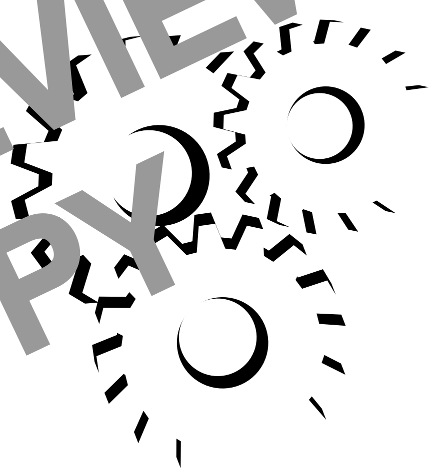
# Table of Contents

JOB ANALYSIS

Lesson One

# Machining Cylindrical Shapes





TPC Training Systems

32301

Lesson

# 1 Machining Cylindrical Shapes

## TOPICS

Turning
Shoulder Facing
Machining Fillets
Turning Relief Notches
Turning Tapered Profiles
Knurling
Filing on a Lathe
Facing
Boring and Counterboring
Boring Tapered Holes
Thread Cutting
Thread Forms
Job Analysis

## OBJECTIVES

**After studying this Lesson, you should be able to…**

- Explain the procedures for turning single and multiple diameters, including shoulders, fillets, and relief notches.
- Compute tapers, including the use of the setover method, compound rest method, and taper attachment method.
- Show how to use the cutting tools and machines for knurling, filing, and polishing.
- List the procedures for facing chucked work and work mounted between centers on the lathe.
- Describe the procedures for boring and counterboring in a lathe and boring mill.



## KEY TECHNICAL TERMS

**Turning** 1.01 any operation which reduces the diameter or changes the profile of a cylindrical workpiece

**Fillet** 1.10 a gradual change from one diameter to another at the shoulder on a workpiece

**Relief notch** 1.11 a groove or undercut in the workpiece used to separate two defined diameters

**Taper** 1.13 a progressive difference in diameter on a cylindrical workpiece

**Knurling** 1.21 a diamond-shaped pattern produced on the surface of cylinders such as those used as grips on hand tools

**Facing** 1.27 squaring the ends of a cylinder so that they are at right angles to the longitudinal axis of the workpiece

**Boring** 1.35 a process that produces a clean, straight hole in a previously drilled workpiece

**Many tasks you perform in the maintenance machine shop involve working with cylindrical workpieces. Shafts, collars, flanges, drums, and similar parts are all variations of cylinders. To repair or duplicate any of these parts, you must know which machines to use. This Unit will help you learn how to select the right machine for every operation.**

**This Lesson describes the operations most commonly performed on round stock. By the time you finish it, you will be able to decide which method(s) to use to make or modify a particular cylindrical piece. You will also learn several methods for holding the workpieces in particular operations. You will also learn how to produce unusual profiles and finishes with special tooling.**

## Turning

1.01 In a machine shop, the term *turning* describes any operation that reduces the diameter, or changes the profile, of a cylindrical workpiece. The work may be solid or hollow. It may be long, short, or irregular in shape, and it can be made of almost any material.

1.02 Large, cumbersome, or very heavy workpieces (such as pump or compressor housings) are normally turned on vertical boring mills. These are commonly called vertical lathes. However, most maintenance machine shop turning is done on the more popular engine lathe. When properly set up and operated, an engine lathe can produce many profiles, finishes, and special features on a workpiece.

1.03 **Straight turning.** *Straight turning* is one of the most common lathe operations encountered in a maintenance machine shop. In straight turning, the reduced finished diameter always stays the same along the total length of the workpiece. Straight turning is used to clean up raw stock, correct uneven dimensions, or prepare the face of a gear blank prior to cutting gear teeth.

1.04 The shaft shown in Fig. 1-1 has been turned down along its entire length. In this example, the original diameter of 4.50 inches has been reduced to a finished diameter of 4.375 inches. Such an operation requires straight turning, and is done on an engine lathe.

1.05 In straight turning, the cutting tool may either move across the length of the workpiece from left to right or it may move from right to left. If you cut from left to right, a left-hand tool must be used. If you cut from right to left, a right hand tool is used. Straight turning from right to left is the most common practice.

1.06 Another factor to consider when performing straight turning is the overall length and diameter of the cylinder. If you need to turn a long workpiece, make sure the lathe is large enough to handle it. If the workpiece is long and thin and needs to be turned between centers, use *a steady rest* to prevent whipping.

1.07 **Turning multiple diameters.** Many jobs require turning a part to produce more than one diameter. Examples include shafts, drums, and flanges. These operations are also commonly done on a lathe, but are more complex than straight turning.



**Fig. 1-1. Straight turning**

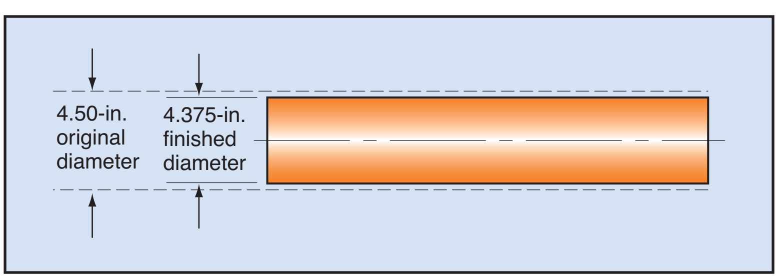
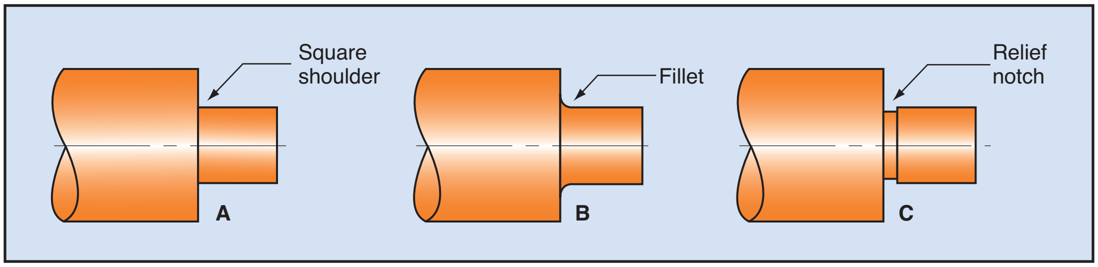


**Fig. 1-2. Examples of shoulders**

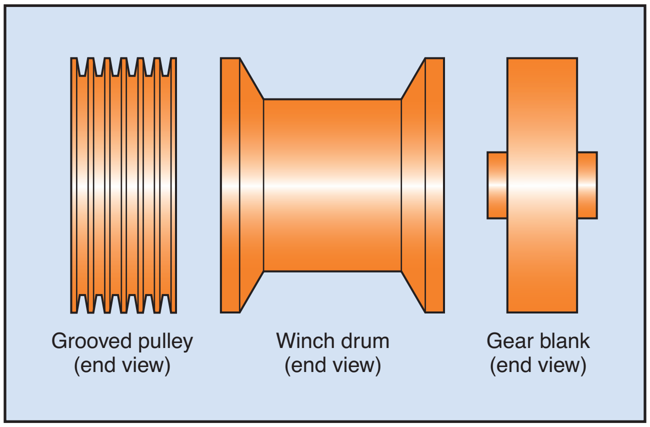



1.08 Production of more than one diameter requires that the lathe be stopped after each diameter is turned. You must also change the cutting tool setup, and, possibly, the speeds and feeds. In addition, changing diameters always creates a *shoulder* at the point of each change. Depending on the use of the finished part, this shoulder must be cut in one of several profiles. Figure 1-2 shows several common shoulder profiles.

### Shoulder Facing

1.09 The *square shoulder* illustrated in Fig. 1-2A is used when another machine part, such as a gear, bearing, or pulley, is mounted on the smaller diameter. The vertical surface of the large diameter's face must be very accurate when parts such as bearings are placed against it. After turning, this surface must be faced with the correct lathe cutting tool to ensure that the vertical surface is perfectly true and square with the horizontal axis. Tool selection will depend on whether it is a right-hand or left-hand cut.

**Fig. 1-3. Turning profiles**




### Machining Fillets

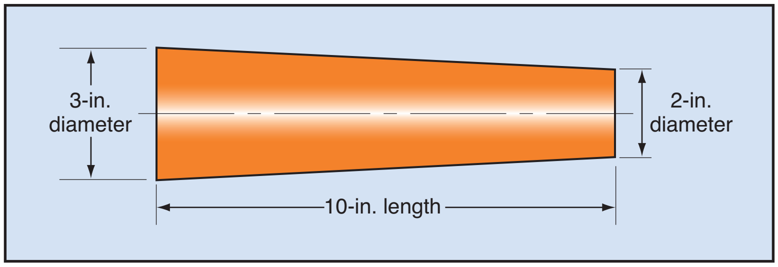
1.10 Figure 1-2B shows a *fillet*, or radius, at the point of diameter change. A fillet produces a gradual change from one diameter to another during turning. You can determine the size of the fillet by using a radius gauge or by checking specifications in a drawing. Fillets greatly reduce the material stresses caused by sharp corners. You can also use a fillet whenever another machine part does not have to fit snugly against a square shoulder face.

### Turning Relief Notches

1.11 The smaller diameter, as illustrated in Fig. 1-2C, is supposed to be ground to a very fine finish along its entire length. As shown, a *relief notch* has been cut into the work at the point of the diameter change to ensure that the grinding tool can advance right up to the square face of the larger diameter. This relief notch provides enough clearance for grinding the entire surface of the smaller diameter. Relief notches should be cut after you have completed finish turning the smaller diameter. You will need to use a cutoff tool for this operation. Try to make notches for other design requirements in the workpiece while the lathe is still set up for the turning work.

1.12 **Additional turning of cylinders.** As shown in Fig. 1-3, several other kinds of machine parts which can be made, or modified, by turning them on a lathe. Work done by turning on a lathe includes cutting grooves into pulley blanks, machining drums to provide a smooth rope or cable contact area, and truing gear blanks. Always study drawings or other specifications carefully to determine the type of turning required for the project at hand. Then decide on the particular type of machine needed to perform the work.

**Fig. 1-4. Taper**




## Turning Tapered Profiles

1.13 Many machines have tapered parts such as shafts, spindles, or axles. *Tapers* are used to provide friction fits between two locking parts. A tapered fit between pieces saves time if they are frequently joined and separated. You can turn and bore most tapered parts on an engine lathe. However, you must first compute the amount of taper required and express it either in inches of taper/inch, or inches of taper/foot. This ratio will enable you to select a lathe with the capacity to turn that particular amount of taper.

1.14 Always express taper in inches. A taper is a uniform increase or decrease in diameter over a given length parallel to its longitudinal axis. An example of a taper is a cone, in that there is a uniform change from its base diameter to its top. In the example shown in Fig. 1-4, a 10 in. shaft has a 3-in. diameter at one end and a diameter of 2 in. at the other. There is a uniform decrease in diameter or taper between these two dimensions of diameter. To compute the inches of taper/inch, use the following formula:

Taper = large diameter - small diameter

Taper = 3 in. - 2 in.

Taper = 1 in.

This 1 in. difference occurs over a 10 in. length. In order to find the amount of taper/inch, divide the difference in diameter (1 in.) by the length of the taper (10 in.):

Inches of taper/inch = 1 in. ÷ 10 in.

Inches of taper/inch = 0. 10

To compute the taper/foot, multiply taper/inch by 12. The inches of taper/foot of the 10 in. shaft shown in Fig. 1-4 equals:

Inches of taper/foot = taper/inch × 12

= 0.10 in. × 12

= 1.20 in.

Inches of taper/foot = 1.20

You can also compute taper when the angle of a taper is given. You will need a calculator with a trigonometry function. To do this, use the following formula:

Inches of taper/inch = 2 × tangent of 0.50 the angle

Inches of taper/inch = $2 \times \tan\left(\frac{A}{2}\right)$

Suppose the angle of taper is 4.25°. Using the trig function on your calculator, compute:

4.25 ÷ 2 = 2.125

press "tan"

= 0.037

Then

0.037 × 2 = 0.074 inches of taper/inch

1.15 After studying the drawing or specifications of the tapered part, compute the taper/inch or taper/foot. Then you must select the method for producing that taper on a lathe in your shop. There are three basic ways to cut tapers in a lathe.

1.16 **Tailstock setover method.** The tailstock setover method for turning tapers is most often used by experienced machinists. This method involves moving the tailstock of the lathe off its normal centerline. The tailstock setover method causes abnormal loadings on the lathe centers and can result in rapid wear of these parts. You must also make sure of proper clearance for the lathe dog when using the setover method.

1.17 Several trial cuts are necessary when the setover method is used. Therefore, time becomes a factor when making "breakdown" replacement parts. Furthermore, never use this method for turning tapers on a large number of identical pieces. It will affect the accuracy of finished parts and cause excessive wear to the lathe centers. Remember that tapers always vary in their length depending on the application. Consequently, the same center offset for one taper will not necessarily be the same setting for the same taper of different length. Once you have calculated the inches of taper/inch, the dead center offset is calculated in the following manner:

TPI = inches of taper/inch

TPF = inches of taper/foot

L = length of taper

$$\text{Offset} = \frac{\text{TPI} \times \text{L}}{2}$$

If you are working in taper/foot, use:

$$\text{Offset} = \frac{\text{TPF} \times \text{L}}{24}$$

**Fig. 1-5. Knurling tool**


1.18 **Compound rest method.** The compound rest on a lathe is used to hold the toolholder and the cutter. Its base can also swivel clockwise or counterclockwise to turn steeper tapers. The compound rest method is more accurate than the setover method. It is most often used for tapers larger than three inches/foot. Because the lathe centers are in line when using this method, they will not wear as much as they do in the tailstock setover method. When using this method, always consider that any angle of taper can be cut, but length is limited by the travel capability of the compound rest.

1.19 **Taper attachment method.** The taper attachment method is the most accurate way to taper longer workpieces. This method involves using a taper attachment which is a separate accessory for your lathe. The taper attachment method is commonly used for making duplicate pieces with tapers up to about three inches/foot. Remember that the length of taper using this method is limited to the taper attachment's length of travel. You will also need to consider the two types of taper attachments. The plain taper attachment is common and limited by its set length of travel. The second type of taper attachment is the telescopic taper attachment. Because of its design, it has a greater degree of travel and eliminates the need for the plain taper attachment. When using this method, your work may be either held between centers or held in a chuck. Always use the power feed on your lathe.



1.20 Before deciding which method of turning a taper to use, determine the amount of taper required on the work, the number of pieces to be made, the length of the workpiece, and the required accuracy of the taper. Be sure that an available lathe is equipped with a taper attachment (as optional equipment) when considering this method.

### Knurling

1.21 *Knurling* on a workpiece produces a diamond-shaped surface such as that found on handles or grips of various hand tools. Do this operation on a lathe using a special knurling tool like the one shown in Fig. 1-5. The knurling tool has two hardened rolls with right and left helically opposed teeth. These rolls must be harder than the workpiece material in order to roll the diamond pattern into the work surface.

1.22 Three grades, or degrees, of knurling are commonly used for tool grips. These are coarse, medium, and fine. Select the correct knurling tool for a given job by the degree or grade required for the project.

### Filing on a Lathe

1.23 Most filing operations done on a lathe "clean up" or remove sharp corners and burrs after machining. However, machine filing is also used to remove tool marks, prepare surfaces for polishing, or improve the fit of tapered parts.

1.24 Use relatively high lathe speeds for filing. Use special lathe files to remove larger amounts of stock. Mount the work in the lathe between centers or hold it in a chuck.

1.25 **Polishing.** After filing, use an emery cloth to produce a smooth, mirrorlike finish. When polishing a piece that has been prepared by filing, you must use higher speeds than those used during filing.

1.26 Polishing is usually required on shaft journals where softer elements, such as rubber seals or felt gaskets, contact the shaft. In such cases, very smooth surfaces are needed to protect the seal materials.



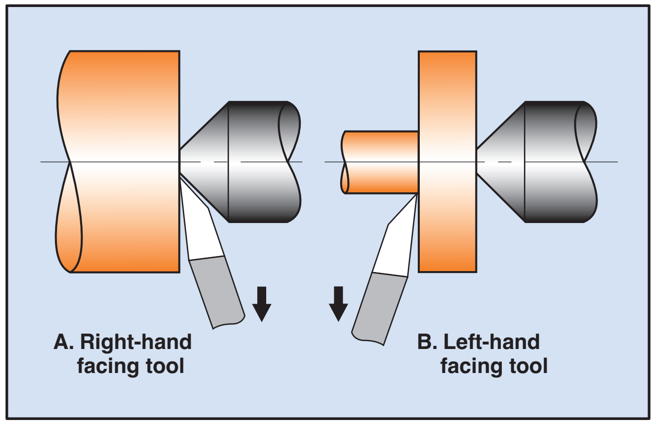
### Facing

1.27 *Facing* cylindrical work is another operation you will frequently encounter in a maintenance machine shop. Facing involves squaring the ends of a workpiece so that they are at right angles to the longitudinal axis of the work. Facing is also used to finish a workpiece to a specific length. Figure 1-6 shows how facing is done on the end of a workpiece held between lathe centers.

1.28 **Facing short cylinders.** The flange and the collar shown in Fig. 1-7 are examples of shorter cylindrical workpieces. Face the ends of these, and similar parts, to make them perfectly square so they will fit properly with other components. Facing short work is also done to repair, or renew, damaged end surfaces.

1.29 You must decide how to hold the workpiece when facing shorter work in a lathe. Generally, hold shorter pieces in a chuck, in a collet, or mounted on a mandrel for facing. Study the drawing or other specifications carefully to help you decide the best method

**Fig. 1-6. Right- and left-hand facing**




for holding work. Overall length of the workpiece and its general configuration are important factors when deciding how to hold the work.

1.30 **Facing multiple surfaces.** The gear blank shown in Fig. 1-8 on the following page is mounted on a lathe mandrel. Both ends of its hub and rim are supposed to be squared with its axis. You can perform these operations without disturbing the setup of the workpiece after turning the outside diameter. Facing the hub and the rim requires only a change of cutter and cutter location after turning.

1.31 **Facing between centers.** After finish turning a workpiece between centers on a lathe, you may need to square the ends with the work axis. You may also need to reduce the piece to its desired finished length. Face the ends of the work while it is still mounted in the lathe between centers. Center alignment is crucial. If it is poor, the ends will not be faced. They could turn out concave or convex due to misalignment of centers. Use a steady rest when you are holding long, thin workpieces between the centers.

**Fig. 1-7. Short cylindrical workpieces**

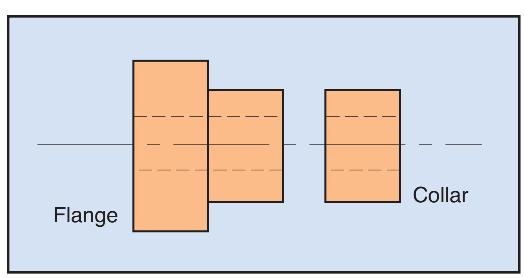


**Fig. 1-8. Facing multiple surfaces**

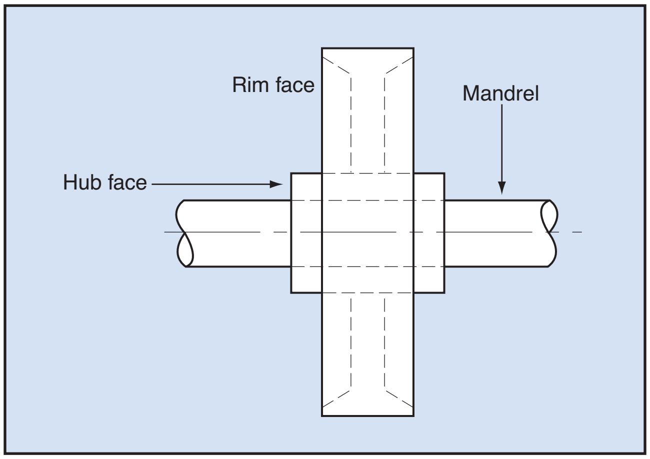
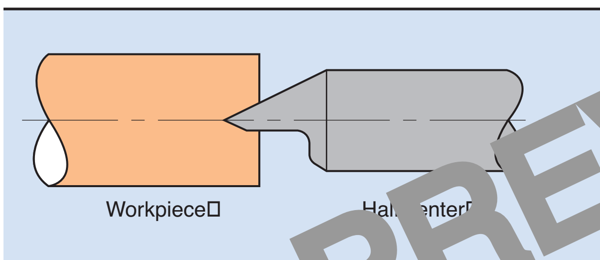



**Fig. 1-9. Workpiece held by half center**




1.32 Use a half-center mounted in the tailstock when facing the ends of work mounted between centers. The half-center, shown in Fig. 1-9, has half of its tip ground away. This allows the cutting tool to be fed right up to the center hole in the work. This also eliminates cutter breakage from contact with the dead center. Never use a half-center for turning diameters on a lathe.

1.33 **Facing chucked work.** Hold shorter work in a chuck for facing. However, do the facing on these pieces before drilling, boring, or reaming. Facing provides clean, square surfaces for locating and inserting drilling or boring tools.

1.34 As when turning work between lathe centers, you can make both roughing cuts and finishing cuts with the workpiece held in a chuck. Also, you can cut off the unwanted stock when the work is chucked. Cutting off is a form of facing, but it should *never* be done when the work is held between centers.

**The Programmed Exercises on the next page will tell you how well you understand the material you have just read. Before starting the exercises, remove the REVEAL KEY from the back of your Book. Read the instructions printed on the Reveal Key. Follow these instructions as you work through the Programmed Exercises.**

| | |
|---|---|
| 1-1. A machining operation which reduces the diameter of a cylindrical workpiece is called __________. | 1-1. TURNING<br>Ref: 1.01 |
| 1-2. A gradual change from one diameter to another on the same workpiece is called a(n) __________. | 1-2. FILLET<br>Ref: 1.10 |
| 1-3. A grinding tool can be advanced to a square face by turning a __________ in the work. | 1-3. RELIEF NOTCH<br>Ref: 1.11 |
| 1-4. Taper is always expressed in __________. | 1-4. INCHES<br>Ref: 1.14 |
| 1-5. The setover method for turning tapers involves moving the __________ of the lathe | 1-5. TAILSTOCK<br>Ref: 1.16 |
| 1-6. A diamond-shaped pattern is produced by a __________ tool. | 1-6. KNURLING<br>Ref: 1.21 |
| 1-7. After turning a piece on a lathe, sharp corners and burrs are removed by __________. | 1-7. FILING<br>Ref: 1.23 |
| 1-8. The operation used to square the ends of a workpiece is called __________. | 1-8. FACING<br>Ref: 1.27 |

## Boring and Counterboring

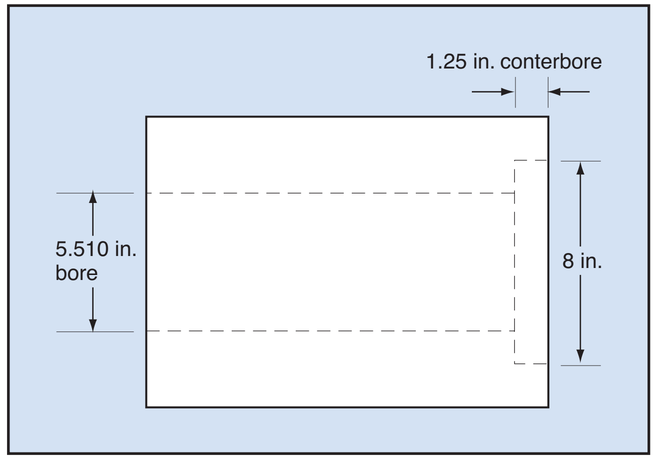
1.35 *Boring* produces a clean, straight hole in work. Boring is normally done to enlarge and accurately size a hole previously made by drilling or during casting. Boring is the opposite of a turning operation. Turning reduces the outside diameter of the work. Boring enlarges the inside diameter.

1.36 Use single-point cutting tools for boring. Small- to medium-size cylindrical work can usually be bored on a lathe. Large workpieces, irregularly shaped workpieces, or workpieces that are very heavy are usually bored on a horizontal boring mill. A thorough analysis of the size and shape of the workpiece will help you decide which kind of machine to use.

1.37 **Counterboring.** *Counterboring* enlarges bored holes to a specific depth. The workpiece shown in Fig. 1-10 has been counterbored to a diameter of 8 in. and to a depth of 1.25 in. This operation was done after the bore of 5.510 in. was machined in the work.

1.38 Always counterbore the workpiece in the same lathe setup used for the primary boring operation. This ensures concentricity of the bored and counterbored holes.

**Fig. 1-10. Counterboring**

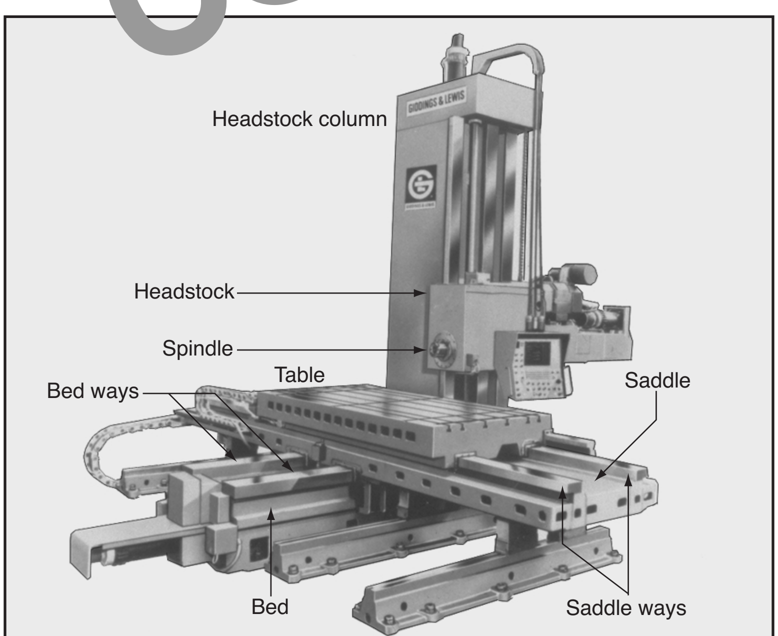



1.39 **Boring large and irregularly shaped workpieces.** If the size and shape of a workpiece preclude mounting in a lathe, boring must be done on a horizontal boring mill. Figure 1-11 shows a table-type horizontal boring mill. This is the most common boring mill used in the maintenance machine shop.

1.40 The workpiece is held stationary on the machine table while the cutting tool rotates against the inside wall of the hole. Use boring mills to make accurate holes in very large workpieces such as engine blocks, valve housings, and transmission cases.



**Fig. 1-11. Horizontal boring mill**



1.41 You should also use boring mills to machine two or more holes in close alignment on a common axis. Use only one setup to do this. Figure 1-12 shows such a project with the cutting tool just entering the last hole in line. Various methods and accessories can be used to mount and hold cumbersome workpieces on the table of a boring mill. In addition, most horizontal boring mills can be equipped with a rotary, or pivoting, table. This feature allows the work to be rotated and repositioned for various operations without unclamping it.

1.42 **Boring on a lathe.** Most maintenance machine shop boring work can be done on a lathe. Hold the workpiece in a chuck or mount it on a faceplate. A stationary cutting tool is held against the hole in the rotating work. This, of course, is different from the boring mill operation, where the tool rotates and the work is held stationary.

1.43 As the workpiece rotates with the lathe spindle and chuck, the boring tool advances into the hole to produce the required diameter and depth of the bore. Operation of the lathe's longitudinal feed controls the rate of tool entry into the hole. Generally, use a medium feed for rough boring operations.

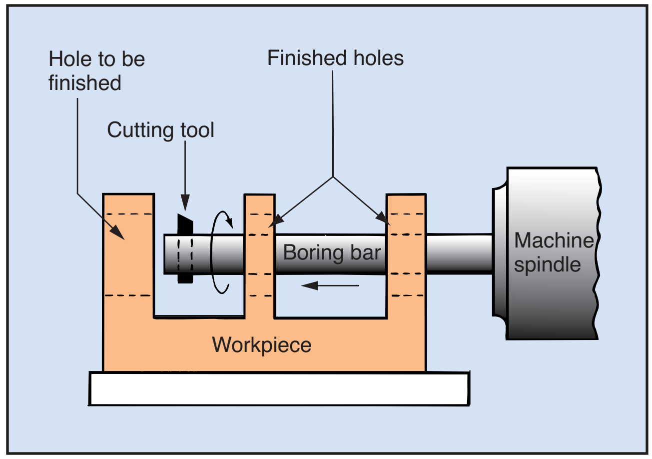
### Boring Tapered Holes

1.44 There are two methods used for boring tapered holes on a lathe. Do not use the tailstock setover method for turning tapered bores. The tailstock setover method cannot be used because of the methods used to hold the workpiece during boring.

1.45 **Compound rest method for boring tapers.** When boring a short, steep taper in a workpiece, the boring tool and holder are mounted on the lathe's compound rest. The compound rest and the boring tool are both set at the desired taper angle. The predrilled straight hole must be large enough to clear the boring tool as it enters the hole.

1.46 **Taper attachment method for boring tapers.** Using the taper attachment on a lathe is the most accurate method for boring tapered holes. If you must hold the tapered bore to very close tolerances, use this method. Hold the work in a chuck, and set the taper attachment for the desired angle. The crossfeed mechanism of the lathe controls the depth of the taper when a telescopic taper attachment is used.

**Fig. 1-12. Multiple boring on horizontal boring mill**




1.47 When boring tapered holes, frequently check the hole and its mating part. This ensures a proper fit between the two.



### Thread Cutting

1.48 The engine lathe is the most popular and effective machine tool used to cut screw threads. Engine lathes are equipped with special features designed exclusively for this purpose. They produce extremely accurate threads in either metric or customary systems of measurement.

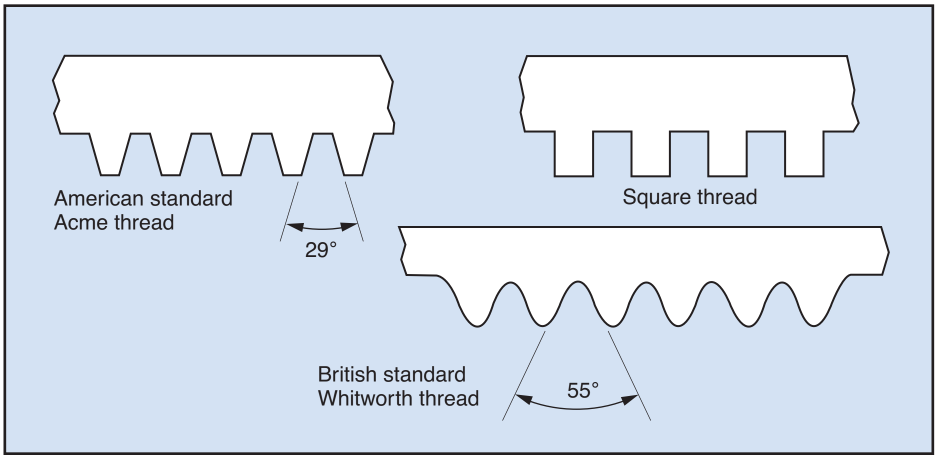
1.49 Thread cutting in a lathe is sometimes called "thread chasing." Use single-point cutting tools for this purpose. This holds tool upkeep and initial cost to a minimum. A lathe's capability to cut threads often eliminates the need to move a workpiece to another machine for this operation. Cut the threads in the workpiece after turning, boring, or facing the piece in the lathe.

1.50 Both external and internal threads can be cut on a lathe. In addition, you can machine single and multiple threads, as well as make tapered threads on a lathe.

### Thread Forms

1.51 Figure 1-13 on the following page shows several common screw thread forms which can be cut on a lathe. These are in addition to the popular American Standard Unified and National Thread classes.

Fig. 1-13. Common screw threads




1.52 The profile of the cutting tool tip controls the shape of the threads to be cut. Adjust the lathe gearing and feed mechanisms to suit the particular thread cutting operation. Hold the workpiece in a chuck or collet, between centers, or mounted on a faceplate.

1.53 **Identifying threads.** The first and most important thing to do before selecting and setting up a lathe for thread cutting is to correctly identify the thread. If you are working from a drawing or print, the specifications will usually give the thread size, type, and general classification.

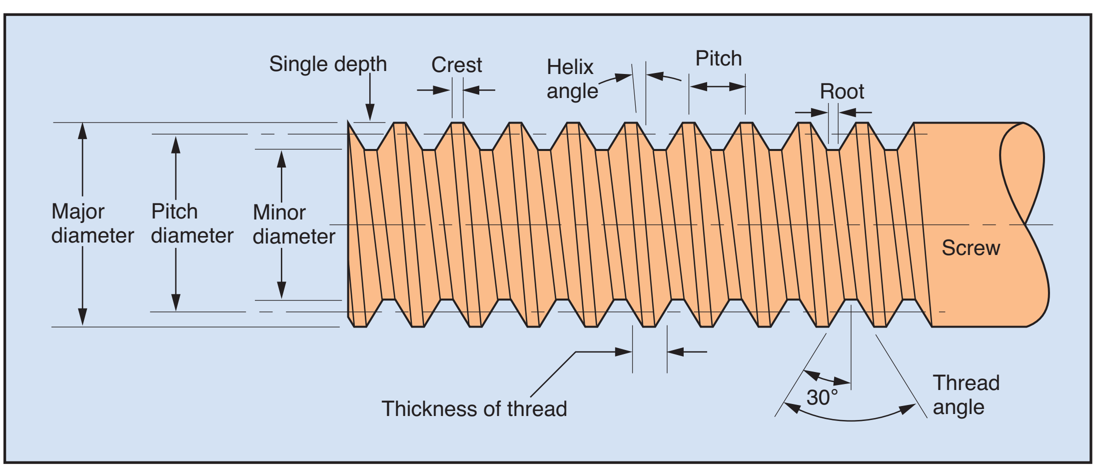
1.54 Duplicating threads on a new part using the old piece as a reference is a little more difficult. This requires measuring the old work carefully to determine all of the thread characteristics. Correctly identify the particular thread for your job by comparing these measurements with specification tables and major dimensions published in machinists' handbooks. There are also many useful gauges and templates available for this purpose.

1.55 It is wise to measure the thread of the workpiece to be duplicated with precision instruments if there is any doubt about the size or type to be machined. The naked eye or simple scales sometimes provide wrong readings, which results in turning the wrong thread for a job.

1.56 Figure 1-14 shows the important dimensions and terms relating to a common screw thread. While the thread profiles differ among the various classes and types of threads, the basic terminology is the same.



Fig. 1-14. Screw thread terms



### Job Analysis

1.57 The workpiece shown in Fig. 1-15 initially appears to be a rather complex part. Machining this piece to produce the shape and profile shown involves numerous operations and techniques. Yet the part can be made completely as shown on a lathe using different setups and tooling.

1.58 Upon careful analysis, you will see that this part is actually composed of several cylinders of various shapes and sizes. Study each cylinder separately and analyze the requirements to machine that portion. Then develop a list of operations needed for that particular area.

1.59 For example, the right-hand extension of the shaft shown in Fig. 1-15 is to be tapered. The extreme end of this extension must be threaded to specification. A fillet of 0.25 in. must be machined into it where the diameter changes to 4.9375 in. That end must also be faced.

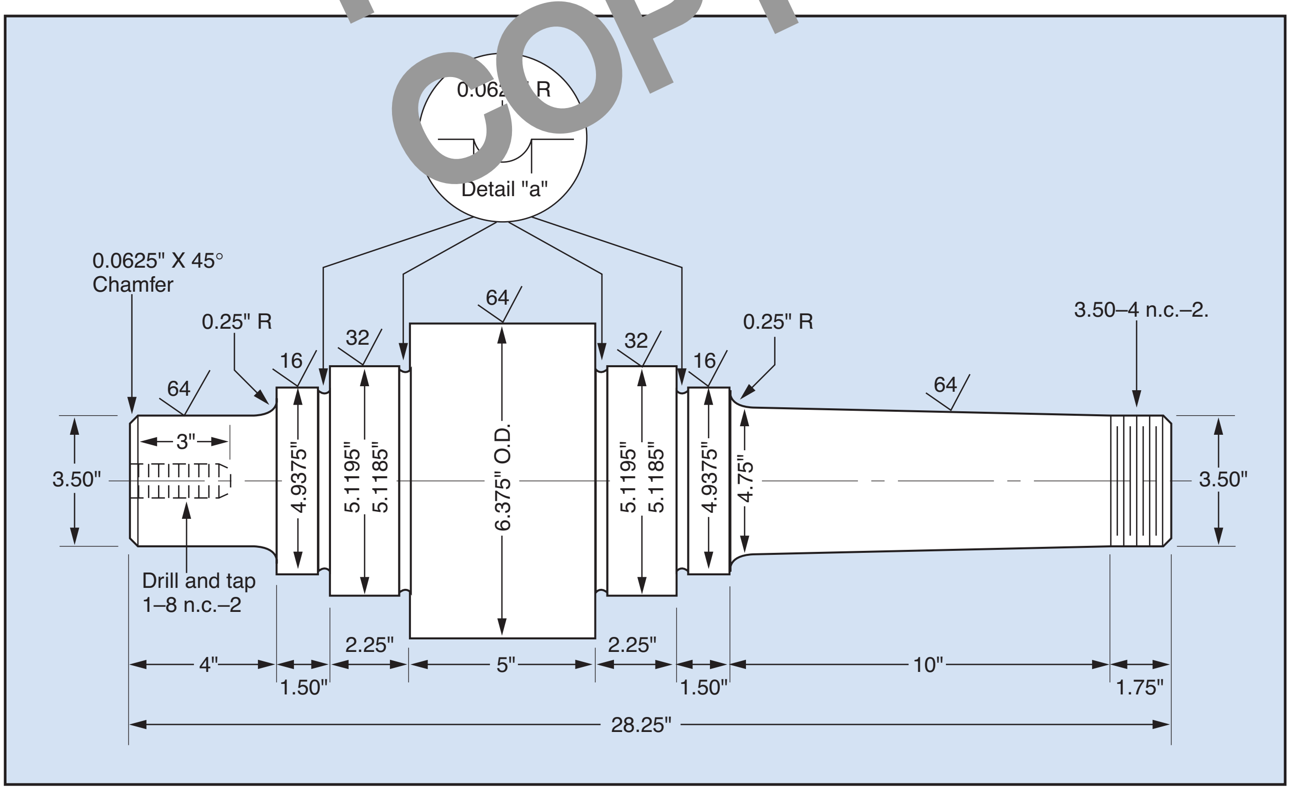
1.60 The opposite end of the shaft has an extension 4 in. long which must be drilled and tapped to specification. The tapping operation is the only procedure that will be performed after the piece is finished on the lathe.

1.61 Study Fig. 1-15 closely and you will see that cylinders of various lengths and diameters make up the rest of the shaft. You will make each individual cylinder by turning the workpiece in the lathe to meet the required specifications for finish and facing. You will find that the technique of dividing a complex part into separate sections in order to analyze a job is quite effective. It will be helpful in your machining work on cylindrical, as well as other, workpieces.

1.62 Boring operations, whether they are done on a lathe or a boring mill, can be broken down in a similar manner. First, decide which of these machines is best suited for the job at hand. Carefully analyze the total job in light of the type and number of operations needed to make a part. Compare these with the available machines in your maintenance machine shop. Keep in mind that many basic machines can be equipped with various attachments which greatly expand their ranges of operation.



**Fig. 1-15. Job analysis of workplace**



1.63 On the other hand, never attempt to do a job on a machine tool when the equipment is not intended for that purpose. Do not try to make unorthodox modifications to a lathe or boring machine in an attempt to adapt it to a particular project.

1.64 Always select the proper setup and working tools. Be certain they are in the best operating condition. If you have any doubts about the overall condition of a machine, consult with the last operator for his or her opinion.

1.65 The material and other characteristics of the workpiece are important considerations when analyzing a project before machining. These considerations will affect the selection of the correct tooling for the job as well as the setup and operation of the machine. A clear understanding of what is to be done is essential before deciding how to do a job.

| | |
|---|---|
| 1-9. When facing work between lathe centers, you should use a __________ mounted in the tailstock. | 1-9. HALF-CENTER<br>Ref: 1.32 |
| 1-10. When facing shorter workpieces on a lathe, the work should be held in a __________. | 1-10. CHUCK<br>Ref: 1.33 |
| 1-11. Work should never be held between lathe centers when __________. | 1-11. CUTTING OFF<br>Ref: 1.34 |
| 1-12. Large or irregularly shaped workpieces are usually bored on a(n) __________. | 1-12. HORIZONTAL BORING MILL<br>Ref: 1.36 |
| 1-13. Enlarging a bored hole to a specific depth is called __________. | 1-13. COUNTERBORING<br>Ref: 1.37 |
| 1-14. When boring on a lathe, the workpiece is held in a chuck or mounted on a __________. | 1-14. FACEPLATE<br>Ref: 1.42 |
| 1-15. The most accurate method of boring tapers on a lathe is by using the __________. | 1-15. TAPER ATTACHMENT<br>Ref: 1.46 |
| 1-16. Cutting threads on a lathe is also called thread __________. | 1-16. CHASING<br>Ref: 1.49 |

**Answer the following questions by marking an "X" in the box next to the best answer.**

1-1. Machining work to the same outside diameter along its entire length is called

- ☐ a. boring
- ☐ b. tapering
- ☐ c. straight turning
- ☐ d. shoulder turning

1-2. Which of the following can be cut at the point of a change in diameters?

- ☐ a Relief notch
- ☐ b. Shoulder
- ☐ c. Fillet
- ☐ d. All of the above

1-3. The most accurate method for turning tapers on longer workpieces is the

- ☐ a. tailstock setover method
- ☐ b. taper attachment method
- ☐ c. compound rest method
- ☐ d. cutoff method

1-4. When facing shorter work, the piece is held in a chuck or collet, or it is mounted

- ☐ a. between centers
- ☐ b. on a mandrel
- ☐ c. on a spindle
- ☐ d. on the steady rest

1-5. The half-center should be used when

- ☐ a. facing work between centers
- ☐ b. turning diameters on a lathe
- ☐ c. cutting relief notches and fillets
- ☐ d. all of the above

1-6. Which of the following tasks should be completed first?

- ☐ a. Drilling
- ☐ b. Boring
- ☐ c. Facing
- ☐ d. Reaming

1-7. To ensure concentricity of the bored and counterbored holes,

- ☐ a. use the same lathe setup for both
- ☐ b. do not face workpieces first
- ☐ c. use the same cutting tools for both
- ☐ d. do not use single-point cutting tools

1-8. When boring two or more holes in close alignment on a common axis, use a(n)

- ☐ a. lathe with a compound rest
- ☐ b. slab mill
- ☐ c. horizontal boring mill
- ☐ d. ball mill

1-9. The tailstock setover method for turning tapers cannot be used for ________ tapers.

- ☐ a. smaller
- ☐ b. boring
- ☐ c. polishing
- ☐ d. grinding

1-10. The shape and form of screw threads cut on a lathe is controlled by the profile of the

- ☐ a. lead screw
- ☐ b. dead center
- ☐ c. cutter tip
- ☐ d. cross feed

## SUMMARY

**Cylinders of all shapes and sizes will be machined in your maintenance machine shop. The machinist must know what specific cutting operations are needed as well as what specific machines to use. It is also necessary to know what device to use for holding the workpiece during these cutting operations.**

**The majority of work on cylinders will affect the end workpiece in two ways. The profile is changed because the outside diameter has been reduced or changed through turning. The other effect is just the opposite. The inside diameter of a hollow or previously drilled cylinder is enlarged from boring or counterboring.**

**A careful analysis of the project's drawings is necessary for correct machining. You must know the individual cutting technique for each particular finished shape on the cylinder's profile.**

### Answers to Self-Check Quiz

1-1. c. Straight turning. Ref: 1.03

1-2. d. All of the above. Ref: Fig. 1-2

1-3. b. Taper attachment method. Ref: 1.19

1-4. b. On a mandrel. Ref: 1.29

1-5. a. Facing work between centers. Ref: 1.32

1-6. c. Facing. Ref: 1.33

1-7. a. Use the same lathe setup for both. Ref: 1.38

1-8. c. Horizontal boring mill. Ref: 1.41

1-9. b. Boring. Ref: 1.44

1-10. c. Cuttertip. Ref: 1.52



**Contributions from the following sources are appreciated:**

Figure 1-5. Armstrong Bros. Tool Co.
Figure 1-11. Giddings & Lewis Machine Co.