

# Introduction to Turning Tools and their Application

*Identification and application of cutting tools for turning*



The variety of cutting tools available for modern CNC turning centers makes it imperative for machine operators to be familiar with different tool geometries and how they are applied to common turning processes.

This course curriculum contains 16-hours of material for instructors to get their students ready to identify different types of turning tools and their uses.



# Table of Contents

Introduction .....	2
Audience .....	2
Purpose .....	2
Lesson Objectives.....	2
Anatomy of a turning tool.....	3
Standard Inserts .....	3
ANSI Insert Designations.....	3
Insert Materials.....	9
Seat .....	10
Tool Holders .....	11
Boring Bars .....	13
Drills, Taps and Reamers.....	13
Twist Drills.....	13
Indexable Drills.....	14
Spot Drills and Center Drills .....	14
Taps and Single Point Thread Tools .....	15
Reamers .....	15
Feeds and Speeds.....	16
Cutting Speed.....	16
Feed Rate .....	17
Selection of Tools, Feeds, and Speeds .....	18

# Introduction

---

Turning generates axially symmetric shapes with a single-point tool. A single-point tool removes material by means of one cutting edge. In most cases the tool is held in a fixed position with the workpiece rotating about a turning axis. There are also tools held on the spindle centerline (drills, reamers, taps) for hole-making applications that have speed and feed limitations.

## Audience

This class is intended for entry-level turning operators and students in a turning operator training program who have a basic understanding of turning machines and their operation. This lesson is also useful to anyone interested in the metalworking industry who wants to gain knowledge about material removal in CNC machining.

## Purpose

Learn how to identify the components of common turning tools and how those tools are used in everyday turning processes. Students are introduced to ANSI turning insert nomenclature, tool holders and boring bars. They also learn about tools used for centerline machining such as twist drills, taps and reamers. Students finish with an introduction to the concepts of spindle speeds and feed rates.

## Lesson Objectives

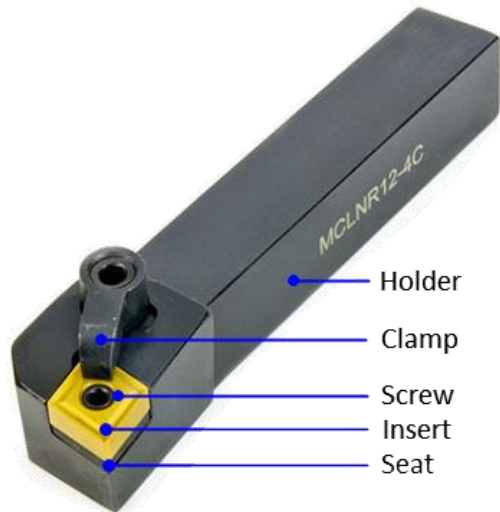
At the end of this lesson, you will know how to:

- Identify the components of common turning tools
- Identify the basic shapes of turning inserts
- Identify basic tool holders for external and internal turning
- Identify common types of tools for drilling and threading
- Identify the main groups of cutting tool materials
- Understand the applications for common turning tools
- Understand the difference between cut speed and feed rate
- Understand the difference between Revolutions Per Minute (RPM) and Constant Surface Speed (CSS)
- Understand the difference between Inches per Revolution (IPR) and Inches per Minute (IPM)

## Anatomy of a turning tool

---

Most turning is done using a replaceable insert that is gripped in a turning tool body, which is then mounted on the lathe turret.



## Standard Inserts

---

Turning inserts employ highly engineered composite structures, coatings, and geometry features to achieve great accuracy and high material removal rates.

The benefits of using replaceable inserts for turning tools include:

- Some inserts can be indexed to use other edges when one becomes worn
- Inserts are quickly and easily replaced at the machine

## ANSI Insert Designations

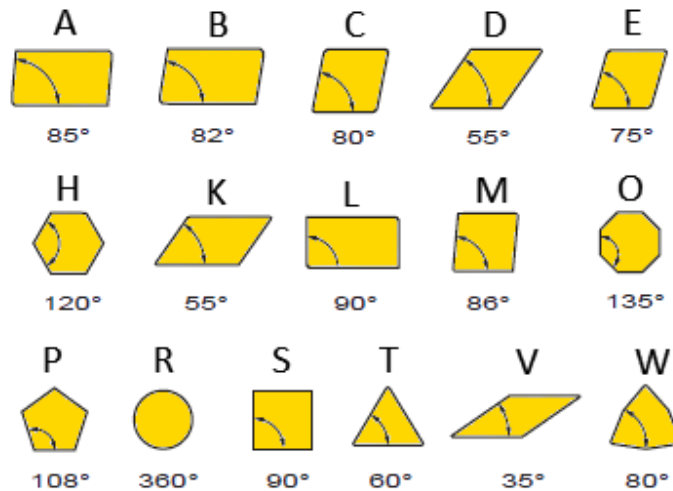
The American National Standards Institute (ANSI) has developed a coding system of numbers and letters to describe the shape, dimensions, and important parameters of turning inserts.

<b>C</b>	<b>N</b>	<b>M</b>	<b>G</b>	<b>-</b>	<b>4</b>	<b>3</b>	<b>2</b>
1	2	3	4		5	6	7
Shape	Clearance	Tolerance	Type		I.C. Size	Thickness	Nose Radius

## 1. Insert Shape

Turning inserts are manufactured in a variety of shapes, sizes and thicknesses. The shape can be round to maximize edge strength, diamond-shaped to allow a sharp point to cut fine features, square, or even octagonal to increase the number of separate edges that can be applied as one edge after another wears out.

- A = parallelogram 85°
- B = parallelogram 82°
- C = diamond 80°
- D = diamond 55°
- E = diamond 75°
- H = hexagon
- K = diamond 55°
- L = rectangle
- M = diamond 86°
- O = octagon
- P = pentagon
- R = round
- S = square
- T = triangle
- V = diamond 35°
- W - trigon 80°



C and W type turning inserts are often used for rough machining due to their larger point angle, which makes them more rigid. Inserts with a smaller point angle, such as D and V, are often used for finish machining. Although they have less strength, the smaller angle can reach more part details.

Large point angle:

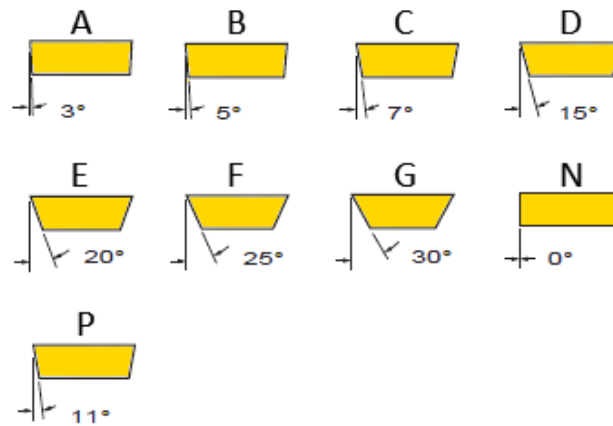
- Stronger cutting edge
- Higher feed rates
- Increased cutting forces
- Increased vibration

Small point angle:

- Weaker cutting edge
- Increased access to part details
- Decreased cutting forces
- Decreased vibration

## 2. Clearance

Most inserts have drafted faces on the walls. Clearance prevents the walls of the insert from rubbing against the part, which will give poor machining. However, a turning insert with a 0° clearance angle is mostly used for rough machining.



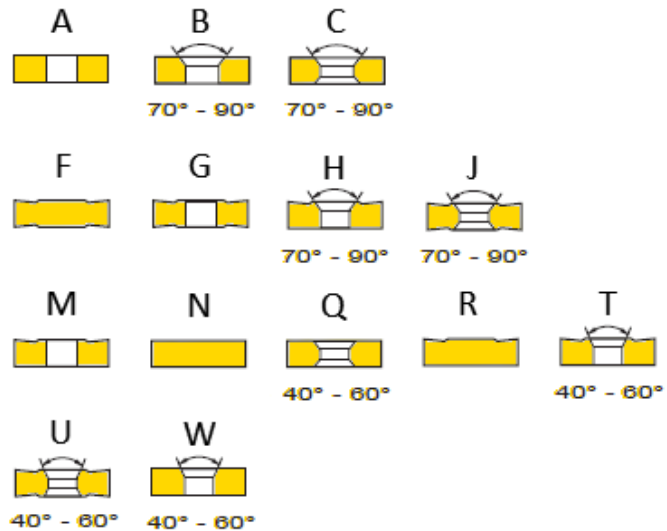
## 3. Tolerance

Letter	Inscribed Circle (I.C.)	Thickness	
A	±.0002	±.001	<p>The diagram shows a yellow turning insert with a square cross-section. A circle is inscribed within the square, labeled 'I.C.'. To the right of the insert, a vertical dimension line with arrows at both ends indicates the thickness of the insert.</p>
B	±.0002	±.005	
C	±.0005	±.001	
D	±.0005	±.005	
E	±.001	±.001	
F	±.002 to ±.004	±.002	
G	±.001	±.005	
M	±.002 to ±.010	±.005	
U	±.005 to ±.012	±.005	

#### 4. Type

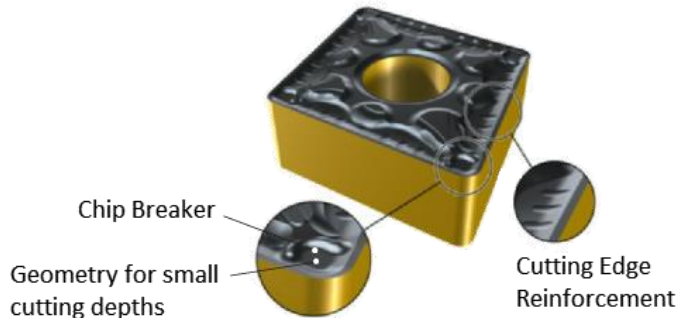
The turning insert hole shape and chip breaker type.

- A = Cylindrical hole
- B = 70-90° countersink hole
- C = Double countersink
- F = No hole; Double-sided chip breaker
- G = Cylindrical; Double-sided chip breaker
- H = 70-90° countersink; Single-sided
- J = Double countersink; Double-sided
- M = Cylindrical; Single-sided
- N = No hole; No chip breaker
- P = Cylindrical; Hi-Double-Positive Chip breaker
- Q = 40-60° Double Countersink; No chip breaker
- R = No hole; Single-sided
- S = Cylindrical; Hi-double positive
- T = 40-60° double countersink; Single-sided
- U = 40-60° double countersink; Double-sided
- W = 40-60° double countersink
- X = Special Design



#### 4. Type

A chip breaker is a feature in the face of the insert that disrupts the flow of chips such that they break into short segments, rather than forming a long, stringy chip.



#### 5. Size

This numeric value tells us the cutting edge length of the turning insert.



For equal sided inserts  $1/4''$  I.C. or over:

Size = Number of  $1/8''$  increments.

Examples:

2 =  $1/4''$   
 3 =  $3/8''$   
 4 =  $1/2''$

For equal sided inserts less than  $1/4''$  I.C.:

Size = Number of  $1/32''$  increments.

Examples:

2 =  $1/16''$   
 3 =  $3/32''$   
 4 =  $1/8''$

For rectangles and parallelograms, 2 digits are necessary (Width and Length).

Digit 1: Width in  $1/8''$  increments

Digit 2: Length in  $1/4''$  increments

Example:  
 12 =  $1/8'' \times 1/2''$

**Important:** The depth-of-cut for roughing should never exceed  $1/2$  the inscribed circle of the insert.



## 6. Thickness

This value tells us the thickness of the insert.

For inserts 1/4" I.C. or over: Thickness in 1/16" increments

For inserts less than 1/4" I.C.: Thickness in 1/32" increments

## 7. Corner

Form on the corner in 1/16" increments for inserts with a corner radius.

To reduce vibration, it is often an advantage to choose a nose radius that is smaller than the depth of cut.

0 = Sharp corner (.005R)

1 = 1/64" radius

2 = 1/32

3 = 3/64

4 = 1/16

5 = 5/64

6 = 3/32

7 = 7/64

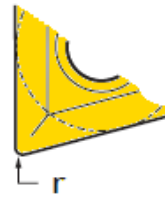
8 = 1/8

10 = 5/32

12 = 3/16

14 = 7/32

16 = 1/4



Large nose radius:

- Increased feed rates
- Large depths of cut
- Strong edge security
- Increased radial pressures

Small nose radius:

- Small cutting depths
- Reduced vibration
- Weak cutting edge

## Insert Materials

Insert material is typically carbide, though ceramic, cermet or diamond inserts can be applied to more demanding applications. A variety of protective coatings also help these insert materials cut faster and last longer.

Insert Material	Characteristics
Cemented carbide (HW, HC) <ul style="list-style-type: none"> <li>• HW: Uncoated</li> <li>• HC: Coated</li> </ul>	The most common material used in the industry today. It is offered in several “grades” containing different proportions of tungsten carbide and binder (usually cobalt). High resistance to abrasion.
Cermets (HT, HC) Cermet containing primarily titanium carbides (TiC) or titanium nitrides (TiN) or both <ul style="list-style-type: none"> <li>• HT: Uncoated</li> <li>• HC: Coated</li> </ul>	Another cemented material, based on titanium carbide (TiC). Binder is usually nickel. It provides higher abrasion resistance compared to tungsten carbide at the expense of some toughness. Extremely high resistance to abrasion.
Ceramics (CA, CM, CN, CC) <ul style="list-style-type: none"> <li>• CA Oxide ceramics containing primarily aluminum oxide (<math>Al_2O_3</math>)</li> <li>• CM Mixed ceramics containing primarily aluminum oxide (<math>Al_2O_3</math>) but containing components other than oxides</li> <li>• CN Nitride ceramics containing primarily silicon nitride (<math>Si_3N_4</math>)</li> <li>• CC Nitride ceramics containing primarily silicon nitride (<math>Si_3N_4</math>), but coated</li> </ul>	Chemically inert and extremely resistant to heat, ceramics are usually desirable in high speed applications, the only drawback being their high fragility. The most common ceramic materials are based on alumina (aluminium oxide), silicon nitride and silicon carbide.

Insert Material	Characteristics
Cubic boron nitrides (BN)	The second hardest substance. It offers extremely high resistance to abrasion at the expense of much toughness. It is generally used in a machining process called "hard machining", which involves running the tool or the part fast enough to melt it before it touches the edge, softening it considerably.
Polycrystalline diamonds (DP, HC) <ul style="list-style-type: none"> <li>• DP : Uncoated</li> <li>• HC : Coated</li> </ul>	The hardest substance. Superior resistance to abrasion but also high chemical affinity to iron which results in being unsuitable for steel machining. It is used where abrasive materials would wear anything else.

## Seat

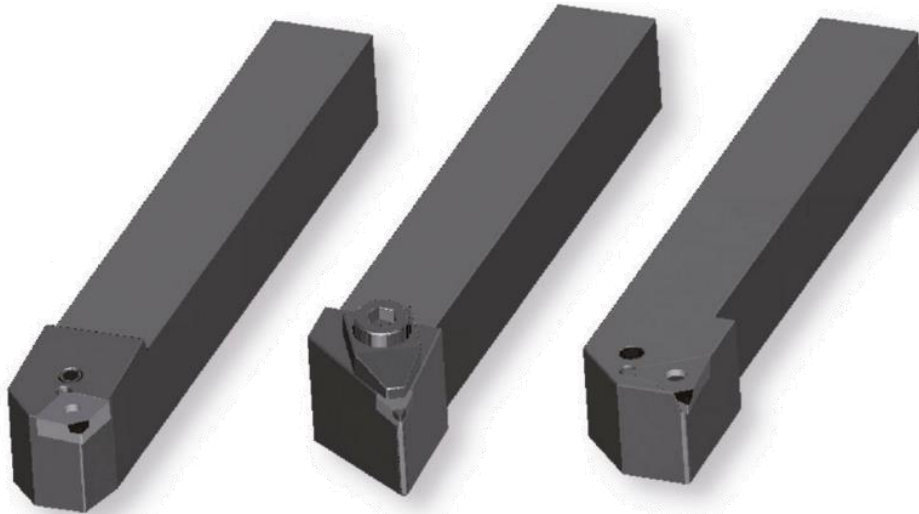
---

A piece of carbide, the same size as the insert it supports, placed between the insert and the bottom of the pocket where the insert fits in the tool holder.

## Tool Holders

---

It is essential that the insert be supported in a strong, rigid manner to minimize deflection and possible vibration. Consequently, turning tools are supported in various types of heavy, forged steel tool holders.



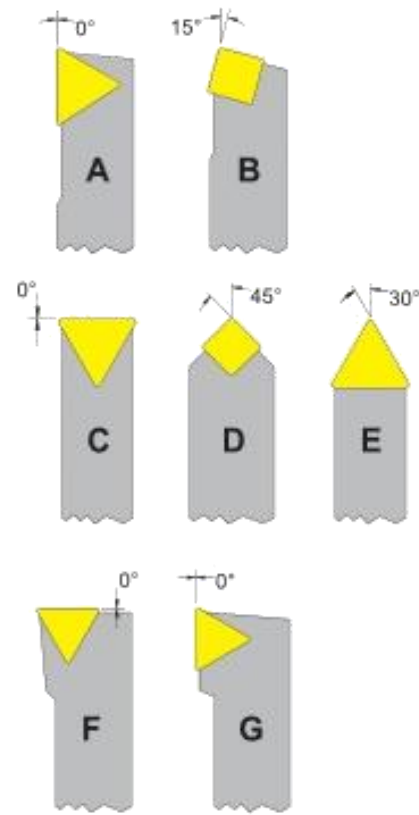
The turning tool body generally does not feature quite so much engineering as the insert, but even here there are a range of choices for fine-tuning the process. Quick-change tools involve modular bodies that allow replacement tool bodies to be swapped in and out and locked in place quickly to minimize setup time. The turning tool body can also channel high-pressure coolant more effectively to the cutting edge of the tool.

The ANSI numbering system for turning tool holders has assigned letters to specific geometries in terms of lead angle and end cutting edge angle. The primary lathe machining operations of turning, facing, grooving, threading and cutoff are covered by one of the seven basic tool styles outlined by the ANSI system.

The designations for the seven primary tool styles are A, B, C, D, E, F and G.

## Tool Holders A - G

- A = 0° side-cutting edge angle, straight shank
- B = 15° side-cutting edge angle, straight shank
- C = 0° end-cutting edge angle, straight shank (for cutoff and grooving operations)
- D = 45° side-cutting edge angle, straight shank
- E = 30° side-cutting edge angle, straight shank (for threading operations)
- F = 0° end-cutting edge angle, offset shank (for facing operations)
- G = 0° side-cutting edge angle; offset shank (this tool is an 'A' style tool with additional clearance built in for turning operations close to the lathe chuck)



**Tip:** The most commonly used insert/holder combination for O.D. rough turning and facing is the C type 80° diamond insert with a 3-5° negative lead tool holder. It is often selected because it is the best compromise between strength of insert and end-angle clearance.

## Boring Bars

---

Boring bars are cylindrical bars used for internal machining, typically to produce precise holes. Boring bars are also used for internal turning. Boring bars are available in steel, solid carbide, and carbide-reinforced steel.



Boring tools consist of a round shaft with one insert pocket designed to reach into a part hole or cavity to remove internal stock. Boring tools enlarge a hole made by a previous process.

Boring bars have the following advantages for hole making:

- Ensuring precision and straightness
- Producing a hole larger than standard drill bit sizes
- Producing a non-standard size hole
- Producing a hole with a good finish

## Drills, Taps and Reamers

---

High Speed Steel (HSS) tools such as drills, taps and reamers are commonly used on CNC machining centers for hole making operations.

### Twist Drills

Twist drills have a conical cutting point at the tip of a cylindrical shaft that has one or more helical flutes. The flutes are designed to evacuate chips out of the hole being drilled. Twist drills are available in a variety of sizes, lengths, materials and coatings, but the most common are High Speed Steel (HSS) and solid carbide.



Tool coatings have an influence on the cutting process by increasing cutting speed and tool life. Coatings such as titanium nitride (TiN) increase the initial cost but reduce wear and increase tool life. Applied as a thin coating, TiN is used to harden and protect cutting surfaces.

Twist drills coated with titanium nitride (TiN) are easily identified by a gold like color. This coating increases the hardness of the bit and adds a self-lubricating property.



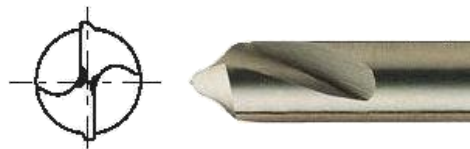
## Indexable Drills

The indexable drill is a two-fluted, center-cutting tool with indexable carbide inserts.



## Spot Drills and Center Drills

Spot drills are designed to be extremely rigid so that they can precisely spot a hole for a twist drill. The goal is use the spot drill to make a little dimple in the workpiece that keeps the twist drill from walking so that the hole winds up in the right place.



Center drills are intended to be used to create a 60° center in the end of lathe stock. They have a 2-part tip that has a small pilot as well as the larger countersinking area of the bit.



If you use a carbide drill, spotting is typically not needed. The carbide itself is so rigid compared to HSS that the drill will go where it is pointed.

## Taps and Single Point Thread Tools

Taps are used to cut internal threads of a specific size and pitch. A tap requires a hole be drilled first to the size of the minor diameter.



A single point thread mill is a cutter whose shape is the thread form. On a lathe, that cutter is synchronized to the rotation of the part so it perfectly tracks the helix that is a thread. Successive passes cut deeper and deeper to form threads on the interior or exterior of the stock.



## Reamers

Reamers are used to enlarge an existing hole to a precise tolerance and to add a high quality surface finish. Reamers require a hole be drilled first that is fairly close to the final size so that the reamer actually removes relatively little material. Reamers ensure a hole has an accurate diameter, roundness, and good surface finish.





# Feeds and Speeds

---

Feeds and speeds refer to two separate velocities for machine tools: feed rate and cutting speed. They are used together because of their combined effect on the cutting process.

- Cut speed is the speed at the outside edge of the part as it is rotating, also known as surface speed
- Feed rate is the velocity at which the cutter is advanced along the spinning workpiece

## Cutting Speed

Cutting speed is the speed that the material moves past the cutting edge of the tool. Cut speed can be defined as revolutions per minute (RPM) or as surface feet per minute (SFM).

Revolutions Per Minute (RPM) relates directly to the speed, or velocity, of the spindle. It annotates the number of turns completed in one minute around a fixed axis. RPM maintains the same revolutions per minute throughout the entire operation.

RPM mode is useful for:

- Center cutting operations (drilling)
- When the diameter at the beginning and end of a cut only differs slightly
- During threading to allow the perfect synchronization between spindle revolution and Z-axis motion to allow precise threads

Surface Feet Per Minute (SFM) is a combination of the cut diameter and RPM. The faster the spindle turns, and/or the larger the part diameter, the higher the SFM.

If two round pieces of different sizes are turning at the same revolutions per minute, the larger piece has a greater surface speed because it has a larger circumference and has more surface area. As the tool plunges closer to the center of a workpiece, the same spindle speed will yield a decreasing surface speed. This is because each revolution represents a smaller circumferential distance, but takes the same amount of time.

Most CNC lathes have CSS (constant surface speed) to counteract the natural decrease in surface speed, which speeds up the spindle as the tool moves closer to the turning axis. CSS adjusts the revolutions per minute to maintain a constant surface speed at every distance from the center.

CSS is useful for:

- A uniform surface finish
- When the diameter at the beginning a cut will differ significantly from the diameter at the end of the cut
- Better tool life and machining time because tools will always cut at the appropriate speed

Materials will run better at specific SFMs. SFM is a constant, with RPM as a variable based upon cut diameter.

When the SFM constant is known for a specific material, the formulas below can be used to determine spindle speed:

$$\text{RPM} = \text{SFM} \times (12/\text{Pi}) / \text{Cut Diameter}$$

So, for an SFM of 400 and a cut diameter of 5", RPM can be found:

$$\text{RPM} = 400 \times 3.82 / 5 = 306 \text{ RPM}$$

When RPM is known, SFM can be found as follows:

$$\text{SFM} = \text{RPM} \times \text{Cut Diameter} \times \text{Pi}/12$$

For example, if you have a cut diameter of 5" and a spindle speed of 306 RPM, then:

$$\text{SFM} = 306 \times 5 \times .262 = 400 \text{ SFM}$$

## Feed Rate

Feed rate is the velocity at which the cutter is advanced along the workpiece. Feed rate is expressed as units of distance (inch) per minute or per single revolution.

Feed rate can be defined as inch per minute (IPM) or inch per revolution (IPR). IPR is more commonly used.

Values for IPR and IPM are easily converted with the following formulas:

$$\text{IPM} = \text{IPR} \times \text{RPM}$$

$$\text{IPR} = \text{IPM} / \text{RPM}$$

So, for a spindle speed of 306 RPM and a feed rate of .01 IPR, the IPM can be calculated as follows:

$$\text{IPM} = .01 \times 306 = 3 \text{ IPM}$$

## Selection of Tools, Feeds, and Speeds

Cutting tool selection has a direct impact on the proper programming of feeds and speeds at the machine.

However, many other variables that affect feeds and speeds are:

- Workpiece material class and condition
- Workpiece diameter
- Cutter material
- Cutter geometry
- Type of cut
- Depth of cut
- Condition of the machine

Cutting tool manufacturers publish the general feeds and speeds and recommended usage for the application. Cutting tool manufacturers are often a good place to start for recommendations on tool selection and feeds/speeds since they rely on customer loyalty. The customer (or potential) should select an insert and grade based on the vendor's recommendation.