

# FEW ADVANCES IN MILLING CUTTERS

Satyajit Chatterjee<sup>1</sup> and Dr. Santanu Das<sup>2</sup>

<sup>1</sup>. 2<sup>nd</sup> year, M. Tech. (Prod. E.),

<sup>2</sup>. Head of the Dept, Dept. of Mechanical Engineering,  
Kalyani Govt. Engineering College, Kalyani – 741235

E-mail : sdas\_me@rediffmail.com

## Introduction:

Machining or metal removal processes refer to the removal of certain selected areas from a part to obtain desired shape, finish and integrity. Chips are formed by interaction of a cutting tool (wedge shaped) with the material being machined. The cutting tool is mounted on a machine tool that provides the required movements of the tool relative to the work to accomplish the process desired.

There are seven basic machining processes after reference [1],

1. Shaping,
2. Drilling,
3. Turning,
4. Milling,
5. Sawing,
6. Broaching,
7. Abrasive machining.

Historically, eight basic types of machine tools were developed to accomplish the basic processes. These are,

1. Shapers (planers),
2. Drill presses,
3. Lathes,
4. Boring machines,
5. Milling machines,
6. Saws,
7. Broaches,
8. Grinders.

Many of these machine tools are capable of performing more than one of the basic machining processes. This obvious advantage has led to the development of *Machining Centers*, specifically designed to combine many of the basic processes.

Milling is a basic and one of the most widely applied machining processes by which a flat surface is generated by progressive chip removal. It is estimated that in an average shop, milling constitutes 28% of the total number of operations and 30% of the total machining time [2].

In a milling operation, the work piece is fed into a rotating cutting tool. Sometimes the work piece remains stationary and the cutter is fed into the work. Unlike turning, milling employs a multipoint cutting tool in nearly all cases. The rotary cutting motion is imparted to the work piece.

With a multipoint cutter, milling leads to high material removal. Often the desired surface is obtained in a single pass of the cutter work and because very good surface finish can be obtained, milling is particularly well suited and widely used for mass production work. Unquestionably, more flat surfaces are produced by milling than by any other machining processes. Here, equally spaced peripheral teeth will intermittently engage and machine the work piece causing what is called *Interrupted Cutting*.

## High Definition Tool Materials :

The latest developments in machining and metal cutting have provided us with new and improved tool materials like Poly-crystalline Diamond (PCD), cubic Boron Nitride (cBN), Cermets etc. the costs of these new tool materials are high as compared to the conventional tool materials like HSS. To compensate for their high cost, their utilization factor should be improved with a higher level of productivity through higher machining parameters. And the selection of higher machining parameters is always combined with effective dynamic balancing which in turn require finer accuracy level on the tool.

## **Milling Cutters with High Definition Tool Materials :**

Generally, the word *milling* indicates towards a process with low cutting speeds, wider tolerances, low surface finish, poor tool life and so on. But today with the introduction of poly-crystalline diamond (PCD) inserts for the non-ferrous milling applications, most of these limitations are being eliminated.

The poly-crystalline diamond tools are used with cutting speeds of 300-2500 m/min; low coefficient of friction tends towards low built up edge (BUE) formation and high resistance to abrasive wear and it can be ground to very sharp and smooth edge. These result in longer tool life, lower cutting forces and less heat generation. But effective machining at this high speed requires high-speed spindles and high precision bearings. Also the cutters fitted with PCD inserts must be dynamically balanced to secure it against the centrifugal force. And the PCD tools are to be re-grindable to have optimum utilization to compensate for their high cost.

The face true running accuracy of a milling cutter requires finer adjustments of the inserts. In the existing design of the milling cutters with PCD inserts, with all adjustments, could guarantee a face true running accuracy of 30 microns that lead to vibration at the cutting edges, which have detrimental effect on the cutter life as well. To have better accuracy, the vibration level should be reduced to a bare minimum. This will not only give a smooth cut but will also result in optimum tool life.

Researches [3] in this field were fruitful, as a new design of clamping method provided the way of finest adjustments of the PCD inserts on the cutter body. The concept of differential pitch mechanism was employed here in the newly designed clamping screw. The screw along with an angled insert support shim could adjust the

insert on the cutter body within a very small value (a fraction of a micron). This new design of cutter could be set approximately 3 microns of face true running accuracy.

Such high precision milling cutters will surely enhance the quality level of the components machined. Besides, higher cutting speed and higher tool life result in higher productivity and better machine utilization.

## **High Performance Milling Cutters:**

Cutting operation conducted at high speed and feed results in high production costs. However, high production techniques employing high cutting speeds (500-700 m/min for machining steel and up to 2500 m/min for softer materials) considerably increase the amount of heat generated in the cutting process and reduce the time available for its dissipation. Therefore, the tool point is heated to a very high temperature by the chip; the temperature of the cutting edge may reach 800-900°C or higher. Since, high-speed HSS tools lose their form stability at a much lower temperature. They are, obviously, unsuitable for high-speed machining. On the other hand, cemented carbide tipped tools can operate satisfactorily at temperatures of 800-1100°C. Therefore, carbide tipped tools are usually employed on high production cutting tools. Ceramic tipped tools, capable of withstanding temperatures of 1100-1200°C, are also high production cutting tools.

Carbide inserts with proper coating can also improve performance in trying conditions. During the high-speed dry milling of Inconel 718 by a ball nose end milling cutter, TiAlN coating gave a better performance and tool life as it has the higher oxidation resistance, higher room temperature hardness, lower coefficient of friction and high adhesion strength [4].

*The major problems encountered in a high*

*production machining are caused by insufficient tool life, chipping of the cutting edges, poor quality of surface finish and vibration or chatter. The causes are very much interrelated.*

### **Features of High-Production Milling Cutters:**

Milling cutters for high production rates have specially designed features which enable them to perform at high speed and feed. Some of the more important design considerations for high production cutting tools are as follows.

#### **1. Use of negative rake for edge**

**strengthening :** As cemented carbide (the primary tool material for high production tools) is brittle, it is advantageous to reduce the rake angle or use a negative rake to strengthen the tip and reduce chipping. Also, larger wedge angle in the negative rake tool improves heat conduction from the cutting edge to the tool shank. This betterment of heat dissipation improves tool life.

#### **2. Stability of the system should be high :**

The traditional theory of regenerative chatter is not adequately suitable to describe the stability of highly interrupted/intermittent type machining like milling where the ratio of time spent for cutting to non-machining time is low. Research works are going on in this field where to develop a new model for the stability of interrupted machining like milling [5]. The concept is based upon modeling the system as a kicked harmonic oscillator with delay.

### **High Performance Milling Cutters in Mould Making:**

Traditional mould making can be a slow, multi-step process, especially if the mould maker uses

the “drill and whittle” method. This operation requires the mould maker to drill an access hole in the workpiece and then, after changing tools, slowly whittle away material with an endmill. Productivity drops with each tool change, and it can be impaired further if excess tool overhang, cutter vibration, or insecure fixturing forces the mould maker to reduce cutting speed.

Some manufacturers producing medium-to-large moulds are tackling these problems by using a high performance line of milling cutters. These cutters eliminate the need to drill an access hole by offering a variety of workpiece-entry methods. Where space permits it, the tool can machine holes in solid material using helical interpolation. In other situations, the cutter can ramp into the work at an angle or plunge into it in the z-axis.

Once one of these preliminary steps is completed, the same tool can cut into the work along the x- and y-axes the way a standard facemill can. A mould maker with a single ramp/plunge/facemill (Kennametal Inc. trademark RPF™) cutter [6] and CNC programming can perform operations that formerly might have taken a number of tools and setups and significantly longer machining time.

When ramping and plunging, the inserts cut with their inside and outside edges at the same time. This simultaneous cutting action can reduce machining time by as much as 80% in cavity production operations compared to other types of cutters. Productivity is enhanced by the RPF cutter's design, which incorporates high-shear insert geometry to promote free cutting action and improve the surface finish, as well as by large chip gullets and screw-on inserts to facilitate chip flow. Because the high-positive geometry of the tools expels chips well (Fig 1), their performance is nearly equivalent on either horizontal or vertical milling machines.

Currently, RPF cutters are available in

diameters ranging from 1" to 6" as stocked standards, and up to 8" diameters as specials. Inserts specially designed for the cutters are available in round and elongated round styles with various nose radii.

### **Working with RPF Cutters :**

To benefit from the use of RPF tools, the mould maker must program the CNC to take advantage of the tool's unique cutting action. RPF cutters should be programmed to cut in a climb mode. This is especially important when machining work-hardening materials. Climb milling is not mandatory when using a RPF cutter on other materials, but it is still recommended.

The next consideration will be to select the method used to enter the workpiece and create the cavity. RPF cutters can enter workpiece with helical interpolation, ramping, or plunging (Fig 2).

### **Helical Interpolation :**

Helical interpolation is the most efficient of the three methods when using a 3-axis machine. To helically interpolate a hole, the cutter moves in a circle that is slightly less than twice the cutter's diameter. As it moves around the cut, the cutter also moves down along the z-axis, so that it is constantly cutting deeper into the workpiece. With this method, the tool enters and exits the workpiece only once, maintaining a more uniform depth-of-cut (DOC) and chip load without dwelling on the z-axis.

Compared to other entry methods, helical interpolation offers longer tool life, greater chip control on entry, increased metal-removal rates, and reduced cycle times. In addition, a mould maker using helical interpolation can best utilize available horsepower during entry by adjusting the helical ramp angle (the depth of the tool's z-axis penetration per revolution).

With helical interpolation, the depth of the entry operation is independent of the DOC and used during subsequent facemilling operations along the x- and y-axes. The mould maker uses helical interpolation to mill down to the full depth of the cavity first and then retracts the cutter to begin facemilling the remainder of the cavity. Multiple x/ y passes are needed to reach the cavity's full depth.

Performing the entry operation to the full depth of the cavity first and then facemilling allows the mould maker to use as much of the machine's horsepower as possible during the facemilling stage because he can program the machine to facemill with greater DOCs than he could if each facemilling pass had to begin with a ramping or z-axis plunge operation. For example, a 2" RPF cutter with elongated-round inserts can plunge in the z-axis to a depth of only 0.250" before the interior structure of the cutter bottoms out against the workpiece. The same cutter and insert combination can facemill the x- and y-axes to a depth of 0.437". A mould maker with a moderate-to-high horsepower machine could facemill to this maximum DOC if the entry hole already has been machined using helical interpolation. But if the cutter plunge into the workpiece first and then facemill the x- and y-axes, the plunging operation will limit the facemilling DOC to 0.250".

Ideally, cutting into a workpiece using helical interpolation should leave a flat bottom with no post or cone in the center of the machined surface that must be removed with another pass. This is accomplished by limiting the diameter of the machined hole to a specific range.

The maximum diameter hole that can be cut to produce a flat bottom can be calculated using the formula  $(2D - IC)$ , where IC equals the width of the insert. For example, using a 3" diameter cutter and a (0.625"-IC) insert, the largest diameter that could be cut would be a 5.375

(2 x 3" diameter)-0.625" insert IC. A mould maker could cut a larger-diameter hole, if necessary, but a post will remain in the middle of the cut that is the same diameter as the amount the hole exceeds the calculated maximum diameter.

The minimum diameter is calculated with the formula  $(2D - 2IC)$ . This is an absolute minimum. If a mould maker attempts to helically interpolate a bore that is smaller than this, the portion of the workpiece at the center of the hole that remains uncut will crash into the cutter.

### **Ramping :**

If a mould cavity is not large enough to accommodate helical interpolation, then ramping or plunging will be necessary. To ramp a cutter into the workpiece, the mould maker gradually increases the DOC as the tool moves deeper (Fig 2). The rate of increase is governed by the maximum angle each RPF cutter diameter is capable of. Ramping is preferable to plunging, because chatter and workhardening are less likely to be problems during this operation, and less machine horsepower is required.

But ramping can be used only if there is sufficient room in either the x- or y-axis. In some cases, there may be enough room to ramp into the workpiece, but not enough to ramp to the full depth the cutter is capable of. If the length or width of the cavity does not permit ramping to the full depth, it also will limit the DOC that can be used during the x- and y-axis facemilling that is performed with each pass. Milling to a specified depth, in this case, will require a greater number of passes (Fig 3).

### **Z-Axis Plunge :**

Entering the workpiece with a z-axis plunge is the least attractive alternative, but it may be necessary if space in the cavity is too small to

allow helical interpolation or ramping. In a z-axis plunge, the tool is fed straight into the workpiece to the maximum depth allowed by the cutter's design and then moves it along the x- and y-axes. To mill to the cavity's full depth, repeated plunges and x/y facemilling operations are required.

When using the plunge method to enter difficult-to-machine materials, a 0.5-second dwell must be programmed every 0.020" into the cut to break the continuous chip that forms during the plunge. These dwells will increase cycle time.

### **Facemilling :**

To facemill in the x- and y-axes successfully, mould makers must follow certain guidelines, regardless of the entry method employed. Cutters work most efficiently if an equal width of cut is maintained throughout the cavity. This can be achieved by offsetting the point of entry into the cavity. The ideal width is 60% to 80% of the cutter's diameter, moving to the lower end of the range to cut workhardening materials or to facemill with a low-horsepower machine. Mould maker can extend tool life by avoiding widths of cut equal to the cutter's diameter on workhardening or abrasive materials.

Chatter may be a problem as those points where the RPF cutter makes a turn. To eliminate this problem, the mould maker should program a corner radius larger than the cutter radius. This will minimize radial engagement between the cutter and the workpiece. This programming change also may eliminate the momentary dwell that can occur at the corners with older CNC machines.

### **Machine & Fixturing :**

As engineers, programmers, and users seek

the most productive combination of fixturing, machine, coolant system, and cutter, they must remember that RPF tools perform differently from conventional round-insert facemills. RPF cutters remove large amounts of material, and this generates increased cutting forces. As a result, fixtures that may be rigid enough for conventional milling cutters may not be rigid enough for use with RPF cutters. Insufficiently rigid workholding may result in workpiece deflection or chatter when using conventional machining techniques.

The horsepower requirements will increase as the diameter of the tool becomes larger, and may limit the size of cutter that can be used with a specific machine. Although a cavity may be large enough to accept a 6" cutter, the machine may lack the power to run the cutter productively. The DOC and feed rate would have to be scaled back to such an extent that tool life and metal removal efficiency would be sacrificed. Table 1 offers some general guidelines for matching the cutter diameter to the machine's available horsepower.

**Table 1: General guidelines for selecting cutter diameter**

Horsepower recommended	Cutter diameter
8-10	1.25" to 1.50"
10-15	2.00"
15-20	3.00"
20-30	4.00"
30+	6.00"

When possible, coolant should be used to flush chips from the workpiece cavity. If a facemill does not feature through-coolant capability, drill a 0.125" diameter hole axially through the shell mill retention screw to allow coolant to flush the chips from the bottom of the cavity outward. This modification is particularly helpful in applications that involve milling a deep, small-diameter hole.

Before using an RPF cutter, the user first should check the shop's inventory for a shell-mill adapter that is the proper size for the machine. Both the pilot diameter and the key size must be considered. The mould maker also may need to take into account the depth of the cavity. If it is too deep, the shell-mill adapter may not be long enough to reach the bottom. In this case, the manufacturer may be able to supply an adapter of the proper length. To avoid tool deflection, better not to use an adapter that is substantially longer than necessary.

Because of the RPF cutter's higher metal-removal rates and the reduction in machining steps, there is growing interest in the use of RPF cutters among mould makers and machinists in the aerospace industry. With appropriate modifications to programming, machining conditions, and fixturing, RPF cutters can meet user demands for high productivity and cost effectiveness in mould milling.

### Conclusion :

In this article, first, the general issues of milling cutters are discussed. Next, few typical milling cutters developed recently are reviewed and their principles of operation are presented.

### References :

- [1] T. A. Sadasivan, 'Cutting tools for production machining', e1, WIDIA (India) Ltd. Bangalore, 1997.
- [2] E. P. Degarmo, J. T. Black, R. A. Koshner, 'Materials & Processes in Manufacturing', e8, Prentice Hall of India Private Ltd., New Delhi, 1997.
- [3] Ramamurthy K. N., Karunagara J., (1997), "High precision milling cutter", Proceedings of 17th AIMTDR Conf., 1997, pp: 77-79.
- [4] Ng E. G., Lee D. W., Sharman A. R. C.,

Dewas R. C., Aspinwal D. K., 2000, "High speed ball-nose end milling of Inconel 718", Ann. of CIRP, 49/1, pp: 41-46.

[5] Davies M. A., Pratt J. R., Dutterer B.S., Burns T.J., 2000, "The stability of low radial

immersion milling", Ann of CIRP, 49/1, pp: 37-40.

[6] <http://www.manufacturingtalk.com>, Newsletter, 'High performance milling cutters'.

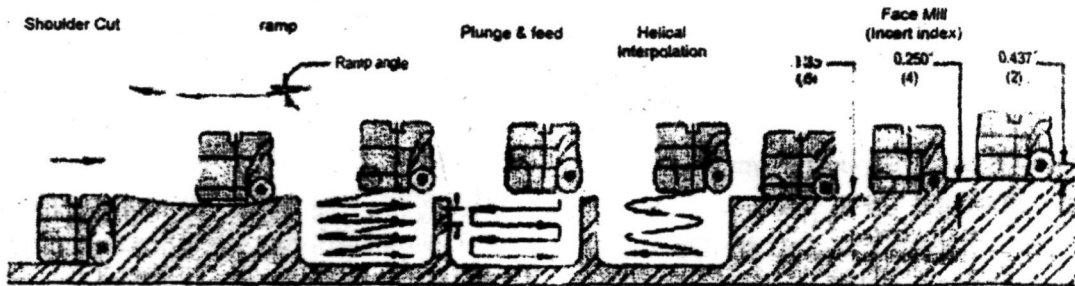


Fig 1: Milling using RPF Cutter (Kennametal Inc.)

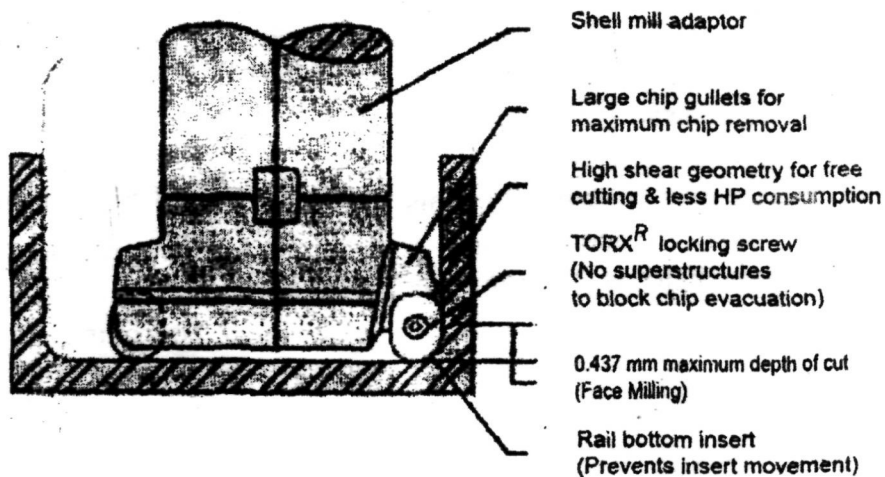


Fig 2: Cutter motion for milling with RPF Cutters

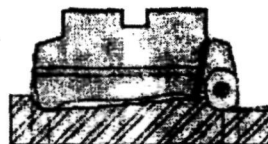


Fig 3: Ramping with a RPF Cutter