Selecting the Proper Gear Milling Cutter Design for the Machining of High-Quality Parallel Axis, Cylindrical Gears and Splines

By Brent K. Marsh

A comprehensive, yet practical, view is presented to assist the manufacturing engineer or process planner in successfully choosing the design of gear milling cutters to make cost-effective cylindrical gears to the appropriate quality desired.

THERE ARE A NUMBER OF DECISIONS THE MANUFACTURING engineer faces when planning the necessary processes for producing parallel axis, cylindrical spur or helical gears and splines. Hobbing, of course, is the most common metal-removal process to create the proper tooth space that is needed to produce these gears. This process has been around for more than 100 years and it has proven to be an effective method. It is a form-generating process by way of successive and incremental cuts that create the proper form via a number of straight profile rack-style cutting teeth. With the hobbing process, the gear rotates as the hob rotates and advances axially across its face.

But there is another way to accomplish this task. Gear milling cutters that mill an entire tooth space with the gear wheel stationary are effective as well. The questions many engineers ask are: When is gear milling preferred over hobbing? What type of machine is needed for gear milling? How do I select the right tool for gear milling? How is the cycle time estimated? What quality concerns should I have? What about workholding, tool mounting, and speeds and feeds?

Armed with the proper information, sound planning, and the right tools and equipment, the answers to these questions can be found. Stable, cost-effective gear milling or gashing processes can be applied when appropriate. This paper is intended for the manufacturing engineer who is perhaps new to gear manufacturing. It is not high-level scientific analysis. It is practical in nature and intended to provide valuable guidance for process development in gear fabrication.

BASICS OF THE HOBBING PROCESS

The manufacturing engineer must first understand the basics of hobbing in order to make a rational process decision of gashing versus hobbing. As already stated, hobbing is a generating process. The nature of hobbing involves complex cutting kinematics. The best way to understand this is to observe the generating patterns produced by hobbing (see Figure 1). Note the trochoidal looping pattern as indicated.

As shown, each tooth of the hob cuts in a different position in the tooth space and has different chip-forming characteristics. As the gear wheel gets smaller in diameter and has fewer teeth for a given size diametral pitch (DP) or module, the tooth space gets bigger. (See Figure 2.)

As the tooth space gets larger and the gear diameter decreases, fewer hob teeth are engaged in the cutting process as the chordal length of the hob decreases; therefore, the chip thickness increases for a given feed rate per gear revolution. Also, as the gap gets larger, each hob tooth now must take a larger bite. The area of the chip cross-section is larger.

This means that pinions with fewer teeth and bigger gaps than their mating gear wheel can be quite demanding. This is when gear milling cutters begin to become a better process alternative to hobbing.

The other factor with hobbing to consider is the root configuration and any necessary undercut. Undercuts are required when a successive finishing operation such as grinding is required. Root undercuts are accomplished with a feature on the tip of the hob tooth referred to as protuberance. This protuberance feature creates the root undercut with a sweeping trochoidal pattern of cutting action. (See Figure 3.)
The amount of undercut from hobbing low numbers of teeth, or course pitch, can be controlled using short lead hob designs. Special “off lead” hob designs are considered custom tools, perhaps with a longer lead time and higher cost than conventional hobs. This has been a successful method of controlling excessive undercut for many years, but in-depth analysis of these hobs is beyond the scope of this paper.

**BASICS OF THE GEAR MILLING OR GASHING PROCESS**

With gashing, the gear wheel is held steady while a milling cutter advances axially along the face width of the gear blank. This process can be used to rough or semi-finish the tooth space with extra stock allowance for successive hobbing, grinding, shaving, or honing operations. Or, it can complete the tooth space to final form. It can be done in one or multiple passes. Gear milling cutters can be mounted in tandem to produce two or more spaces, and they can be constructed of steel bodies with removable carbide inserts. In some rare cases, they are made to accept re-grindable carbide blades, but with modern precision grinding techniques for carbide inserts, there is little (if any) justification for such tools. Solid carbide tools are sometimes used, but due to the high cost and size limitations of solid carbide, they too are becoming less popular. High-speed steel cutters can be made at a relatively low cost, but they offer limited speed capability and are best for very low-volume gears and smaller module sizes, as the high cost of high-speed steel material makes larger size cutters impractical. (Examples of various gear milling cutter designs are shown in Figures 5–8.)

**Figure 2: Tooth space variation based on tooth count**

**Figure 3: Root undercut created by protuberance**

**Figure 4: Example of gear milled profile with flank grind stock, but finished root**

Gear milling cutters with tangentially mounted inserts are quite common. They offer good economy, as most have carbide inserts with multiple edges. Most are designed for module sizes over 8 (DP 3). Horsepower and torque requirements are higher, as their cutting geometries traditionally involve negative axial and radial rake angles. Some newer designs have inserts with positive cutting geometry, but this is generally limited to roughing and semi-finishing tools that do not require...
gets thinner (see Figure 3). The thicker root compared to hobbing offers a design strength advantage for gashing versus hobbing.

External spline milling tool and inserts are shown in Figures 9 and 10. In the past, whether involute or straight-side splines, hobbing was almost the exclusive method of cutting. With advancements in CNC machines with precise 4th axis indexing, gashing is now emerging as the process of choice.

Grinding relief (finish milled) in the root area to prevent a step in the active profile (see Figure 4) may be done with gear milling cutters as well. As the hobbing process produces a more pronounced undercut as the number of gear teeth goes down and the pitch circle diameter decreases, the cross-sectional area of the root

SELECTING GASHING OVER HOBBING
If single indexing machines, whether dedicated hobbing/gashing machines or CNC equipment with precise indexing capabilities are available, gashing/gear milling is possible. The choice of hobbing versus gashing must take into account these factors:

- **Number of teeth on the gear wheel.** With pinions and gears with a small number of teeth (given their module or DP size), the tooth space widens as the arc of the pitch circle decreases. The chordal length of engagement of a hob tool will be less than it would be on a larger diameter gear with more teeth and a larger pitch circle arc. This means that fewer hob teeth are engaged in the work to generate the necessary gap. This limits the feed of the hob, as the hob teeth become overloaded quickly. Gashing can complete the form in one or more passes, depending on machine and setup rigidity, horsepower, and torque. Time for indexing the cutter from one tooth space to the next is reduced along with the tooth count. This works in the favor of gashing cycle time.

- **Root fillet requirements.** Root fillets are generated with the hobbing process and are somewhat limited due to the trochio- dal loop previously discussed. If grinding, shaving, or honing of the flanks is required in subsequent operation, then proper allowance in the root area is required. The degree of flexibility is important. Hobbing can create a true undercut, as shown in Figure 3. Gashing will produce a straight relief in the root. This is preferred in many cases due to increasing the cross-sectional thickness of the tooth at the root area below the start of active profile. With pinions having few teeth, the trochioiodal loop may generate a very large undercut with hobbing. Gashing is preferred for strength in such cases.

- **Having conventional CNC equipment only.** If there is no hobbing machine available, but a CNC machine tool with indexing capability is available, gashing is often a good option. The consideration here is
that the machine has a very good quality indexing (4th or 5th axis) table that can achieve the tooth spacing error tolerances per requirement. Modern multitask mill/turn machines are gaining popularity and offer tremendous potential. Power at the spindle and rigidity must be considered as well. High-quality tool-to-spindle interfaces must be considered. No. 50 ISO taper. Coromant Capto, or HSK are good spindle-to-tool interface choices to increase accuracy and stiffness. CNC machines will not have outboard arbor support like a traditional gear cutting machine; therefore, one may need to take multiple passes to reduce forces. On the other hand, spline milling is usually much less demanding, as the whole depths are often half of the same gear size. With spline milling, the cycle time is often 50 percent less than traditional horizontal HSS hobbing, especially if the CNC machine selected has a high rapid traverse rate. This can be less than one second per tooth.

**COMBINING GASHING WITH HOBBING**

Gashing and hobbing tools can be combined on the same machine. Many modern hobbing/gashing machines have the control functionality to allow a gashing tool mounted on the same arbor as a hob to single index and rough or semi-finish the tooth spaces. The hobbing head can then shift over — without losing track of tooth position on the workpiece and location of the hob cutter — to finish hobbing the gear. This method is useful by using a carbide indexable gasher to remove most of the stock when work material is hard and difficult to machine with a traditional HSS hob. The hob tool only finishes the gear by removing a minimal amount of stock. The length of the hob can be shortened, and the time between regrinding can be increased. (See Figure 11.)

**DETERMINING CHIP THICKNESS WHEN GASHING**

The most important factor in proper application of gear milling cutters is the determination of chip thickness. Because the arc of engagement of a gear milling cutter is generally very short, the portion of the cutter engaged in the work is very small, relative to overall circumference (see Figure 12). The feed velocity in milling is often stated in terms of feed per tooth. Due to this small arc of engagement, the actual chip thickness will be significantly less than the feed per tooth.

With this being said, actual chip thickness must be calculated. Actual chip thickness is referred to as $H_{ax}$. Calculation of an $f_z$ modification factor must be determined. This factor is stated as a multiplier of actual feed per tooth, referred to as $f_z$. The first step is to determine the diameter of the cutter, or $D_c$. Then, determine the actual whole depth of the tooth space, or depth you are milling to (if multiple passes are required), which is referred to as $A_e$. Of course, the proper $H_{ax}$ value must be determined as recommended by the tool manufacturer.

The formula for $f_z$, modification factor, is:

$$f_z = \frac{0.5 \left( \frac{D_c}{A_e} \right)}{\frac{D_c}{\sqrt{A_e}} - 1}$$

Equation 1

An example would be an 8-inch-diameter cutter, $D_c = 8$.

One pass milling to a depth of 0.7-inch tooth space, $A_e = 0.7$.

Thus, $f_z$ modification factor = $5.7143 / 3.229$ = 1.769.

Therefore, if $H_{ax}$ of 0.008 is needed, then $f_z$ of 0.008 x 1.769 = 0.014.

**CHIP THINNING IN THE FLANK VERSUS ROOT**

The previous section on chip thickness refers to the root area. The root area is where maximum chip thickness is calculated. The chips generated in the flank area of the tooth form are much thinner. A simple way to look at this is to consider a basic rack V-form. In the case of a 20-degree pressure angle gear, you would simply take the tangent of a 20-degree angle, which is 0.364. Using this factor, a chip thickness calculated at the root of 0.008 would be multiplied by 0.364. This would indicate a flank chip thickness to be around 0.003 inch.

Because the flanks are not a straight V-shape (unless milling a straight rack) with the typical involute curve design, it is impractical to calculate the exact chip thickness at the flank. This method of using the tangent of the pressure angle is acceptable for process planning purposes. This factor of thinner chips at the flank is also cause for lighter cutting edge preparation on the flank inserts. Too much edge hone could cause rubbing and smearing of the metal, with too much pressure and inadequate shearing action.

For this reason, when tools are made with separate root and flank inserts, the designers often increase the number of root inserts relative to flank inserts. Root-to-flank ratios of 2:1 and 3:1 are common. The root inserts produce roughly three times the work of the flank inserts, so this concept makes sense for balancing tool wear of all inserts involved.

**FULL-FORM GASHING**

Referring back to Figure 6, full-form gashing tools have merit in many cases. With full-form gashing, the carbide inserts are ground to the total form of the tooth space (root and flanks) being produced. This is the most accurate option due to the all-in-one design. In the case of a separate root, left and right flank inserts, each insert has a tolerance, plus the tool body insert pockets have tolerances; therefore, the stack-up is greater than the full-form insert where there is one insert and one pocket required.

Another advantage to the full-form design is that chip evacuation is straightforward and effective. Tangential insert
designs with a number of different root and flank inserts create various chip formations that curl in various directions. Predicting these patterns of chip formation and subsequent evacuation can be challenging to even the most experienced tool designers. Sometimes chip smearing and re-cutting is encountered. Full-form gear milling cutters have up to two times more effective teeth than tangential solutions; therefore, they are more productive.

A minor drawback to the full-form design is that the root-to-flank ratio is 1:1, which, as mentioned in the previous section, is not the ideal balance for tool wear. Productivity, quality, and chip evacuation improvements usually outweigh these concerns. As the gear module or DP size increases, the size of the required carbide blank size gets larger, and it eventually becomes impractical to produce, both technically and economically.

If AGMA, DIN, or other gear quality specifications are a concern with the tool designer or manufacturing engineer, the full-form option should be considered.

**CLIMB MILLING VERSUS CONVENTIONAL MILLING**

Common milling best practice is to employ a climb milling technique for gear gashing. The other process, commonly referred to as conventional milling, is sometimes employed. A visualization of the two methods is shown in Figure 13.

Climb milling allows for the cutting insert to enter the work with some immediate chip forming action and depart the cutting zone with no chip thickness. As carbide performs best under compressive load, this method has been proven to be superior in tool life. The unloading on exit is less detrimental to the tool, as the exit shock is minimal.

With conventional milling, the tool enters the cut with no chip thickness and progressively forms the chip as it advances into the work. It exits the cut with some degree of chip thickness, so the unloading is sudden and detrimental to tool life. Also, the rubbing effect as the tool enters the work causes more heat generation and thermal effect. There’s also a significant increase in tool pressure with this method. One measurable benefit is that the surface finish is typically superior in conventional milling due to the compressive, burnishing action as the tool begins forming the chip. Lab results at Sandvik Coromant have proven a 20-30 percent reduction in Ra when conventional milling is used versus climb cutting.

**MILLING WITH CUTTING FLUIDS VERSUS DRY CUTTING**

Best milling practice with carbide tooling is, with few exceptions, to cut dry. Cutting with fluids will thermally shock the carbide tool as it exits the cut. The effect of thermal shock is typically detrimental to tool life. An example of this failure mechanism is shown in Figure 14. Here, repeated heating and cooling cycles eventually lead to perpendicular edge cracks that will eventually allow insert material to release, leading to rapid breakdown. Water-soluble cutting fluids have the most detrimental effect on thermal shock as the water rapidly cools the tool. Cutting oils also have significant cooling effects, but they do not remove the BTUs as rapidly as water. Therefore, they are somewhat better than water for tool life. Of course, there are environmental costs with both oil- and water-based fluids, and the goal should be to eliminate them whenever feasible.

Application of compressed shop air, or even vortex cooling of compressed air, can assist in chip evacuation and cooling of the tool and work. An example of this method of cooling is shown in Figure 15. This is a proven method of providing cooling. Eliminating heat buildup and removal of chips is the main reason cutting fluids are still employed.

Tools can be designed with internal air passages as shown in Figure 16. Such designs assist in chip evacuation and cooling of the
tool. Such features, however, considerably increase the complexity and cost of the tool. This method of internal air should be considered carefully, as the necessary machine spindle and arbor modifications, plus the tool cost and complexity issues of through-the-tool air flow, are significant. With proper tool design for free chip flow combined with proper cutting strategy, internal air holes likely can be avoided.

CUTTING FORCES
Power and torque requirements must be determined to effectively apply gear milling cutters for a stable machining process. To estimate machining forces properly, one must determine the cross-sectional chip area and specific cutting force for a given material. To calculate the exact cross-sectional area for a given tooth space is somewhat complex and is best done with the aid of computer software capable of creating the exact flank profile and root configuration. Such calculations are beyond the scope of this paper. Instead, the intention is to find a close, usable approximation. To do that, one can examine the basic rack configuration (V-shape) of a given pressure angle without the complexity of an involute curve profile on the flanks. Note: When using this method of estimation for milling gears or pinions with low tooth count, as previously discussed, the gap will open up as the pitch circle gets smaller. In such cases, force calculation results might need to be padded to the high side.

The first step in the calculation of the chip cross-section is to define the basic rack. (See Figure 17.)

NOMENCLATURE:

\[
\begin{align*}
H &= \text{Whole depth of tooth space (mm)} \\
\alpha &= \text{Pressure angle (degrees)} \\
V &= \text{Flank offset from root (mm)} \\
B &= \text{Chip thickness at flank (mm)} \\
D &= \text{Root width (mm)} \\
A &= \text{Tooth space area (mm}^2) \\
f_z &= \text{Feed per tooth (mm)} \\
h_{\text{ex}} &= \text{Maximum chip thickness (mm)} \\
h_{\text{m}} &= \text{Average chip thickness (mm)} \\
v_c &= \text{Cutting velocity (m/min.)}
\end{align*}
\]
The following example shows how to calculate power and cutting forces. The first step is to determine cutter rpm \( (n) \) based on a recommended cutting speed \( (v_c) \).

The formula is:

\[
 n = \frac{v_c (1000)}{D_c \pi}
\]

Equation 2

For this example:

\[
 n = \frac{180(1000)}{350 \pi}
\]

Next, the \( \nu_f \) (mm/min. feed) must be determined. The formula is:

\[
 \nu_f = n(f_z)(z_c)
\]

Equation 3

In this example, we assume that:

\[
 \nu_f = 164(0.40)(8) = 524 \text{mm/min}
\]

Next, \( k_c \) must be determined. The formula is:

\[
 k_c = \frac{k_{c1}}{m_c}
\]

Equation 4

In this example, we use:

\[
 k_{c1} = 1900 \text{N/mm}^2 \text{ and } m_c = 0.25.
\]
This factor is available in the materials sections of Sandvik Coromant catalogs and technical literature in print and online. (See Figure 18.)

Average chip thickness, or $h_m$, is determined by the formula:

$$h_m = \frac{2a_d f_z}{D_c \arccos \left(1 - \frac{2a_d}{D_c}\right)}$$  

Equation 5

In this example, we use a whole depth ($H$) of 36 mm. We assume a one-pass operation, so $a_e = H$ and $f_z$ (feed per tooth in mm) of 0.04 mm:

$$h_m = \frac{(2)(36)(0.40)}{350 \arccos \left(1 - \frac{2(72)}{350}\right)}$$

Reduced to:

$$h_m = \frac{28.8}{350 \arccos 0.7943}$$

This can be further reduced to:

$$h_m = \frac{28.8}{350(0.6529)} = 0.13$$

So for specific cutting force, we have $k_c = 1900 / (0.130.25) = 3188.93$

Next, tooth space area must be determined. Referring back to Figure 17, we must first determine $C$. The formula for this is:

$$C = \frac{M_n (2 + 0.2)}{\cos \alpha}$$  

Equation 6

This example assumes module size, $M_n = 16$ and $\alpha = 20$:

$$C = \frac{16(2 + 0.2)}{\cos 20} = 37.459$$

Next, from Figure 17, $D$ must be determined. The formula is:

$$D = M_n \left[\frac{\pi}{2} - 2 \tan \alpha\right]$$  

Equation 7

In this example:

$$D = 16 \left[\frac{\pi}{2} - 2 \tan 20\right] = 13.486$$

Again from Figure 17, $V$ must be determined. The formula is:

$$V = \sqrt{c^2 - H^2}$$  

Equation 8
In this example:

\[ V = \sqrt{37.459^2 - 36^2} = 10.353 \]

Next, from Figure 17, \( A \) must be determined, so the formula is:

\[ A = V \times H + D \times H \]  
Equation 9

In this example, it is:

\[ A = 10.353 \times 36 + 13.486 \times 36 = 858.204 \]

With these steps completed, we can calculate power at the spindle, or \( P_c \). The formula is:

\[ P_c = (k_c \times A \times v_f) + (60 \times 10^4) \]  
Equation 10

In this example:

\[ P_c = \frac{(3188.93)(858.204)(524)}{60000} = 23.9\text{Kw (multiply by 1.341 for HP)} \]

The next important calculation is torque, or \( M_c \). The formula for this calculation is:

\[ M_c = \frac{P_c (9550)}{n} \]  
Equation 11

In this example:

\[ M_c = \frac{23.9(9550)}{164} = 1392\text{Nm (divide Nm by 1.3558 to get ft. lbs.)} \]

In summary, a module 16 mm, 350 mm diameter cutter with eight effective teeth, running at 180 m/min. cutting speed, feeding at a 524 mm/min. feed rate, cutting normal low-alloy steel at 300 BHN hardness will consume approximately 24 Kw at the spindle with a torque requirement of 1392 Nm.

A simple computer spreadsheet program can be written to automate these calculations in order to save time and prevent calculation errors. This will greatly assist the gear manufacturing engineer when planning a process. The same spreadsheet could be integrated with cycle time calculations as well.

CONCLUSION

Gear milling or gashing operations in gear machining is a well-proven, efficient, and stable method of fabrication for gear production. The possibilities for improving productivity, reducing cost, and creating quality gears are evident. Assistance from qualified tooling vendors will help guide the engineer in the proper direction. New tooling concepts, advancing machine tool concepts, heat treatment methods, and gear materials constantly change the production environment. And a new generation of gear professionals entering the workplace bring fresh ideas and a willingness to embrace new methods from the young engineers who are steering the technology shift.

Along with these factors, multitask CNC machines are being deployed for gear-making in ever-increasing numbers. The future of gear manufacturing is bright, and the coming years are set to be an exciting time for those in the industry.

REFERENCES


ABOUT THE AUTHOR: Brent K. Marsh is the manager of business development of Gear Milling Solutions at Sandvik Coromant, Americas. He has 21 years of cutting tool industry experience in sales, application engineering, grade development, key account management, project management, and business development. Prior to that, he has had over 10 years in manufacturing engineering (foundry and diesel engine production). Marsh received his Bachelor of Science in Business Administration and Economics from Missouri Valley College and his Master of Science in Industrial Technology with concentration in automated manufacturing from University of Central Missouri.