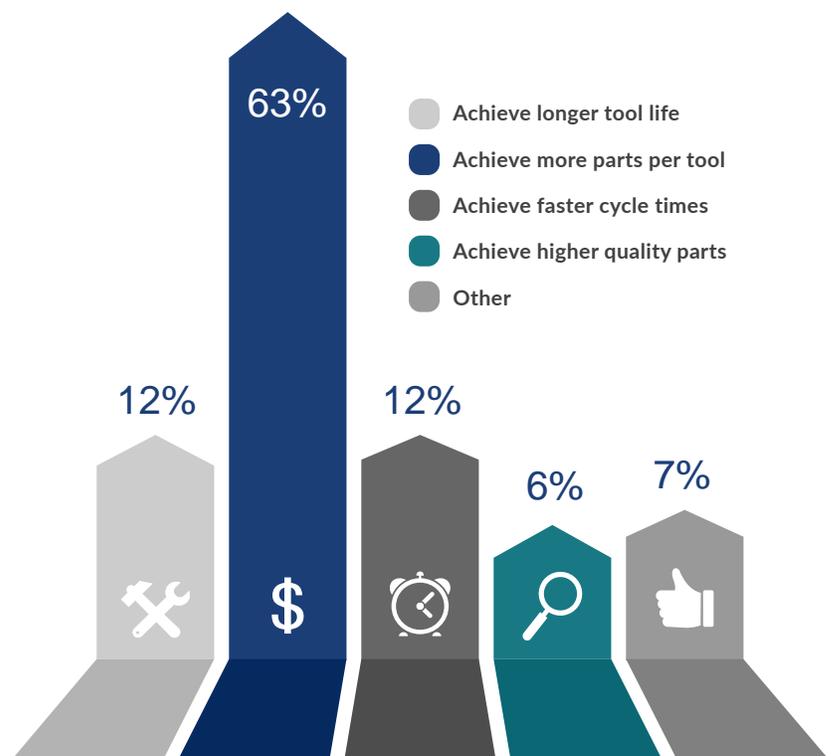
Harvey Tool and Helical Solutions surveyed thousands of machinists and programmers to get their insights on High Efficiency Milling and how it's been used in their shops. We compiled hundreds of results to uncover the trends of HEM and why machinists are choosing to rely on this toolpath more and more.

Does your shop use HEM toolpaths?



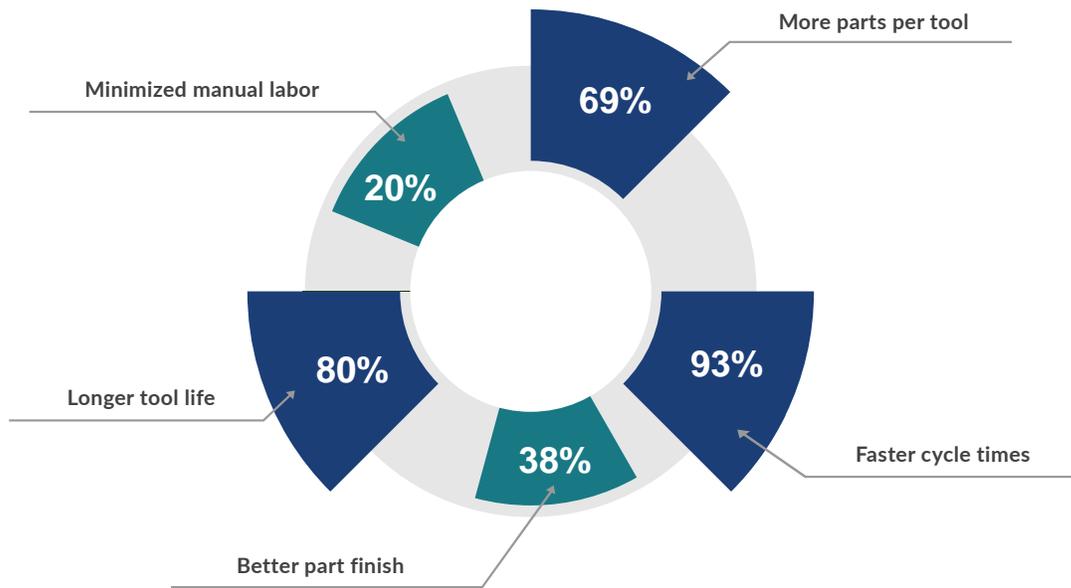
What is the primary reason your shop uses HEM?

Respondents were encouraged to select just one primary reason their shop chose to use HEM toolpaths. Of those who selected "Other," many indicated that all of the reasons went into their decision to use HEM. Other responses included "more reliable toolpaths," "decreased cost per part," "ease of programming," "better machining control," and "less likelihood of breaking tools."



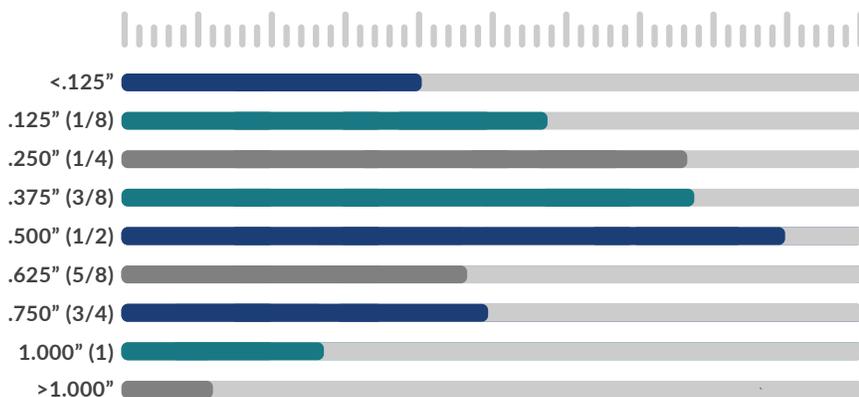
In using HEM, which of the following results has your shop experienced?

Respondents were invited to select all of the results that their shop experienced when utilizing HEM toolpaths and techniques.

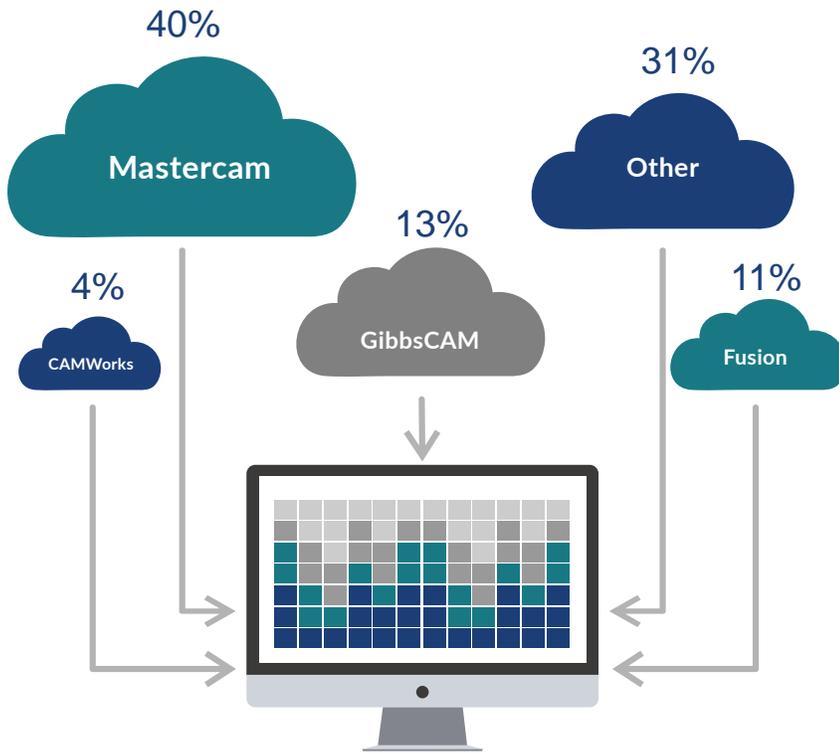


What cutter diameters have you used with HEM?

Respondents were invited to select all of the cutter diameters they've used with HEM toolpaths and techniques.



Curious how to implement HEM strategies with miniature end mills? See the HEM & Micromachining section starting on page 32.

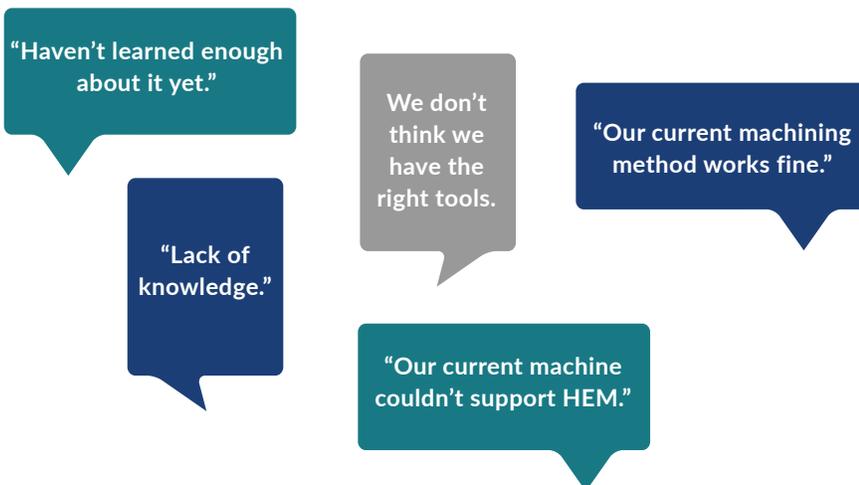


What CAM software does your shop use for HEM?

Respondents were invited to name the CAM software that they use for HEM toolpaths. Respondents who indicated “Other,” listed such programs as Esprit, HSM Works, SolidCAM, Volumill, and others.

Why doesn't your shop use HEM?

Respondents who indicated their shops did not use HEM were asked to elaborate on why this milling strategy was not being utilized. 26% of respondents indicated that they were happy with their current machining methods. Other popular reasons included “not having the right tools,” “not having the right machine,” and “not having time to learn the strategy.”



Thinking you don't have the right tooling, software, or machines? Learn about what's required in “Intro to HEM,” starting on page 7.

2

INTRODUCTION TO HIGH EFFICIENCY MILLING





INTRODUCTION TO HIGH EFFICIENCY MILLING

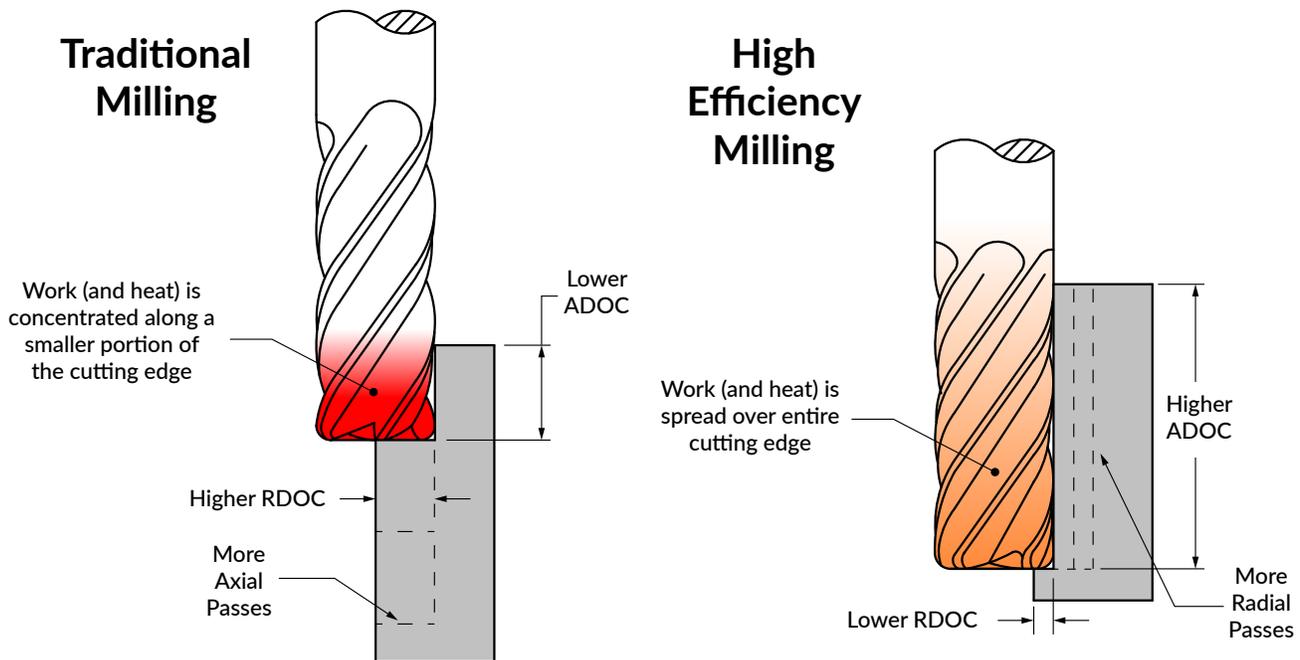
High Efficiency Milling (HEM) is a strategy that is rapidly gaining popularity in the metalworking industry. Most CAM packages now offer modules to generate HEM toolpaths, each with their own proprietary name. In these packages, HEM can also be known as Dynamic Milling or High Efficiency Machining, among others. HEM can result in profound shop efficiency, extended tool life, greater performance, and cost savings. High performance end mills designed to achieve higher speeds and feeds will help machinists to reap the full benefits of this popular machining method.

High Efficiency Milling Defined

HEM is a milling technique for roughing that utilizes a lower Radial Depth of Cut (RDOC) and a higher Axial Depth of Cut (ADOC). This spreads wear evenly across the cutting edge, dissipates heat, and reduces the chance of tool failure.

HEM is a milling technique for roughing that utilizes a lower Radial Depth of Cut (RDOC) and a higher Axial Depth of Cut (ADOC).

This strategy differs from traditional or conventional milling, which typically calls for a higher RDOC and lighter ADOC. Traditional milling causes heat concentrations in one small portion of the cutting tool, expediting the tool wear process. Further, while Traditional Milling calls for more axial passes, HEM toolpaths use more passes radially.



Built-In CAM Applications

Machining technology has been advancing with the development of faster, more powerful machines. In order to keep up, many CAM applications have developed built-in features for HEM toolpaths, including Trochoidal Milling, a method of machining used to create a slot wider than the cutting tool's cutting diameter.

HEM is largely based on the theory surrounding Radial Chip Thinning, or the phenomenon that occurs with varying RDOC, and relates to the chip thickness and feed per tooth. HEM adjusts parameters to maintain a constant load on the tool through the entire roughing operation, resulting in more aggressive material removal rates (MRR). In this way, HEM differs from other high performance toolpaths, which involve different methods for achieving significant MRR.

HEM adjusts parameters to maintain a constant load on the tool through the entire roughing operation, resulting in more aggressive MRR.

Virtually any CNC machine can perform HEM – the key is a fast CNC controller. When converting from a regular program to HEM, about 20 lines of HEM code will be written for every line of regular code. A fast processor is needed to look ahead for the code, and keep up with the operation. In addition, advanced CAM software that intelligently manages tool load by adjusting the IPT and RDOC is also needed.

Virtually any CNC machine can perform HEM – the key is a fast CNC controller.

HEM Case Studies

The following example shows the result a machinist had when using a Helical Solutions HEV-5 tool to perform an HEM operation in 17-4PH stainless steel (35 Rc). While performing HEM, this ½” diameter, 5-flute end mill engaged the part just 12% radially, but 100% axially. This machinist was able to reduce tool wear and was able to complete 40 parts with a single tool, versus only 15 with a traditional roughing toolpath.



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 | 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% |

The effect of HEM on a roughing application can also be seen in the case study below. While machining 6061 aluminum with Helical's H45AL-C-3, a 1/2", 3-flute rougher, this machinist was able to finish a part in 3 minutes, versus 11 minutes with a traditional roughing toolpath. One tool was able to make 900 parts with HEM, a boost of more than 150% over the traditional method.



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

| | RPM | IPM | RDOC | ADOC | MRR | Cycle Time per Part | Parts per 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% |

Importance of Tooling to HEM

Generally speaking, HEM is a matter of running the tool – not the tool itself. Virtually every tool can perform HEM, but using tooling built to withstand the rigors of HEM will result in greater success. While you can run a marathon in any type of shoes, you'd likely get the best results and performance from running shoes.

HEM is often regarded as a machining method for larger diameter tooling because of the aggressive MRR of the operation and the fragility of tooling under 1/8" in size. However, miniature tooling can be used to achieve HEM, too.

Using miniature tooling for HEM can create additional challenges that must be understood prior to beginning your operation.

Best Tools for HEM

- ▶ High flute count for increased MRR
- ▶ Large core diameter for added strength
- ▶ Tool coating optimized for the workpiece material for increased lubricity
- ▶ Variable pitch/variable helix design for reduced harmonics

Key Takeaways

HEM is a machining operation which continues to grow in popularity in shops worldwide. A milling technique for roughing that utilizes a lower RDOC and higher ADOC than traditional milling, HEM distributes wear evenly across the cutting edge of a tool, reducing heat concentrations and slowing the rate of tool wear. This is especially true in tooling best suited to promote the benefits of HEM.

3

HIGH SPEED MACHINING VS. HEM





HIGH SPEED MACHINING VS. HEM

Advancements in the metalworking industry have led to new, innovative ways of increasing productivity. One of the most popular ways of doing so (creating many new buzzwords in the process) has been the discovery of new, high-productivity toolpaths. Terms like trochoidal milling, high speed machining, adaptive milling, feed milling, and High Efficiency Milling are a handful of the names given to these cutting-edge techniques.

With multiple techniques being described with somewhat similar terms, there is some confusion as to what each is referring to. High Efficiency Milling (HEM) and High Speed Machining (HSM) are two commonly used terms and techniques that can often be confused with one another. Both describe techniques that lead to increased material removal rates and boosted productivity. However, the similarities largely stop there.

High Speed Machining

High speed machining is often used as an umbrella term for all high productivity machining methods including HEM. However, HEM and HSM are unique, separate machining styles. HSM

High Efficiency Milling (HEM) and High Speed Machining (HSM) are two commonly used terms and techniques that can often be confused with one another.

encompasses a technique that results in higher production rates while using a much different approach to depth of cut and speeds and feeds. While certain HEM parameters are constantly changing, HSM uses constant values for the key parameters. A very high spindle speed paired with much lighter axial depths of cut results in a much higher allowable feed rate. This is also often referred to as feed milling. Depths of cut involve a very low axial and high radial components. The method in general is often thought of as z-axis slice machining, where the tool will step down a fixed amount, machine all it can, then step down the next fixed amount and continue the cycle.

High speed machining techniques can also be applied to contoured surfaces using a ball profile or corner radius tool. In these situations, the tool is not used in one plane at a time, and will follow the 3 dimensional curved surfaces of a part. This is extremely effective for using one tool to bring a block of material down to a final (or close to final) shape using high resultant material removal rates paired with the ability to create virtually any shape.

Effectively, HEM uses the depths associated with a traditional finishing operation but boosts speeds and feeds, resulting in much higher material removal rates.

High Efficiency Milling

HEM has evolved from a philosophy that takes advantage of the maximum amount of work that a tool can perform. Considerations for chip thinning and feed rate adjustment are used so that each cutting edge of a tool takes a consistent chip thickness with each rotation, even at varying radial depths of cut and while interpolating around curves. This allows machinists the opportunity to utilize a radial depth of cut that more effectively uses the full potential of a given tool. Utilizing the entire available length of cut allows tool wear to be spread over a greater area, prolonging tool life and lowering production costs. Effectively, HEM uses the depths associated with a traditional finishing operation but boosts speeds and feeds, resulting in much higher material removal rates (MRR). This technique is typically used for hogging out large volumes of material in roughing and pocketing applications.

In short, HEM is somewhat similar to an accelerated finishing operation in regards to depth of cut, while HSM is more of a high feed contouring operation. Both can achieve increased MRR and higher productivity when compared to traditional methods. While HSM can be seen as an umbrella term for all high efficiency paths, HEM has grown in popularity to a point where it can be classified on its own. Classifying each separately takes a bit of clarification, showing they each have power in certain situations.

4

COMBAT CHIP THINNING & BOOST TOOL POTENTIAL





HOW TO COMBAT CHIP THINNING

Defining Chip Thinning

Chip Thinning is a phenomenon that occurs with varying Radial Depths Of Cut (RDOC), and relates to chip thickness and feed per tooth. While these two values are often mistaken as the same, they are separate variables that have a direct impact on each other. Feed per tooth translates directly to your tool feed rate, and is commonly referred to as Inches Per Tooth (IPT) or chip load.

Chip Thickness

Chip thickness is often overlooked. It refers to the actual thickness of each chip cut by a tool, measured at its largest cross-section. Users should be careful not to confuse chip thickness and feed per tooth, as these are each directly related to the ideal cutting conditions.

Users should be careful not to confuse chip thickness and feed per tooth, as these are each directly related to the ideal cutting conditions.

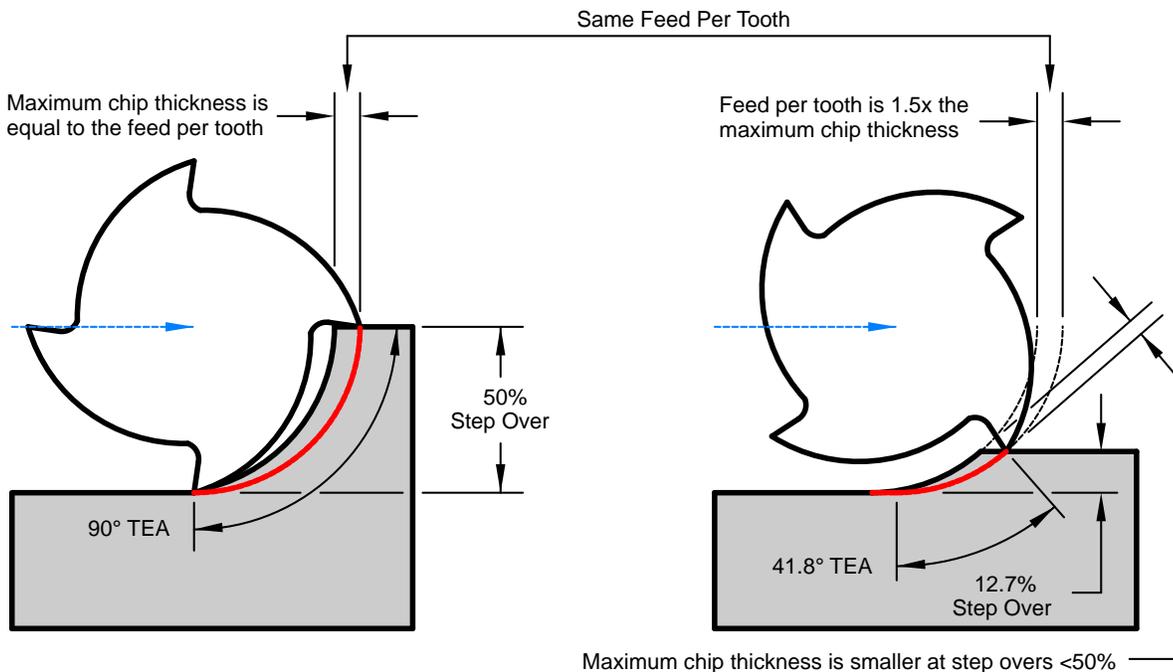
How Chip Thinning Occurs

When using a 50% step over (left side of Figure 1), the chip thickness and feed per tooth are equal to each other. Each tooth will engage the workpiece at a right angle, allowing for the most effective cutting action, and avoiding rubbing as much as possible.

Once the RDOC falls below 50% of the cutter diameter (right side of Figure 1), the maximum chip thickness decreases, in turn changing the ideal cutting conditions of the application. This can lead to poor part finish, inefficient cycle times, and premature tool wear. Properly adjusting the running parameters can greatly help reduce these issues.

Properly adjusting the running parameters can greatly help reduce issues like poor part finish and inefficient cycle times.

Figure 1



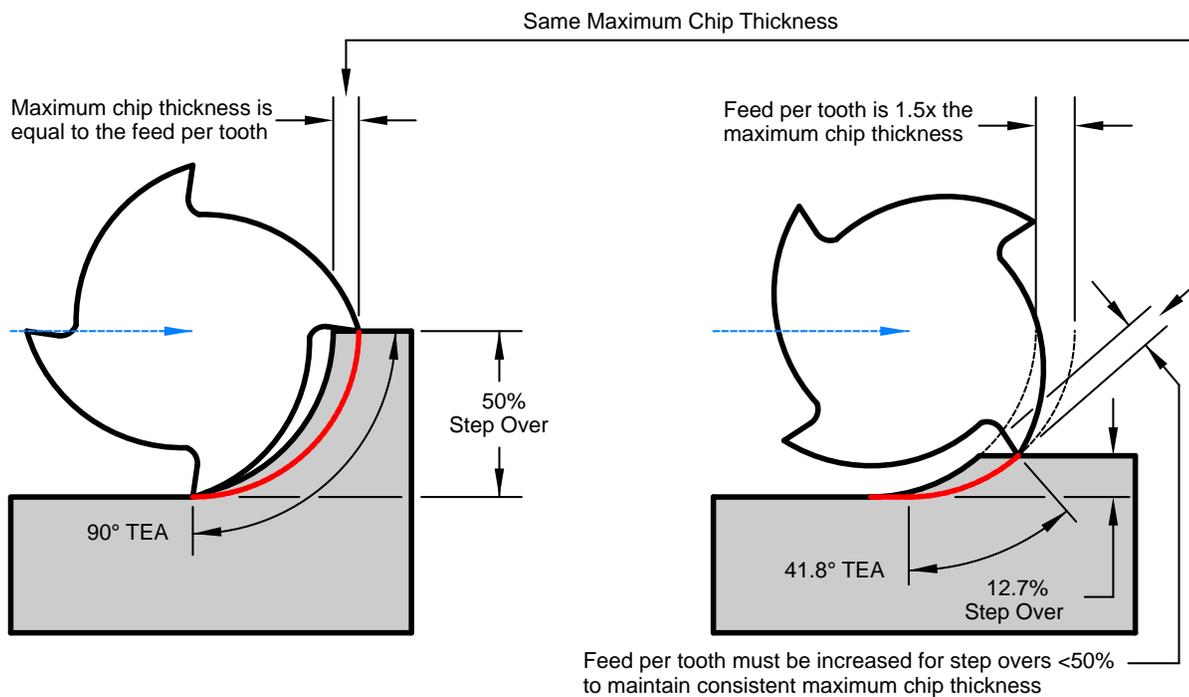
The aim is to achieve a constant chip thickness by adjusting the feed rate when cutting at different RDOC. This can be done with the following equation using the Tool Diameter (D), RDOC, Chip Thickness (CT), and Feed Rate (IPT). For chip thickness, use the recommended value of IPT at 50% step over. Finding

an adjusted feed rate is as simple as plugging in the desired values and solving for IPT. This keeps the chip thickness constant at different depths of cut. The adjustment is illustrated in Figure 2.

Inches Per Tooth
(Chip Thinning Adjustment)

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

Figure 2



Lasting Benefits

In summary, the purpose of these chip thinning adjustments is to get the most out of your tool. Keeping the chip thickness constant ensures that a tool is doing as much work as it can within any given cut. Other benefits include: reduced rubbing, increased material removal rates, and improved tool life.

5

DIVING INTO DEPTH OF CUT



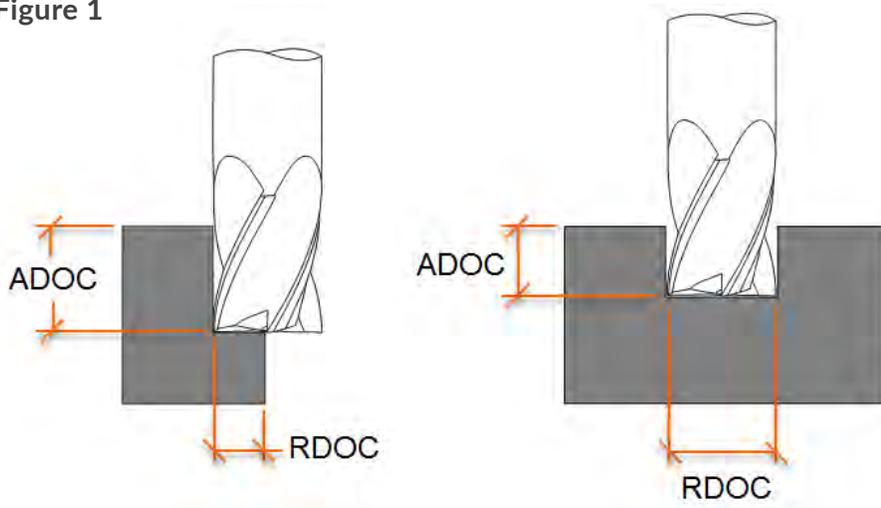


DIVING INTO DEPTH OF CUT

Every CNC machining operation requires a radial and axial depth of cut strategy. Radial depth of cut (RDOC), the distance a tool is stepping over into a workpiece; and Axial depth of cut (ADOC), the distance a tool engages a workpiece along its centerline, are the backbones of machining. Machining to appropriate depths – whether slotting or peripheral milling (profiling, roughing, and finishing) – is vital to your machining success (Figure 1).

Machining to appropriate depths – whether slotting or peripheral milling – is vital to your machining success.

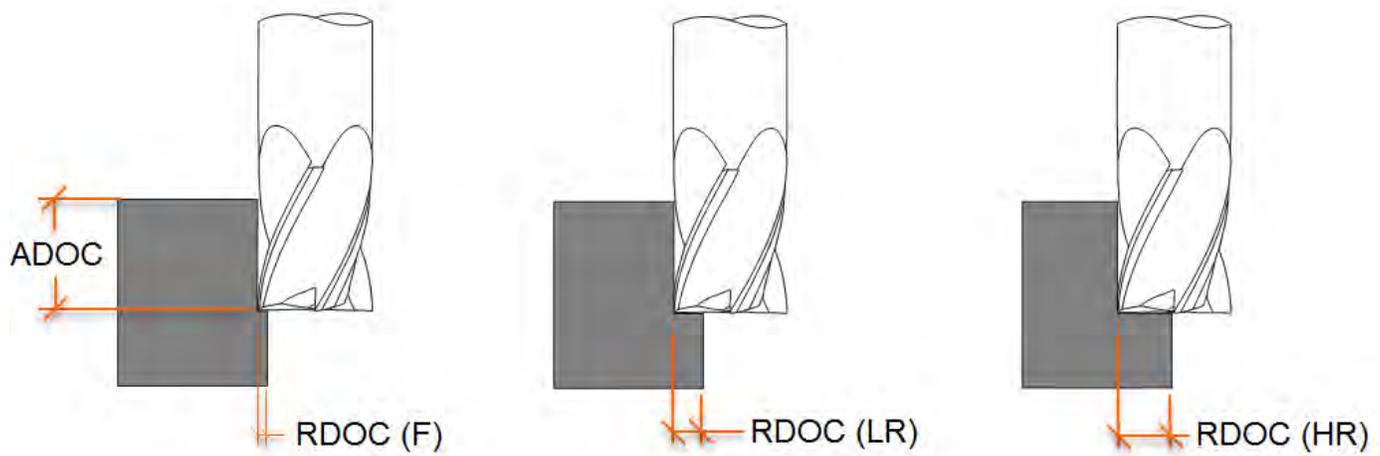
Figure 1



Peripheral Milling Styles & Appropriate RDOC

The amount a tool engages a workpiece radially during peripheral milling is dependent upon the operation being performed (Figure 2). In finishing applications, smaller amounts of material are removed from a wall, equating to about 3-5% of the cutter diameter per radial pass.

Figure 2



In heavy roughing applications, 30-50% of the tool's cutter diameter is engaged with the part. Although heavy roughing involves a higher RDOC than finishing, the ADOC is most often smaller than for finishing due to load on the tool.

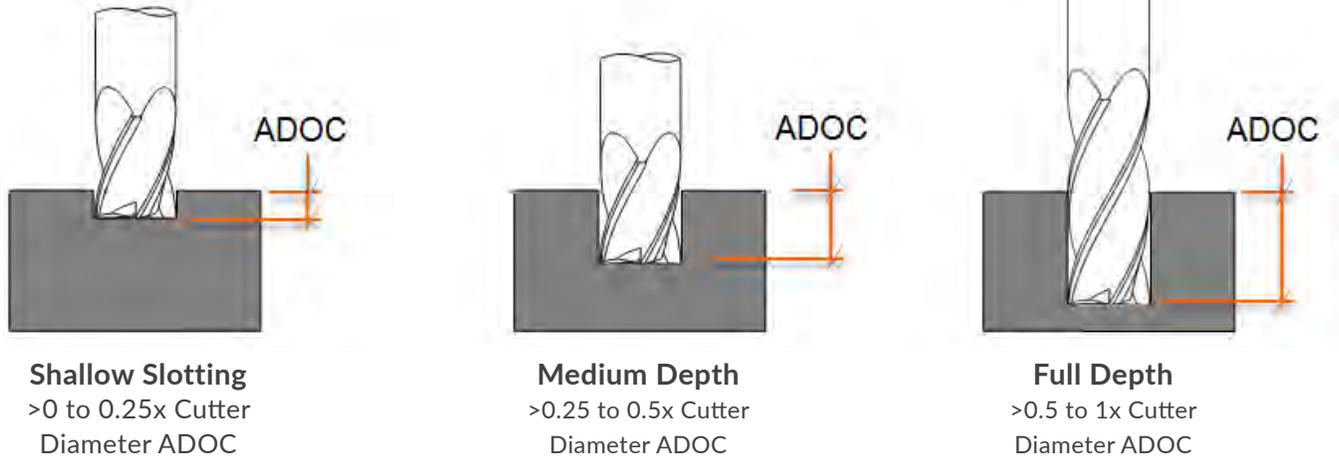
Slotting Styles & Appropriate ADOC

The amount a tool engages a part axially during a slotting operation must be appropriate for the tool being used (Figure 3). Using an inappropriate approach could lead to tool deflection and damage, and poor part quality.

End mills come in various length of cut options, as well as numerous reached options. Choosing the tool that allows the completion of a project with the least deflection, and highest productivity, is critical. As the ADOC needed to slot can be lower, a stub length of cut is often the strongest and most appropriate tool choice. As slot depths increase, longer lengths of cut become necessary, but reached tooling should be used where allowable.

Choosing the tool that allows the completion of a project with the least deflection, and highest productivity, is critical.

Figure 3



Slotting Styles & Appropriate ADOC Engagement

The amount a tool engages a part axially during a slotting operation must be appropriate for the tool being used (Figure 3). Using an inappropriate approach could lead to tool deflection, damage, and poor part quality.

End mills come in various length of cut options, as well as numerous reached options. Choosing the tool that allows the completion of a project with the least deflection, and highest productivity, is critical. As the ADOC needed to slot can be lower, a stub length of cut is often the strongest and most appropriate tool choice. As slot depths increase, longer lengths of cut become necessary, but reached tooling should be used where allowable.

Depth of Cut Strategy for High Efficiency Milling

When High Efficiency Milling, using a light RDOC and heavy ADOC is essential. With this machining style, feed rates can be increased and cuts are kept uniform to evenly distribute stresses across the cutting portion of the tool, prolonging tool life.

Traditional Strategy

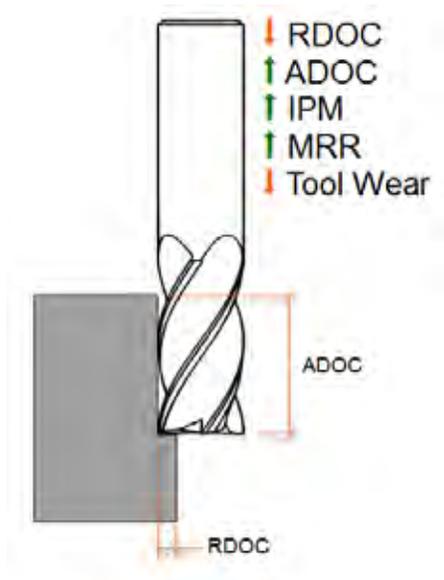
- ▶ Heavy RDOC
- ▶ Light ADOC
- ▶ Conservative Feed Rate

High Efficiency Milling (HEM)

- ▶ Light RDOC
- ▶ Heavy ADOC
- ▶ Increased Feed Rate

HEM involves using 7-30% of the tool diameter radially and up to twice the cutter diameter axially, paired with increased feed rates (Figure 4).

Figure 4



Accounting for chip thinning, this combination of running parameters can result in noticeably higher material removal rates (MRR). Modern CAM software often offers a complete high performance solution with built-in features for HEM toolpaths. These principals can also be applied to trochoidal toolpaths for slotting applications.

With HEM, feed rates can be increased and cuts are kept uniform to evenly distribute stresses across the cutting portion of the tool.

6

PREVENTING TOOL WEAR





PREVENTING TOOL WEAR

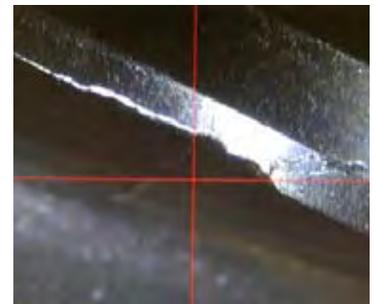
Defining Tool Wear

Tool wear is the breakdown and gradual failure of a cutting tool due to regular operation. Every tool will experience tool wear at some point in its life. Excessive wear will show inconsistencies and have unwanted effects on your workpiece, so it is important to avoid tool wear in order to achieve optimal end mill performance. Tool wear can also lead to failure, which in turn can lead to serious damage, rework, and scrapped parts.

To prolong tool life, identifying and mitigating the various signs of tool wear is key. Both thermal and mechanical stresses cause tool wear, with heat and abrasion being the major culprits. Learning how to identify the most common types of tool wear and what causes them can help machinists remedy issues quickly and extend tool longevity.



An example of a carbide cutting tool with no wear.



An example of a carbide cutting tool with excessive wear.

Abrasive Wear

The wear land is a pattern of uniform abrasion on the cutting edge of the tool, caused by mechanical abrasion from the workpiece. This dulls the cutting edge of a tool, and can even alter dimensions such as the tool diameter. At higher speeds, excessive heat becomes more of an issue, causing more damage to the cutting edge, especially when an appropriate tool coating is not used.



If the wear land becomes excessive or causes premature tool failure, reducing the cutting speed and optimizing coolant usage can help.

If the wear land becomes excessive or causes premature tool failure, reducing the cutting speed and optimizing coolant usage can help. High Efficiency Milling (HEM) toolpaths offer another way to combat tool wear by spreading the work done by the tool over its entire length of cut. This prevents localized wear and will prolong tool life by using the entire cutting edge available.

Chipping

Chipping can be easily identified by a nicked or flaked edge on the cutting tool, or by examining the surface finish of a part. A poor surface finish can often indicate that a tool has experienced some sort of chipping, which can lead to eventual catastrophic tool failure if it is not caught.

Chipping is typically caused by excessive loads and shock-loading during operation, but it can also be caused by thermal cracking, another type of tool wear which is explored in further detail below. To counter chipping, ensure the milling operation is completely free of vibration and chatter. Taking a look at the speeds and feeds can also help. Interrupted cuts and repeated part entry can also have a negative impact on a tool. Reducing feed rates for these situations can mitigate the risk of chipping.



To counter chipping, ensure the milling operation is completely free of vibration or chatter.

Thermal Cracking

Thermal cracking is often identified by cracks in the tool perpendicular to the cutting edge. Cracks form slowly, but they can lead to both chipping and premature tool failure.



Thermal cracking, as its name suggests, is caused by extreme temperature fluctuations during milling. Adding a proper coating to an end mill is beneficial in providing heat resistance and reduced abrasion on a tool. HEM toolpaths provide excellent protection against thermal cracking, as these toolpaths spread the heat across the cutting edge of the tool, reducing the overall temperature and preventing serious fluctuations in heat.

Fracture

Fracture is the complete loss of tool usage due to sudden breakage, often as a result of improper speeds and feeds, an incorrect coating, or an inappropriate depth of cut. Tool holder issues or loose work holding can also cause a fracture, as can inconsistencies in workpiece material properties.



Adjusting the speeds, feeds, and depth of cut and checking the setup for rigidity will help to reduce fracturing. Optimizing coolant usage can also be helpful to avoid hot spots in materials which can dull a cutting edge and cause a fracture. HEM toolpaths prevent fracture by offering a more consistent load on a tool. Shock loading is reduced, causing less stress on a tool, which lessens the likelihood of breakage and increases tool life.

HEM toolpaths provide excellent protection against thermal cracking, as these toolpaths spread the heat across the cutting edge of the tool.

Preventing Tool Wear

It is important to monitor tools and keep them in good, working condition to avoid downtime and save money. Wear is caused by both thermal and mechanical forces, which can be mitigated by running with appropriate running parameters and HEM toolpaths to spread wear over the entire length of cut. While every tool will eventually experience some sort of tool wear, the effects can be delayed by paying close attention to speeds and feeds and depth of cut. Preemptive action should be taken to correct issues before they cause complete tool failure.

The effects of tool wear can be delayed by paying close attention to speeds and feeds and depth of cut.

7

APPLYING HEM TO MICROMACHINING





APPLYING HEM TO MICROMACHINING

Benefits of Using HEM with Miniature Tooling

High Efficiency Milling (HEM) is a technique for roughing that utilizes a lower Radial Depth of Cut (RDOC), and a higher Axial Depth of Cut (ADOC). This delays the rate of tool wear, reducing the chance of failure and prolonging tool life while boosting productivity and Material Removal Rates (MRR). Because this machining method boosts MRR, miniature tooling (<.125") is commonly overlooked for HEM operations. Further, many shops also do not have the high RPM capabilities necessary to see the benefits of HEM for miniature tooling. However, if used properly, miniature tooling can produce the same benefits of HEM that larger diameter tooling can. Benefits of HEM include:

- ▶ Extended tool life and performance
- ▶ Faster cycle times
- ▶ Overall cost savings

If done properly, miniature tooling can reap the same benefits of HEM that larger diameter tooling can.

Preventing Common Challenges

Utilizing miniature tooling for HEM, while beneficial if performed correctly, presents several challenges that all machinists must be mindful of. Knowing what to keep an eye out for is a pivotal first step to success.



Tool Fragility & Breakage

Breakage is one of the main challenges associated with utilizing HEM with miniature tooling due to the fragility of the tool. Spindle runout and vibration, tool deflection, material inconsistencies, and uneven loading are just some of the problems which can lead to a broken tool. To prevent this, more attention must be paid to the machine setup and material to ensure the tools have the highest chance of success.

As a general rule, HEM should not be considered when using tools with cutting diameters less than .031". While possible, HEM may still be prohibitively challenging or risky at diameters below .062", and your application and machine must be considered carefully.

HEM may still be prohibitively challenging or risky at diameters below .062", and your application and machine must be considered carefully.

Techniques to Prevent Tool Failure

- ▶ Ensure work piece is secure and supported
- ▶ Use the shortest overall length and length of cut as possible
- ▶ Check tool runout in the spindle and utilize shrink fit holders if possible
- ▶ Choose a coating optimized for your material

Excessive Heat & Thermal Shock

Due to the small nature of miniature tooling and the high running speeds they require, heat generation can quickly become an issue. When heat is not controlled, the workpiece and tooling may experience thermal cracking, melting, burning, built up edge, or warping.

To combat high heat, coolant is often used to decrease the surface temperature of the material as well as aid in chip evacuation and lubricity. However, care must be taken to ensure that using coolant doesn't cool the material too quickly or unevenly. If an improper coolant method is used, thermal shock can occur. Thermal shock happens when a material expands unevenly, creating micro fractures that propagate throughout the material and can crack, warp, or change the physical properties of the material.

Techniques to Prevent Heat & Thermal Shock

- ▶ Run your coated tool dry or with compressed air while ensuring sufficient chip evacuation.
- ▶ Choose a coating optimized for your material.
- ▶ Use tooling with geometry specific to your workpiece material.
- ▶ Decrease speed (RPM).

If performed properly, miniature tooling (<.125") can reap the same benefits of HEM that larger diameter tooling can: reduced tool wear, accelerated part production rates, and greater machining accuracy. However, more care must be taken to monitor the machining process and to prevent tool fragility, excessive heat, and thermal shock.

Thermal shock occurs when a material expands differently in some areas than others, creating micro fractures propagating throughout the material.

8

BEST PRACTICES FOR TROCHOIDAL MILLING





BEST PRACTICES FOR TROCHOIDAL MILLING

What Is Trochoidal Milling?

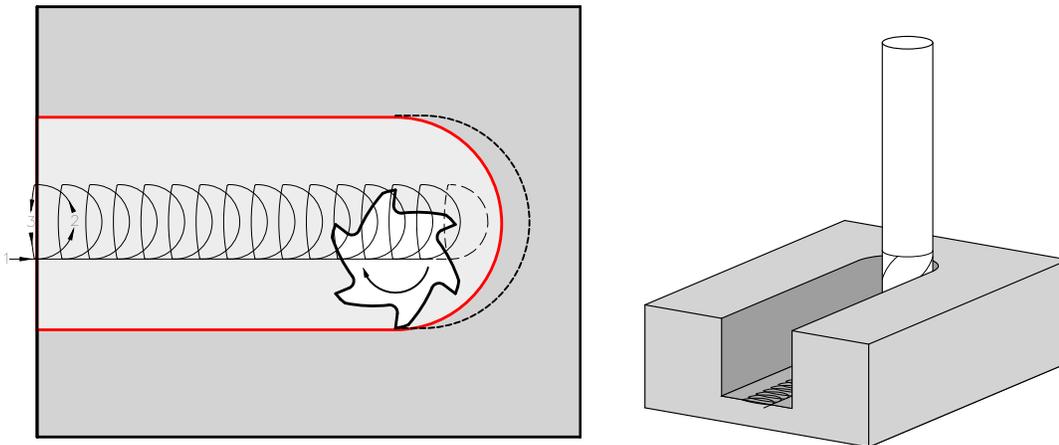
Trochoidal milling is a method of machining used to create a slot wider than the cutting tool's cutting diameter. This is accomplished using a series of circular cuts known as a trochoidal tool path. A form of High Efficiency Milling (HEM), trochoidal milling leverages high speeds while maintaining a low radial depth of cut (RDOC) and a high axial depth of cut (ADOC).

Trochoidal milling is largely based on the theory surrounding chip thinning in machining. Conventional thinking suggests that cutting tools have an optimal chip load that determines the ideal width and size of the chips produced. The concept of combating chip thinning involves machining with a chip load that is larger than "optimal" in order to maintain a constant maximum chip thickness.

Trochoidal milling leverages high speeds while maintaining a low radial depth of cut (RDOC) and a high axial depth of cut (ADOC).

In contrast to a completely linear radial tool path in conventional machining, trochoidal milling takes advantage of a spiral tool path with a low RDOC to reduce load and wear on the tool (Figure 1).

Figure 1



Advantages of Trochoidal Milling

Trochoidal milling can be very advantageous in certain applications. The reduced radial engagement of the cutting edge decreases the amount of heat produced in the cut while also decreasing the cutting forces and load on the spindle. The reduced radial forces allow for greater accuracy during production and make it possible to machine finer and more precise features on a part. Other advantages include:

- ▶ Decreased cutting forces
- ▶ Reduced heat
- ▶ Greater machining accuracy
- ▶ Improved tool life
- ▶ Faster cycle times
- ▶ One tool for multiple slot sizes

In addition, the lower radial depth of cut allows for a higher axial depth of cut, meaning that the entire length of the cutting edge can be utilized. This ensures that heat and cutting forces are distributed across the tool's cutting edge, rather than

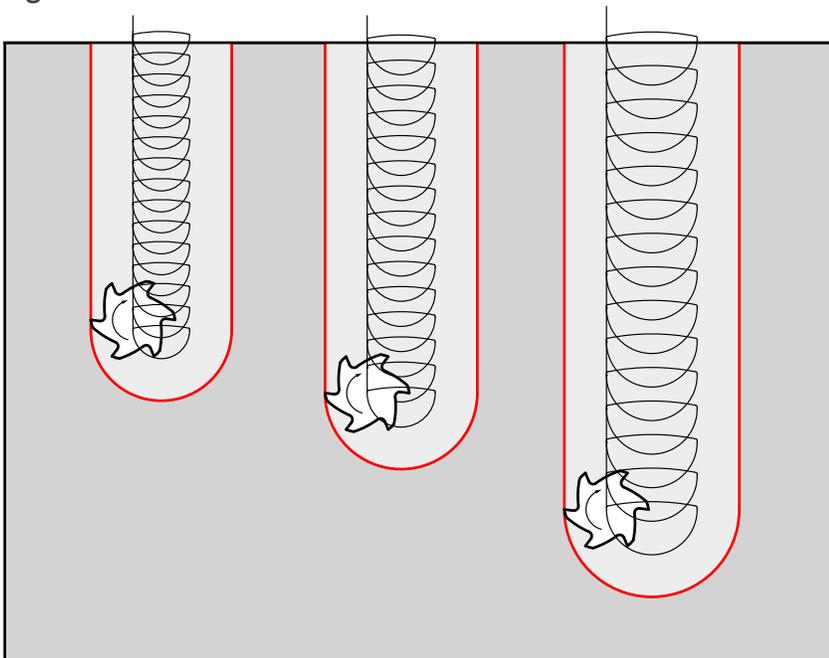
The reduced radial engagement of the cutting edge decreases the amount of heat produced in the cut while also decreasing the cutting forces.

concentrated on a single section. The reduced heat and wear, combined with their uniform spread on the cutting edge, result in significantly improved tool life over conventional slotting methods.

Given the reduced destructive forces, the cutting tool's speeds can be increased. Since the entire length of cut is utilized, trochoidal milling can eliminate the need for multiple axial depths of cut. Increased running parameters and a reduced number of passes greatly reduce cycle time.

Since trochoidal milling uses a tool to machine a slot wider than its cutting diameter, the same tool can be used to create slots of varying sizes, rather than just one. This can free up space in your tool carousel and save time on tool change outs, depending on the requirements of the part (Figure 2).

Figure 2



Although slotting is a roughing operation, the reduced radial depth of cut and decreased cutting forces from trochoidal milling often result in an improved finish over a conventional slotting toolpath. However, a finishing pass along the walls of the workpiece might be required to remove any cusps left from the spiral motion of the cutting tool.

The reduced radial depth of cut and decreased cutting forces from trochoidal milling often result in an improved finish over a conventional slotting toolpath.

Challenges of Trochoidal Milling

The challenges of trochoidal milling are typically found with the machinery and software. The right machine to take advantage of trochoidal milling will not only be capable of high speeds and feeds, but will also be capable of a constantly changing feed rate as the tool moves along its spiral path. Inability to have a changing feed rate will cause chip thinning which can yield non-ideal results and potentially cause tool breakage.

Special software might also be required to program tool paths and feed rates for this process. This is further complicated by factors like the ratio of the cutter diameter to the size of the groove, as well as the radial depth of cut for these different ratios. Most figures suggest the cutter diameter be 50%-70% of the final slot width, while the radial depth of cut should equal 10%-35% of cutter diameter (Table 1), but the safest option is always to consult the tool manufacturer.

Inability to have a changing feed rate will cause chip thinning which can yield non-ideal results.

Table 1

| Cutter Diameter | RDOC (Minimum) | RDOC (Maximum) | Slot Width (Minimum) | Slot Width (Maximum) |
|-----------------|----------------|----------------|----------------------|----------------------|
| 1/2" | .050" | .175" | .714" | 1.000" |
| 1/4" | .025" | .087" | .357" | .500" |
| 1/8" | .012" | .043" | .178" | .250" |
| 1/16" | .006" | .021" | .089" | .125" |
| 1/32" | .003" | .010" | .044" | .062" |

Trochoidal Milling & Micromachining

Benefits When Micromachining

Micromachining can also benefit from trochoidal milling. The decreased radial engagement and lower cutting forces produced during a trochoidal tool path put less force on the cutting tools. This is especially important for smaller diameter tools, as they are weaker and less rigid, and the reduced cutting forces decrease the chance of deflection and breakage.

Challenges When Micromachining

While trochoidal milling with miniature tooling is theoretically beneficial, there are additional challenges associated with smaller tools. Miniature cutting tools are much more susceptible to breakage due to spindle runout and vibration, material inconsistencies, uneven loading, and many other variables that arise during machining. Depending on your application, it may be worth using the tool with the greatest diameter for the extra strength. Although there are potential benefits at the miniature level, more attention must be paid to the machine setup and material to ensure the tools have the highest chance of success.

Just like HEM, as a general rule, trochoidal milling should not be considered when using tools with cutting diameters less than .031". While possible, trochoidal milling may still be prohibitively challenging or risky at diameters below .062", and your application and machine must be considered carefully.

Depending on your application, it may be worth using the tool with the greatest diameter for the extra strength.

9

HEM FAQs





HEM FAQs

What is High Efficiency Milling (HEM)?

HEM is a strategy for milling that utilizes a lower Radial Depth of Cut (RDOC) and a higher Axial Depth of Cut (ADOC) paired with a variable, optimized feed rate. This technique spreads wear evenly across the cutting edge, dissipates heat, and reduces the chance of tool failure.

Are there other names for HEM that are used in the industry?

Most CAM packages now offer modules to generate HEM toolpaths, each with their own proprietary names. In these packages, HEM can also be known as Dynamic Milling or High Efficiency Machining, among others.

Is HEM the same thing as High Speed Machining (HSM)?

HEM and HSM are both very popular machining methods that boost material removal rates (MRR) and increased productivity. While HEM is similar to an accelerated finishing operation in regards to depth of cut, HSM is more comparable to a high feed contouring operation.

While HEM is similar to an accelerated finishing operation in regards to depth of cut, HSM is more comparable to a high feed contouring operation.

What are the benefits of HEM?

HEM can result in profound shop efficiency, extended tool life, greater performance, and cost savings.

How does HEM differ from traditional milling?

Traditional or conventional milling typically calls for higher RDOC and lower ADOC. Traditional methods call for more axial passes, while HEM toolpaths use more passes radially. Traditional paths use constant feed rates, while HEM compensates for chip thinning and varying depth of cut with a changing feed rate.

What is Trochoidal Milling?

Trochoidal Milling is a form of HEM that creates a slot wider than the cutting tool's cutting diameter. This method leverages high speeds while maintaining a low RDOC and high ADOC. It is largely based on the concept of chip thinning.

What is Chip Thinning?

Chip Thinning is a phenomenon that occurs with varying RDOC. When using a step over that falls below 50 percent of the cutter diameter, the maximum chip thickness decreases and changes the ideal cutting conditions of the application. This can lead to subpar finish and accelerated tool wear. To combat chip thinning, adjust parameters to machine with a chip load that is larger than "optimal" to ensure that your tool is maintaining a constant, efficient chip thickness.

What are the best tools for HEM?

High performance end mills designed to achieve higher speeds and feeds will help machinists to reap the full benefits of HEM. In addition, tools with high flute counts, large core diameters, and Variable Pitch/Variable Helix designs have proven to be better for achieving great results with HEM.

What machinery is needed for HEM?

Virtually any CNC machine can perform HEM. However, a fast CNC controller is needed to read the increased number of lines of code needed for HEM. When converting from a regular program to HEM, about 20 lines of HEM code will be written for every line of traditional code.

While HEM is similar to an accelerated finishing operation in regards to depth of cut, HSM is more comparable to a high feed contouring operation.

10

PRO TIPS





PRO TIPS

Need more reasons to start using high efficiency milling in your shop? We asked hundreds of CNC machinists and programmers to share their best advice about using HEM techniques. Use these pro tips to improve your current process and to increase your shop efficiency.

BENEFITS

“Once you start using it, you will be amazed at the tool life.”

“[HEM] cuts cycle times and [is] easier on tooling and machines.”

“Try it, it makes a huge difference in hogging out material fast.”

“Great for hard to machine materials.”

“It’s the best way to rough low carbon steels like 1018. The abrasiveness of the material is the limiting factor in ideal conditions. Spreading the wear out is the only way to cope.”

“It is phenomenal for tool life and deep cuts where the tool’s total LOC is fully engaged. It is great for some cutting operations.”

“It looks scary at first but when you see the reliability of these toolpaths you will be a believer.”

“You can’t afford to NOT figure out where [HEM] benefits your shop, your machine, and your parts, because if you don’t your competition most certainly will.”

SPEED & FEEDS

“Trust the feeds and speeds given to you - step over is biggest thing that can change outcome.”

“More depth of cut, less width, more feed, and watch the cycle times drop.”

“Don’t be skeptical about the Speed & Feed numbers. Give them a try.”

“Make sure to get your settings correct, and this approach to machining will blow you mind. Deep axial cuts with light radial step overs is the wave of the future.”

“Don’t just slow feed rate down when proving program out due to the higher RPM’s in harder material. It will burn a cutter up really fast. Make sure to adjust spindle and feed together.”

“Start on the low end of the recommended FPT and once the cutter is in the material, increase the feed.”

“Start at a low end of surface speed, don’t forget to feed or you’ll burn your tools. Use good tool holders when ramping it up! Tools will pull out on you. Hydraulic or Weldon is the way to go!”

“Be prepared to see feed rates and depth of cuts you never thought possible, and then go even faster!”

“I like to use the Helical Milling Advisor for a speed, feed, & DOC. It has been a great resource.”

“Read up on the technique and get advice regarding speeds & feeds, RDOC, and ADOC.”

“If you don’t break off a cutter once every few months, you’re most likely not programming or machining at your machine’s capability limit. Don’t be afraid to load up that cutter.”

“Watch your spindle load. Using high axial depths of cut, especially with high helix tools, creates a lot more torque than conventional milling. High quality tool holders are important.”

“Don’t be scared! Surface speed and chipload seem high, but that’s why it works so well.”

“Jump in and never look back. It is worth it. Also, invest in a good speeds and feeds calculator. They prove invaluable with HEM milling.”

“It’s not hard, just get used to using calculators and don’t just put random feeds and speeds. Let the technology do the work for you.”

“Ensure the actual stock size is the same as modeled. Know your machine tool capabilities. Adjust the Helical Milling Advisor to your machine capabilities. Trust the calculations and let it rip.”

SETUP & SOFTWARE

“Make sure tools are running true (minimal run out) and held securely. The investment in good collet chucks, collets, and other high quality tool holding will pay for itself quickly. Trust your CAM package.”

“Provided your machine/CAM software can handle it, you should be using it!”

“Use a good backplotter software, in addition to your CAM. Trust [your] CAM software and your ability with it.

“Check cycle times between HEM and traditional machining, as sometimes the HEM paths go far in excess of traditional methods. Use Helical’s Milling Adviser to get some starting data.”

“Maximize the CNC machine tool memory. HEM programs are very long and memory hogs.”

“Expect long G-code programs.”

“Make sure you have software that will produce good tool paths for HEM.”

“Buy good software and good tools designed for the application with no or light hones”

“Chip thinning must be considered and calculated for, and requires you to run much higher feed. Which can be hard to get comfortable with.”

“Give it a shot, be conservative to start and make sure you test the toolpath types with your machines to find the sweet spot for DOC/WOC/RPM/IPM with a given material. It will pay off if you give it a genuine shot!”

“Try it. Hold the tool tight. Lots of cutting pressure. Define your stock to your largest piece.”

TOOLING

“Buy good tools with the proper geometry for the material.”

“Use a smaller tool than you used to use for that pocket corner radius.”

“Watch for tool deflection.”

“Set a baseline with proven tooling, then try HEM. Use the Helical Milling Advisor, and then let it eat.”

“Work with tooling reps to achieve optimal cutting parameters for each machine tool on the floor.”

“Don’t be afraid to push the tools harder than conventional feed and speed recommendations.”

“Push your tools!”

GENERAL

“Ask questions. Read about proper HEM techniques.”

“Make sure you have good swarf evacuation. You are going to need it.”

“Use it. :) Don’t forget chip thinning.”

“Don’t be afraid to try something new.”

“It requires a different train of thought [versus] the conventional methods of machining.”

“Don’t be afraid to be aggressive.”

“Get used to it, it is the future of machining.”

“Do it. Buy extra tools and just try to break ‘em. You’ll be surprised just how far you can push them.”

“Learn it, technology is not going to go away. Proper holders, no run out. use proper coatings for work materials.”

“You’ll wonder why you haven’t been doing it all along.”

“Calculate for chip thinning to achieve longer life and faster cycle times.”

“It is a MUST in machining now days. This IS the future.”



ABOUT THE BRANDS

Harvey Performance Company is dedicated to providing world class products, services, and solutions that increase productivity for our customers in the manufacturing and metalworking industries.

Our industry-leading brands, Harvey Tool and Helical Solutions serve specialty needs and markets with a shared commitment to delivering high quality products and superior service.



 **HARVEY TOOL**

Over 19,000 fully stocked miniature and specialty end mills. Ship today, in your machine tomorrow.



Helical 

Material-optimized high performance carbide end mills. Run faster, push harder, machine smarter.

*Harvey
Performance
Company strives
to offer unique and
innovative products
to solve challenging
machining
requirements for
our customers.*



CONTACT US

The technical support representatives and application engineers at Harvey Tool and Helical Solutions are always available to walk you through milling strategies, application support, troubleshooting, and tool selection.



CALL: 800-645-5609

EMAIL: harveytech@harveyperformance.com

VISIT: www.harveytool.com



CALL: 866-543-5422

EMAIL: helicaltech@harveyperformance.com

VISIT: www.helicaltool.com