

# Chip Thinning; Chip Thickness Control

What's the difference, and what does it mean to me?

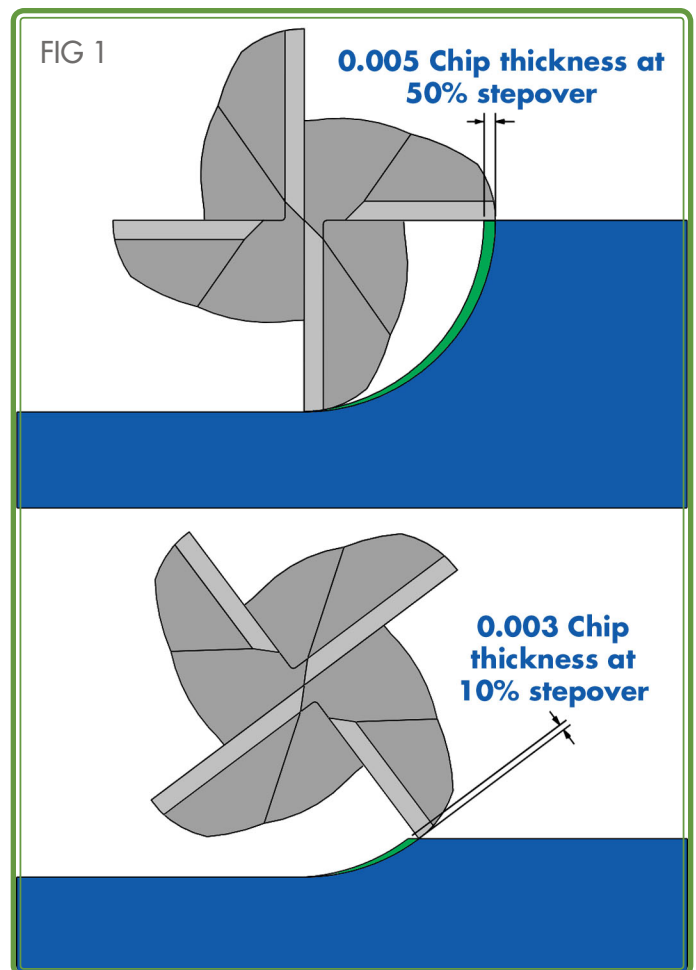
by Glenn Coleman

## Introduction

Radial chip thinning, a common topic of discussion over the past decade or so, happens in your shop all day every day, with or without your involvement. It is a real phenomenon, easily defined and illustrated, but one whose benefits have proven elusive. Chip thickness control, on the other hand, is an action you can take to significantly improve your milling operations. It is a powerful process that provides you with a great opportunity for improved productivity and reduced costs.

## Background

It is common knowledge that a chip produced through a milling process is not of uniform thickness. Assuming climb milling, the chip is thicker towards its beginning than its end. Every chip has a maximum thickness at a single point and gets gradually thinner from there. Given a constant spindle speed and feedrate, the thickness of a chip is a function of its length; the longer the chip, the thicker the chip. And the length of a chip is a function of the radial depth of cut, or cut width, established in most CAM systems with the stepover parameter. See Figure 1.



Chip load is often confused with chip thickness. In reality, however, chip load, in the prevalent use of the term, is nothing more than a feedrate expressed in inches per tooth (IPT).  $IPT \times RPM \times \#flutes = IPM$ . The IPT value is equal to the thickness of the chip if and only if the width of cut is greater than or equal to 50% of the tool diameter. When the cut width is less than 50% of tool diameter, the maximum chip thickness is less than the IPT value. Again see Figure 1. The feed per tooth values that cutting tool manufacturers recommend are valuable in calculating a feedrate, but they are not the thickness of the chips that will be produced.

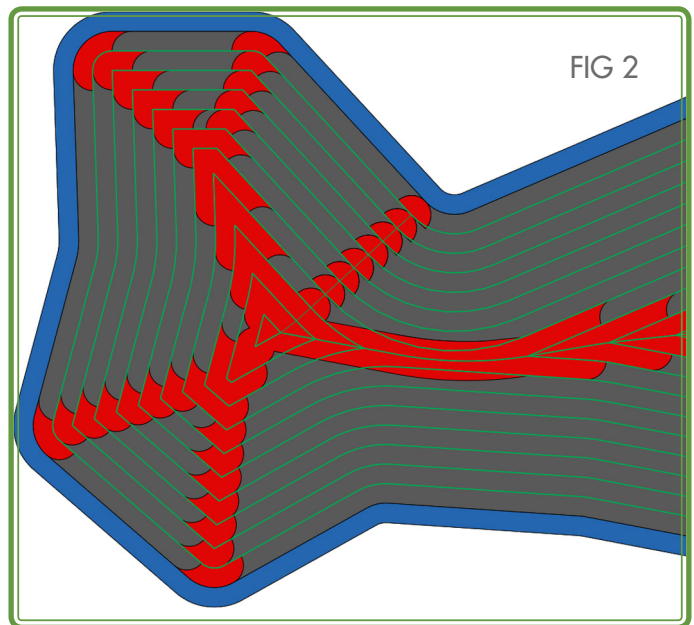
The limiting factor in the application of the radial chip thinning phenomenon is, as per usual, one of practicality rather than one of theory. The calculation is straightforward, but there are serious limitations: The chip thinning calculation is only applicable when the width of cut remains close to constant, and when the cut width is quite small to begin with. In other words, it's really only useful when finish milling vertical walls, where the cut width is small and minimally variable, for reasons put forth below. This means, of course, that this popular concept can only be used for removing about 1% of the material that gets milled on a daily basis. It is not applicable to rough milling, which is where about 99% of all material gets removed.

If radial chip thinning could be applied to rough milling, huge productivity gains could be had. Though discussions of radial chip thinning are nothing new, the ability to predictably control the maximum thickness of a chip during roughing operations very much is. The key is to forget about chip thinning, and focus on active chip thickness control.

## Varying Cut Widths

Cut widths regularly vary in rough milling. Although it is physically impossible to have no variance in cut width when rough milling, the toolpath that drives the tool can have a huge impact in minimizing that variation.

All machinists know through experience that traditional toolpaths cause uneven tool loads. Any time there is a directional change, a connection from the end of one cut to the beginning of the next, or motion through the center of an area, examples of each of which are depicted in red in Figure 2, the load increases. These increases in load are the direct result of an increase in cut width. As the cut width increases, the chip gets longer, and its thickness increases accordingly. If you have increased the feedrate so that

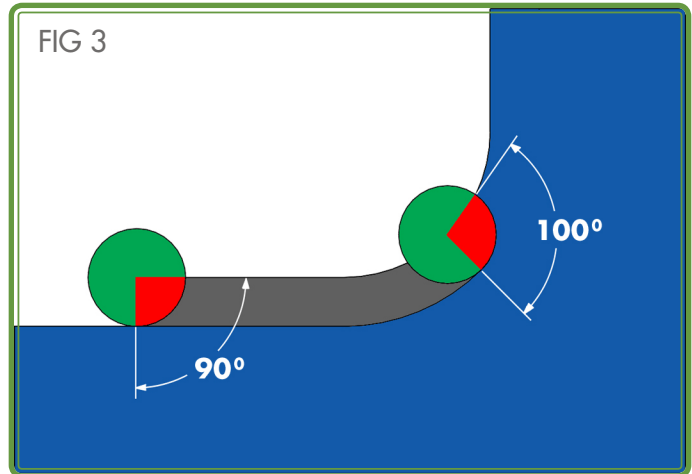


the chip thickness is at its rated maximum, the results can be catastrophic when these load increases occur, and they will occur. So applying chip thinning calculations to traditional rough milling processes is simply a non-starter.

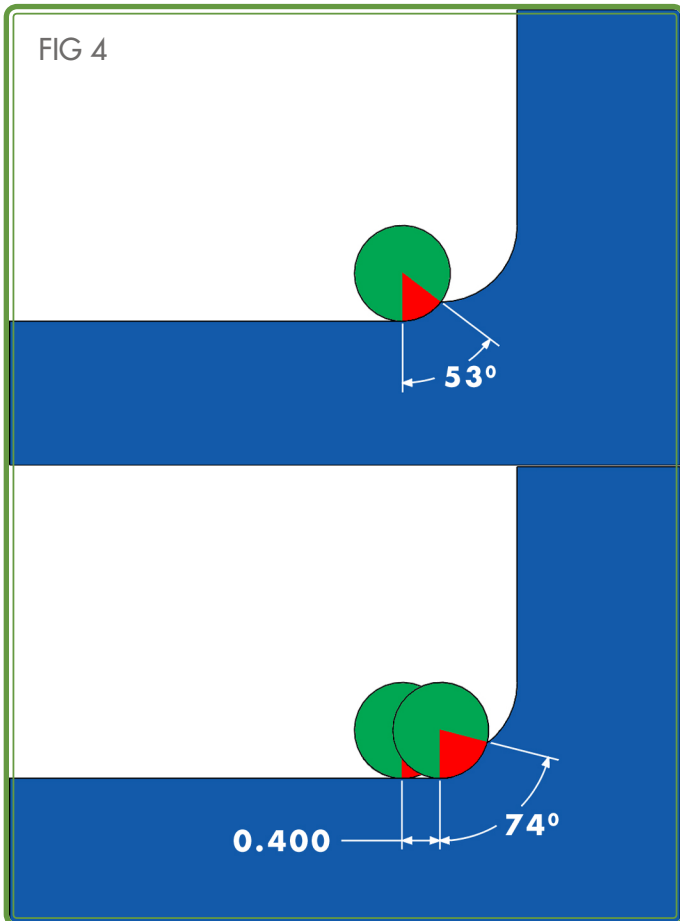
There are high-speed milling toolpaths on the market that address these changes in cut width to varying degrees, but most of these fail to produce a constant cut width. The fact that the load spikes depicted in Figure 2 (previous page) are predominantly avoided enables the use of more aggressive milling parameters. But, with one exception, these toolpath technologies actually reduce the percentage of toolpath length that cuts with a constant width. Traditional toolpaths are at least perfectly constant in cut width between their numerous instances of terribleness. Most of these new high speed technologies, in contrast, have virtually no constant cut widths. Don't be misled; these new toolpaths are superior to traditional methods, at least in specific geometric configurations, in that they avoid many of the load spikes. But they actually move further away from the ideal machining conditions of a constant-width cut, rather than closer to them.

Importantly, there is another common, though not well known, circumstance where the chip thickness, and therefore load on the tool, increases, one that happens with all existing toolpath technologies. And since it is an artifact of simple geometric axioms, it cannot

be avoided. Whenever a cutting tool transitions from cutting along a straight line onto a path of convex curvature (generally a G3 move, assuming climb milling), the engagement angle between the tool and the material increases. This happens even when the radial depth of cut, the cut width, remains perfectly constant. See Figure 3. This happens with any type of toolpath, even those designed to actively manage the



tool engagement angle. It is a simple function of cutting material with a round cutting tool, and therefore cannot be circumvented in the world of milling. The fact that this increase in engagement angle is unavoidable is clear. But the most detrimental aspect of this dynamic is that it happens so quickly. As shown in Figure 4 (see next page), the increase occurs in the span of distance/time from when the periphery of the tool intersects the point of tangency between the linear and circular portions of the current in-process material



boundary, as established by the previous cut, to when the center of the cutter is normal to the same point on the current cut, a short span indeed. This dynamic is in play because of the concentric-natured construction of toolpaths: Whenever arcs are offset from one another in successive cuts within a toolpath, they are concentric with each other. It is this geometric configuration that ensures that this spike in tool load is ever present.

To produce chips of consistent thickness, it is imperative to drive cutting tools along a path that maintains a constant width of cut, and that spreads the unavoidable

increase in tool load inherent with entering CCW arcs across a larger distance, and therefore longer time span. Fortunately this is now possible with the latest version of some new toolpath technology. The ideal cutting conditions that these constant-cut-width toolpaths provide has been proven in thousands of cases over the past few years. Combining those well established machining benefits with non-concentric spacing of arcs of adjacent cuts makes chip thickness control possible. Instead of just observing chips getting thinner as they taper to their ends, it is now possible to actively control the maximum thickness of the chips, enabling machine tools and cutting tools to perform even better, and reducing cycle times even further.

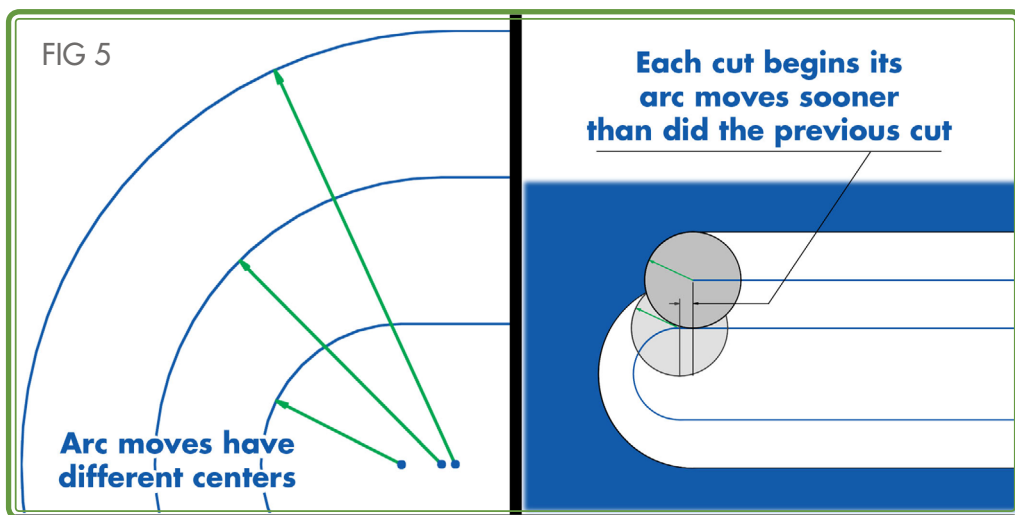
## Active Chip Thickness Control (ACTC)

Version 4.0 of VoluMill, from Celeritive Technologies, Inc., adds a specific user parameter to control the maximum thickness of the chip, a parameter that is tightly linked with the traditional parameters of spindle speed, feedrate, and stepover (cut width). Before we discuss the details of the user-control over chip thickness and discover its potential, we should address some characteristics of the VoluMill toolpath, including a key enhancement for version 4.0, that help make this possible.

Maintaining a perfectly constant width of cut during rough milling is virtually impossible except when making parallel, straight cuts, but VoluMill toolpaths

contain a much higher percentage of constant-width cuts than any other toolpath technology, high speed or traditional. And, with version 4.0, VoluMill toolpaths now avoid the inclusion of concentric arcs in their construction. Rather, arcs of successive cuts are deliberately spaced non-concentrically, in a manner designed to mitigate the problem of rapidly increasing chip thickness, a problem that toolpaths with concentric-arc construction actually exacerbate.

ness Control™ (ACTC™) is now possible. ACTC is much more than just a chip thinning calculator. It opens a new world of possibilities for controlling your milling processes and achieving significant productivity improvements over what even VoluMill itself made possible. In fact, the highest levels of productivity increases are expected to come from using this new control in a way that chip thinning was never intended to be used, chip thinning in reverse, if you will.



The new VoluMill toolpath design ensures that each CCW arc begins earlier than its inwardly adjacent neighbor, as shown in Figure 5. This doesn't eliminate the increase in engagement angle, but it does spread it over a larger area. In addition, the increase is only momentary, beginning to decrease immediately, rather than continuing throughout the length of the arc, as is the case with all other toolpath technology. With the inclusion of this non-concentric construction for VoluMill 4.0, along with the legacy benefits of predominantly constant widths of cut and automatic feedrate reductions in CCW arcs, Active Chip Thick-

Based on the values for spindle speed, feedrate, stepover, and the number of flutes on the cutting tool, VoluMill ACTC calculates and displays the thickness of the chips that the toolpath will produce. Many new possibilities now exist. In a classic chip thinning approach, the user may choose to adjust the actual chip thickness to be equal to the IPT and recalculate the feedrate. This will increase the IPM (and IPT) value so

that chips equal in maximum thickness to the original IPT will be produced, which will decrease cycle time over the original values. Or, rather than adjusting the feedrate to produce this new thickness of chip, the spindle speed can be increased, or the stepover. Any single value can be calculated on-the-fly based on the manipulation of any of the other three.

The above manipulations are well understood, but the nature of the VoluMill toolpath makes them much more applicable. Having the options in a simple user interface is certainly convenient, but there is one benefit of

the new user interface combined with the toolpath enhancements that is proving to be especially powerful. VoluMill users have found possible the use of aggressive combinations of milling parameters, combinations that result in cycle time reductions of over 90% in the best cases, and that average in the 50% to 70% range. Typically, the axial depth of cut is significantly deeper, and the spindle speed and feedrate are several times faster than is possible with traditional methods, but the cut width is usually much reduced. For example, many customers regularly machine 6Al4V titanium at 400 SFM, at .0065 IPT (for a ½-inch diameter, 5-flute cutter), two tool-diameters deep, with a stepover of 7% of tool diameter, which equates to 3,056 RPM, 100 IPM, 1.00 inch deep, with a .035 cut width. Some customers cut more aggressively than this, but these values are common, with cycle time reductions in the 65% range being the norm.

With the new capabilities in VoluMill 4.0, these cycle times can be reduced even further in the following way. This combination of parameters yields an actual chip thickness of .0033, slightly more than half of the IPT. Producing chips of this thickness in this material has long been proven effective. With the ACTC enhancement, it now takes just a single parameter adjustment to improve performance. The speeds and feeds work well together, providing uniform chip formation, good chip clearance, superior heat evacuation, and excellent tool life. So an opportunity presents itself for increasing the cut width.

Choosing to calculate a new feedrate, while doubling the stepover and maintaining the actual chip thickness

of .0033, produces a cycle time savings of approximately 35%. The doubled stepover reduces the path length by 50%. Maintaining the actual chip thickness at .0033 with the doubled stepover requires a feedrate reduction of only 26.5%, however, thus yielding the significantly reduced machining time. Since the 5-flute tool can easily clear the chips at the increased cut width, this modification in parameters works very well. So a cycle time that was originally reduced by 65% by using VoluMill toolpaths is now reduced 35% further by taking advantage of the ACTC feature in Version 4.0.

The above is just one example. As always in the wonderful world of milling, there will be physical limitations that come into play. It may be tempting to increase the cut width even further, for example, but care must be taken not to impinge the tool's ability to clear the chips as they get longer. The specific capabilities and limitations of your machining hardware will come into play, but this added toolpath generation capability will help extract the full potential from your machining assets.

## Conclusion

The shop floor-proven advantages of using VoluMill toolpaths for your rough milling needs have become even stronger. The new Active Chip Thickness Control, in combination with the now non-concentric toolpath construction, puts even more productivity-enhancing power at your fingertips.