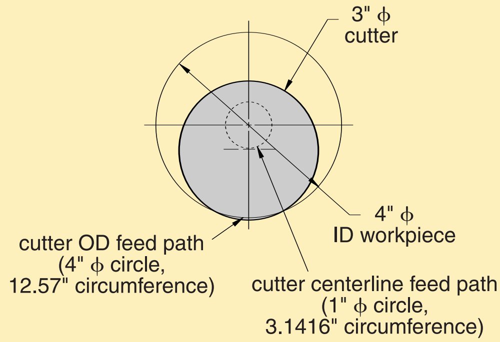
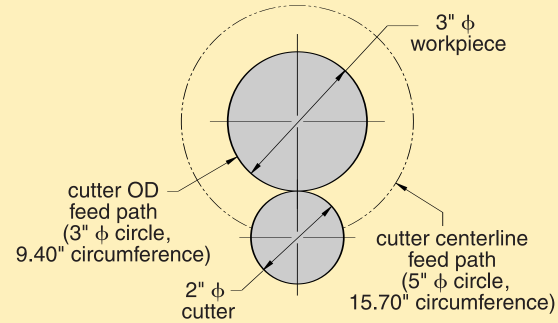
## ID and OD Circular and Helical Interpolation

**circular interpolation:** Consists of a cutter rotating about its own axis while traveling in an orbiting motion about an ID or OD workpiece circumference without any vertical shift during the operation. This orbiting movement utilizes the “X” and “Y” axis.



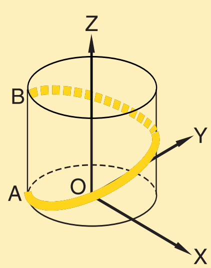

**ID circular interpolation**




**OD circular interpolation**

**helical interpolation:** This application requires a milling machine with three-axis control capability. The operation consists of a cutter rotating about its own axis together in an orbiting motion about an ID or OD workpiece circumference in the “X” and “Y” plane. The circular movement about the “X” and “Y” plane, with a simultaneous linear movement in the Z-axis plane (which is perpendicular to the “X” and “Y” plane), creates the helical movement. For example, the path from point A to point B on the envelope of the cylinder combines a circular movement in the “X” and “Y” plane with a linear movement in the “Z” direction. On most CNC systems, this function can be executed in two different ways:

- GO2: helical interpolation in a clockwise direction.
- GO3: helical interpolation in a counterclockwise direction.




**helical interpolation**

**calculation of feed rate for circular and helical interpolation:** On most CNC machines, the feed rate required for programming contour (circular or helical) milling is calculated based on the centerline of the tool. When dealing with linear tool movement, the feed rate at the cutting edge and centerline are identical, but with circular tool movement, this is not the case.

**calculate feed rate at the cutting edge:** First calculate the tool feed rate at the cutting edge with the following formula.

$$F_1 = ipt \times nt \times rpm$$

$F_1$ = tool feed rate at the cutting edge (in./min.)

ipt = inch per tooth (chip load)

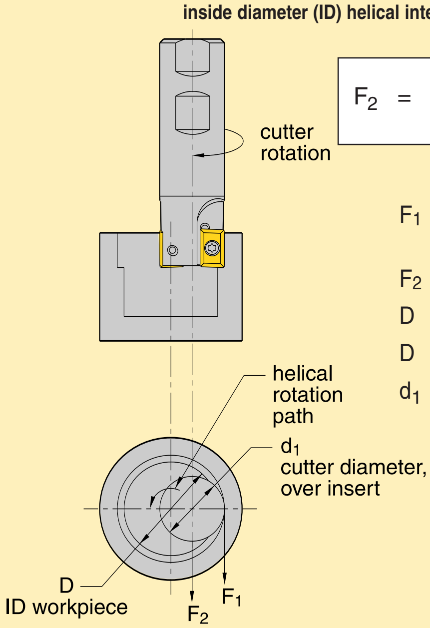
nt = number of effective inserts in the cutter

rpm = revolutions per minute

(See page M445 for converting sfm to rpm.)

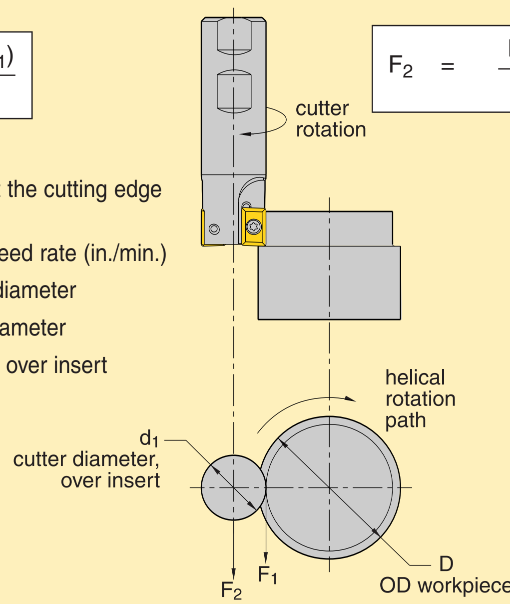
**Calculation of the Feed Rate at the Tool Centerline:** Use the following equations to define the relationship between feed rates at the cutting edge and at the tool centerline.

**inside diameter (ID) helical interpolation**




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

**outside diameter (OD) helical interpolation**




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

$F_1$ = 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

## ID and OD Circular and Helical Interpolation (cont'd.)



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

**example for ID**

D = 4" 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.**

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

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

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% less feed than the cutting edge feed rate ($F_1$).

In 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" 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% more feed rate than the cutting edge feed rate ($F_1$).

### Large Surfaces

Interpolating with a smaller cutter may be faster than using a manually-loaded, large-diameter cutter. Also, keep the cutter in contact with the workpiece rather than exiting and re-entering.

### Maximize Metal Removal Rate

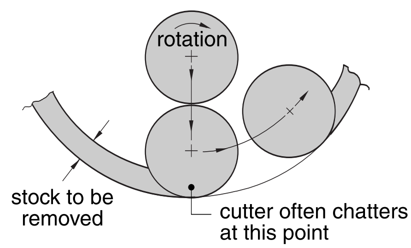
Concentrate on mrr (metal removal rate), not just on higher sfm (speeds). Increasing spindle speed without increasing chip load will not improve mrr. However, by doubling ipm, mrr does increase and horsepower consumption only increases by approximately 50%.

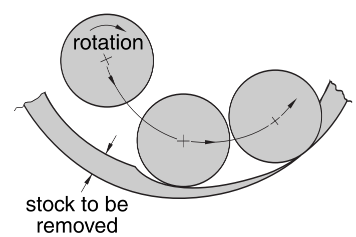
### Preset

Use cutter preset areas for proper setting of cutters rather than indexing cutters at the machine, if possible.

### Ramp In and Out

As shown below, ramping gradually into the cut will provide greater tool life. Also, by keeping the cutter constantly moving when entering and exiting the cut, dwell marks will be eliminated on the workpiece.




### Safety and Over Travel

Program the milling cutter to rapid advance up to the part, within a range of .125 before engaging the workpiece. This allows the machine to reach its proper operating parameters before actual chip making begins.

Rapid advance to the next cutting location, when the cutter is .020 to .040 past the edge of the part. If the spindle has built-in tilt or programmed runout, the cutter can be advanced to the next cutting location while the back half of the cutter is still over the finished milled surface.