CHAPTER **14**

The Lathe

Chapter Outline

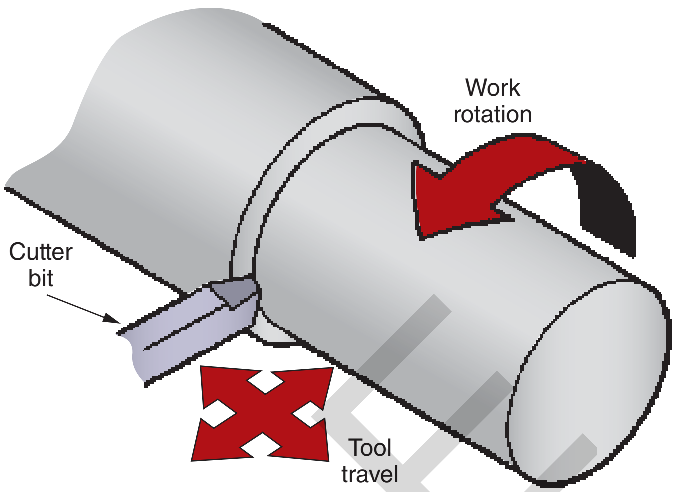
Learning Objectives

After studying this chapter, you will be able to:
- Describe how a lathe operates.
- Identify the various parts of a lathe.
- Safely set up and operate a lathe using various work-holding devices.
- Calculate correct cutting speeds and feeds for lathe operations.
- Perform basic machining operations on a lathe, including facing, turning, parting, and grooving.
- Explain the information that can be obtained from studying the chips produced during the machining process.

Technical Terms

chipbreaker
compound rest
cross-slide
cutting speed
depth of cut
facing
feed rate
grooving
headstock
indexable insert cutting tool
lathe center
lathe dog
parting
plain turning
roughing cut
single-point cutting tool
spindle
swing
tailstock
tool post
ways

The lathe operates on the principle of rotating the work against the edge of a cutting tool, **Figure 14-1**. It is one of the oldest and most important machine tools. The cutting tool is controllable and can be moved lengthwise on the lathe bed and across the revolving work at any desired angle. See **Figure 14-2**.



Goodheart-Willcox Publisher

Figure 14-2. Operating principle of the lathe. The cutting tool is fed into the revolving work.

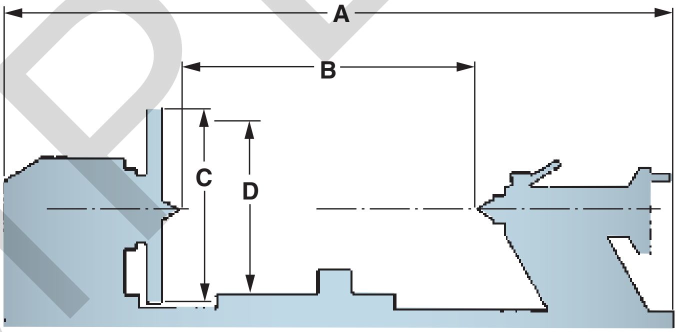
14.1 Lathe Size

Lathe size is determined by the swing and the length of the bed, **Figure 14-3**. The ***swing*** is the largest diameter that can be turned over the ***ways*** (the flat or V-shaped bearing surface that aligns and guides the movable part of the machine). Bed length is the entire length of the ways.

Bed length must not be mistaken for the maximum length of the work that can be turned between centers. The longest piece that can be turned is equal to the length of the bed minus the distance taken up by the headstock and tailstock. Refer to measurement B in **Figure 14-3**.

As an example, consider the capacity and clearance of a 13″ × 6′ (325 mm × 1800 mm) lathe:

Swing over bed: 13″ (325 mm)
Swing over cross-slide: 8 3/4″ (218 mm)
Bed length: 72″ (1800 mm)
Distance between centers: 50″ (1240 mm)



Goodheart-Willcox Publisher

Figure 14-3. Lathe measurements. A—Length of bed. B—Distance between centers. C—Diameter of work that can be turned over the ways. D—Diameter of work that can be turned over the cross-slide.

Jet Equipment & Tools

Figure 14-1. A basic metal-cutting lathe. All controls are operated manually. Most machinists begin their training on this type of lathe.

14.2 Major Parts of a Lathe

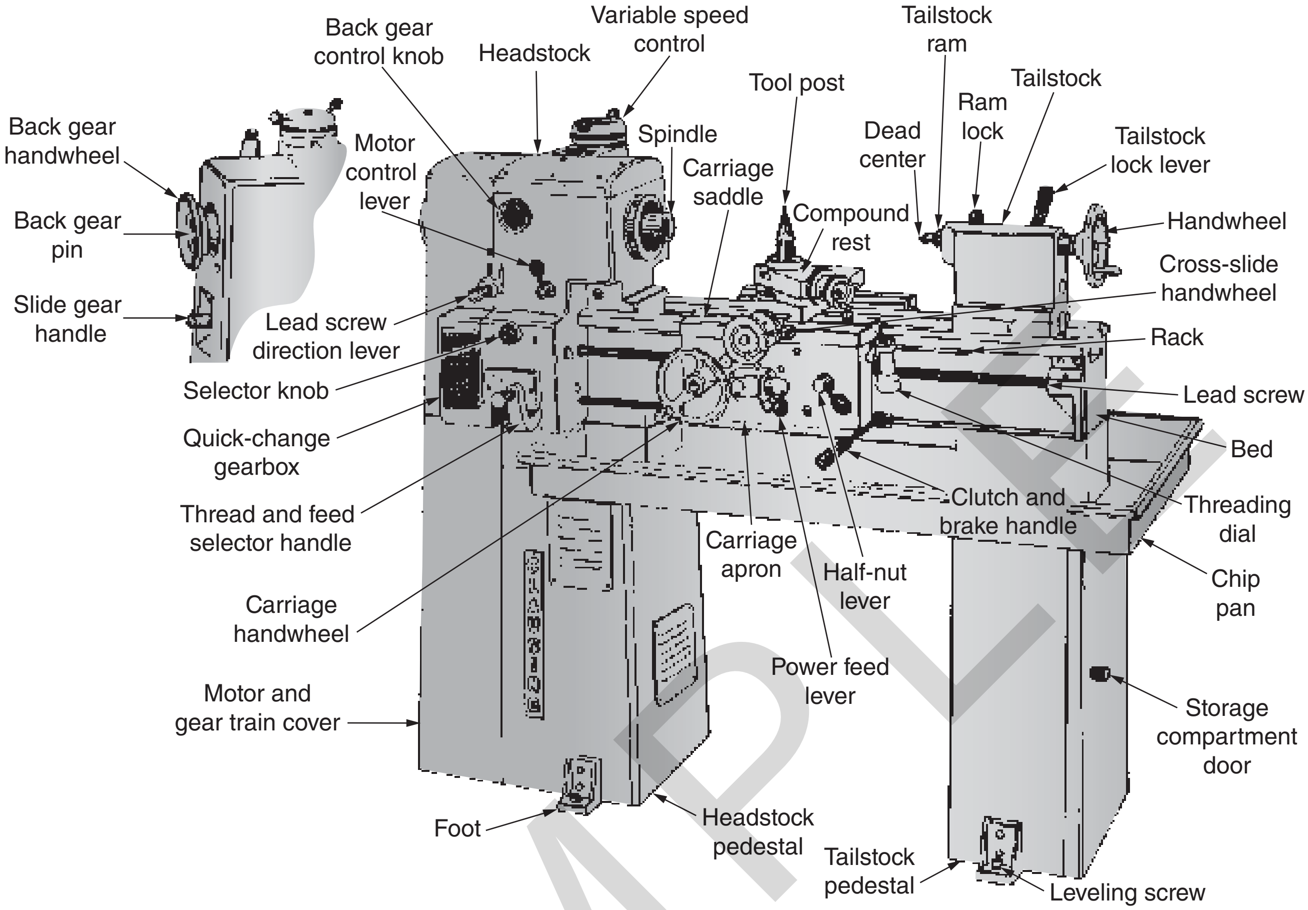
The chief function of any lathe, no matter how complex it may appear to be, is to rotate the work against a controllable cutting tool. Each of the lathe parts in **Figure 14-4** can be assigned to one of the following three functions:

- Driving the lathe.
- Holding and rotating the work.
- Holding, moving, and guiding the cutting tool.

14.2.1 Driving the Lathe

Power is transmitted to the drive mechanisms by a belt drive or gear train. Spindle speed can be varied by any of the following:

- Shifting to a different gear ratio, **Figure 14-5**.
- Adjusting a split pulley to another position, **Figure 14-6**.



Clausing Industrial, Inc.

Figure 14-4. The engine lathe and its major parts.

- Moving the drive belt to another pulley ratio (seldom used today).
- Controlling the speed hydraulically.

Slower speeds with greater power are obtained on some machines by engaging a back gear. To avoid damaging the lathe's drive system, do not engage the back gear while the spindle is rotating.

14.2.2 Holding and Rotating the Work

The ***headstock*** contains the spindle to which the various work-holding attachments are fitted, **Figure 14-7**. The ***spindle*** revolves in heavy-duty bearings and is rotated by belts, gears, or a combination of the two. The front of the hollow spindle is tapered internally to receive tools and attachments with taper shanks, **Figure 14-8**. The hole through the spindle permits long stock to be turned without dangerous overhang. It also allows use of a knockout bar to remove taper-shank tools.

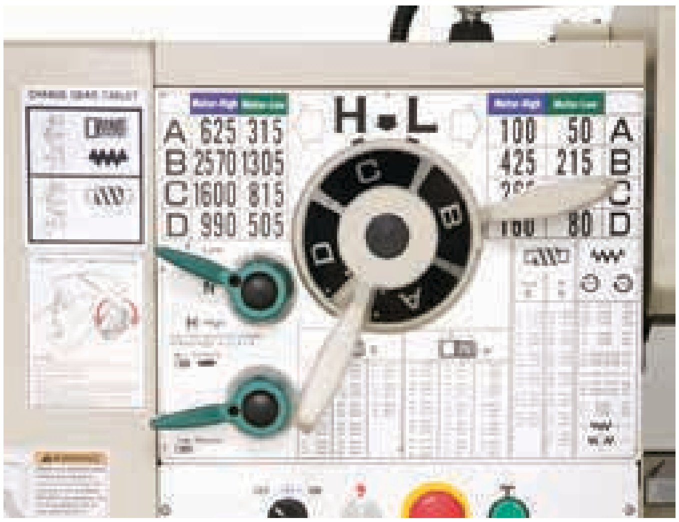
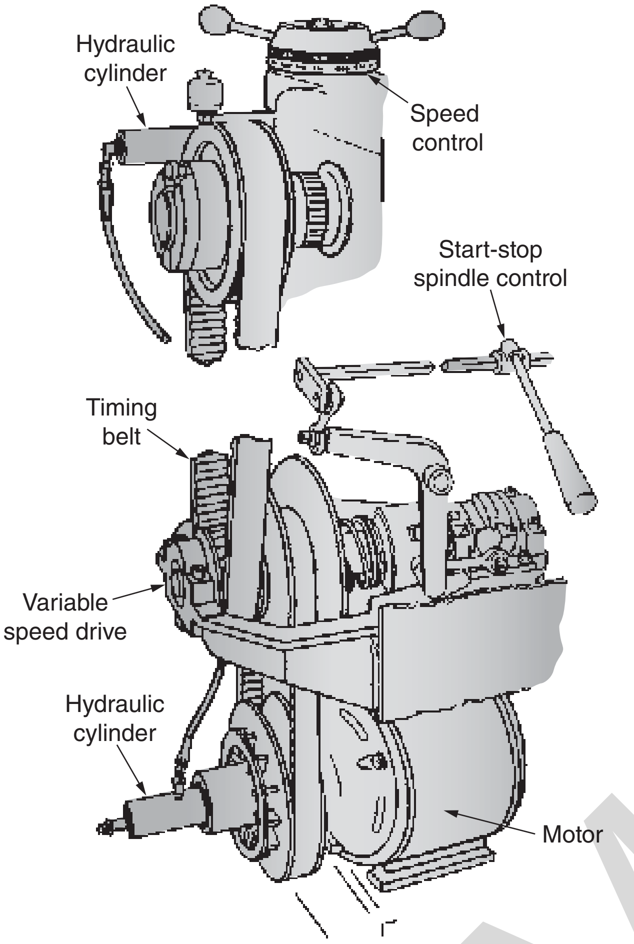


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 14-5. Spindle speed control. Speed is increased or decreased by shifting to different gear ratios. On this machine, the desired speed is dialed in.

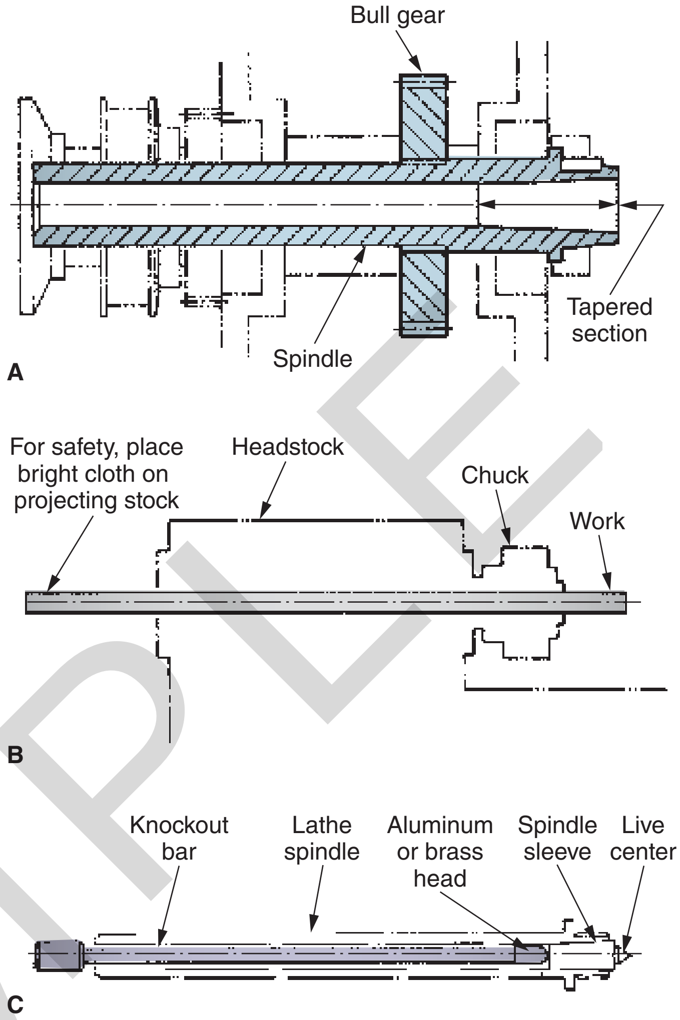


Clausing Industrial, Inc.

Figure 14-6. This split pulley is hydraulically actuated from the top of the machine by a speed control. A split pulley is used to control spindle speeds on many lathes.

Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 14-7. The headstock is the driving end of the lathe.



Goodheart-Willcox Publisher

Figure 14-8. Lathe spindle. A—Hollow spindle construction allows long stock to be turned without dangerous overhang. B—To prevent accidents that could cause injury, a bright flag should be tied to the portion of stock that projects from the rear of the spindle. C—A knockout bar is used to tap tapered shank lathe accessories out of the spindle.

On the front end, a spindle may be threaded externally or fitted with a tapered spindle nose to receive work-holding attachments. See **Figure 14-9**. A threaded spindle nose is seldom used on modern lathes. It permits mounting an attachment by screwing it directly onto the threads until it seats on the spindle flange.

The cam-lock spindle nose has a short taper that fits into a tapered recess on the back of the work-holding attachment. A series of cam locking studs, located on the back of the attachment, are inserted into holes in the spindle nose. The studs are locked by tightening the cams located around the spindle nose.

A long taper key spindle nose has a protruding long taper and key that fit into a corresponding taper and keyway in the back of the work-holding device. To mount a work-holding device (a chuck or faceplate), the spindle is rotated until the key is on top. The keyway in the back of the work-holding device is slid over the key to support the device until the threaded spindle collar can be engaged with the threaded section of the device and tightened.

SAFETY NOTE

Attachment points on the spindle nose and work-holding attachment must be cleaned carefully before mounting the device.

Work is held in the lathe by a chuck, faceplate, or collet, or by mounting it between centers. These attachments will be described in detail later in this chapter.

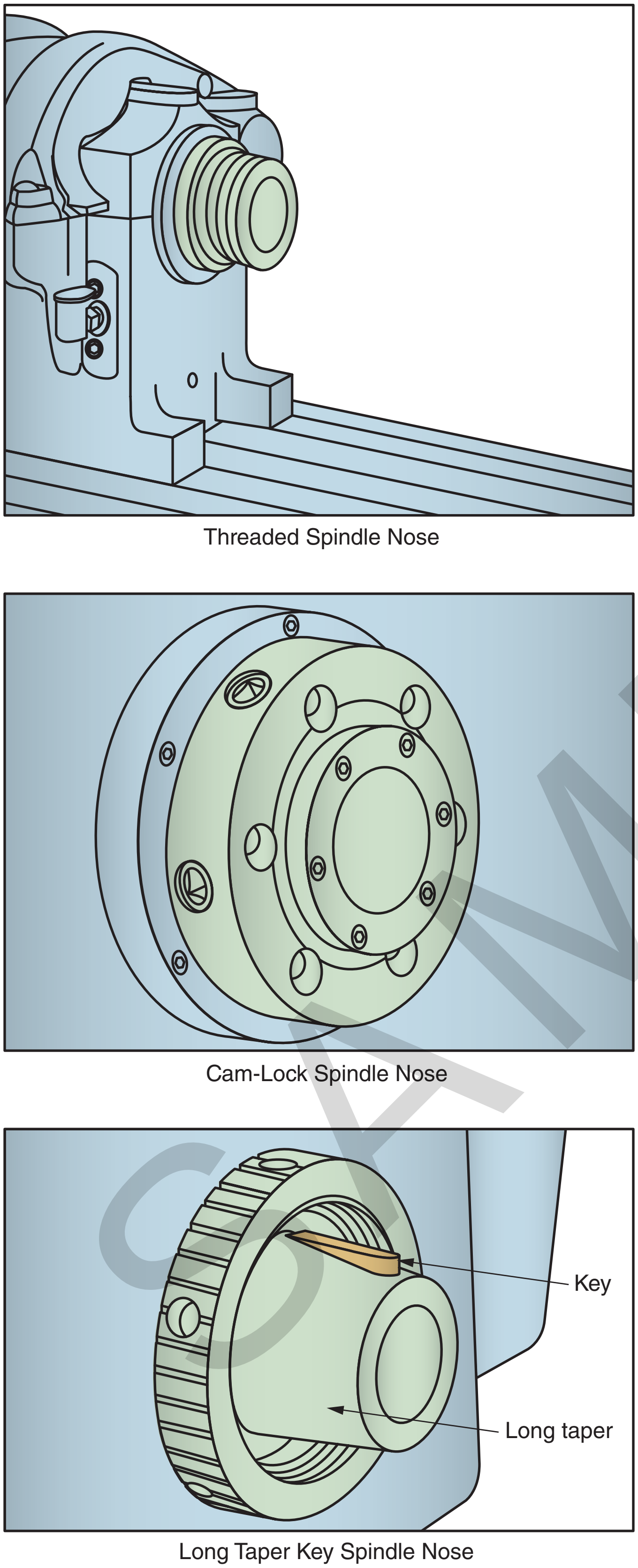
The outer end of the work is often supported by the lathe's ***tailstock***, **Figure 14-10**. The tailstock can be adjusted along the ways to accommodate different lengths of work.

The tailstock is used to mount the lathe center. It can also be fitted with tools for drilling, reaming, and threading and can be offset for taper turning.

The tailstock is locked onto the ways by tightening a clamp bolt nut or binding lever. The tailstock spindle is positioned by rotating the handwheel. It can be locked in position by tightening a binding lever.

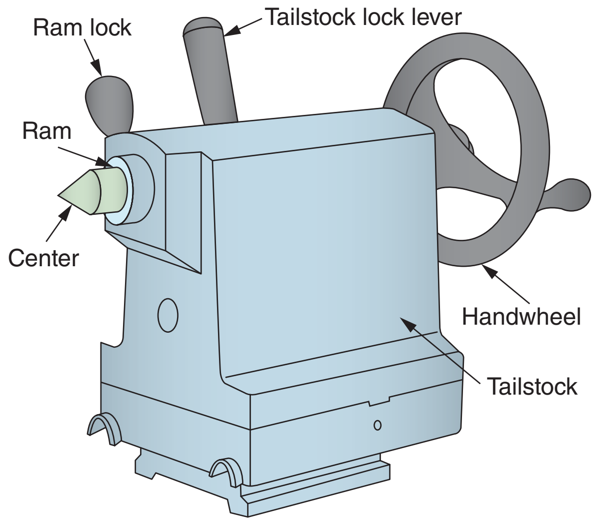
14.2.3 Holding, Moving, and Guiding the Cutting Tool

The bed, **Figure 14-11**, is the foundation of a lathe. All other parts are fitted to it. Ways are integral with the bed. The V-shaped rails maintain precise alignment of the headstock and tailstock and guide the travel of the carriage.



Goodheart-Willcox Publisher

Figure 14-9. Types of spindle noses. The threaded spindle nose is seldom used today.



Goodheart-Willcox Publisher

Figure 14-10. Parts of a tailstock.

The carriage, **Figure 14-12**, controls and supports the cutting tool. It is composed of the following parts:

- The saddle is fitted to the ways and slides along them.
- The apron contains a drive mechanism to move the carriage along the ways, using hand or power feed.
- The ***cross-slide*** permits transverse tool movement (movement toward or away from the operator, at a right angle to the axis of the lathe).
- The ***compound rest*** permits angular tool movement.
- The ***tool post*** is used to mount the cutting tool.

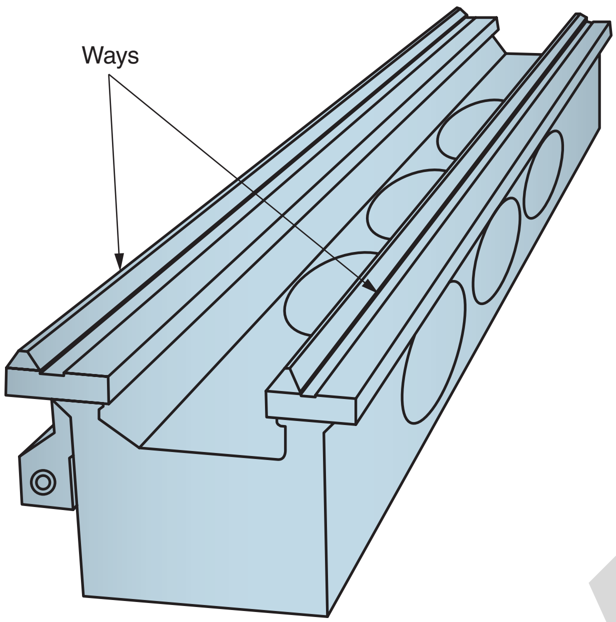
Both the cross-slide and the compound rest sit on dovetailed slide bearings for smooth and precise movement. Over time the slides wear down. This can have a negative effect on the accuracy of the slide. Small, tapered pieces of iron or steel known as *gibs* are used to adjust the slide and compensate for this wear. Gibs are adjusted with a screw. Tightening the screw pushes the gib forward, reducing the clearance within the slide.

Power is transmitted to the carriage through the feed mechanism, which is located at the left (headstock) end of the lathe. Power is transmitted through a train of gears to the quick-change gearbox. This device, **Figure 14-13**, regulates the amount of tool travel per revolution of the spindle. The gear train also contains gears for reversing tool travel.

The quick-change gearbox is located between the spindle and the lead screw. It contains gears of various ratios that make it possible to machine different pitches of screw threads without physically removing and replacing gears. Longitudinal (back-and-forth) travel and cross (in-and-out) travel are controlled in the same manner.

An index plate provides instructions for setting the lathe shift levers for various thread cutting and feed combinations. It is located on the face of the gearbox. Index plates often show the lever positions for both inch and metric feeds and thread pitches.

The lead screw transmits power to the carriage through a gearing and clutch arrangement in the carriage apron, **Figure 14-14**. Feed change levers on the apron control power longitudinal feed and power cross-feed, **Figure 14-15**.



Goodheart-Willcox Publisher

Figure 14-11. The bed is the foundation of the lathe. The ways maintain alignment of the headstock and tailstock.

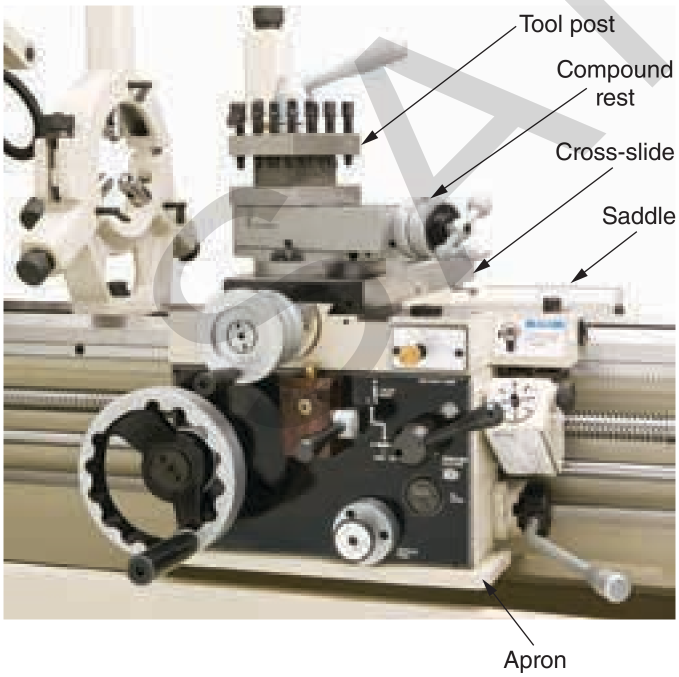


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 14-12. The V-shaped ways guide the carriage. The cutting tool is mounted on the carriage.

Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 14-13. A quick-change gearbox for cutting both inch and metric size threads.

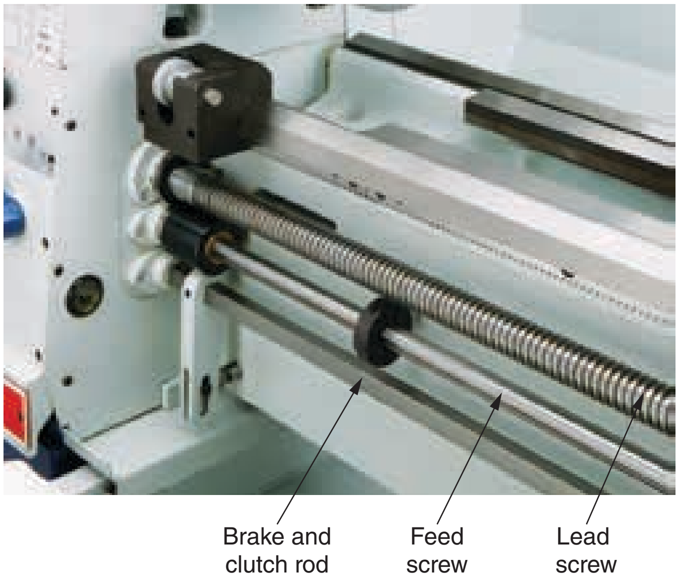
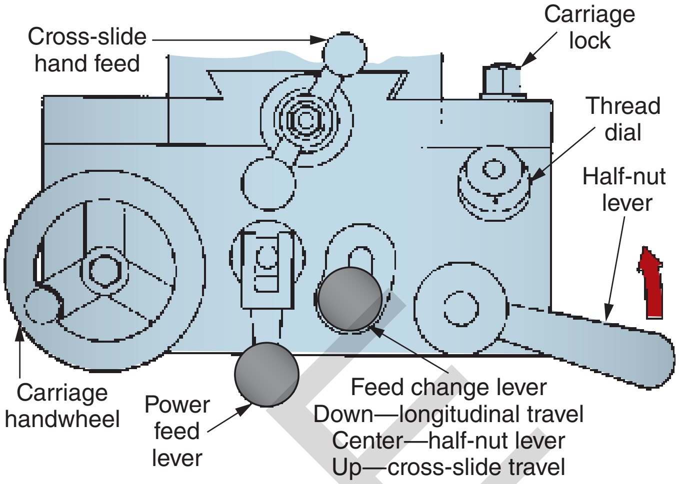


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 14-14. The lead screw.



Goodheart-Willcox Publisher

Figure 14-15. Power feed functions are controlled by the feed change levers on the apron. The half-nut lever is engaged only for thread cutting, sometimes called *thread chasing*.

When the feed change lever is placed in neutral, the half-nuts may be engaged for thread cutting. The gear arrangement makes it possible to engage the power feed and half-nuts simultaneously. The half-nuts are engaged only for thread cutting. They are not used as an automatic feed for regular turning.

14.3 Work-Holding Attachments

One of the reasons the lathe is such a versatile machine tool is the great variety of ways that work may be mounted in or on it. The most common way is to mount the work so that it revolves, permitting the cutting tool to move across the work's surface. Large or odd-shaped pieces are sometimes mounted on the carriage and machined with a cutting tool that is mounted in the rotating spindle.

Most work is machined while supported by one of the following methods, as shown in **Figure 14-16**:

- Between centers using a faceplate and lathe dog.
- Held in one of the three types of chucks.
  - 3-jaw universal chuck.
  - 4-jaw independent chuck.
  - Jacobs chuck.
- Held in a collet.
- Bolted to the faceplate.

14.3.1 Work-Holding between Centers

Considerable lathe work is done with the workpiece supported between centers. Because work-holding between centers requires the work to be supported at both ends, only external machining operations can be performed.

A ***lathe center***, or center, is a pointed work-holding device used to accurately align the workpiece along an axis, **Figure 14-17**. Heavy-duty ball bearings allow live centers to

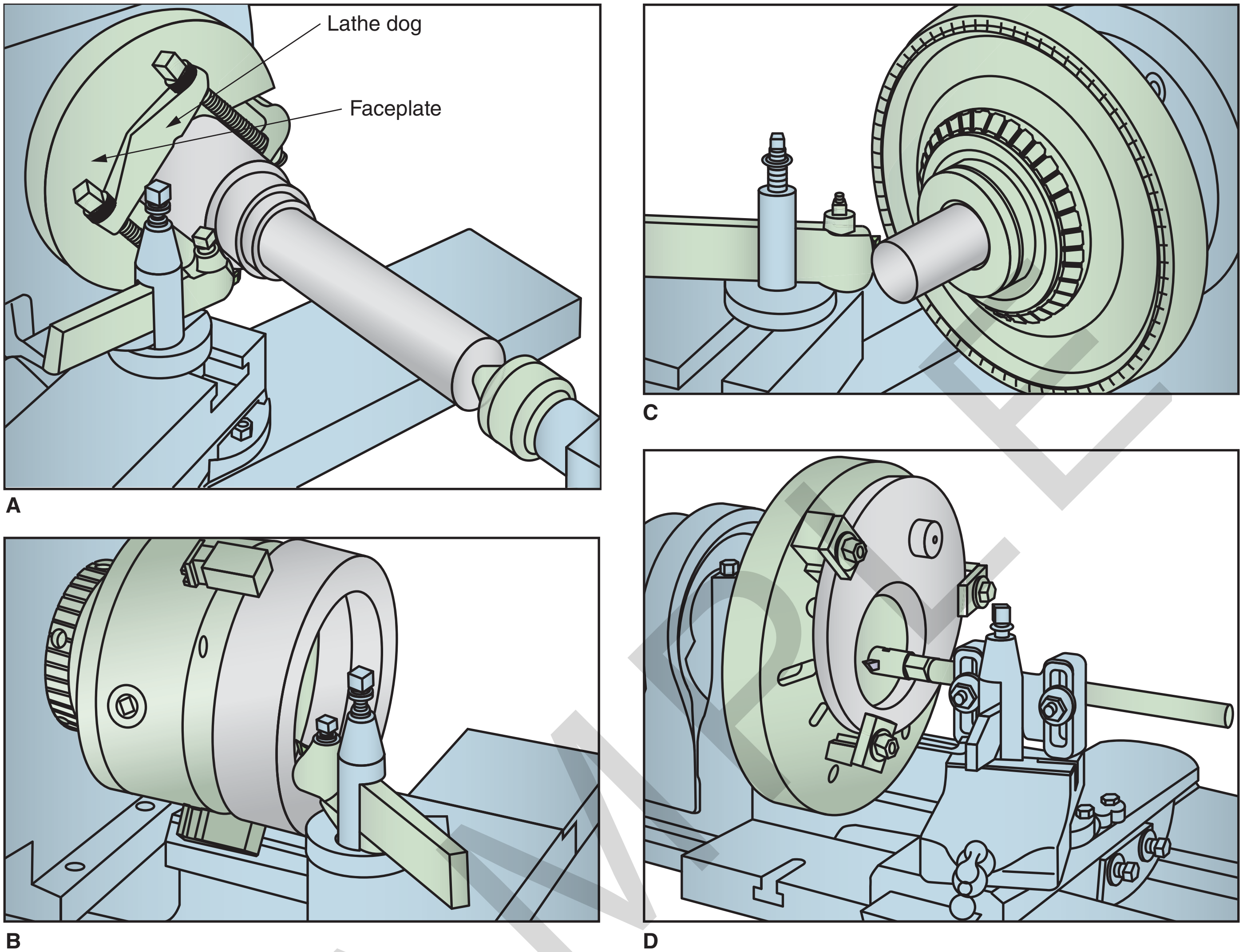
WORKPLACE SKILLS

Attitude on the Job

Your attitude can often determine your success in your job. Your *attitude* is your outlook on life. It is reflected by how you react to the events and people around you. A smile and courteous behavior can make customers and fellow employees feel good about themselves and you. Customers prefer to do business in friendly environments. Being friendly may take some effort on your part, but it does pay off.

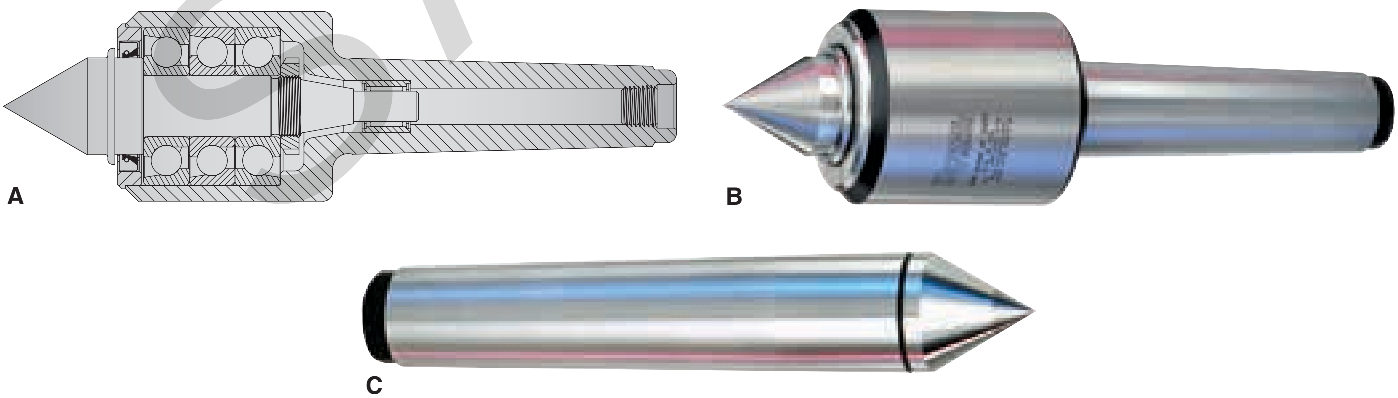
Enthusiasm spreads easily from one person to another. Usually, *enthusiasm* means a person enjoys what he or she is doing. In an office setting, enthusiasm builds a team spirit for working together.

People who do a good job feel pride in their work. They feel a sense of accomplishment and a desire to achieve more. This attitude can inspire others as well.



Goodheart-Willcox Publisher

Figure 14-16. Work-holding methods. A—Work being machined between centers. B—Work held in a chuck for machining. C—Work being machined while held in a collet. D—Work bolted to a faceplate for machining.



Royal Products, Division of Curran Manufacturing Corporation

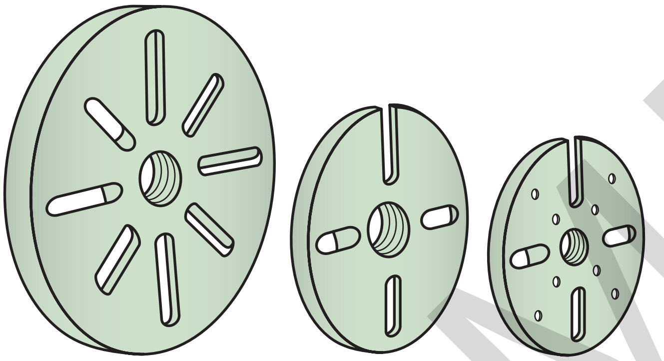
Figure 14-17. Lathe centers. A—Sectional view shows construction of a heavy-duty ball bearing live center. B—High-precision quad-bearing live center. C—A dead center does not rotate. The carbide tip provides great wear resistance.

rotate freely with the workpiece. Dead centers do not rotate. When supporting the workpiece between centers, a faceplate, **Figure 14-18**, is often attached to the spindle nose. A sleeve and dead center are inserted into the spindle opening, **Figure 14-19**.

Either a live or dead center is fitted into the tailstock spindle to support one end of the work. The ends of the stock are drilled to fit over the center points.

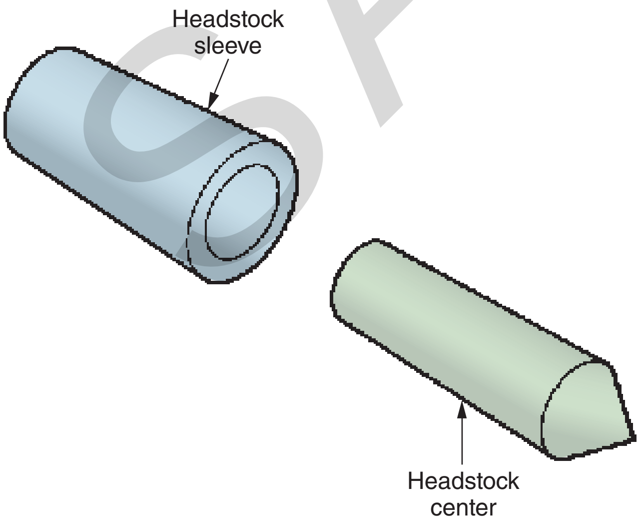
A ***lathe dog*** is a device clamped to one end of the material to drive the workpiece. The three types of lathe dogs, as shown in **Figure 14-20**, are as follows:

- The bent-tail standard dog has an exposed setscrew.
- The bent-tail safety dog has a recessed setscrew. This type of dog is usually preferred over the standard lathe dog.
- The clamp-type dog is used for turning square or rectangular work.



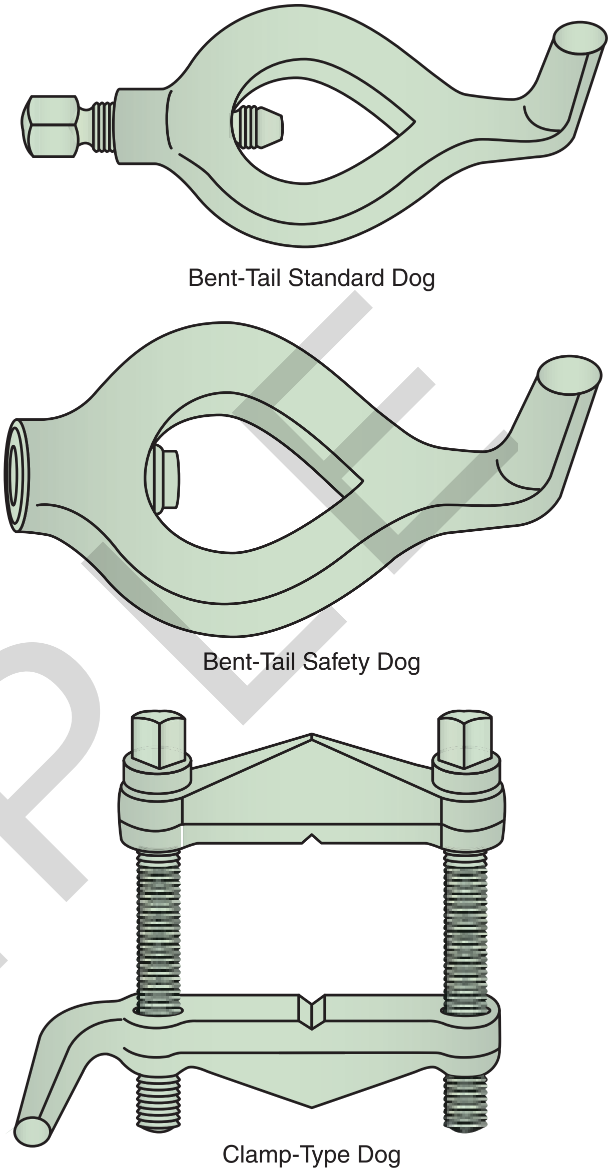
Goodheart-Willcox Publisher

Figure 14-18. Lathe faceplates come in various sizes.



Goodheart-Willcox Publisher

Figure 14-19. Sleeve and headstock center.



Goodheart-Willcox Publisher

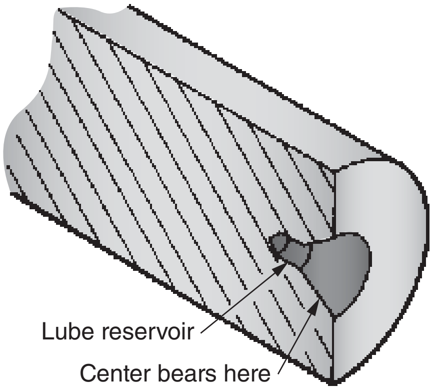
Figure 14-20. Lathe dogs.

Drilling Center Holes

Before work can be mounted between centers, it is necessary to locate and drill center holes in each end of the stock, **Figure 14-21**. Several methods for locating the center of round stock are shown in **Figure 14-22**.

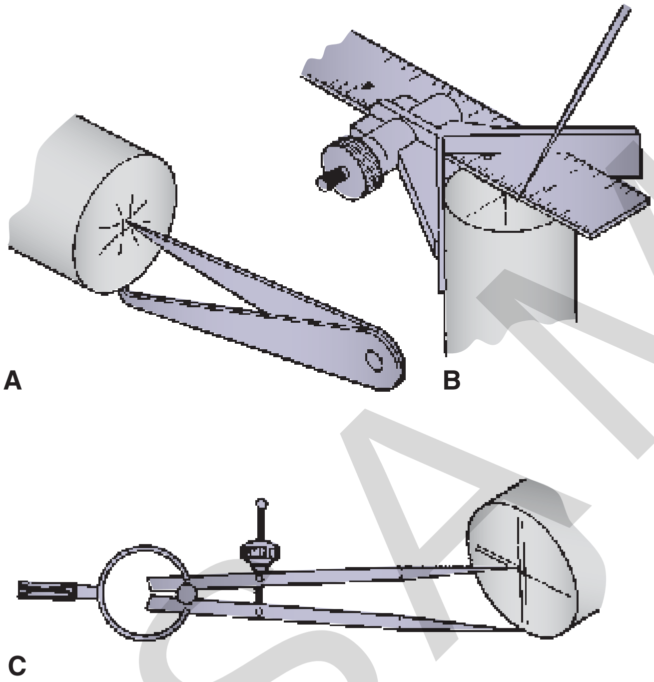
Center holes are usually drilled with a center drill, **Figure 14-23**. The drill angle is identical to that of the center point. The straight drill provides clearance for the center point and serves as a reservoir for a lubricant. The chart provides the information needed to select the correct size center drill.

The center holes can be drilled on the lathe with the work centered in the chuck, on the lathe with the center drill held in the headstock, or on a drill press. Some work can be held in a lathe chuck for center drilling. The work is centered



Goodheart-Willcox Publisher

Figure 14-21. The tailstock center rides in the drilled and countersunk center hole. If a dead center is used, a supply of lubricant is placed in the reservoir. The lubricant will expand and lubricate the center as the metals heat up.

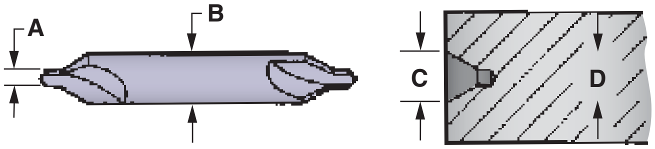


Goodheart-Willcox Publisher

Figure 14-22. Several ways to locate the center of round stock. A—With a hermaphrodite caliper. B—With center head and rule of a combination set (recommended method). C—With dividers.

in a lathe chuck mounted in the headstock. The center drill is fitted in a Jacobs chuck mounted to the tailstock.

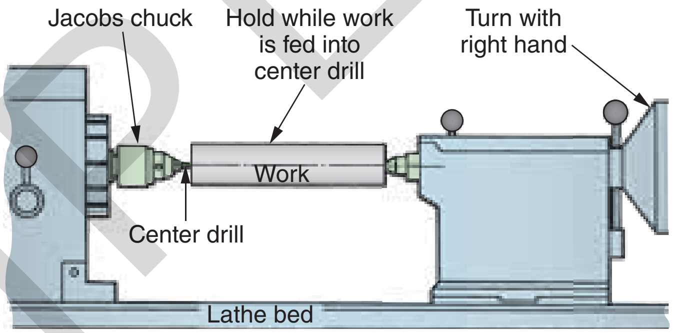
Center holes can be drilled in large stock by mounting a Jacobs chuck in the headstock. Locate the center point of each end and center punch. Support one end on tail center and feed the other end into the center drill mounted in the Jacobs chuck. Repeat the operation on the second end. See **Figure 14-24**.



Center Drill Number	A	B	C	D
1	1/16	13/64	1/8	3/16 to 5/16
2	3/32	3/16	3/16	3/8 to 1
3	1/8	1/4	1/4	1 1/4 to 2
4	5/32	7/16	5/16	2 1/4 to 4

Goodheart-Willcox Publisher

Figure 14-23. This size chart contains the information needed to select the correct size center drill. A center drill makes the hole and countersinks it in one operation.



Goodheart-Willcox Publisher

Figure 14-24. Center holes can be drilled in large stock by mounting a Jacobs chuck in the headstock.

When using a drill press, mount the work in a V-block for support. The center holes should be drilled deep enough to provide adequate support, **Figure 14-25**.

Checking Center Alignment

Accurate turning between centers requires centers that run true and are in precise alignment. Since the work must be reversed to machine its entire length, care must be taken to make the live center run true. If the live center does not run true, the diameters will be eccentric (not aligned on the same centerline), **Figure 14-26**. This can be prevented by truing the live center. If the center is not hardened, a light truing cut can be made. A tool post grinder is needed to true a hardened center.

Approximate alignment can be determined by checking centers visually. Bring their points together, or check the witness lines on the base of the tailstock for alignment. See **Figure 14-27**.

A more precise method for checking alignment is needed if close tolerance work is to be done. Three such methods are as follows:

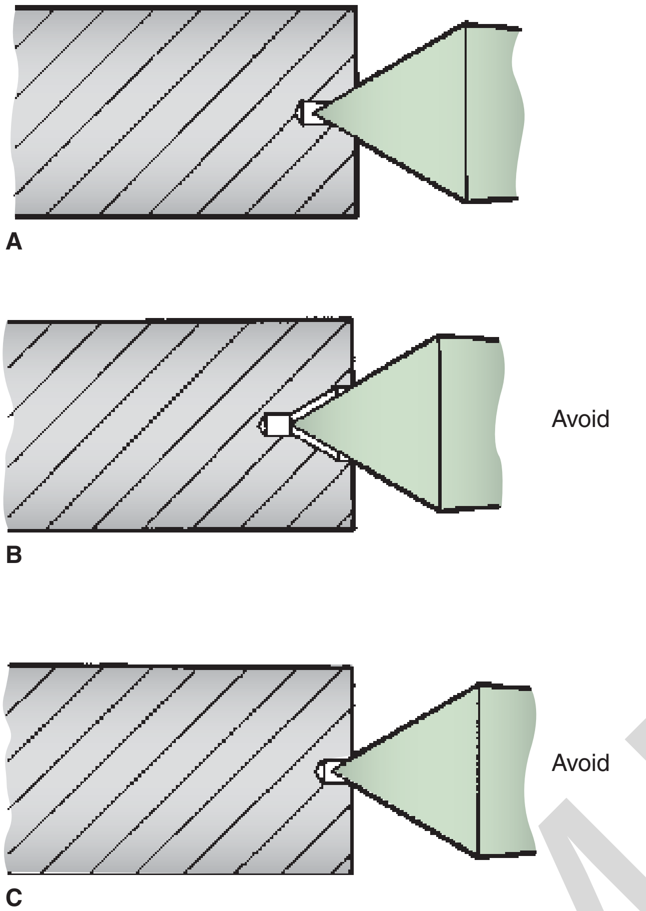


Figure 14-25. Correctly and incorrectly drilled center holes. A—Properly drilled center hole. B—Hole drilled too deep. C—Hole not drilled deep enough. Does not provide enough support. If used with a dead center, the center point will burn off.

- Make a light trial cut across a few inches of the material. Check the diameter at each end with a micrometer, **Figure 14-28**. The centers are aligned if the readings are identical.
- Use a steel test bar and dial indicator, **Figure 14-29**. Mount the test bar between centers and position the dial indicator in the tool post at right angles to the work. Move the indicator contact point against the test bar until a reading is shown. Move the indicator along the test bar. If the readings remain constant, the centers are aligned.
- Machine a section of scrap, **Figure 14-30**. Set the cross-feed screw to make a light cut at the right end of the piece. With the same tool setting, move it to the left and continue the cut. Identical micrometer readings indicate center alignment.

Adjusting screws (one on each side at the base of the tailstock) are used to align the centers if checks indicate they are not correctly aligned. Make adjustments gradually. See **Figure 14-31**.

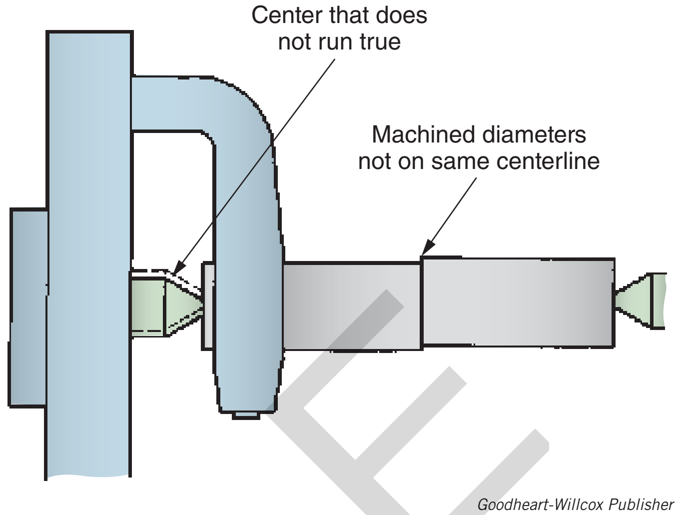


Figure 14-26. The workpiece must be reversed in the lathe dog so it can be machined for its entire length. If the live center does not run true, eccentric diameters will result.

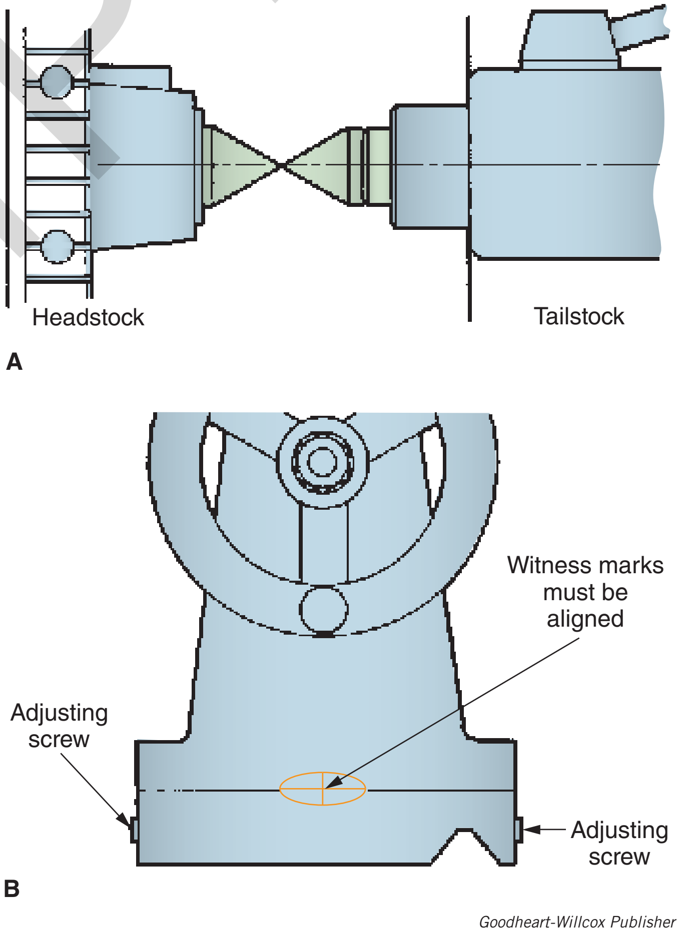
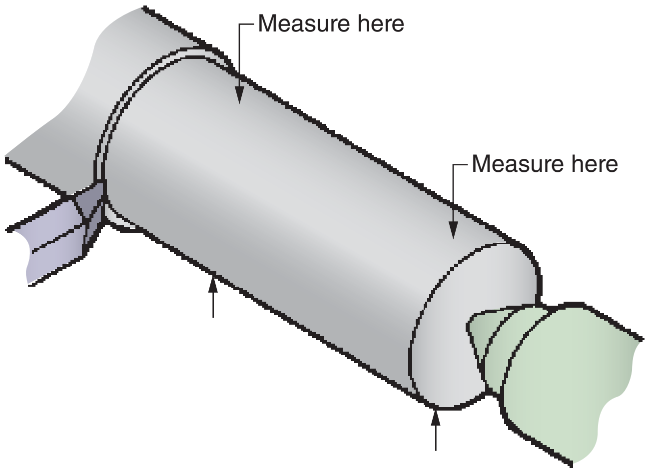
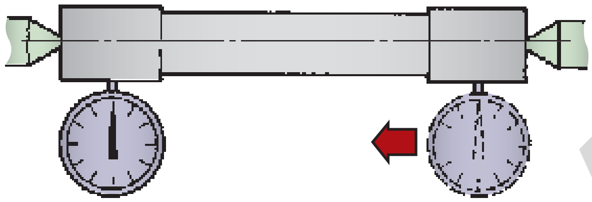


Figure 14-27. Checking center alignment. A—Checking alignment by bringing center points together. This view is looking down on the top of the centers. B—Alignment can be determined by checking witness lines on the base of the tailstock.



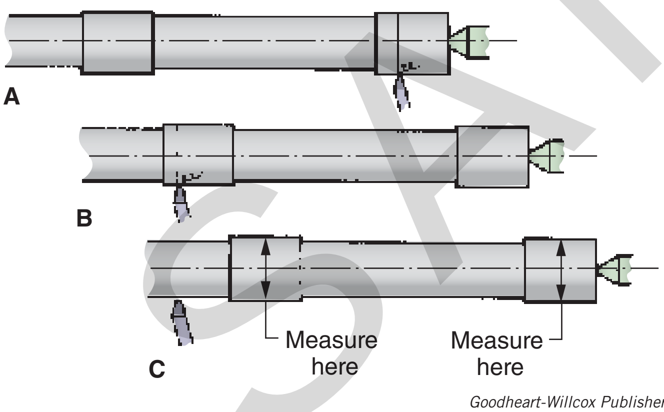
Goodheart-Willcox Publisher

Figure 14-28. Make a light cut on the stock and measure the diameter at two points to check alignment. Measurements must be equal.



Goodheart-Willcox Publisher

Figure 14-29. Using a test bar and dial indicator to check center alignment.



Goodheart-Willcox Publisher

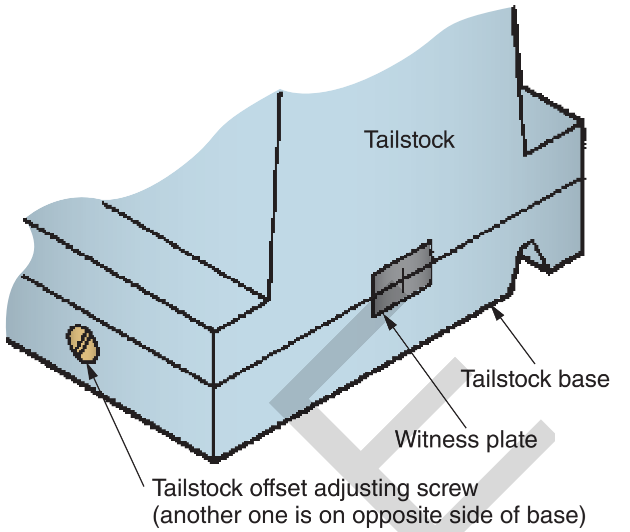
Figure 14-30. Checking for center alignment on a scrap piece. A—Machine two shoulders on the test piece. B—Keep the same tool setting and make a cut on both shoulders. C—Measure the resulting diameters using a micrometer.

Mounting Work between Centers

Clamp a dog to one end of the work. Place a lubricant (graphite and oil or a commercial center lubricant) in the center hole on the other end. Mount the piece on the centers and adjust the tailstock spindle until the work is snug. If the work is too loose, it will “clatter.” If adjusted too tightly, it will score or burn the center point.

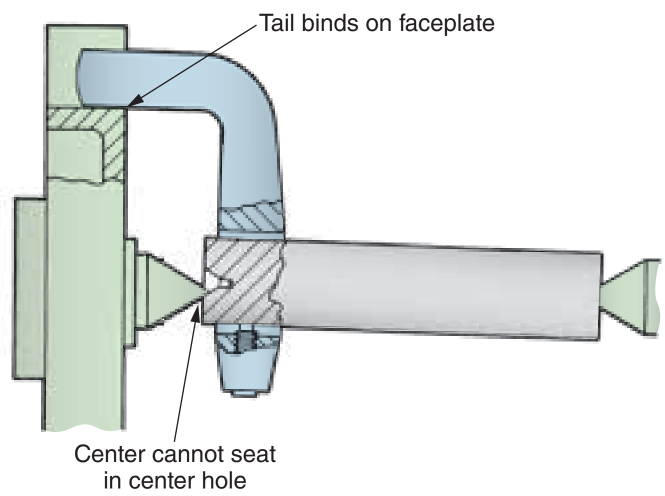
Check the adjustment from time to time because heat generated by the machining process will cause the work to expand. Using a live center, instead of a dead center, will reduce or eliminate many of the problems involved in working between centers.

Check to see if the dog tail binds on the faceplate slot, **Figure 14-32**. This can cause the work to be pulled off center. When machined, this will produce a surface that is not



Goodheart-Willcox Publisher

Figure 14-31. Adjusting screws are located on both sides of tailstock base. Their primary purpose is to shift the tailstock for taper turning.



Goodheart-Willcox Publisher

Figure 14-32. The diameter of the turned surface will not be concentric with the center holes if center hole is not seated properly on the live center. A binding lathe dog is a common cause of this problem.

concentric with the center hole. If binding is occurring, use a different faceplate.

14.3.2 Using Lathe Chucks

A chuck is another device for holding work in a lathe. Chucking is the most rapid method of mounting work, and it is widely preferred for that reason. Other operations, such as drilling, boring, reaming, and internal threading, can be done while the work is held in a chuck. Additional support can be obtained for the piece by supporting the free end with the tailstock center.

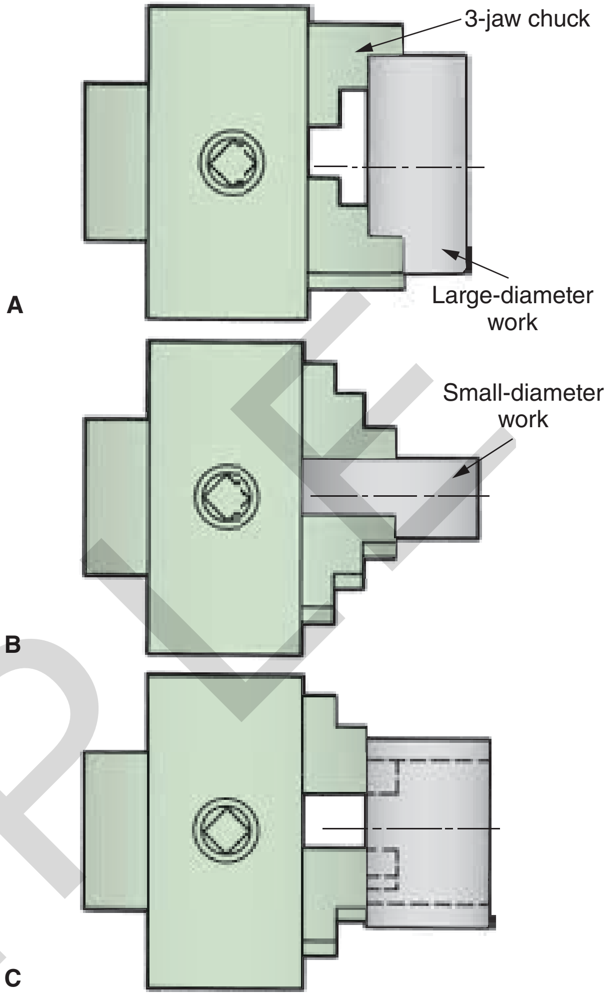
Three-Jaw Universal Chuck

The 3-jaw universal chuck is designed so that all jaws operate at the same time, **Figure 14-33**. It automatically centers round or hexagonal shaped stock.

Two sets of jaws are supplied with each universal chuck. One set is used to hold large-diameter work. The other set is used for small-diameter work, **Figure 14-34**. The jaws in each set are numbered 1, 2, and 3, as are the slots in which they are fitted. To center the work accurately, the jaw number must correspond with the slot number. Sets of jaws are made for a specific chuck and are not interchangeable with other chucks. Make sure the chuck and jaws have the same serial number!

Installing Chuck Jaws

Before installing jaws, clean the jaws, jaw slots, and scroll (spiral thread in the jaw slots). Turn the scroll until the first thread does not quite show in jaw slot 1. Slide the matching jaw into the slot as far as it will go. Then turn the scroll until the spiral engages with the first tooth on the bottom of the jaw. Repeat the operation at slots 2 and 3, making sure the proper jaws are inserted.



Goodheart-Willcox Publisher

Figure 14-34. Chuck jaws. A—One of the sets of jaws supplied with a 3-jaw universal chuck is used to mount large-diameter work. B—Holding work using the set of jaws supplied for smaller workpieces. C—Another method of mounting work in the chuck.

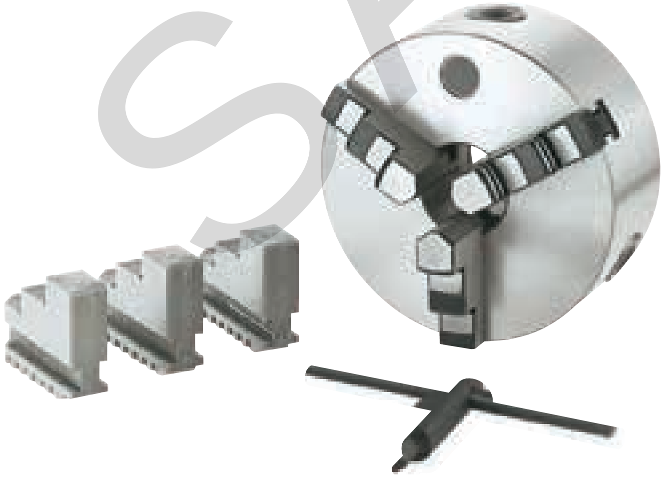


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 14-33. The 3-jaw universal chuck automatically centers round or hexagonal stock.

SAFETY NOTE

Remove the chuck key when you finish using it. If the key is left in the chuck when the lathe is turned on, it could become a dangerous missile. Make it a habit to never let go of a lathe chuck key unless you are placing it on the tool tray or lathe board.

The jaws of a universal chuck lose their centering accuracy as the scroll wears. Accuracy is also affected when too much pressure is used to mount the work, or when work is gripped too near the front of the jaws.

SAFETY NOTE

Avoid gripping work near the front of the jaws. The work can fly out and cause injuries.

Four-Jaw Independent Chuck

Each of the jaws of a 4-jaw independent chuck, **Figure 14-35**, operates individually, instead of being coupled with the other jaws as in the 3-jaw universal chuck. This permits square, rectangular, and odd-shaped work to be centered. Unlike those of a 3-jaw chuck, the jaws of a 4-jaw chuck can be removed from their slots and reversed. This reversing feature permits the jaws to be used to hold large-diameter work in one position and small-diameter work when reversed, **Figure 14-36**.

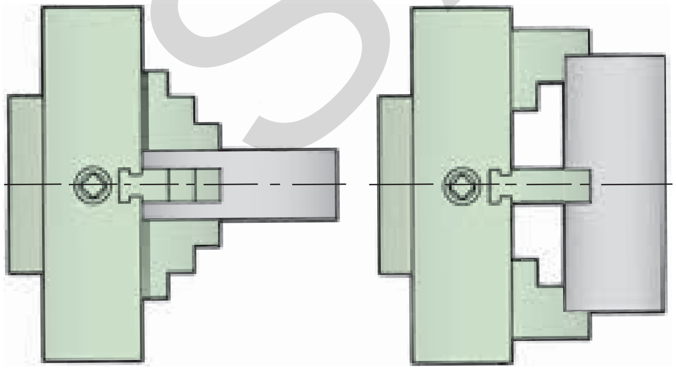
Unlike the 3-jaw chuck, the 4-jaw type is not self-centering. The most accurate way to center round work in this type of chuck is to use a dial indicator. The piece is first centered approximately, using the concentric rings on the chuck face as a guide. A dial indicator is then mounted in the tool post, **Figure 14-37**. The jaws are adjusted until the indicator needle does not fluctuate (move back and forth) when the work is rotated by hand. After the piece has been centered, all jaws must be tightened securely.

Goodheart-Willcox Publisher

Figure 14-37. Centering work in a 4-jaw chuck using a dial indicator. The machine shown is a small model maker's lathe.

AMT—The Association for Manufacturing Technology

Figure 14-35. A 4-jaw independent chuck. The jaws on this type of chuck are reversible.



Goodheart-Willcox Publisher

Figure 14-36. The reversing feature of jaws in a 4-jaw independent chuck makes it possible to turn work having extreme differences in diameter without difficulty.

Another centering method uses chalk. Rotate the work while bringing the chalk into contact with it. The chalk mark indicates the "high point." Slightly loosen the jaws opposite the chalk mark. Then tighten the jaws on the side where the chalk mark appears. Continue this operation until the work is centered. If the work is oversize enough, a cutting tool may be used instead of chalk.

Avoid trying to center stock in one or two adjustments. Rather, work in small increments (steps). When making the final small adjustment, it may be necessary to loosen the jaw on the low side and retighten it, then retighten the high side. This last method for making final adjustment applies, in particular, when centering work with a dial indicator.

Jacobs Chuck

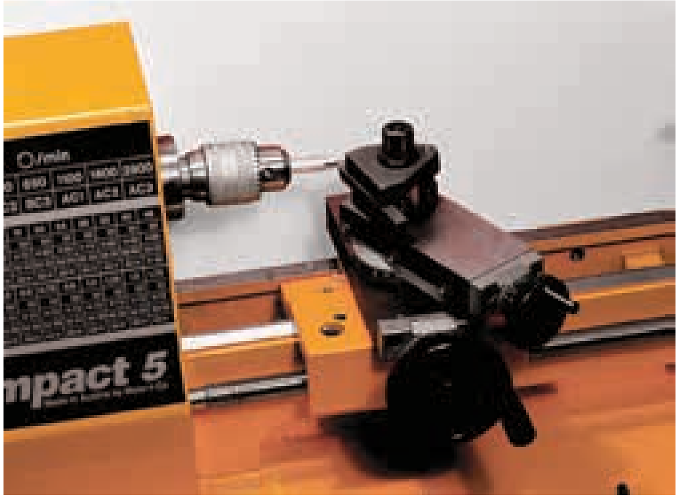
When turning small-diameter work, such as screws or pins, the Jacobs chuck can be used. This chuck, **Figure 14-38**, is better suited for such work than the larger universal or independent chuck.

A standard Jacobs chuck is usually fitted in the tailstock for drilling. However, it also can be mounted by fitting it in a sleeve and then placing the unit in the headstock spindle. Wipe the chuck shank, sleeve, and spindle hole with a clean, soft cloth before fitting them together.

A headstock spindle Jacobs chuck is similar to the standard Jacobs chuck, but it is designed to fit directly onto a threaded spindle nose, **Figure 14-39**. The chuck has the advantage of not interfering with the compound rest, making it possible to work very close to the chuck.

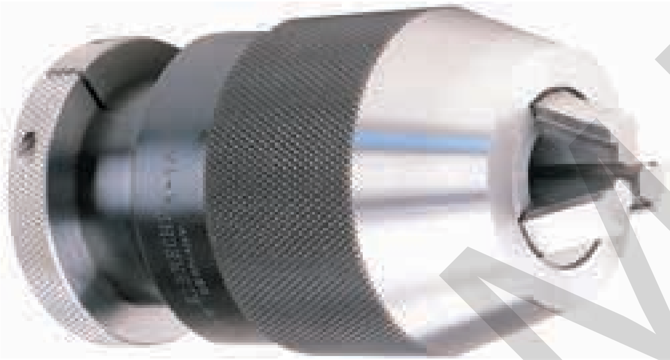
Draw-In Collet Chuck

The draw-in collet chuck is a work-holding device for securing work small enough to pass through the lathe spindle, **Figure 14-40**. Collets are accurately made sleeves with one end threaded and the other split into three even sections.



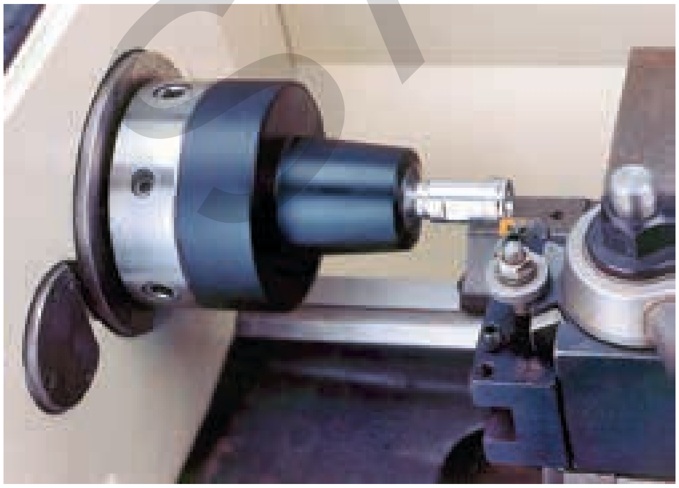
Goodheart-Willcox Publisher

Figure 14-38. Turning small-diameter work in a Jacobs chuck.



Royal Products, Division of Curran Manufacturing Corporation

Figure 14-39. This Jacobs chuck can be mounted on a lathe with a threaded spindle nose.



Royal Products, Division of Curran Manufacturing Corporation

Figure 14-40. A draw-in collet chuck.

The slots are cut slightly more than half the length of the collet and permit the jaws to spring in and clamp the work. The standard collet has a circular hole for round stock, but collets for holding square, hexagonal, and octagonal material are available.

The chief advantages of collets are their ability to center work automatically and to maintain accuracy over long periods of hard usage. They have the disadvantage of being expensive, since a separate collet is needed for each different size or stock shape, **Figure 14-41**.

A collet chuck that has steel segments bonded to rubber is also available. An advantage of this chuck is that each collet has a range of 0.100″ (2.5 mm), rather than being a single size, like steel collets. However, these collets are available only for round work.

Mounting Chucks

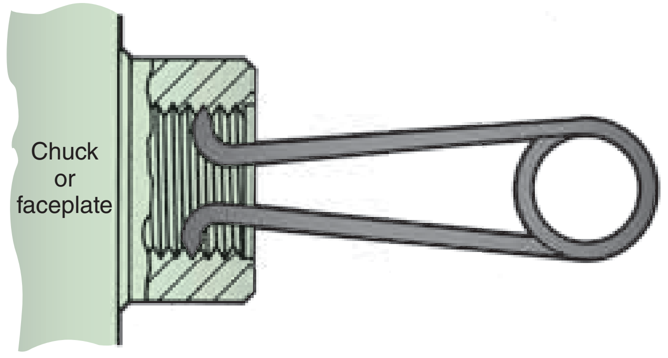
If a chuck is not installed on the spindle nose correctly, its accuracy is affected. To install a chuck, remove the center and sleeve, if they are in place. Hold the center and sleeve with one hand and tap them loose with a knockout bar. Carefully wipe the spindle end clean of chips and dirt. Apply a few drops of spindle oil. Clean the portion of the chuck that fits on the spindle. On a chuck that is fitted to a threaded spindle nose, clean the threads with a spring cleaner. See **Figure 14-42**.

For a tapered key spindle nose, rotate the spindle until the key is in the up position. Slide the chuck into place and tighten the threaded ring. Fit the pins on the cam-lock spindle nose into place and lock them.

Fitting a chuck onto a threaded spindle nose requires a different technique. Hold the chuck against the spindle nose with the right hand and turn the spindle with the left hand. Screw the chuck on until it fits firmly against the shoulder. To avoid possible injury, do not spin the chuck on rapidly or use power. Release belt tension, if possible, to eliminate any chance of power being transferred to the spindle.

Maswerks, Inc.

Figure 14-41. A variety of collets is necessary to clamp different stock sizes.



Goodheart-Willcox Publisher

Figure 14-42. A spring cleaner is used to clean the threads in a chuck before mounting it on a threaded spindle nose.

During installation, place a board on the ways under the chuck. The board will protect your hands and prevent damage to the machine ways if the chuck is dropped.

Removing a Chuck from Threaded Spindle

There are several accepted methods of removing chucks from a threaded headstock spindle. The first step in any method, regardless of the type of spindle nose, is to place a wooden lathe board across the ways beneath the chuck for support, **Figure 14-43A**. Then use one of the following techniques:

- Lock the spindle in back gear and use a chuck key to apply leverage.
- Place a suitable size adjustable wrench on one jaw and apply pressure to the wrench.
- If neither of the preceding methods works, place a block of hardwood between the rear lathe ways and one of the chuck jaws. Engage the back gear and give the drive pulley a quick rearward turn, **Figure 14-43B**.

Removing a Chuck from Other Spindle Noses

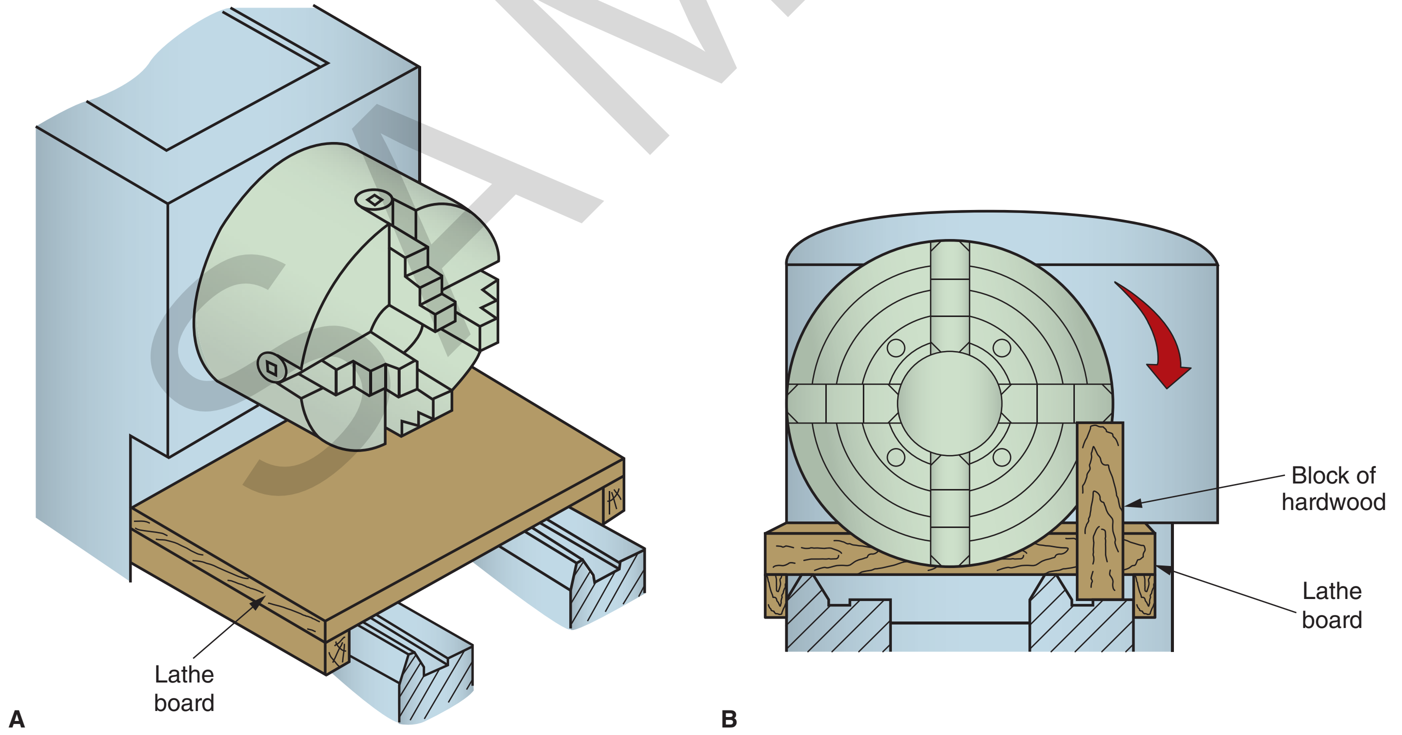
Removing a chuck from tapered and cam-lock spindle noses is not difficult. For tapered spindle noses, first lock the spindle in back gear, then place the appropriate spanner wrench in the locking ring. Give it a tap or two with a leather or plastic mallet. Turn the ring until the chuck is released.

SAFETY NOTE

Place a wooden cradle under the chuck before attempting to remove it from the spindle. This makes removal easier and helps avoid hand injuries.

14.4 Cutting Tools and Toolholders

To operate a lathe efficiently, the machinist must have a thorough knowledge of cutting tools and know how they must be



Goodheart-Willcox Publisher

Figure 14-43. Removing a chuck. A—A lathe board placed under a chuck when mounting or removing it will protect your hand and the machine ways if you drop the chuck. B—In truly stubborn cases, reverse the chuck against a block of hardwood.

shaped to machine various materials. The cutting tool is held in contact with the revolving work to remove material from the work. Before indexable insert cutting tools were invented, a ***single-point cutting tool*** of high-speed steel (HSS) was used for most applications.

The square cutter-bit body is inserted in a lathe toolholder. Toolholders are made in straight, right-hand, and left-hand models, **Figure 14-44A**. To tell the difference between right-hand and left-hand toolholders, hold the head of the tool in your hand and note the direction the shank points. The shank of the right-hand toolholder points to the right, and the shank of the left-hand toolholder points to the left.

A turret holder may also be utilized, **Figure 14-44B**. Turret holders typically have four cutter bits. A bit can be changed by loosening the lock (handle) and pivoting the holder so the new bit is in cutting position, then locking it in place.

14.4.1 High-Speed Steel Cutting Tool Shapes

Figure 14-45 shows the parts of a cutter bit and the correct terminology for those parts. To get the best performance, the bit must have a keen, properly shaped cutting edge. The shape depends on the type of work—roughing or finishing—and on the metal to be machined.

Most cutter bits are ground to cut in only one direction (left or right). The exception is the round-nose tool, which can cut in either direction. Some cutting tools used for general-purpose turning are shown in **Figure 14-46**.

14.4.2 Types of Cutting Tools

Various types of cutting tools are available for lathe work. These include tools for making rough and finish cuts, as well as for working with different materials.

Roughing Tools

The deep cuts made to remove considerable material from a workpiece are called ***roughing cuts***. Roughing tools have a tool shape (shape of cutting tip) that consists of a straight cutting edge with a small, rounded nose. This shape permits deep cuts at heavy feed rates. The slight side relief provides ample support to the cutting edge.

The left-cut roughing tool cuts most efficiently when it travels from left to right. The right-cut roughing tool operates just the opposite, from right to left. See **Figure 14-47A**.

Finishing Tools

The nose of a finishing tool is more rounded than the nose of the roughing tool. See **Figure 14-47B**. If the cutting edge

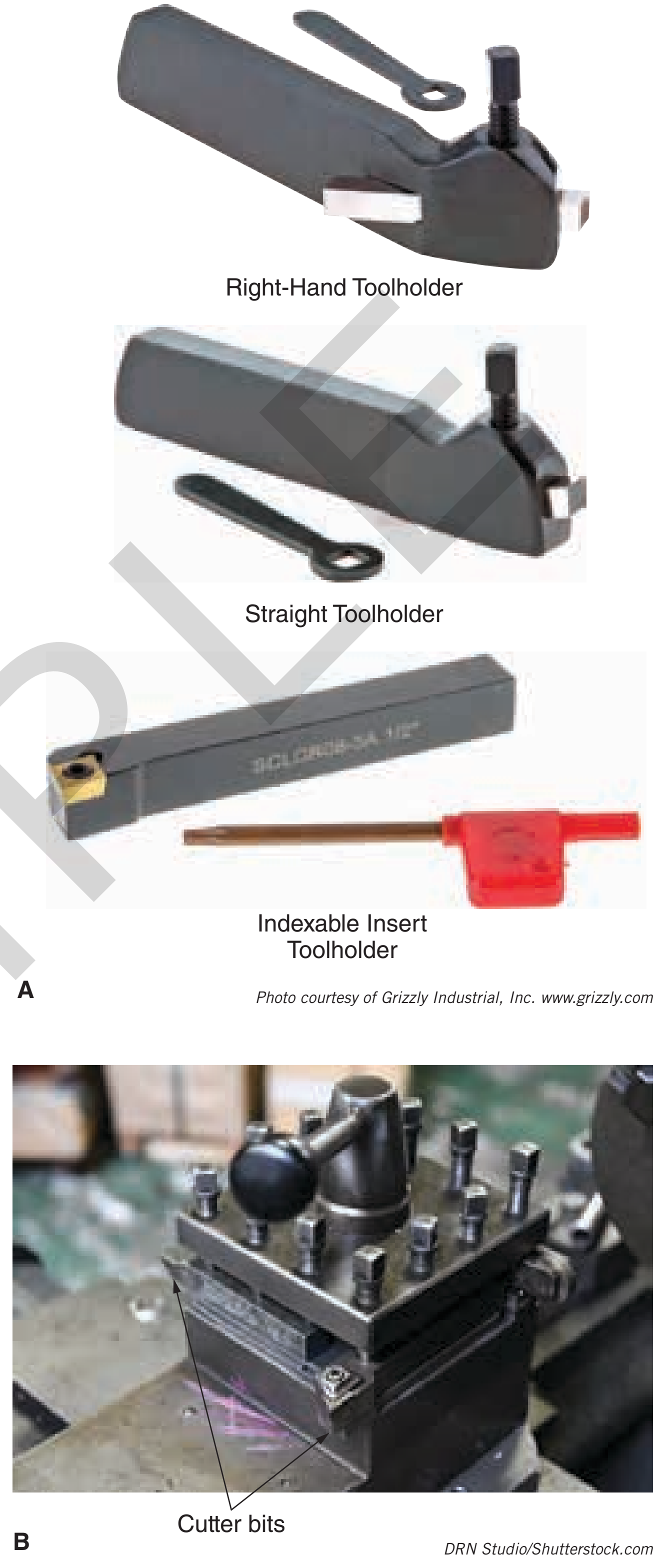
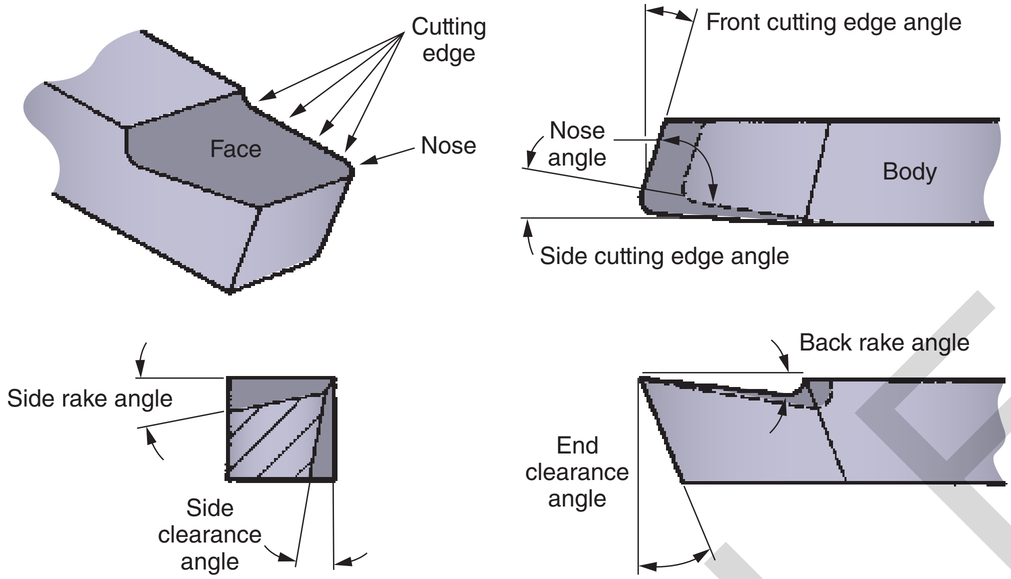
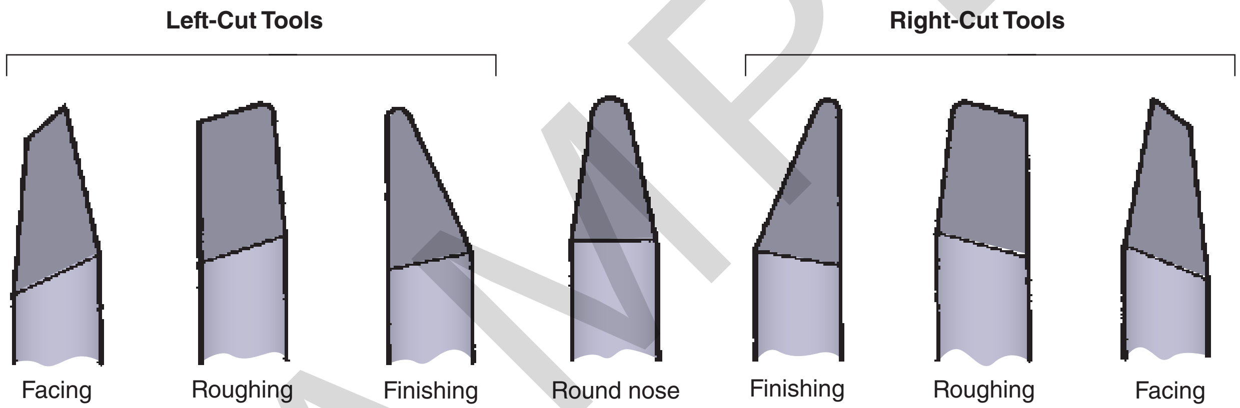


Figure 14-44. A—Examples of toolholders. B—A turret holder with four cutter bits.



Goodheart-Willcox Publisher

Figure 14-45. Cutter bit nomenclature.



Goodheart-Willcox Publisher

Figure 14-46. Standard HSS cutting tool shapes.

is honed with a fine oilstone after grinding, a finishing tool produces a smooth finish on the workpiece. A light cut and a fine feed must be used. Like roughing tools, finishing tools are made in left-hand and right-hand models.

Facing Tool

The facing tool is ground to prevent interference with the tailstock center. The tool point is set at a slight angle to the face of the work with the point leading slightly. See **Figure 14-48A**.

Round-Nose Tool

A round-nose tool is designed for lighter turning and is ground flat on the face (without back or side rake) to permit cutting in either direction. See **Figure 14-48B**. A slight variation of the round-nose tool, with a negative rake ground on the face, is excellent for machining brass, **Figure 14-49**.

Machining aluminum requires a tool with a considerably different shape from those previously described. As shown in **Figure 14-50**, the tool is set slightly above center to reduce any tendency to "chatter" (vibrate rapidly). The tool designs illustrated are typical of cutting tools used to machine aluminum alloys.

Brazed-Tip Single-Point Cutting Tools

Brazed-tip single-point cutting tools are made by brazing a carbide cutting tip onto a shank made from less costly

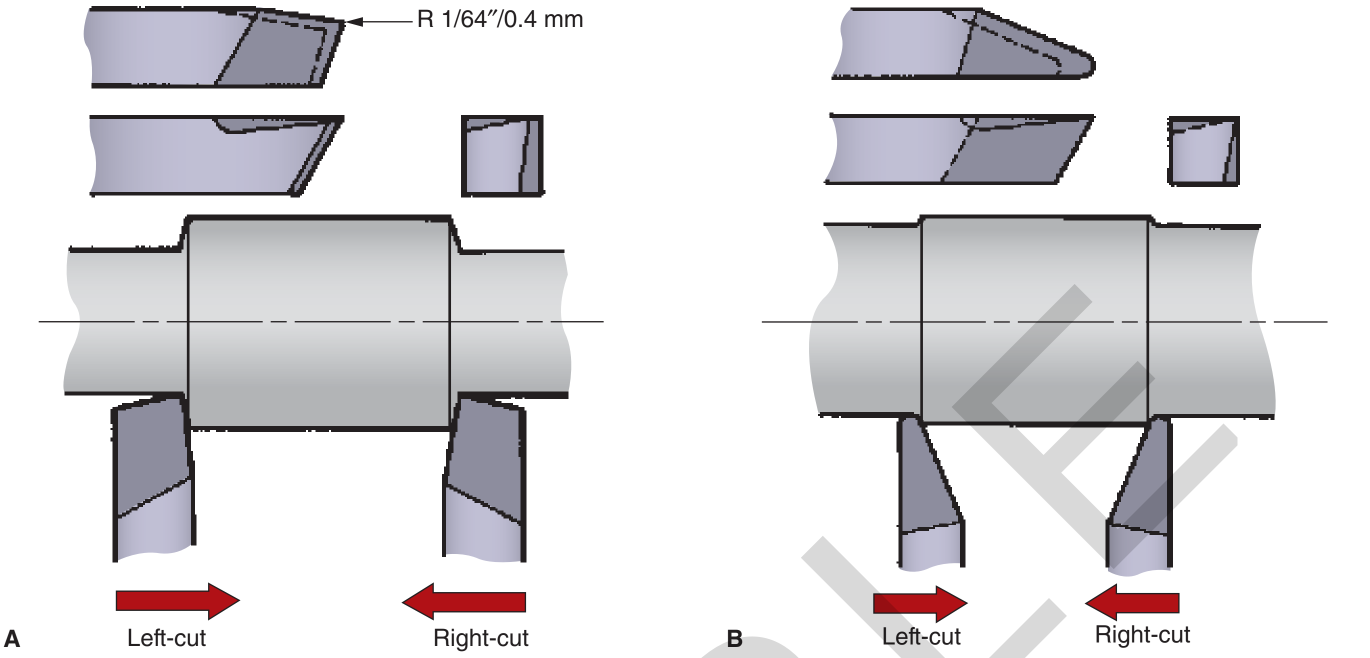


Figure 14-47. Lathe tools. A—The roughing tool is used for rapid material removal. B—The finishing tool produces a smooth surface.

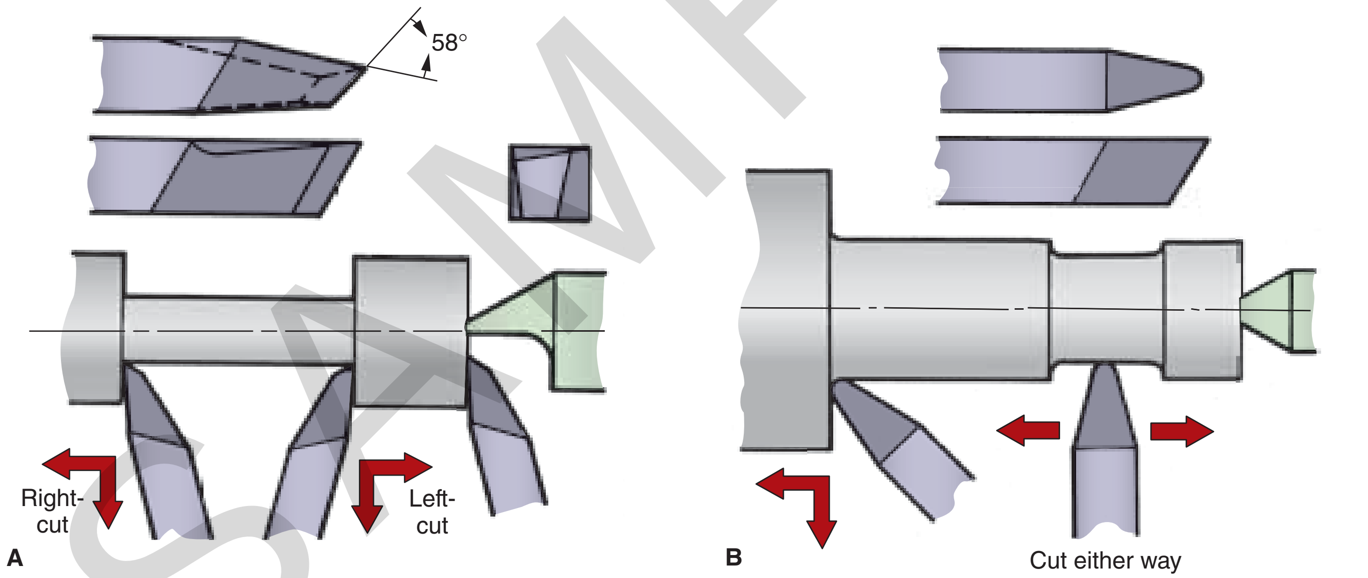
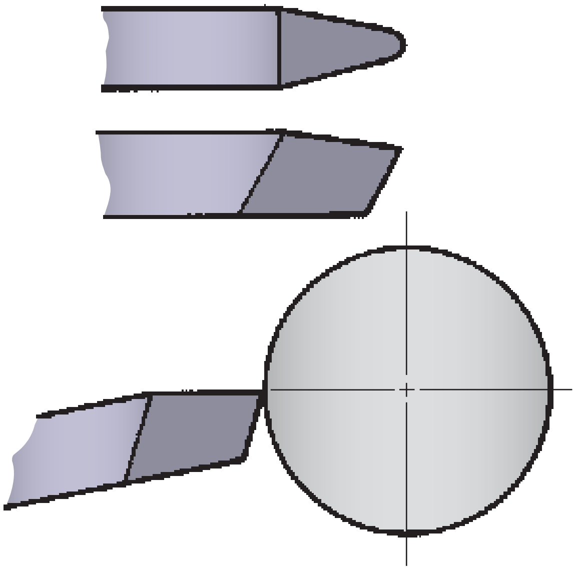


Figure 14-48. Lathe tools. A—A facing tool is used to machine surfaces perpendicular to the spindle centerline. B—A round-nose tool produces fillets. Its shape permits it to cut either left or right.

material, **Figure 14-51**. Many tip shapes (tool blanks) are available.

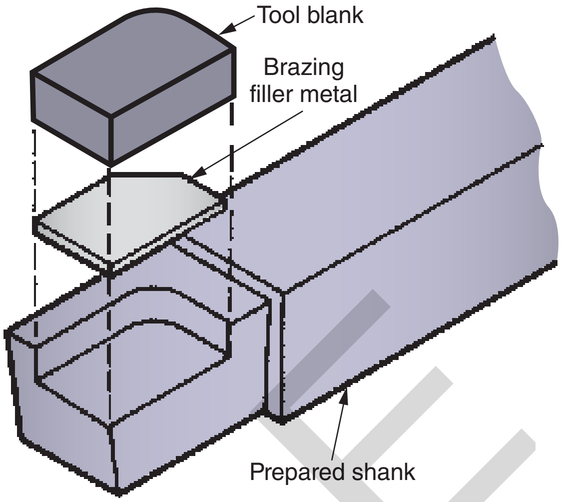
Cutting speeds can be increased by 300% to 400% when using carbide cutting tools. Powders of tungsten, carbon, and cobalt are molded into tool blanks and heated to extremely high temperatures. The hardness and strength of the blank can be controlled by varying the amount of cobalt that is used to cement (bind together) the tungsten and carbon particles.

For best results, these tools should be sharpened on a special silicon carbide or diamond-charged grinding wheel in which diamond dust particles or chips are embedded. A special type of grinder must be used, **Figure 14-52**.



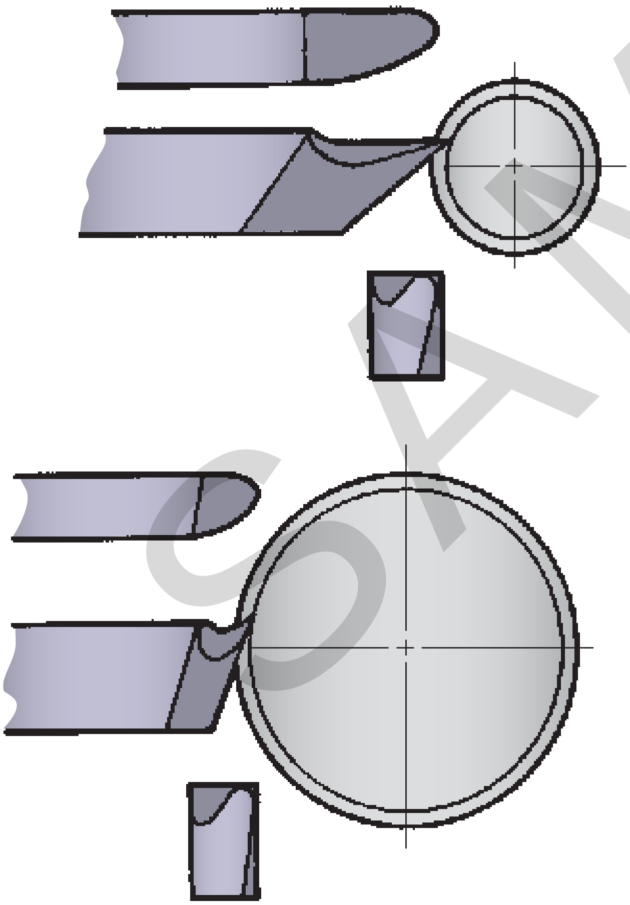
Goodheart-Willcox Publisher

Figure 14-49. Cutter nose shape with negative rake for machining brass.



Goodheart-Willcox Publisher

Figure 14-51. Brazing a carbide tool blank into place on a prepared shank. The brazing must be done properly or the tool blank will not be solidly attached, causing it to wear rapidly. Tungsten carbide tool blanks are available in a wide selection of shapes, sizes, and degrees of hardness.



Goodheart-Willcox Publisher

Figure 14-50. For machining aluminum, cutter bit shapes different from those used for other metals are necessary. The tool is set slightly above center for a smoother cut.

Goodheart-Willcox Publisher

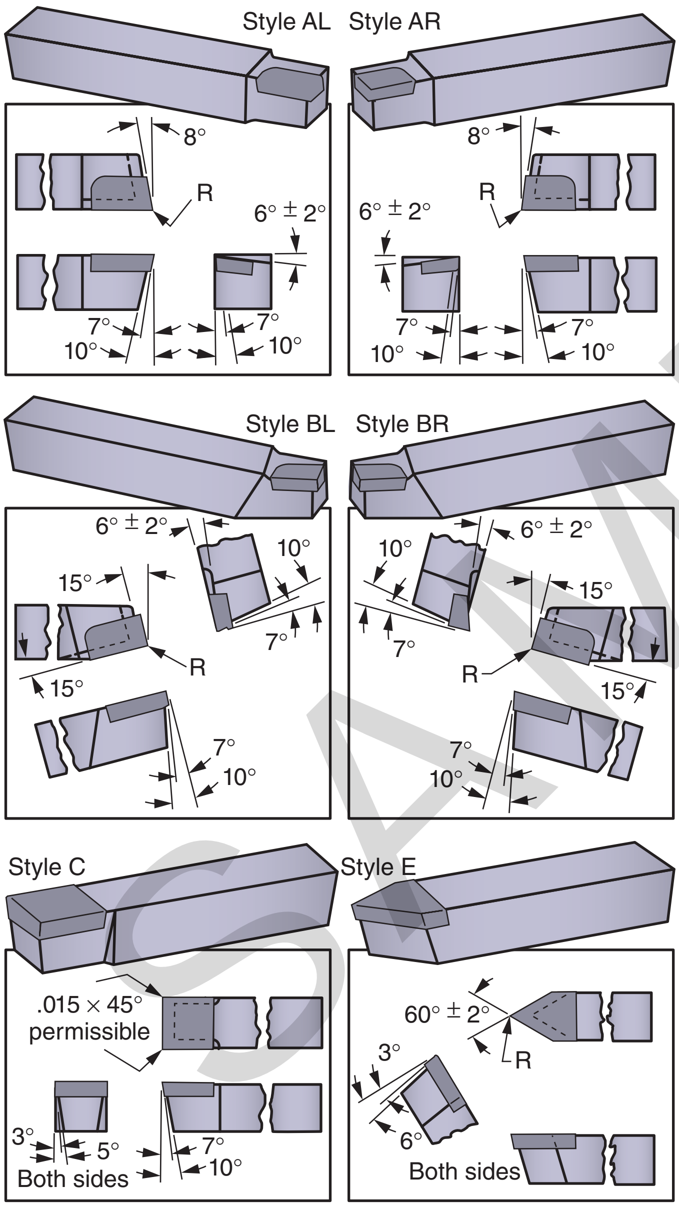
Figure 14-52. A grinder designed for sharpening carbide, cermet, cubic boron nitride, and polycrystalline diamond (PCD) cutting tools. It uses diamond-charged wheels and is fitted with a microscope inspection system.

Cutting tools designed for machining steel are chamfered 0.002″ to 0.003″ (0.050 to 0.075 mm) by honing them lightly with a silicon carbide or diamond hone. If the tools are not honed, the irregular edge produced by grinding will crumble when used. Honing, if done properly, does not interfere with the cutting action.

Carbide-Tipped Straight Turning Tools

The cutting tools shown in **Figure 14-53** are general-purpose tools for facing, turning, and boring. The square nose shape permits machining to a square shoulder. The clearance angles of the carbide tools described are not as great as those required for HSS cutting tools.

Also shown is a carbide-tipped threading tool (Style E). This tool has a 60° included angle that conforms to the Unified National 60° included-angle thread. It is used for V-grooving and chamfering.



Carboloy

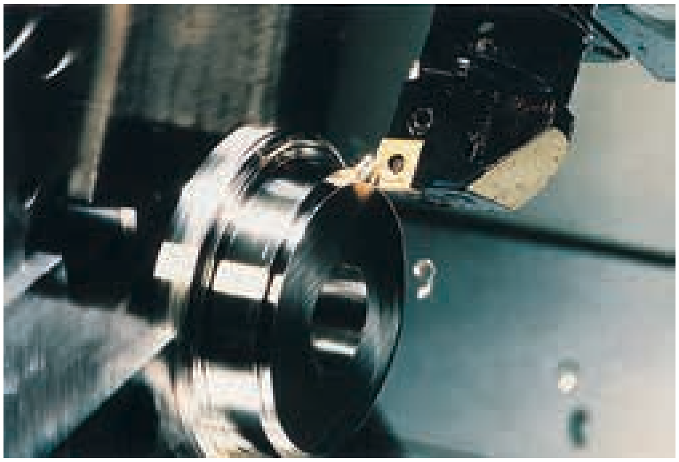
Figure 14-53. Typical standard cemented-carbide single-point tools. Style E is a carbide-tipped threading tool.

Indexable Insert Cutting Tools

Brazed-tip cutting tools have almost completely been replaced by mechanically clamped ***indexable insert cutting tools***, **Figure 14-54**. Indexable insert cutting tools are widely used for both turning and milling operations. The inserts are manufactured in a number of shapes and sizes, **Figure 14-55**, for different turning geometries. **Figure 14-56** shows six of the most commonly used standard shapes in order of decreasing strength.

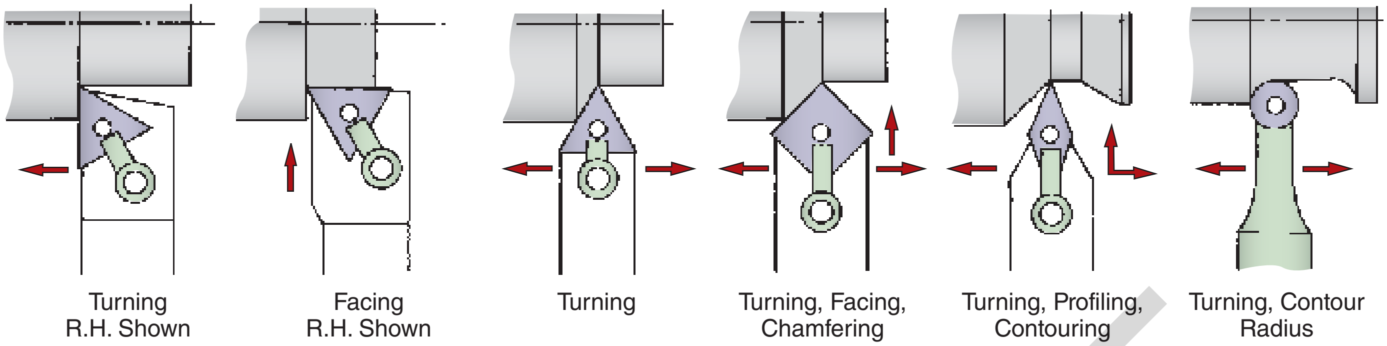
Indexable inserts clamp to toolholders, **Figure 14-57**. As an edge dulls, the next edge is rotated into position until all edges are dulled. Since it is less costly to replace inserts made from some materials than to resharpen them, they are usually discarded after use.

Inserts are manufactured from a number of materials, with each designed for a different metal requirement. See **Figure 14-58**. Carbide inserts are more versatile and have higher abrasion resistance, chemical stability, and lubricity when they are coated with various combinations of titanium carbide (TiC), titanium nitride (TiN), and alumina.



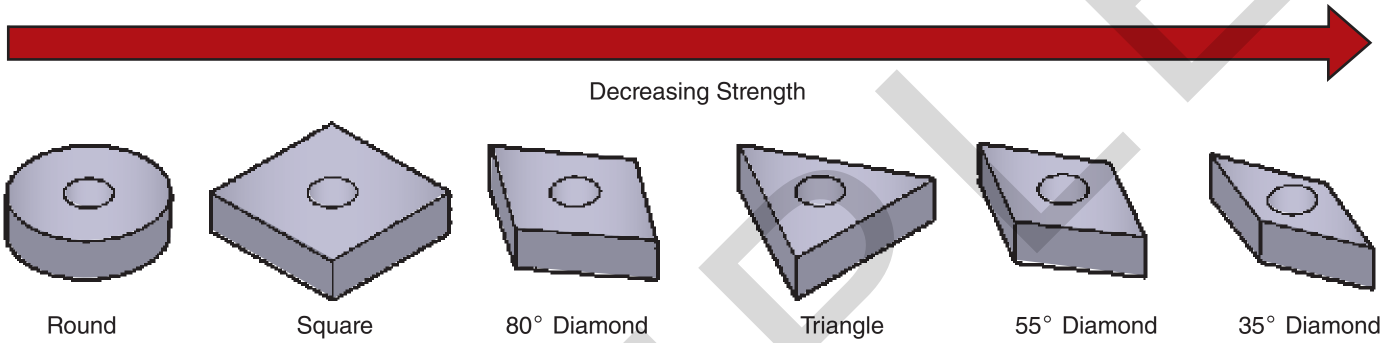
Sandvik Coromant Co.

Figure 14-54. Indexable insert cutting tools of carbide or sintered oxides (often referred to as *cermets*) are mechanically clamped into toolholders to perform cutting tasks. This insert is being used to machine stainless steel.



Goodheart-Willcox Publisher

Figure 14-55. Indexable inserts are manufactured in a number of different shapes and sizes for different turning operations.



Goodheart-Willcox Publisher

Figure 14-56. Most commonly used indexable insert shapes are shown in order of decreasing strength.

Carboloy

Figure 14-57. A selection of typical toolholders and replaceable carbide insert cutting tools for the lathe. Each insert has three or four cutting tips. The inserts are clamped in place on the holder and can be indexed (rotated into position) to present a new tip when the one in use becomes dull.

Chipbreakers

Machining some metals produces long, continuous chips unless some method is used to break the chips into smaller pieces. This is accomplished by a small step or groove, called a ***chipbreaker***, which is located on the top of the cutter at the cutting edge. Most inserts manufactured today have molded-in chipbreakers. Single-point cutting tools must have a chipbreaker ground into the top face of the tool, **Figure 14-59**.

Other Types of Cutting Tools

Diamonds, both natural and manufactured, are used as single-point cutting tools on materials whose hardness or abrasive qualities make them difficult to machine with other types of cutting tools. These diamonds are known as *industrial diamonds*.

14.4.3 Grinding High-Speed Cutter Bits

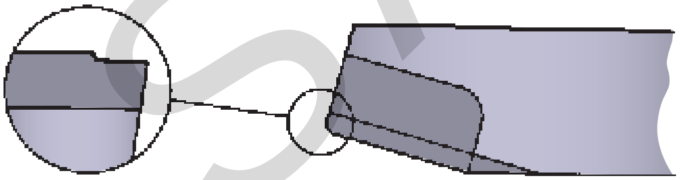
When first attempting to grind a cutter bit, it may be best if you first practice on sections of cold-finished steel square stock. You may also want to use chalk or bluing and draw the desired tool shape on the front portion of the blank. The lines will serve as guides for grinding.

Figure 14-60 depicts the recommended grinding sequence for a cutter bit. Side clearance, top clearance, and end relief may be checked with a clearance and cutting angle gage, **Figure 14-61**.

Material	Strengths	Weaknesses	Typical Applications
HSS	Superior resistance Versatility	Poor speed capabilities Poor wear resistance	Screw machine and other low-speed operations, interrupted cuts, low-horsepower machining.
Carbide	Most versatile cutting material High shock resistance	Limited speed capabilities	Finishing to heavy roughing of most materials, including irons, steels, exotics, and plastics.
Coated Carbide	High versatility High shock resistance Good performance at moderate speeds	Limited to moderate speeds	Same as carbide, except with higher speed capabilities.
Cermet	High versatility Good performance at moderate speeds	Low shock resistance Limited to moderate speeds	Finishing operation on irons, steels, stainless steels, and aluminum alloys.
Ceramic—Hot/Cold Pressed	High abrasion resistance High-speed capabilties Versatility	Low mechanical shock resistance Low thermal shock resistance	Steel mill-roll resurfacing, finishing operations on cast irons and steels.
Ceramic—Silicon Nitride	High shock resistance Good abrasion resistance	Very limited applications	Roughing and finishing operations on cast irons.
Ceramic—Whisker	High shock resistance High termal shock resistance	Limited versatility	High-speed roughing and finishing of hardened steels, chilled cast iron, high-nickel superalloys.
Cubic Boron Nitride	High hot hardness High strength High thermal shock resistance	Limited performance on materials below 38 on the Rockwell C hardness scale Limited applications High cost	Hardened work materials in 45–70 Rockwell C hardness range.
Poly-Crystalline Diamond	High abrasion resistance High speed capabilities	Limited applications Low mechanical shock resistance	Roughing and finishing operations on abrasive nonferrous or nonmetallic materials.

Valenite, Inc.

Figure 14-58. The nine basic categories of cutting tool materials.



Goodheart-Willcox Publisher

Figure 14-59. Typical chipbreaker on a single-point tool.

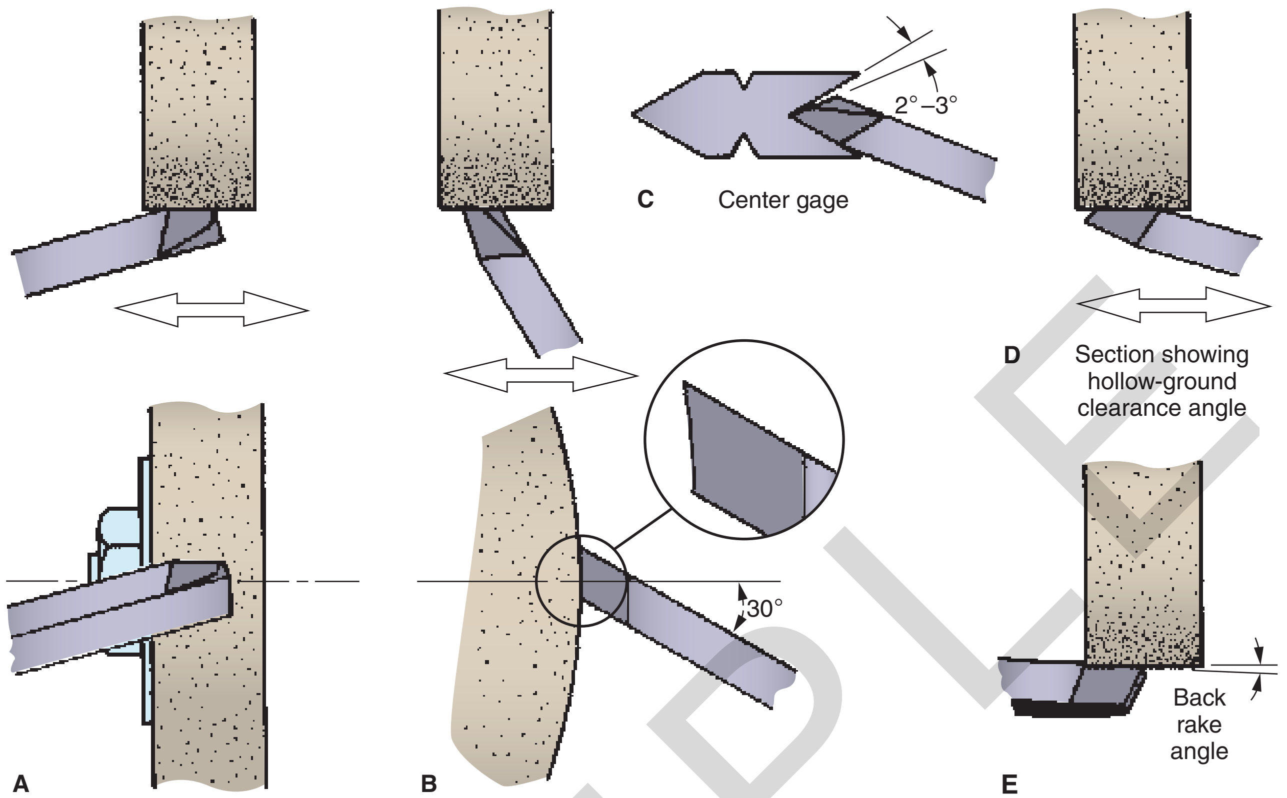
14.5 Cutting Speeds and Feeds

The matter of cutting speed and feed rate is important because these factors govern the length of time required to machine the work and the quality of the surface finish. ***Cutting speed*** is the distance the work moves past the cutting tool, expressed in feet per minute (fpm) or meters per minute (mpm). Measuring is done on the circumference of the work.

To explain it differently: if a lathe were to cut one long chip, the length of the chip cut in one minute (measured in either feet or meters) would be the cutting speed of the lathe. Cutting speed is *not* the revolutions per minute (rpm) of the lathe.

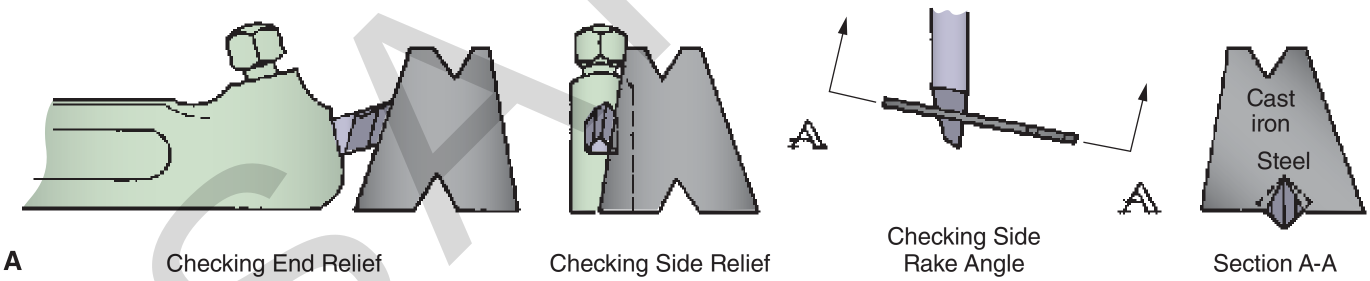
Feed rate is the distance that the cutter moves lengthwise along the lathe bed during a single revolution of the work. There are a number of factors that must be considered when determining the correct cutting speed and feed rate, including the following:

- Material used for the cutting tool.
- Kind of material being machined.
- Desired finish.



Goodheart-Willcox Publisher

Figure 14-60. Grinding sequence for a cutter bit. A—Two views showing how to position a cutter bit blank on the grinding wheel to shape the side clearance angle and side cutting edge angle. B—Shaping the end clearance angle and front cutting edge angle. C—Center gage being used to check the nose angle. D—Grinding the other side clearance angle, when required. E—Grinding the back/side rake angles. Accuracy of the clearance angles can be checked with a cutter bit gage.



Rake and Clearance Angle for Lathe Tools (High-Speed Steel)							
	Cast Iron	**Low-Carbon Steel**	**High-Carbon Steel**	**Alloy Steels**	**Soft Brass**	**Aluminum**	**Copper**
Back Rake	6–8°	8–12°	4–6°	5–8°	0–2°	25–50°	10–12°
Side Rake	10–12°	14–18°	8–10°	10–15°	0–2°	10–20°	20–25°
Clearance*	6–9°	8–10°	6–8°	6–8°	10–15°	7–10°	6–8°

*The end and side clearance angles are usually the same.

B

Goodheart-Willcox Publisher

Figure 14-61. Cutter bit gage. A—Bit gage being used to check accuracy after grinding the cutter tip. B—This table provides rake and clearance angles for lathe tools to machine different metals.

- Condition of the lathe.
- Rigidity of the workpiece.
- Kind of coolant being used (if any).
- Shape of the material being machined.
- Depth of cut.

If the machining is done with a cutting speed that is too slow, extra time will be needed to complete the job. If speed is too high, the cutting tool will dull rapidly and the finish will be substandard.

A speed and feed chart takes into consideration the many factors listed earlier. **Figure 14-62** is a chart for use with HSS cutter bits. The speeds for rough turning are presented as a starting point. The speed should be as high as the machine and work will withstand. The finishing speed depends on the desired finish quality. Cutting speeds and feed rates on the chart can be increased by 50% if a coolant is used and by 300% to 400% if a cemented carbide cutting tool is used. Depending on the condition of the machine, however, the cutting speed may have to be increased or decreased until optimum cutting conditions are obtained.

14.5.1 Calculating Cutting Speeds

Cutting speed (CS) is given in feet per minute (fpm) or meters per minute (mpm). Speed of the work (spindle speed) is given in revolutions per minute (rpm). Thus, the peripheral speed (speed at the circumference or outside edge of the work) must be converted to rpm to determine the required spindle speed. The following formulas are used:

Material to Be Cut	Roughing Cut 0.01″ to 0.020″ (0.25 mm to 0.50 mm) Feed		Finishing Cut 0.001″ to 0.010″ (0.025 mm to 0.25 mm) Feed	
	fpm	mpm	fpm	mpm
Cast Iron	70	20	120	36
Steel				
Low carbon	130	40	160	56
Med carbon	90	27	100	30
High carbon	50	15	65	20
Tool Steel (Annealed)	50	15	65	20
Brass—Yellow	160	56	220	67
Bronze	90	27	100	30

Goodheart-Willcox Publisher

Figure 14-62. Cutting speeds and feeds suggested for turning various metals with HSS tools.

Inch-based.

$$\text{rpm} = \frac{\text{CS} \times 4}{\text{D}}$$

Where:

rpm = Revolutions per minute.

CS = Cutting in feet per minute.

D = Diameter of the work in inches.

Cutting speed problem. What spindle speed is required to finish-turn 4″ diameter aluminum alloy?

CS = Table recommends a cutting speed of 1000 fpm for finish-turning aluminum alloy.

D = 4

$$\text{rpm} = \frac{\text{CS} \times 4}{\text{D}}$$

$$\text{rpm} = \frac{1000 \times 4}{4}$$

$$= 1000 \text{ rpm}$$

Adjust the spindle speed to as close to this speed (1000 rpm) as possible. Increase or decrease the speed as needed to obtain the desired surface finish.

Metric-based.

$$\text{rpm} = \frac{\text{CS} \times 333}{\text{D}}$$

Where:

rpm = Revolutions per minute.

CS = Cutting speed in meters per minute (mpm).

D = Diameter of work in millimeters (mm).

Cutting speed problem. What spindle speed is required to finish-turn 100 mm diameter aluminum alloy?

CS = Table recommends a cutting speed of 300 mpm for finish-turning aluminum alloy.

D = 100 mm

$$\text{rpm} = \frac{\text{CS} \times 333}{\text{D}}$$

$$\text{rpm} = \frac{300 \times 333}{100}$$

$$= 999 \text{ rpm}$$

Adjust the spindle to as close to this speed (1000 rpm) as possible. Increase or decrease speed as needed to obtain desired surface finish.

14.5.2 Roughing Cuts

Roughing cuts are taken to reduce work diameter to approximate size. The work is left 1/32″ (0.08 mm) oversize for finish turning. Since the finish obtained on the roughing cut is of little importance, use the highest speed and coarsest feed consistent with safety and accuracy.

14.5.3 Finishing Cuts

The finishing cut brings the work to the required diameter and surface finish. A high spindle speed, sharp cutting tool, and fine feed are used.

14.5.4 Depth of Cut

The ***depth of cut*** is the distance the cutter is fed into the work surface. The depth of cut varies greatly with lathe condition, material hardness, speed, feed rate, amount of material to be removed, and whether it is to be a roughing or finishing cut.

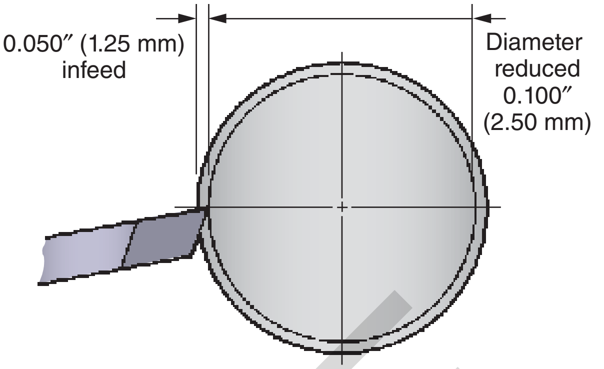
Depth of the cut can be set accurately with the micrometer dials on the cross-slide and compound rest, **Figure 14-63**. The micrometer dial is usually graduated in 0.001″ or 0.02 mm increments. This means that a movement of one graduation feeds the cutting tool into the piece 0.001″ or 0.02 mm. However, material is removed around the periphery (outside edge) of the rotating work at double the depth adjustment. For each 0.001″ (0.02 mm) of infeed, for example, the workpiece diameter is reduced by 0.002″ (0.04 mm). See **Figure 14-64**. This must not be forgotten or twice as much material as intended will be removed.

Some lathes, however, have a micrometer dial set up so that the number of graduations the cutter is fed into the work will equal the amount that the work diameter will be reduced. That is, if the cutter is fed in 0.005″ (0.10 mm) or 5 graduations, the work diameter will be reduced 0.005″ (0.10 mm). Check the lathe you will be using to be sure which system it uses.

A common mistake when using a lathe is to remove too little material at too slow a speed. Cuts as deep as 0.125″ (3 mm) can be handled by light lathes. Cuts of 0.250″ (6 mm) and deeper can be made by heavier machines without overtaxing the lathe.

Ed Phillips/Shutterstock.com

Figure 14-63. Graduated micrometer dials on the cross-slide and compound rest handwheels of a lathe are used to set the depth of cut.



Goodheart-Willcox Publisher

Figure 14-64. Material is removed from the work on each cut at two times the infeed distance.

14.6 Preparing the Lathe for Operation

Before an aircraft is permitted to take off, the pilot and crew must go through a checkout procedure to determine whether the engines, controls, and safety features are in first-class operating condition. The same applies to the operation of a machine tool, such as a lathe. The operator should inspect the machine for safe and proper operation. The checkout procedure for the lathe should include the following actions:

- Clean and lubricate the machine. Use lubricant types and grades specified by the manufacturer. Many recommend a specific lubricating sequence to reduce any possibility of missing a vital lubrication point.
- Be sure all guards are in position and locked in place.
- Turn the spindle over by hand to be sure it is not locked or engaged in back gear (unless you intend to use back gear).
- Move the carriage along the ways. There should be no binding.
- Check cross-slide movement. If there is too much play, adjust the gibs.
- Mount the desired work-holding attachment. Clean the spindle nose with a soft brush. For a threaded spindle nose, apply a drop of lubricating oil before attaching the chuck or faceplate.

- Adjust the drive mechanism for the desired speed and feed.
- If the tailstock is used, check it for proper alignment, **Figure 14-65**.
- Clamp the cutter bit into an appropriate toolholder and mount it in the tool post. Do not permit excessive compound rest overhang because this often causes tool chatter and results in a poorly machined surface, **Figure 14-66**.
- Mount the work. Check for adequate clearance between the work and the various machine parts.

SAFETY NOTE

Sleeves should be rolled up and all jewelry removed before beginning to use the lathe.

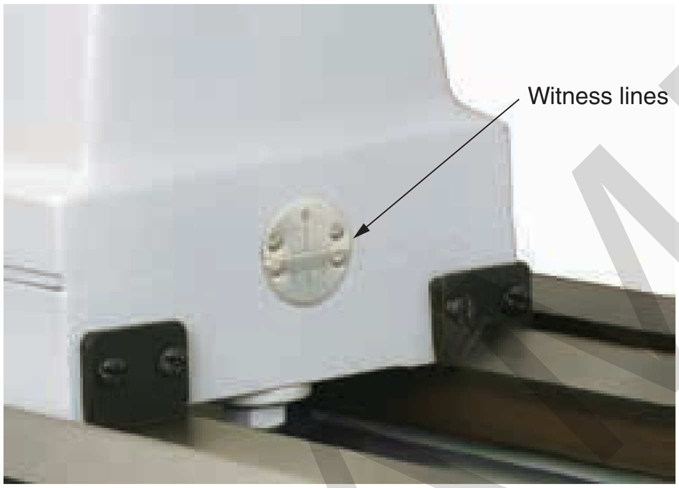
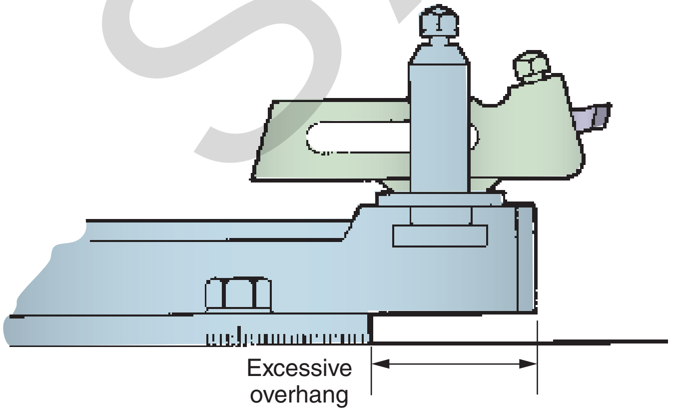


Photo courtesy of Grizzly Industrial, Inc. www.grizzly.com

Figure 14-65. Witness lines on the tailstock indicate whether the tailstock is aligned properly with the headstock.



Goodheart-Willcox Publisher

Figure 14-66. Excessive overhang of the compound rest usually causes tool chatter, resulting in a surface that is poorly machined.

Loose tools must never be placed on the lathe ways or carriage. A lathe board helps organize and hold the tools and measuring instruments needed for the job. Some lathes come pre-equipped with a lathe tray built into the headstock.

14.7 Cleaning the Lathe

To maintain the accuracy built into a lathe, it must be thoroughly cleaned after each work period. Use a 2″ paintbrush (not a dust brush) to remove the accumulated chips.

SAFETY NOTE

Lathe chips are sharp. Do not remove them with your hands. Never use an air hose to remove chips. The flying particles could injure you or others.

Wipe all painted surfaces with a soft cloth. To complete the job, move the tailstock to the extreme right end of the ways. Use a soft cloth to remove any remaining chips, oil, and dirt from the machined surfaces.

To prevent rust until the next time the machine is used, apply a light coating of machine oil to all machined surfaces. The lead screw occasionally needs cleaning. To do so, adjust the screw to rotate at a slow speed, then place a heavy cord around it and start the machine. With the lead screw revolving, permit the cord to feed along the thread. Hold the cord just tightly enough to remove the accumulated dirt.

SAFETY NOTE

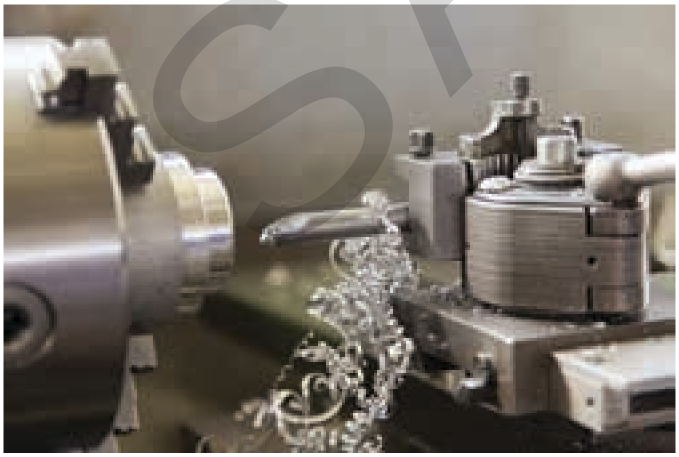
Never wrap the cord around your hand. The cord could catch and cause serious injury.

14.8 Lathe Safety

Follow these safety guidelines while operating a lathe:

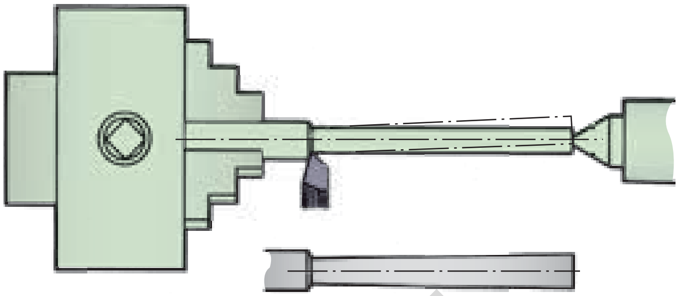
- Do not attempt to operate a lathe until you know the proper procedures and have been checked out on its safe operation by your instructor.
- Never attempt to operate a lathe while your senses are impaired by medication or other substances.
- Dress appropriately! Remove any necklaces or other dangling jewelry, wristwatch, or rings. Secure any loose-fitting clothing and roll up long sleeves. Wear an apron or a properly fitted shop coat. Safety glasses are a must!
- Clamp all work solidly. Use the correct size tool and work-holding device for the job. Get help when handling large sections of metal and heavy chucks and attachments.

- Check work frequently when it is being machined between centers. A workpiece expands as it heats up from friction and could damage the tailstock center.
- Be sure all guards are in place before attempting to operate the machine. Never attempt to defeat or bypass a safety switch.
- Turn the faceplate or chuck by hand to be sure there is no binding or danger of the work striking any part of the lathe.
- Keep the machine clear of tools.
- Always stop the machine before making measurements and adjustments.
- Metal chips are sharp and can cause severe cuts. Do not try to remove them with your hands when they become "stringy" and build up on the tool post, **Figure 14-67**. Stop the machine and remove them with pliers.
- Do not permit small-diameter work to project too far from the chuck without support from the tailstock. Without support, the work will be tapered, or worse, spring up over the cutting tool and could break. See **Figure 14-68**.
- Do not run the cutting tool into the chuck or dog. Check any readjustment of the work or tool to make sure there is ample clearance when the cutter has been moved leftward to the farthest point that will be machined.
- Stop the machine before attempting to wipe down its surface, so the cloth does not become caught on rotating parts. When knurling, keep the coolant brush clear of the work.
- Before repositioning or removing work from the lathe, move the cutting tool clear of the work area. This will prevent accidental cuts on your hands and arms from the cutter bit.



movit/Shutterstock.com

Figure 14-67. To avoid injury, always remove stringy chips with pliers. Never use your hands.



Goodheart-Willcox Publisher

Figure 14-68. If a small-diameter workpiece is not properly supported by a tailstock center, it will spring away from the cutting tool and be machined on a slight taper.

- Avoid talking to anyone while running a lathe. Do not permit anyone to adjust the machine while you are operating it. You are the only one who should turn the machine on or off or make any adjustments.
- If the lathe has a threaded spindle nose, never attempt to run the chuck on or off the spindle using power. It is also a dangerous practice to stop such a lathe by reversing the direction of rotation. The chuck could spin off and cause serious injury.
- Before engaging the half-nuts or automatic feed, you should always be aware of the direction of travel and speed of the carriage.
- Always remove the key from the chuck. Make it a habit to never let go of the key until it is out of the chuck and clear of the work area.
- Do not place tools on the lathe ways. Use a tool board or place them on the lathe tray.
- When doing filing on a lathe, make sure the file has a securely fitting handle.
- If any odd-sounding noise or vibration develops during lathe operation, stop the machine immediately. If you cannot locate the trouble, get help from your instructor. Do not operate the machine until the trouble has been corrected.
- Remove sharp edges and burrs from the workpiece before dismounting it from the machine. Burrs and sharp edges can cause painful cuts.
- Use care when cleaning the lathe. Chips sometimes stick in recesses. Remove them with a paintbrush or wooden stick, not a dust brush. Never clean a machine tool with compressed air.

14.9 Facing Operations

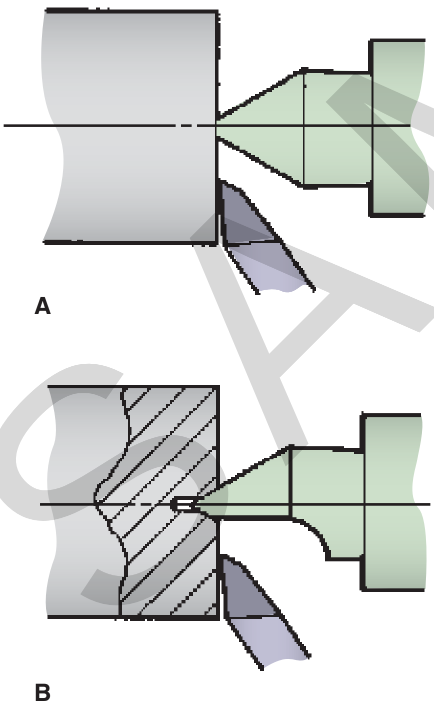
Facing is an operation that machines the end of the work square and reduces the work to a specific length. Facing is often the first operation performed in order to clean up the face of the workpiece before other machining operations are performed.

It is standard machining practice to cut stock slightly longer than needed. A steel rule may be used if the dimension is not critical. For more accuracy, a vernier caliper or large micrometer may be used. The difference between the rough length and the required length is the amount of material that must be removed.

14.9.1 Facing Work Held between Centers

Facing can be performed with the work mounted between centers. At times, considerable material must be removed. In this situation, it is best to leave the work longer than finished size and drill deeper center holes for better support during the roughing operation.

Face the work to length before starting the finish cut. Use a right-cut facing tool. The 58° point on this tool provides a slight clearance between the center point and the work face, **Figure 14-69**. Be careful not to damage the cutting tool point by running it into the center. A half center makes the operation easier, but is used only for facing. A half center does not provide an adequate bearing surface for general work and will not hold lubricant.



Goodheart-Willcox Publisher

Figure 14-69. Facing stock. A—Relationship of the cutter bit to the work face when making a facing cut. B—Using a half center provides more clearance when facing the end of stock.

Set the compound rest at 30°, **Figure 14-70**. Bring the cutting tool up until it just touches the surface to be machined, and then lock the carriage. Remove material from each end of the stock until the specified length is attained.

14.9.2 Facing Stock Held in a Chuck

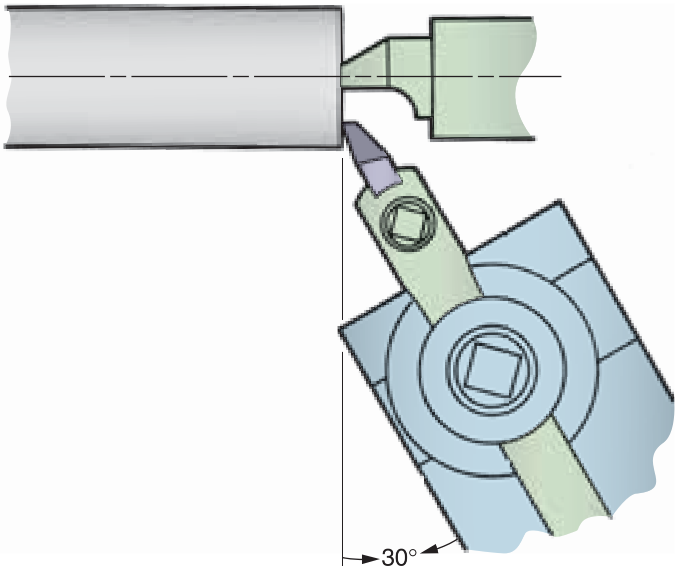
A round-nose cutting tool, held in a straight toolholder, is used to face stock held in a chuck. Pivot the compound rest 30° to the right. Set the toolholder to less than 90° to face the work, and make sure the cutter bit is exactly on center. Then move the carriage into position and lock it to the way. See **Figure 14-71A**.

A facing cut can be made in either direction. The tool may be started in the center and fed out, or the reverse may be done. The usual practice is to start from the center and feed outward. If the material is over 1 1/2″ (38 mm) in diameter, automatic feed may be used.

With the cutting tool on center, a smooth face will result from the cut. A rounded "nubbin" (remaining piece of unmachined face material) will result if the tool is slightly above center, **Figure 14-71B**. A square-shoulder nubbin indicates that the cutter is below center, **Figure 14-71C**. Reposition the tool and repeat the operation if either condition occurs.

14.10 Turning Operations

Turning is a machining operation that reduces the outside diameter of the work. Turning is one of the most commonly



Goodheart-Willcox Publisher

Figure 14-70. Recommended compound rest setting when facing stock to length.

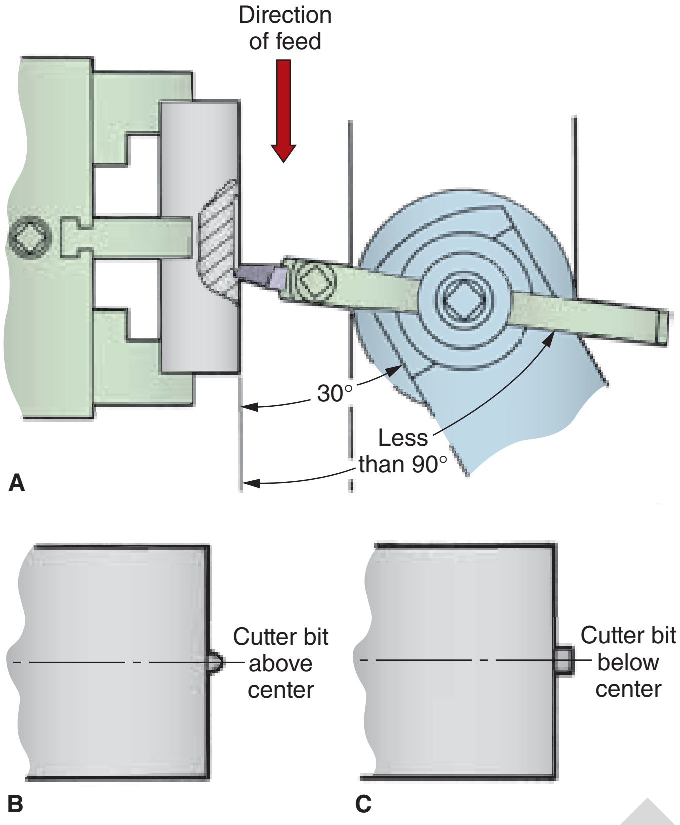


Figure 14-71. Facing in a chuck. A—Correct tool and toolholder positions for facing. B—Rounded nubbin left by above-center cutter. C—Square-shoulder nubbin left by below-center cutter.

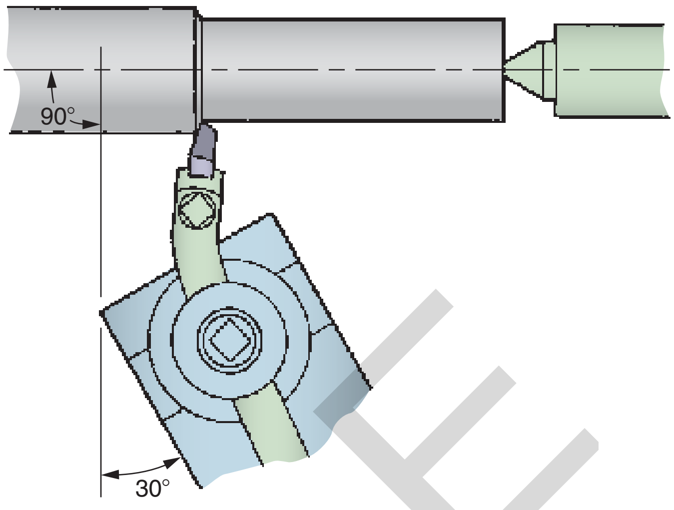


Figure 14-72. Compound rest setting used for rough turning.

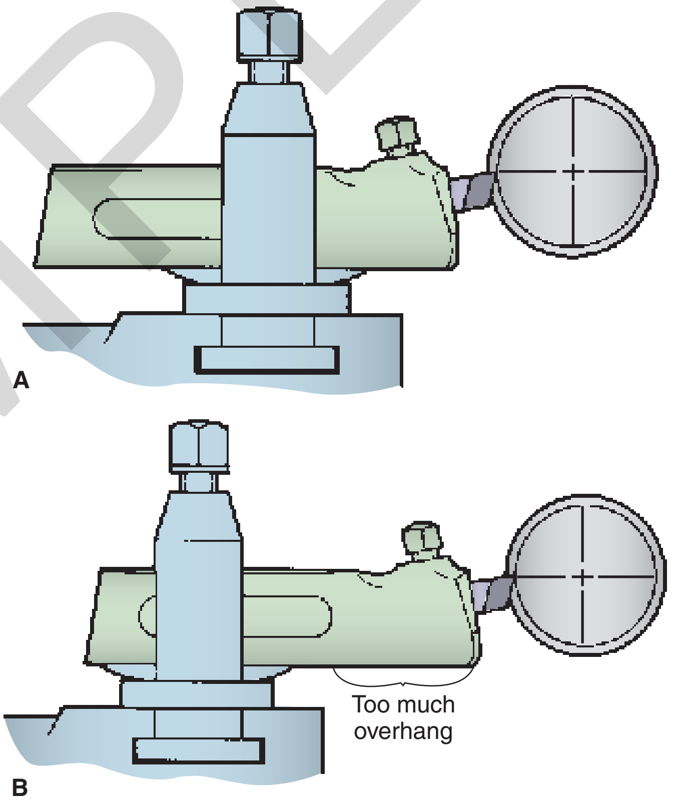


Figure 14-73. Correct and incorrect mounting of toolholder. A—Toolholder and cutter bit in proper position. B—Too much overhang causes the tool to chatter and produces a rough machined surface.

performed machining operations on a lathe. Turning operations can be performed with the work held between centers or in a chuck.

14.10.1 Rough Turning between Centers

Rough turning is an operation in which excess material is cut away rapidly with little regard for the quality of surface finish. The diameter is reduced to within 1/32″ (0.8 mm) of required size by using deep cuts and coarse feeds.

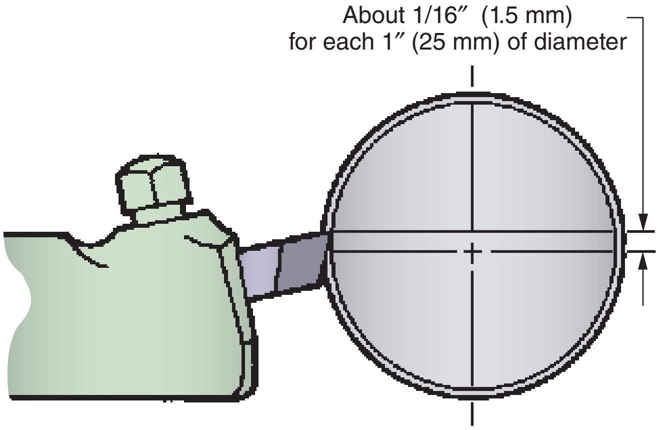
Set the compound rest at 30° from a right angle to the work, **Figure 14-72**. This will permit the tool to cut as close as possible to the left end of the work without the dog striking the compound rest.

SAFETY NOTE

Caution: Always check the maximum distance the compound rest can be fed toward the dog or chuck without striking them before you start the lathe.

Use a left-hand toolholder. Position the tool post as far to the left as possible in the compound rest T-slot. Avoid excessive tool overhang, **Figure 14-73**.

Locate the cutting edge of the tool about 1/16″ (1.5 mm) above the center of the work for each inch of diameter, **Figure 14-74**. It can be set by comparing it with the tail center point or with an index line scribed on the tailstock ram of some lathes.



Goodheart-Willcox Publisher

Figure 14-74. Set the tool slightly above center when rough turning.

The toolholder must be positioned correctly. If it is not, the heavy side pressure developed during machining will cause it to turn in the tool post, forcing the cutting tool deeper into the work. When the toolholder is correctly positioned, the cutting tool pivots away from the work. See **Figure 14-75**.

Make a trial cut to true up the stock. Measure the resulting diameter. The difference between the diameter and the required rough diameter is twice the distance the tool must be fed into the work. If the piece is greatly oversize, it will be necessary to make two or more cuts to bring it to size.

When depth of cut has been determined, engage the power feed. Observe the condition of the chips. They should be in small sections and slightly blue in color. Long, stringy chips indicate a cutting tool that is not properly sharpened. Stop the machine and remove stringy chips with pliers. Replace the cutting tool with one that is properly sharpened. After each cut, measure the work diameter to prevent excess metal removal.

SAFETY NOTE

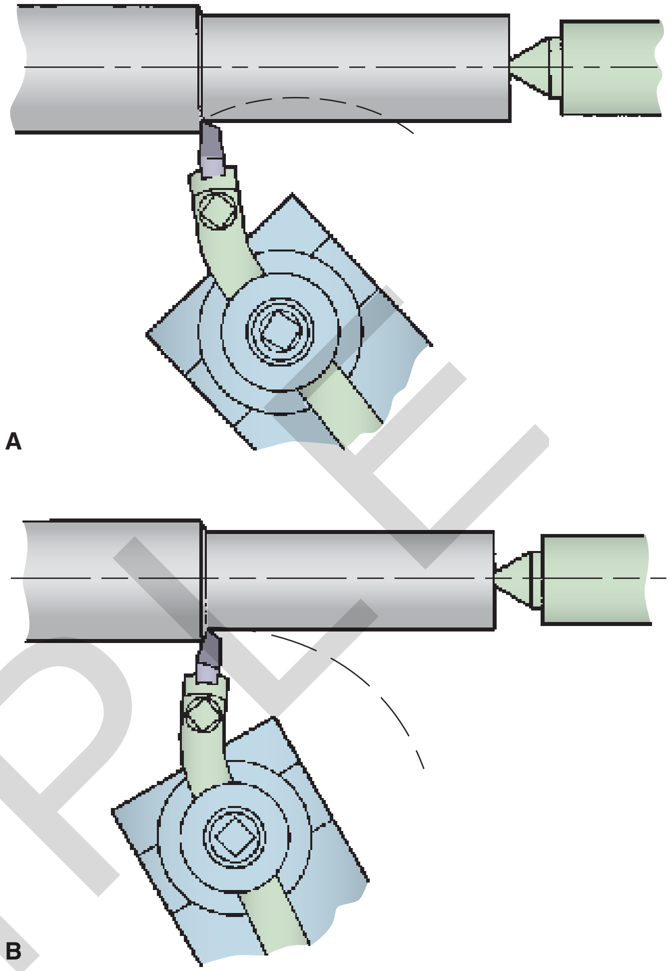
Always stop the machine before making measurements or cleaning out chips.

If a dead center is used in the tailstock, lubricate the center frequently. Stop the machine immediately if the center heats up and starts to smoke or "squeal."

14.10.2 Finish Turning

After rough turning, the work is still oversize. It must be machined to the specified diameter and to a smooth surface finish by finish turning, **Figure 14-76**.

Fit a right-cut finishing tool into the toolholder. All rough and finish machining should be done toward the headstock (right to left) because the headstock offers a more solid base than the tailstock. Position the tool on center and



Goodheart-Willcox Publisher

Figure 14-75. Toolholder positioning. A—An incorrectly positioned tool cuts deeper into work if the toolholder slips in the tool post. B—A correctly positioned tool swings clear of the work if the toolholder slips.

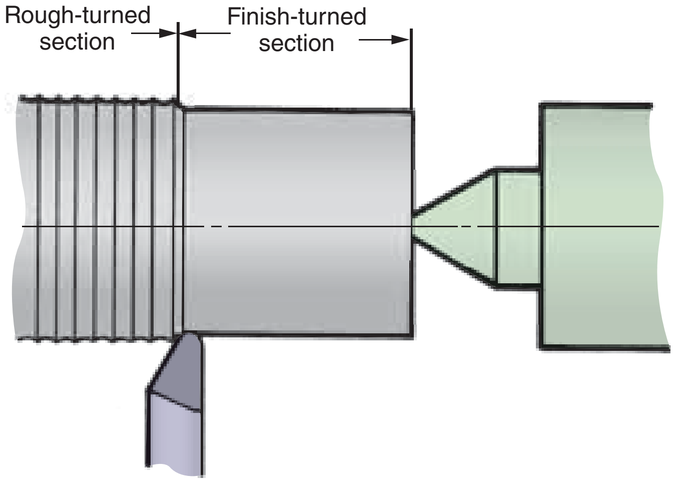
check for adequate clearance between the compound rest and the revolving lathe dog.

Adjust the lathe for a faster spindle speed and a fine feed. Run the cutting tool into the work until a light cut is being made, then engage the power feed. After a sufficient distance has been machined, disengage the power feed and stop the lathe.

SAFETY NOTE

Never reverse a lathe. Brake it to a stop.

Do not interfere with the cross-slide setting. Use a micrometer to measure the diameter of the machined area. The difference between the measurement and the specified diameter is the amount of material that must be removed. Move the cutting tool clear of the work and feed it in one-half the amount that must be removed. For example, if the



Goodheart-Willcox Publisher

Figure 14-76. After roughing work to approximate size, turn it to the required size with a finishing tool. Make the cut from right to left.

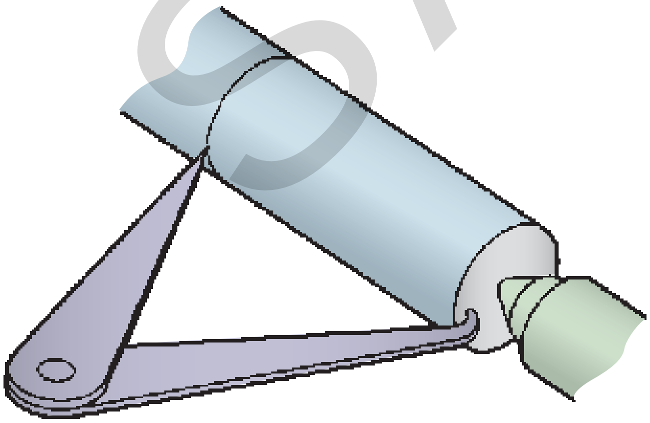
diameter is 0.008″ (0.20 mm) oversize, tool infeed should be 0.004″ (0.10 mm). Make another cut about 1/2″ (13 mm) in width at the new depth setting. Measure again to make sure the correct diameter will be machined.

When reversing the work to permit machining its entire length, avoid marring the finished surface by touching the lathe dog setscrew. Insert a small piece of soft aluminum or copper sheet between the setscrew and the workpiece.

14.10.3 Turning to a Shoulder

Up to this point, only ***plain turning*** has been described. This is turning in which the entire length of the piece is machined to a specified diameter. However, it is frequently necessary to machine a piece to several different diameters.

Locate the points to which the different diameters are to be cut. Scribe lines with a hermaphrodite caliper that has been set to the required length, **Figure 14-77**.



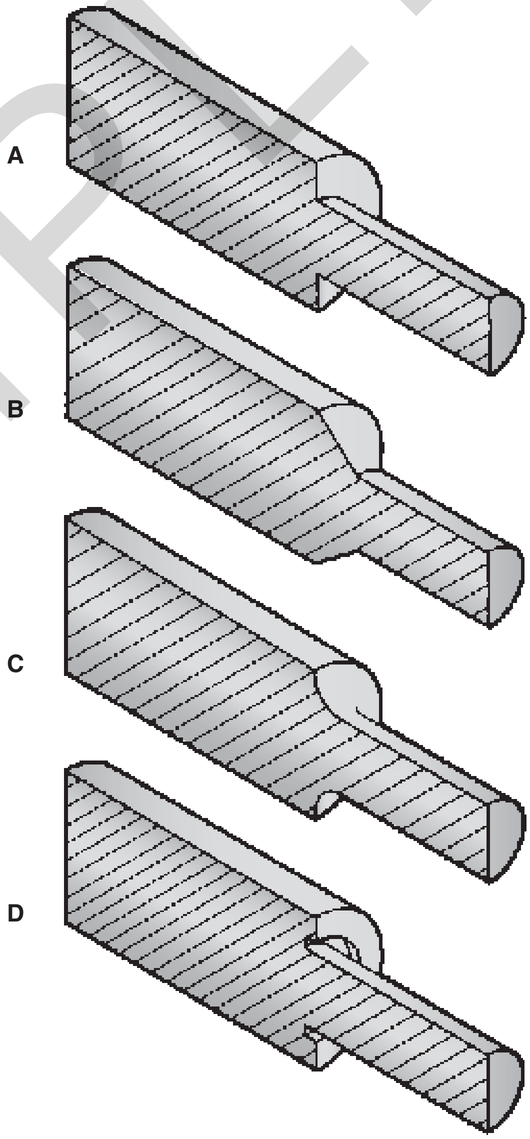
Goodheart-Willcox Publisher

Figure 14-77. Scribing reference lines on a workpiece with a hermaphrodite caliper.

Machining is done as previously described, with the exception of cutting the shoulder, which is the point at which the diameters change. The four types of shoulders, as shown in **Figure 14-78**, are as follows:

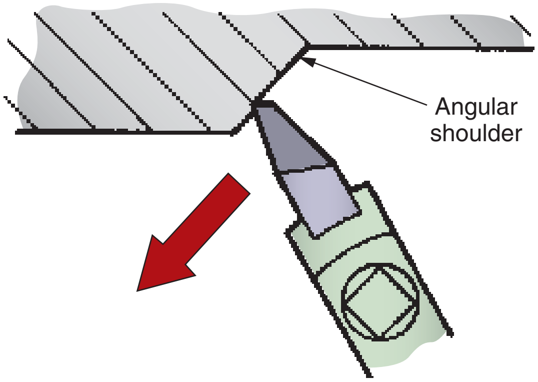
- Square.
- Angular.
- Filleted.
- Undercut.

Use a right-cut tool to make the square and angular shoulders. See **Figure 14-79** and **Figure 14-80**. To machine a filleted shoulder, grind a round-nose tool to the required radius using a fillet or radius gage to check radius accuracy. See **Figure 14-81**.



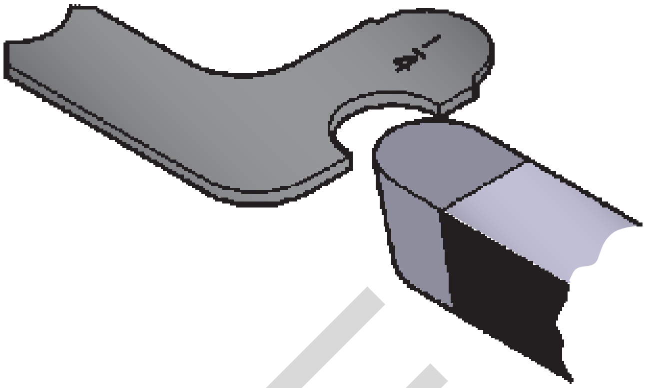
Goodheart-Willcox Publisher

Figure 14-78. The four types of shoulders. A—Square. B—Angular. C—Filleted. D—Undercut.



Goodheart-Willcox Publisher

Figure 14-79. To machine an angular shoulder, make the cut from the smaller diameter to the larger diameter.



Goodheart-Willcox Publisher

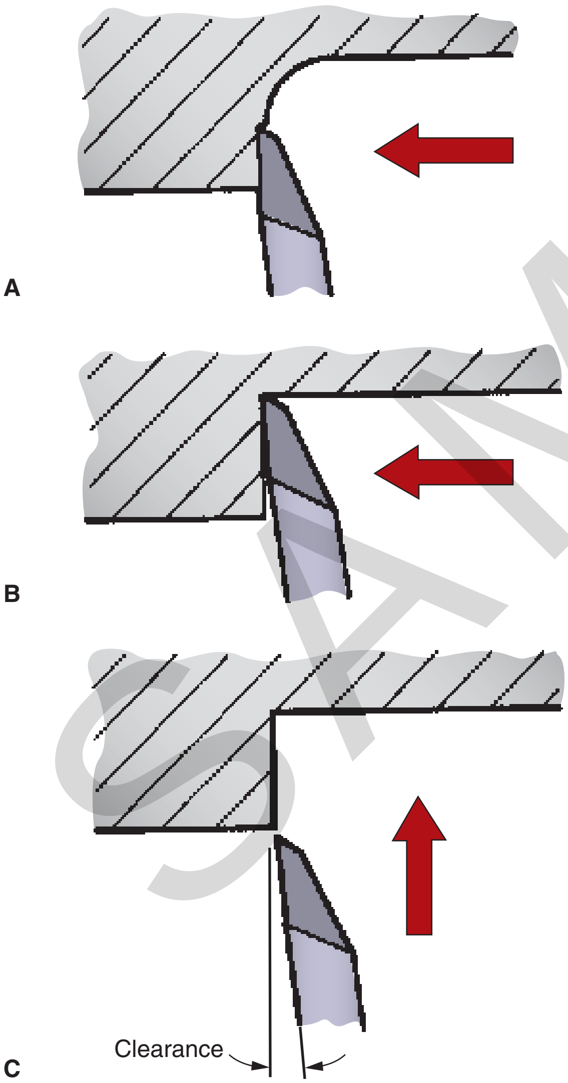
Figure 14-81. The radius on a cutter bit can be checked with a fillet gage.

14.10.4 Turning Work Held in a Chuck

Work mounted in a chuck is machined in the same manner as if it were between centers. To prevent "springing" (flexing) while it is being machined, long work should be center-drilled and supported with a tailstock center, **Figure 14-82**.

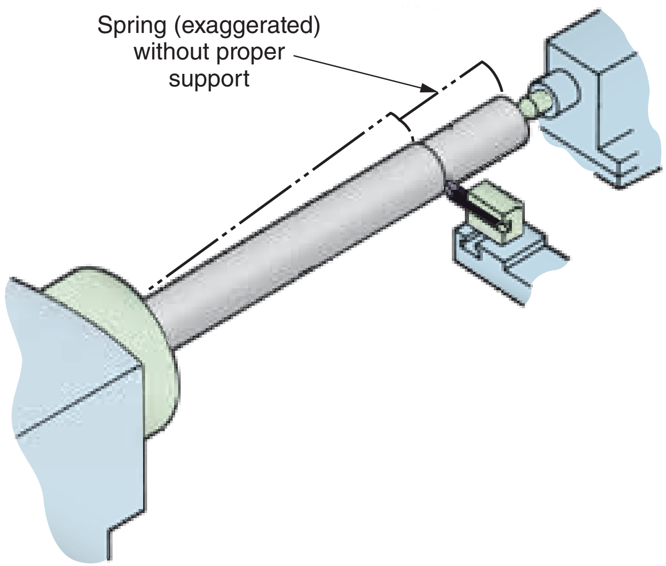
14.11 Parting and Grooving Operations

Parting and grooving are two common machining operations that can be performed on a lathe. ***Parting*** is the



Goodheart-Willcox Publisher

Figure 14-80. Machining sequence used to cut a square shoulder. A—First cut. B—Second cut. C—Facing cut.



Goodheart-Willcox Publisher

Figure 14-82. For accurate turning, long work must be supported with a tailstock center.

operation of cutting off material after it has been machined, **Figure 14-83**. ***Grooving*** is used to cut recesses or grooves into the surface of the workpiece.

Cutting tools for parting or grooving are held in a straight or offset toolholder. They must be ground with the correct clearance (front, side, and end). A concave rake is ground on top of the cutter to reduce chip width and to prevent it from seizing (binding) in the groove.

Keep the tool sharp. This will permit easy penetration into the work. If the tool is not kept sharp, it may slip and, as pressure builds up, dig in suddenly and break.

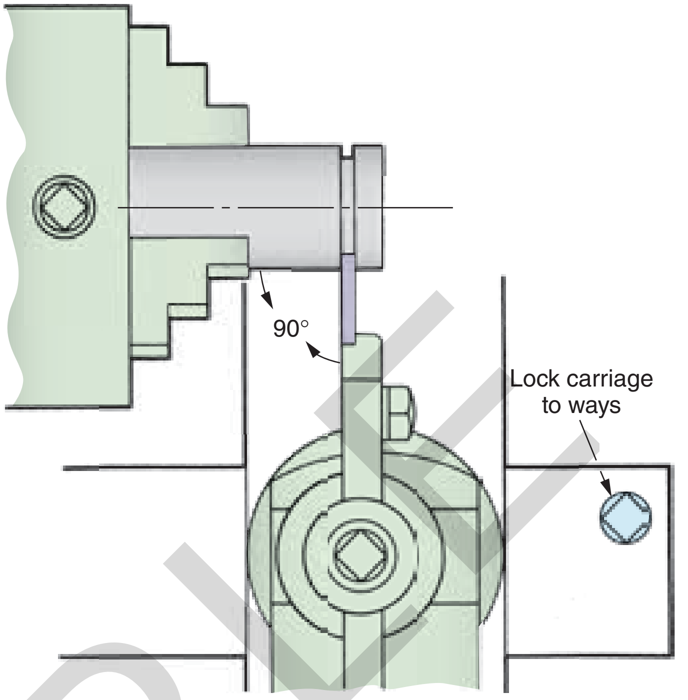
14.11.1 Parting Operations

Parting is one of the more difficult operations performed on a lathe. The cutoff blade is set at exactly 90° to the work surface, **Figure 14-84**. The cutting edge should be set on center when parting stock 1″ (25.0 mm) in diameter. For larger pieces, the cutting edge should be positioned 1/16″ (1.5 mm) above center for each 1″ (25.0 mm) of diameter. The tool must be lowered as work diameter is reduced, unless the center of the piece has been drilled out.

Spindle speed is about one-third of the speed used for conventional turning. The compound rest and cross-slide must be tightened to prevent play.

Iscar Metals, Inc.

Figure 14-83. Parting is one of the more difficult jobs performed on the lathe. This illustration shows parting of thick-wall tubing. The replaceable tool has a helical, twisted geometry to prevent binding during parting operations.



Goodheart-Willcox Publisher

Figure 14-84. Work is held close in the chuck for the parting operation. The parting tool blade is set at a 90° angle to cut, and the carriage is locked to the ways.

SAFETY NOTE

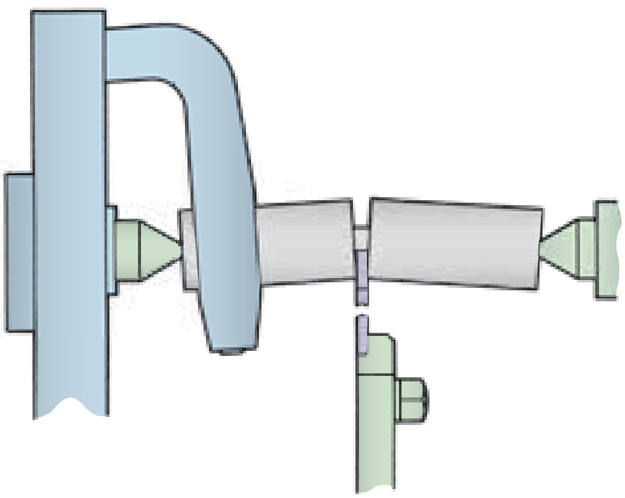
Do not forget to lock the carriage to the ways during a parting operation.

Feed should be ample to provide a continuous chip. If feed is too slow, "hogging" (the cutter digging in and taking a very heavy cut) can result. The tool will not cut continuously, but will ride on the surface of the metal for a revolution or two, then bite suddenly. If the machine is in good condition, automatic cross-feed may be used.

When parting, apply ample quantities of cutting fluid. Whenever possible, hold the work close in the chuck and, if necessary, use an offset toolholder.

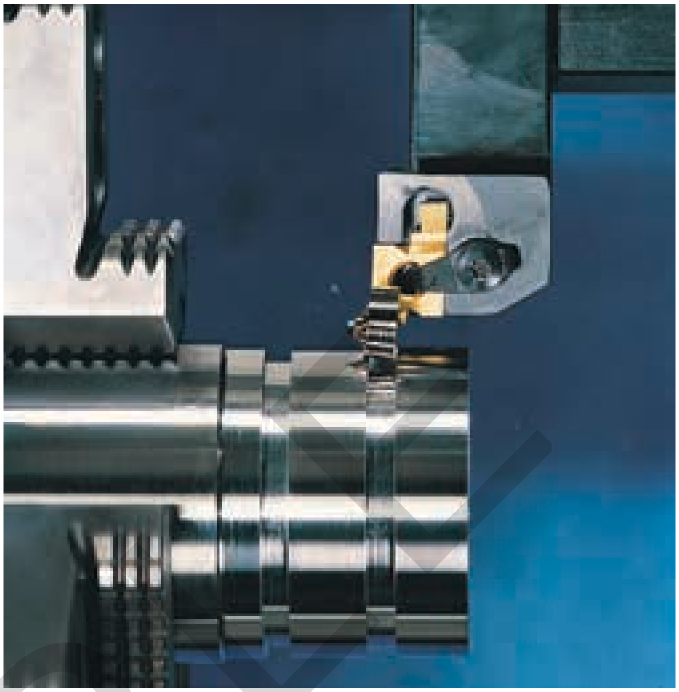
SAFETY NOTE

Never attempt to part work that is held between centers. This can cause serious trouble. See **Figure 14-85**.



Goodheart-Willcox Publisher

Figure 14-85. Work cannot be parted safely while being held between centers.



Carboloy

Figure 14-86. A groove or neck can be cut into a shaft with a grooving or parting tool. Toolholders for grooving and parting can be straight or offset. The shape of the groove may be square, angular (square with sloping sides), or round.

14.11.2 Grooving or Necking Operations

It is sometimes necessary to cut a groove or neck on a shaft to terminate a thread or to provide adequate clearance for mating parts, **Figure 14-86**. Any recess cut into a surface has a tendency to weaken a shaft. It is therefore better to make the groove round, rather than square.

The tool is set on center and fed in until it just touches the work surface. Set the cross-feed micrometer dial to zero and feed in the tool to the specified depth. Square grooves can be machined with a parting tool or HSS tool ground to the correct width.

14.11.3 Cutting Grooves with a Parting Tool

When tolerances and surface finish requirements permit, grooves of various widths can often be machined with a parting tool. **Figure 14-87** illustrates two techniques that can be used. The parting tool should be mounted in the toolholder to reduce chatter and tool flexing.

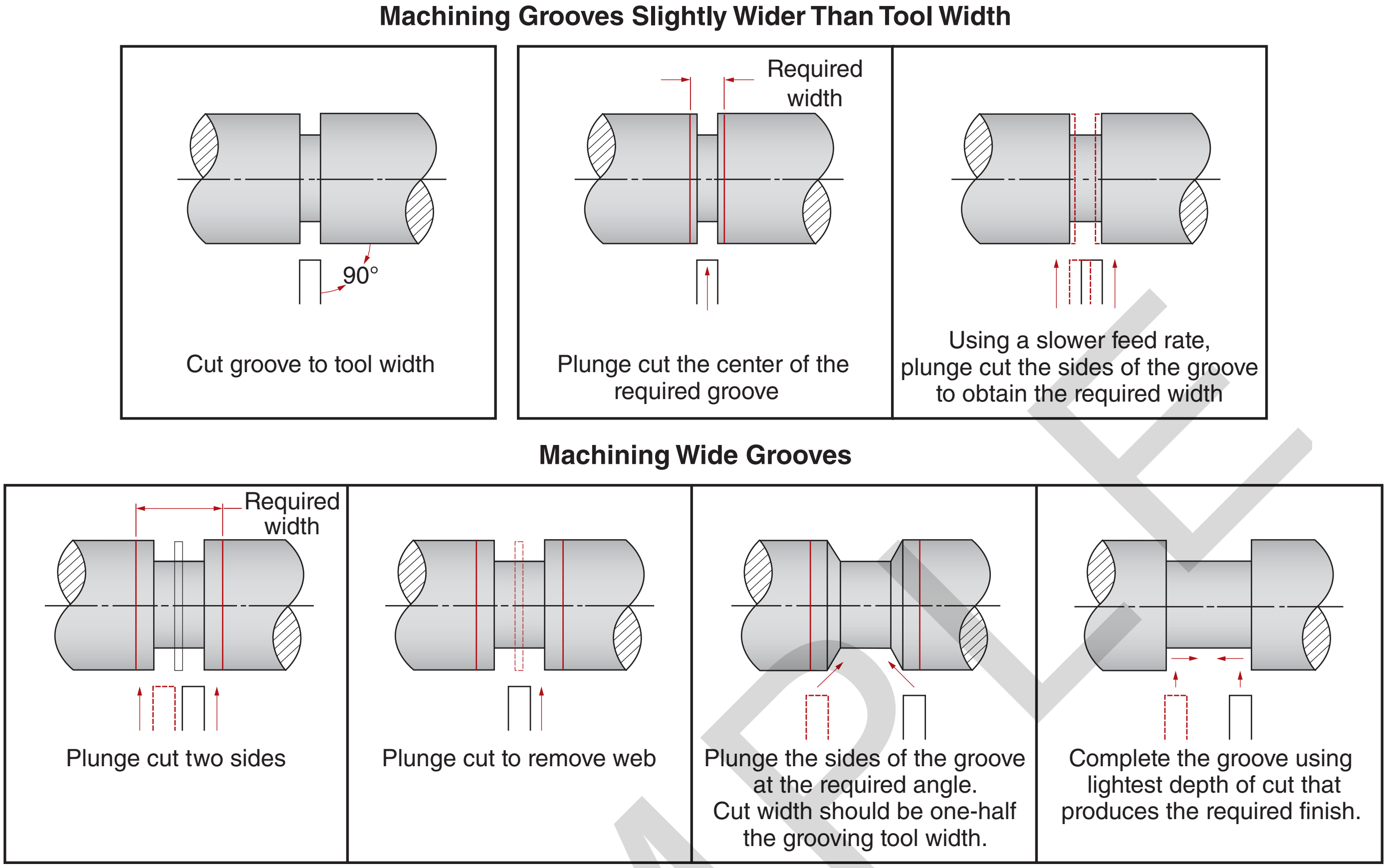
14.12 Gathering Information from Chips

Studying the chips created during the machining process can help indicate whether the parameters being used in the machining process have been properly selected. For example, long, stringy chips that do not break on themselves are usually a sign of a feed rate that has been set too slow. Ideally, chips should be of medium thickness and should break against themselves during the machining process. Chips that are thick and intertwine are usually a result of a feed rate that is too fast for the depth of cut selected. This can result in damage to the workpiece or to the tool.

Likewise, different lead angles can cause different chip formation. Small lead angles, with the tool at or near 90° to the center axis of the workpiece, create higher cutting forces that result in short, wide chips. Higher cutting angles, where the tool is at a 30° to 45° angle to the center axis of the workpiece, place less stress on the cutting surface. This results in smoother chips with less curl.

The color of the chips can also indicate problems with machining parameters. Different materials produce chips of different colors due to the chemical reactions that take place in the heat generated by the machining process. Ideally, in most steels, the chips that are generated should have a light brown tint. The chips then change to a blue or even purple hue, depending on the type of steel.

If the chip turns blue immediately after it clears the cutting area, the chip load and/or the feed rate is excessive. This indicates too much heat at the point of machining, and the workpiece is absorbing some of the heat, which can lead to straightness and diameter defects. Either the chip load or the feed rate must be changed so that less heat is generated. Alternatively, coolant can be used to help disburse some of the heat.



Goodheart-Willcox Publisher

Figure 14-87. These two techniques can be used to machine grooves of various widths using a parting tool.

Chapter Review

Summary

- Lathes operate by rotating the workpiece against the edge of a cutting tool.
- The major parts of a lathe include the parts that drive the lathe, the work-holding and rotating parts, and the parts used to hold, move, and guide the cutting tools.
- Work can be held on a lathe between centers, can be held in a chuck or a collet, or can be bolted to a faceplate.
- A variety of cutting tools and toolholders can be used on a lathe.
- Proper cutting speeds and rates of feed are very important to successful turning operations.
- Always properly maintain lathes and follow all safety precautions to ensure safe and efficient turning operations.
- Facing, facing to length, rough turning, finish turning, turning to a shoulder, parting, grooving, and necking are all operations that can be performed on a lathe.
- Information about the parameters used in the machining process can be obtained by studying the chips produced.

Review Questions

Answer the following questions using the information provided in this chapter.

Know and Understand

1. *True or False?* The basic principle of lathe operation is the rotation of the edge of a cutting tool against the work.
2. *True or False?* Lathe size is determined by the swing and the length of the bed.
3. The ways of a lathe bed _____.
   A. are flat or V-shaped bearing surfaces
   B. align and guide the movable part of the machine
   C. take up the entire bed length
   D. All of the above.

The carriage supports and controls the cutting tool and is composed of a number of parts. For questions 4–7, match each description with the correct term.

4. Fitted to the ways and slides along them
5. Permits transverse tool movement
6. Permits angular tool movement
7. Used to mount the cutting tool

A. Compound rest
B. Saddle
C. Tool post
D. Cross-slide

8. Before work can be mounted between centers, a _____ must be drilled at each end of the stock.
   A. pilot hole
   B. center hole
   C. counterbore
   D. None of the above.
9. *True or False?* Chucking is widely preferred because it is the most rapid method of mounting work.
10. *True or False?* The reversing feature of the 3-jaw chuck permits the holding of small-diameter work when reversed.
11. _____ is used to indicate the distance that the cutter moves in one revolution of the work.
   A. Cutting speed
   B. Spindle speed
   C. Feed rate
   D. Peripheral speed
12. Which of the following is *not* one of the four types of shoulders?
   A. Square
   B. Finished
   C. Filleted
   D. Undercut
13. When using the parting tool, the spindle speed of the machine is about _____ the speed used for conventional turning.
   A. one-third
   B. one-half
   C. one-fourth
   D. twice
14. Which of the following actions are considered dangerous when operating a lathe?
   A. Measuring with work rotating
   B. Operating lathe with most guards in place
   C. Using compressed air to clean machine
   D. All of the above.
15. Which operation machines the end of a workpiece square and reduces it to a specific length?
   A. Parting
   B. Facing
   C. Grooving
   D. None of the above.

16. Cutting speeds can be increased 300% to 400% by using _____.
    A. coolant
    B. cemented carbide cutting tools
    C. high speed steel cutting tools
    D. Any of the above.
17. For parting or grooving, a _____ is ground on top of the cutter to reduce chip width and prevent seizing.
    A. concave rake
    B. way
    C. neck groove
    D. mating thread
18. When examining the chips created by a machining process, you note that the chips are thick and intertwining. What does this tell you about the parameters used for the operation?
    A. The feed rate is too slow.
    B. The feed rate is too fast.
    C. The tool is dull.
    D. The spindle rpm is too slow.
19. *True or False?* Sleeves should be rolled up and all jewelry removed before beginning to use the lathe.
20. Accumulated metal chips and dirt are cleaned from the lathe with _____, *never* with your hands.
    A. a 2″ paintbrush
    B. a dust brush
    C. compressed air
    D. a vacuum
21. Some lathes come equipped with a _____, allowing for slower speeds with greater power.
    A. torque converter
    B. back gear
    C. Posi-Lock gear
    D. None of the above.
22. What determines the longest piece that can be turned between centers on a lathe?
    A. The swing
    B. The length of the bed
    C. The length of the bed minus the distance taken up by the headstock and tailstock
    D. None of the above.

Apply and Analyze

1. What is the spindle speed (rpm) required to finish-turn 2 1/2″ diameter aluminum alloy? A rate of 1000 fpm is the recommended speed for finish-turning the material.
2. What is the spindle speed (rpm) required to rough-turn 1″ diameter tool steel? The recommended rate for rough-turning the material is 50 fpm.
3. What spindle speed is required to finish-turn 200 mm diameter aluminum alloy? Recommended cutting speed for the material is 300 mpm.
4. Most work is machined while supported by one of four methods. List them.
5. When the work is turned between centers, a tapered piece will result if the centers are not aligned. Approximate alignment can be determined by two methods. What are they?
6. Briefly describe the three methods for checking center alignment if close tolerance work is to be done between centers.

Critical Thinking

1. Explain what cutting speed is and how it is used as a standard for spindle rpm calculations.
2. If a fellow machinist complains about the accuracy and movement of the cross-slide of a lathe carriage, what might you recommend as a first step in troubleshooting?