

Cutting Tool Applications

Cutting Tool Applications

By George Schneider, Jr. CMfgE



Upcoming Chapters

Metal Removal

Cutting-Tool Materials
Metal Removal Methods
Machinability of Metals

Single Point Machining

Turning Tools and Operations
Turning Methods and Machines
Grooving and Threading
Shaping and Planing

Hole Making Processes

Drills and Drilling Operations
Drilling Methods and Machines
Boring Operations and Machines
Reaming and Tapping

Multi Point Machining

Milling Cutters and Operations
Milling Methods and Machines
Broaches and Broaching
Saws and Sawing

Abrasive Processes

Grinding Wheels and Operations
Grinding Methods and Machines
Lapping and Honing



George Schneider, Jr. CMfgE

Professor Emeritus
Engineering Technology
Lawrence Technological University

Former Chairman
Detroit Chapter ONE
Society of Manufacturing Engineers

Former President
International Executive Board
Society of Carbide & Tool Engineers

Lawrence Tech. Univ.: <http://www.ltu.edu>
Prentice Hall: <http://www.prenhall.com>

Chapter 4

Turning Tools

& Operations

4.1 Introduction

Turning is a metal cutting process used for the generation of cylindrical surfaces. Normally the workpiece is rotated on a spindle and the tool is fed into it radially, axially, or both ways simultaneously, to give the required surface. The term 'turning', in the general sense, refers to the generation of any cylindrical surface with a single point tool. More specifically it is often applied just to the generation of external cylindrical surfaces oriented primarily parallel to the workpiece axis. The generation of surfaces oriented primarily perpendicular to the workpiece axis are called 'facing'. In turning the direction of the feeding motion is predominantly axial with respect to the machine spindle. In facing a radial feed is dominant. Tapered and contoured surfaces require both modes of tool feed at the same time often referred to as 'profiling'.

Turning facing and profiling operations are shown in Figure 4.1

The cutting characteristics of most turning applications are similar. For a given surface only one cutting tool is used. This tool must overhang its holder to some extent to enable the holder to clear the rotating workpiece. Once the cut starts, the tool and the workpiece are usually in contact until the surface is completely generated. During this time the cutting speed and cut dimensions will be constant when a cylindrical surface is being turned. In the case of facing operations the cutting speed is proportional to the work diameter, the speed decreasing as the center of the piece is approached. Sometimes a spindle speed changing mechanism is provided to increase the rotating speed of the workpiece as the tool moves to the center of the part.

In general, turning is characterized by steady conditions of metal cutting. Except at the beginning and end of the cut, the forces on the cutting tool and the tool tip temperature are essentially constant. For the special case of facing, the varying cutting speed will affect the tool tip temperature. Higher temperatures will be encountered at the larger diameters on the workpiece. However, since cutting speed has only a small effect on cutting forces, the forces acting on a facing tool may be expected to remain almost constant during the cut.

4.2 Related Turning Operations

A variety of other machining operations can be performed on a lathe in addition to turning and facing. These include the following, as shown in Figure 4.2a through 4.2f.

Single point tools are used in most operations performed on a lathe. A short description of six additional lathe operations are given below:

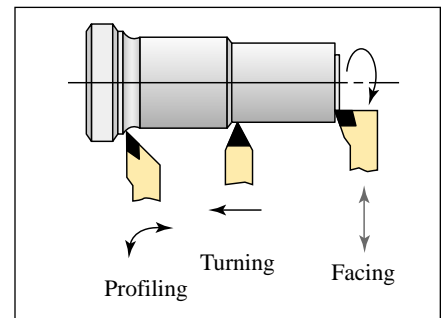


FIGURE 4.1: Diagram of the most common lathe operations: facing, turning, and profiling.

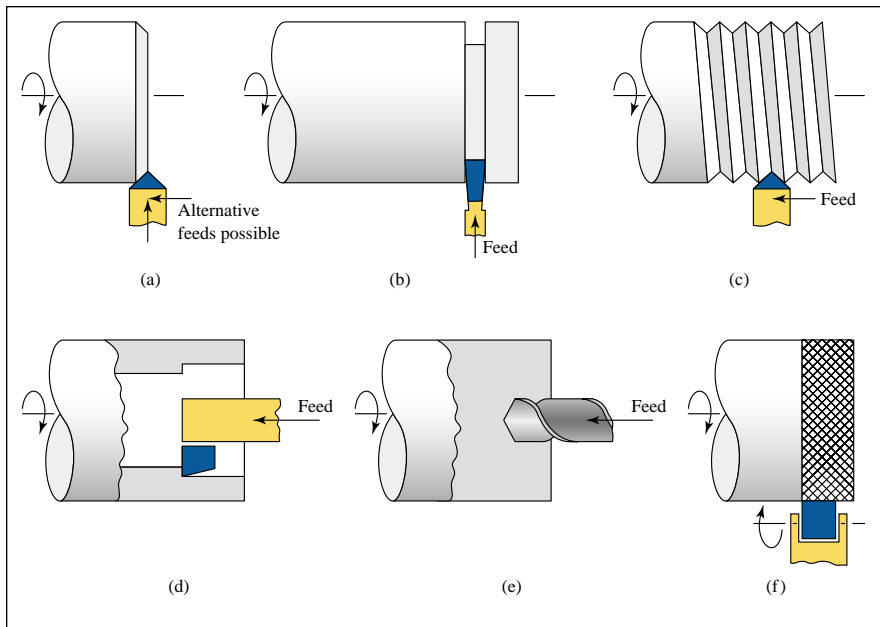


FIGURE 4.2: Related turning operations: (a) chamfering, (b) parting, (c) threading, (d) boring, (e) drilling, (f) knurling.

Chamfering: The tool is used to cut an angle on the corner of a cylinder.

Parting: The tool is fed radially into rotating work at a specific location along its length to cut off the end of a part.

Threading: A pointed tool is fed linearly across the outside or inside surface of rotating parts to produce external or internal threads.

Boring: Enlarging a hole made by a previous process. A single point tool is fed linearly and parallel to the axis

of rotation.

Drilling: Producing a hole by feeding the drill into the rotating work along its axis. Drilling can be followed by reaming or boring to improve accuracy and surface finish.

Knurling: Metal forming operation used to produce a regular cross-hatched pattern in work surfaces.

Chamfering and profiling operations are shown in Figures 4.3a and 4.3b respectively.

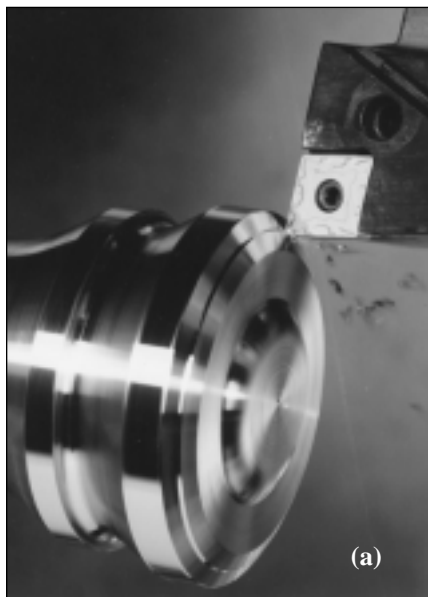


FIGURE 4.3: Chamfering (a) and profiling (b) operations, typically performed on a lathe or a machining center. (Courtesy Valenite Inc.)

4.3 Turning Tool Holders

Mechanical Tool Holders and the ANSI Identification System for Turning Tool Holders and indexable inserts were introduced in Chapter 2. A more detailed discussion of Toolholder Styles and their application will be presented here.

4.3.1 Toolholder Styles

The ANSI numbering system for turning toolholders 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.

A STYLE - Straight shank with 0 degree side cutting edge angle, for turning operations.

B STYLE - Straight shank with 15 degree side cutting edge angle, for turning operations.

C STYLE - Straight shank with 0 degree end cutting edge angle, for cutoff and grooving operations.

D STYLE - Straight shank with 45 degree side cutting edge angle, for turning operations.

E STYLE - Straight shank with 30 degree side cutting edge angle, for threading operations.

F STYLE - Offset shank with 0 degree end cutting edge angle, for facing operations.

G STYLE - Offset shank with 0 degree side cutting edge angle; this tool is an 'A' style tool with additional clearance built in for turning operations close to the lathe chuck.

There are many other styles of turning tools available in addition to those shown here, as detailed by the ANSI numbering system (see Figure 2.35). The seven basic tools are shown in operation in Figure 4.4

Right and Left Hand Toolholders

The toolholder styles discussed here and shown above represent a fraction of those standard styles available from most indexable cutting tool manufacturers. ANSI standard turning tools can be purchased in either right or left hand styles. The problem of identifying a right hand tool from a left hand tool can be resolved by remembering that when

holding the shank of a right hand tool as shown in Figure 4.5 (insert facing upward), will cut from left to right.

4.3.2 Turning Insert Shapes

Indexable turning inserts are manufactured in a variety of shapes, sizes, and thicknesses, with straight holes, with countersunk holes, without holes, with chipbreakers on one side, with chipbreakers on two sides or without chipbreakers. The selection of the appropriate turning toolholder geometry accompanied by the correct insert shape and chip breaker geometry, will ultimately have a significant impact on the productivity and tool life of a specific turning operation.

Insert strength is one important factor in selecting the correct geometry for a workpiece material or hardness range. Triangle inserts are the most popular shaped inserts primarily because of their wide application range. A triangular insert can be utilized in any of the seven basic turning holders mentioned earlier. Diamond shaped inserts are used for profile turning operations while squares are often used on lead angle tools. The general rule for rating an insert's strength based on its shape is: 'the larger the included angle on the insert corner, the greater the insert strength'.

The following list describes the different insert shapes from strongest to weakest. The relationship between insert shapes and insert strength was shown in Chapter 2. (see Fig. 2.28)

Insert Letter Designation	Insert Description	Insert Included Angle
R	Round	N/A
O	Octagon	135
H	Hexagon	120
P	Pentagon	108
S	Square	90
C	Diamond	80
T	Triangle	60
D	Diamond	55
V	Diamond	35

Six common turning tool holders are shown in Figure 4.6a and five common indexable insert shapes with molded chip breakers are shown in Figure 4.6b.

4.4 Operating Conditions

Operating conditions control three important metal cutting variables: metal removal rate, tool life, and surface

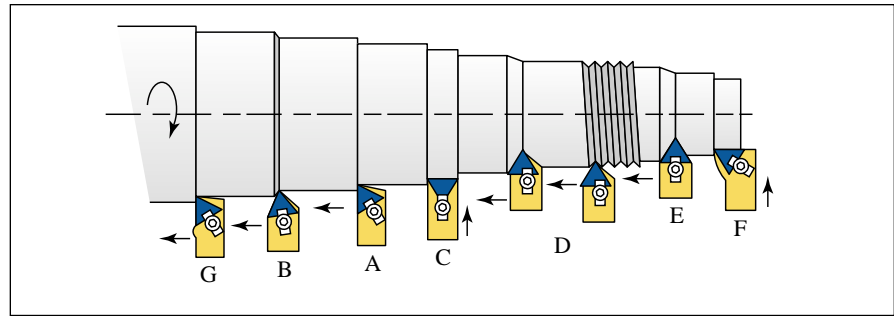


FIGURE 4.4: The primary lathe machining operations of turning, facing, grooving, threading and cut-off are performed with one of seven basic toolholder styles.

finish. Correct operating conditions must be selected to balance these three variables and to achieve the minimum machining cost per piece, the maximum production rate, and/or the best surface finish whichever is desirable for a particular operation.

The success of any machining operation is dependent on the set-up of the workpiece and the cutting tool. Set-up becomes especially important when the workpiece is not stiff or rigid and when the tooling or machine tool components must be extended to reach the area to be machined.

Deflection of the workpiece, the cutting tool, and the machine, is always present and can never be eliminated totally. This deflection is usually so minimal that it has no influence on an operation, and often goes unnoticed. The deflection only becomes a problem when it results in chatter, vibration, or distortion. It is therefore, very important to take the necessary time and effort to ensure that the set-up is as rigid as possible for the type of operation to be performed. This is especially important when making heavy or interrupted cuts.

Balancing should be considered when machining odd-shaped workpieces, especially those workpieces that have uneven weight distribution and those which are loaded off-center. An unbalanced situation can be a safety hazard and can cause work inaccuracies, chatter, and damage to the machine tool. While unbalance problems may not be apparent, they may exist at low speed operations and will become increasingly severe as the speed is increased. Unbalance condi-

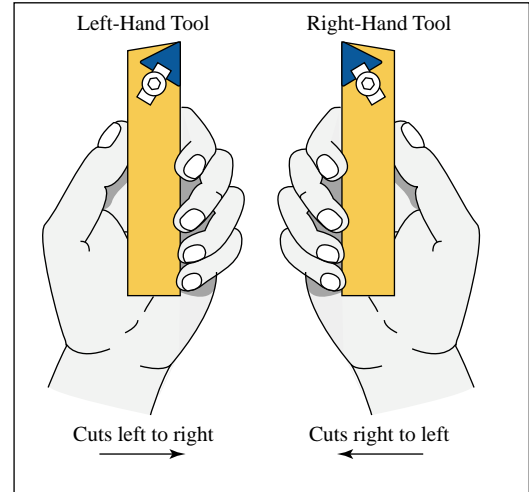


FIGURE 4.5: Identification method for right- and left-hand turning toolholders.

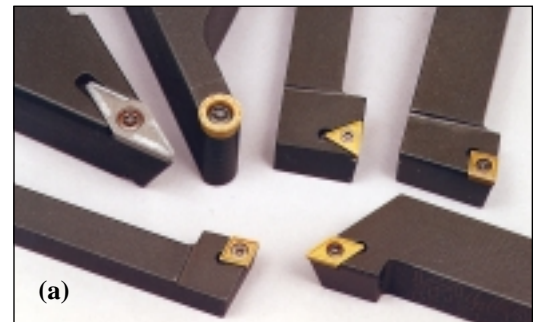


FIGURE 4.6: Common turning toolholders (a) and common indexable insert shapes (b) with molded chipbreakers are shown. (Courtesy Valenite Inc.)



FIGURE 4.7: Operating conditions become very important when machining very large parts. (Courtesy Sandvik Coromant Corp.)

tions most often occur when using turntables and lathe face plates.

As material is removed from the workpiece, the balance may change. If a series of roughing cuts causes the workpiece to become unbalanced, the problem will be compounded when the speed is increased to take finishing cuts. As a result, the reasons for problems in achieving the required accuracy and surface finish may not be apparent until the machining operation has progressed to the finishing stage. Operating conditions become very important when machining very large parts as shown in Figure 4.7.

4.4.1 Work Holding Methods

In lathe work the three most common work holding methods are:

- Held in a chuck



FIGURE 4.8: Many of the various work-holding devices used on a lathe for turning operations. (Courtesy Kitagawa Div. Sumikin Bussan International Corp.)

- Held between centers
- Held in a collet

Many of the various work holding devices used on a lathe are shown in Figure 4.8.

Chucks: The most common method of work holding, the chuck, has either three or four jaws (Fig. 4.9) and is mounted on the end of the main

spindle. A three jaw chuck is used for gripping cylindrical workpieces when the operations to be performed are such that the machined surface is concentric with the work surfaces.

The jaws have a series of teeth that mesh with spiral grooves on a circular plate within the chuck. This plate can be rotated by the key inserted in the square socket, resulting in simultaneous radial motion of the jaws. Since the jaws maintain an equal distance from the chuck axis, cylindrical workpieces are automatically centered when gripped.

Three jaw chucks, as shown in Figure 4.10, are often used to automatically clamp cylindrical parts using either electric or hydraulic power.

With the four jaw chuck, each jaw can be adjusted independently by rotation of the radially mounted threaded screws. Although accurate mounting of a workpiece can be time consuming, a four jaw chuck is often necessary for non-cylindrical workpieces. Both three and four jaw chucks are shown in Figure 4.8.

Between Centers: For accurate turning operations or in cases where the work surface is not truly cylindrical, the workpiece can be turned between centers. This form of work holding is illustrated in Fig. 4.11. Initially the workpiece has a conical center hole drilled at each end to provide location for the lathe centers. Before supporting the workpiece between the centers (one in the headstock

and one in the tailstock) a clamping device called a ‘dog’ is secured to the workpiece. The dog is arranged so that the tip is inserted into a slot in the drive plate mounted on the main spindle, ensuring that the workpiece will rotate with the spindle.

Lathe centers support the workpiece between the headstock and the tailstock. The center used in the headstock spindle is called the ‘live’ center. It rotates with the headstock spindle. The ‘dead’ center is located in the tailstock spindle. This center usually does not rotate and must be hardened and lubricated to withstand the wear of the revolving work. Shown in figure 4.12 are three kinds of dead

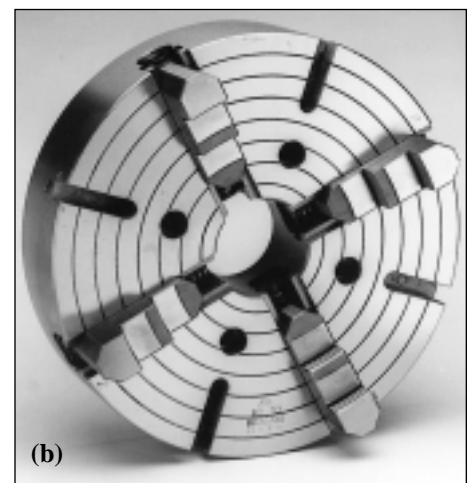


FIGURE 4.9: The most common method of work holding, the chuck, has either three jaws (a) or four jaws (b). (Courtesy Kitagawa Div. Sumikin Bussan International Corp.)

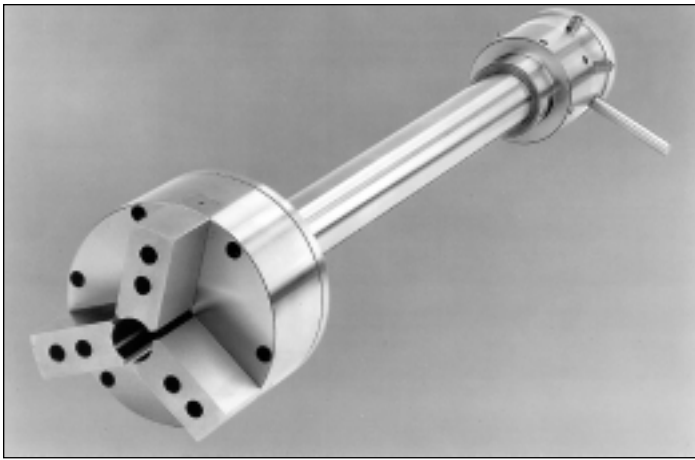


FIGURE 4.10: Three-jaw chucks are often used in automated machining systems pneumatically or hydraulically clamp cylindrical parts. (Courtesy Royal Products)



FIGURE 4.12: Hardened “dead” centers are mounted in the tailstock; they do not rotate with the workpiece and must be lubricated. (Courtesy Stark Industrial, Inc.)

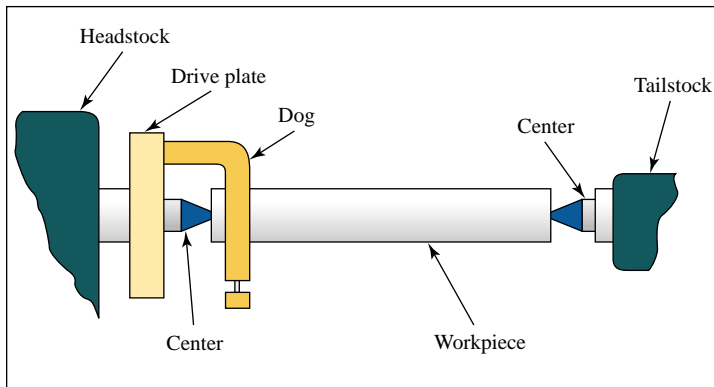


FIGURE 4.11: For accurate machining, cylindrical parts can be turned between centers.

centers.

As shown in Figure 4.13, some manufacturers are making a roller-bearing or ball-bearing center in which the center revolves.

The hole in the spindle into which the center fits, is usually of a Morse standard taper. It is important that the hole in the spindle be kept free of dirt and also that the taper of the center be clean and free of chips or burrs. If the taper of the live center has particles of dirt or a burr on it, it will not run true. The centers play a very important part in lathe operation. Since they give support to the workpiece, they must be properly ground and in perfect alignment with each other. The workpiece must have perfectly drilled and countersunk holes to receive the centers. The center must have a 60 degree point.

Collets: Collets are used when smooth bar stock, or workpieces that have been machined to a given diameter, must be held more accu-

ately than normally can be achieved in a regular three or four jaw chuck. Collets are relatively thin tubular steel bushings that are split into three longitudinal segments over about two thirds of their length (See Fig. 4.14a). The smooth internal surface of the split end is shaped to fit the piece of stock that is to be held. The external surface at the split end is a taper that fits within an internal taper of a collet sleeve placed in the spindle hole. When the collet is pulled inward into the spindle, by means of the draw bar that engages threads on the inner end of the collet, the action of the two mating

tapers squeezes the collet segments together, causing them to grip the workpiece (Fig. 4.14b).

Collets are made to fit a variety of symmetrical shapes. If the stock surface is smooth and accurate, collets will provide accurate centering; maximum runout should be less than 0.0005 inch. However, the work should be no more than 0.002 inch larger or 0.005 inch smaller than the nominal size of the collet. Consequently, collets are used only on drill rod, cold drawn, extruded, or previously machined material.

Another type of collet has a size range of about 1/8 inch. Thin strips of hardened steel are bonded together on their sides by synthetic rubber to form a truncated cone with a central hole. The collet fits into a tapered spindle sleeve so that the outer edges of the metal strips are in contact with the inner taper of the sleeve. The inner edges bear against the workpiece. Pulling the collet into the adapter sleeve causes the strips to grip the work. Because of their greater size range, fewer of these collets

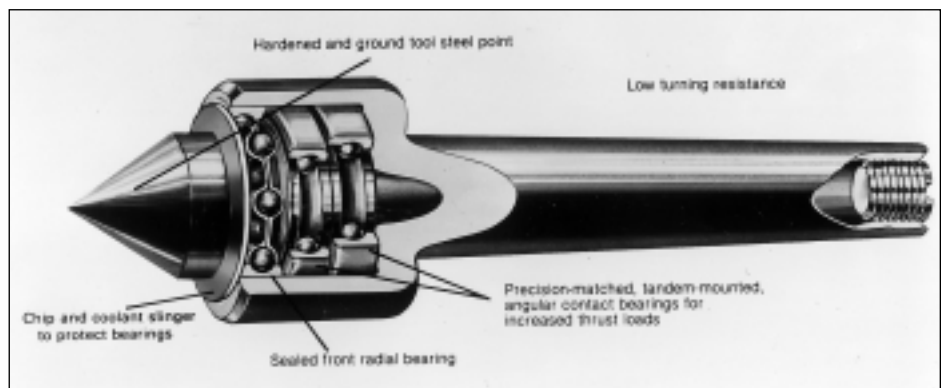
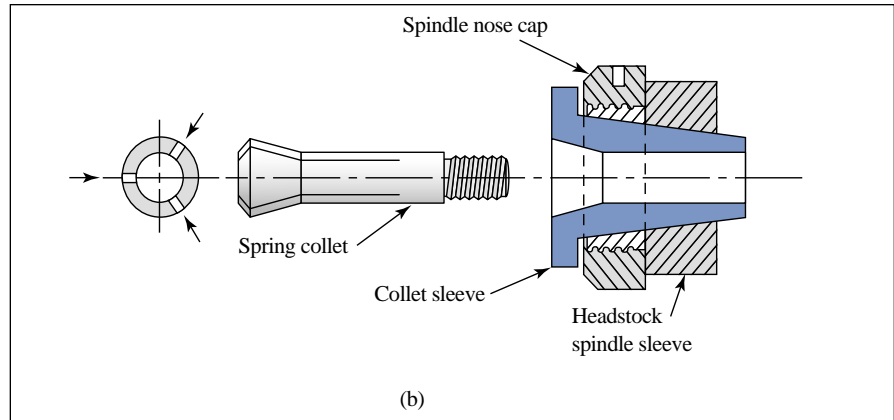


FIGURE 4.13: Hardened “live” centers are mounted in the tailstock; they rotate with the workpiece and do not need lubrication. (Courtesy Royal Products)



FIGURE 4.14: A collet (a) and a collet mounting assembly (b) are shown here. (Courtesy Lyndex Corp.)



are required than with the ordinary type.

4.4.2 Tool Holding Devices

The simplest form of tool holder or post is illustrated in Figure 4.15a and is suitable for holding one single-point tool. Immediately below the tool is a curved block resting on a concave spherical surface. This method of support provides an easy way of inclining the tool so that its corner is at the correct height for the machining operation. In Figure 4.15a the tool post is shown mounted on a compound rest. The rest is a small slideway that can be clamped in any angular position in the horizontal plane and is mounted on the cross slide of the lathe. The compound rest allows the tool to be hand fed at an oblique angle to the lathe bed and is required in operations like screw-threading and the machining of short tapers or chamfers.

Another common form of tool post, the square turret, is shown in Figure 4.15b. It also is mounted on the compound rest. As its name suggests, this four-way tool post can accommodate as many as four cutting tools. Any cutting tool can be quickly brought into position by unlocking the tool post with the lever provided, rotating the tool post, and then reclamping with the lever.

All standard tool holders are designed to cut with the cutting point located on the centerline of the machine and workpiece. If the cutting point is not on the centerline, as shown in Figure 4.16a, the clearance angle between the tool holders and the workpiece will be reduced. The lack of clearance will lead to poor tool life and poor surface finish. It will also force the workpiece away from the tool when working with small diameters.

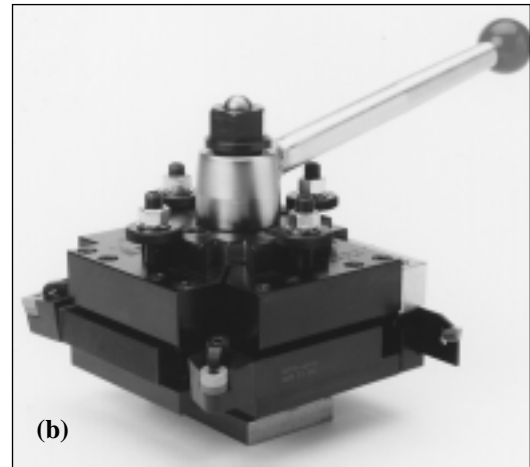
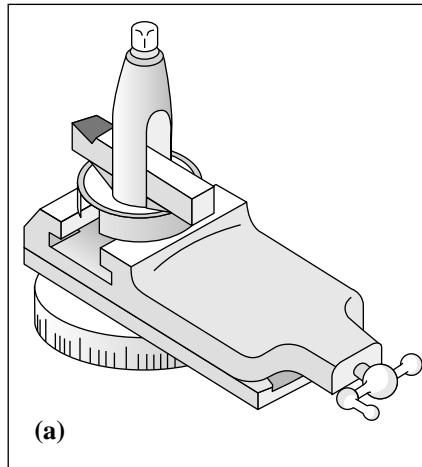


FIGURE 4.15: A toolpost for single-point tools (a) and a quick change indexing square turret, which can hold up to four tools (b). (Courtesy Dorian Tool)

On the other hand, if the cutting edge is positioned below the centerline, as shown in Figure 4.16b, the rake angle becomes more negative. Very high cutting forces will be generated and the chip will be directed into a tight curl. Insert fracture can very easily occur and a small diameter workpiece can even climb over the top of the tool and be torn from the machine.

Occasionally, however, moving the cutting point off centerline can solve a problem. An example is in situations when machining flimsy parts or when

(2% to 4% of the workpiece diameter) will change the rake angle slightly, and this in turn, will reduce cutting forces and make chatter less of a danger.

Interrupted cuts present special problems, particularly when machining large diameter workpieces. It is best to position the cutting point slightly below the centerline to present the insert in a stronger cutting position. A lead angle should also be used whenever possible. Moving the cutting point slightly below the centerline and using a lead angle, allows the workpiece to contact the tool

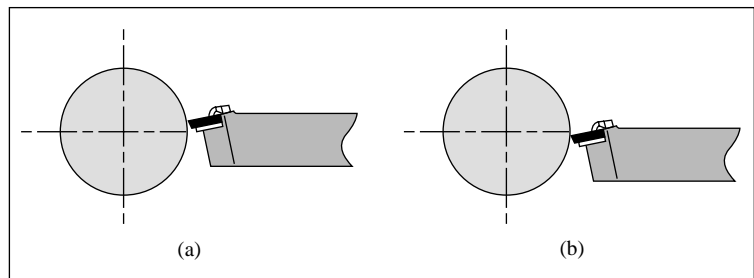


FIGURE 4.16: Cutting edge above workpiece centerline (a) and cutting edge below workpiece centerline (b). Both conditions result in poor performance.

on a stronger part of the insert, behind the nose.

4.5 Cutting Conditions

After deciding on the machine tool and cutting tool, the following main cutting conditions have to be considered:

- Cutting speed
- Depth of cut
- Feed rate

The choice of these cutting conditions will affect the productivity of the machining operation in general, and the following factors in particular: the life of the cutting tool; the surface finish of the workpiece; the heat generated in the cutting operation (which in turn affects the life of the tool and the surface integrity of the machined parts); and the power consumption.

Cutting Speed: Cutting speed refers to the relative surface speed between tool and work, expressed in surface feet per minute. Either the work, the tool, or both, can move during cutting. Because the machine tool is built to operate in revolutions per minute, some means must then be available for converting surface speeds into revolutions per minute (RPM). The common formula for conversion is:

$$\text{SFPM} = \frac{D \times \pi \times (\text{RPM})}{12}$$

where D is the diameter (in inches) of the workpiece or the rotating tool, π equals the constant 3.1416, and RPM is a function of the speed of the machine tool in revolutions per minute. For example:

If a lathe is set to run at 250 RPM and the diameter of the workpiece is 5 inches, then:

$$\text{SFPM} = \frac{3.1416 \times 5 \times 250}{12} = \frac{3927}{12}$$

Answer = 327.25 or 327 SFPM

The RPM can be calculated when another cutting speed is desired. For example, to use 350 SFPM with a workpiece which is 4 inches in diameter.

$$\text{RPM} = \frac{12 \times \text{SFPM}}{\pi \times D}$$
$$\text{RPM} = \frac{12 \times 350}{3.1416 \times 4} = \frac{4200}{12.57}$$

Answer = 334.13 or 334 RPM

All tool materials are meant to run at a certain SFM when machining various work materials. The SFPM range recommendations for tool and work materials are given in many reference publications.

Depth of Cut: The depth of cut relates to the depth the tool cutting edge engages the work. The depth of cut determines one linear dimension of the area of cut. For example: to reduce the outside diameter (OD) of a workpiece by .500 inches, the depth of cut would be .250 inches.

Feed Rate: The feed rate for lathe turning is the axial advance of the tool along the work for each revolution of the work expressed as inches per revolution (IPR). The feed is also expressed as a distance traveled in a single minute or IPM (inches per minute). The following formula is used to calculate the feed in IPM:

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

Feed, speed, and depth of cut have a direct effect on productivity, tool life, and machine requirements. Therefore these elements must be carefully chosen for each operation. Whether the objective is rough cutting or finishing will have a great influence on the cutting conditions selected.

Roughing Cuts: When roughing, the goal is usually maximum stock removal in minimum time with minor consideration given to tool life and surface finish. There are several important points to keep in mind when rough cutting.

The first is to use a heavy feed because this makes the most efficient use of power and, with less tool contact, tends to create less chatter. There are some exceptions where a deeper cut is more advantageous than a heavy feed, especially where longer tool life is needed. Increasing the depth of cut will increase tool life over an increase in feed rate. But, as long as it is practical and chip formation is satisfactory, it is better to choose a heavy feed rate.

A heavy feed or deeper cut is usually preferable to higher speed, because the machine is less efficient at high speed. When machining common materials, the unit horsepower (HP) factor is reduced in the cut itself, as the cutting speed increases up to a certain critical value. But the machine inefficiencies will overcome any advantage when

machining heavy workpieces.

Even more important, tool life is greatly reduced at high cutting speeds unless coated carbide or other modern tool materials are used, and these also have practical speed limits. Tool life is decreased most at high speeds, although some decrease in tool life occurs when feed or depth of cut is increased. This stands to reason, because more material will be removed in less time. It becomes a choice then, between longer tool life or increased stock removal. Since productivity generally outweighs tool costs, the most practical cutting conditions are usually those which first, are most productive, and second, will achieve reasonable tool life.

Finishing Cuts: When taking finishing cuts, feed rate and depth of cut are of minor concern. The feed rate cannot exceed that which is necessary to achieve the required surface finish and the depth of cut will be light. However, the rule about speed will still apply. The speeds will generally be higher for finish cuts, but they must still be within the operating speed of the tool material.

Tool life is of greater concern for finish cuts. It is often better to strive for greater tool life at the expense of material removed per minute. If tool wear can be minimized, especially on a long cut, greater accuracy can be achieved, and matching cuts which result from tool changes, can be avoided.

One way to minimize tool wear during finishing cuts is to use the maximum feed rate that will still produce the required surface finish. The less time the tool spends on the cut, the less tool wear can occur. Another way to minimize tool wear during a long finishing cut is to reduce the speed slightly. Coolant, spray mist, or air flow, will also extend tool life because it reduces the heat of the tool.

4.6 Hard Turning

As the hardness of the workpiece increases, its machinability decreases accordingly and tool wear and fracture, as well as surface finish and integrity, can become a significant problem. There are several other mechanical processes and nonmechanical methods of removing material economically from hard or hardened metals. However, it is still possible to apply traditional cutting processes to hard metals and alloys by selecting an appropriate

tool material and machine tools with high stiffness and high speed spindles.

One common example is finish machining of heat-treated steel machine and automotive components using polycrystalline cubic boron nitride (PCBN) cutting tools. This process produces machined parts with good dimensional accuracy, surface finish, and surface integrity. It can compete successfully with grinding the same components, from both technical and economic aspects. According to some calculations grinding is over ten times more costly than hard turning.

Advanced cutting tool materials such as polycrystalline cubic boron nitride (PCBN) and ceramics (discussed in Chapter 1 - Cutting Tool Materials), have made the turning of hardened steel a cost effective alternative to grinding. Many machine shops have retired their cylindrical grinders in favor of less expensive and more versatile CNC lathes.

Compared to grinding, hard turning:

- permits faster metal removal rates, which means shorter cycle times.
- eliminates the need for coolant (dry vs. wet machining will be discussed later).
- shortens set up time and permits multiple operations to be performed in one chucking.

Today's sophisticated CNC lathes offer accuracy and surface finishes comparable to what grinders provide.

Hard turning requires much less energy than grinding, thermal and other damage to the workpiece is less likely to occur, cutting fluids may not be necessary and the machine tools are less

expensive. In addition, finishing the part while still chucked in the lathe eliminates the need for material handling and setting the part in the grinder. However, work holding devices for large and slender workpieces for hard turning can present problems, since the cutting forces are higher than in grinding.

Furthermore, tool wear and its control can be a significant problem as compared to the automatic dressing of grinding wheels. It is thus evident that the competitive position of hard turning versus grinding must be elevated individually for each application and in terms of product surface integrity, quality, and overall economics.

4.6.1 Dry vs. Wet Machining

Just two decades ago, cutting fluids accounted for less than 3 percent of the cost of most machining processes. Fluids were so cheap that few machine shops gave them much thought. Times have changed.

Today, cutting fluids account for up to 15 percent of a shop's production costs, and machine shop owners constantly worry about fluids.

Cutting fluids, especially those containing oil, have become a huge liability. Not only does the Environmental Protection Agency (EPA) regulate the disposal of such mixtures, but many states and localities also have classified them as hazardous wastes, and impose even stricter controls if they contain oil and certain alloys.

Because many high-speed machining operations and fluid nozzles create air-

borne mists, governmental bodies also limit the amount of cutting fluid mist allowed into the air. The EPA has proposed even stricter standards for controlling such airborne particulate, and the Occupational Safety and Health Administration (OSHA) is considering and advisory committee's recommendation to lower the permissible exposure limit to fluid mist.

The cost of maintenance, record keeping and compliance with current and proposed regulations is rapidly raising the price of cutting fluids. Consequently, many machine shops are considering eliminating the costs and headaches associated with cutting fluids altogether by cutting dry.

The decision to cut wet or dry must be made on a case-by-case basis. A lubricious fluid often will prove beneficial in low-speed jobs, hard-to-machine materials, difficult applications, and when surface finish requirements are demanding. A fluid with high cooling capacity can enhance performance in high-speed jobs, easy-to-machine materials, simple operations, and jobs prone to edge-buildup problems or having tight dimensional tolerances.

Many tough times though, the extra performance capabilities that a cutting fluid offers is not worth the extra expense incurred, and in a growing number of applications, cutting fluids are simply unnecessary or downright detrimental. Modern cutting tools can run hotter than their predecessors and sometimes compressed air can be used to carry hot chips away from the cutting zone.