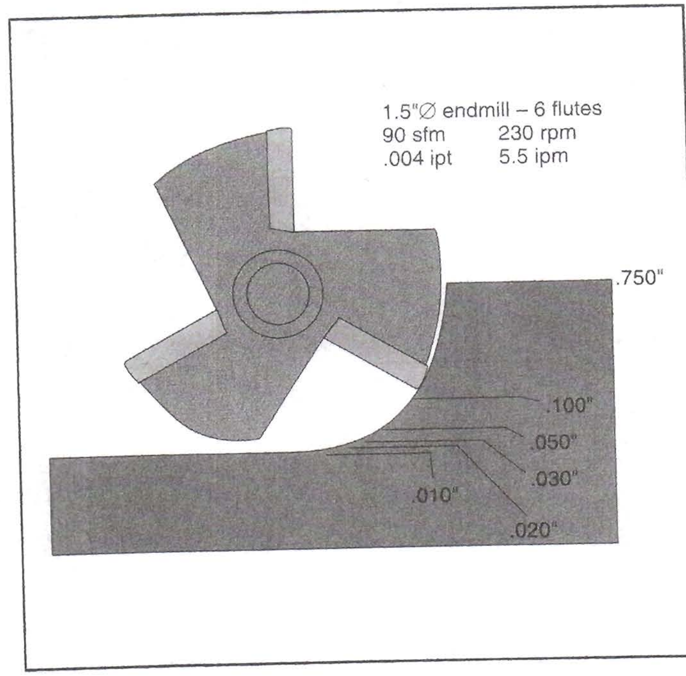
Feed Rate Compensation

Operations such as **periphery milling with a light radial depth of cut or slotting with an arbor mounted cutter** require a calculation for feed rate compensation to maintain a proper chip load on the insert edge at entry into the cut. The calculated chip load and actual chip load can be dramatically different, depending on the radial depth and the cutter diameter. For instance, the actual chip load on entry for a 3/4" diameter cutter taking a .010" radial depth cut is only 23 percent of the calculated chip load. It is not uncommon to encounter built-up edge, work hardening or chatter problems if the following formula is not applied. Minimal cutter run out is critical to obtaining an equal chip load on each flute of the cutter too. A side benefit to applying this formula is increased productivity as feed rates can increase dramatically.

radial depth of cut	actual chip load (ipt)	feed required (ipm) to maintain .004 ipt	increase
.750	.0040	5.5	0%
.100	.0200	11.5	109%
.050	.0014	15.3	178%
.030	.0011	19.6	256%
.020	.0009	23.9	335%
.010	.0006	33.8	515%

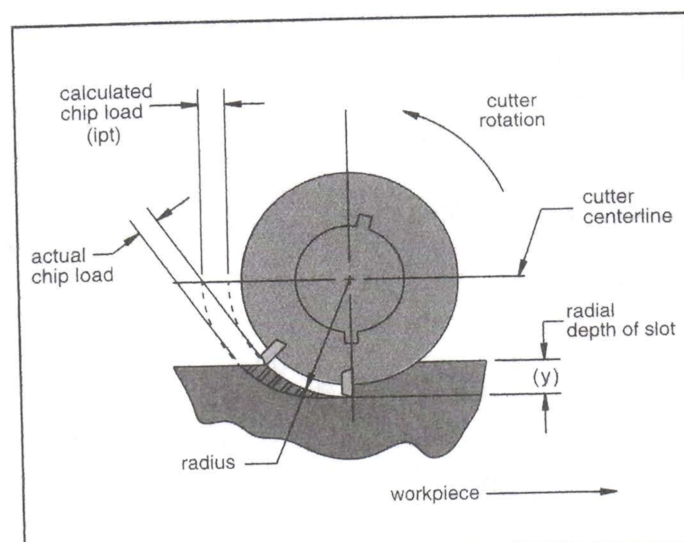


Slotting or Periphery Milling

True, or actual chip, load on the cutting edge of the insert is equal to the programmed chip load only when 50 percent or more of the diameter of the cutter is engaged in the cut (lead angle not considered). Anything less than half the diameter of the cutter means that the actual chip load is reduced by some percentage. The smaller the radial depth of cut, the greater the decrease in actual chip load.

It is very important to maintain a chip load which is great enough to ensure heat dissipation and prevent work hardening. A sufficient chip load will also create stability between the cutter and the workpiece.

The formulas shown below are used to determine the programmed chip load, or feedrate, necessary to obtain the desired load on the insert cutting edge as it enters the workpiece. These formulas should be applied whenever an arbor mounted slotting cutter is being used, or when less than half the diameter of a face mill or end mill is engaged in the cut. The lighter the radial depth of cut, the more important it becomes to apply the productivity formulas.



Productivity Formulas

$$\text{chip load (ipt)} = \frac{\left(\dfrac{\sqrt{(\text{dia.} - y) \times (y)}}{\text{radius}}\right) \times \left(\dfrac{\text{ipm}}{\text{rpm}}\right)}{\text{nt}}$$

or

$$\text{ipm} = \frac{\text{rpm} \times \text{nt} \times \text{ipt}}{\left(\dfrac{\sqrt{(\text{dia.} - y) \times (y)}}{\text{radius}}\right)}$$

Milling Applications

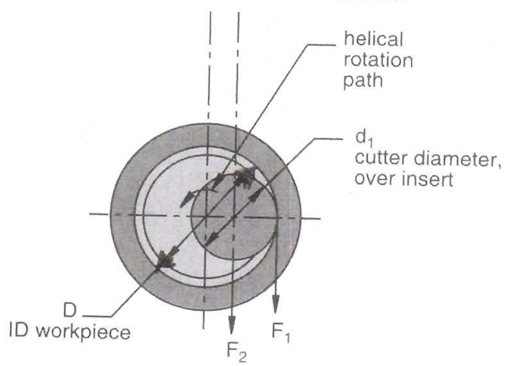
ID and OD Circular and Helical Interpolation

When we are doing inside or outside circular interpolation or helical interpolation and the radial depth of cut is small relative to the cutter diameter, the feed rate should be adjusted to maintain a proper chip load on the insert edge as it enters the work. The formula for this calculation and examples are shown below.

F_1 = tool feed rate at the cutting edge (in./min.)
F_2 = tool centerline feed rate (in./min.)
D = OD workpiece diameter
D = ID workpiece diameter
d_1 = cutter diameter, over insert

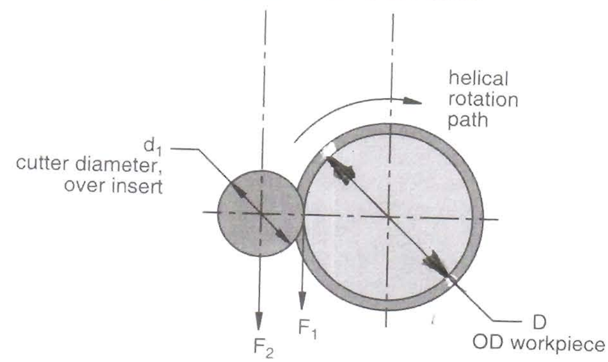
inside diameter (ID) helical interpolation

$$F_2 = \frac{F_1 \times (D - d_1)}{D}$$



outside diameter (OD) helical interpolation

$$F_2 = \frac{F_1 \times (d_1 + D)}{D}$$



On ID contour applications, you will find the tool centerline feed is always less than the cutting edge feed rate.

example for ID

D = 4.0" ID workpiece
d_1 = 3" cutter diameter
ipt = .008"
rpm = 637
nt = 7 effective inserts

1. Calculate feed rate at the cutting edge.

F_1 = ipt x nt x rpm

F_1 = .008 x 7 x 637 = 35.7 in./min.

2. Calculate feed rate at the tool centerline.

$$F_2 = \frac{F_1 \times (D - d_1)}{D}$$

$$F_2 = \frac{35.7 \; (4.0 - 3.0)}{4.0} = 8.9 \text{ in./min.}$$

To have (F_1) 35.7 in./min. at the cutting edge feed rate, we must program the machine tool for (F_2) 8.9 in./min. at the tool centerline feed rate. This is a difference of approximately 75 percent less feed than the cutting edge feed rate (F_1).

On OD contour applications, you will find the tool centerline feed rate is always larger than the cutting edge feed rate.

example for OD

D = 5.0" OD workpiece
d_1 = 2" cutter diameter
ipt = .008"
rpm = 955
nt = 5 effective teeth

1. Calculate feed rate at the cutting edge.

F_1 = ipt x nt x rpm

F_1 = .008 x 5 x 955 = 38.2 in./min.

2. Calculate feed rate at the tool centerline.

$$F_2 = \frac{F_1 \times (d_1 + D)}{D}$$

$$F_2 = \frac{38.2 \times (2 + 5)}{5} = 53.5 \text{ in./min.}$$

To have (F_1) 38.2 in./min. at the cutting edge feed rate, we must program the machine tool for (F_2) 53.5 in./min. at the tool centerline feed rate. This translates to an increase of about 40 percent more feed rate than the cutting edge feed rate (F_1).