Machining Fundamentals

From Basic to Advanced Techniques

by

John R. Walker

Publisher
The Goodheart-Willcox Company, Inc.
Tinley Park, Illinois

Library of Congress Card Catalog Number 99-17776
International Standard Book Number 1-56637-662-9

3 4 5 6 7 8 9 10 00 03 02 01 00

Cover photo ©Westlight (K. Tiedge)

Library of Congress Cataloging in Publication Data

Walker, John R.
Machining Fundamentals: from basic to advanced techniques /
by John R. Walker

p. cm.
Includes index.
ISBN 1-56637-662-9
1. Machine-shop practice. 2. Machining.
I. Title.
TJ1160.W25 2000
671.3'5--dc21

99-17776
CIP

Introduction

Machinists are highly skilled men and women. They use drawings, hand tools, precision measuring tools, drilling machines, grinders, lathes, milling machines, and other specialized machine tools to shape and finish metal and nonmetal parts. Machinists must have a sound understanding of basic and advanced machining technology, which includes:

- Proficiency in safely operating machine tools of various types (manual, automatic, and computer controlled).
- Knowledge of the working properties of metals and nonmetals.
- The academic skills (math, science, English, print reading, metallurgy, etc.) needed to make precision layouts and machine set-ups.

Machining Fundamentals provides an introduction to this important area of manufacturing technology. The text explains the "how, why, and when" of numerous machining operations, set-ups, and procedures. Through it, you will learn how machine tools operate and when to use one particular machine instead of another. The advantages and disadvantages of various machining techniques are discussed, along with their suitability for particular applications.

Machining Fundamentals details the many common methods of machining and shaping parts to meet given specifications. It also covers newer processes such as laser machining and welding, water-jet cutting, high-energy-rate forming (HERF), cryogenics, chipless machining, electrical discharge machining (EDM), electrochemical machining (ECM), robotics, and rapid prototyping. The importance of computer numerical control (CNC) in the operation of most machine tools, and its role in automated manufacturing is explored thoroughly.

This new edition of ***Machining Fundamentals*** has many features that make it easy to read and understand. A numbering system for headings has been adopted to make it easier to locate information in a chapter. Learning objectives are presented at the beginning of each chapter, along with a list of selected technical terms important to understanding the material in that chapter. Throughout the book, technical terms are highlighted in bold italic type as they are introduced and defined. Several hundred of these terms are also listed and defined in a *Glossary of Technical Terms* at the end of this text. Review questions covering the content taught are presented at the end of each chapter.

Color is employed extensively in this new edition to enhance understanding and to emphasize safety precautions. A consistent color coding has been employed in the hundreds of line illustrations (most made especially for the text) to help you visualize more clearly the machining operations and procedures. Many of the black and white photographs in the text have been replaced with new, full-color photos showing the most current types of equipment and processes.

Machining Fundamentals is a valuable guide to anyone interested in machining, since the procedures and techniques presented have been drawn from all areas of machining technology.

John R. Walker

Machining Fundamentals Color Key

Colors are used throughout *Machining Fundamentals* to indicate various materials or equipment features. The following key shows what each color represents.

- ◼ **Metals (surfaces)**
- ⬜ **Metals (in section)**
- 🟦 **Machines/machine parts**
- 🟩 **Tools**
- 🟩 **Cutting edges**
- 🟪 **Work-holding and tool-holding devices**
- 🟨 **Rulers and measuring devices**
- 🟥 **Direction or force arrows, dimensional information**
- 🟨 **Fasteners**
- 🟫 **Abrasives**
- 🟩 **Fluids**
- 🟧 **Miscellaneous**

IMPORTANT SAFETY NOTICE

Work procedures and shop practices described in this book are effective, but general, methods of performing given operations. Always use special tools and equipment as recommended. Carefully follow all safety warnings and cautions (they are printed in red type for greater legibility). Note that these warnings are *not* exhaustive. Proceed with care and under proper supervision to minimize the risk of personal injury or injury to others. Also follow specific equipment operating instructions.

This book contains the most complete and accurate information that could be obtained from various authoritative sources at the time of publication. Goodheart-Willcox Publisher cannot assume responsibility for any changes, errors, or omissions.

Contents

Chapter 6. Hand Tools 91

Chapter 7. Fasteners 127

Chapter 8. Jigs and Fixtures 143

Chapter 9. Cutting Fluids 149

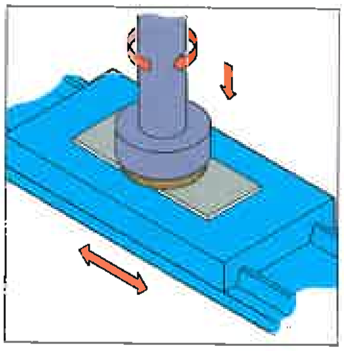
Chapter 10. Drills and Drilling Machines 153

Chapter 11. Offhand Grinding 183

Chapter 17. The Milling Machine 285

Chapter 18. Milling Machine Operations 317

Chapter 19. Precision Grinding 353



Chapter 20. Band Machining 383

Chapter 28. Nontraditional Machining Techniques 511

Chapter 29. Other Processes 525

Chapter 30. Occupations in Machining Technology. 547

Chapter 1

An Introduction to Machining Technology

LEARNING OBJECTIVES

After studying this chapter, you will be able to:

- ○ Discuss how modern machine technology affects the workforce.
- ○ Give a brief explanation of the evolution of machine tools.
- ○ Provide an overview of machining processes.
- ○ Explain how CNC machining equipment operates.
- ○ Describe the role of the machinist.

IMPORTANT TERMS

band machining
computer numerical control (CNC)
drill press
lathe
machine tools
machinist
milling machine
numerical control (NC)
precision grinding
skill standards

Figure 1-1. *Machine tools have made it possible to manufacture parts with the precision and speed necessary for low-cost mass production. Without machine tools, most products on the market today would not be available or affordable. (Courtesy of SURFCAM by Surfware)*

A study of technology will show that industry has progressed from the time when everything was made by hand to the present fully automated manufacturing of products. Machine tools have played an essential role in all technological advances.

Without machine tools, **Figure 1-1**, there would be no airplanes, automobiles, television sets, or computers. Many of the other industrial, medical, recreational, and domestic products we take for granted would not have been developed. For example, if machine tools were not available to manufacture tractors and farming implements, farmers might still be plowing with oxen and hand-forged plowshares.

It is difficult to name a product that does not require, either directly or indirectly, the use of a machine tool somewhere in its manufacture. Today, no country can hope to compete successfully in a global economy without making use of the most advanced machine tools.

There is one very important point that must be emphasized concerning modern manufacturing technology. The high-paying skilled jobs in manufacturing, such as tool-and-die making and precision machining, *require aptitudes comparable to those of college graduates*. Jobs that require few or no skills have almost disappeared.

1.1 THE EVOLUTION OF MACHINE TOOLS

Machine tools are the class of machines which, taken as a group, can reproduce themselves (manufacture other machine tools). There are many variations of each type of machine tool, and they are available in many sizes. Tools range from those small enough to fit on a bench top to machines weighing several hundred tons.

The evolution of machine tools is somewhat akin to the old question, "Which came first, the chicken or the egg?" You could also ask, "How could there be machine tools when there were no machine tools to make them?"

1.1.1 Early Machine Tools

The first machine tools, the bow lathe and bow drill, were hand-made. They have been dated back to about 1200 BC. Until the end of the 17th Century, the lathe could only be used to turn softer materials, such as wood, ivory, or at most, soft metals like lead or copper. All of them were human-powered. Eventually, the bow lathe with its ***reciprocating*** (back-and-forth) ***motion*** gave way to treadle power, which made possible work rotation that was continuous in one direction. Later, machines were powered by a "great wheel" turned by flowing water or by a person or animal walking on a treadmill. Power was transmitted from the wheel to one or more machines by a belt and pulley system.

When inventor James Watt first experimented with his steam engine, the need for perfectly bored cylinders soon became apparent. This brought about the development of the first *true* machine tool. It was a form of the lathe and was called a "boring mill," **Figure 1-2.** The water-powered tool was developed in 1774 by Englishman John Wilkinson.

This machine was capable of turning a cylinder 36" in diameter to an accuracy of a "thin-worn shilling"(an English coin). However, operation of the boring mill, like all metal cutting lathes at the time, was hampered by the lack of tool control. The "mechanic" (the first ***machinist***) had to unbolt and reposition the cutting tool after each cut.

About 1800, the first lathe capable of cutting accurate screw threads was designed and constructed by Henry Maudslay, an English master mechanic and machine toolmaker. As shown in **Figure 1-3,** a hand-made screw thread was geared to the spindle and moved a cutting tool along the work. Maudslay also devised a slide rest and fitted it to his lathe. It allowed the cutting tool to be accurately repositioned after each cut. Maudslay's lathe is considered the "granddaddy" of all modern chip-making machine tools.

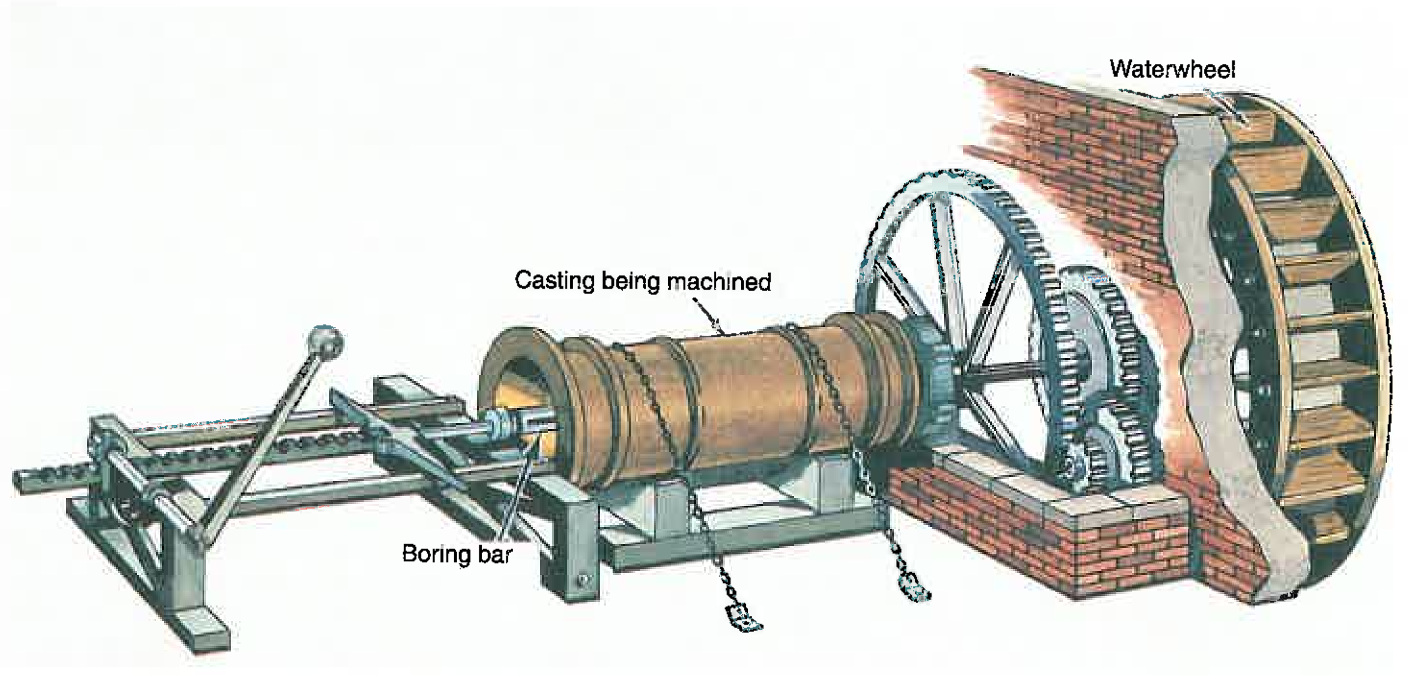


Figure 1-2. The first true machine tool is thought to be the boring mill invented by John Wilkinson in 1774. It enabled James Watt to complete the first successful steam engine. The boring bar was rigidly supported at both ends, and was rotated by waterpower. It could bore a 36" diameter cylinder to an accuracy of less than 1/16". (DoALL Co.)

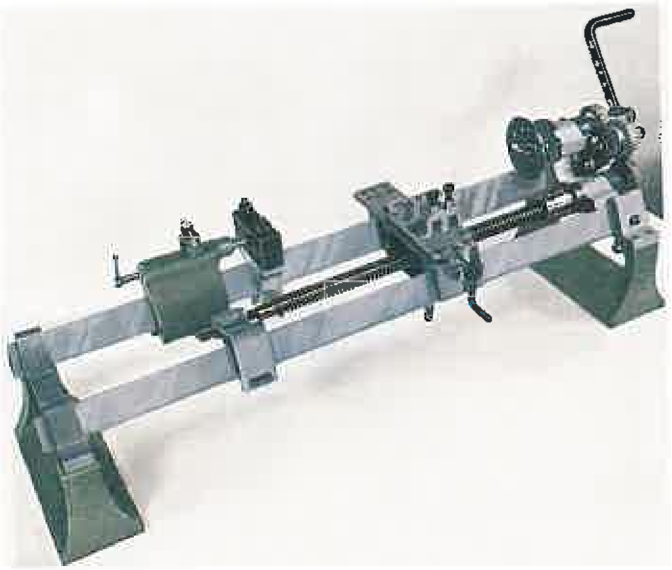


Figure 1-3. *Henry Maudslay's screw-cutting lathe. This machine tool, constructed on a heavy frame, combined a master lead screw and a movable slide rest. The lead screw had to be changed when a different thread pitch was required. (DoALL Co.)*

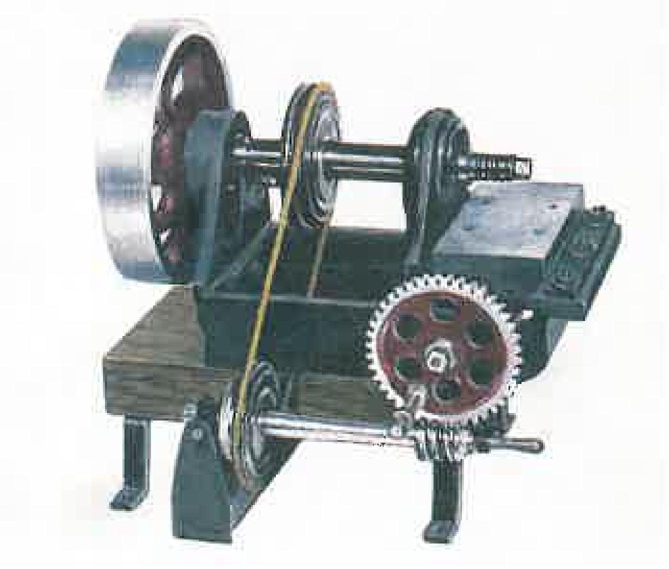


Figure 1-4. *One of the first practical milling machines manufactured in America. Eli Whitney used it and similar machines to mass-produce musket parts that were interchangeable. (DoALL Co.)*

In retrospect, the Industrial Revolution could not have taken place if there had not been a cheap, convenient source of power: the *steam engine*. Until the advent of the steam engine, industry had to locate near sources of water power. This was often some distance from raw materials and workers. With cheap power, industry could locate where workers were plentiful and where the products they produced were needed. The steam engine, in turn, would not have been possible without machine tools. Until the boring mill and lathe were developed to the point where metal could be machined with some degree of accuracy, there could be no steam engine.

The milling machine was the next important development in machine tools. It also evolved from the lathe. In 1820, Eli Whitney, an American inventor and manufacturer, devised a system to mass produce muskets (guns). Whitney began using a milling machine, **Figure 1-4**, to make interchangeable musket parts. Until then, muskets were made individually by hand, so parts from one musket would not fit in another. Whitney's milling machine even had power feed, but it had one defect. There was no provision to raise the worktable. The part had to be raised by shimming after each cut. Since each machine was used to produce the same part again and again, this shortcoming was not a great problem. It wasn't too much later that this problem was corrected.

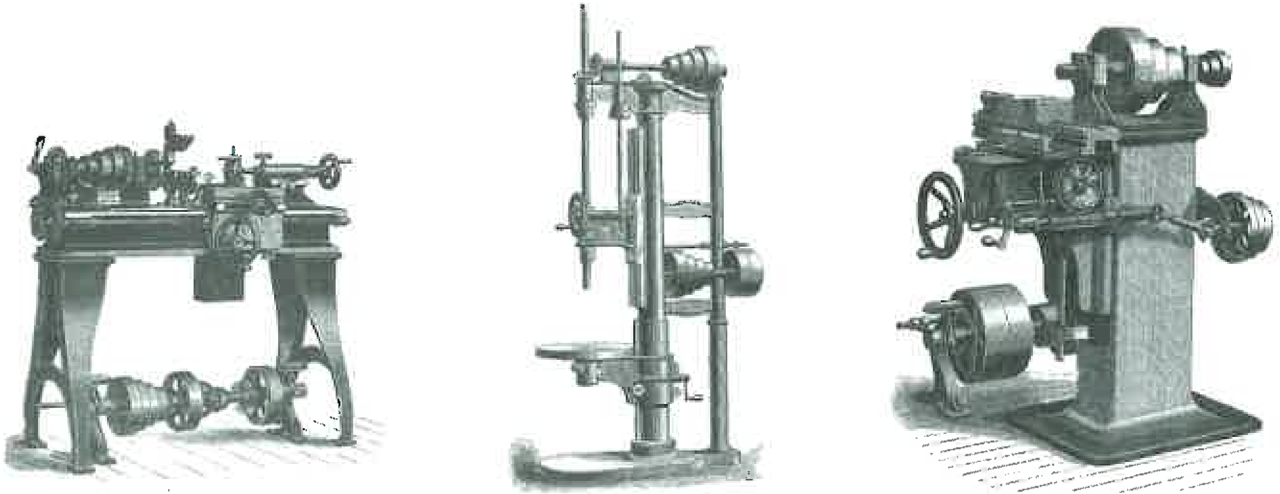
Whitney had another problem, however. His ideas were used in several armories producing gun parts. There was no standard of measurement at that time, so parts made in one armory were not interchangeable with parts in another armory. It was not until the mid-1860s that the United States adopted a standard measuring system.

By 1875, basic machine tools such as the lathe, the milling machine, and the drill press, **Figure 1-5**, were capable of attaining accuracies of one one-thousandth of an inch. America was well on its way to becoming the greatest industrial nation in the world.

1.1.2 Power Sources

As machine tools were improved, so was the way they were powered. At first, the changes were very slow, taking hundreds of years. The great changes have come only in the last 150 years or so.

- *Hand power.* The bow lathe and bow drill are examples. Direction of rotation changed at each stroke of the bow.
- *Foot power.* A treadle or a treadmill made possible continuous rotation of the work in one direction.
- *Animal power.* Treadmills were used to power early devices for boring cannon barrels. Human foot power was not sufficiently strong for this work.



13 INCH WEIGHTED LATHE.

No. 2 UPRIGHT DRILL.

No. 1 MILLING MACHINE.

Figure 1-5. *Illustrations of Pratt & Whitney machine tools from an 1876 advertisement. Built from heavy iron castings, the machines were driven by overhead pulleys and belting. A central steam engine or large electric motor powered the overhead pulleys in factories until the 1920s.*

- *Water power.* Not always dependable as a power source, because of lack of water during dry seasons.
- *Steam power.* The first real source of dependable power. A centrally located steam engine turned shafts and overhead pulleys that were belted to the individual machines.
- *Central electrical power.* Large electric motors simply replaced the steam engines. Power transmission to the machines did not change.
- *Individual electrical power.* Motors were built into the individual machine tools. Overhead belting was eliminated.

1.2 BASIC MACHINE TOOL OPERATION

Almost all machine tools have evolved from the ***lathe,*** **Figure 1-6**. This machine tool performs one of the most important machining operations. It operates on the principle of work being rotated against the edge of a cutting tool, **Figure 1-7**. Many other operations—drilling, boring, threadcutting, milling, and grinding—can also be performed on a lathe. The most advanced version of the lathe is the ***CNC turning center,*** **Figure 1-8**. See Chapters 13-15 for basic lathe operations, and Chapters 21 and 22 for automated machining.

Figure 1-6. *A modern lathe using digital technology to perform operations such as maintaining a constant surface speed, automatic threading cycle, automatic radius cutting, and taper turning. Note the safety shield that moves with the carriage. Except those tools that perform nontraditional machining operations, all machine tools have evolved from the lathe. (Harrison/Rem Sales, Inc.)*

1.2.1 Drill Press

A ***drill press,*** **Figure 1-9**, rotates a cutting tool (***drill***) against the material with sufficient pressure

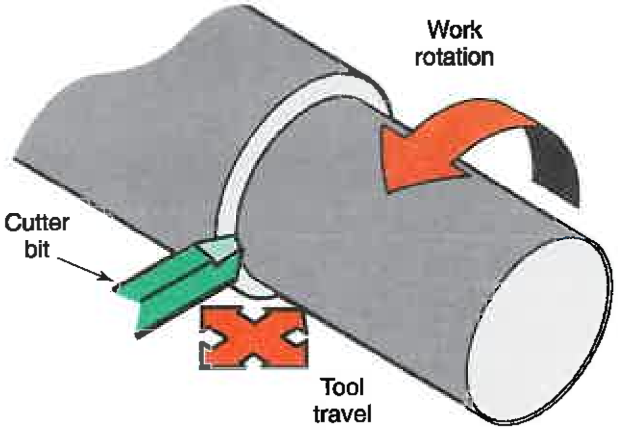


Figure 1-7. *The lathe operates on the principle of the work being rotated against the edge of a cutting tool.*

Figure 1-8. *Slant bed CNC lathe with hydraulic chucking and an electronically indexed 12 station turret. (Clausing Industrial, Inc.)*

Figure 1-9. *A typical 20" variable-speed gear head drill press with power feed. It can drill holes up to 1 1/2" in diameter in cast iron. (Willis Machinery and Tools Corp.)*

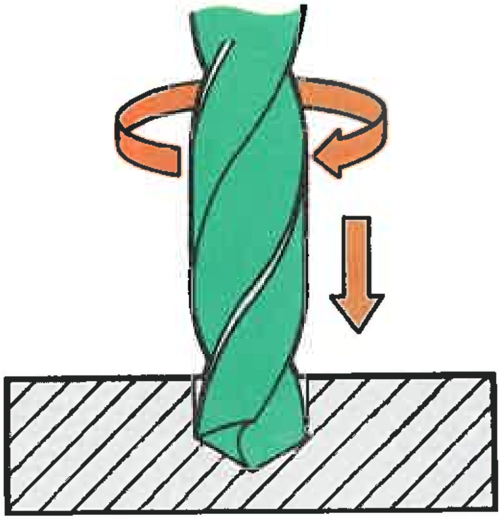


Figure 1-10. *A drill press operates by rotating a cutting tool (drill) against the material with sufficient pressure to cause the tool to penetrate the material.*

to cause the tool to penetrate the material. It is primarily used for cutting round holes. See **Figure 1-10**. Drill presses are available in many versions. Some are designed to machine holes as small as 0.0016" (0.04 mm) in diameter. See Chapter 10.

1.2.2 Grinding Machines

Grinding, **Figure 1-11**, is an operation that removes metal by rotating a grinding wheel or abrasive belt against the work. The process falls into two basic categories:

- ***Offhand grinding.*** Work that does not require great accuracy is hand-held and manipulated until ground to the desired shape. See Chapter 11.

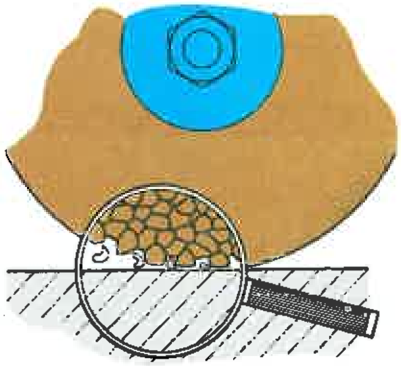


Figure 1-11. *Grinding is a cutting operation, like turning, drilling, milling, or sawing. However, instead of the one, two, or multiple-edge cutting tools used in other applications, grinding employs an abrasive tool composed of thousands of cutting edges.*

- ***Precision grinding.*** Only a small amount of material is removed with each pass of the grinding wheel, so that a smooth, accurate surface is generated. Precision grinding is a finishing operation. See Chapter 19.

1.2.3 Band Machines

Band machining, **Figure 1-12**, is a widely employed technique that makes use of a continuous saw blade. Chip removal is rapid and accuracy can be held to close tolerances, eliminating or minimizing many secondary machining operations. See Chapter 20.

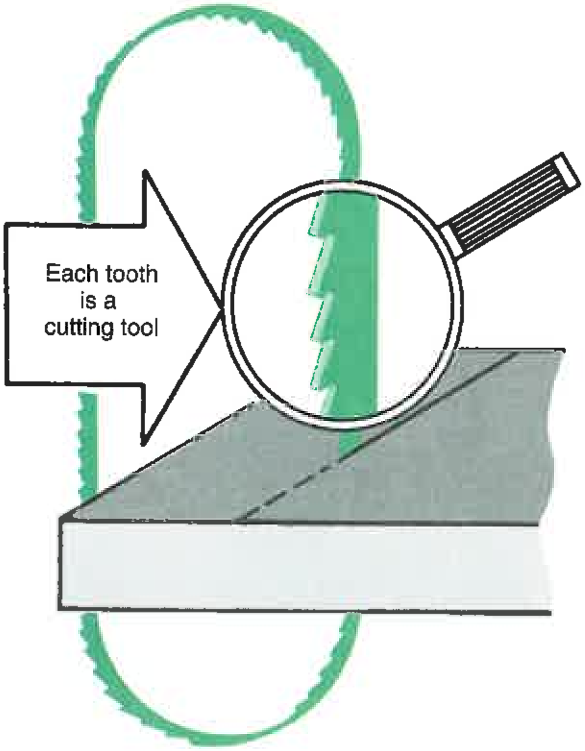


Figure 1-12. *Band machining makes use of a continuous saw blade, with each tooth functioning as a precision cutting tool.*

1.2.4 Milling Machine

A ***milling machine*** rotates a multitoothed cutter into the work, **Figure 1-13**. A wide variety of cutting operations can be performed on milling machines. See Chapters 17 and 18.

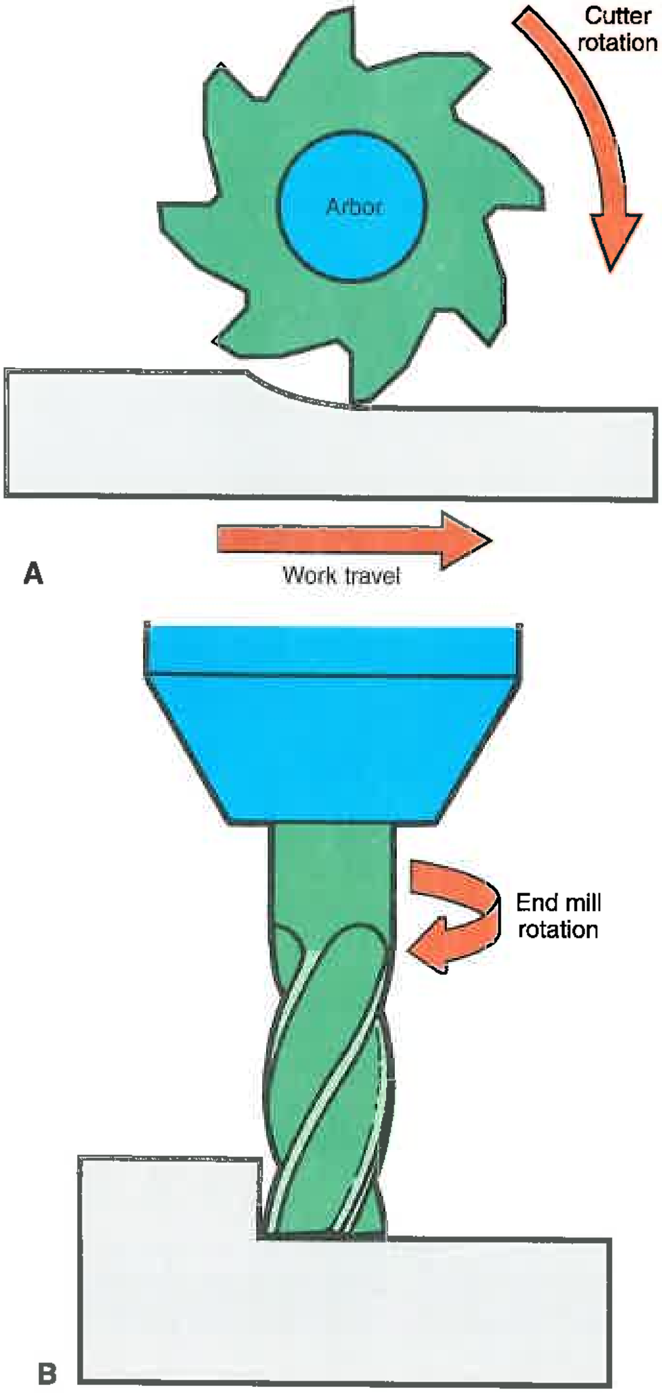


Figure 1-13. *Milling removes material by rotating a multitoothed cutter into the work. A—With peripheral milling, the surface being machined is parallel to periphery of the cutter. B—End mills have cutting edges on the circumference and the end.*

1.2.5 Broaching Machines

Broaching machines are designed to push or pull a multitoothed cutter across the work, **Figure 1-14**. Each tooth of the ***broach*** (cutting tool) removes only a small amount of the material being machined.

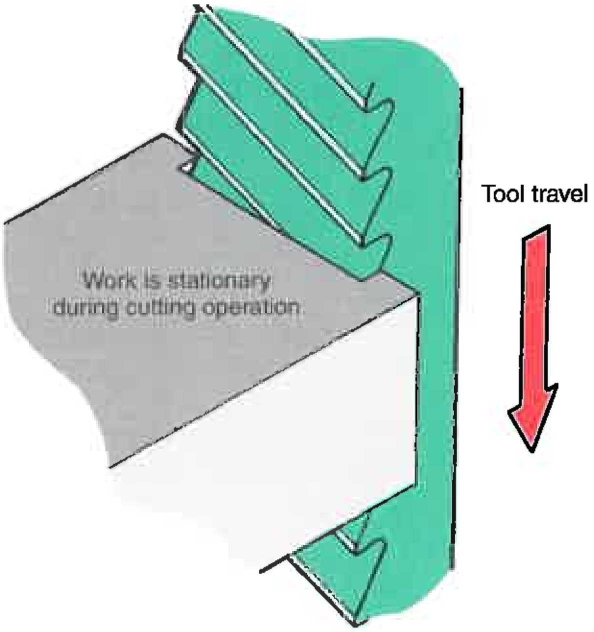


Figure 1-14. *A broach is a multitoothed cutting tool that moves against the work. Each tooth removes only a small portion of the material being machined. The cutting operation may be on a vertical or horizontal plane.*

1.3 NONTRADITIONAL MACHINING PROCESSES

There are a number of machining operations that have not evolved from the lathe. They are classified as ***nontraditional machining processes***. These processes include:

- ***Electrical discharge machining (EDM).*** An advanced machining process that uses a fine, accurately controlled electrical spark to erode metal.
- ***Electrochemical machining (ECM).*** A method of material removal that shapes a workpiece by removing electrons from its surface atoms. In effect, ECM is exactly the opposite of electroplating.
- ***Chemical milling.*** A process in which chemicals are employed to etch away selected portions of metal.
- ***Chemical blanking.*** A material removal method in which chemicals are employed to produce small, intricate, ultrathin parts by etching away unwanted material.
- ***Hydrodynamic machining (HDM).*** A computer-controlled technique that uses a 55,000 psi water jet to cut complex shapes with minimum waste. The work can be accomplished with or without abrasives added to the jet.
- ***Ultrasonic machining.*** A method that uses ultrasonic sound waves and an abrasive slurry to remove metal.
- ***Electron beam machining (EBM).*** A thermoelectric process that focuses a high-speed beam of electrons on the workpiece. The heat that is generated vaporizes the metal.
- ***Laser machining.*** The laser produces an intense beam of light that can be focused onto an area only a few microns in diameter. It is useful for cutting and drilling.
- ***Hexapods.*** CNC has made possible unconventional machine tools that use new work-positioning and tool-positioning concepts. See **Figure 1-15**. These tools, already available to industry, utilize the same movement principles developed for the flight simulators that train aircraft pilots. They offer basic advantages in stiffness, accuracy, speed, dexterity, and scaling (making larger or smaller versions of the same part).

1.4 AUTOMATING THE MACHINING PROCESS

In the late 1940s, the United States Air Force was searching for ways to increase production on complex parts for the new jet aircraft and missiles then going into production.

The Parsons Corporation, a manufacturer of aircraft parts, had developed a two-axis technique for generating data to check helicopter blade airfoil patterns. This system used punched-card tabulating equipment. To determine the accuracy of the data, a pattern was mounted on a Bridgeport milling machine. With a dial indicator in place, the X and Y points were called out to a machinist operating the machine's X-axis handwheel and another machinist who controlled the Y-axis handwheel. With enough reference points established, the generated data proved accurate to ±0.0015" (0.038 mm).

1.4.1 The Development of Numerical Control

Parsons realized that the technique might also be developed into a two-axis, or even three-axis, machining system. With an Air Force contract to manufacture a contoured integrally stiffened aircraft wing section, the Parsons Corporation subcontracted with the Servomechanism Laboratory at the Massachusetts Institute of Technology to design a three-axis machining system. MIT eventually took over the entire NC development project.

By 1952, MIT had designed a control system and mounted it on a vertical spindle machine tool. The system operated on instructions coded in the binary number system on punched (perforated) tape. Programming required the use of an early computer upon which MIT was also experimenting.

Later in that year, MIT demonstrated the first machine tool capable of executing simultaneous cutting tool movement on three axes. Since mathematical information was the basis of the concept, MIT coined the term ***numerical control (NC).*** The first NC machines became available to industry by late 1955.

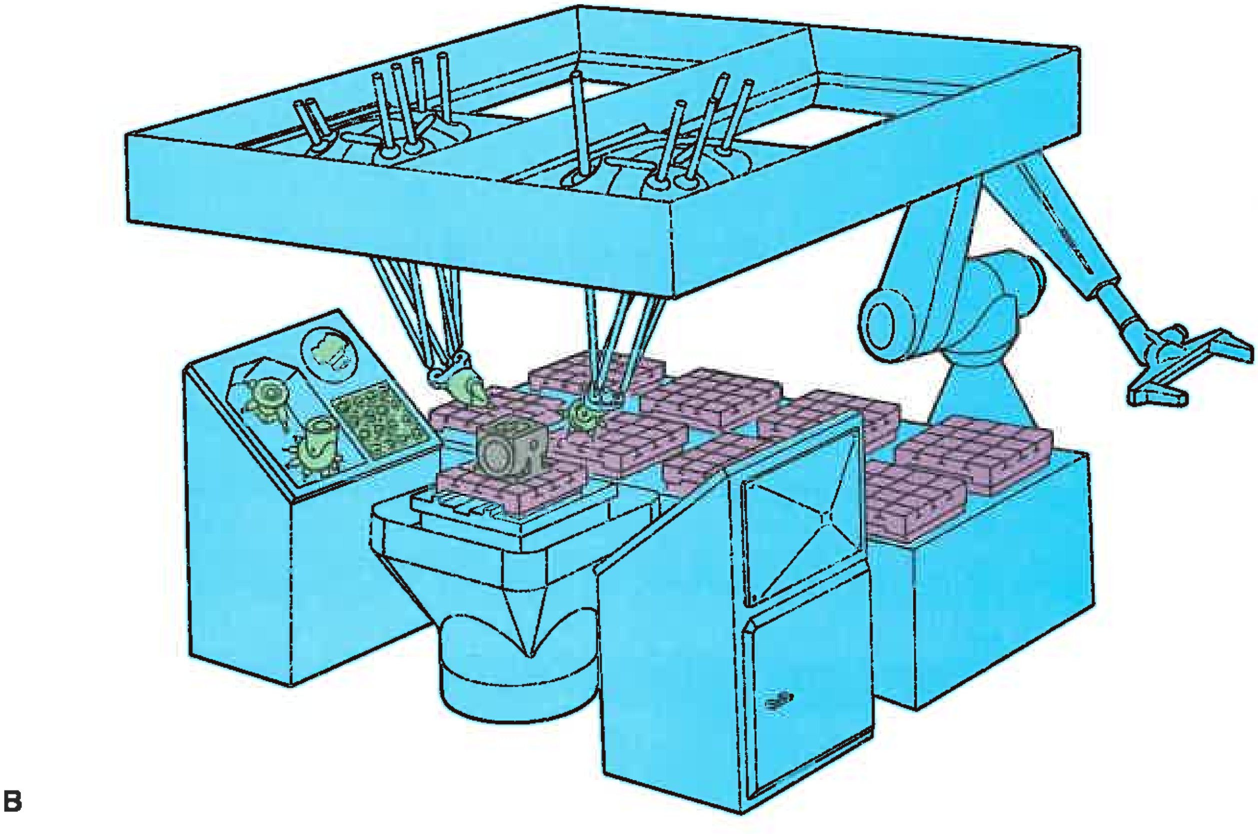


Figure 1-15. *New machine tool concept, the hexapod, seems to defy almost every preconception about what a machine tool should be. A—The hexapod uses an entirely new concept for cutting tool movement and work positioning, with six degrees of freedom provided by a framework of variable length struts. (Renaissance Design, Inc.) B—The hexapod can be configured to perform multiple functions such as milling, drilling, tapping, polishing, grinding, welding, and even mechanical assembly.*

1.4.2 Computer Numerical Control

In the mid-1970s, with the introduction of the microchip, the use of onboard computers on individual machine tools became possible. This led to the introduction of ***computer numerical control (CNC)***, **Figure 1-16**.

CNC machine tools are much easier to use than manually controlled machines. They have menu-selectable displays, advanced graphics (the multifunction screen displays the full operational data as a part is being machined), and a word address format for programming. The program is made up of sentence-like commands. Programs can be entered at the machine, or may be downloaded by direct line from an external computer. Programs on punched tapes are rarely used. A modern CNC ***horizontal machining center (HMC)*** is shown in **Figure 1-17**.

A CNC machine tool offers:

- *Accuracy.* It is capable of producing consistent and accurate workpieces.
- *Repeatability.* It is able to produce any number of identical workpieces once a program is verified.

Figure 1-16. *CNC machine tools are equipped with on-board computers that permit computer-aided or manual programming. All controls needed for complete machine operation are in one location. A CRT screen displays all important machining information, such as the tool path. (Giddings & Lewis, Inc.)*

Figure 1-17. *A state-of-the-art CNC horizontal machining center (HMC) with multiaxis capabilities. It can handle a wide range of workpiece sizes and materials. The center is fitted with a multiple pallet work storage system (foreground) that automatically transfers workpieces into and out of the machine upon command from the CNC unit. (Giddings & Lewis, Inc.)*

- *Flexibility.* Changeover to running another type of part requires only a short period of nonproductive machine downtime.

The use of ***robotic systems*** for loading and unloading permits some machine tools to operate unattended during the entire machining cycle. Robots also have many other industrial applications, **Figure 1-18.** They can:

- Operate in hazardous and harsh environments.
- Perform operations that would be tedious for a human operator.
- Handle heavy materials.
- Position parts with great repetitive precision.

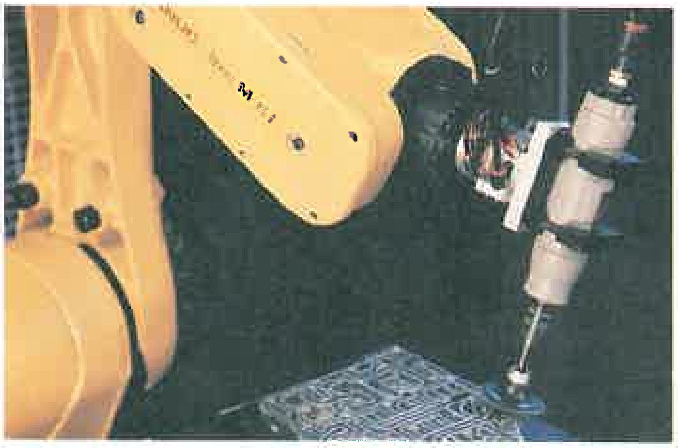


Figure 1-18. *A robot is a programmable, multifunctional manipulator designed to move material, tools, or specialized devices through programmed motions for the performance of a variety of tasks. This robot is deburring a complex part following machining operations. (Fanuc Robotics)*

The automotive industry makes extensive use of robots in the manufacture and assembly of motor vehicles, **Figure 1-19.**

1.5 THE EVOLVING ROLE OF THE MACHINIST

In recent years, the number of highly skilled machinists has been in decline. CNC machine tools have compensated for this trend to some degree. Since these machines operate under programmed control, the men and women who use them do not require the same level of skill or training as a skilled machinist.

However, because of these same CNC machine tools, the demand for machinists has not diminished. Machinists understand machining technology and what machine tools are capable of accomplishing. For these reasons, they make the best programmers and setup personnel.

Figure 1-19. *The automotive industry makes extensive use of robots for positioning parts, welding, painting, and performing quality control tasks.*

There is still another reason for the high demand for machinists: although CNC equipment is found in almost all machine shops, surveys consistently show that there is still considerable work being produced on conventional manually operated machine tools.

Whether planning an NC program or preparing to produce work on a conventional machine tool, a machinist must make many decisions and determinations on how to manufacture a part in the most economical way. The machinist must:

- Make a thorough study of the print.
- Determine the machining that must be done.
- Ascertain tolerance requirements.
- Plan the machining sequence.
- Determine how the setup will be made.
- Select the machine tool, cutter(s), and other tools and equipment that will be needed.
- Calculate cutting speeds and feeds.
- Select a proper cutting fluid for the material being machined.

All of this is possible because of the skill, knowledge, and experience of the machinist. Essentially, a machinist is able to *visualize the machining program.* When NC and CNC came along, most machinists quickly adapted to the new technology because they were already experienced in machining technology.

1.5.1 Acquiring Machining Skills and Knowledge

The skills and knowledge needed by the machinist are not acquired in a short time. It normally requires taking part in a multiyear salaried apprentice program. In addition to machine tool training under an experienced machinist, the program also involves related subjects such as English, algebra, geometry, trigonometry, print reading, safety, production techniques, and CNC principles and programming. Refer to Chapter 30 for additional information on machining technology occupations.

The *National Tooling and Machining Association,* with the aid of the metalworking industry, has developed three levels of ***skill standards*** reflecting industry skill requirements. A major goal of the Metalworking Skills Standards program is performance testing. The standards will provide skilled workers with certification that will afford them industry recognition.

TEST YOUR KNOWLEDGE

Please do not write in this text. Write your answers on a separate sheet of paper.

1. One of the first machine tools, the bow lathe:
   a. Could only turn softer materials.
   b. Has been dated back to about 1200 BC.
   c. Eventually gave way to treadle power.
   d. None of the above.
   e. All of the above.

2. The Industrial Revolution could not have taken place without the cheap, convenient power of the ________.

3. List seven power sources in the order they have evolved over the last 150 years or so.

4. Almost all machine tools have evolved from the _____.

5. Jobs such as tool-and-diemaking and precision machining require aptitudes comparable to those of _____ .
   a. High school graduates.
   b. College graduates.
   c. High school equivalency graduates.
   d. All of the above
   e. None of the above.

6. Eli Whitney's mass-production system for muskets had a major problem because _____.
   a. There were no skilled workers
   b. There was no good source of power.
   c. There was no standard of measurement.
   d. All of the above.
   e. None of the above.

7. What occurred in the mid-1860s that was very important to the development of machining technology in the United States?

8. List four types of nontraditional machining processes and briefly describe their operation.

9. The introduction of the microchip in the mid-1970s led to the introduction of _____ machine tools.

10. List four industrial applications of robots.

The role of the computer in manufacturing has expanded greatly in recent years. In addition to computer numerical control of machine tools, many production operations include computer-controlled robotic assembly lines like this one. (Giddings & Lewis, Inc.)

Chapter 2

Shop Safety

LEARNING OBJECTIVES

After studying this chapter, you will be able to:

- ○ Give reasons why shop safety is important.
- ○ Explain why it is important to develop safe work habits.
- ○ Recognize and correct unsafe work practices.
- ○ Apply safe work practices when employed in a machine shop.
- ○ Select the appropriate fire extinguisher for a particular type of fire.

IMPORTANT TERMS

adequate ventilation
approved-type respirator
back injuries
combustible materials
electrocution
OSHA
protective clothing
safety equipment
unsafe practice
warning signs

Shop safety is *not* something to be studied at the start of a training program and then forgotten; most accidents are caused by carelessness or by not observing safety rules. Remember this when your instructor insists on safe work practices. If you are diligent and follow instructions with care, machining operations can be safe and enjoyable. Safe work practices should become a force of habit!

Since it is not possible to include *every* safety precaution, the safety practices in this chapter are general. Safety precautions for specific tools and machines are described in the text where they apply, along with the description and operation of the equipment. Refer to **Figure 2-1**.

Study all safety rules carefully and constantly apply them. When in doubt about any task, get help! *Do not* take chances!

2.1 SAFETY IN THE SHOP

Keep the shop *clean*. Metal scraps should be placed in the scrap bin. Never allow them to remain on the bench or floor.

Exercise extreme care when you are machining unfamiliar materials. For example, magnesium chips burn with great intensity under certain conditions. Applying water to the burning magnesium chips only intensifies the fire. Machining equipment can be damaged beyond repair and very serious burns can result.

Inhaling fumes or dust from some of the newer space-age and exotic materials can cause serious respiratory ailments. *Do not* machine a material until you *know what it is* and how it should be handled.

An approved-type respirator and special ***protective clothing*** must be worn when machining some materials. Machines must be fitted with effective vacuum systems as needed.

The shop is a place to work, not play. It is *not* a place for horseplay. A "joker" in a machine shop is

Figure 2-1. *None of the pilots in this precision flying team would think of taking off to give a flight demonstration until all the plane's systems were in safe operating condition. In the same way, you should never operate a machine tool until you have determined that it is in safe operating condition.*

a *walking hazard* to everyone. Daydreaming also increases your chances of injury.

If you have been ill and are using ***medication,*** check with your doctor or school clinic to determine whether it is safe for you to operate machinery. For example, many cold remedies recommend that you do not operate machinery while taking the medication because of possible drowsiness.

Avoid using ***compressed air*** to remove chips and cutting oil from machines. Flying chips can cause serious eye injuries. Also, oil that has been vaporized by the stream of air can ignite, resulting in painful burns and property damage.

Oily rags must be placed in an ***approved safety container*** (a metal can with metal lid). See **Figure 2-2**. Rags or waste used to clean machines will also have metal slivers embedded in them, posing an additional hazard. Placing them in a safety container will help make sure they will not be used again. Dispose of the rags daily. This will minimize the possibility of spontaneous combustion (ignition by rapid oxidation or burning of oil without an external source of heat).

Figure 2-2. *Oily rags used for cleaning machines or soaking up spills should be placed in an approved safety container like this one to minimize fire hazards. The container should be emptied daily and contents disposed of properly.*
(Justrite Manufacturing Company)

Keep ***hand tools*** in good condition. Store tools in such a way that people cannot be injured while they are removing the tools from the tool panel or storage rack.

Use care when handling long sections of metal stock — accidentally contacting a light fixture with the stock, for example, could cause severe electrical burns or even death by ***electrocution*** (electric current passing through body tissues). An electric shock has been compared to "being hit by a truck."

When moving heavy machine accessories or large pieces of metal stock, always secure help. The ***back injuries*** that result from improper lifting are usually long-term injuries!

Dress properly for working around machinery — severe injuries or even death can result if clothing, hair, or jewelry gets caught in moving parts. Avoid wearing loose-fitting sweaters or similar clothing that could catch in machinery. A snug-fitting shop coat or apron can be worn to protect your street clothes, **Figure 2-3**. Keep sleeves rolled up. Rings and other jewelry should be removed before working around machinery. If you have long hair, wear a cap or use other means of containing it.

For jobs where dust and fumes are a hazard, ensure ***adequate ventilation***. Return solvents and oils to proper storage after use. Wipe up spilled oil or solvent right away. If the spill area is extensive, use an approved-type oil absorbent. See **Figure 2-4**.

Figure 2-3. *This trainee is properly dressed for the job she is doing. She is wearing approved eye protection and a snug-fitting apron. The machine was carefully checked before she began to operate it. (Millersville University)*

Figure 2-4. This worker in a manufacturing plant is using a special oil-absorbent material to wrap a leaking machine component. Note that the material, which is packaged in roll form, has also been used to soak up oil on the machine base and floor. (3M Company)

Wear appropriate ***safety equipment,*** **Figure 2-5.** In noisy areas, use earplugs or another type of hearing protection. Disposable plastic gloves will protect your hands when handling oils, cutting fluids, or solvents. Wear a dust mask when machining produces airborne particles, such as those from sand castings, plastics, and some grinding operations. A disposable dust mask is *not* suitable in areas where machining operations produce a mist of oil or coolants. An ***approved-type respirator*** must be worn in such situations. See **Figure 2-6.** Suitable personal protective equipment must also be worn when handling sharp, hot, or contaminated materials.

Figure 2-5. Wear appropriate safety equipment. Shown are approved eye protection, an apron to protect clothing, plastic gloves for handling oils and solvents, a hearing protector, earplugs, and a dust mask.

Figure 2-6. Whenever fine airborne mists of oil, coolant, or other materials are present, an approved respirator is required. This spray painter is wearing a respirator supplied with clean air through a tube from a central source. It is also important to use proper eye protection, such as the safety glasses worn by this worker. (3M Company)

Take no chances! Always protect your eyes. Eyesight that has been damaged or destroyed cannot be replaced. Wear safety glasses, goggles, or face shields approved by ***OSHA*** (the United States Occupational Safety and Health Administration).

Wear eye protection whenever you are in the shop. Protective eyewear should be used when conditions call for it. It is good practice to have your own personal safety glasses. The cost is reasonable. Your instructor can help you determine the style best suited for your needs, **Figure 2-7.** If you wear glasses, special safety lenses are available that can be ground to your prescription. Your eye doctor or optician can help get them.

Know your job! It is foolish and disastrous to operate machines without first receiving proper instruction. If you are not sure what must be done, or how a task should be performed, get help.

Figure 2-7. *Safety glasses are available in a number of styles. The model at left is similar to regular eyeglasses, but has "wings" at each side and on top to guard against flying particles. The goggle-style model at right fits tightly against the face and can be worn over regular eyeglasses.*

2.1.1 Safety Aids

Awareness barriers, while they all do not provide physical protection from machine hazards, serve to remind the operator of an area that is dangerous. In simplest form, a barrier may be nothing more than red or yellow lines painted on the floor. More complex barriers stop the machine when a light beam or electronic beam is broken by someone entering the danger area.

Machine shields, **Figure 2-8**, provide protection from flying chips and splashing cutting fluids or coolants. Many CNC machine tools are fitted with large sliding shields that cover the entire machining area, **Figure 2-9**.

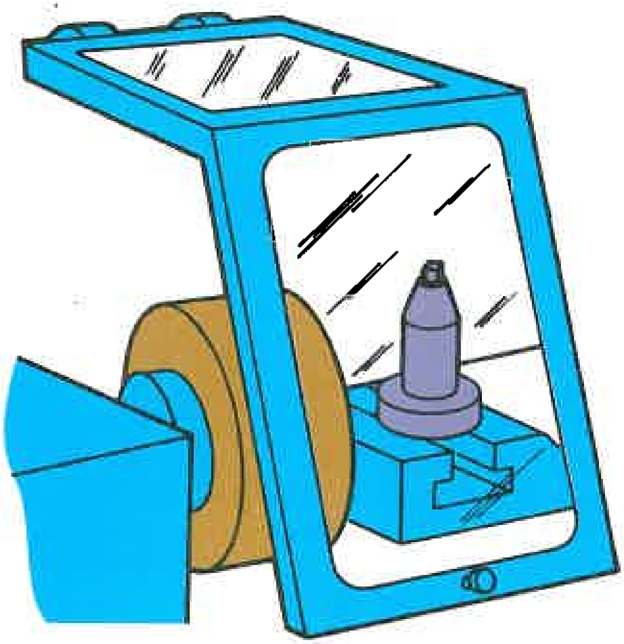


Figure 2-8. *Specially designed safety shields are available for almost all machine tools. They should always be in place before the machine is operated.*

Figure 2-9. *Many CNC machine tools use sliding shields over the machining area to protect the machinist from flying chips and vaporized coolant. (Bridgeport Machines, Inc.)*

Many different types of ***warning signs***, **Figure 2-10**, are used to notify workers of potential hazards. *No Smoking* signs must be posted in areas where inflammable or combustible materials are used and stored.

2.2 GENERAL MACHINE SAFETY

- Never operate a machine until all guards are in place!
- Always stop your machine to make adjustments or measurements! Resist the urge, while the machine is running, to touch a surface that has been machined. Severe lacerations can result.
- Keep the floor around your machine clear of oil, chips, and metal scrap.
- It is considered an ***unsafe practice*** to talk to anyone while you are operating a machine. You might become distracted and injure yourself, or someone else.
- *Never* attempt to remove chips or cuttings with your hands or while the machine is operating. Use a brush, **Figure 2-11**. Pliers are one of the safest ways to remove long, stringy chips from a lathe. Better still, learn how to grind the cutting tool to break chips off in shorter pieces. This is explained later in the text.

Figure 2-10. *Signs can remind workers to be alert for potential hazards. (Idesco Corp.)*

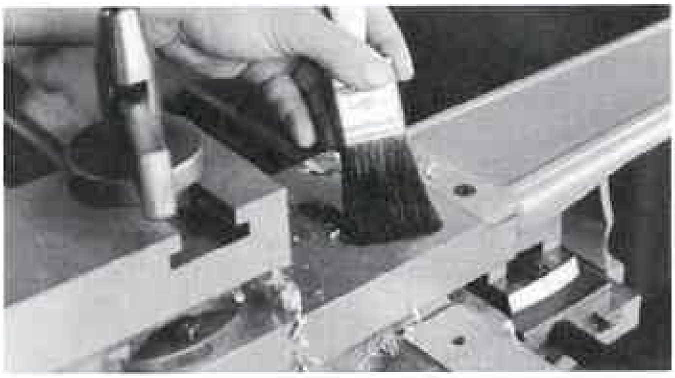


Figure 2-11. *Use a brush,* not *your hands, to remove accumulated chips!*

- Secure prompt medical attention for any cut, bruise, scratch, burn, or other injury. No matter how minor the injury may appear, report it to your instructor!

2.3 GENERAL TOOL SAFETY

- *Never carry sharp-pointed tools in your pockets!* When using sharp tools, lay them on the bench in such a way that you will not injure yourself when you reach for them, **Figure 2-12.**
- Make sure tools are properly sharpened, in good condition, and fitted with suitable handles.

Figure 2-12. *Arrange sharp pointed tools on the bench in a way that they will not injure you when you reach to pick them up.*

2.4 FIRE SAFETY

Combustible materials are classified into four categories, **Figure 2-13.** Extinguishers should have color-coded symbols to identify their appropriateness for a particular type of fire.

Class A Fires. Those involving ordinary combustible materials—paper, wood, textiles, etc. They require the cooling and quenching effect of water, or solutions containing a large percentage of water. Do not use Class A extinguishers on Class C and D fires.

Class B Fires. Flammable liquid and grease fires require the blanketing or smothering effects of dry chemicals or carbon dioxide.

Class C Fires. Electrical equipment fires require nonconducting extinguishing agents that will smother the flames. Do not use Class A extinguishers on electrical fires.

Class D Fires. Extinguishers containing specially prepared heat-absorbing dry powder are used on flammable metals, such as magnesium and lithium. Do not use Class A extinguishers on flammable metal fires.

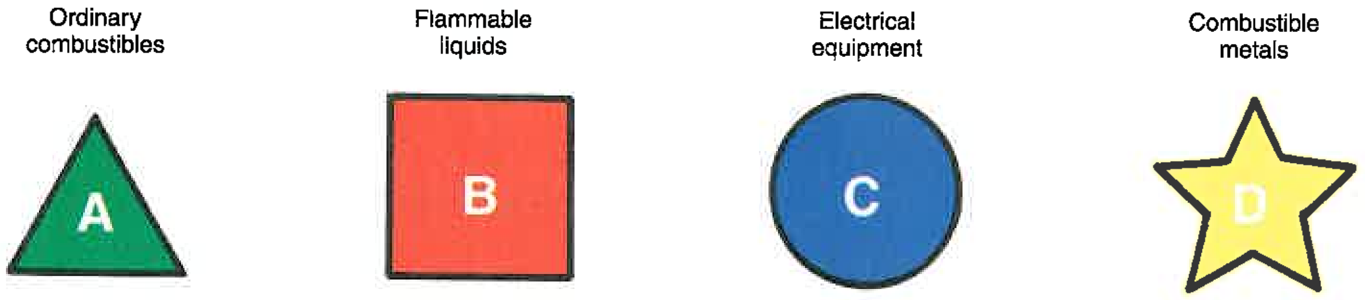


Figure 2-13. *Symbols coded by color and shape identify the four classifications of fire and the extinguishers that can be used on them.*

2.4.1 Dealing with a Fire

Know what to do in case of a fire! Be familiar with the location of your building's fire exits and how they are opened. Be aware of alternate escape routes.

In some situations, students are trained in the use of fire extinguishers by the local fire department. If you are one of those students, make sure that you know where the fire extinguishers are located. If you have *not* received such training, get out of the fire area immediately.

SPECIAL SAFETY NOTE: *Think before acting!* It costs nothing, and you may be saved from painful injury that could result in a permanent disability. If you think it tiring to sit through an hour-long class in school, think what it would be like to spend your entire life in a wheelchair!

TEST YOUR KNOWLEDGE

Please do not write in this text. Write your answers on a separate sheet of paper.

1. Why is shop safety so important?
2. Most shop accidents are caused by _____.
3. Safety glasses should be worn:
   a. most of the time.
   b. only when working on machines.
   c. the entire time you are in the shop.
   d. none of the above.
4. Oily rags should be placed in a safety container to prevent _____.
5. Why should compressed air *not* be used to clean chips from machine tools?
6. Never attempt to operate a machine until _____.
7. Always stop machine tools before making _____ and _____.
8. Use a _____ to remove chips and shavings, *not* your _____.
9. When working in an area contaminated with dust or solvent fumes, be sure there is _____. A(n) _____ should also be worn when working in a dusty area.
10. Secure prompt _____ for any cut, bruise, scratch, or burn.
11. Get help when moving _____.
12. What should you do before operating a machine tool if you are taking medication of any sort?
13. Why is it necessary to take special precautions when handling long sections of metal stock?

Chapter 3

Understanding Drawings

LEARNING OBJECTIVES

After studying this chapter, you will be able to:

- ○ Read drawings that are dimensioned in fractional inches, decimal inches, and in metric units.
- ○ Explain the information found on a typical drawing.
- ○ Describe how detail, subassembly, and assembly drawings differ.
- ○ Point out why drawings are numbered.
- ○ Explain the basics of geometric dimensioning and tolerancing.

IMPORTANT TERMS

actual size
American National Standards Institute
bill of materials
dual dimensioning
geometric dimensioning and tolerancing
revisions
scale drawings
SI Metric
US Conventional
working drawings

Many products manufactured today are an assembly of parts supplied by a number of different industries. These industries may be in distant geographic locations, **Figure 3-1.**

It would not be possible for industry to manufacture a complex product without using drawings. ***Drawings*** show the craft worker what to make and identify the standards that must be followed so the various parts will fit together properly. The resulting parts will also be interchangeable with similar components on equipment already in service.

Drawings range from a simple freehand sketch, **Figure 3-2,** to detailed drawings for complex products, **Figure 3-3.**

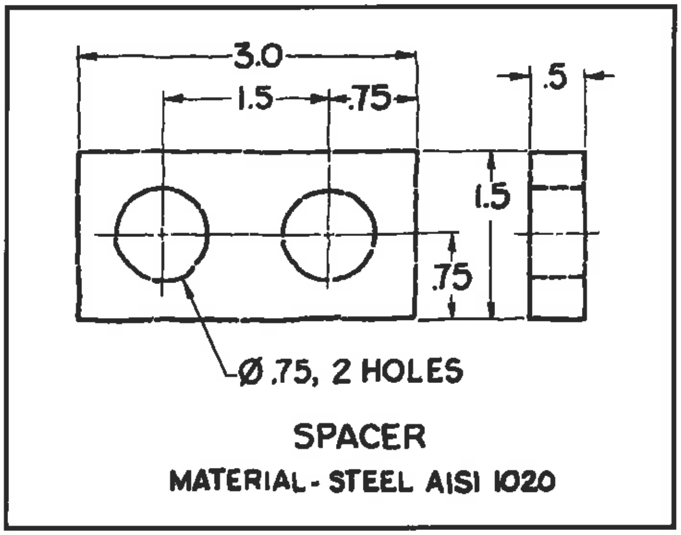


Figure 3-2. *Some drawings are as simple as this freehand sketch.*

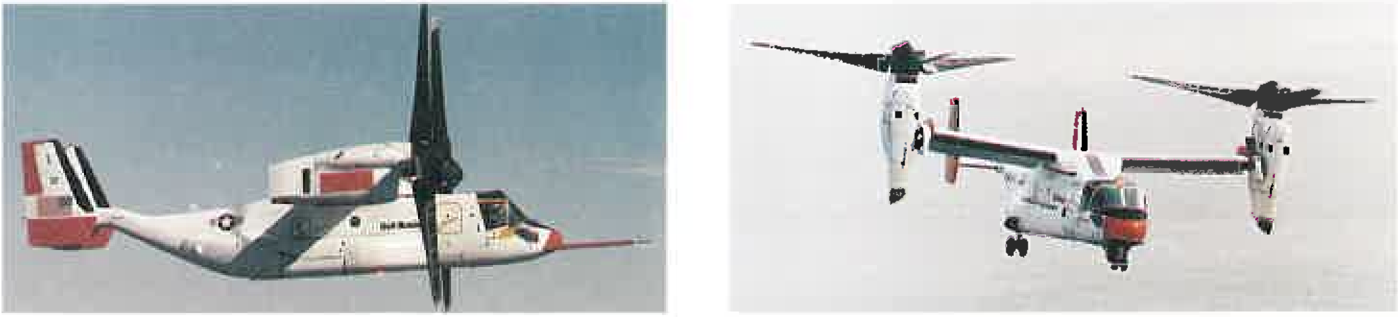


Figure 3-1. *Thousands of drawings were required in the design and construction of this vertical takeoff and landing aircraft. Standards and specifications had to be exact because components were manufactured in several geographic locations.*
(Bell Helicopter Textron/Boeing Helicopters)

Symbols, lines, and figures are employed to give drawings meaning, **Figure 3-4.** They have been standardized so they have the same meaning wherever drawings are made and used.

These symbols, lines, and figures have been devised by the ***American National Standards Institute,*** better known as ANSI. The symbols, lines, and figures on drawings are known as the *"language of industry."*

Periodically, ANSI changes standard drawing symbols. Craft workers must be familiar with past and present practices because only recently made

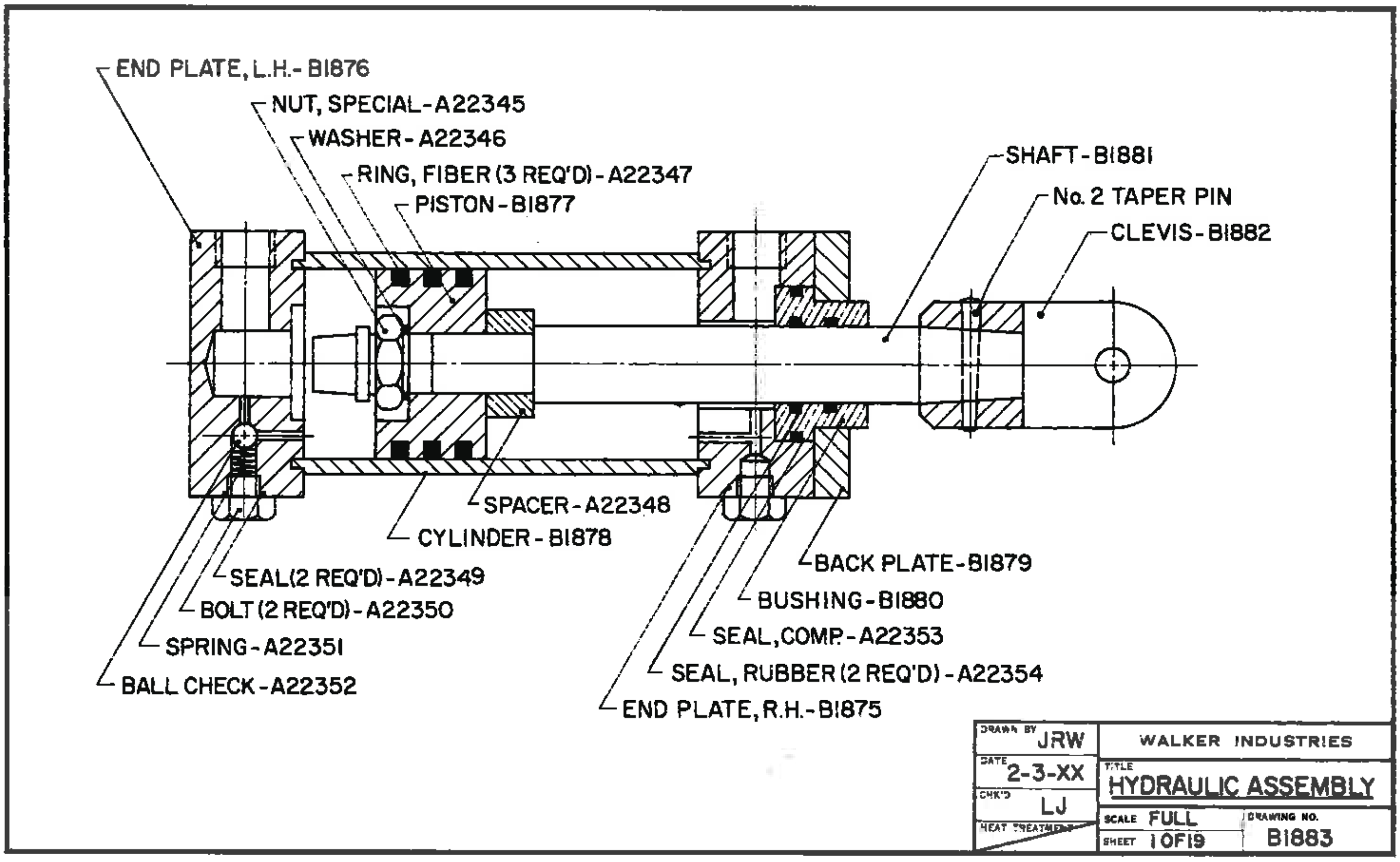


Figure 3-3. *Each manufactured product may require dozens of drawings, one for each part. Even the smallest screw, washer, or pin may require a drawing.*

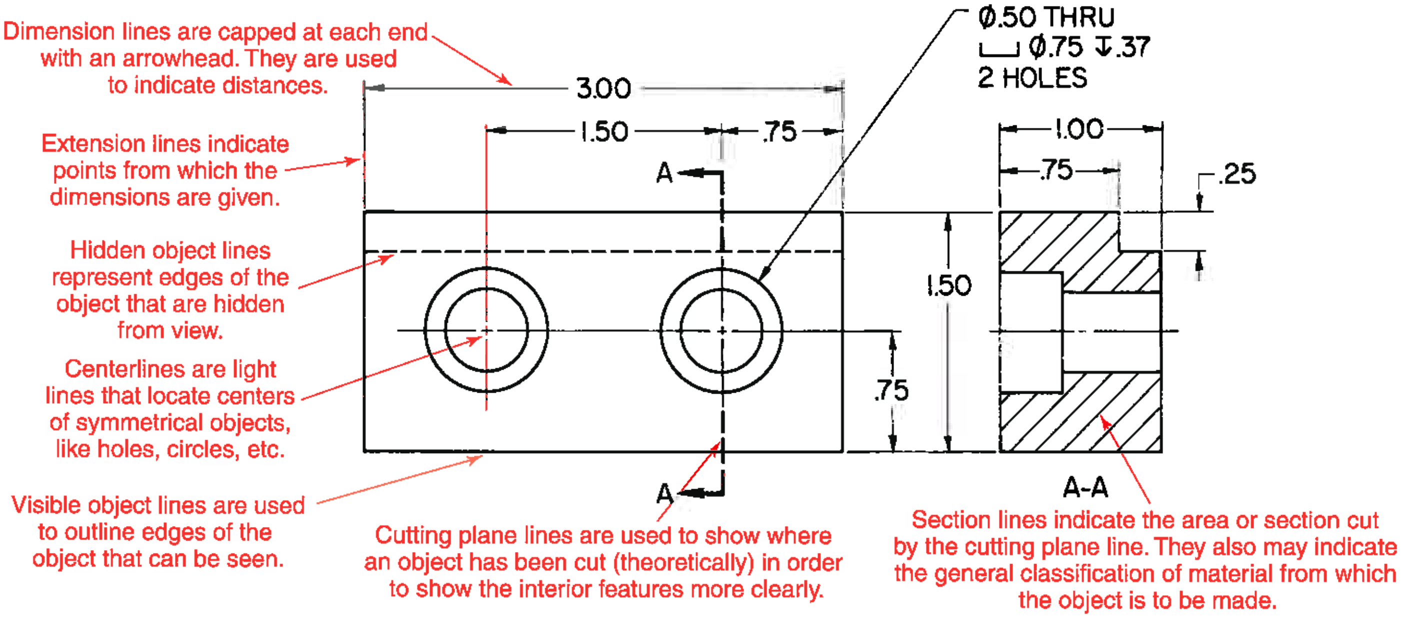


Figure 3-4. *Many types of lines, symbols, and figures are used to give a drawing exact meaning.*

drawings will follow the new standards. It is too expensive to revise the millions of drawings made before the new standards were devised. **Figure 3-5** shows past and present metalworking symbols.

Lines are used to draw views that fully describe the object to be manufactured. In addition, the drawing usually includes other information needed to make the product. Details often show threads, for example. **Figure 3-6** shows several methods of showing threads on a drawing.

3.1 DIMENSIONS

A proper drawing includes all ***dimensions*** (sizes or measurements) in proper relation to one another. The dimensions are needed to produce the part or object.

Until recently, drawings were only dimensioned in decimal or fractional parts of an inch, **Figure 3-7.** However, some industries in the United States are in the process of converting to the metric system of measurement. During this transition period, craft workers will need to understand drawings dimensioned in more than one system.

3.1.1 Fractional Dimensioning

Drawings using ***fractional dimensioning*** usually show objects that do not require a high degree of precision in their manufacture. Greater precision is indicated when dimensions are given in decimal parts of an inch.

3.1.2 Dual Dimensioning

Dual dimensioning is a system that employs the ***US Conventional*** ("English") system of fraction or decimal dimensions and metric dimensions on the same drawing, **Figure 3-8.** If the drawing is intended primarily for use in the United States, the decimal inch will appear above the metric dimension, as in **Figure 3-9A.** The reverse is true if the drawing is to be used in a metric-oriented country, as in **Figure 3-9B.** Some companies place the metric dimension within brackets, as in **Figure 3-9C.**

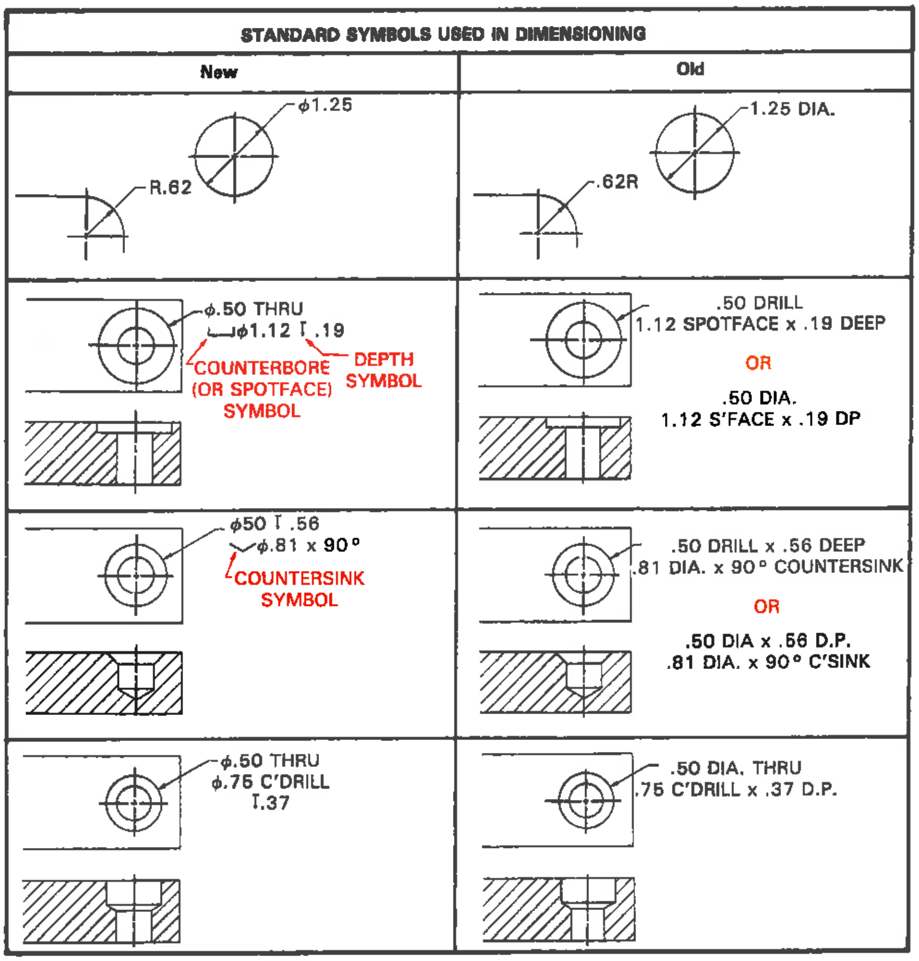
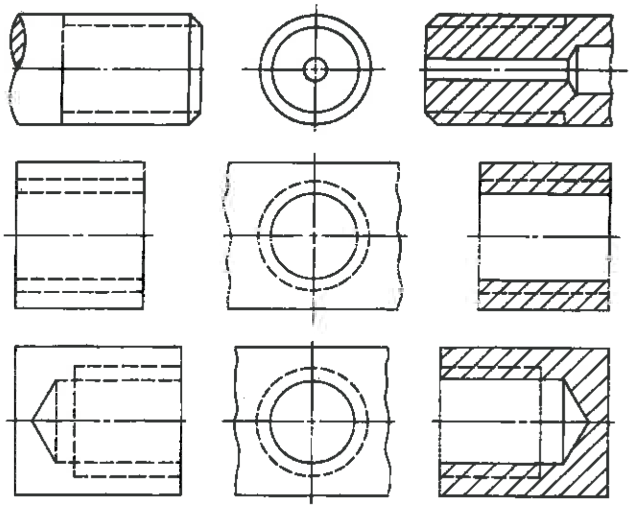
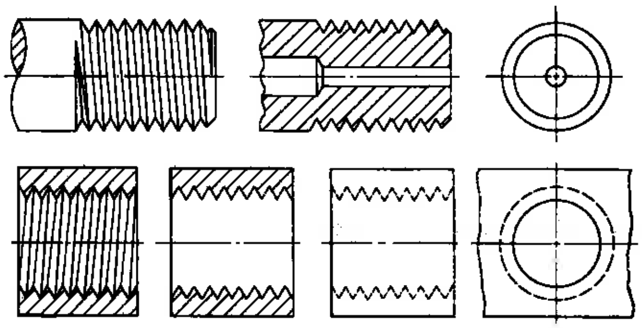


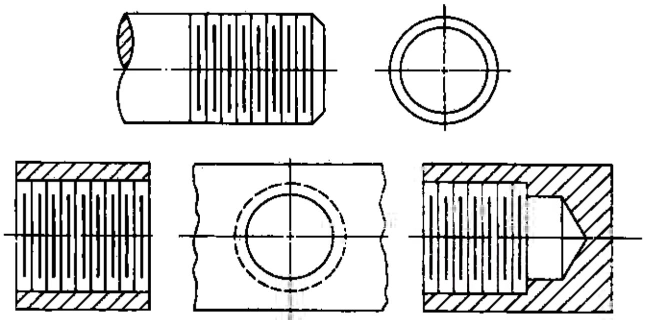
Figure 3-5. *Standard ANSI symbols are changed periodically. You must be familiar with both the old and new symbols because either may be used on the drawings. Compare these examples.*



Simplified representation



Detailed representation



Schematic representation

Figure 3-6. *Methods employed to depict threads on drawings. Only one type will be found on a drawing. The simplified version is the most common style.*

3.1.3 Metric Dimensioning

With ***metric dimensioning,*** all of the dimensions on the drawing are in the ***SI Metric*** system, usually in millimeters. However, until there is full conversion to metric in the United States, a ***conversion chart*** (equivalents for millimeter and inch dimensions) will appear on the drawing, **Figure 3-10.**

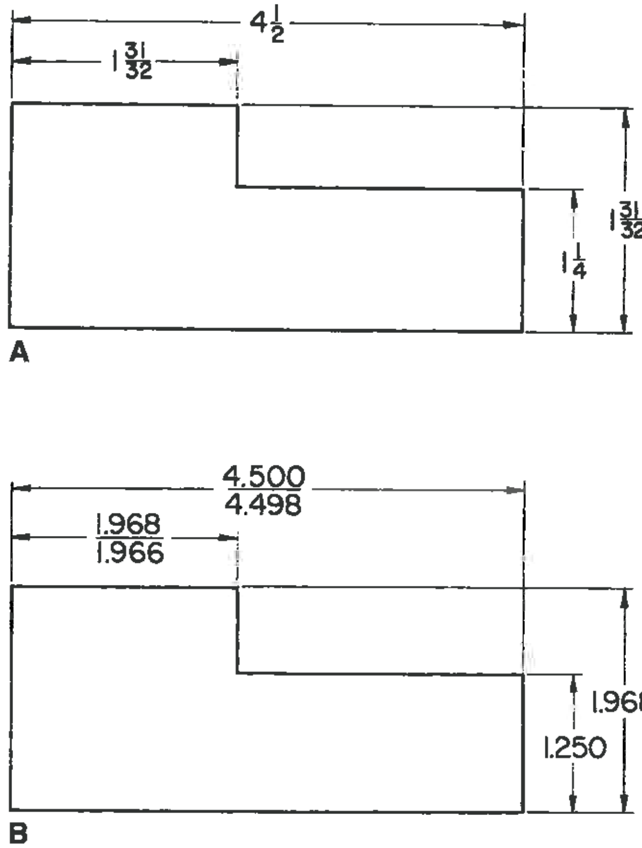


Figure 3-7. *Dimensional styles. A—Fractional dimensions do not require tolerances closer than ±1/64". B—Decimal dimensions normally have tolerances of ±0.010" or 0.015".*

3.2 INFORMATION INCLUDED ON DRAWINGS

Drawings contain additional information to inform the machinist of the material to be used, required surface finish, tolerances, etc. It is important for you to be familiar with this information.

3.2.1 Materials

The general classification of materials to be used in the manufacture of an object may be indicated by the type of section line on the drawing or plan, **Figure 3-11.** Exact material specification is included in a section of the title block, **Figure 3-12(A).** Sometimes, the material specification may be found in the notes shown elsewhere on the drawing.

3.2.2 Surface Finishes

The quality of the ***surface finish*** (degree of surface smoothness) is important in the manufacture of many products. The smoothness of the bore of an engine cylinder is an example. Usually, the more

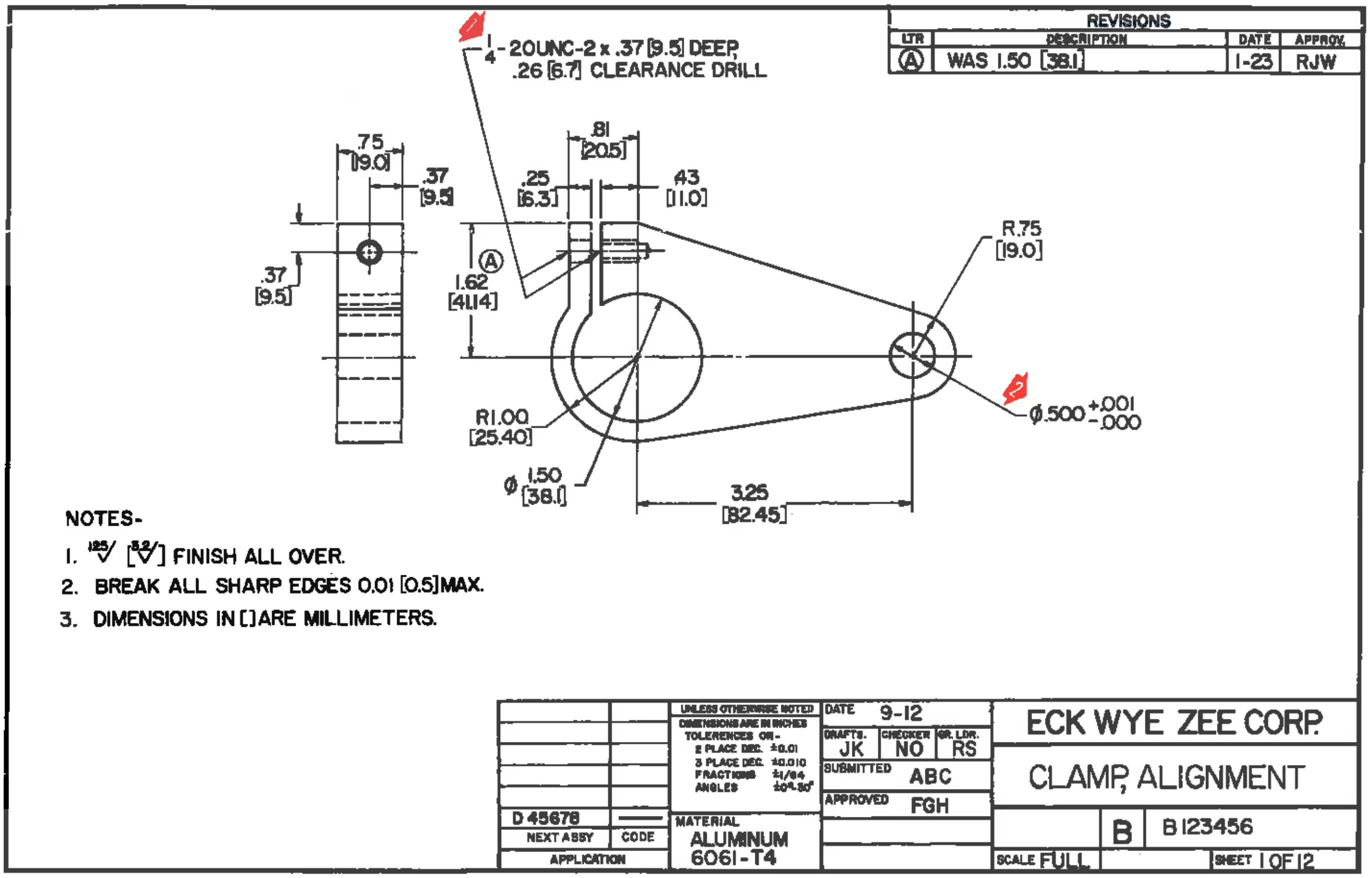


Figure 3-8. Dual-dimensioned drawing. 1—A metric thread size has not been given. There is no metric thread that is equal to this size fractional thread. 2—There is no metric reamer equal to this size.

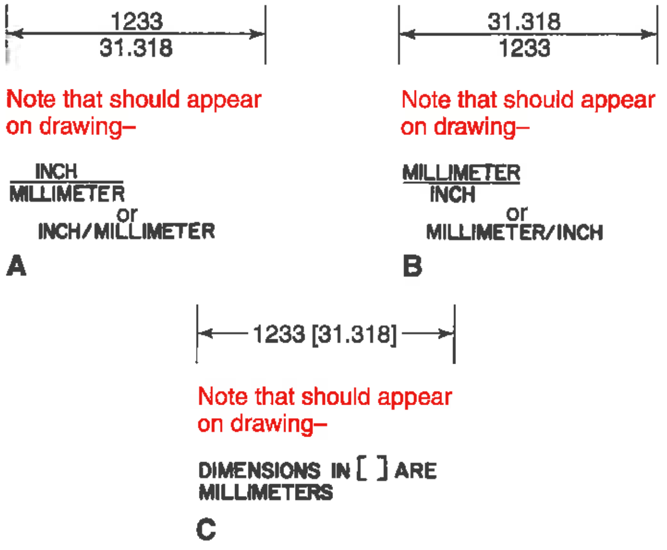


Figure 3-9. Indicating inch and millimeter dimensions on a dual-dimensioned drawing. A—When the drawing is to be used in the United States, the inch value appears on top. B—When the drawing is to be used primarily in a country that uses the metric system, the millimeter value appears on top. C—Sometimes, brackets are used to indicate the metric equivalent on a drawing to be used in the United States.

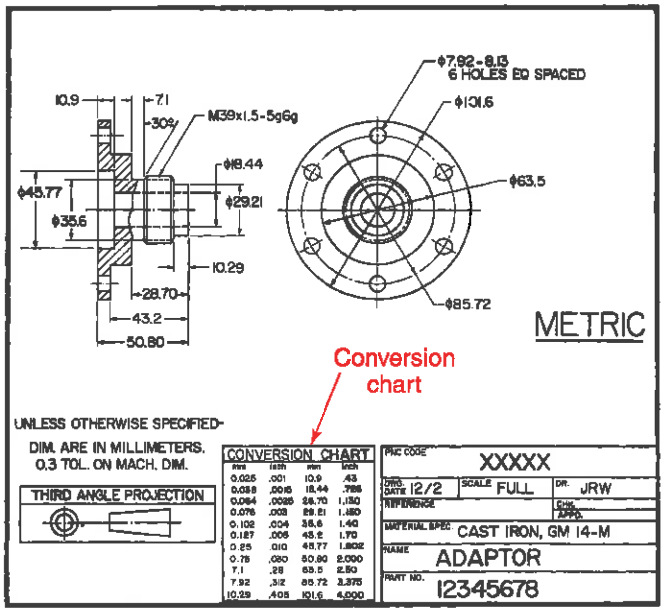


Figure 3-10. A drawing using metric units of measurement. Until the United States has fully converted to metric measurement, a conversion chart will usually appear on the drawing.

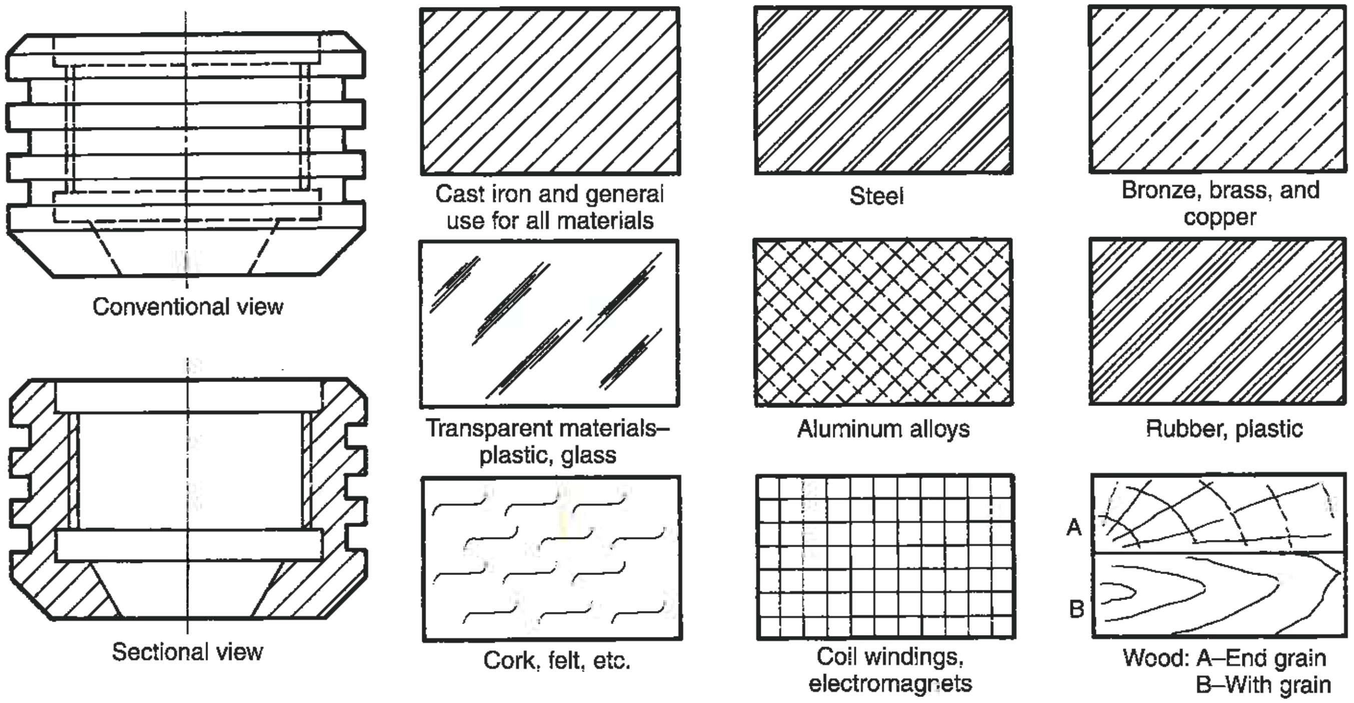


Figure 3-11. *Sectional views make a drawing easier to understand because internal details are shown more clearly. Various materials are identified with unique section lines. However, many section views use general section lining regardless of the material being used.*

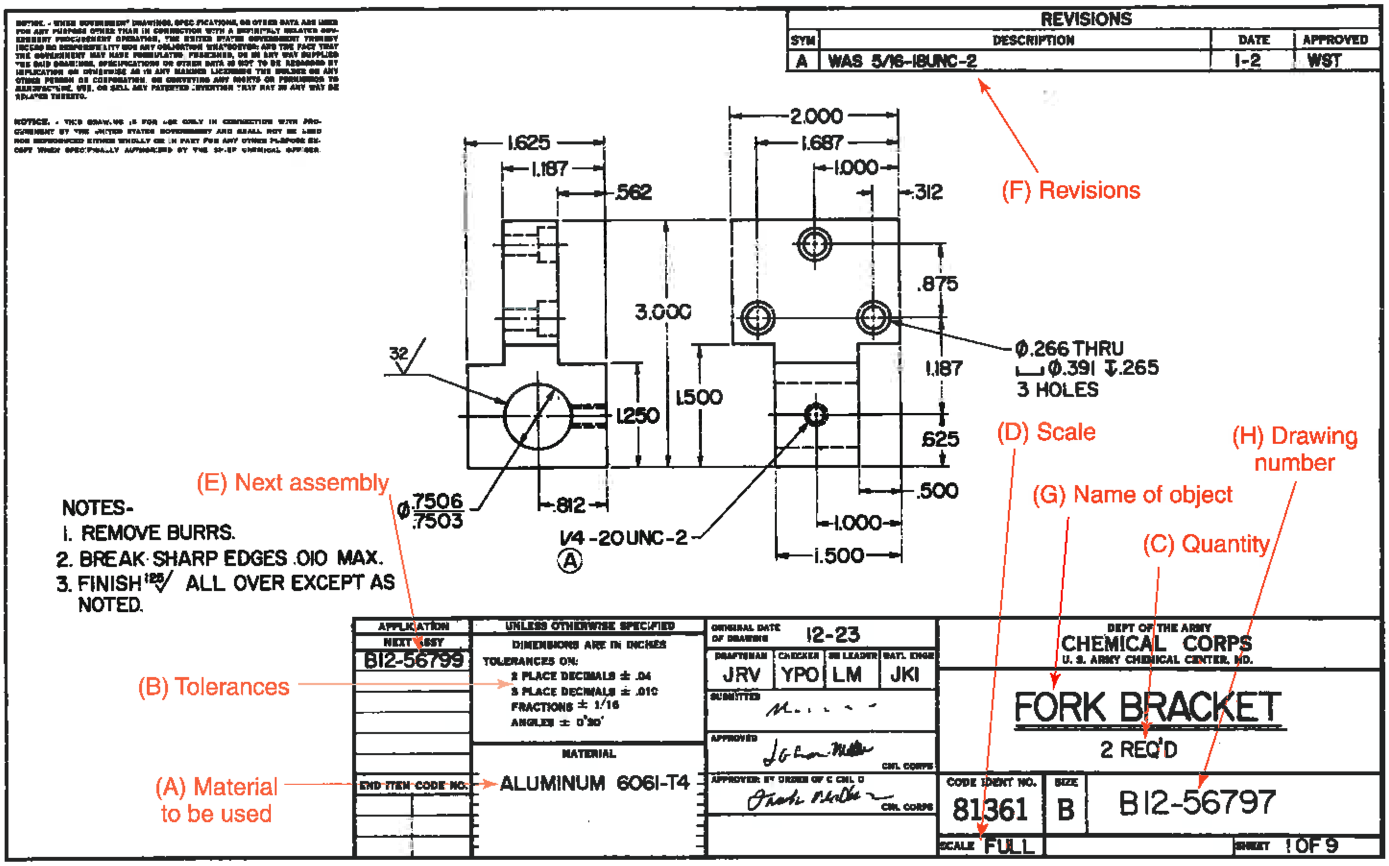


Figure 3-12. *A great deal of information is contained in the drawing's title block. The components highlighted here are standard on most drawings.*

superior (smoother) the finish of a machined surface, the more expensive it is to manufacture.

In the past, symbols were employed to indicate machined surfaces, **Figure 3-13.** These symbols may still be found on some older drawings. With so many machining techniques now in use, symbols such as these do not indicate, in sufficient detail, the quality of the surface finish required on a part.

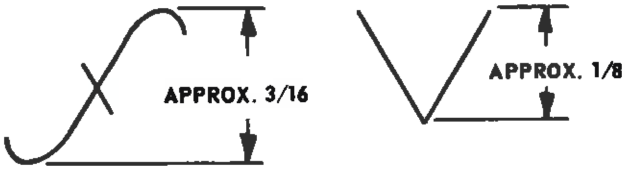


Figure 3-13. *These are older-style finish marks. They do not indicate the degree of smoothness required; they simply specify that the surface is machined. Finish marks of these types are still found on older drawings.*

The method presently used provides more complete surface information. Shown in **Figure 3-14,** a *check mark* and *number* are used to indicate surface roughness in microinches or micrometers. A ***microinch*** is one-millionth of an inch (0.000001"). A ***micrometer*** (micron) is one-millionth of a meter (0.000001 m) and is abbreviated μm.

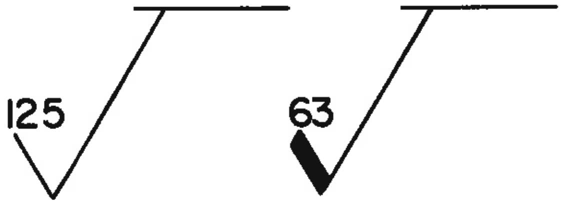


Figure 3-14. *Current surface finish marks. The number indicates the degree of smoothness in microinches—the larger the number, the rougher the finish.*

A machinist compares surface finishes to required specifications by using a ***surface roughness comparison standard*** as a guide, **Figure 3-15.** If the surface finish is critical, as it is in some jet engine components, the finish is measured electronically with a device called a ***profilometer,*** **Figure 3-16.**

3.2.3 Tolerances

The control of dimensions to achieve interchangeable manufacturing is known as tolerancing. This controls the size of the features of a part. A standard system of geometric dimensioning and tolerancing has been established.

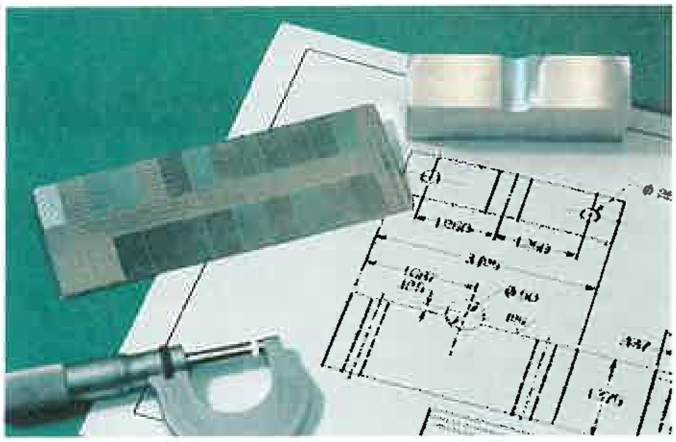


Figure 3-15. *A surface roughness comparison standard is used to check whether a milled surface meets the required specifications.*

Figure 3-16. *Surface roughness is best determined with a profilometer or electronic surface roughness gage. The probe on the unit is moved across the work and measures surface roughness electronically. A digital display presents the measured roughness value in microinches or micrometers. (Federal Products Co.)*

Tolerances are allowances, either oversize or undersize, that are permitted when machining or making a part. Refer again to **Figure 3-12(B).** Acceptable tolerances are shown on drawings in several different ways.

When the dimension is given in fractional inch units, the permissible tolerances can be assumed to be ±1/64", unless otherwise indicated.

The symbol "±" means that the machined surface can be *plus* (larger) or *minus* (smaller) by the dimension that follows and still be acceptable. In the example above, the dimension may be up to 1/64" larger or smaller than the dimension given on the drawing. This "plus *and* minus" tolerance is called a ***bilateral tolerance.***

If it is permissible to machine the part larger, but not smaller, the dimension on the drawing might read

$$2\,1/2^{\,+1/64}$$

If only a minus tolerance is permitted, the dimension might read

$$2\,1/2^{\,-1/64}$$

When the tolerance is only plus or only minus (one direction), it is called a ***unilateral dimension.***

Drawings dimensioned in decimal inches usually indicate that the work must be machined more precisely than dimensioning in fractional inches. The part can be used as long as the machined dimensions measure within these limits. Unless otherwise indicated, the tolerances can be assumed to be ±0.001″.

A *plus* tolerance may be shown as:

$$2.500^{\,+.001} \text{ or } \frac{2.501}{2.500}$$

A *minus* tolerance may be shown as:

$$2.500^{\,-.001} \text{ or } \frac{2.500}{2.499}$$

Metric tolerances are presented in the same way as decimal tolerances, **Figure 3-17.**

3.2.4 Quantity of Units

Also shown on the drawing is the *number of parts* (quantity) needed in each assembly. Refer to **Figure 3-12(C).** A work order, included with the job information received by the shop, gives the total number of units to be manufactured. This facilitates ordering the necessary materials, and will help in determining the most economical way to manufacture the pieces.

3.2.5 Drawing Scale

Drawings made other than actual size (1:1) are called ***scale drawings.*** The scale is usually shown in a section of the title block, **Figure 3-12(D).** A drawing made one-half size would have a scale of 1:2 (one-to-two). A scale of 2:1 (two-to-one) would mean that the drawing is twice the size of the actual part.

3.2.6 Assembly or Subassembly

Assembly or ***subassembly*** information is necessary to correctly fit the various parts together. The

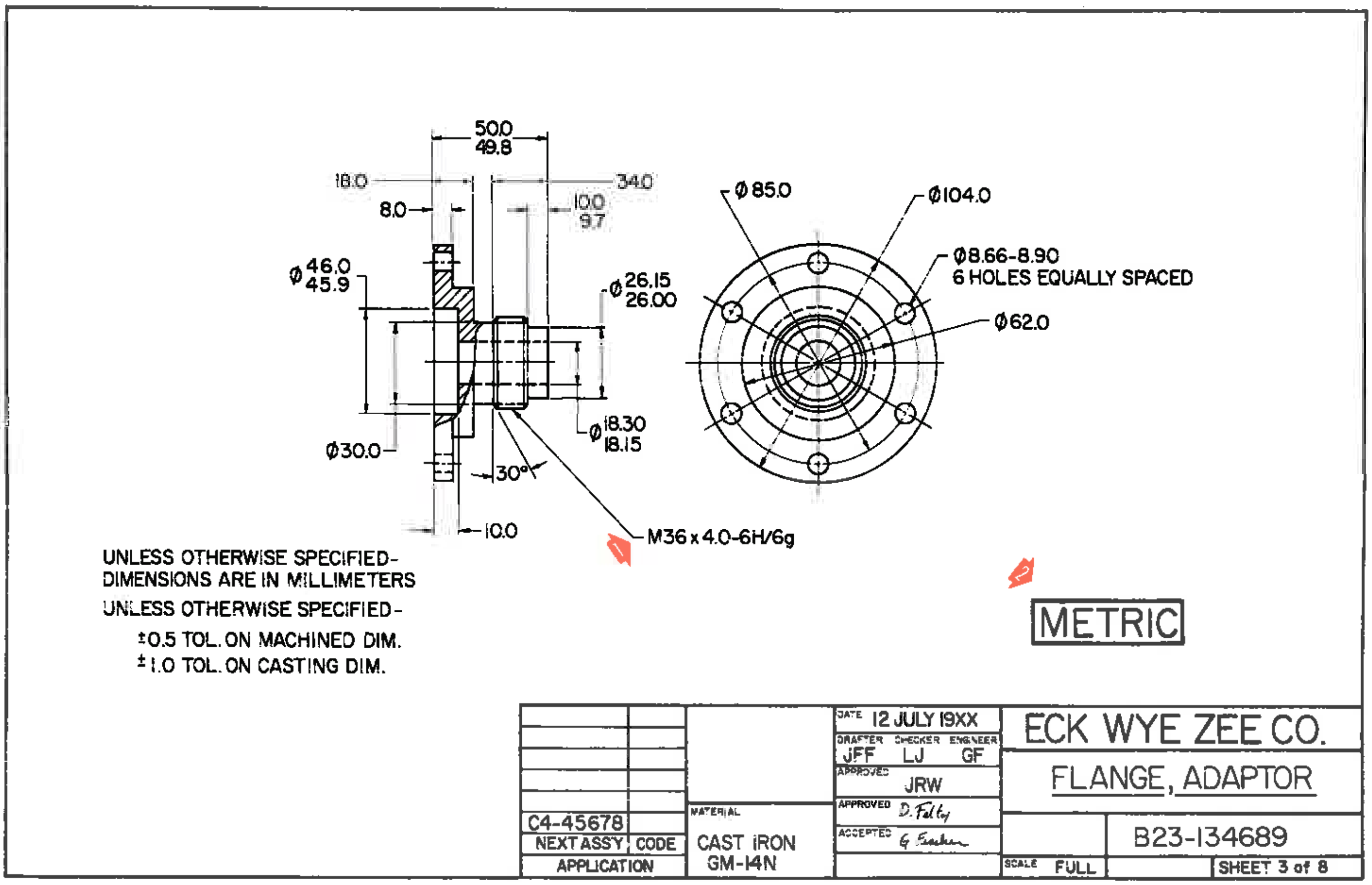


Figure 3-17. *A metric detail drawing. 1—Note that metric thread specifications are different from the more familiar UNC (coarse) and UNF (fine) series threads. The letter "M" denotes standard metric screw threads. The 36 indicates the nominal thread diameter in millimeters. The 4.0 denotes thread pitch in millimeters. The 6H and 6g are tolerance class designations. 2—To avoid possible misunderstanding, metric is shown on the drawing in large letters.*

term *application* is sometimes used in place of the term *next assembly*, **Figure 3-12(E).**

3.2.7 Revisions

Revisions indicate what changes were made to the original drawing and when they were made. Refer again to **Figure 3-12(F).**

3.2.8 Name of the Object

A portion of the title block provides this information. It tells the machinist the correct name of the piece, **Figure 3-12(G).**

3.3 TYPES OF PRINTS

The original drawings are seldom used in the shop because they might be lost, damaged, or destroyed. On many jobs, several sets of plans are required. There are several methods of duplicating original drawings:

- ***Blueprints.*** The term *blueprint* is often used to refer to all types of prints. An *actual* blueprint has white lines on a blue background. However, the blueprint process is seldom used today because of the time required to make a print.
- ***Diazo process.*** These are direct positive copies (dark lines on a white background) of the original drawing. They are often referred to as ***whiteprints*** or ***bluelines.***
- ***Xerographic (electrostatic) process.*** This process makes a copy of the original drawing. The print can be enlarged or reduced in size if necessary. Full color copies can be made on some xerographic machines.
- ***Microfilm process.*** This is a technique in which the original drawing is reduced by photographic means. Finished negatives can be stored in roll form or on cards, **Figure 3-18.** To produce a working print, the microfilm image is retrieved from files and enlarged onto photographic paper. The print is discarded or destroyed when it is no longer needed. Microfilms can also be viewed on a ***reader*** (machine for making enlarged projection on display screen). This technique is still widely employed for the storage of older drawings that would be too expensive to convert to computer data.

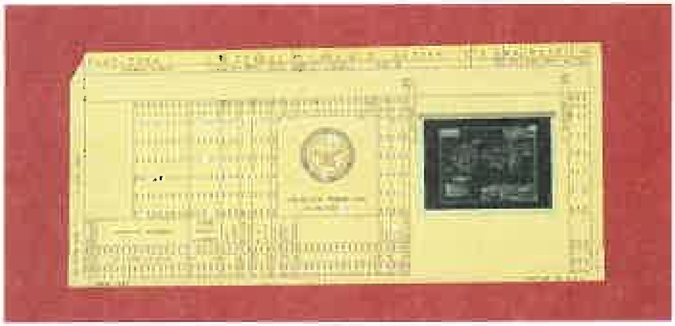


Figure 3-18. *The small negative on the microfilm aperture card is enlarged by a photographic process to the desired print size on a microfilm reader/printer. The enlarged print can be verified or confirmed on the view screen.*

- ***Computer-generated prints.*** The prints are generated on a plotter (automatic drafting machine) from information stored electronically in computer memory, **Figure 3-19.**

```
Regenerating paperspace.
Regenerating drawing.
Command:
```

Figure 3-19. *Computer-generated print. Left—Many companies now use computer-aided design and drafting (CAD) techniques to prepare drawings. (Autodesk, Inc.) Right—Prints are generated on a plotter from CAD-developed information. (Hewlett-Packard Marketing Communications)*

This same information can also be used to control machine tools, using *CAM* (computer-aided manufacturing). When these methods are used, the overall manufacturing technique is called *CIM* (computer-integrated manufacturing). More information on computers in manufacturing will be included in later chapters.

3.4 TYPES OF DRAWINGS USED IN THE SHOP

Working drawings, also called ***prints,*** establish the standards for the product and show the craft worker what to make. There are two major kinds of working drawings:

- ***Detail drawings.*** These consist of a drawing (usually multiview) of the part with dimensions and other information for making the part, **Figure 3-20.**
- ***Assembly drawings.*** These drawings show where and how the parts, described on detail drawings, fit into the completed assembly. See **Figure 3-21.**

On large or complex products, ***subassembly drawings*** are used to show the assembly of a small portion of the completed object, **Figure 3-22.**

Some assembly and subassembly drawings are shown as ***exploded pictorial drawings*** (a drawing with parts separated, but in proper relationship). One is shown in **Figure 3-23.**

In most instances, a detail drawing provides information on just one item. However, if the mechanism is small in size or if it is composed of only a few parts, the detail and assembly drawings may appear on the same sheet, **Figure 3-24.**

3.5 PARTS LIST

Parts are identified by *circled numbers* and are listed in a note. Some drawings will also include a ***parts list*** or ***bill of materials*** listing all of the parts used in the assembly. See **Figure 3-25.**

3.6 DRAWING SIZES

Most firms centralize the preparation and storage of drawings in the engineering department. Generally, engineers and drafters prepare drawings

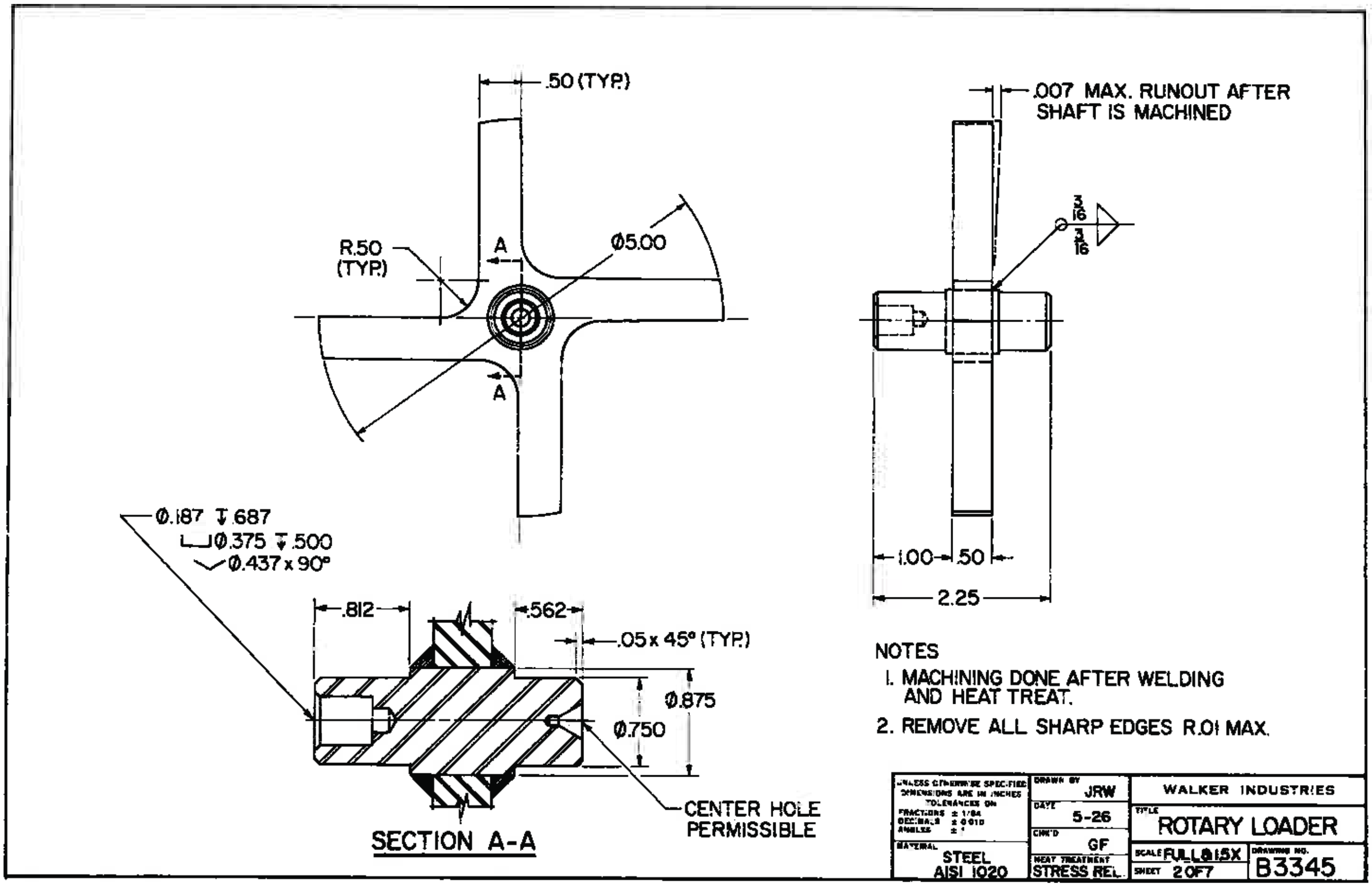


Figure 3-20. *A detail drawing contains all of the information needed to produce the part.*

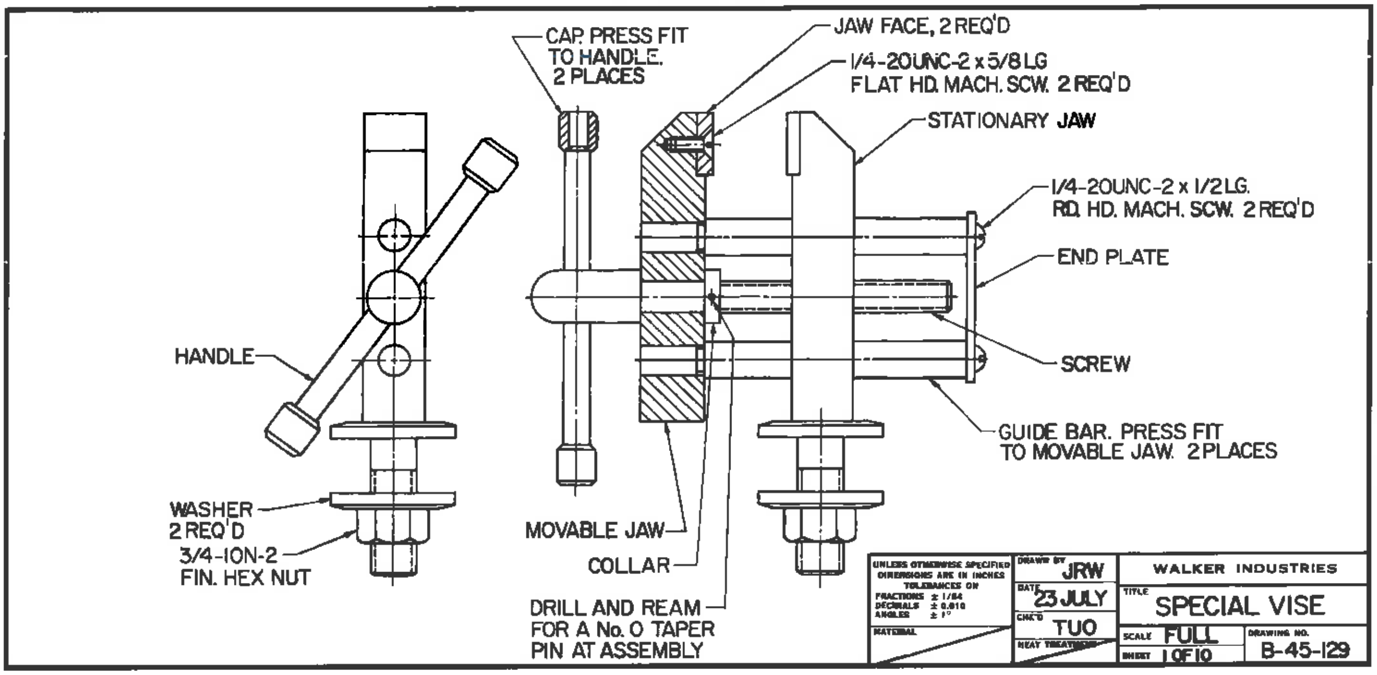


Figure 3-21. An assembly drawing shows how various parts fit together.

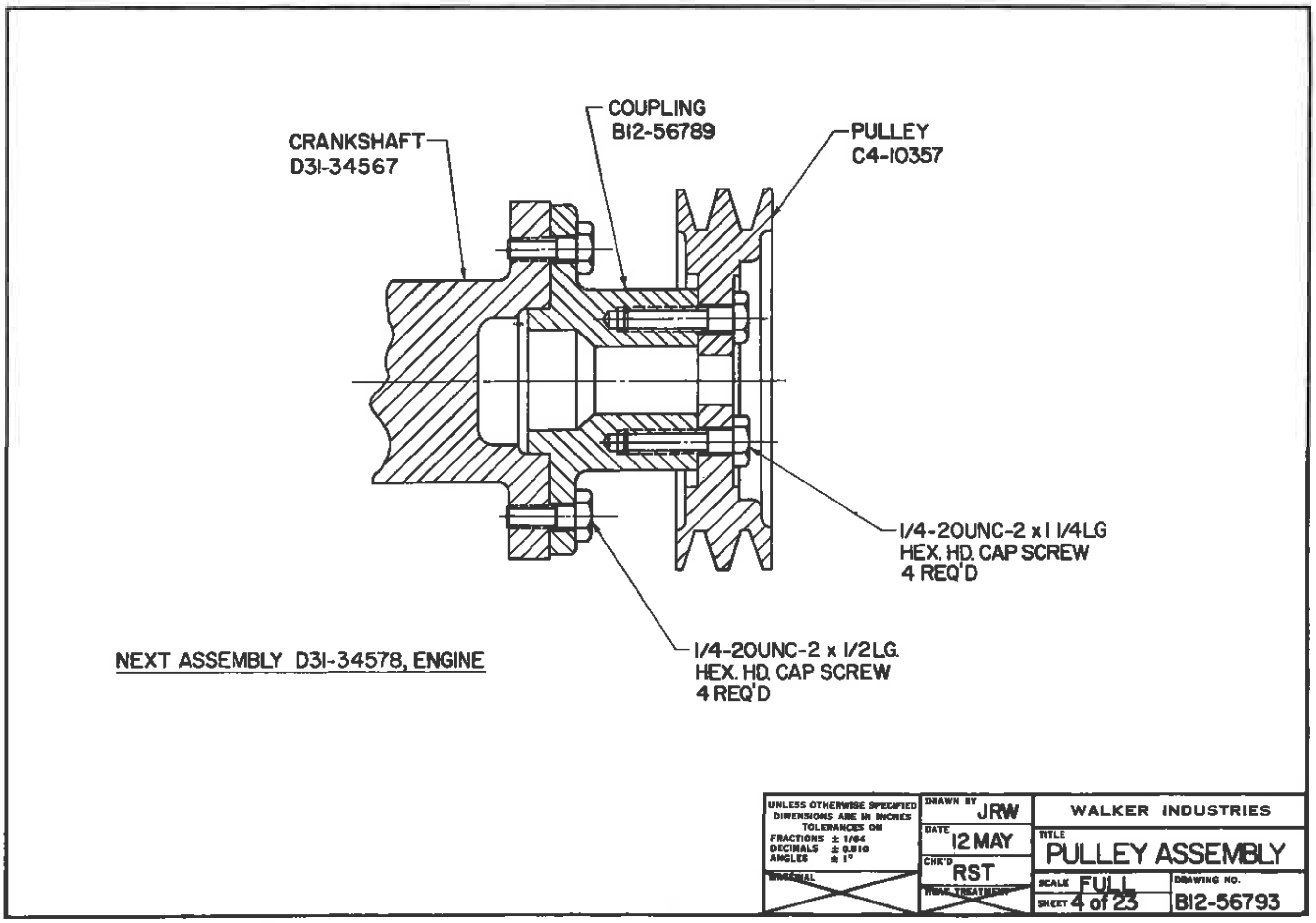


Figure 3-22. A subassembly drawing contains the assembly of only a portion of the entire product.

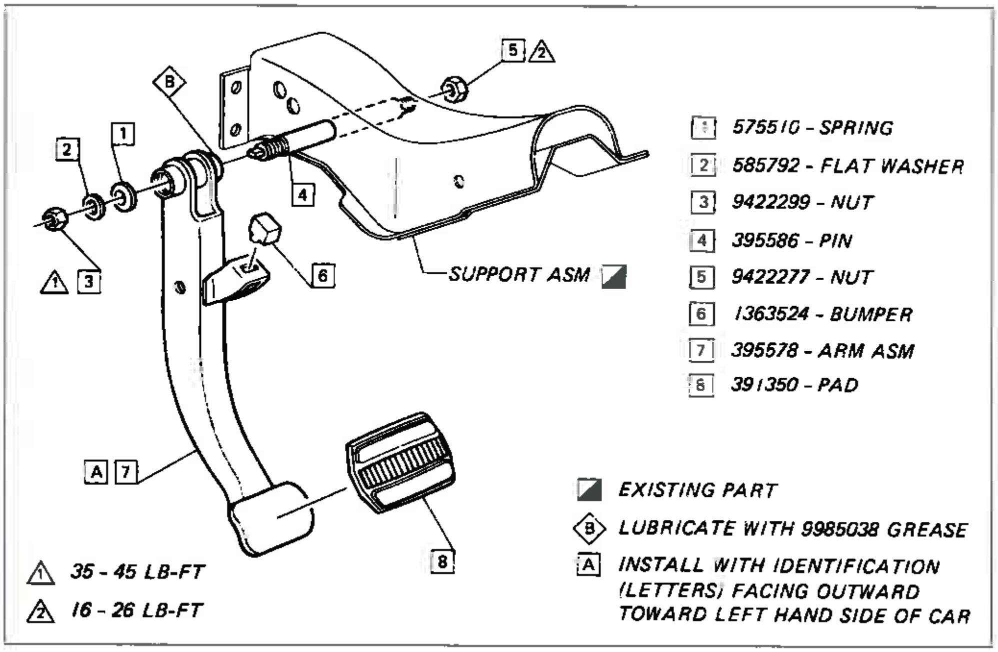


Figure 3-23. *Exploded pictorial drawings are often used with semiskilled workers who have received a minimum of training in print reading. (General Motors Corp.)*

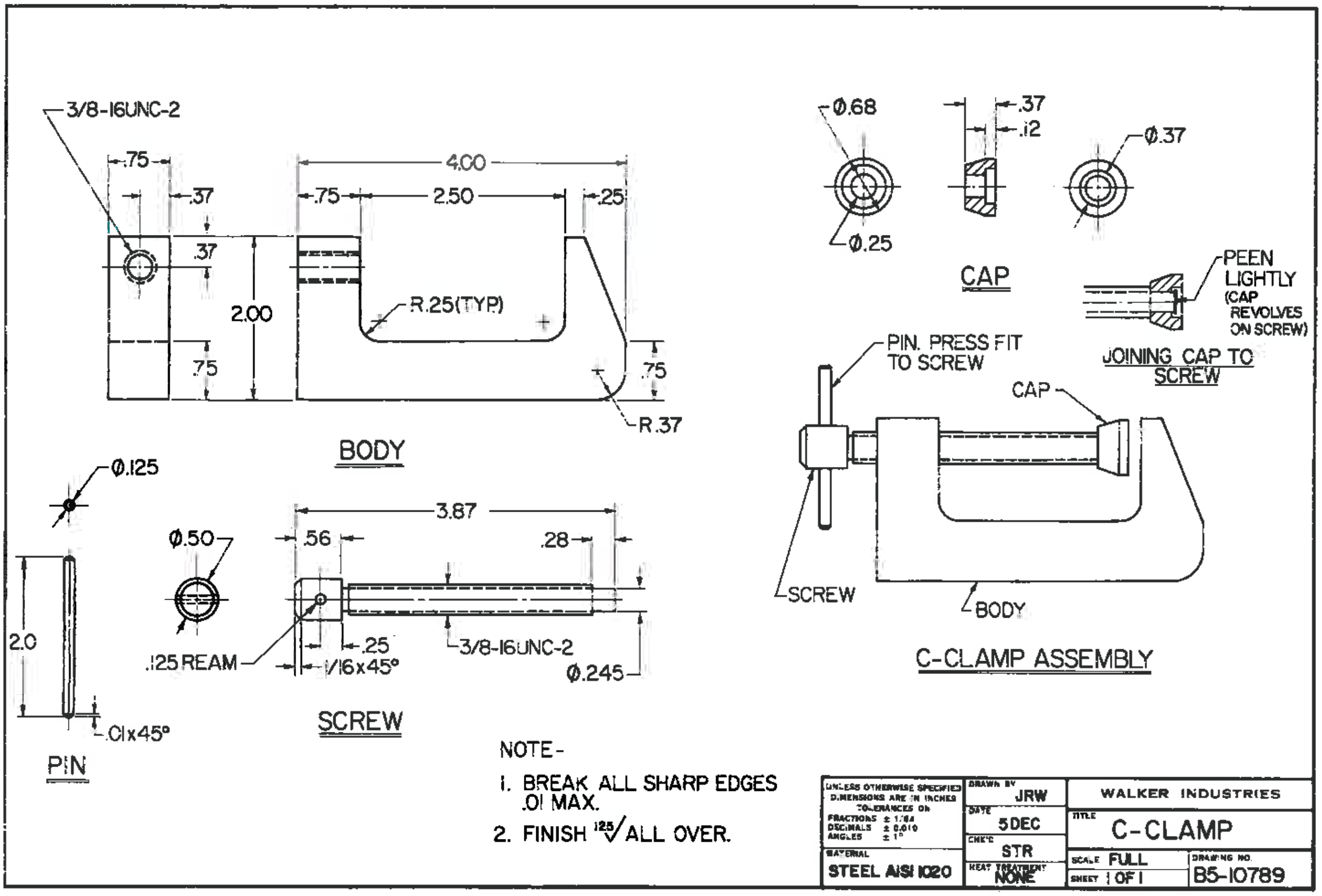
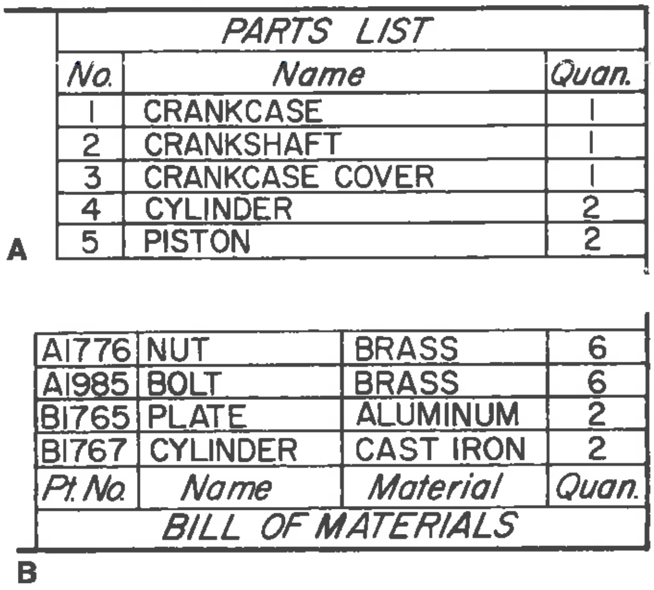


Figure 3-24. *A detail and assembly drawing on the same sheet.*



PARTS LIST

No.	Name	Quan.
1	CRANKCASE	1
2	CRANKSHAFT	1
3	CRANKCASE COVER	1
4	CYLINDER	2
5	PISTON	2

A

A1776	NUT	BRASS	6
A1985	BOLT	BRASS	6
B1765	PLATE	ALUMINUM	2
B1767	CYLINDER	CAST IRON	2
Pt. No.	Name	Material	Quan.

BILL OF MATERIALS

B

Figure 3-25. *A list of parts and materials is normally included with the drawings for a project. A—A typical, but partial, parts list. B—An example of a partial bill of materials.*

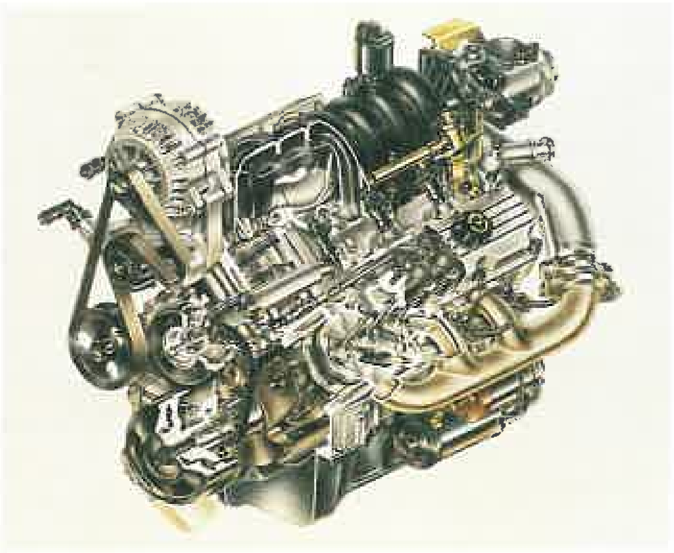


Figure 3-26. *A high degree of precision is needed to produce the parts used in this engine. The tolerances allowed for the shape and location of features on the parts must not be exceeded. (Buick Div. of GMC)*

on standard-size sheets. This simplifies the stocking, handling, and storage of the completed drawings.

Standard sizes for drawing sheets include the following:

US CONVENTIONAL SHEET SIZES

A size = 8 1/2″ × 11″
B size = 11″ × 17″
C size = 17″ × 22″
D size = 22″ × 34″
E size = 34″ × 44″

SI METRIC SHEET SIZES

A4 size = 210 × 297 mm
A3 size = 297 × 420 mm
A2 size = 420 × 594 mm
A1 size = 594 × 841 mm
A0 size = 841 × 1189 mm

Also, for convenience in filing and locating drawings in storage, each drawing has an identifying number, **Figure 3-12(H).**

3.7 GEOMETRIC DIMENSIONING AND TOLERANCING

Conventional tolerancing is appropriate for many products. However, for accurately machined parts, the amount of variation (tolerances) in form (shape and size) and position (location) may need to be more strictly defined. This definition provides the precision needed to allow for the most economical manufacture of parts. See **Figure 3-26.**

Geometric dimensioning and tolerancing is a system that provides additional precision compared to conventional dimensioning. It ensures that parts can be easily interchanged.

Only a brief introduction to geometric dimensioning and tolerancing is included in this text. Detailed information can be found in the publication ASME Y14.5M-1994.

3.7.1 Definitions

Geometric characteristic symbols are employed to provide clarity and precision in communicating design specifications. See **Figure 3-27.** These symbols are standardized by the American Society of Mechanical Engineers (ASME). ***Geometric tolerance*** is a general term that refers to tolerances which control form, profile, orientation, location, and runout.

A ***basic dimension*** is a numerical value denoting the exact size, profile, orientation, or location of a feature. The ***true position*** of a feature is its theoretically exact location as established by basic dimensions. A ***reference dimension*** is a dimension provided for information only. It is not used for production or inspection purposes. See **Figure 3-28.**

Datum is an exact point, axis, or plane. It is the origin from which the location or geometric characteristic of features of a part is established. It is identified by a solid triangle with an identifying letter. See **Figure 3-29.** ***Feature*** is a general term applied to a physical portion of a part, such as a surface, pin,

Symbol for:	ASME Y14.5
Straightness	⏤
Flatness	⏥
Circularity	○
Cylindricity	⌭
Profile of a line	⌒
Profile of a surface	⌓
All-around profile	◄─⊖─
Angularity	∠
Perpendicularity	⟂
Parallelism	//
Position	⌖
Concentricity/coaxiality	◎
Symmetry	⌯
Circular runout	*↗
Total runout	*⌰
At maximum material condition	Ⓜ
At least material condition	Ⓛ
Regardless of feature size	NONE
Projected tolerance zone	Ⓟ
Diameter	Ø
Basic dimension	$\boxed{30}$
Reference dimension	(30)
Datum feature	[A]─◄
Datum target	Ø6 A1
Target point	X
Dimension origin	⌱
Feature control frame	|⌖|Ø0.5Ⓜ|A|B|C|
Conical taper	⌲
Slope	⌳
Counterbore/spotface	⌴
Countersink	⌵
Depth/deep	↧
Square (shape)	□
Dimension not to scale	$\underline{15}$
Number of times/places	8X
Arc length	$\overset{\frown}{105}$
Radius	R
Spherical radius	SR
Spherical diameter	SØ

* May be filled

Figure 3-27. *Symbols used to specify positional and form tolerances in geometric dimensioning. (American National Standards Institute)*

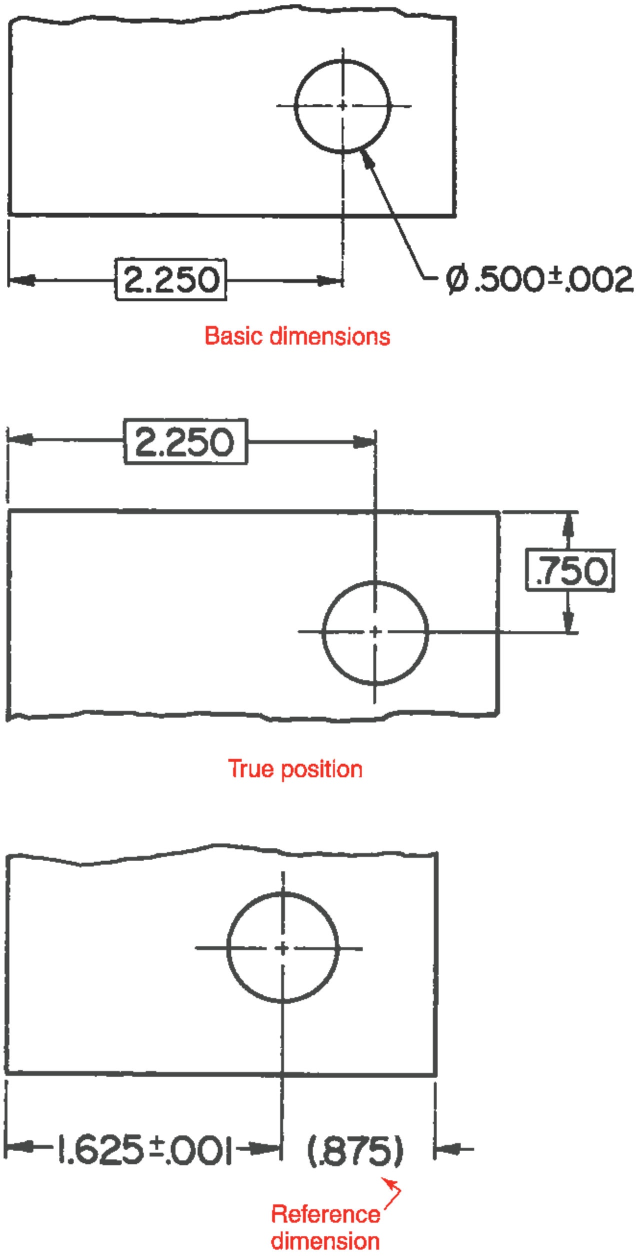


Figure 3-28. *Basic dimensions are usually indicated by being enclosed in a rectangular frame. They are not toleranced. True position is the theoretical exact location of feature. It is established by basic dimensions. Reference dimensions are not used for production or inspection purposes. On a drawing, they are shown enclosed in parentheses.*

hole, or slot. A ***datum feature*** is the actual feature of a part used to establish a datum. See **Figure 3-30.**

Maximum material condition (MMC) is the condition in which the size of a feature contains the maximum amount of material within the stated limits of size. Examples include a minimum hole diameter and maximum shaft diameter, both of which result in the greatest possible amount of material

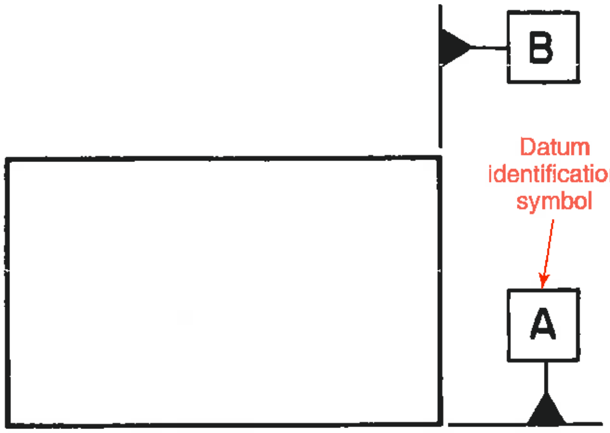


Figure 3-29. *Datums are exact points, axes, or planes from which features of a part are located.*

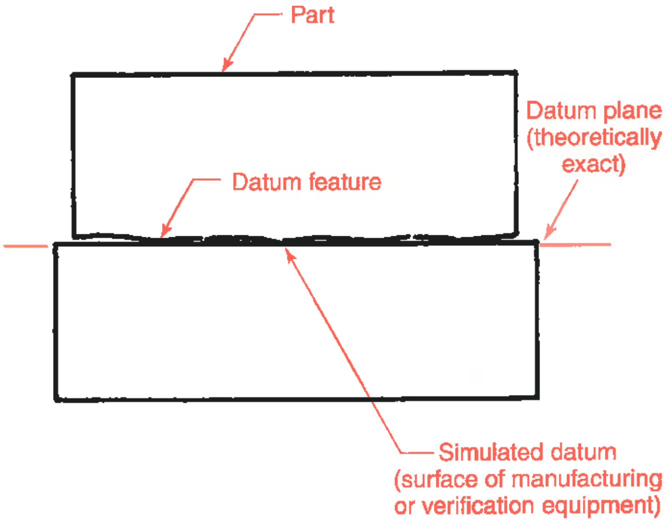


Figure 3-30. *A datum feature is a physical feature on a part used to establish a datum.*

being used. See **Figure 3-31.** MMC is indicated by an *M* within a circle.

Least material condition (LMC) is the condition in which the size of a feature contains the least amount of material within the stated tolerance limits. Examples include a maximum hole diameter and a minimum shaft diameter. See **Figure 3-32.** LMC is indicated by an *L* within a circle.

Regardless of feature size (RFS) specifies that the size of a feature tolerance must not be exceeded. RFS is assumed for all geometric tolerances unless otherwise specified.

The maximum and minimum sizes of a feature are called the ***limits of size.*** See **Figure 3-33.** The measured size of a part after it is manufactured is the ***actual size.***

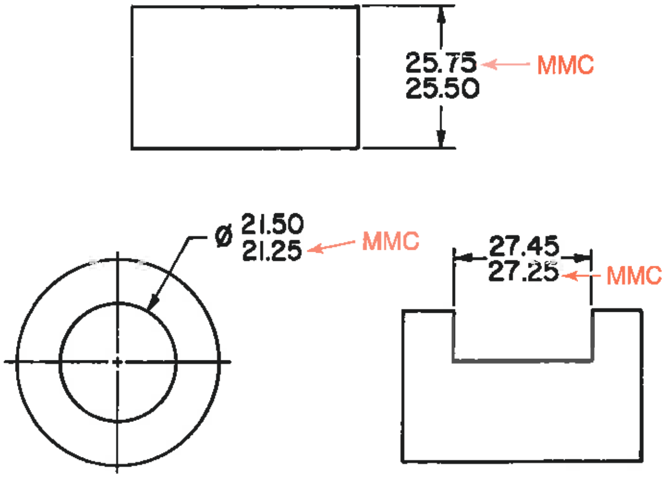


Figure 3-31. *Maximum material condition (MMC) indicates that the size of a feature contains the maximum amount of material within the stated tolerance limits.*

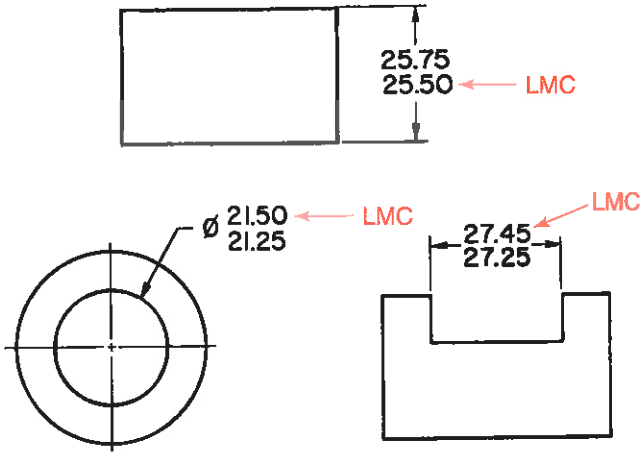


Figure 3-32. *Least material condition (LMC) indicates that the size of a feature contains the least amount of material within the stated limits of size.*

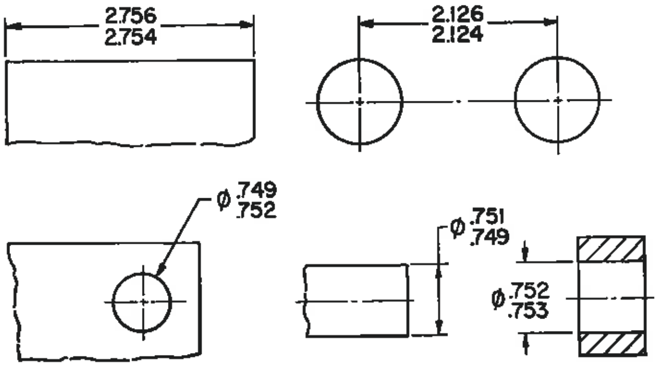


Figure 3-33. *Limits of size are the maximum and minimum sizes of a feature.*

3.7.2 Application of Geometric Dimensioning and Tolerancing

Datum identification symbol. A datum identifying symbol, **Figure 3-34,** consists of a square frame that contains the *datum reference letter.* All letters but *I, O,* and *Q* may be used. A rectangular frame with the datum reference letter preceded and followed by a dash may be found on older drawings.

Feature control frame. A feature control frame is used when a location or form tolerance is related to a datum. It contains the geometric symbol, allowable tolerance, and the datum reference letter(s). It is connected to an extension line of the feature, a leader running to the feature, or below a leader-directed note of the feature, **Figure 3-35.**

Datum references indicated on the right end of the feature control frame are read from left to right. The letters signify datum preference. They establish three mutually perpendicular planes, **Figure 3-36.**

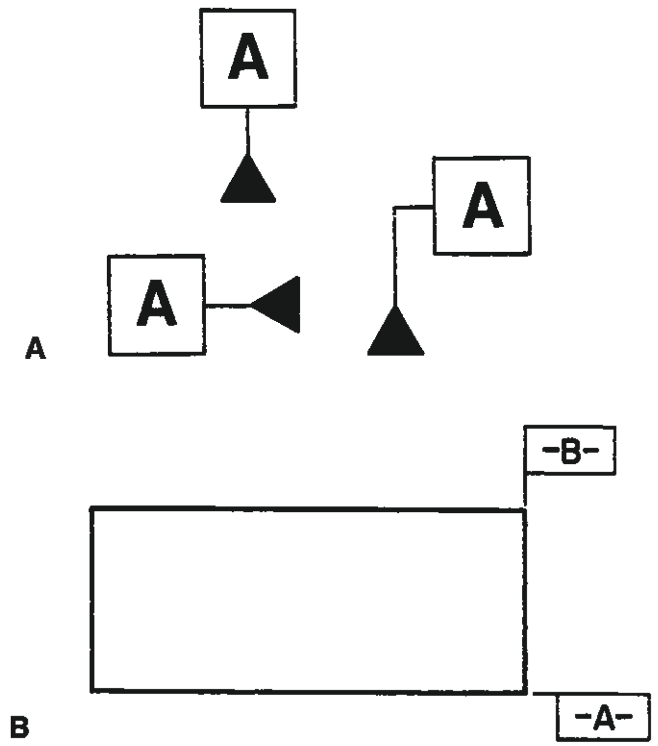
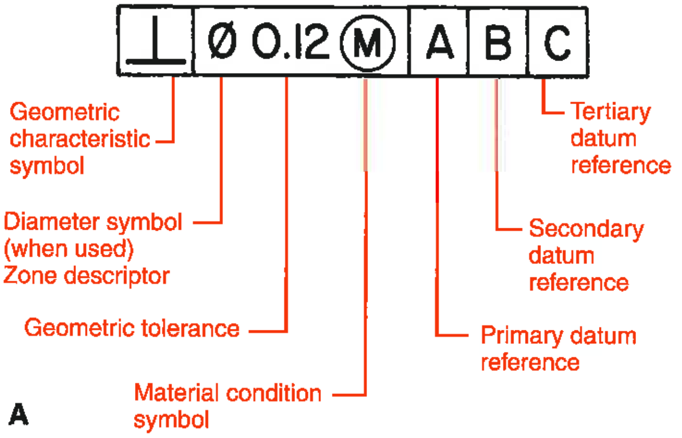


Figure 3-34. *Datum points and surfaces are identified by a datum identification symbol. A—Datum identification symbols used on new drawings. B—This type of datum symbol is not used currently, but is still found on old drawings.*



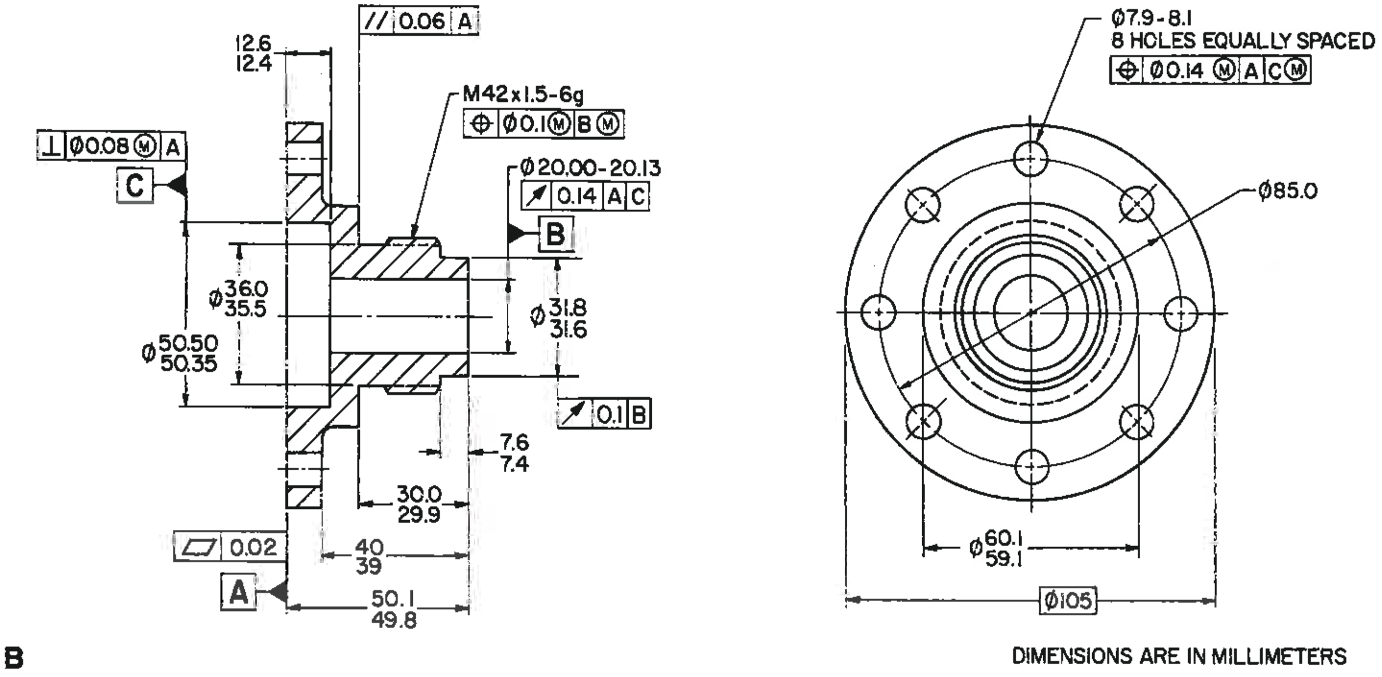


Figure 3-35. *A feature control frame is employed when a location or form tolerance is related to a datum. A—Components of a feature control frame. B—Feature control frames are used to specify tolerances on this drawing.*

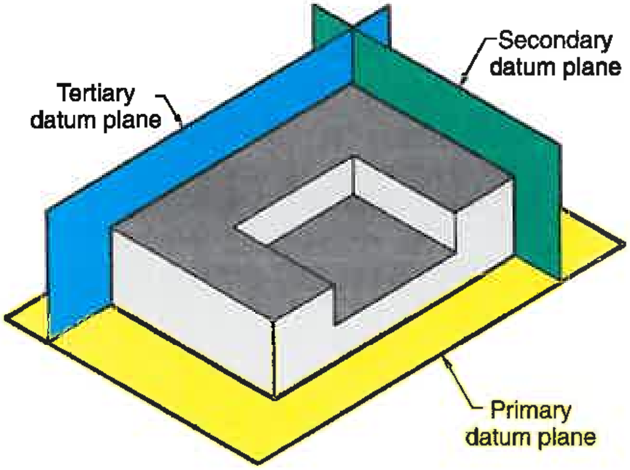


Figure 3-36. *Datum references are perpendicular planes. The first datum referenced is the primary datum, followed by the secondary and tertiary datums.*

3.7.3 Form Geometric Tolerances

Form geometric tolerances control flatness, straightness, circularity (roundness), and cylindricity. They are indicated by the symbols shown in **Figure 3-37.** Form tolerances control only the variation permitted on a single feature and are used when form variation is less than that permitted by size tolerance.

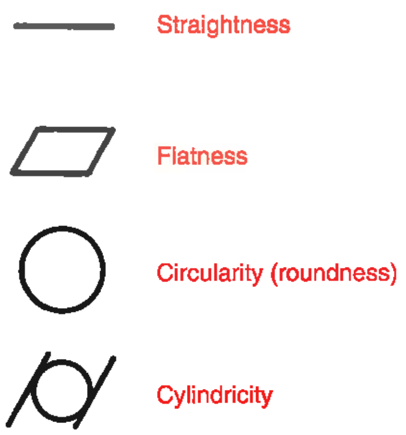


Figure 3-37. *Form geometric symbols.*

Flatness is a measure of the variation of a surface perpendicular to its plane. The flatness geometric tolerance specifies the two parallel planes within which all points of a surface must lie, **Figure 3-38.**

Straightness describes how closely the surface of an object is to a line. A straightness geometric tolerance establishes a tolerance zone of uniform width along a line. All elements of the surface must lie within this zone, **Figure 3-39.**

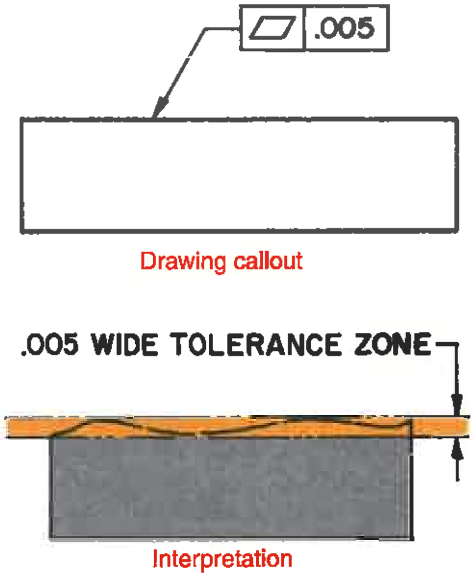


Figure 3-38. *The flatness geometric form tolerance specifies the two parallel planes within which a surface must lie.*

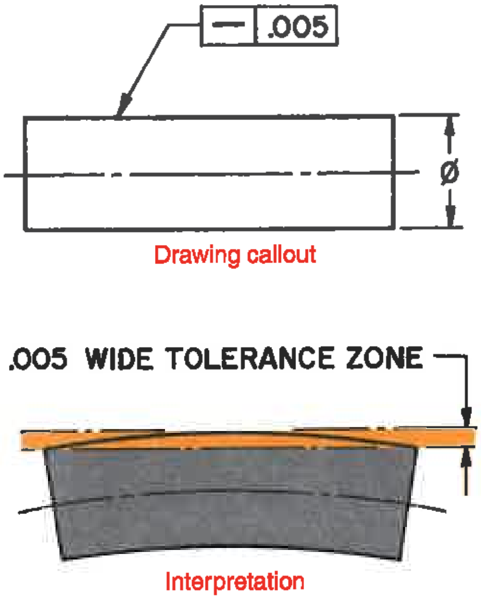
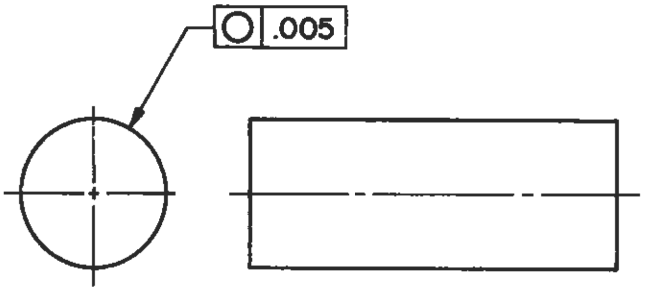
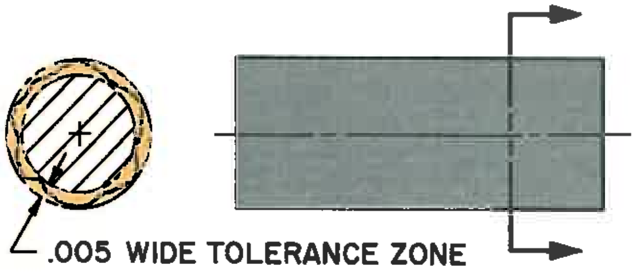


Figure 3-39. *A straightness geometric form tolerance establishes a tolerance zone of uniform width along a straight line. All elements of the surface must lie within this zone.*

Circularity is characterized by any given cross section taken perpendicular to the axis of a cylinder or a cone, or through the common center of a sphere. A circularity (roundness) geometric tolerance specifies a tolerance zone bounded by two concentric circles, indicated on a plane perpendicular to the axis of a cylinder or a cone, within which each circular element must lie. It is a single cross-sectional tolerance. See **Figure 3-40.**



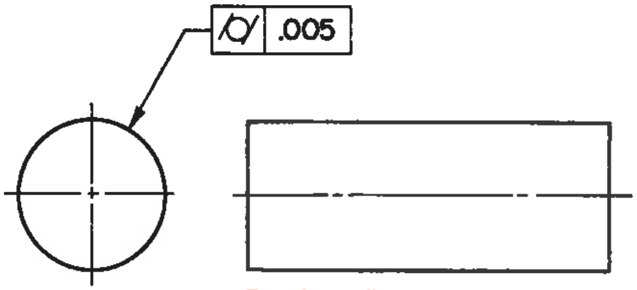
Drawing callout



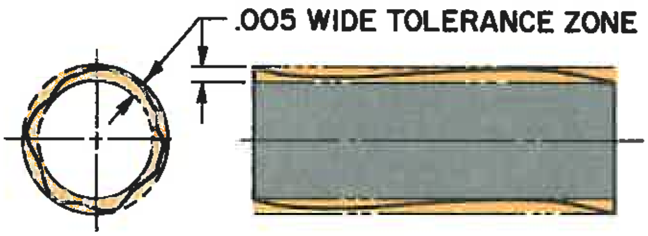
Interpretation

Figure 3-40. *A circularity geometric tolerance specifies a tolerance zone bounded by two concentric circles on a plane perpendicular to the axis of a cylinder or cone, within which each circular element must lie.*

Cylindricity represents a surface in which all points are an equal distance from a common center. The cylindricity geometric tolerance establishes a tolerance zone that controls the diameter of a cylinder throughout its entire length. It consists of two concentric cylinders within which the actual surface must lie. This tolerance covers both the circular and longitudinal elements. See **Figure 3-41.**



Drawing callout



Interpretation

Figure 3-41. *The cylindricity geometric tolerance establishes a tolerance zone that controls the diameter of a cylinder throughout its entire length.*

3.7.4 Profile Geometric Tolerances

A ***profile geometric tolerance*** controls the outline or contour of an object and can be represented by an external view or by a cross section through the object. It is a boundary along the true profile in which elements of the surface must be contained. The symbols used to indicate profile tolerances are shown in **Figure 3-42.**

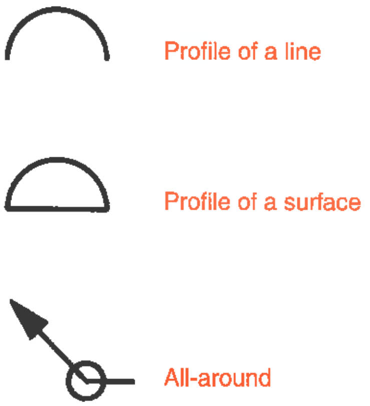


Figure 3-42. *Profile geometric tolerance symbols. When a tolerance is specified for all sides of an object, the "all-around" symbol is used.*

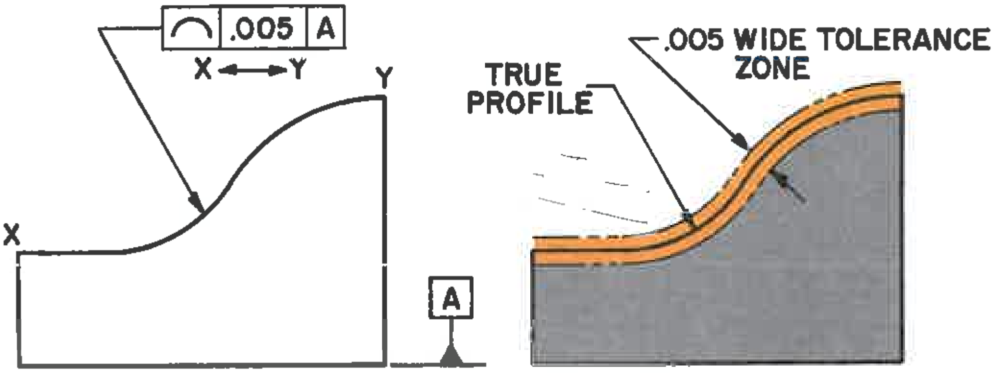
A *profile line geometric tolerance* is a two-dimensional (cross-sectional) tolerance zone extending along the length of the element. It is located using basic dimensions, **Figure 3-43.**

The *profile surface geometric tolerance* is three-dimensional and extends along the length and width of the surface. For proper orientation of the profile, a datum reference is usually required, **Figure 3-44.**

3.7.5 Orientation Geometric Tolerances

Orientation geometric tolerances control the degree of parallelism, perpendicularity, or angularity of a feature with respect to one or more datums. There are three orientation tolerances, **Figure 3-45.**

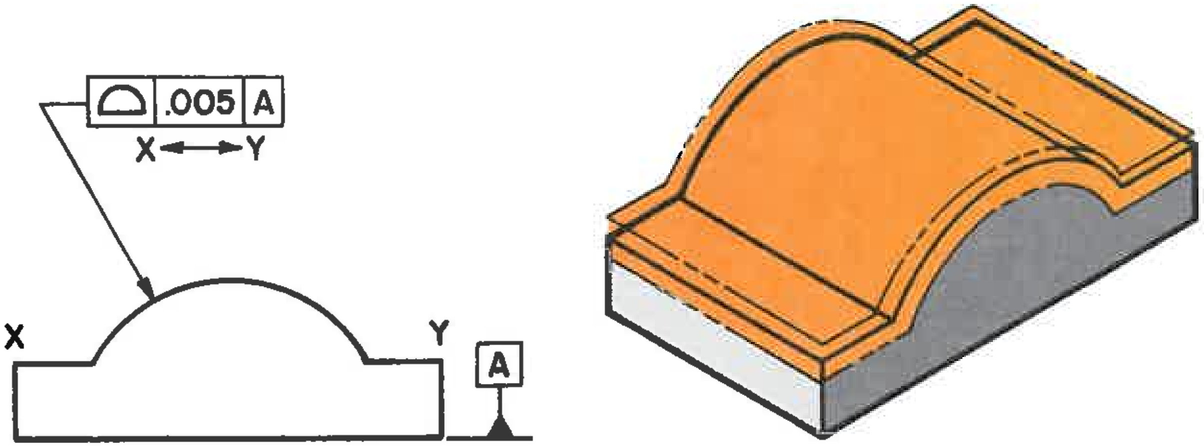
Angularity is concerned with the position of a surface or axis at a specified angle to a datum plane or axis. The specified angle must be other than 90°. An *angularity geometric tolerance* establishes a tolerance zone defined by two parallel lines, planes, or a cylindrical zone at a specified basic angle other than 90°. The line elements, surface, or axis of the considered feature must lie within this zone, **Figure 3-46.**



Drawing callout

Interpretation

Figure 3-43. *A profile line geometric tolerance is a two-dimensional tolerance zone extending along the length of the considered element.*



Drawing callout

Interpretation

Figure 3-44. *The profile surface geometric tolerance is three-dimensional and extends along the length and width of the surface.*

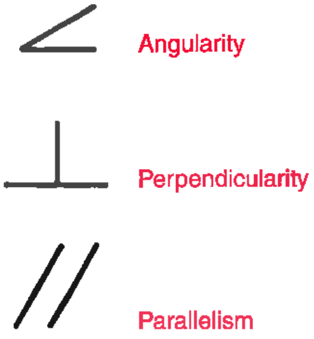


Figure 3-45. *Orientation geometric tolerance symbols.*

A *perpendicularity geometric tolerance* specifies a tolerance zone at right angles to a given datum or axis. It is described by two parallel lines, planes, or a cylindrical tolerance zone. The line, surface, or axis of the considered feature must lie within this zone, **Figure 3-47.**

Parallelism describes how close all elements of a line or surface are to being parallel (equidistant) to a given datum plane or axis. A *parallelism geometric tolerance* is a tolerance zone defined by two lines parallel to a datum within which the elements of a surface or axis must lie, **Figure 3-48.**

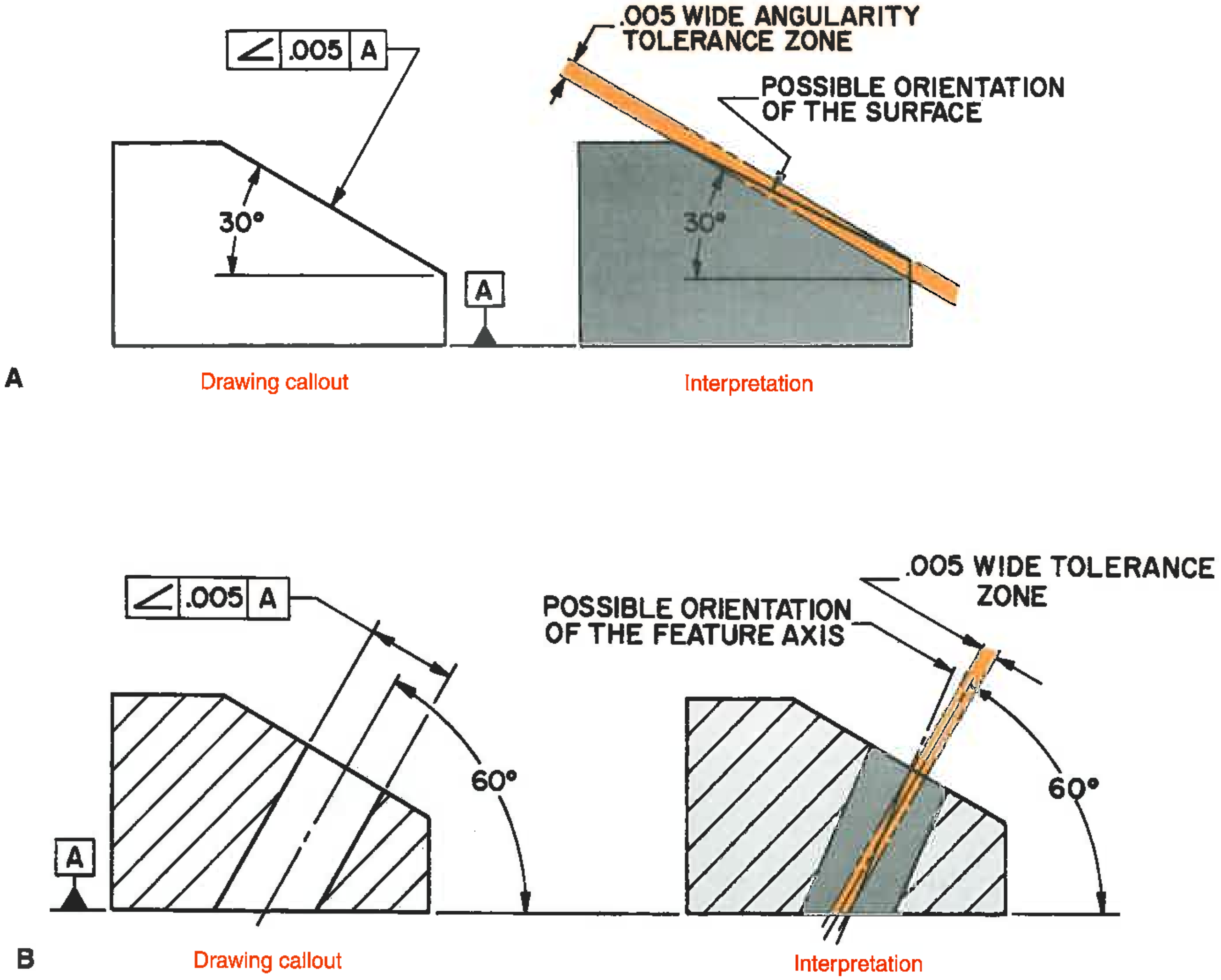


Figure 3-46. *An angularity geometric tolerance establishes a tolerance zone defined by two parallel lines, planes, or a cylindrical zone at a specified basic angle other than 90°. A—Angularity of a surface. B—Angularity of an axis.*

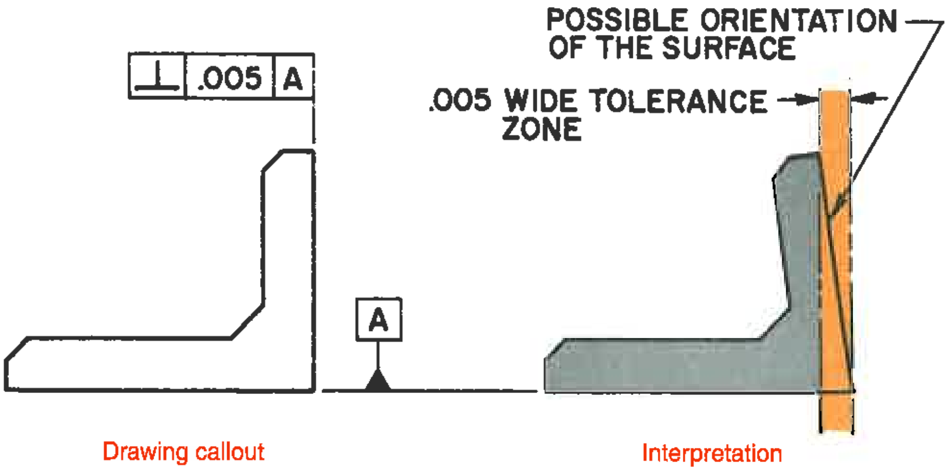
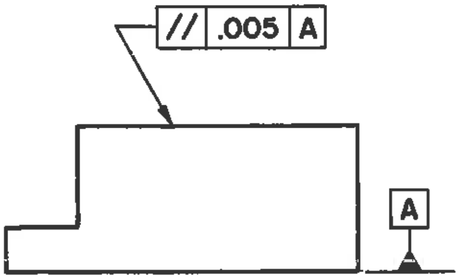
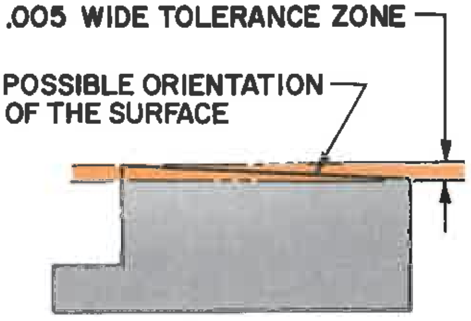


Figure 3-47. *The line, surface, or axis of a considered feature must lie within the perpendicularity geometric tolerance zone.*



Drawing callout



Interpretation

Figure 3-48. *A parallelism geometric tolerance is a tolerance zone defined by two lines parallel to a datum within which the elements of a surface or axis must lie.*

3.7.6 Location Geometric Tolerances

Location geometric tolerances are employed to establish the location of features and datums. They define the zone within which the center, axis, or center plane of a feature may vary from a true (theoretically exact) position. Location tolerances are also known as *positional tolerances* and include position, concentricity, and symmetry. See **Figure 3-49.**

Basic dimensions establish the true position of a feature from specified datums and related features. A *positional geometric tolerance* establishes how far a feature may vary from its true position, **Figure 3-50.**

Concentricity defines the relationship between the axes of two or more of an object's cylindrical features. A *concentricity geometric tolerance* is expressed

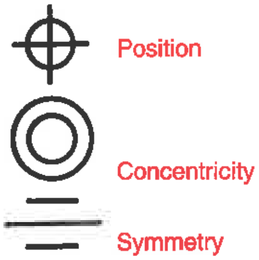
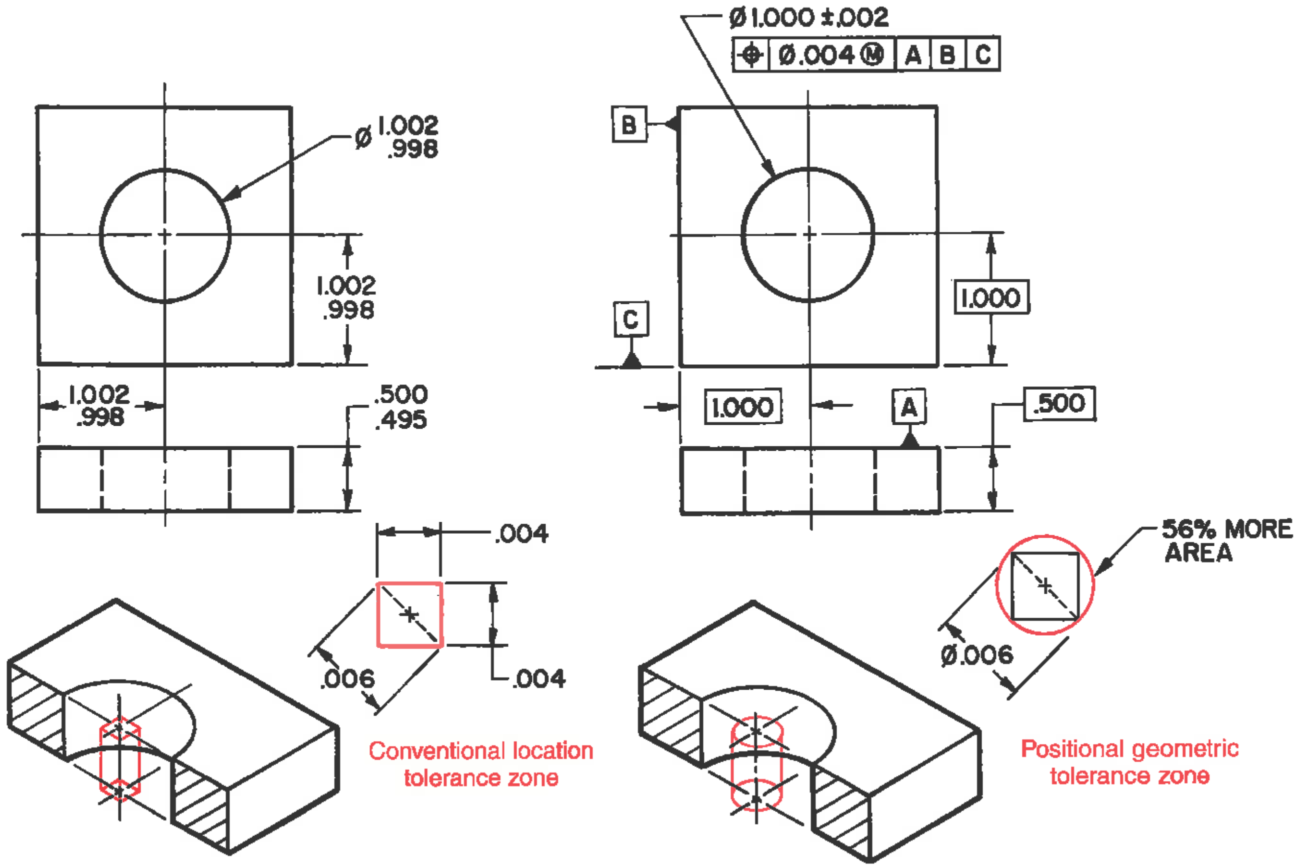


Figure 3-49. *Location or positional tolerance symbols.*



Conventional location tolerance zone

Positional geometric tolerance zone

Figure 3-50. *A positional geometric tolerance establishes how far a feature may vary from its true position.*

as a cylindrical tolerance zone. The axis or center point of this zone coincides with a datum axis, **Figure 3-51.**

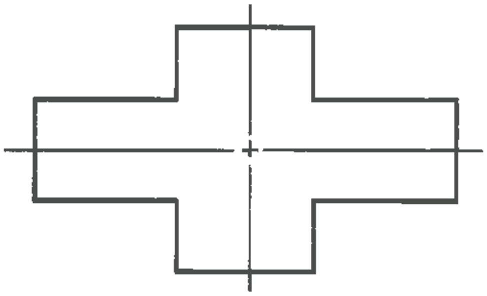
Since this tolerance is sometimes difficult and time-consuming to verify, runout or positional geometric tolerances are often used instead.

Symmetry indicates equal or balanced proportions on either side of a central plane or datum, **Figure 3-52.** A *symmetry geometric tolerance* is a zone within which the symmetrical surfaces align with the datum of a center plane or axis, **Figure 3-53.**

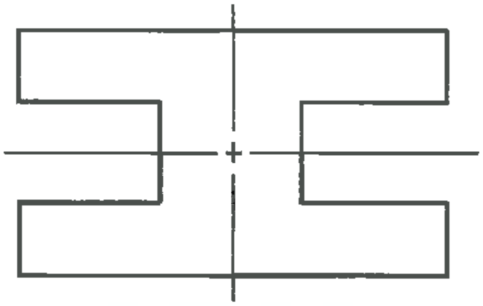
3.7.7 Runout Geometric Tolerances

There are two types of ***runout geometric tolerances***—total runout and circular runout. These tolerances are indicated by the symbols shown in Figure 3-54. Runout tolerances are used to control runout of surfaces around or perpendicular to a datum axis.

Total runout controls circularity, straightness, angularity, and cylindricity of a part when applied to surfaces rotated around a datum axis, **Figure 3-55.** The entire surface must lie within the tolerance zone.

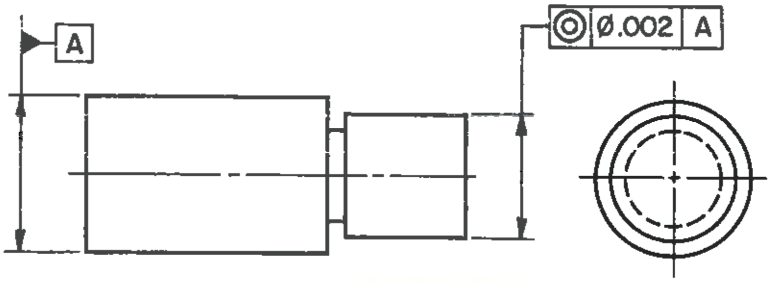


Symmetrical double tab

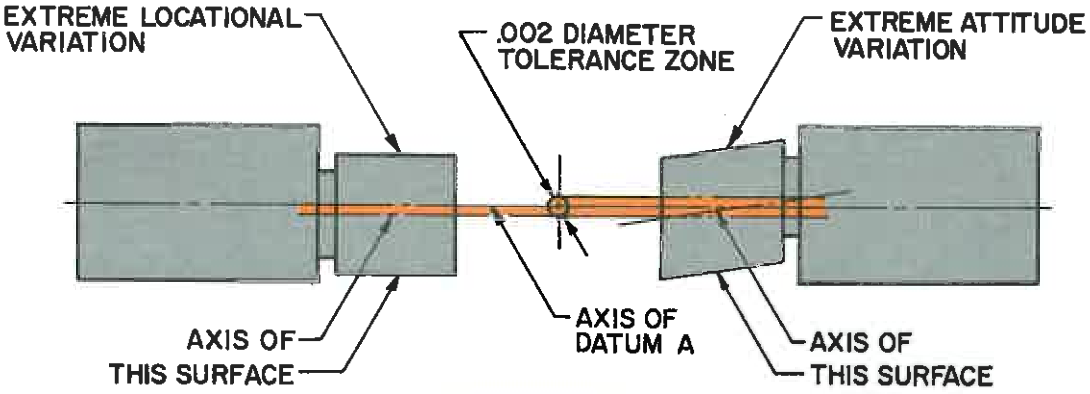


Symmetrical double slot

Figure 3-52. *Symmetry indicates equal or balanced proportions on either side of a central plane.*

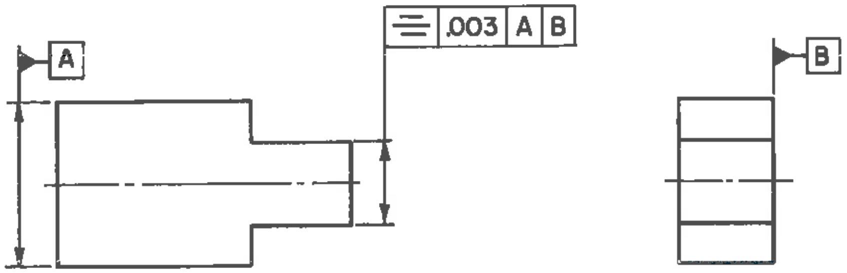


Drawing callout

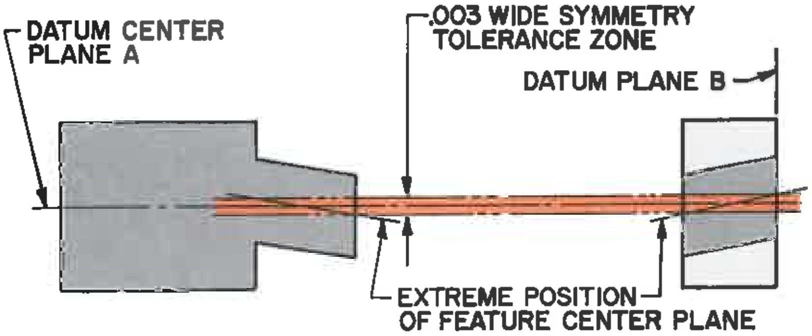


Interpretation

Figure 3-51. *A concentricity geometric tolerance is expressed as a cylindrical tolerance zone. The axis or center point of this zone coincides with a datum axis.*



Drawing callout



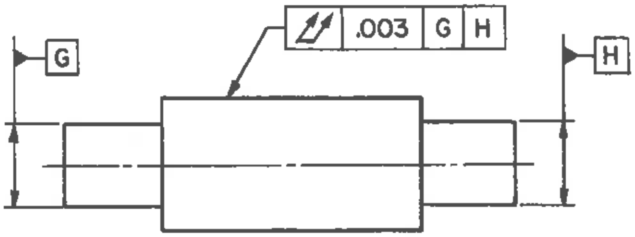
Interpretation

Figure 3-53. *A symmetry geometric tolerance is a zone within which the symmetrical surfaces align with the datum of a center plane or axis.*

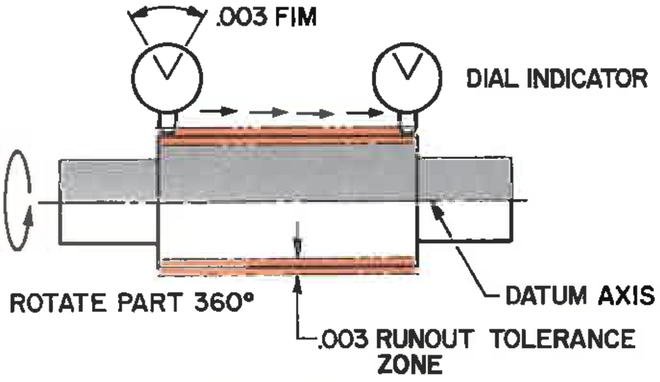
Circular runout

Total runout

Figure 3-54. *Runout geometric tolerance symbols. Arrows may be filled or unfilled.*



Drawing callout



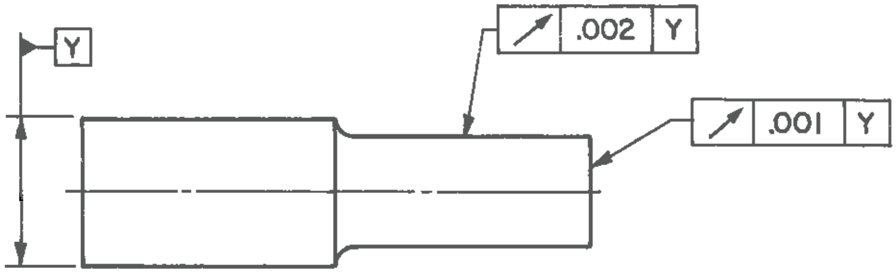
Interpretation

Figure 3-55. *Total runout controls circularity, straightness, angularity, and cylindricity of a part when applied to surfaces rotated around a datum. The entire surface must lie within the tolerance zone.*

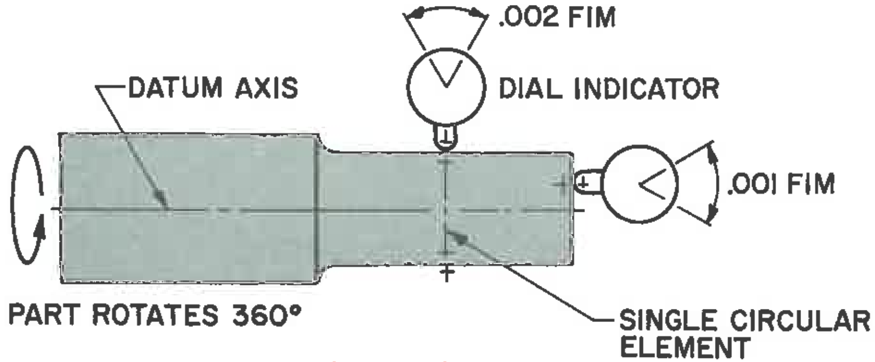
Circular runout is applied to features independently and controls circularity of a single circular cross section, **Figure 3-56.** The tolerance is measured by the *full indicator movement (FIM)* of a dial indicator when it is placed at several positions as the part is rotated.

3.7.8 Summary

Geometric dimensioning and tolerancing is far more involved than described on the preceding pages. As you progress in machining technology, you should consider purchasing a text on the subject, studying a copy of ASME Y14.5M–1994, or enrolling in a class on geometric dimensioning and tolerancing.



Drawing callout



Interpretation

Figure 3-56. *Circular runout controls circularity of a single circular cross section.*

TEST YOUR KNOWLEDGE

Please do not write in the text. Write your answers on a separate sheet of paper.

1. Drawings are used to:
    a. Show, in multiview, what an object looks like before it is made.
    b. Standardize parts.
    c. Show what to make and the sizes to make it.
    d. All of the above.
    e. None of the above.
2. The symbols, lines, and figures that make up a drawing are frequently called the _____.
3. A microinch is _____ of an inch.
4. A micrometer is _____ of a meter.
5. How can surface roughness of a machined part be checked against specifications on the drawing? How can it be measured electronically?
6. When tolerances are plus and minus, it is called a _____ tolerance.
7. When tolerances are only plus or only minus, it is called a _____ tolerance.
8. Tolerances are:
    a. The different materials that can be used.
    b. Allowances in either oversize or undersize that a part can be made and still be acceptable.
    c. Dimensions.
    d. All of the above.
    e. None of the above.

9. Drawings made other than actual size are called _____.

10. A subassembly drawing differs from an assembly drawing by:
    a. Showing only a small portion of the complete object.
    b. Making it possible to use smaller drawings.
    c. Showing the object without all needed dimensions.
    d. All of the above.
    e. None of the above.

11. Why are prints used in place of the original drawings?

12. The craft worker is given all of the information needed to make a part on a _____ drawing.

13. What does an assembly drawing show?

14. Why are standard size drawing sheets used?

15. All dimensions have a tolerance except _____ dimensions.

16. Dimensions placed between parentheses are _____ dimensions.

17. When is a feature control frame employed?

18. Sketch the form geometric tolerance symbols and indicate what they mean.

19. Define the term *maximum material condition* (*MMC*). Use a sketch if necessary.

20. Define the term *least material condition* (*LMC*). Use a sketch if necessary.

Extremely accurate measurements of small parts (up to 30″ × 30″ × 27″ or 762 mm × 762 mm × 686 mm) can be done on the shop floor by a bridge-type coordinate measuring machine like this one. The CMM is computer-controlled, and can make repeated measurements with an accuracy of 0.00012″ (0.003 mm). (Giddings & Lewis, Inc.)

Chapter 4

Measurement

LEARNING OBJECTIVES

After studying this chapter, you will be able to:

- ○ Measure to 1/64″ (0.5 mm) with a steel rule.
- ○ Measure to 0.0001″ (0.002 mm) using a Vernier micrometer caliper.
- ○ Measure to 0.001″ (0.02 mm) using Vernier measuring tools.
- ○ Measure angles to 0°5′ using a universal Vernier bevel.
- ○ Identify and use various types of gages found in a machine shop.
- ○ Use a dial indicator.
- ○ Employ the various helper measuring tools found in a machine shop.

IMPORTANT TERMS

dial indicators
gage blocks
gaging
graduations
helper measuring tools
International System of Units
metrology
micrometer caliper
steel rule
Vernier caliper

Without some form of accurate measurement, modern industry could not exist. The science that deals with systems of measurement is called ***metrology.*** Today, industry can make measurements accurate to one microinch (one-millionth of an inch).

If a microinch were as thick as a dime, one inch would be as high as *four* Empire State Buildings (about 5000′ total). An engineer once estimated, with tongue in cheek, that a steel railroad rail supported at both ends would sag one-millionth of an inch when a "fat horsefly" landed on it in the middle.

In addition to using US Conventional units of measure (inch, foot, etc.), industry is gradually converting to metric units of measure (millimeter, meter, etc.), called the ***International System of Units*** (abbreviated SI). A micrometer is one-millionth of a meter (0.000001 m).

All of the familiar measuring tools are available with scales graduated in metric units, **Figure 4-1.** An SI Metric (millimeter) rule is compared with conventional fractional and decimal rules in **Figure 4-2.** Metric-based measuring tools should offer no problems for the user. As a matter of fact, they are often *easier* to read than inch-based measuring tools.

Although you will measure in very tiny units when you go to work in industry, you must first learn to read a rule to 1/64″ and 0.5 mm. Then, you can progress through 1/1000″ (0.001″) and 1/100 mm (0.01 mm) by learning to use micrometer and Vernier-type measuring tools. Finally, you can progress to 1/10,000″ (0.0001″) and 1/500 mm (0.002 mm) by using the Vernier scale on some micrometers.

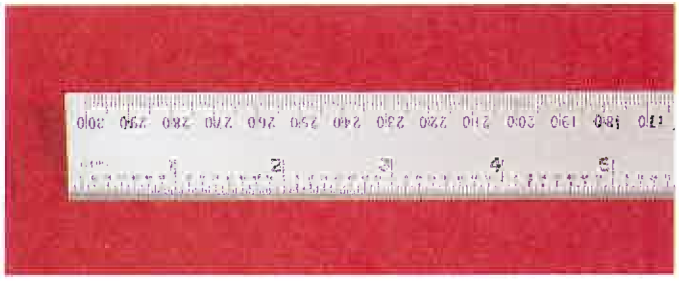


Figure 4-1. *This rule can be used to make measurements in both US Conventional and SI Metric units.*

4.1 THE RULE

The ***steel rule,*** often incorrectly called a *scale,* is the simplest of the measuring tools found in the shop. **Figure 4-2** shows the three basic types of rule graduations. A few of the many rule styles are shown in **Figure 4-3.**

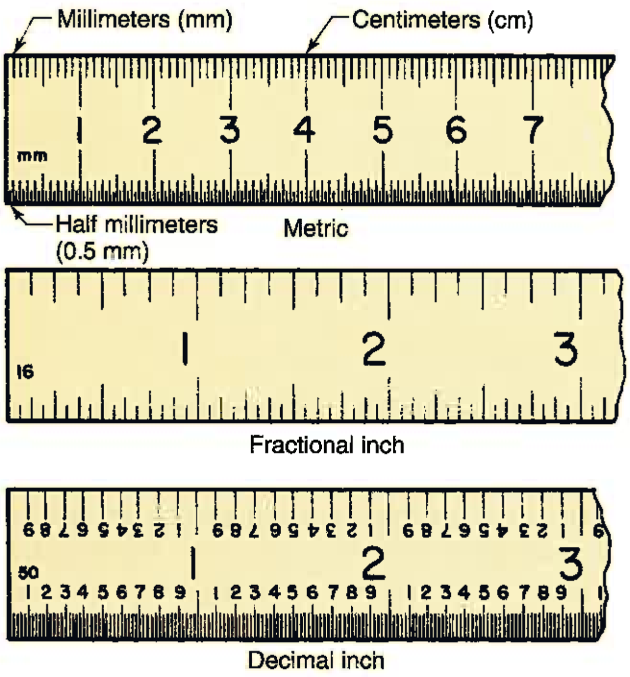


Figure 4-2. *Compare the metric (millimeter-graduated) rule with the more familiar rules graduated in fractional and decimal inch units.*

4.1.1 Reading the Rule (US Conventional)

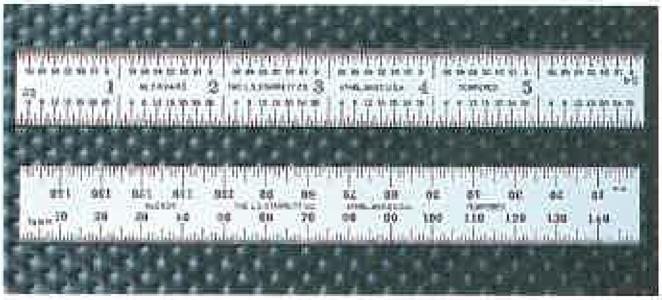
A careful study of the enlarged rule section will show the different fractional divisions of the inch from 1/8 to 1/64, **Figure 4-4.** The lines representing the divisions are called ***graduations.*** On many rules, every fourth graduation is numbered on the 1/32 edge, and every eighth graduation on the 1/64 edge.

To become familiar with the rule, begin by measuring objects on the 1/8 and 1/16 scales. Once you become comfortable with these scales, begin using the 1/32 and 1/64 scales. Practice until you can quickly and accurately read measurements. Some rules are graduated in 10ths, 20ths, 50ths, and 100ths. Additional practice will be necessary to read these rules.

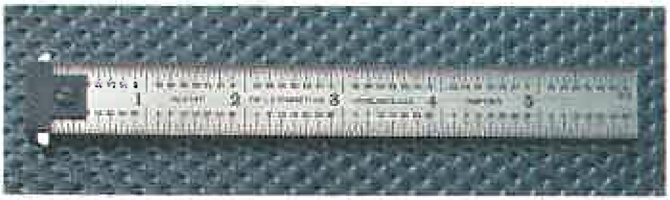
Fractional measurements are *always* reduced to the lowest terms. A measurement of 14/16″ is reduced to 7/8″, 2/8″ becomes 1/4″, and so on.

4.1.2 Reading the Rule (Metric)

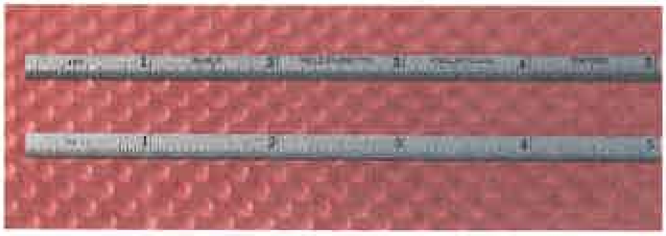
Most metric rules are divided into millimeter or one-half millimeter graduations. They are



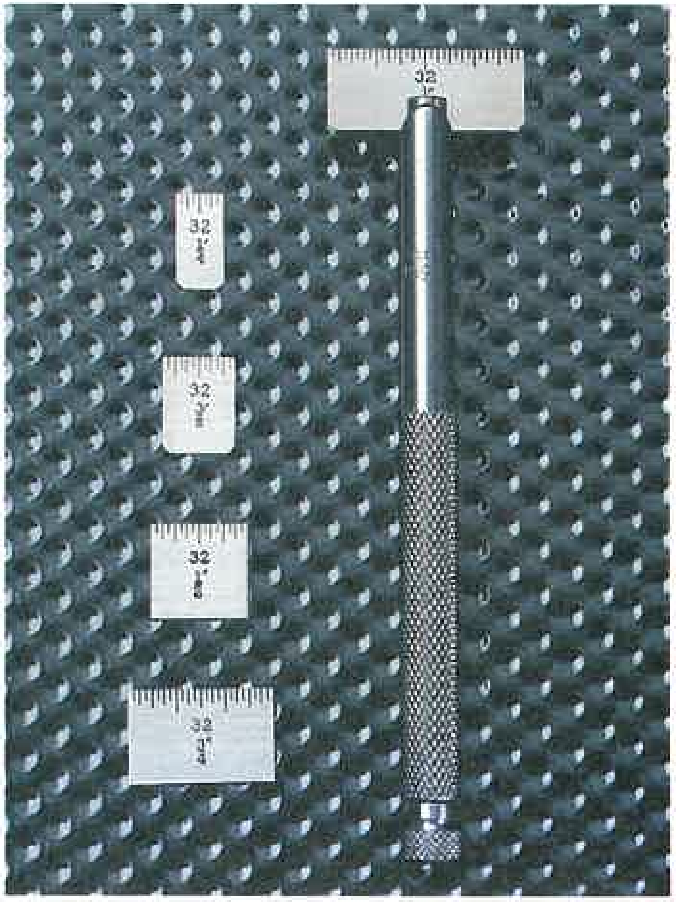
6″ steel rule



Rule with adjustable hook



Narrow rule



Small rules with holder

Figure 4-3. *Many different types of rules are used to make measuring quicker and more accurate. (L. S. Starrett Co.)*

numbered every 10 mm. See **Figure 4-5.** The measurement is determined by counting the number of millimeters.

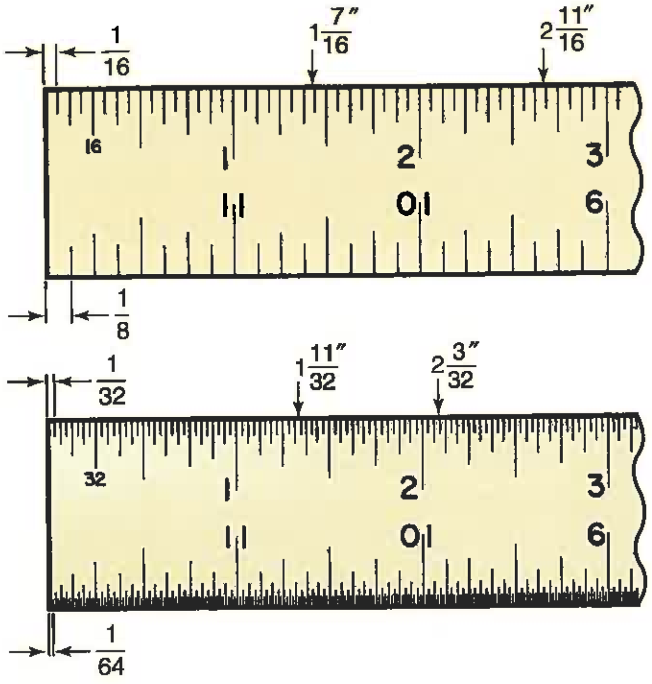


Figure 4-4. *These are the fractional graduations found on a rule. Measurements are taken by counting the number of graduations.*

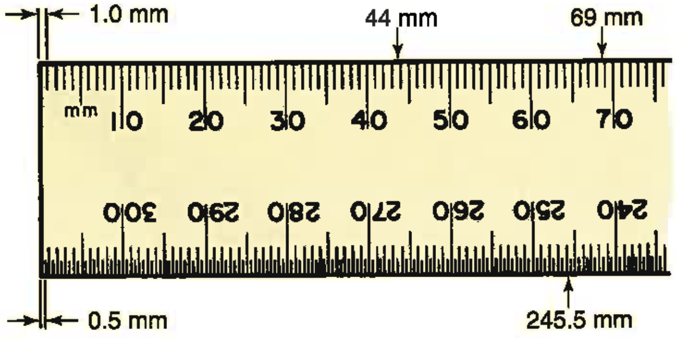


Figure 4-5. *Most metric rules are graduated in millimeters and half-millimeters. They are available in a variety of sizes.*

4.1.3 Care of the Rule

The steel rule is precision-made and, like all tools, its accuracy depends upon the care it receives. Here are a few suggestions:

- Use the rule for measurements only. Do not adjust screws or open paint cans with it. Be careful not to bend your rule.
- Keep the rule clear of moving machinery. Never use it to clean metal chips as they form on the cutting tool. This is extremely dangerous and will ruin the rule.
- Avoid laying other tools on the rule.
- Wipe steel rules with an oily cloth before storing. This will prevent rust. If the rule is to be stored for a prolonged period, coat it with wax or rust preventative.
- Clean the rule with steel wool to keep the graduations legible.
- Make measurements and tool settings from the 1" line (10 mm line on a metric rule) or other major graduations, rather than from the end of the rule.
- Store rules separately. Do not throw them in a drawer with other tools.
- Use the rule with care to protect the ends from nicks and wear.
- Use the correct rule for the job being done.

4.2 THE MICROMETER CALIPER

A Frenchman, Jean Palmer, devised and patented a measuring tool that made use of a screw thread, making it possible to read measurements quickly and accurately without calculations. It incorporated a series of engraved lines on the sleeve and around the thimble. The device, called *Systeme Palmer,* is the basis for the modern micrometer caliper, **Figure 4-6.**

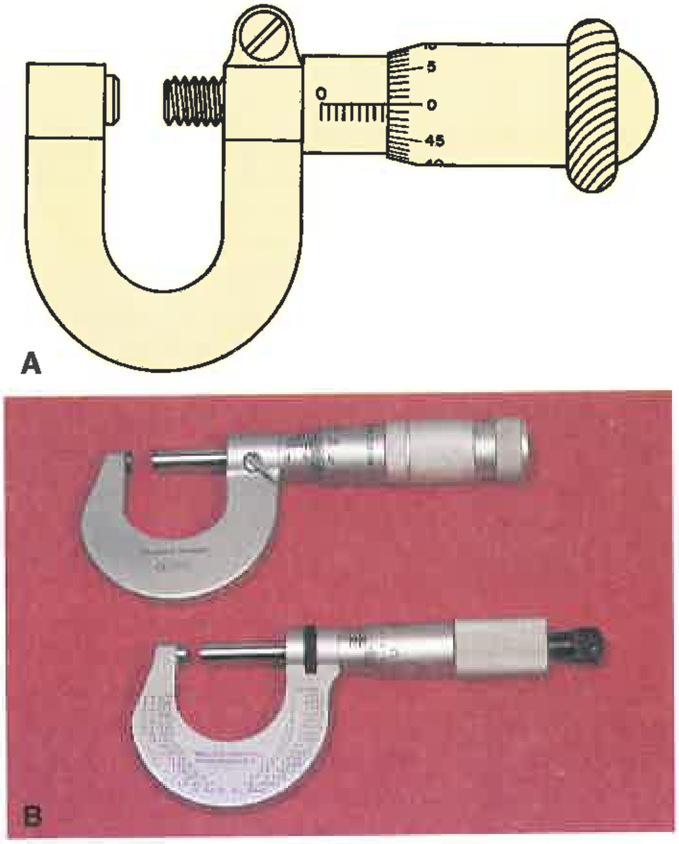


Figure 4-6. *The micrometer caliper, past and present. A—A drawing of the* Systeme Palmer *measuring device. B—These modern micrometer calipers operate on the same principle as the original 1848 invention.*

The *micrometer caliper,* also known as a "mike," is a precision tool capable of measuring to 0.001″ or 0.01 mm. When fitted with a Vernier scale, it will read to 0.0001″ or 0.002 mm.

4.2.1 Types of Micrometers

Micrometers are produced in a wide variety of models. Digital display is included in many micrometers, making measuring easier. Some of the most popular models are the following:

- An ***outside micrometer*** measures external diameters and thickness, **Figure 4-7.**

Figure 4-7. *This digital outside micrometer can be used to measure in both US Conventional and SI Metric units. (Mitutoyo/MTI Corp.)*

- An ***inside micrometer*** has many uses, including measuring internal diameters of cylinders, rings, and slots. The range of a ***conventional inside micrometer*** can be extended by fitting longer rods to the micrometer head. The range of a ***jaw-type inside micrometer*** is limited to 1″ or 25 mm. The jaw-type inside micrometer has a scale graduated from right to left. See **Figure 4-8.**
- A ***micrometer depth gage*** measures the depths of holes, slots, and projections. See **Figure 4-9.** The measuring range can be increased by changing to longer spindles. Measurements are read from right to left.
- A ***screw-thread micrometer*** has a pointed spindle and a double-V anvil shaped to contact the screw thread, **Figure 4-10.** It measures the pitch diameter of the thread, which equals the outside (major) diameter of the thread minus the depth of one thread. Since each thread micrometer is designed to measure only a limited number of threads per inch, a set of thread micrometers is necessary to measure a full range of thread pitches.

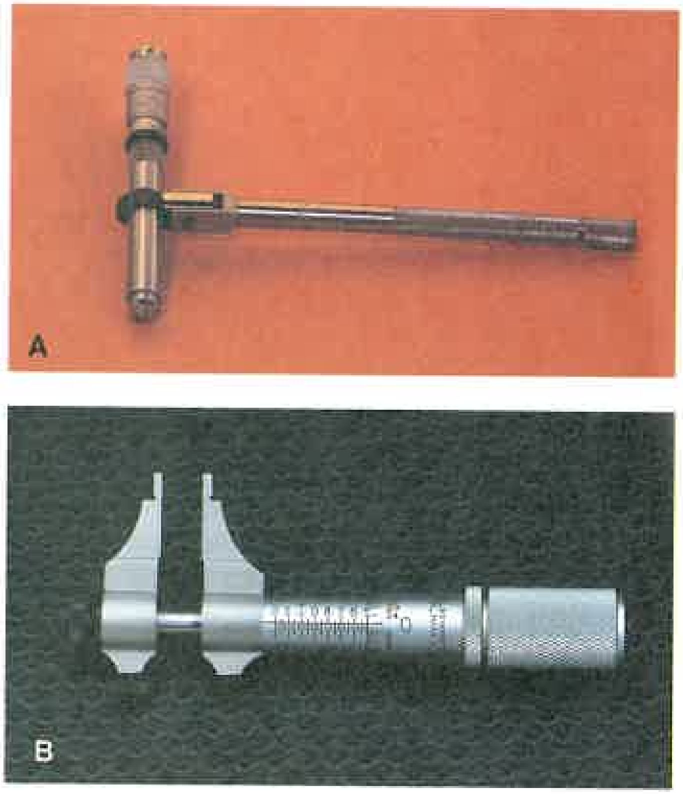


Figure 4-8. *Inside micrometers. A—A conventional inside micrometer. B—The caliper jaws on this inside micrometer allow quick and accurate measurements. The divisions on the sleeve are numbered in the reverse order of a conventional outside micrometer. (L. S. Starrett Co.)*

Figure 4-9. *A standard micrometer depth gage.*

- A ***chamfer micrometer*** will accurately measure countersunk holes and other chamfer-type measurements. With fastener

Figure 4-10. *This screw thread micrometer can measure threads as wide as 7/8″. (L. S. Starrett Co.)*

tolerances so critical on some aerospace and other applications, it is important that countersunk holes and tapers on fasteners meet specifications. A chamfer micrometer makes it possible to check these critical areas.

Special micrometers are available for other applications. These micrometers are devised to handle nonstandard measurement tasks.

4.2.2 Reading an Inch-Based Micrometer

A micrometer uses a very precisely made screw thread that rotates in a fixed nut. The screw thread is ground on the ***spindle*** and is attached to the ***thimble.*** The spindle advances or recedes from the ***anvil*** as the thimble is rotated. See **Figure 4-11.** The threaded section has 40 threads per inch; therefore, each revolution of the thimble moves the spindle 1/40″ (0.025″).

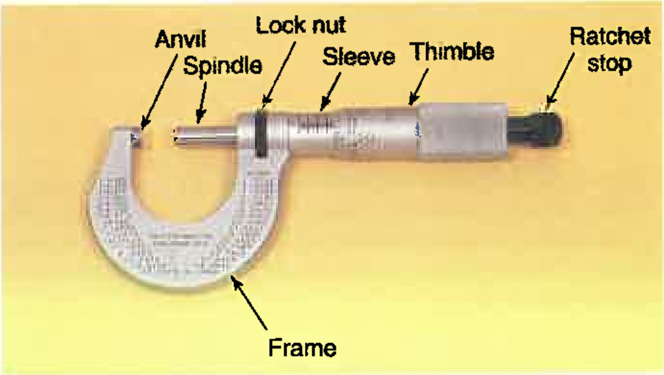
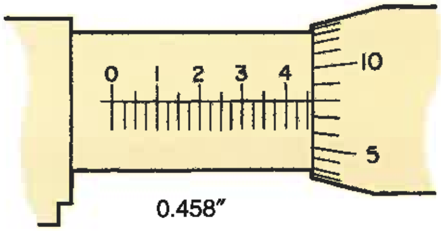


Figure 4-11. *Basic parts of a micrometer caliper.*

The line engraved lengthwise on the sleeve is divided into 40 equal parts per inch (corresponding to the number of threads per inch on the spindle). Each vertical line equals 1/40″, or 0.025″. Every fourth division is numbered, representing 0.100″, 0.200″, etc.

The beveled edge of the thimble is divided into 25 equal parts around its circumference. Each division equals 1/1000″ (0.001″). On some micrometers, every division is numbered, while every fifth division is numbered on others.

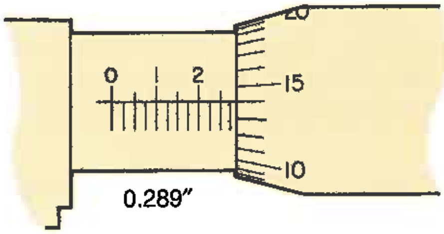
The micrometer is read by recording the highest number on the sleeve (1 = 0.100, 2 = 0.200, etc.). To this number, add the number of vertical lines visible between the number and thimble edge (1 = 0.025, 2 = 0.050, etc.). To this total, add the number of thousandths indicated by the line that coincides with the horizontal sleeve line.



Example 1

Add the readings from the sleeve and the thimble:

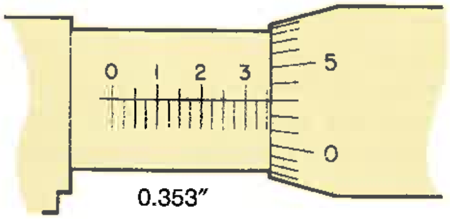
4 large graduations: 4 × 0.100 = 0.400
2 small graduations: 2 × 0.025 = 0.050
8 thimble graduations: 8 × 0.001 = 0.008
Total mike reading = 0.458″



Example 2

Add the readings from the sleeve and the thimble:

2 large graduations: 2 × 0.100 = 0.200
3 small graduations: 3 × 0.025 = 0.075
14 thimble graduations: 14 × 0.001 = 0.014
Total mike reading = 0.289″



Example 3

Add the readings from the sleeve and the thimble:

3 large graduations: 3 × 0.100 = 0.300
2 small graduations: 2 × 0.025 = 0.050
3 thimble graduations: 3 × 0.001 = <u>0.003</u>
Total mike reading = 0.353″

4.2.3 Reading a Vernier Micrometer

On occasion, it is necessary to measure more precisely than 0.001″. A Vernier micrometer caliper is used in these situations. This micrometer has a third scale around the sleeve that will furnish the 1/10,000″ (0.0001″) reading. See **Figure 4-12.**

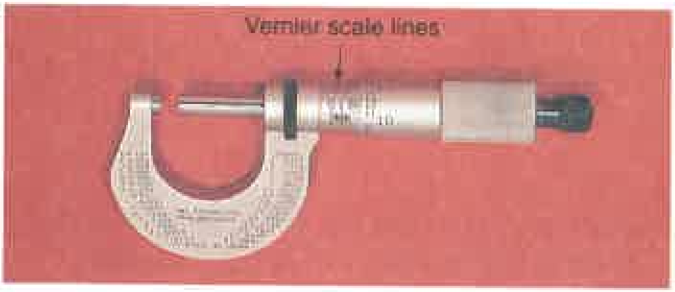


Figure 4-12. A Vernier micrometer caliper includes a Vernier scale on the sleeve.

The Vernier scale has 11 parallel lines that occupy the same space as 10 lines on the thimble. The lines around the sleeve are numbered 1 to 10. The difference between the spaces on the sleeve and those on the thimble is one-tenth of a thousandth of an inch.

To read the Vernier scale, first obtain the thousandths reading, then observe which of the lines on the Vernier scale coincides (lines up) with a line on the thimble. Only one of them can line up. If the line is 1, add 0.0001 to the reading; if line 2, add 0.0002 to the reading, etc. See **Figure 4-13.**

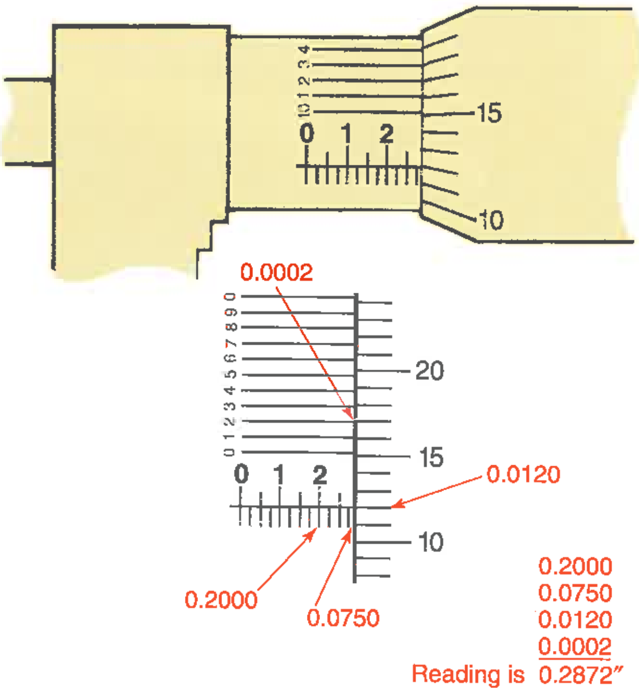


Figure 4-13. How to read a Vernier micrometer caliper. Add the total reading in thousandths, then observe which of the lines on the Vernier scale coincides with a line on the thimble. In this case, it is the second line, so 0.0002 is added to the reading.

4.2.4 Reading a Metric-Based Micrometer

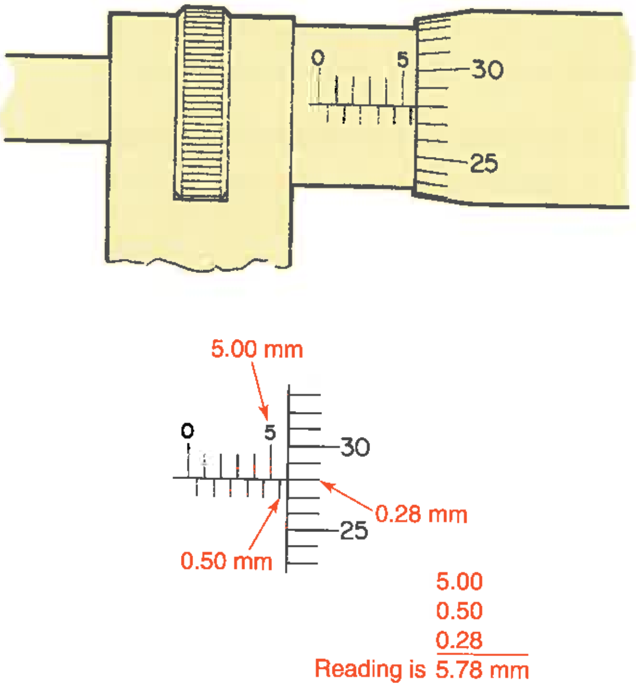


Figure 4-14. To read a metric micrometer, add the total reading in millimeters visible on the sleeve to the reading of hundredths of a millimeter, indicated by the graduation on the thimble. Note that the thimble reading coincides with the longitudinal line on the micrometer sleeve.

The metric-based micrometer is read as shown in **Figure 4-14.** If you are able to read the conventional inch-based micrometer, reading the metric-based tool will offer no difficulties.

4.2.5 Reading a Metric Vernier Micrometer

Metric Vernier micrometers are read in the same way as standard metric micrometers. However, using the Vernier scale on the sleeve, an additional reading of two-thousandths of a millimeter can be obtained, **Figure 4-15.**

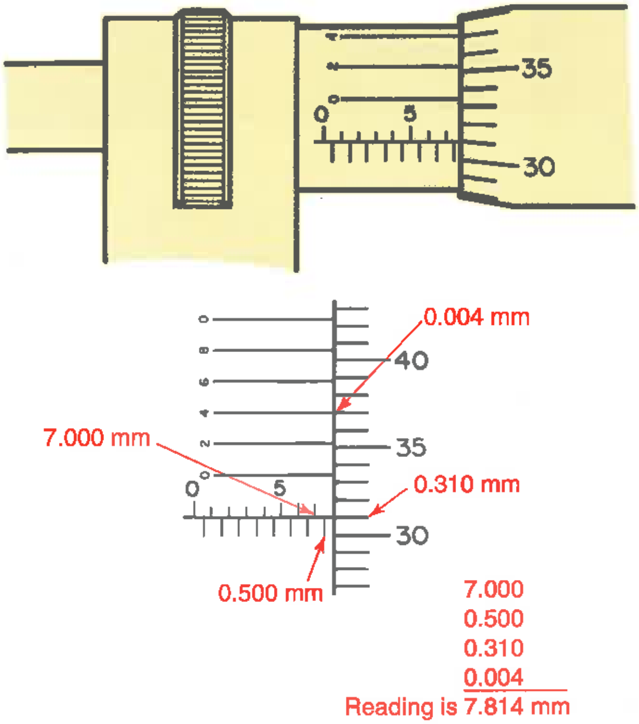


Figure 4-15. *Reading a metric-based Vernier micrometer caliper. To the regular reading in hundredths of a millimeter (0.01), add the reading from the Vernier scale that coincides with a line on the thimble. Each line on the Vernier scale is equal to two thousandths of a millimeter (0.002 mm).*

4.2.6 Using the Micrometer

The proper way to hold a micrometer when making a measurement is shown in **Figure 4-16.** The work is placed into position, and the thimble rotated until the part is clamped *lightly* between the anvil and spindle. Guard against excessive pressure, which will cause an erroneous reading. Some micrometers have features to help regulate pressure:

- A ***ratchet stop*** is used to rotate the spindle. When the pressure reaches a predetermined amount, the ratchet stop slips and prevents further spindle turning. Uniform contact pressure with the work is ensured, even if different people use the same micrometer. Refer again to **Figure 4-11.**
- A ***friction thimble*** may be built into the upper section of the thimble. This produces the same results as the ratchet stop but permits one-handed use of the micrometer.

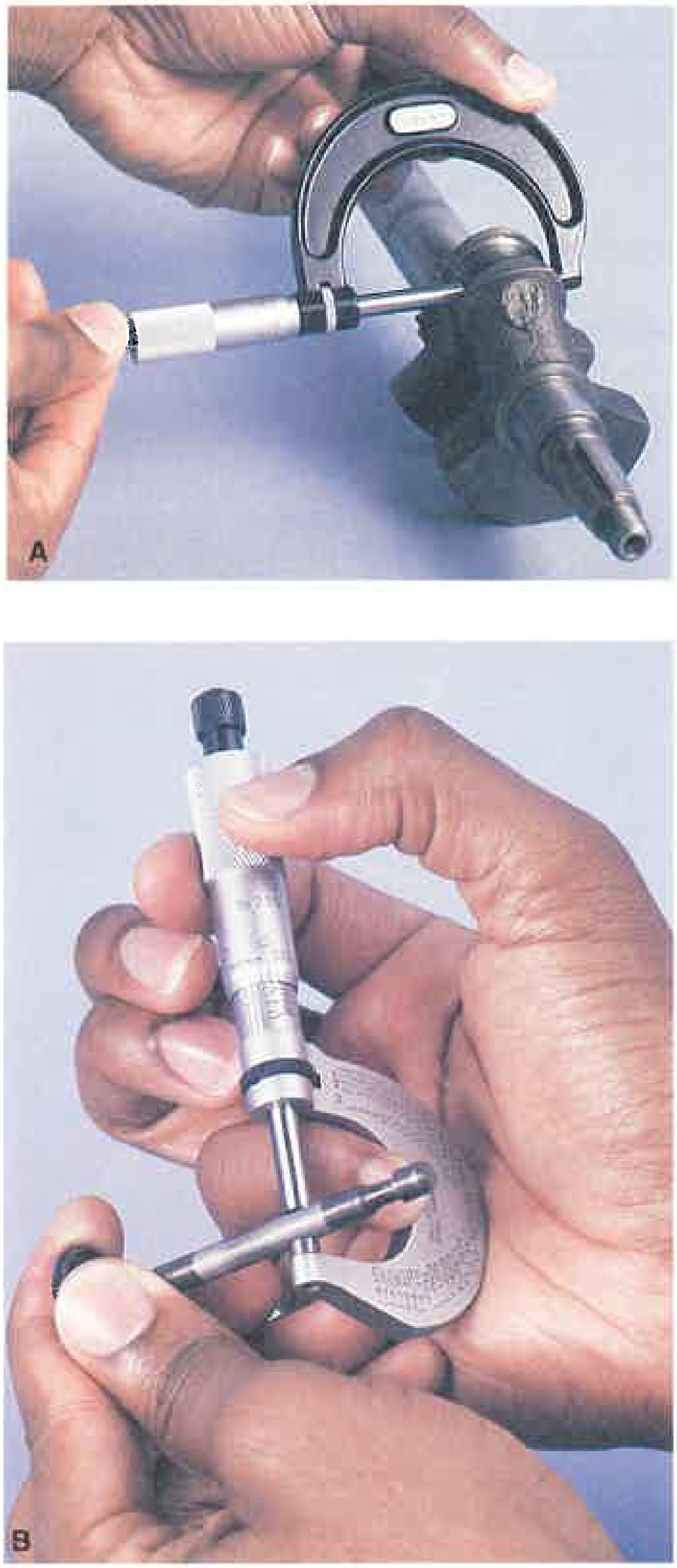


Figure 4-16. *Proper technique of handling a micrometer. A—Use very light pressure when turning the thimble. B—When the piece being measured must also be held, position the micrometer as shown, with a finger in the micrometer frame.*

- A ***lock nut*** is used when several identical parts are to be gaged. Refer again to **Figure 4-11.** The nut locks the spindle into place. Gaging parts with a micrometer locked at the proper setting is an easy way to determine whether the pieces are sized correctly.

4.2.7 Reading an Inside Micrometer

To get a correct reading with an inside micrometer, it is important that the tool be held square across the diameter of the work. It must be positioned so that it will measure across the diameter on the exact center, **Figure 4-17.**

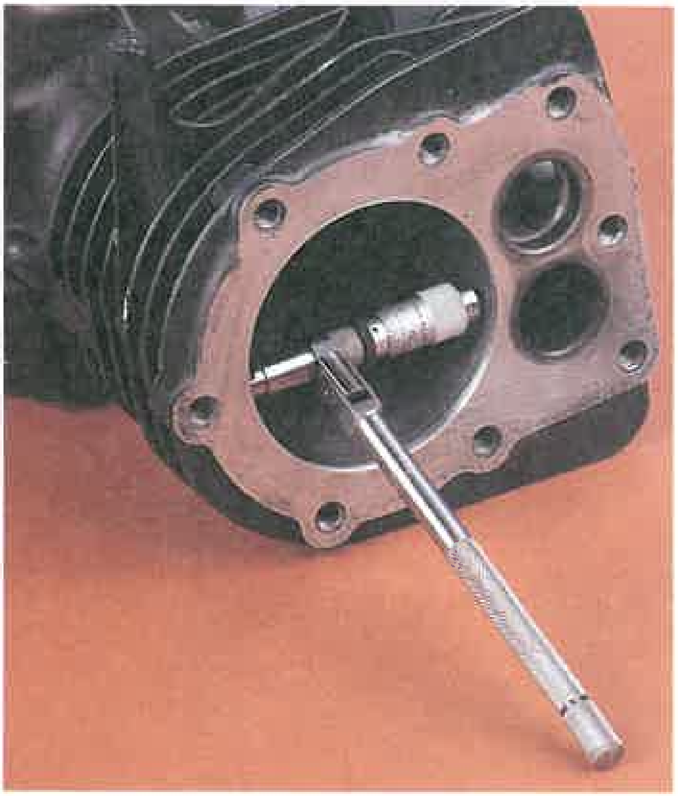


Figure 4-17. *Using an inside micrometer. Extension rods can be added to increase the tool's measuring range.*

Measurement is made by holding one end of the tool in place and then "feeling" for the maximum possible setting by moving the other end from left to right, and then in and out of the opening. The measurement is made when no left or right movement is felt, and a slight drag is noticeable on the in-and-out swing. It may be necessary to take several readings and average them.

4.2.8 Reading a Micrometer Depth Gage

Be sure to read a micrometer depth gage correctly. The graduations on this measuring tool are in *reverse order* of the graduations on an outside micrometer. See **Figure 4-18.** The graduations under the thimble must be read, rather than those that are exposed.

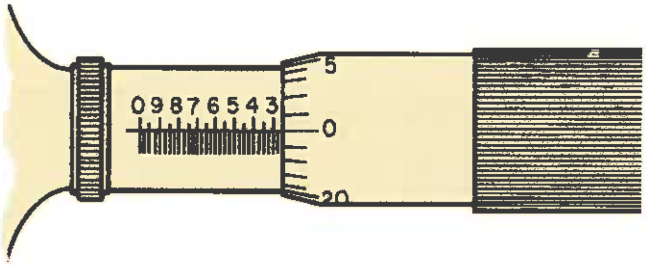


Figure 4-18. *A micrometer depth gage. When making measurements with a depth gage, remember that the graduations are in reverse order. This gage indicates a depth of 0.250.*

4.2.9 Care of a Micrometer

Micrometers are precision instruments and must be handled with care. The following techniques are recommended:

- Place the micrometer on the work carefully so the faces of the anvil and spindle will not be damaged. The same applies when removing the tool after a measurement has been made.
- Keep the micrometer clean. Wipe it with a slightly oiled cloth to prevent rust and tarnish. A drop of light oil on the screw thread will keep the tool operating smoothly.
- Avoid "springing" a micrometer by applying too much pressure when you are making a measurement.
- Clean the anvil and spindle faces before use. This can be done with a soft cloth or by *lightly* closing the jaws on a clean piece of paper and drawing the paper out.
- Check for accuracy by closing the spindle gently on the anvil and note whether the zero line on the thimble coincides with the zero on the sleeve. If they are not aligned, follow the manufacturer's recommended adjustments.
- Avoid placing a micrometer where it may fall on the floor or have other tools placed on it.
- If the micrometer must be opened or closed a considerable distance, do not "twirl" the frame; gently roll the thimble with your palm. See **Figure 4-19.**
- Never attempt to make a micrometer reading until a machine has come to a complete stop.
- Clean and oil the tool if it is to be stored for some time. If possible, place the micrometer in a small box for protection.

Figure 4-19. Micrometers must be treated carefully. Roll the micrometer thimble on the palm of your hand if the instrument must be opened or closed a considerable distance.

4.3 VERNIER MEASURING TOOLS

The Vernier principle of measuring was named for its inventor, Pierre Vernier, a French mathematician. The ***Vernier caliper*** can make accurate measurements to 1/1000″ (0.001″) and 1/50 mm (0.02 mm). See **Figure 4-20.**

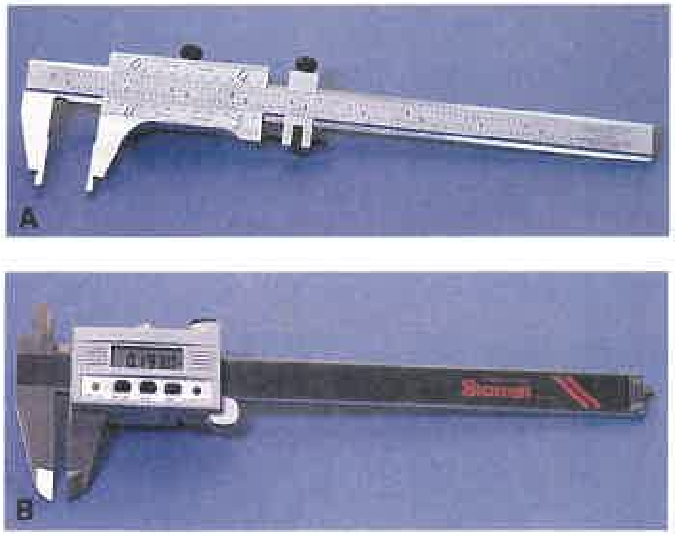


Figure 4-20. Vernier calipers can be used to make very accurate measurements. A—Standard Vernier caliper. B—Modern digital calipers are easier to read than mechanical instruments.

The design of the tool permits measurements to be made over a large range of sizes. It is manufactured as a standard item in 6″, 12″, 24″, 36″, and 48″ lengths. SI Metric Vernier calipers are available in 150 mm, 300 mm, and 600 mm lengths. The 6″, 12″, 150 mm, and 300 mm sizes are most commonly used. Unlike the micrometer caliper, the Vernier caliper can be used for both inside and outside measurements, **Figure 4-21.**

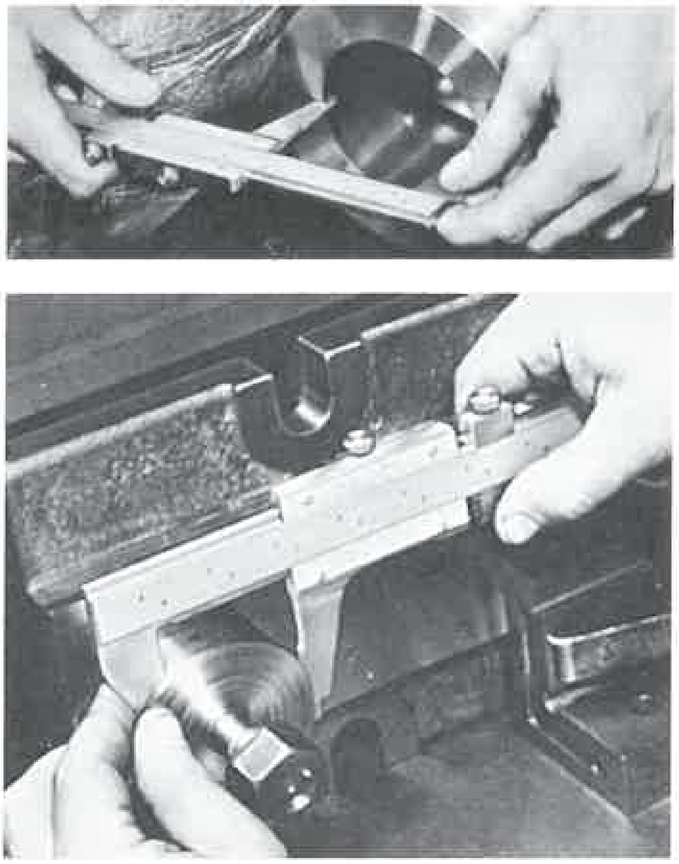


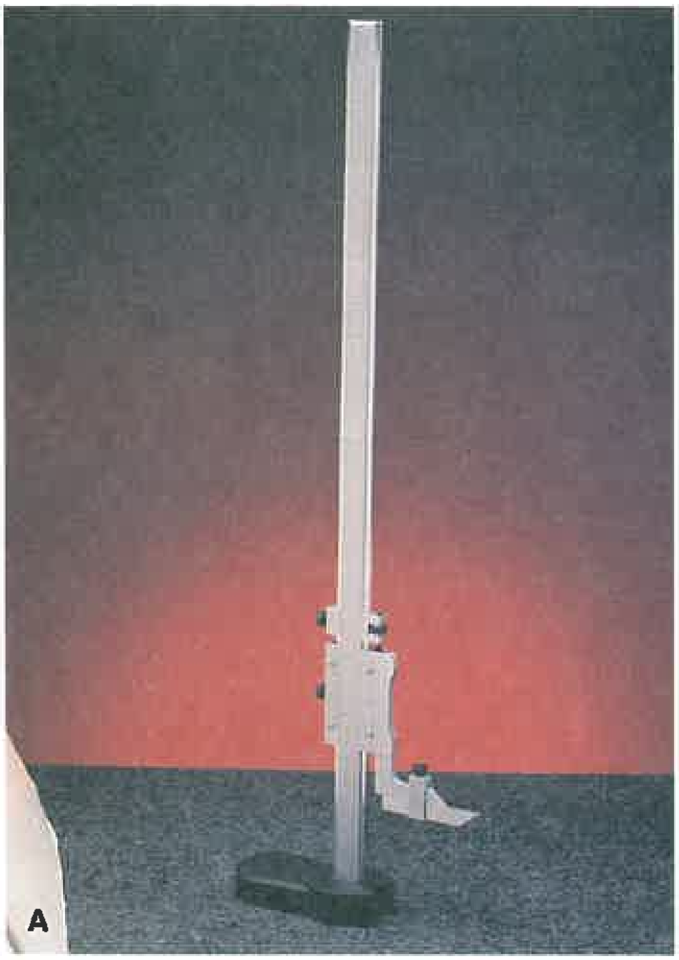
Figure 4-21. Vernier calipers can be used to make both internal and external measurements. (L. S. Starrett Co.)

The following measuring instruments may include a Vernier scale:

- Height and depth gages are used for layout work and to inspect the locations of features. See **Figure 4-22.**
- Gear tooth calipers are used to measure gear teeth and threading tools, **Figure 4-23.**
- Universal Vernier bevel protractors are used for the layout and inspection of angles, **Figure 4-24.**

Vernier measuring tools, with the exception of the Vernier bevel protractor, consist of a graduated beam with fixed jaw or base and a Vernier slide assembly. The Vernier slide assembly is composed of a movable jaw or scribe, Vernier plate, and clamping screws. The slide moves as a unit along the beam.

Unlike other Vernier measuring tools, the beam of the Vernier caliper is graduated on both sides. One side is for making outside measurements, the other for inside measurements. Many of the newer Vernier measuring tools are graduated to make both inch and millimeter measurements.



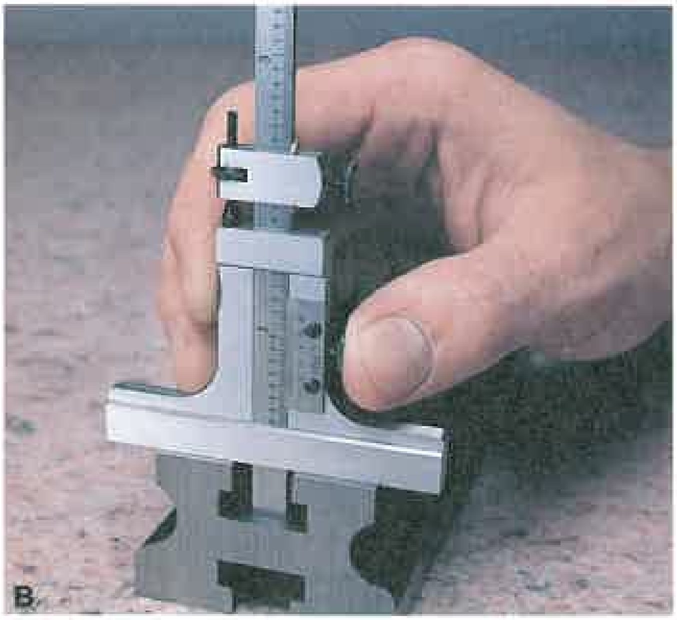


Figure 4-22. *Many instruments are equipped with a Vernier scale. A—Height gage. B—Depth gage. C—The digital readout on this type of height gage serves the same function as a standard Vernier scale. (L. S. Starrett Co.)*

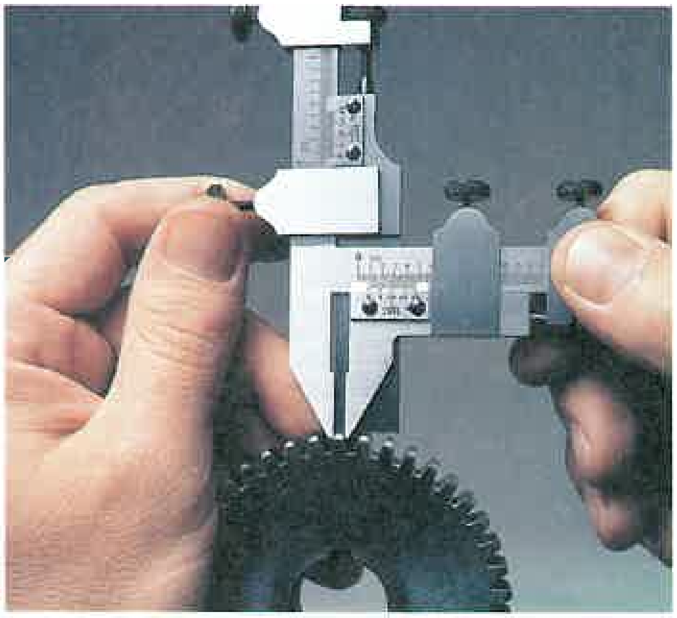


Figure 4-23. *Gear tooth Vernier calipers are used to measure gear teeth, form tools, and threaded tools. (L. S. Starrett Co.)*

4.3.1 Reading an Inch-Based Vernier Scale

These measuring tools are available with either 25-division or 50-division Vernier plates. Both plates can be read to 0.001″.

On measuring tools using the *25-division Vernier plate,* every inch section on the beam is graduated

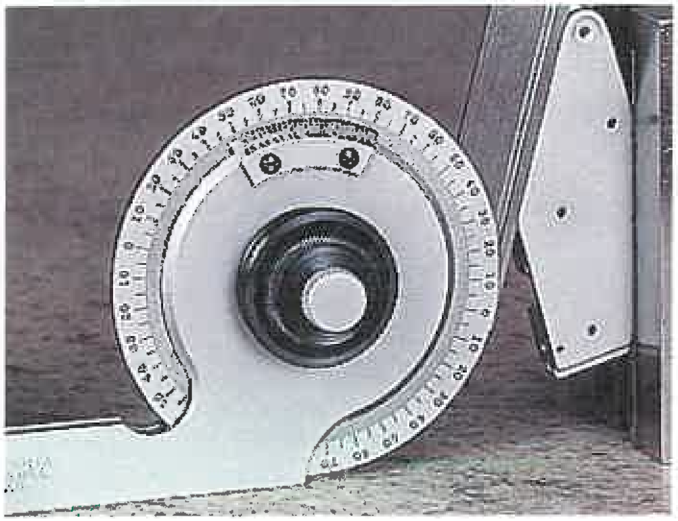
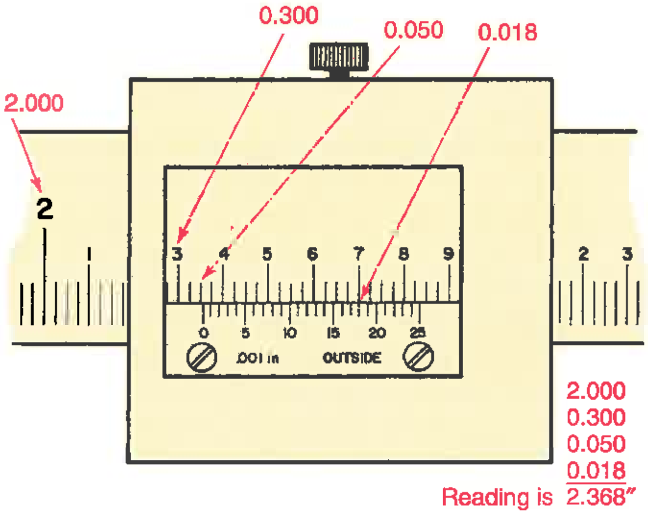


Figure 4-24. *A universal Vernier bevel protractor is used to accurately measure angles. (L. S. Starrett Co.)*

into 40 equal parts. Each graduation is 1/40″ (0.025″). Every fourth division, representing 0.100″, is numbered.

There are 25 divisions on the Vernier plate. Every fifth line is numbered: 5, 10, 15, 20, and 25. The 25 divisions occupy the same space as 24 divisions on the beam. This slight difference, equal to 0.001 (1/1000″) per division, is the basis of the Vernier principle of measuring.

To read a 25-division Vernier plate measuring tool, note how many inches (1, 2, 3, etc.), tenths (0.100, 0.200, etc.), and fortieths (0.025, 0.050, or 0.075) there are between the "0" on the Vernier scale and the "0" line on the beam, then add them. Then count the number of graduations (each graduation equals 0.001″) that lie between the "0" line on the Vernier plate and the line that coincides (corresponds exactly) with a line on the beam. Only one line will coincide. Add this to the above total for the reading.



The "0" line on the Vernier plate is:

Past the 2: 2 × 1 = 2.000

Past the 3: 3 × 0.100 = 0.300

Plus 2 graduations: 2 × 0.025 = 0.050

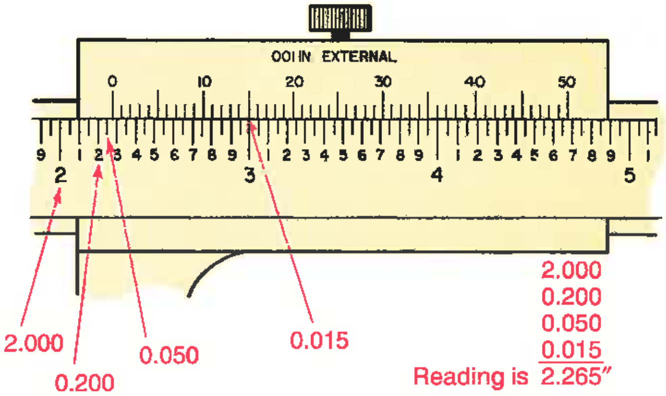
Plus 18 Vernier scale graduations: 18 × 0.001 = 0.018

Total reading = 2.368″

On the *50-division Vernier plate*, every second graduation between the inch lines is numbered, and equals 0.100″. The unnumbered graduations equal 0.050″.

The Vernier plate is graduated into 50 parts, each representing 0.001″. Every fifth line is numbered: 5, 10, 15 . . . 40, 45, and 50.

To read a 50-division Vernier measuring tool, first count how many inches, tenths (0.100), and twentieths (0.050) there are between the "0" line on the beam, and the "0" line on the Vernier plate. Then add them. Then count the number of 0.001 graduations on the Vernier plate from its "0" line to the line that coincides with a line on the beam. Add this to the above total.



The "0" line on the Vernier plate is:

Past the 2: 2 × 1.000 = 2.000

Past the 2: 2 × 0.100 = 0.200

Plus one graduation: 1 × 0.050 = 0.050

Plus 15 Vernier scale graduations: 15 × 0.001 = 0.015

Total reading = 2.265″

4.3.2 Reading a Metric-Based Vernier Scale

The principles used in reading metric Vernier measuring tools are the same as those used for US Conventional measure. However, the readings on the Vernier scale are obtained in 0.02 mm precision. A 25-division Vernier scale is illustrated in **Figure 4-25,** while a 50-division scale is described in **Figure 4-26.**

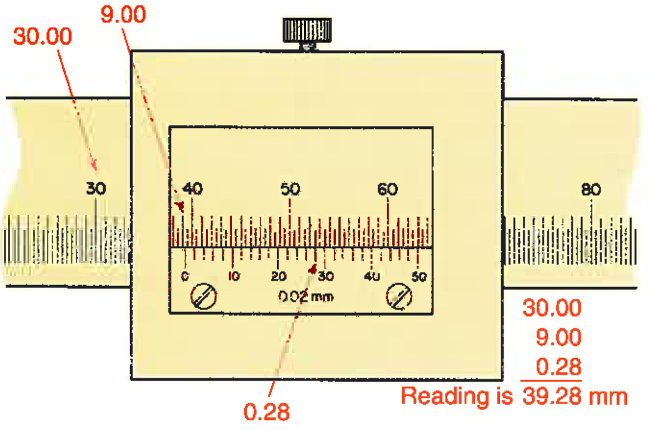


Figure 4-25. *How to read a 25-division metric-based Vernier scale. Readings on the scale are obtained in units of two hundredths of a millimeter (0.02 mm).*

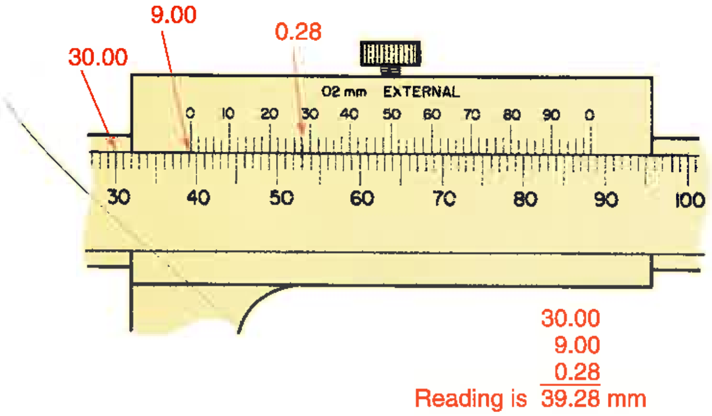


Figure 4-26. *How to read a 50-division metric-based Vernier scale. Each division equals two hundredths of a millimeter (0.02 mm).*

4.3.3 Using the Vernier Caliper

As with any precision tool, a Vernier caliper must not be forced on the work. Slide the Vernier assembly until the jaws nearly contact the section being measured. Lock the clamping screw. Make the tool adjustment with the fine adjusting nut. The jaws must contact the work firmly, but not tightly.

Lock the slide on the beam. Carefully remove the tool from the work and make your reading. For precise layout work, divider and trammel point settings are located on the outside measuring scale and on the slide assembly.

Dial calipers. These direct-reading instruments resemble Vernier calipers. They can be used to make outside, inside, and depth measurements (with the addition of a depth attachment). A lock permits the tool to be employed for repetitive measurements. See **Figure 4-27.**

Figure 4-27. *Dial calipers provide direct readings of measurements. (L.S. Starrett Co.)*

The beam is graduated into 0.10″ increments. The caliper dial is graduated into 100 divisions. The reading is made by combining the division on the beam and the dial reading.

The dial hand makes one full revolution for each 0.10″ movement. Each dial graduation, therefore, represents 1/100 of 0.10″, or 0.001″. On the metric version, each dial graduation represents 0.02 mm.

4.3.4 Universal Vernier Bevel Protractor

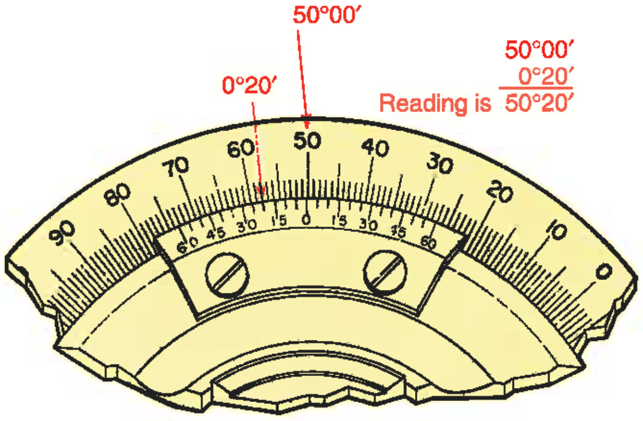
A quick review of the circles, angles, and units of measurement associated with them will help in understanding how to read a universal Vernier bevel protractor.

- **Degree (°)**—Regardless of its size, a circle contains 360°. Angles are also measured by degrees.
- **Minute (′)**—A minute represents a fractional part of a degree. If a degree is divided into 60 equal parts, each part is one minute. A foot mark (′) is used to signify minutes (e.g. 30°15′).
- **Second (″)**—Minutes are divided into smaller units known as seconds. There are 60 seconds in one minute. An angular measurement written in degrees, minutes, and seconds appears as 36°18′22″. This would read "36 degrees, 18 minutes, and 22 seconds."

A ***universal bevel protractor*** has several parts: a dial, a base or stock, and a sliding blade. The dial is graduated into degrees, and the blade can be extended in either direction and set at any angle to the stock. The blade can be locked against the dial by tightening the blade clamp nut. The blade and dial can be rotated as a unit to any desired position, and locked by tightening the dial clamp nut.

The protractor dial is graduated into 360° and reads from 0° to 90° and then back down to 0°. Every ten degree division is numbered, and every five degrees is indicated by a fine line longer than those on either side. The Vernier scale is divided into twelve equal parts on each side of the "0." Every third graduation is numbered (0, 15, 30, 45, 60), representing minutes. Each division equals five minutes. Since each degree is divided into 60 minutes, one division is equal to 5/60 of a degree.

To read the protractor, note the number of degrees that can be read up to the "0" on the Vernier plate. To this, add the number of minutes indicated by the line beyond the "0" on the Vernier plate that aligns exactly with a line on the dial.



In this example the "0" is past the 50° mark, and the Vernier scale aligns at the 20′ mark. Therefore, the measurement is 50°20′.

4.3.5 Care of Vernier Tools

Reasonable care in handling these expensive tools will maintain their accuracy.

- Wipe the instrument with a soft, lint-free cloth before using. This will prevent dirt and grit from being ground in, which could eventually affect the accuracy of the tool.
- Wipe the tool with a lightly oiled, soft cloth after use and before storage.
- Store the tool in its case.
- Never force the tool when you are making measurements.
- Use a magnifying glass or a jeweler's loupe to make Vernier readings. Hold the tool so the light is reflected on the scale.
- Handle the tool as little as possible. Sweat and body acids cause rusting and staining.
- Periodically check for accuracy. Use a measuring standard, Jo-block, or ground parallel. Return the tool to the manufacturer for adjustments and repairs.
- Lay Vernier height gages on their side when not in use. Then there will be no danger that they will be knocked over and damaged.

4.4 GAGES

It is impractical to check every dimension on every manufactured part with conventional measuring tools. Specialized tools, such as plug gages, ring gages, and optical gages are used instead. These gaging devices can quickly determine whether the dimensions of a manufactured part are within specified limits or tolerances.

Measuring requires the skillful use of precision measuring tools to determine the exact geometric size of the piece. ***Gaging*** involves checking parts with various gages. Gaging simply shows whether the piece is made within the specified tolerances.

When great numbers of an item with several critical dimensions are manufactured, it might not be possible to check each piece. It then becomes necessary to decide how many randomly selected pieces must be checked to ensure satisfactory quality and adherence to specifications. This technique is called ***statistical quality control.***

Always handle gages carefully. If dropped or mishandled, the accuracy of the device could be affected. Gages provide a method of checking your work and are very important tools.

4.4.1 Plug Gage

Plug gages are used to check whether hole diameters are within specified tolerances. The ***double-end cylindrical plug gage*** has two gaging members known as *go* and *no-go* plugs, **Figure 4-28.** The ***go plug*** should enter the hole with little or no interference. The ***no-go plug*** should not fit.

The go plug is longer than the no-go plug. A ***progressive plug gage,*** or ***step plug gage,*** has the go and no-go plugs on the same end. This gage is able to check the dimensions in one motion. See **Figure 4-29.**

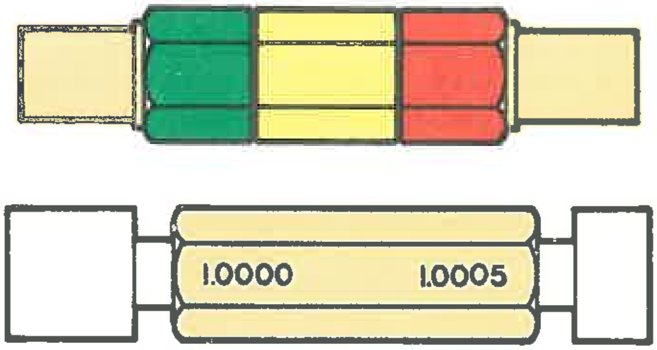


Figure 4-28. *A double end cylindrical plug gage.*

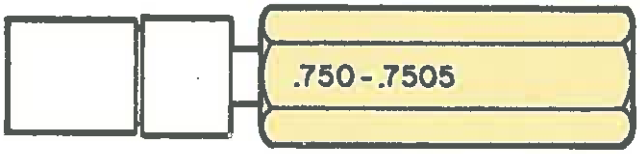


Figure 4-29. *A step plug gage can check for oversize and undersize in a single test.*

4.4.2 Ring Gage

External diameters are checked with ring gages. The go and no-go ring gages are separate units, and can be distinguished from each other by a groove cut on the knurled outer surface of the no-go gage. Refer to **Figure 4-30.**

On ring gages, the gage tolerance is the reverse of plug gages. The opening of the go gage is larger than the opening for the no-go gage.

Figure 4-30. *Ring gages. The larger sizes are cut away to reduce weight. (Standard Tool Co.)*

4.4.3 Snap Gage

A ***snap gage*** serves the same purpose as a ring gage. Snap gages are designed to check internal diameters, external diameters, or both. There are three general types:

- An ***adjustable snap gage*** can be adjusted through a range of sizes. See **Figure 4-31.**
- A ***nonadjustable snap gage*** is made for one specific size. See **Figure 4-32.**

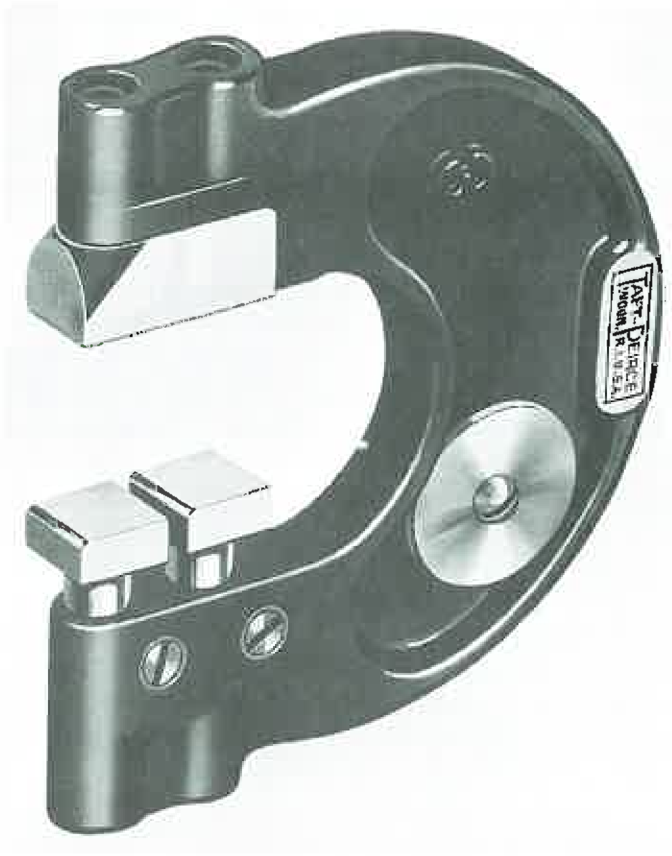


Figure 4-31. *An adjustable snap gage. (Taft-Pierce Co.)*

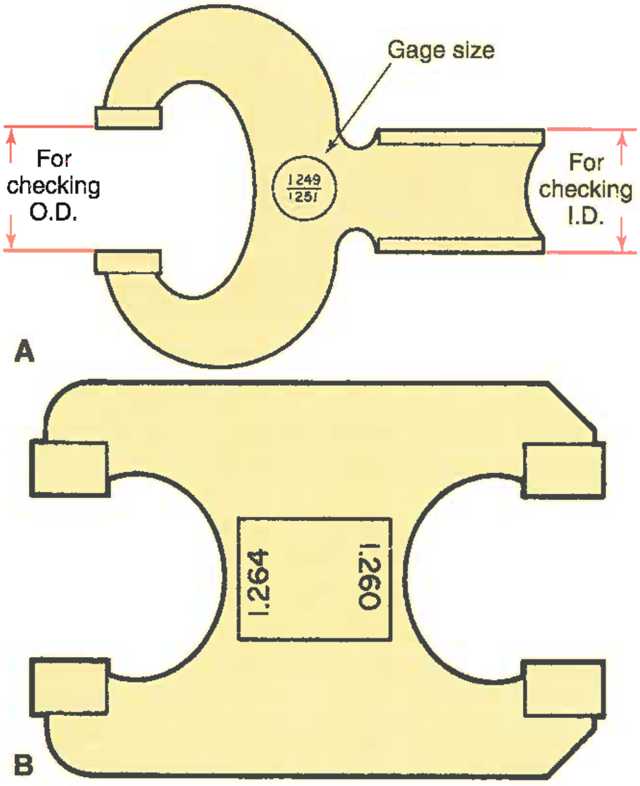


Figure 4-32. *Diagram of a nonadjustable snap gage. A—A combination internal-external nonadjustable snap gage. B—An external nonadjustable snap gage.*

- A ***dial indicator snap gage*** measures the amount of variation in the part measurement. The dial face has a double row of graduations reading in opposite directions from zero. Minus graduations are red and plus graduations are black. Both adjustable and nonadjustable indicating snap gages are available. See **Figure 4-33.**

On snap gages, the anvils should be narrower than the work being measured. This will avoid uneven wear on the measuring surfaces.

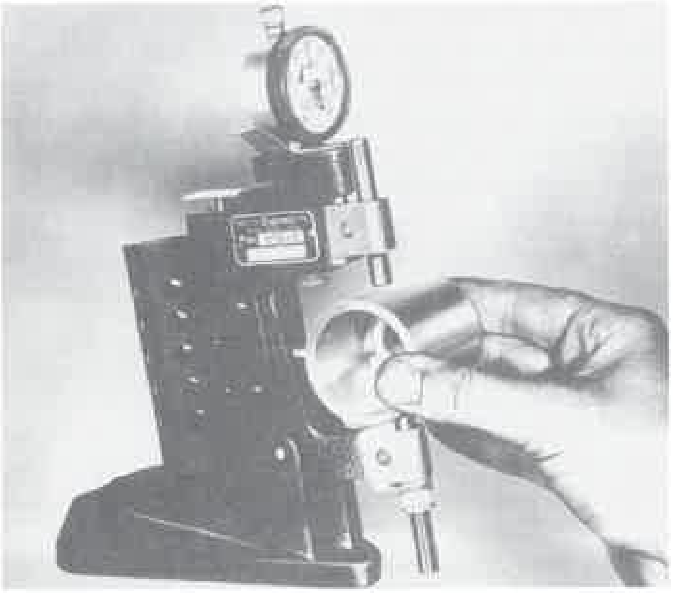


Figure 4-33. *A dial indicator snap gage. (L.S. Starrett Co.)*

4.4.4 Thread Gages

Several types of gages are used to check screw thread fits and tolerances. These gages are similar to the gages already discussed:

- Thread plug gage.
- Thread ring gage.
- Thread roll snap gage.

These gages are illustrated in **Figure 4-34.**

4.4.5 Gage Blocks

Gage blocks, commonly known as ***Jo-blocks or Johansson blocks,*** are precise steel measuring standards. Gage blocks can be purchased in various sets ranging from a few commonly used block sizes to more complete sets. See **Figure 4-35.**

Gage blocks are used to verify the accuracy of master gages. They are also used as working gages and for setting up machining work requiring great accuracy. The Federal Accuracy Grades for gage blocks are shown in **Figure 4-36.**

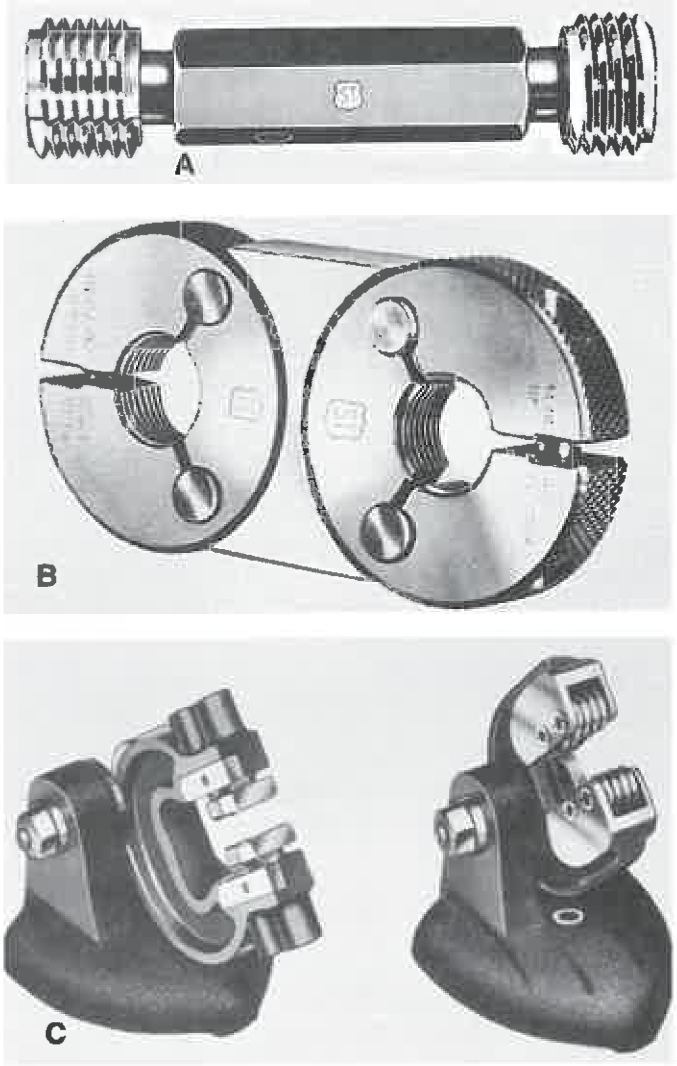


Figure 4-34. *Thread gages. A—Thread plug gage. B—Thread ring gage. C—Go/no-go thread snap gage. (Standard Tool Co. and Taft-Pierce Co.)*

When working with gage blocks, keep the following tips in mind:

- Improper handling can cause temperature changes in the block, resulting in measurement errors. For the most accurate results, blocks should be used in a temperature-controlled room. Handle the blocks as little as possible. When you must handle the blocks, use the tips of your fingers, as shown in **Figure 4-37A.**
- When wringing gage blocks together to build up to desired size, wipe the blocks and then carefully slide them together. They should adhere to each other strongly. Separate the blocks when you are finished. Leaving gage blocks together for extended periods may cause the contacting surfaces to corrode. See **Figure 4-37B.**

Figure 4-35. A typical set of gage blocks. (Federal Products Co.)

Federal Accuracy Grades			
		Tolerance	
Accuracy grade	Former designation	US Conventional system (inch)	Metric system (millimeter)
0.5	AAA	±.000001″	±.00003 mm
1	AA	±.000002″	±.00005 mm
2	A +	+.000004″ –.000002″	+.0001 mm –.00005 mm
3	A&B	+.000006″ –.000002″	+.00015 mm –.00005 mm

Reference temperature: 68°F (20°C)
One inch = 25.4 millimeters exactly

Figure 4-36. Federal Accuracy Grades for gage blocks.

- Wipe gage blocks with a soft cloth or chamois treated with oil. Be sure the oil is one recommended by the gage manufacturer. See **Figure 4-37C.**

4.5 DIAL INDICATORS

Industry is constantly searching for ways to reduce costs without sacrificing quality. Inspection has always been a costly part of manufacturing. To speed up this phase of production without sacrificing accuracy, dial indicators and electronic gages are receiving increased attention.

Dial indicators are designed with shockproof movements and have jeweled bearings (similar to fine watches). There are two types of indicators: balanced and continuous. ***Balanced indicators*** can take measurements on either side of a zero line. ***Continuous indicators*** read from "0" in a clockwise direction. See **Figure 4-38.**

Dial faces are available in a wide range of graduations. They usually read in the following increments:

- 1/1000″ (0.001″)
- 1/100 mm (0.01 mm)
- 1/10,000″ (0.0001″)
- 2/1000 mm (0.002 mm)

Much use is made of dial indicators for centering and aligning work on machine tools, checking for eccentricity, and visual inspection of work. Dial indicators must be mounted to rigid holding devices, **Figure 4-39.**

A ***digital electronic indicator,*** **Figure 4-40,** features direct digital readouts and a traditional graduated dial for fast, accurate reading. These indicators are available as both self-contained and remote readout units.

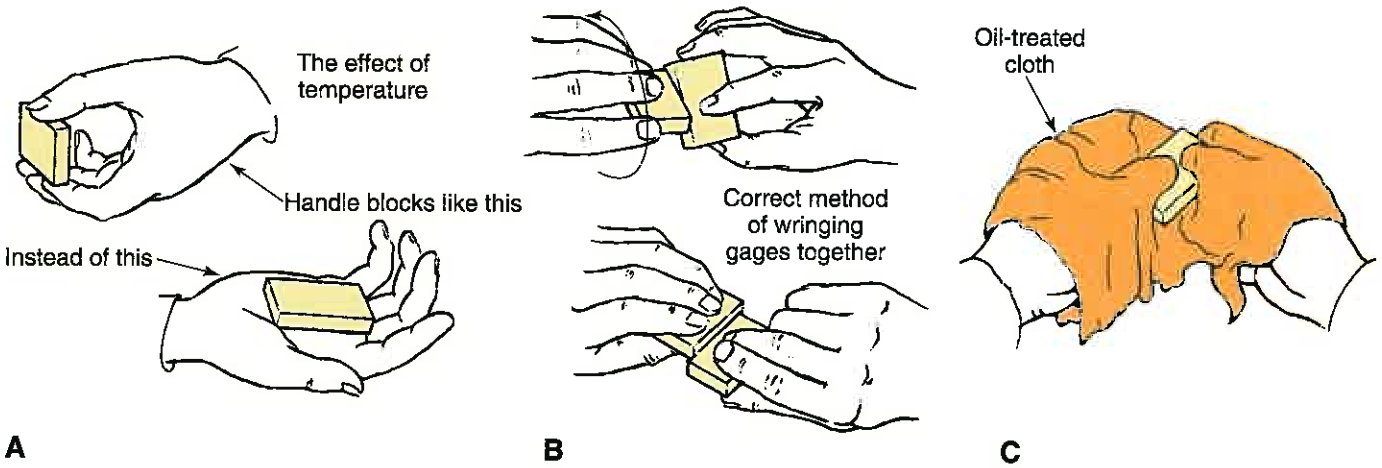


Figure 4-37. Proper care of gage blocks. A—Handling gage blocks. B—Wipe blocks and slide them together. Do not leave blocks together for extended periods. C—Wipe blocks with a soft cloth before storing. (Webber Gage Div., L.S. Starrett Co.)

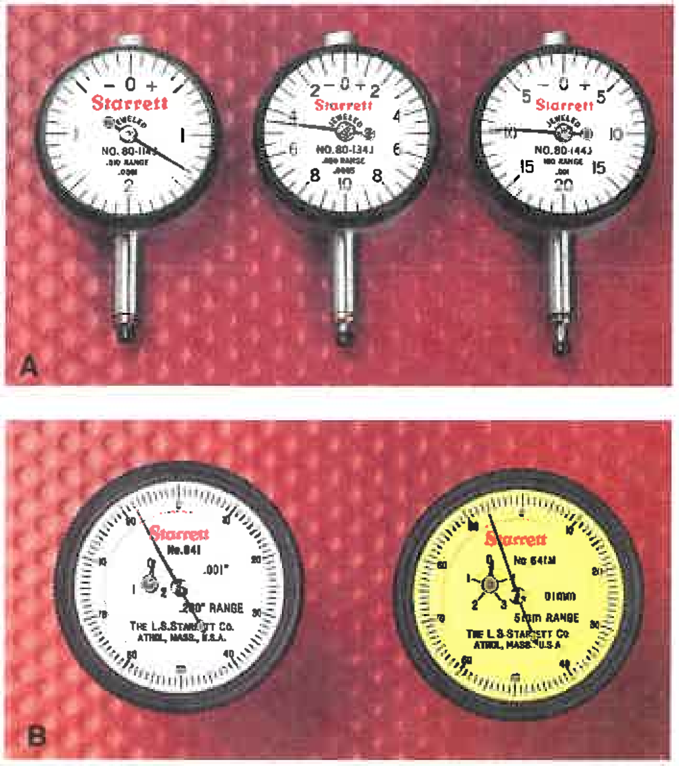


Figure 4-38. The two basic varieties of dial indicators. A—Balanced indicators. B—Continuous indicators. (L. S. Starrett Co.)

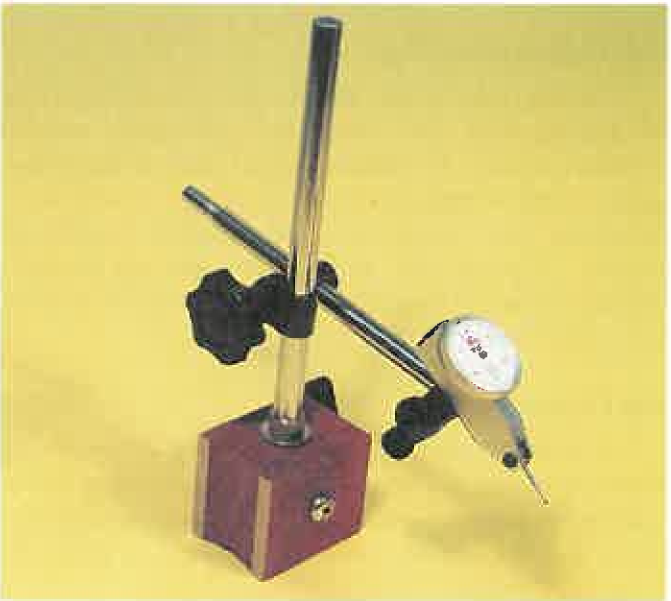


Figure 4-39. Mounting this dial indicator on a magnetic base permits it to be attached to any ferrous metal surface. A push-button releases the magnet.

Figure 4-40. This digital electronic indicator has numeric readouts and a conventional graduated dial. (Federal Products Company)

4.5.1 How to Use a Dial Indicator

The hand on the dial is actuated by a sliding plunger. Place the plunger lightly against the work until the hand moves. The dial face is turned until the "0" line coincides with the hand. As the work touching the plunger is slowly moved, the indicator hand will measure movement.

The dial indicator can show the difference between the high and low points, or the total runout of the piece in a lathe. When machining, adjustments are made until there is little or no indicator movement.

4.6 OTHER GAGING TOOLS

Industry makes wide use of other types of gaging tools. Most of these tools are used for special purposes and are not usually found in a school shop. However, since you might need to use them in industry, it is important to learn about such tools.

4.6.1 Air Gage

An *air gage* uses air pressure to measure hole sizes and hard-to-reach shaft diameters, **Figure 4-41.** This type of gage is especially helpful when measuring deep internal bores. The basic operation of an air gage is illustrated in **Figure 4-42.**

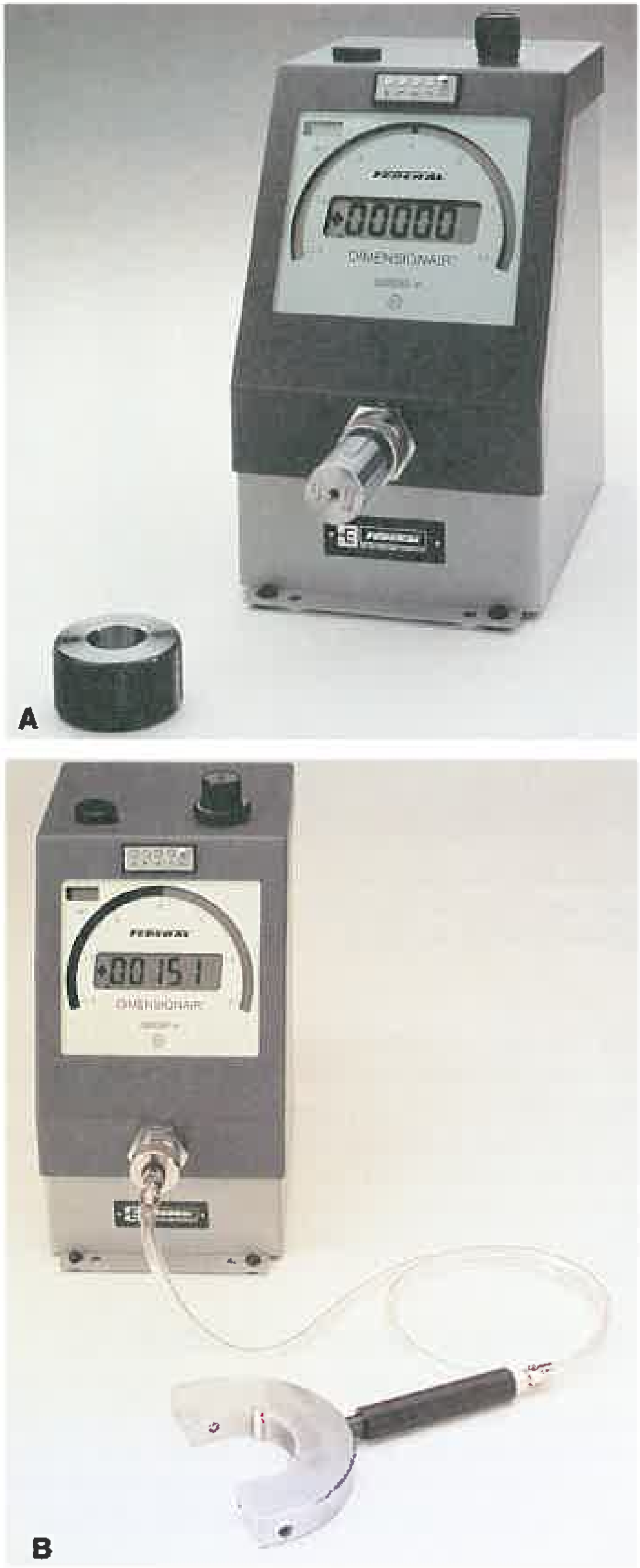


Figure 4-41. Digital air gages are available with either US Conventional or SI Metric readouts. They can check either inside or outside diameters. A—An air gage set up to inspect an internal dimension. The master ring shown with the gage is used to set zero on the readout. B—This gage has an air fork, which is used to check hard-to-reach diameters, such as crankshaft journals. (Federal Products Company)

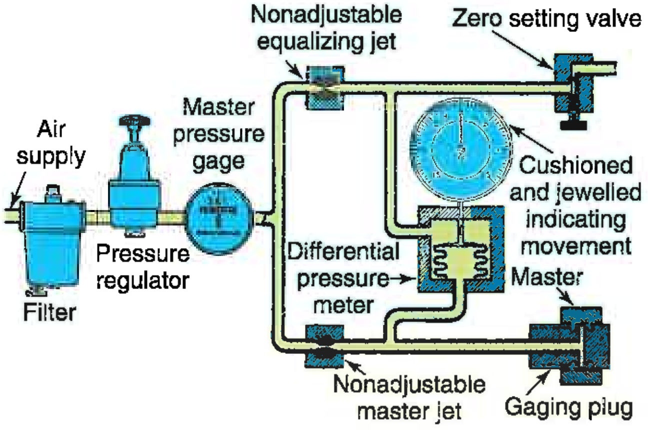


Figure 4-42. This diagram illustrates the operation of an air gage.

There is no actual contact between the measuring gage and wall of the bore being measured. The bore measurement depends on the air leakage between the plug and the hole wall. (The larger the bore diameter, the greater the leakage.) Pressure builds up and the measurement of the back pressure gives an accurate measurement of the hole size.

Change in pressure (air leakage) is measured by a dial indicator, a cork floating on the air stream, or by a ***manometer*** (U-shaped tube in which the height of fluid in the tube indicates pressure).

4.6.2 Electronic Gage

An ***electronic gage,*** **Figure 4-43,** is another type of gaging tool used to make extremely precise measurements. Electronic gages are comparison gages: they compare the size of the work to a reference size. Some are calibrated by means of master gage blocks and others use replaceable gaging probes. These instruments measure in both US Conventional and SI Metric units.

4.6.3 Laser Gaging

A ***laser*** is a device that produces a very narrow beam of extremely intense light. Lasers are used in communication, medical, and industrial applications. Laser is an acronym for *light amplification by stimulated emission of radiation.*

The laser is another area of technology that has moved from the laboratory into the shop. When employed for inspection purposes, it can check the accuracy of critical areas in machined parts quickly and accurately. Refer to **Figure 4-44.**

4.6.4 Optical Comparator

The ***optical comparator*** uses magnification as a means for inspecting parts, **Figure 4-45.** An

Figure 4-43. This electronic bore gaging system can deliver electronic resolution as fine as 0.00001″ (0.0002 mm). Using replaceable gaging probes, the self-contained unit measures diameters ranging from 0.370″ to 2.900″. It also measures in millimeters. It can be linked to a computer for statistical process control (SPC) data collection. (Sunnen Products Company)

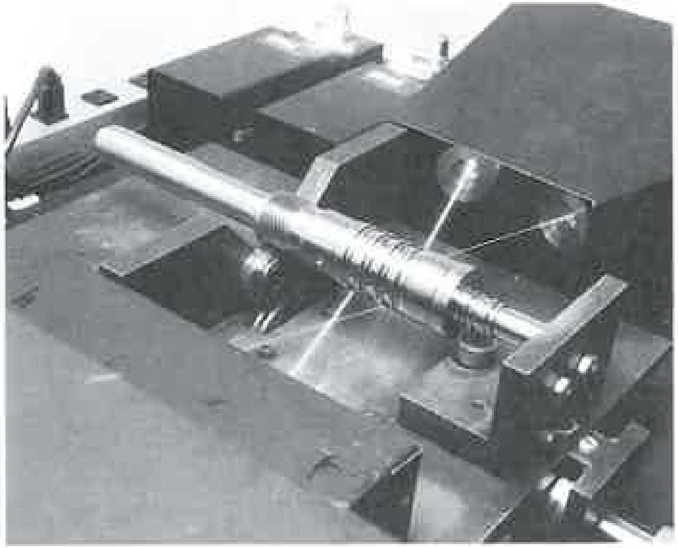


Figure 4-44. This laser is being used to inspect a part from a car's automatic transmission. Manually, one person could inspect no more than four units an hour. The laser can inspect over 120 parts an hour. (Ford Motor Co.)

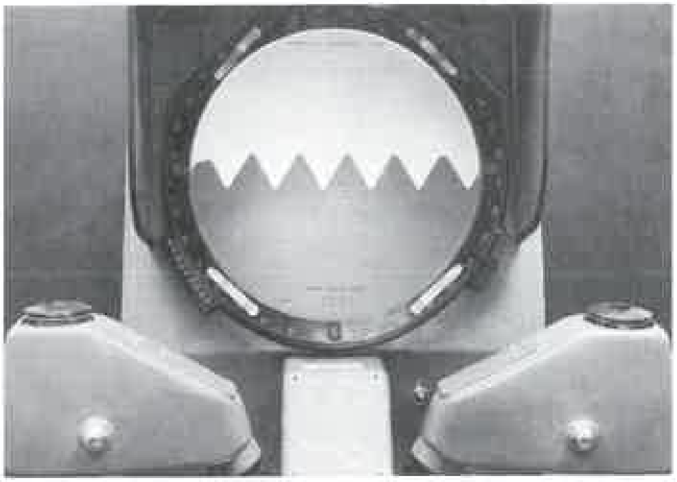


Figure 4-45. This 50-power optical comparator permits a fast check of the tooth formation on a tap.

enlarged image of the part is projected upon a screen for inspection. The part image is superimposed upon an enlarged, accurate drawing of the correct shape and size. The comparison is made visually. Variations as small as 0.0005″ (0.012 mm) can be noted by a skilled operator.

4.6.5 Optical Flats

Optical flats are precise measuring instruments that use light waves as a measuring standard, **Figure 4-46.** The flats are made of quartz and have one face ground and polished to optical flatness. When this face is placed on a machined surface and a special light passed through it, light bands appear on the surface, **Figure 4-47.** The shape of these bands indicate to the inspector the accuracy of the part. See **Figure 4-48.**

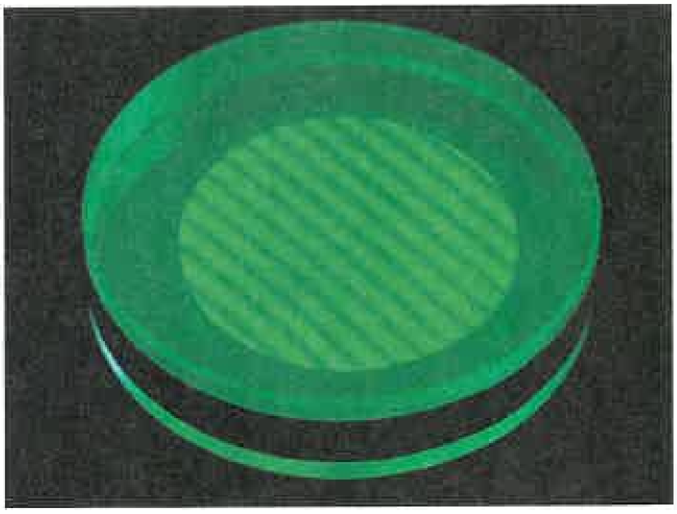


Figure 4-46. Optical flats are used for precision flatness, parallelism, size, and surface variations. (L. S. Starrett Co.)

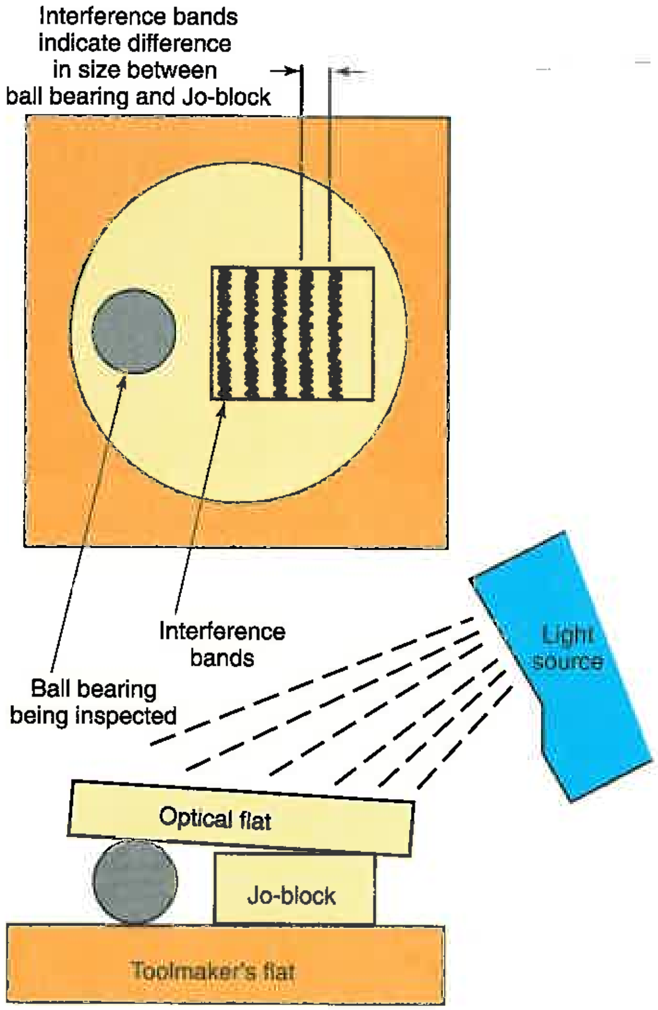


Figure 4-47. Optical flat set-up. Optical flat is placed on top of the work and light is positioned above the flat.

4.6.6 Thickness (Feeler) Gage

Thickness gages are pieces or leaves of metal manufactured to precise thickness, **Figure 4-49.** Thickness gages are made of tempered steel and are usually 1/2″ (12.7 mm) wide.

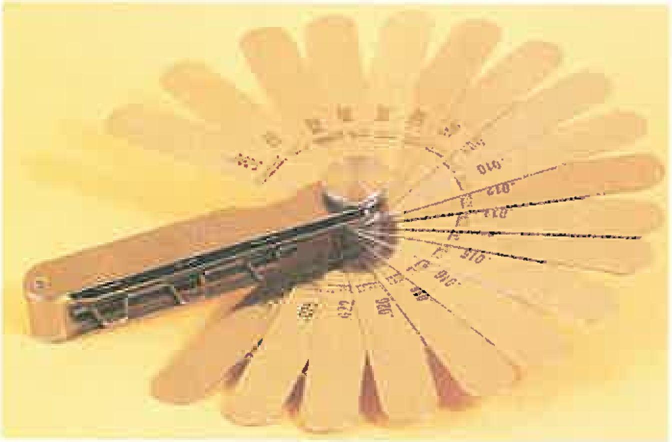
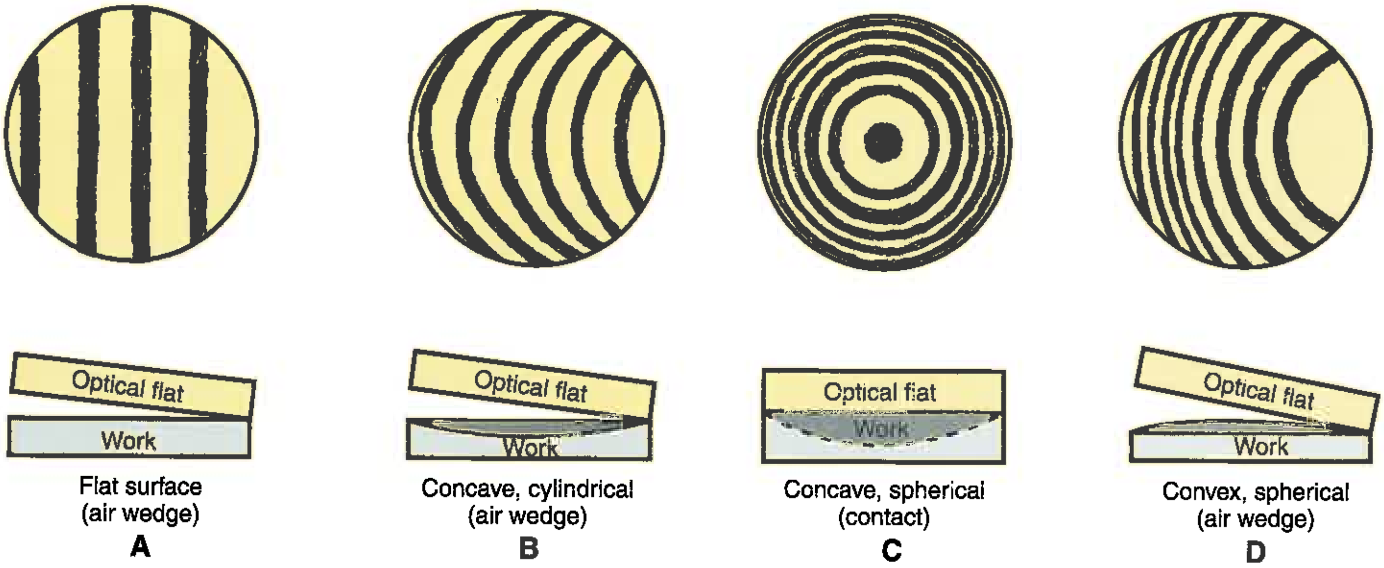


Figure 4-49. Thickness or feeler gages.

Thickness gages are ideal for measuring narrow slots, setting small gaps and clearances, determining fit between mating surfaces, and for checking flatness of parts in straightening operations. See **Figure 4-50.**

4.6.7 Screw Pitch Gage

Screw pitch gages are used to determine the pitch or number of threads per inch on a screw, **Figure 4-51.** Each blade is stamped with the pitch or



Flat surface (air wedge)
A

Concave, cylindrical (air wedge)
B

Concave, spherical (contact)
C

Convex, spherical (air wedge)
D

Figure 4-48. Interference band patterns indicate surface flatness and variations.

Figure 4-50. A thickness gage is used to check part clearance.

Figure 4-51. Screw pitch gages are made for both inch-based and metric threads. (L. S. Starrett Co.)

number of threads per inch. Screw pitch gages are available in US Conventional and SI Metric thread sizes.

4.6.8 Fillet and Radius Gage

The thin steel blades of a ***fillet and radius gage,*** **Figure 4-52,** are used to check concave and convex radii on corners or against shoulders. The gage is used for layout work and inspection, and as a template when grinding form cutting tools. See **Figure 4-53.** The gages increase in radius in 1/64″ (0.5 mm) increments.

4.6.9 Drill Rod

Drill rods are steel rods manufactured to close tolerances to twist drill diameters. They are used to inspect hole alignment, location, and diameter. Drill rods are available in both US Conventional and SI Metric sizes.

Figure 4-52. A set of radius and fillet gages. (L. S. Starrett Co.)

4.7 HELPER MEASURING TOOLS

Some measuring tools are not direct reading and require the help of a rule, micrometer, or Vernier caliper to determine the size of the measurement taken. These are called ***helper measuring tools.***

4.7.1 Calipers

External or internal measurements of 1/64″ (0.4 mm) can be made with calipers, **Figure 4-54.** A caliper does not have a dial or scale that shows a measurement; the distance between points must be measured with a steel rule.

Round stock is measured by setting the caliper square with the work and moving the caliper legs down on the stock. Adjust the tool until the caliper point bears lightly on the center line of the stock. Caliper weight should cause the caliper to slip over the diameter. Hold the caliper next to the rule to make the reading, **Figure 4-55.**

An ***inside caliper*** is used to make internal measurements where 1/64″ (0.4 mm) accuracy is acceptable. Hole diameter can be measured by setting the caliper to approximate size, and inserting the legs into the opening. Hold one leg firmly against the hole wall, and adjust the thumbscrew until the other leg lightly touches the wall exactly opposite the first

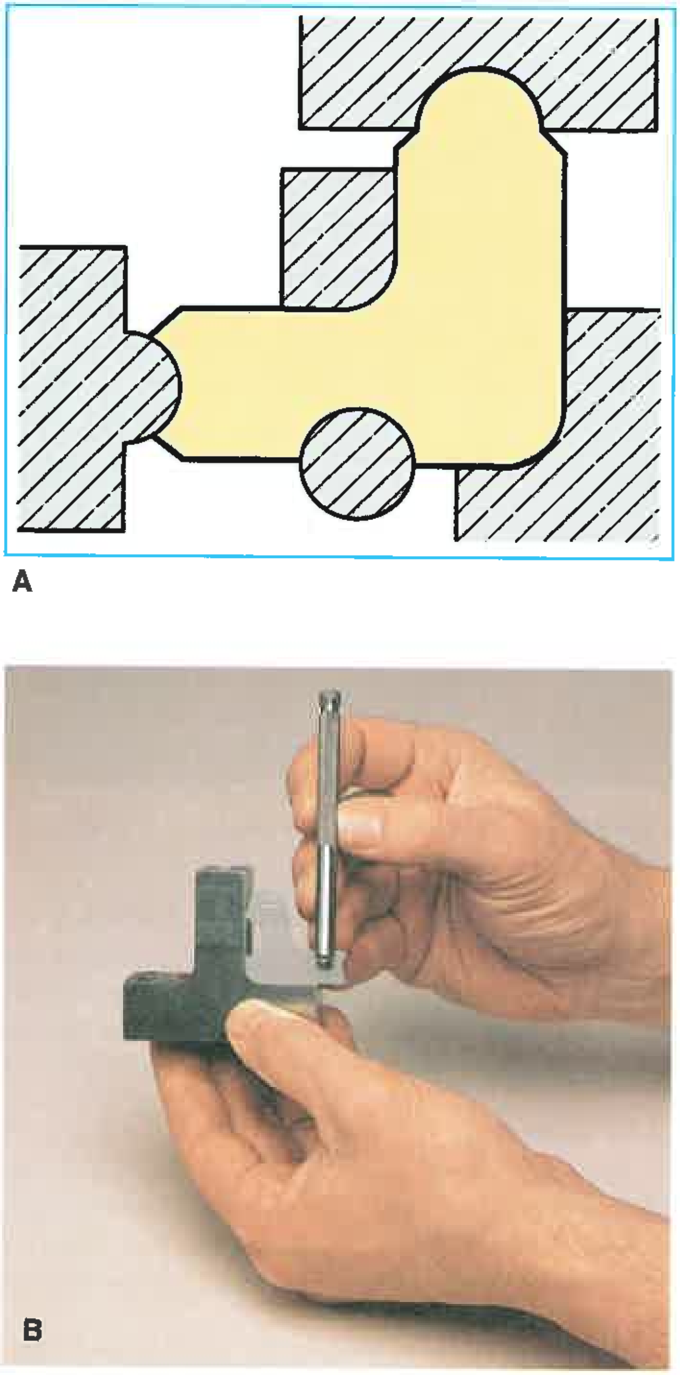


Figure 4-53. *Using a radius gage. A—Various ways a radius gage can be used. B—Using a radius gage holder. (L. S. Starrett Co.)*

leg. The legs should drag slightly when moved in and out, or from side to side.

Considerable skill is required to make accurate measurements with a caliper. See **Figure 4-56.** Much depends upon the machinist's sense of touch. With practice, measurements with accuracy of 0.003" (0.07 mm) can be made. However, a micrometer or Vernier caliper is preferred and must be utilized when greater accuracy is required.

Figure 4-54. *Inside and outside calipers. (L. S. Starrett Co.)*

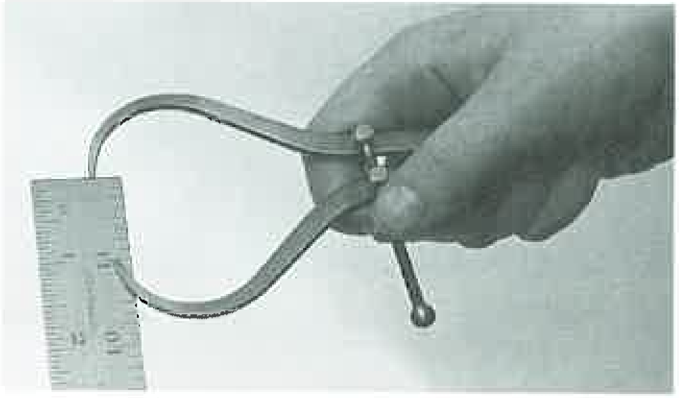


Figure 4-55. *The outside caliper is read with a steel rule.*

4.7.2 Telescoping Gage

A ***telescoping gage*** is intended for use with a micrometer to determine internal dimensions, **Figure 4-57.** Sets of telescoping gages with varying ranges are available, **Figure 4-58.**

To use a telescoping gage, compress the contact legs. The legs telescope within one another under spring tension. Insert the gage into the hole and allow the legs to expand, **Figure 4-59.** After the proper fitting is obtained, lock the contacts into position. Remove the gage from the hole and make your reading with a micrometer, **Figure 4-60.**

4.7.3 Small Hole Gage

A ***small hole gage*** is used to measure openings that are too small for a telescoping gage, **Figure 4-61.** The contacts are designed to allow accurate

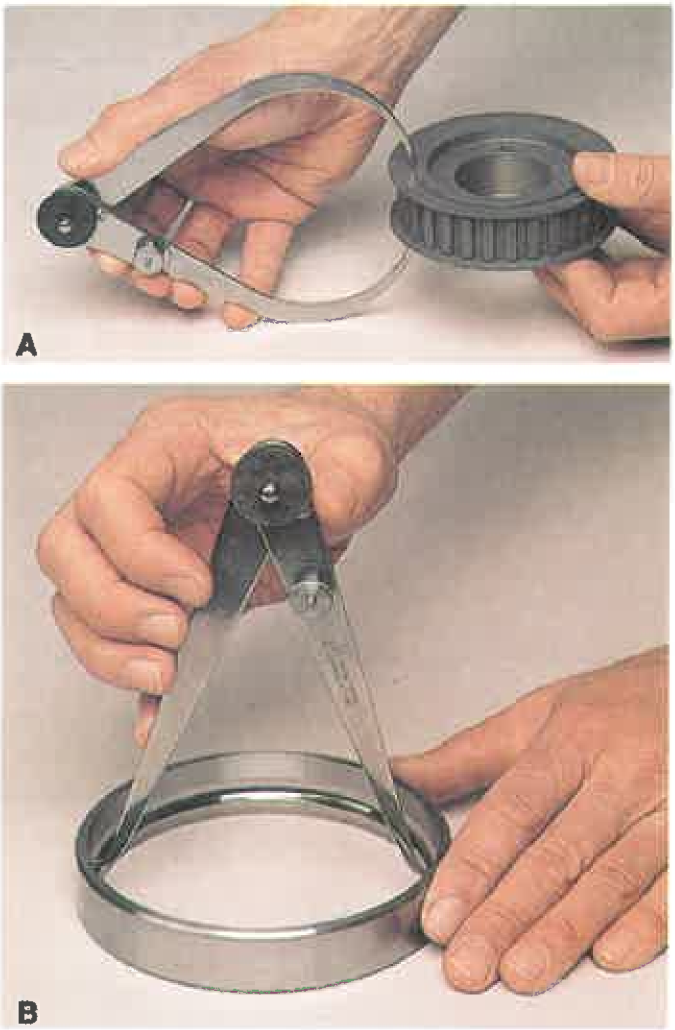


Figure 4-56. Using outside and inside calipers. (L. S. Starrett Co.)

Figure 4-57. A telescoping gage is used with a micrometer.

Figure 4-58. A typical set of telescoping gages.

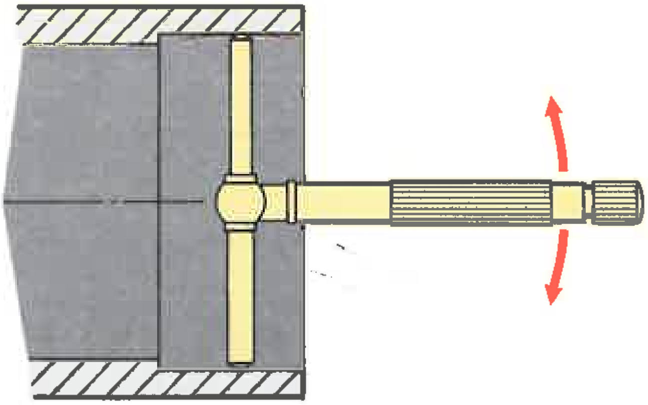


Figure 4-59. Positioning a telescoping gage to measure an inside diameter.

Figure 4-60. After removing the locked telescoping gage, measure it with a micrometer.

Figure 4-61. *Small hole gages are used to measure the diameter of holes that are too small for telescoping gages.*

measurement of shallow grooves, and small diameter holes. They are adjusted to size by the knurled knob at the end of the handle. Measurement is made over the contacts with a micrometer, **Figure 4-62.**

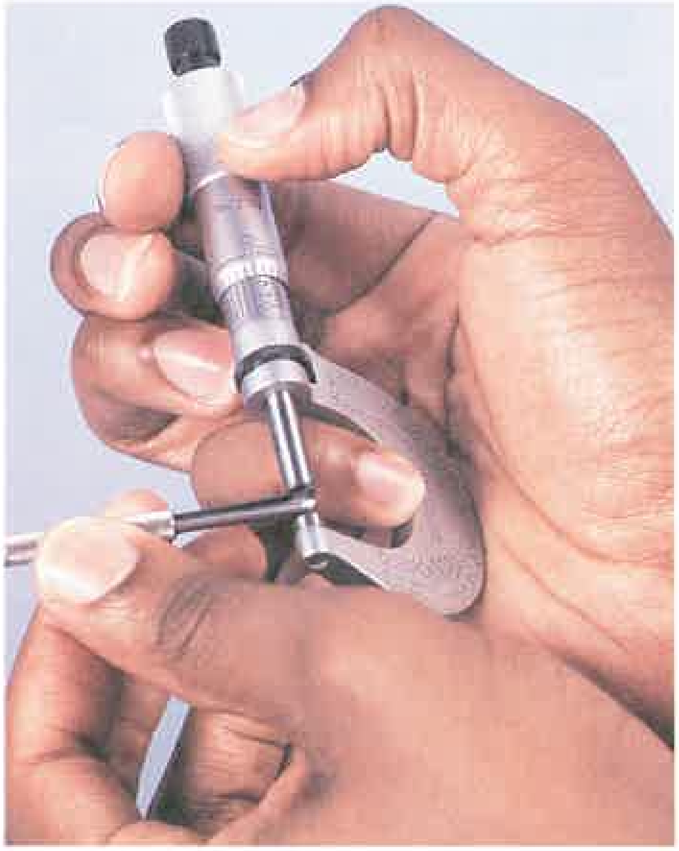
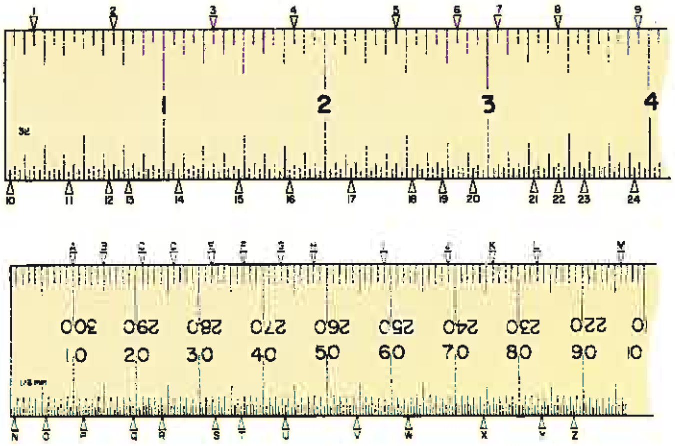


Figure 4-62. *The correct way to measure a small hole gage with a micrometer.*

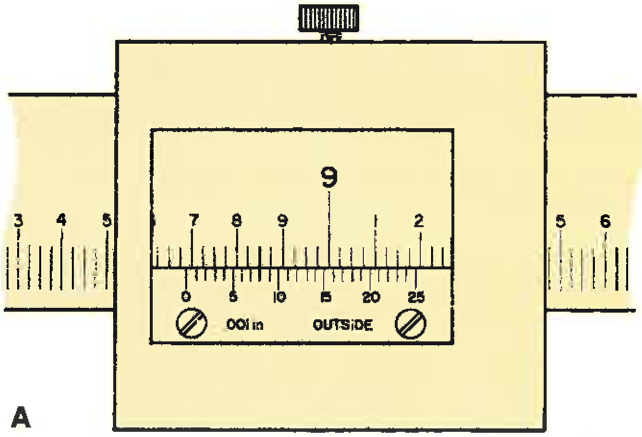
TEST YOUR KNOWLEDGE

Please do not write in this text. Write your answers on a separate sheet of paper.

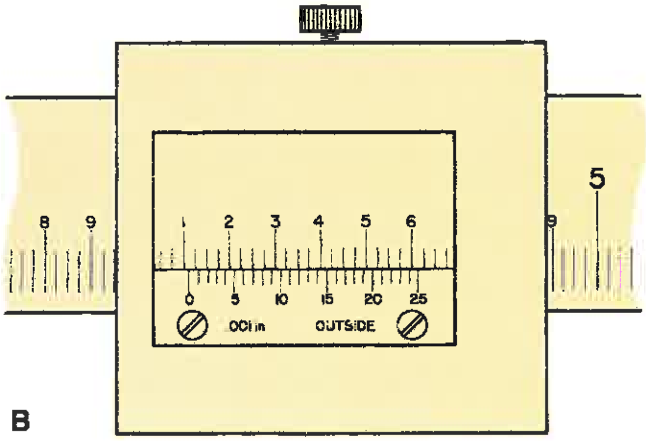
1. Make readings from the rules.



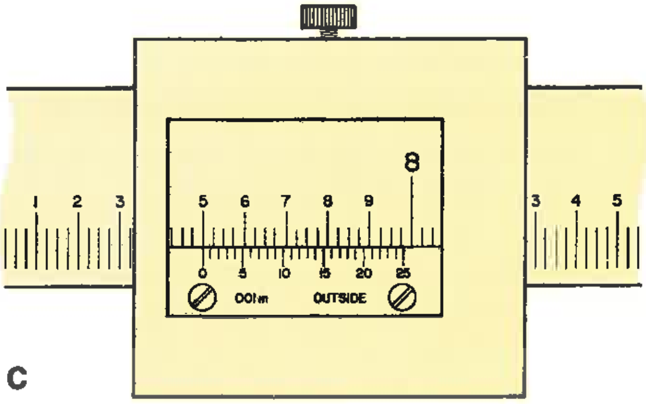
2. Make readings from the Vernier scales shown below.



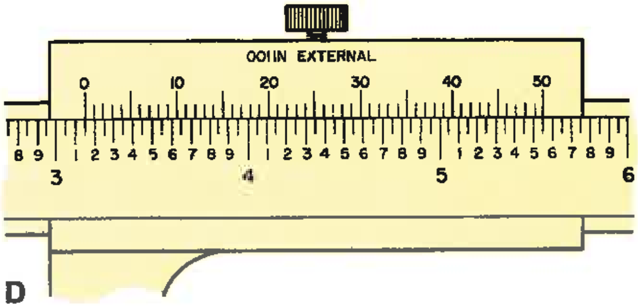
A



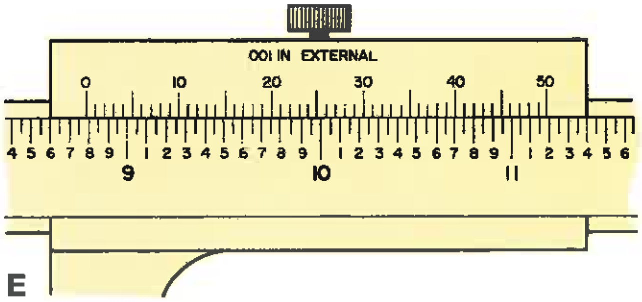
B



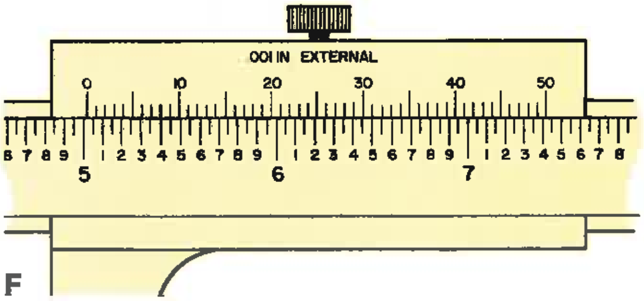
C



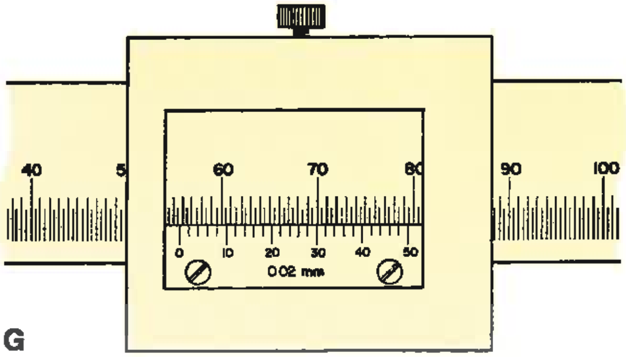
D



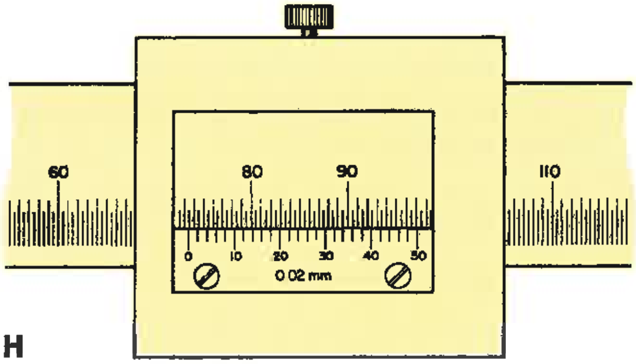
E



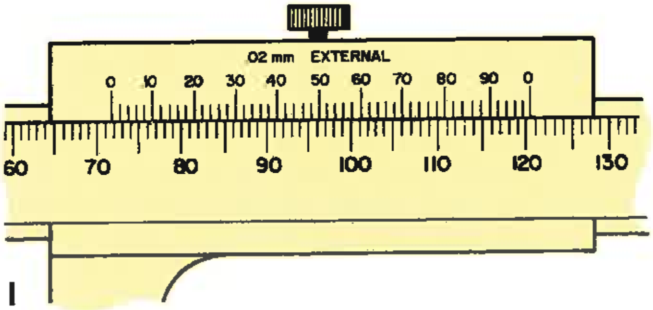
F



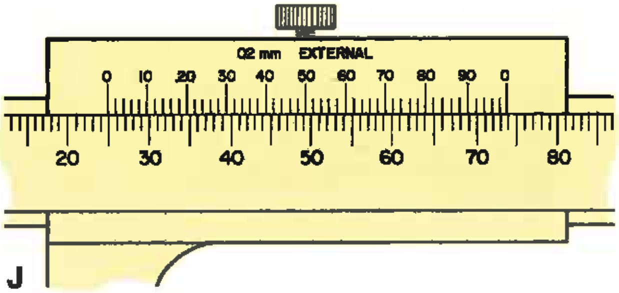
G



H



I



J

Answer the following questions as they pertain to measurement.

3. The micrometer is nicknamed _____.
4. One-millionth part of a standard inch is known as a _____.
5. One-millionth part of a meter is known as a _____.
6. A micrometer is capable of measuring accurately to the _____ and _____ part of standard inch and (in metric versions) to _____ and _____ millimeters.
7. The Vernier caliper has several advantages over the micrometer. List two of them.
8. A Vernier caliper can measure to the _____ part of the inch and (in the metric version) to _____ millimeters.
9. List six precautions that must be observed when using a micrometer or Vernier caliper.

10. The Vernier-type tool for measuring angles is called a _____.

11. How does a double-end cylindrical plug gage differ from a step plug gage?

12. A ring gage is used to check whether _____ are within the specified _____ range.

13. Gage blocks are often referred to as _____ blocks.

14. An air gage employs air pressure to measure deep internal openings and hard-to-reach shaft diameters. It operates on the principle of:
    a. Air pressure leakage between the plug and hole walls.
    b. The amount of air pressure needed to insert the tool properly in the hole.
    c. Amount of air pressure needed to eject the gage from the hole.
    d. All of the above.
    e. None of the above.

15. The dial indicator is available in two basic types. List them.

16. What are some uses for the dial indicator?

17. Name the measuring device that employs light waves as a measuring standard.

18. The _____ is used for production inspection. An enlarged image of the part is projected on a screen where it is superimposed upon an accurate drawing.

19. The pitch of a thread can be determined with a _____.

20. Of what use are fillet and radius gages?

21. What are helper measuring tools?

22. How is a telescoping gage used?

23. Make readings from the micrometer illustrations.


A


B


C


D


E


F

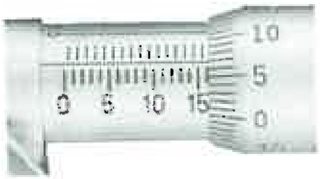

G


H


I


J


K


L

Chapter 5

Layout Work

LEARNING OBJECTIVES

After studying this chapter, you will be able to:

- ○ Explain why layouts are needed.
- ○ Identify common layout tools.
- ○ Use layout tools safely.
- ○ Make basic layouts.
- ○ List safety rules for layout work.

IMPORTANT TERMS

divider
hardened steel square
layout dye
plain protractor
reference line
scriber
straightedge
surface gage
surface plates
V-blocks

Laying out is the term that describes the locating and marking of lines, circles, arcs, and points for drilling holes or making cuts. These lines and reference points on the metal show the machinist where to machine.

The tools for this work are known as ***layout tools.*** Many common hand tools fall into this category. The accuracy of the job will depend upon the proper and careful application of these tools. See **Figure 5-1.**

Figure 5-1. *A few of the tools needed to make a simple layout.*

5.1 MAKING LINES ON METAL

The shiny finish of most metals make it difficult to distinguish layout lines. For this reason, a coating must be placed on the metal before layout.

5.1.1 Layout Dye

There are many coatings used to make layout lines stand out better. Of these coatings, ***layout dye*** is probably the easiest to use. When applied to the metal, this blue-colored fluid offers an excellent contrast between the metal and the layout lines. All dirt, grease, and oil must be removed before applying the dye. If these substances are present on the surface, the dye will not adhere properly.

Chalk will also work on hot finished steel as a layout background. A pencil should not be used because it marks too wide and rubs off.

5.1.2 Scriber

An accurate layout requires fine lines that must be scribed (scratched) into the metal. A scriber will produce these lines, **Figure 5-2.** The point is made of hardened steel, and is kept needle-sharp by frequent honing on a fine oilstone. Many styles of scribers are available.

Never carry a scriber in your pocket. It can puncture the skin easily.

5.1.3 Divider

The scriber is used to draw straight lines. A ***divider*** is used to draw circles and arcs, **Figure 5-3.** It is essential that both legs of the tool be equal in length and kept pointed. Measured distances can be laid out with a divider, **Figure 5-4.** To set the tool to the correct distance, set one point on the inch or centimeter mark of a steel rule, and open the divider until the other leg is set to the proper measurement, **Figure 5-5.**

Figure 5-2. Scribers are used to mark parts during layout. A—The long bent point on this scriber can reach through holes. B—This pocket scriber has a removable point. The point can be reversed when the scriber is not being used, protecting the tip and making the work area safer.

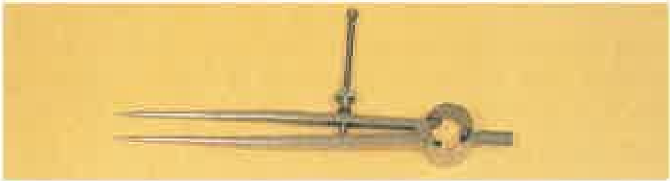


Figure 5-3. A divider is used to mark lines, arcs, and circles.

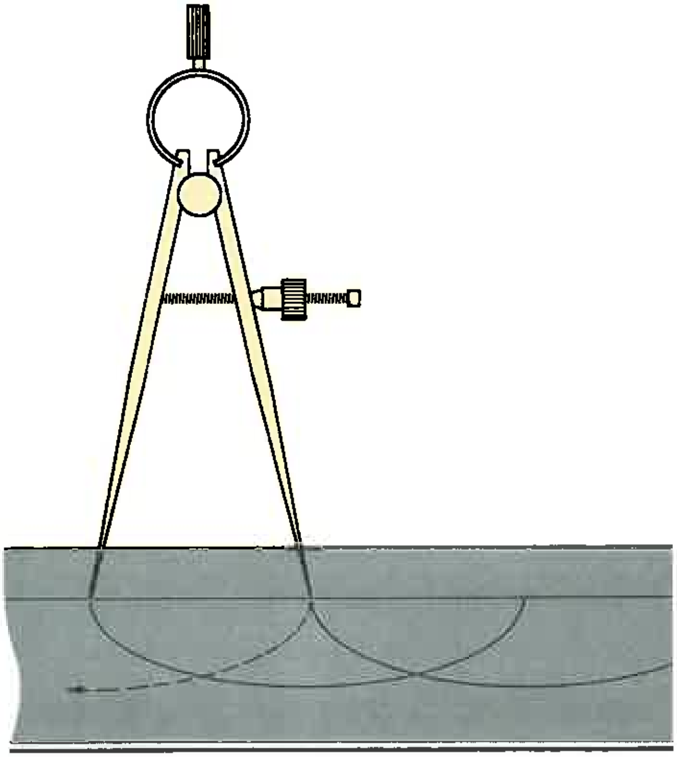


Figure 5-4. Equal spaces can be laid out by "walking" the divider.

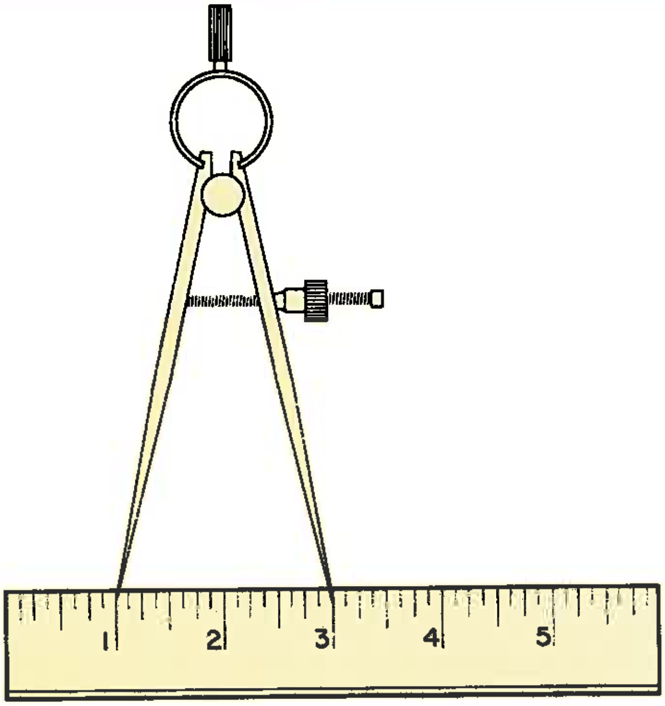


Figure 5-5. To set a divider to a desired dimension, place it on a rule as shown.

Circles and arcs that are too large to be made with a divider are drawn with a ***trammel,*** **Figure 5-6.** This consists of a long thin rod, called a ***beam,*** on which two sliding heads with scriber points are mounted. One head is equipped with an adjusting screw. ***Extension rods*** can be added to the beam to increase the capacity of the tool.

The ***hermaphrodite caliper*** is a layout tool with one leg that is shaped like a caliper and the other pointed like a divider, **Figure 5-7.** Lines parallel to the edge of the material, either straight or curved, can be drawn with the tool, **Figure 5-8.** It can also be used to locate the center of irregularly shaped stock.

5.1.4 Surface Gage

A ***surface gage*** has many uses, but is most frequently employed for layout work, **Figure 5-9.** It consists of a ***base, spindle,*** and ***scriber.*** An adjusting screw is fitted for making fine adjustments. The scriber is mounted in a way that allows it to be pivoted into any position. A surface gage can be utilized to scribe lines at a given height and parallel to the surface, **Figure 5-10.** A V-slot in the base permits the tool to be also employed on a curved surface.

To check whether a part is parallel to a given surface, fit the surface gage with a dial indicator. Set the indicator to the required dimension with the aid of gage blocks. The tool is then moved back and forth along the work, **Figure 5-11.**

A height gage can be used in the same manner as a surface gage. See **Figure 5-12.**

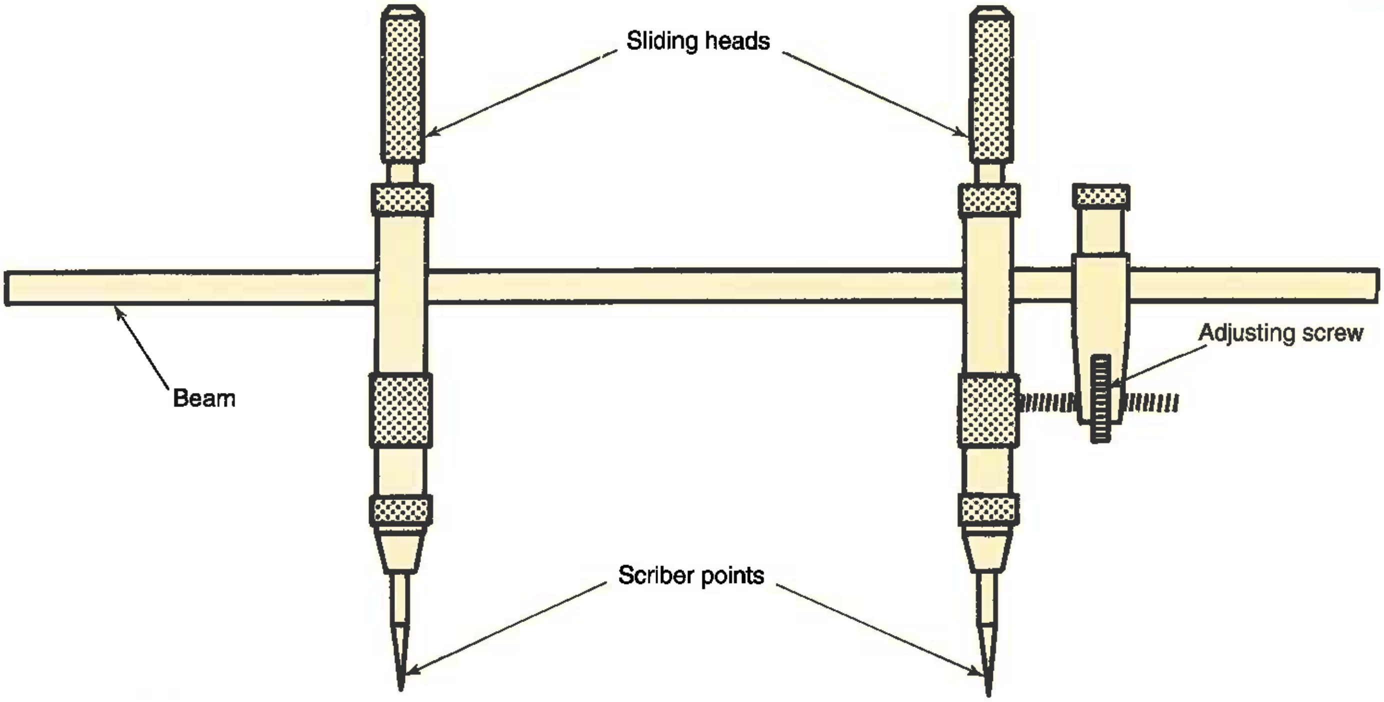


Figure 5-6. *Large circles and arcs are drawn with a trammel.*

Figure 5-7. *A hermaphrodite caliper has a blunt end for the sliding surface and a point for scribing. (L.S. Starrett Co.)*

5.1.5 Surface Plate

Every linear measurement depends upon an accurate reference surface. ***Surface plates*** provide a reference surface (plane) for layout and inspection.

Surface plates can be purchased in sizes up to 72" by 144" (1800 mm by 3600 mm) and in various grades. Surface plate grade differences are given in degrees of flatness:

- Grade AA for laboratories.
- Grade A for inspection.
- Grade B for too room and layout applications.

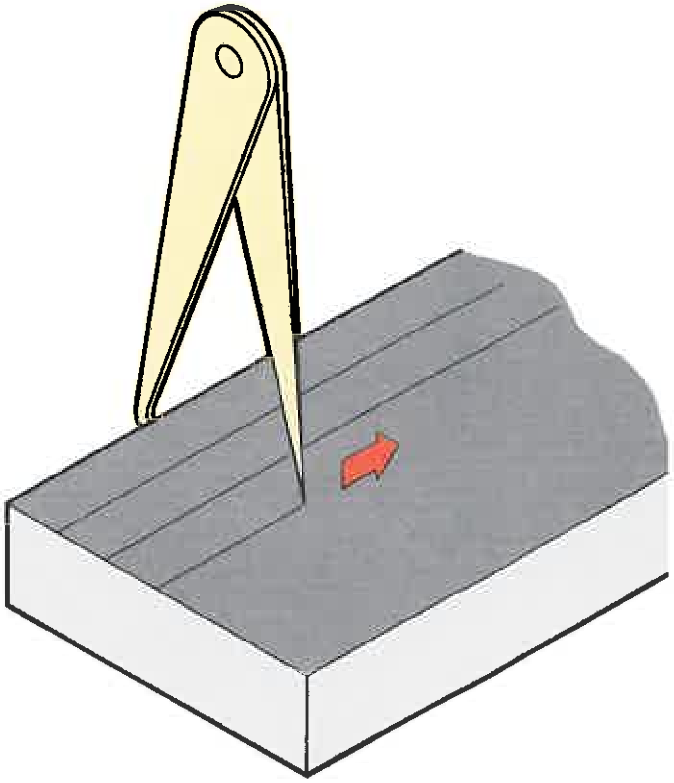


Figure 5-8. *Scribing lines parallel to an edge with a hermaphrodite caliper.*

Figure 5-9. This small surface gage is designed for light work. (L. S. Starrett Co.)

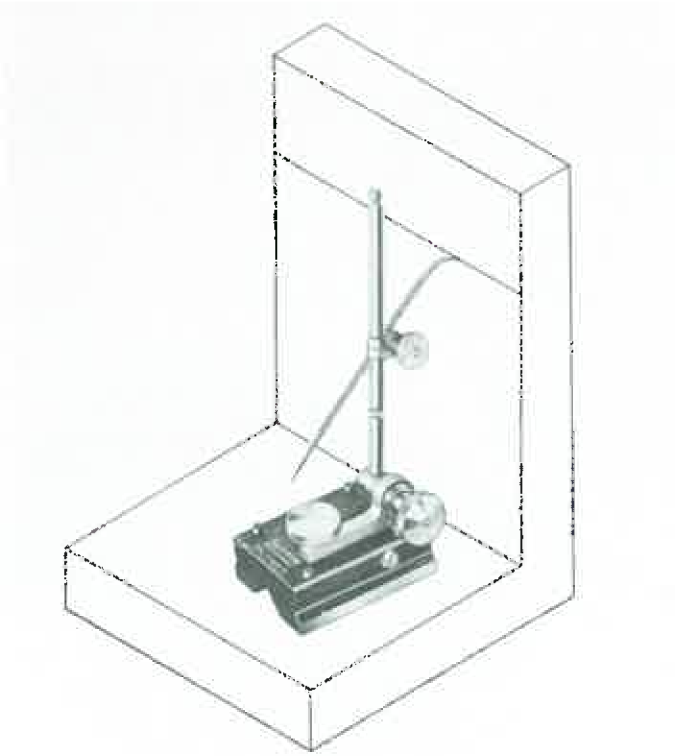


Figure 5-10. Carefully slide the surface gage to scribe lines parallel to the base. Handle the gauge carefully because sharp points can cause injury.

Most surface plates made today are granite but some are semisteel, **Figure 5-13.** Granite is more stable. Semisteel surface plates are more affected by temperature changes.

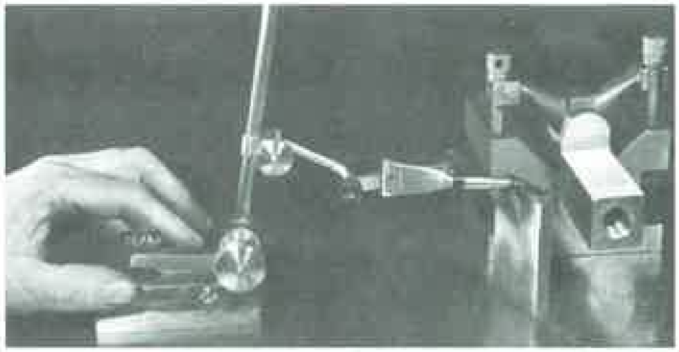


Figure 5-11. A machinist is using gage blocks to set the indicator, which is mounted on the surface gage. (Lufkin Rule Co.)

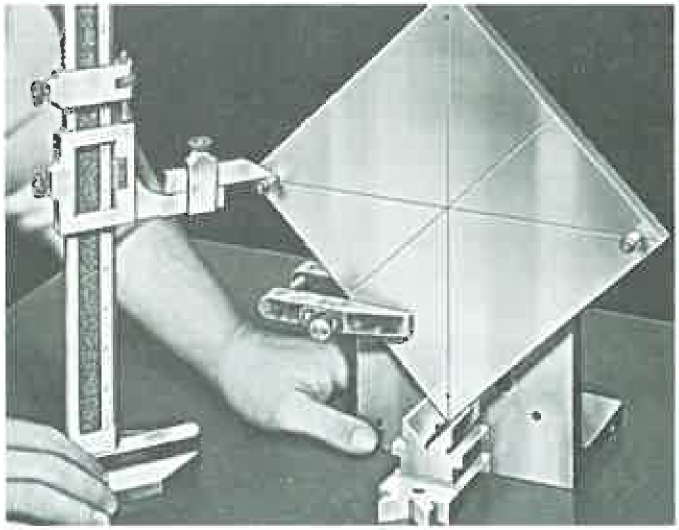


Figure 5-12. A height gage can be used to scribe lines on a workpiece. Here, a machinist is using a Vernier height gage. Note that a V-block and angle plate support the work during layout. (L. S. Starrett Co.)

Surface plates are used primarily for layout and inspection work. They should never be used for any job that could mar or nick the surface.

When square reference surfaces are needed, a ***right angle plate*** is used, **Figure 5-14.** The plates can be placed in any position with the work clamped to the face for layout, measurement, or inspection.

An accurate surface parallel to the surface plate can be obtained using ***box parallels,*** **Figure 5-15.** All surfaces are precision-ground to close tolerances.

5.1.6 V-Blocks

V-blocks support round work for layout and inspection, **Figure 5-16.** They are furnished in matched pairs with surfaces that are ground square to close tolerances. Ribs are cast into the body for weight reduction. The ribs also can be used as clamping surfaces.

Figure 5-13. *Surface plates are available in various grades and materials. A—A granite surface plate. B—A semisteel surface plate. (L. S. Starrett Co. and Challenge Machinery Co.)*

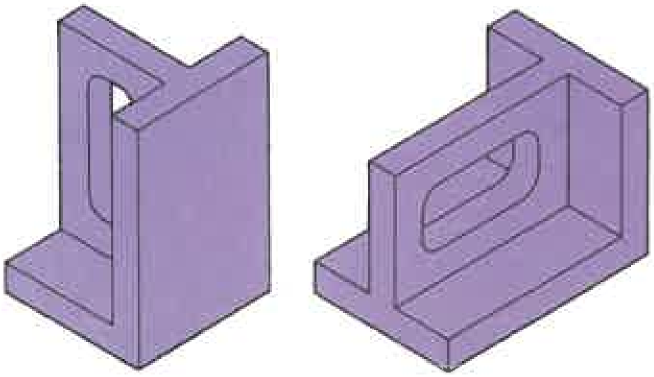


Figure 5-14. *Right angle plates are often used to check perpendicular surfaces.*

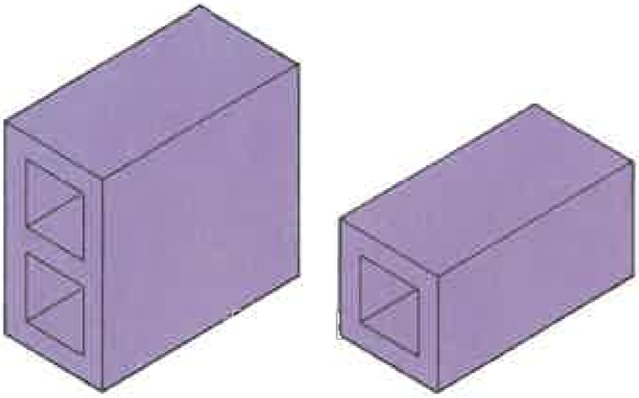


Figure 5-15. *Box parallels are available in a number of sizes.*

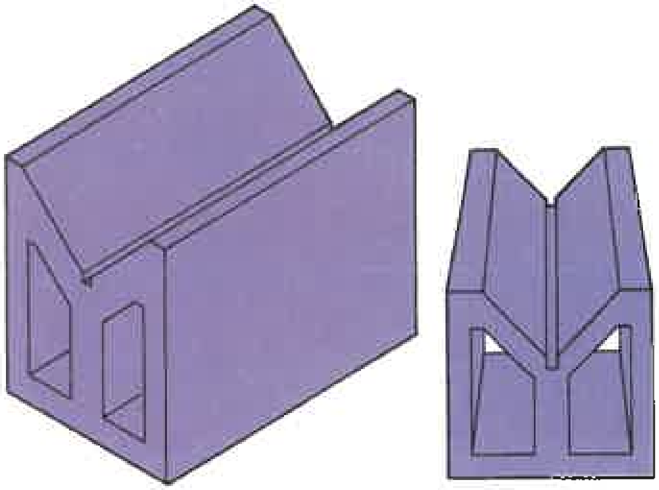


Figure 5-16. *V-blocks can be used to hold round stock for layout and measurement work.*

5.1.7 Straightedge

Long flat surfaces are checked for accuracy with a ***straightedge,*** **Figure 5-17.** This tool is also used for laying out long straight lines. Straightedges can be made from steel or granite, with steel being more common.

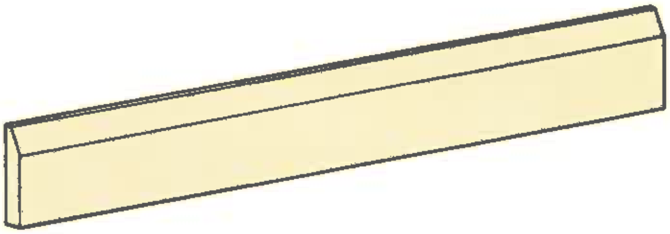


Figure 5-17. *Steel straightedges are very common. Straightedges are also available in granite.*

5.2 SQUARES

The ***square*** is employed to check 90° (square) angles. The tool is also used for laying out lines that must be at right angles to a given edge or parallel to another edge. Some simple machine setups can be made quickly and easily with a square.

Many different types of squares are available. The following are a few of the most common:

- ***Hardened steel square***—This square is recommended when extreme accuracy is required. See **Figure 5-18.** The square has true right angles, both inside and outside. It is accurately ground and lapped for straightness and parallelism. The tool comes in sizes up to 36″ (910 mm).

Figure 5-18. The hardened steel square is handy during layout work. (L. S. Starrett Co.)

- ***Double square***—This is more practical than the steel square for many jobs because the sliding blade is adjustable and interchangeable with other blades, **Figure 5-19.** The tool should not be used where great precision is required. The bevel blade has one angle for checking *octagons* (45° angles), and another for checking *hexagons* (60° angles).

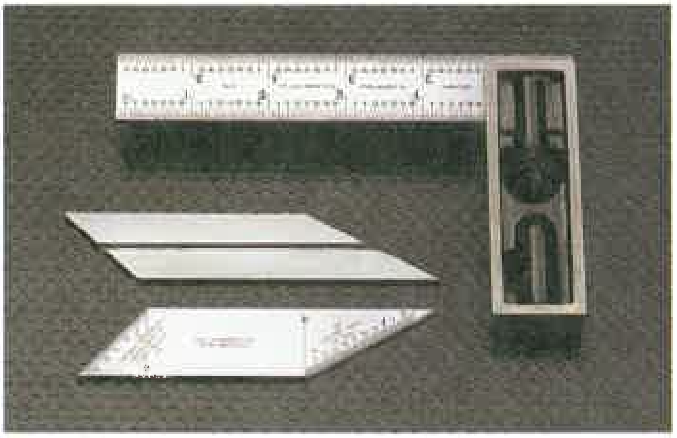


Figure 5-19. This double square has a graduated blade, beveled blade, and a drill grinding blade. (L. S. Starrett Co.)

A drill grinding blade is also available for the double square. One end is beveled to 59° for drill grinding and the other end is beveled at 41° for checking the cutting angles of machine screw countersinks. Both ends are graduated for measuring the length of the cutting lips, to ensure that the cutting tools are sharpened on center.

- ***Combination set***—This tool consists of a hardened blade, square head, center head, and bevel protractor. The blade fits all three heads, **Figure 5-20.** Combination sets are adaptable to a large variety of operations, making them especially valuable in the shop. The square head, which has one 45° edge, makes it possible for the tool to serve as a miter square. By projecting the blade the desired distance below the edge, it also serves as a depth gage, **Figure 5-21.** The spirit level, fitted in one edge, allows it to be used as a simple level.

  With the rule properly inserted, the *center head* can be used to quickly find the center of round stock. This is illustrated in **Figure 5-22.** The *protractor head* can be rotated through 180° and is graduated accordingly. The head can be locked with a locking nut, making it possible to accurately determine and scribe angles, **Figure 5-23.** The head also has a level

Figure 5-20. A combination set will perform various layout tasks.

Figure 5-21. The combination set can be used to check squareness and to measure like a depth gage.

Figure 5-22. *Using a center head and rule to locate the center of a piece of round stock.*

Figure 5-23. *Angular settings on layouts can be made with the protractor head and rule of the combination set.*

built in, making it possible to use it as a level for positioning angles for inspection, layout, or machining.

Handle a square with care. The blade is mounted solidly, but if the tool is dropped, the blade can be "sprung," ruining the square.

5.3 MEASURING ANGLES

In addition to the protractor head of the combination set, other angle measuring tools are employed in layout work. The accuracy required by a job will determine which tool must be used.

When angles do not need to be laid out or checked to extreme accuracy, a ***plain protractor*** can be used, **Figure 5-24.** The head is graduated from 0° to 180° in both directions for easy reading.

A ***protractor depth gage*** is suitable for checking angles and measuring slot depths, **Figure 5-25.**

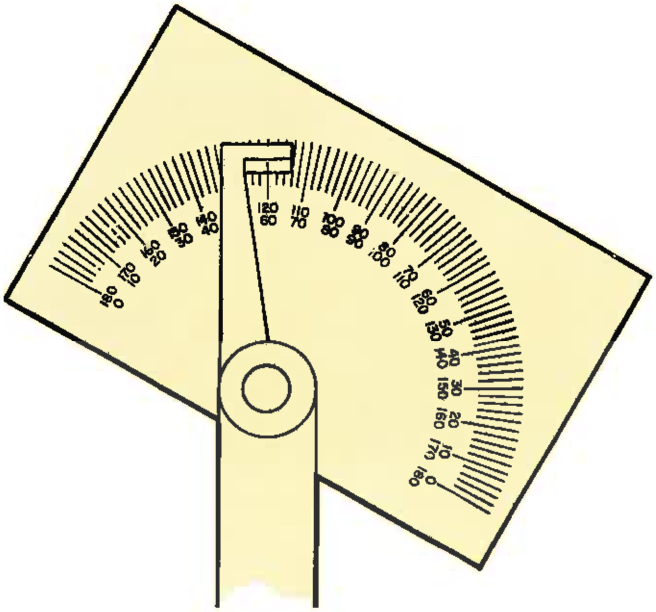


Figure 5-24. *A plain steel protractor will show angles with moderate precision.*

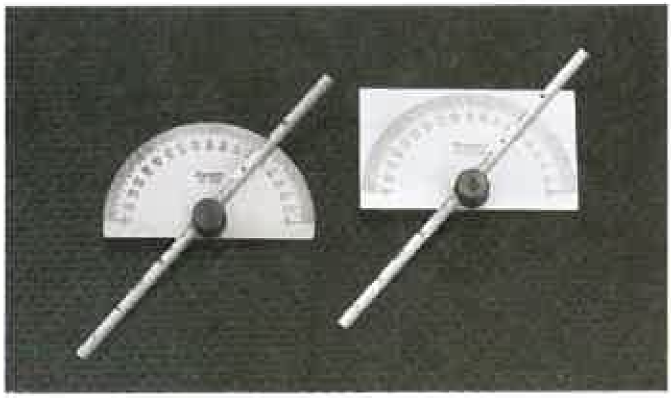


Figure 5-25. *A protractor depth gage. (L. S. Starrett Co.)*

A ***universal bevel*** is useful for checking, laying out, and transferring angles, **Figure 5-26.** Both blade and stock are slotted, making it possible to adjust the blade into any desired position. A thumbscrew locks it tightly in place.

When a job requires extreme accuracy, the machinist uses a ***Vernier protractor,*** **Figure 5-27.** With this tool, angles of 1/12 of a degree (5 minutes) can be accurately measured.

5.4 SIMPLE LAYOUT STEPS

Each layout job requires planning before the operation can be started. **Figure 5-28** shows a typical job. Use the following planning procedure:

1. Carefully study the drawings.
2. Cut stock to size and remove all burrs and sharp edges.

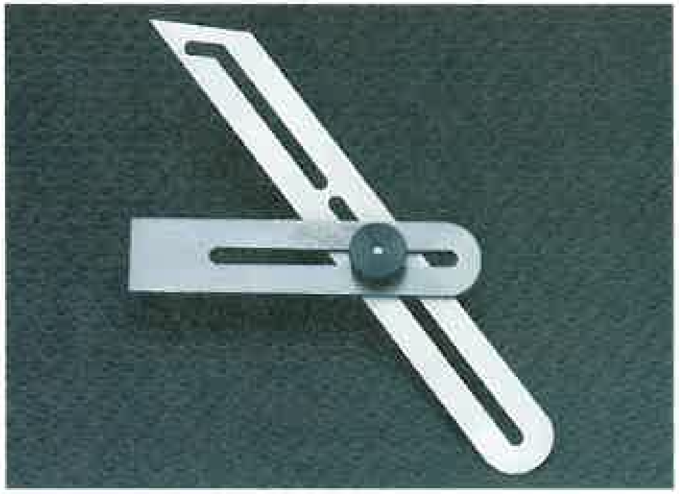


Figure 5-26. *A universal bevel can be locked at various angles. (L. S. Starrett Co.)*

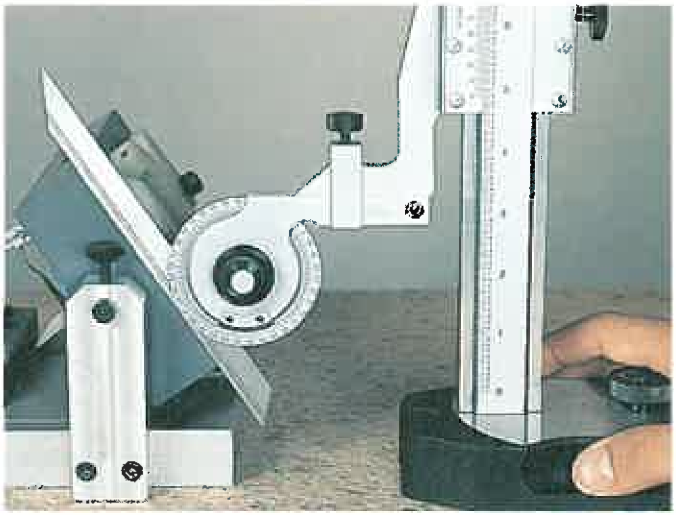
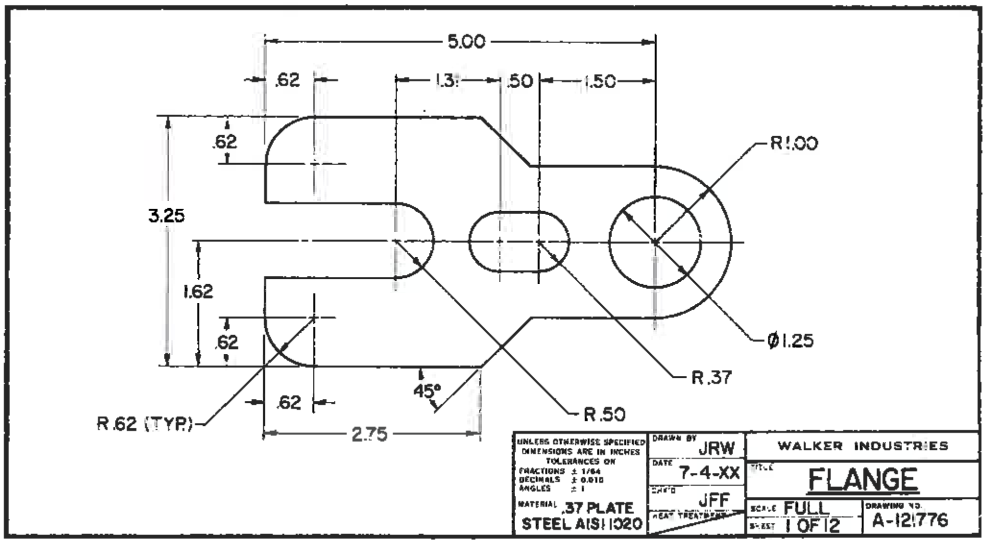
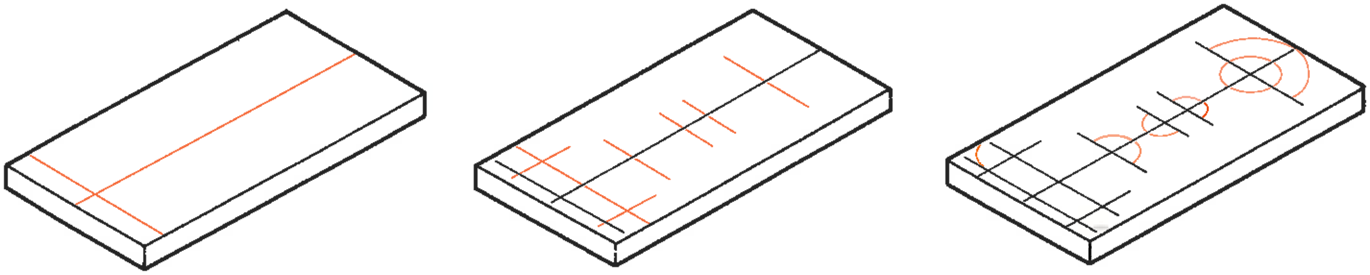
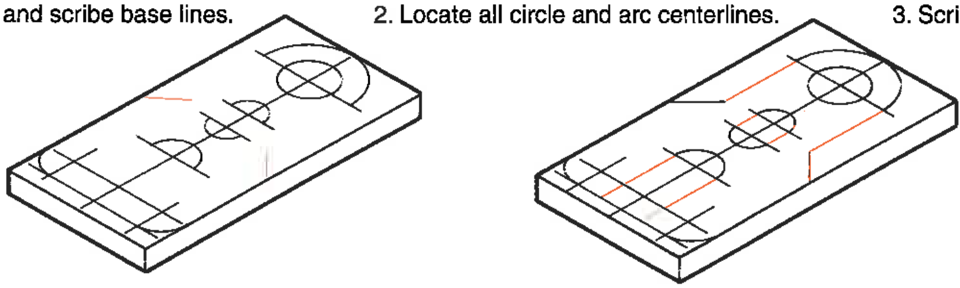


Figure 5-27. *Precise angular measurements are made with a Vernier protractor. In this view, the protractor is mounted on a height gage. (L. S. Starrett Co.)*





1. Locate and scribe base lines.
2. Locate all circle and arc centerlines.
3. Scribe in all circles and arcs.



4. Locate and scribe in angular lines.
5. Connect remaining points

Figure 5-28. *Compare the part drawing with steps involved in laying out the job.*

3. Clean all dirt, grease and oil from the work surface. Apply layout dye.
4. Locate and scribe a ***reference line*** (***base line***). You will make all measurements from this line. If the material has one true edge, it can be used in place of the base line.
5. Locate and center points of all circles and arcs.
6. Use a ***prick punch*** to mark the point where centerlines intersect. The sharp point (30° to 60°) of this punch makes it easy to locate the position. After the prick punch mark has been checked, it is enlarged slightly with a ***center punch***, **Figure 5-29.**

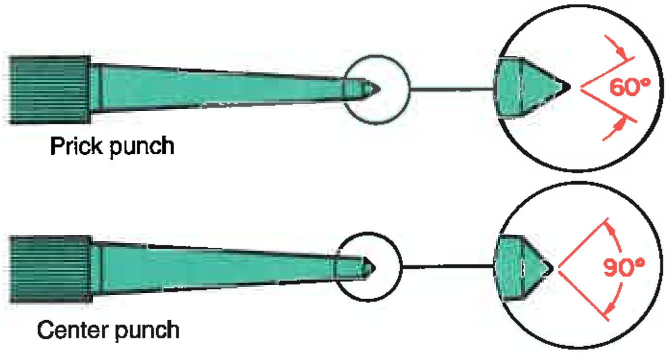


Figure 5-29. *A prick punch has a more sharply angled point than a center punch. The prick punch is used to mark a location. After the prick punch mark is checked, it is enlarged with a center punch.*

7. Scribe in all circles and arcs with a divider or trammel.
8. If angular lines are necessary, scribe them using the proper layout tools. You can also locate the correct points by measuring and connecting them using a rule or straightedge and a scribe.
9. Scribe in all other internal openings.
10. Lines should be clean and sharp. Any double or sloppy line work should be removed by cleaning it off with a solvent. Then apply another coat of dye before scribing the line again.

5.5 LAYOUT SAFETY

- Never carry an open scriber, divider, trammel, or hermaphrodite caliper in your pocket.
- Always cover sharp points with a cork when the tool is not being used.
- Wear goggles when grinding scriber points.
- Get help when you must move heavy items, such as angle plates or V-blocks.
- Remove all burrs and sharp edges from stock before starting layout work.

TEST YOUR KNOWLEDGE

Please do not write in the text. Write your answers on a separate sheet of paper.

1. What is used to make layout lines easier to see?
2. Why are layout lines used?
3. Straight layout lines are drawn with a _____.
4. Circles and arcs are drawn on work with a _____.
5. Large circles and arcs are drawn with a _____.
6. What is wrong with using a pencil to make layout lines on metal?
7. A _____ is the flat granite or steel surface used for layout and inspection work.
8. What layout operations can be performed with a combination set?
9. Round stock is usually supported on _____ for layout and inspection.
10. Long flat surfaces can be checked for trueness with a _____.
11. The center of round stock can be found quickly with the _____ and rule of a combination set.
12. Angular lines that must be very accurate should be laid out with a _____.
13. The _____ punch has a sharper point than the _____ punch.
14. List three safety precautions that you should observe when doing layout work.

A large tap and die set like this one is found in many shops. It includes a complete set of taps and dies, in US Conventional and metric sizes, along with tap wrenches and die stocks. Note that tables matching die and tap sizes to drill sizes are embossed on the inside of the cover for ready reference.

Chapter 6

Hand Tools

LEARNING OBJECTIVES

After studying this chapter, you will be able to:

- ❍ Identify the most commonly used machine shop hand tools.
- ❍ Select the proper hand tool for the job.
- ❍ Maintain hand tools properly.
- ❍ Explain how to use hand tools safely.

IMPORTANT TERMS

abrasive
American National Thread System
blind hole
classes of fits
foot-pounds
newton-meters
number sizes
safe edges
torque
Unified System

Selecting and using hand tools correctly will help you do a job safely, with a minimum expenditure of time. When a hand tool is used *incorrectly,* it can be damaged; more importantly, you or someone else may be injured. It is to your advantage to learn to work properly with hand tools.

6.1 CLAMPING DEVICES

Clamping devices are employed to hold and/or position material while it is being worked on. Several types of clamping devices are used in machining.

6.1.1 Vises

The ***machinist's vise,*** or ***bench vise,*** is used for many holding tasks. It should be mounted on the bench edge far enough out to permit clamping long work in a vertical position. A vise may be a solid base type or may have a swivel base, which allows the vise to be rotated. See **Figure 6-1.**

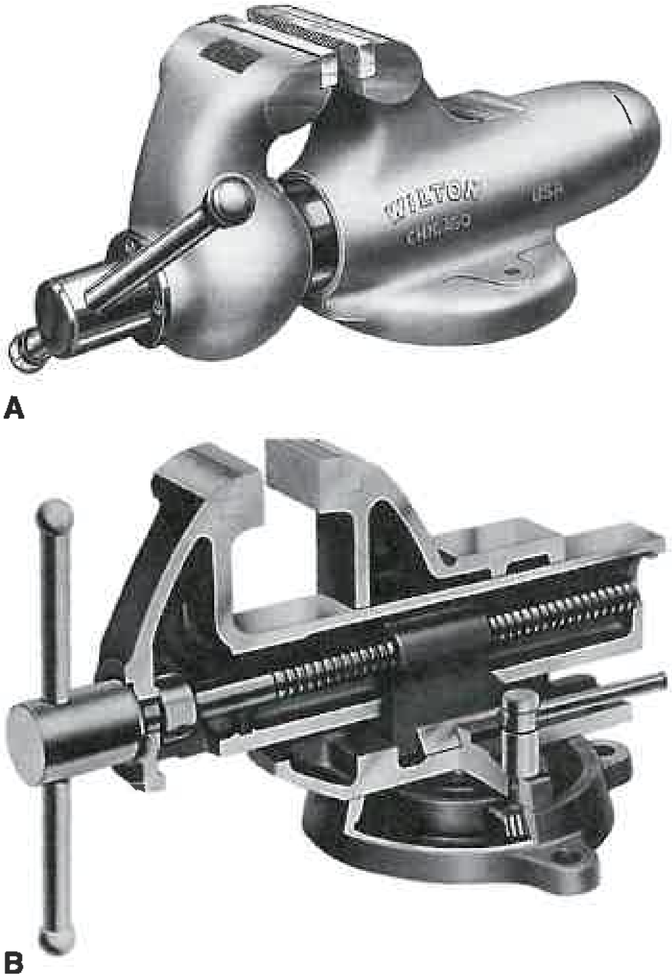


Figure 6-1. *Machinist's (bench) vise. A—Solid base type. (Wilton Tool Mfg., Inc.) B— Cutaway of swivel base vise. The base is made in two parts so that the body can be rotated to any desired position. (Columbian Vise and Mfg. Co.)*

Small precision parts may be held in a small bench vise or ***toolmaker's vise,*** **Figure 6-2.** This type vise can be rotated and tilted to any desired position. Vise *size* is determined by the width of the jaws, **Figure 6-3.**

Figure 6-2. A small vise used by a toolmaker. It can be rotated and pivoted to desired working position. (Wilton Tool Mfg., Inc.)

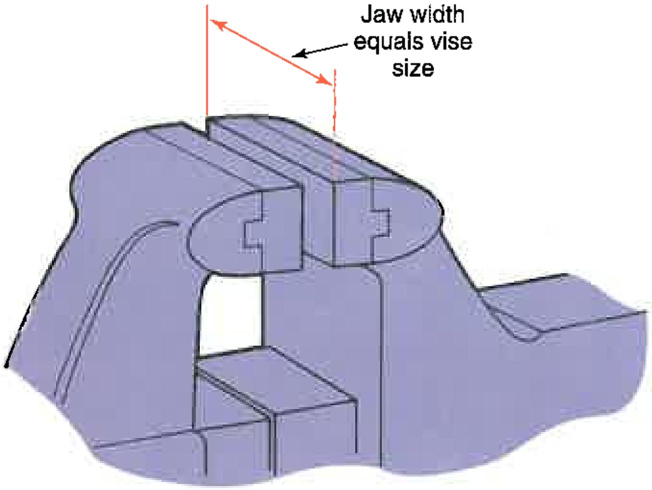


Figure 6-3. Vise size is determined by width of vise jaws.

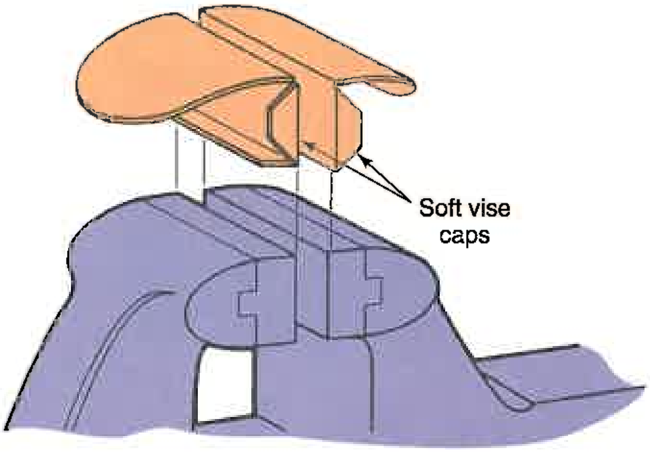


Figure 6-4. Caps made of copper, lead, or aluminum are slipped over hardened vise jaws to protect work from becoming marred or damaged by jaw serrations.

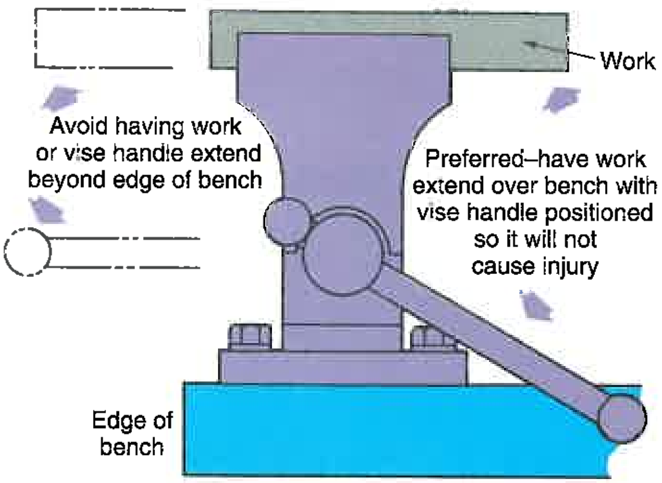


Figure 6-5. To prevent injury, avoid letting the vise handle or work project into the aisle.

A vise's clamping action is obtained from a heavy screw turned by a handle. The handle is long enough to apply ample pressure for any work that will fit the vise. Under *no circumstances* should the vise handle be hammered tight, nor should additional pressure be applied using a length of pipe on the handle for leverage.

Vise jaws are hardened. To clamp work that would be damaged or marred by the jaw serrations, the jaws should be covered with soft copper, brass, or aluminum caps, **Figure 6-4.**

When clamping a job in a vise, do not allow the vise handle or work to project into the aisle, Figure 6-5.

6.1.2 Clamps

The ***C-clamp*** and the ***parallel clamp*** hold parts together while they are worked on. The C-clamp, **Figure 6-6,** is made in many sizes. Jaw opening determines clamp size.

A parallel clamp is ideal for holding small work. For maximum clamping action, the jaw faces must be parallel. See **Figure 6-7.** Placing strips of paper the width of the clamp jaw between the work and the jaws will improve clamping action.

6.2 PLIERS

Combination pliers, also called ***slip-joint pliers,*** are widely used for holding tasks. See **Figure 6-8.**

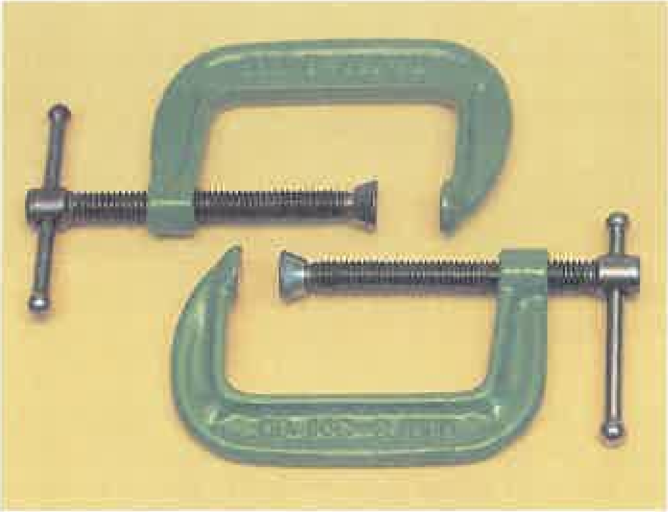


Figure 6-6. C-clamps are available in a range of sizes.

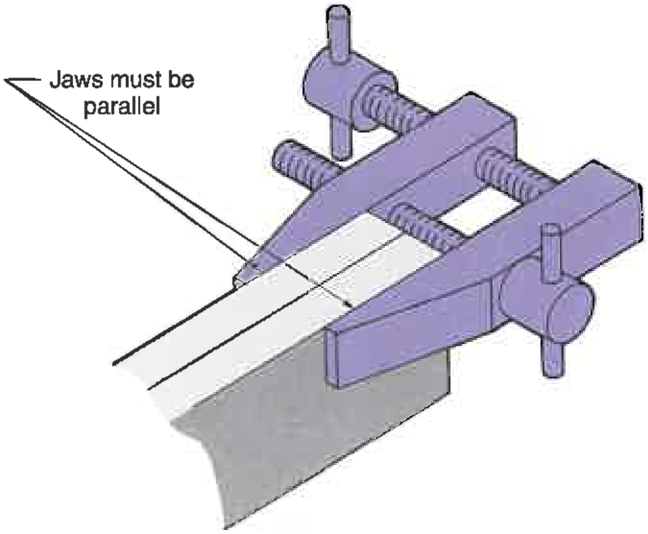


Figure 6-7. For maximum clamping action with a parallel clamp, adjust the jaws until they are parallel.

Figure 6-8. Combination or slip-joint pliers are used for holding tasks.

Figure 6-9. Diagonal pliers will cut flush with a surface.

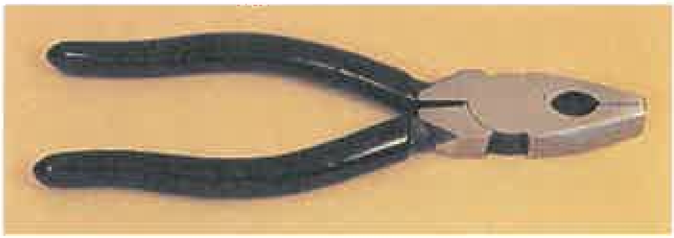


Figure 6-10. Side-cutting pliers have square jaws for holding and cutting tasks.

The slip-joint permits the pliers to be opened wider at the hinge pin to grip larger size work. They are made in 5", 6", 8", and 10" sizes. The *pliers size* indicates the overall length of the tool.

Some combination pliers are made with cutting edges for clipping wire and small metal sections to needed lengths. The better grade pliers are of forged construction.

Diagonal pliers are another widely used tool for light cutting tasks, **Figure 6-9**. The cutting edges are at an angle to permit the pliers to cut *flush* (even) with the work surface. Diagonal pliers are made in 4", 5", 6", and 7" lengths.

Side-cutting pliers are capable of cutting heavier wire and pins, **Figure 6-10.** Some of these pliers have a wire stripping groove and insulated handles. They are made in 6", 7", and 8" lengths.

Round-nose pliers, **Figure 6-11,** are helpful when forming wire and light metal. Their jaws are smooth and will not mar the metal being grasped. Round-nose pliers are available in 4", 4 1/2", 5", and 6" sizes.

Needle-nose pliers, are available in both straight and curved-nose types. They are handy for holding small work and when work space is limited. They will reach into cramped places. See **Figure 6-12.**

Tongue and groove pliers have aligned teeth for flexibility in gripping different size work, **Figure 6-13**. The size of the jaw opening can be adjusted easily. Tongue and groove pliers are made in many different sizes. The 6" size usually has five adjustments, while the larger 16" size has eleven adjustments.

Adjustable clamping pliers are a relatively new addition to the pliers family. On these pliers, the jaw

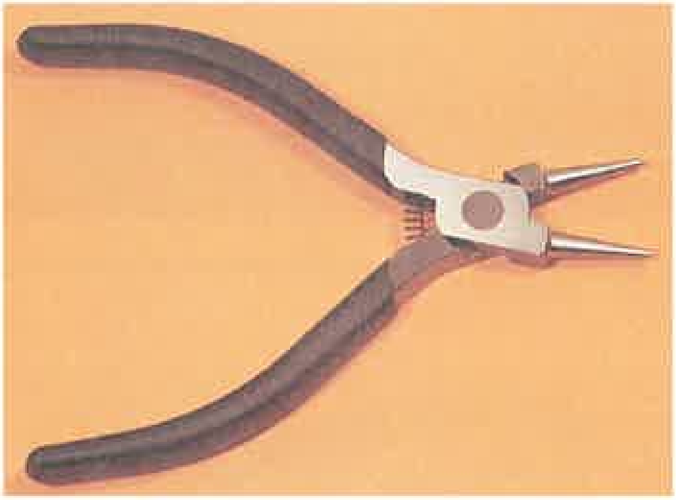


Figure 6-11. Round-nose pliers have smooth jaws.

Figure 6-12. Needle-nosed pliers. A—Straight pliers can be used to grasp smaller hard-to-reach objects. B—Curved pliers are helpful when working in areas with limited space.

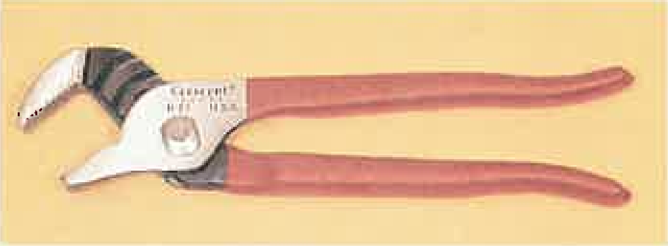


Figure 6-13. Tongue and groove pliers allow the jaws to expand to hold large objects.

Figure 6-14. Adjustable clamping pliers can be locked on work of different sizes.

opening can be adjusted through a range of sizes by using a threaded mechanism on one handle. See **Figure 6-14**. After adjustment, a squeeze of the hand can lock the jaws onto the work with more than a ton of pressure. Jaw pressure can be relieved by using the quick release on the handle. These pliers are made in many sizes with straight, curved, or long-nose jaws. They are known by several names, including locking pliers, Vise Grip® pliers, and Tag-L-Lock® pliers. The newest type of adjustable pliers, called Robo-Grip® pliers, permits one-handed jaw-size adjustment by merely squeezing the handles. See **Figure 6-15.** This type of adjustable pliers does not have a locking feature, however.

Figure 6-15. The newest type of adjustable pliers offers one-handed operation.

6.2.1 Care of Pliers

Like many tools, pliers will give long, useful service if a few simple precautions are taken:

- *Never* use pliers as a substitute for a wrench.
- Do not try to cut metal sizes that are too large, or work that has been heat-treated. Pliers with cutters will deform or break if used in this way. Breakage will also occur if additional leverage is applied to the handles.
- Occasionally clean and oil pliers to keep them in good working condition
- Store pliers in a clean, dry place. Avoid throwing them in a drawer or toolbox with other tools.
- Use pliers that are large enough for the job.

6.3 WRENCHES

Wrenches comprise a family of tools designed for use in assembling and disassembling many types of threaded fasteners. They are available in a vast number of types and sizes. Only the most commonly used wrenches will be covered.

6.3.1 Torque-Limiting Wrenches

Torque is the amount of turning or twisting force applied to a threaded fastener or part. It is measured in force units of ***foot-pounds*** (ft.-lbs.) or the SI Metric equivalent, ***newton-meters*** (N·m). Torque is the product of the force applied times the length of the lever arm. See **Figure 6-16.**

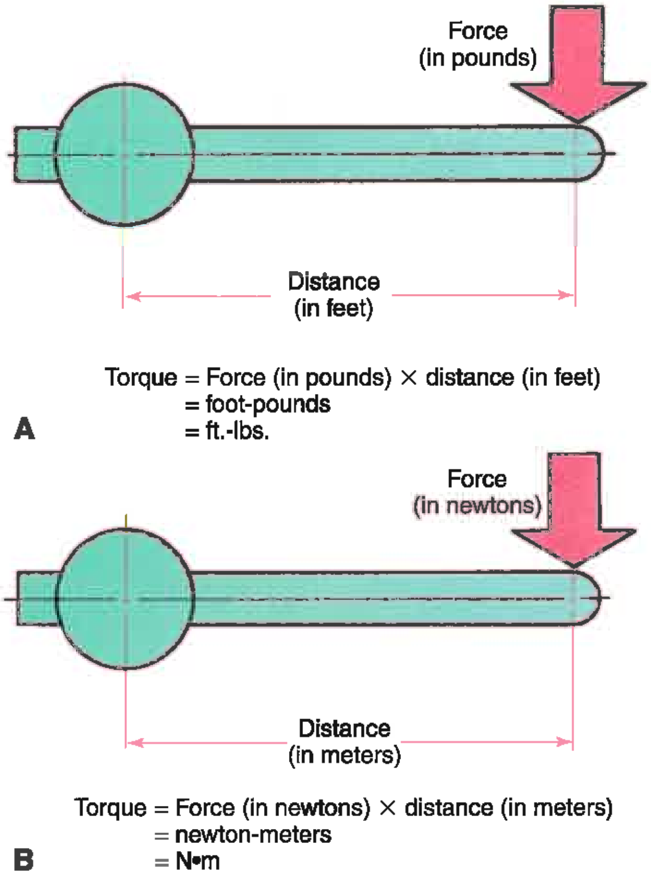


Figure 6-16. *Torque measurement. A—In the US Conventional system, torque is measured in foot-pounds. B—Torque values in SI Metric are given in newton-meters.*

A ***torque-limiting wrench*** allows you to measure the tightening of a threaded fastener in foot-pounds or newton-meters. This provides maximum holding power, without danger of the fastener or part failing or causing the work to warp or spring out of shape. See **Figure 6-17.**

Figure 6-17. *Torque-limiting wrenches are used when fasteners must be tightened to within certain limits to prevent undue stresses and strains from developing in the part.*

There are many types of torque-limiting wrenches, **Figure 6-18.** It is possible to obtain torque wrenches that are direct reading, or that feature a sensory signal (clicking sound or momentary release) when a preset torque is reached.

The right and wrong methods of gripping the wrench handle are shown in **Figure 6-19.** You should *never* lengthen the handle for additional leverage. These tools are designed to take a specific maximum force load. Any force over this amount will destroy the accuracy of the wrench.

Torque-limiting wrenches will provide accurate measurements whether they are pushed or pulled. However, to prevent hand injury, the preferred method is to *pull* on the wrench handle.

6.3.2 Adjustable Wrenches

The term "adjustable wrench" is a *misnomer* (a name not properly applied). Other wrenches, such as the "monkey wrench" and pipe wrench, are also adjustable. However, the wrench that is somewhat like an open-end wrench, but with an adjustable jaw, is commonly referred to as an ***adjustable wrench,*** **Figure 6-20.**

As the name implies, the wrench can be adjusted to fit a range of bolt-head and nut sizes. Although it is convenient at times, the adjustable wrench is *not* intended to take the place of open-end, box, and socket wrenches.

Three important points must be remembered when using the adjustable wrench:

- The wrench should be placed on the bolt head or nut so that the movable jaw *faces the direction* the fastener is to be rotated, **Figure 6-21.**

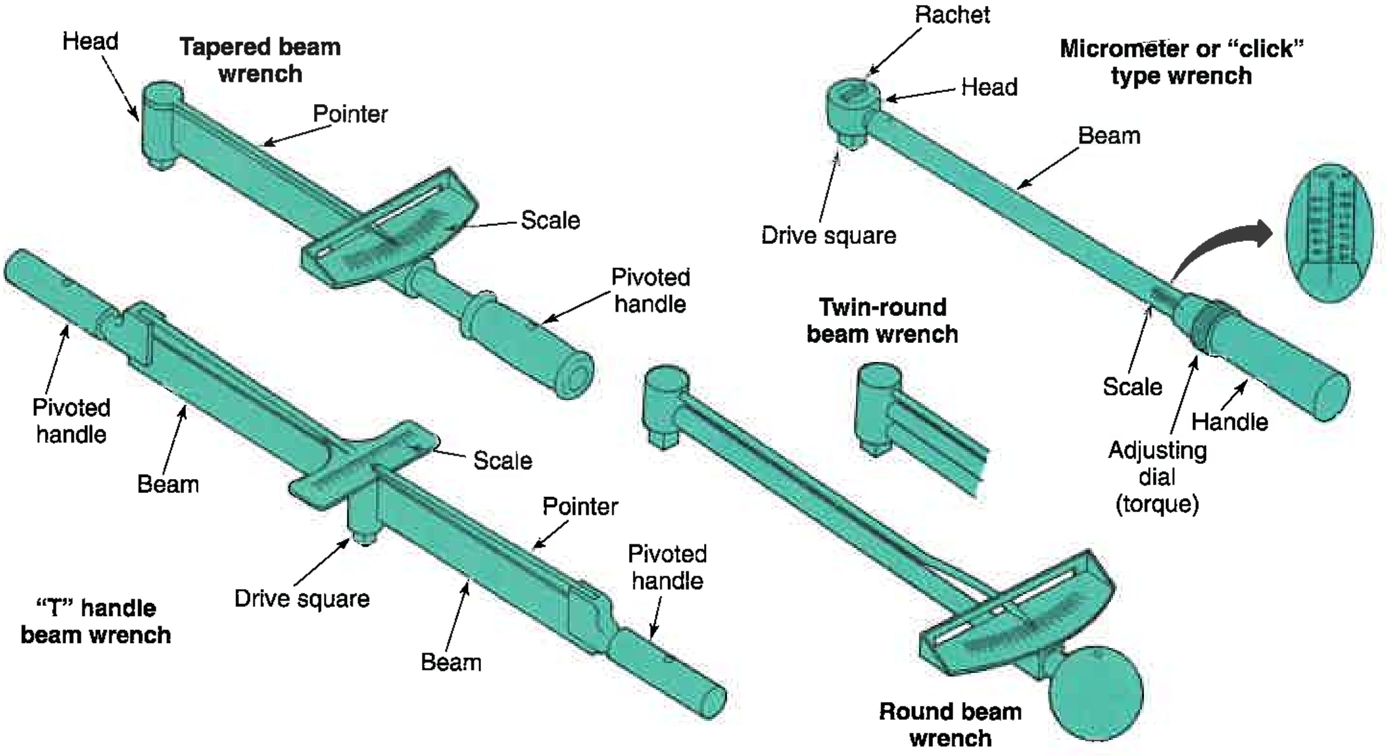


Figure 6-18. Several types of torque-limiting wrenches.

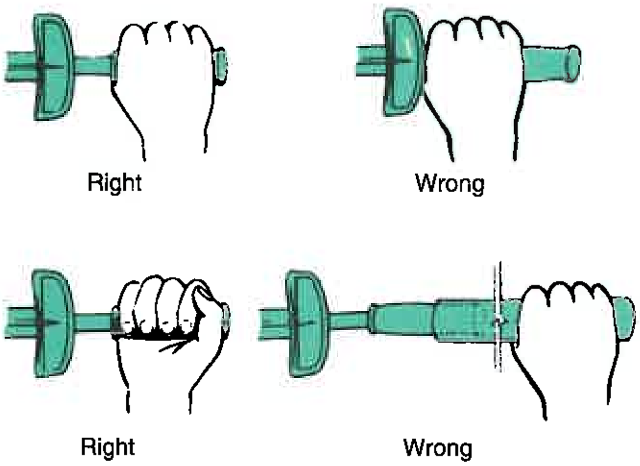


Figure 6-19. *The right and wrong ways to apply pressure to a torque-limiting wrench handle.*

Figure 6-20. *An adjustable wrench is handy when a full wrench set is not available.*

- Adjust the thumbscrew so the jaws fit the bolt head or nut *snugly*, **Figure 6-22.**
- *Do not* place an extension on the wrench handle for additional leverage. Never hammer on the handle to loosen a stubborn fastener. Use the smallest wrench that will fit the fastener on which you are working. This will minimize the possibility of twisting off the fastener.

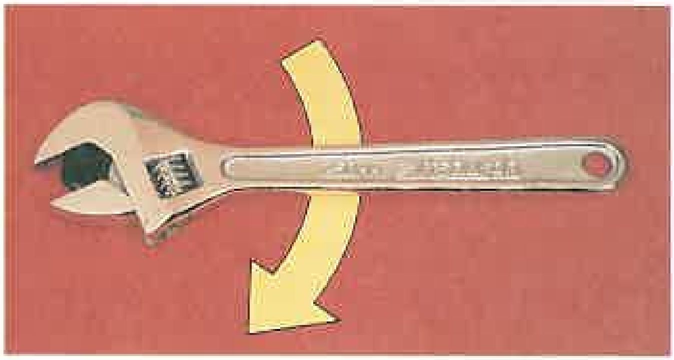


Figure 6-21. *The movable jaw of the wrench should always face the direction of rotation.*

Figure 6-22. A wrench must fit the nut or bolt snugly.

It is dangerous to *push* on, rather than *pull*, any wrench. If the fastener fails or loosens unexpectedly, you will almost always strike and injure your knuckles on the work. This operation is commonly known as "knuckle dusting."

The ***pipe wrench*** is designed to grip round stock, **Figure 6-23.** However, the jaws always leave marks on the work. Do not use a pipe wrench on bolt heads or nuts unless they cannot be turned with another type of wrench. For instance, you might need a pipe wrench to remove a bolt if the corners of its head have been rounded.

Figure 6-23. *The pipe wrench has jaws that will grasp round objects.*

6.3.3 Open-end Wrenches

Open-end wrenches are usually double ended, with two different size openings, **Figure 6-24.** They are made about 0.005" (0.13 mm) oversize to permit them to easily slip on bolt heads and nuts of the specified wrench size. Openings are at an angle to the wrench body, so that the wrench can be applied in close quarters. Standard and metric open-end wrenches are available. Because of the open end, they can be used only when applied torque is low.

6.3.4 Box Wrenches

The body or jaw of the ***box wrench*** completely surrounds the bolt head or nut, so it can be used when higher torque must be applied than is possible with an open-end wrench. See **Figure 6-25.** A properly fitted box wrench will not normally slip. It is preferred for many jobs. Box wrenches are available in the same sizes as open-end wrenches and with straight and offset handles.

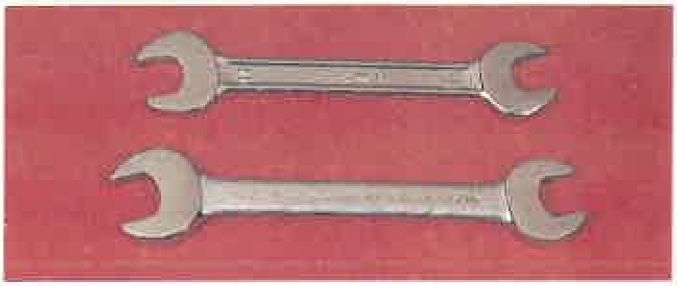


Figure 6-24. *An open-end wrench is acceptable when the torque applied is low.*

Figure 6-25. *A box wrench can handle more torque than an open-end wrench.*

6.3.5 Combination Open-end and Box Wrenches

A ***combination open-end and box wrench*** has an open-end wrench at one end of the handle and a box wrench at the other end. These wrenches are made in standard and metric sizes, **Figure 6-26.**

Figure 6-26. *Combination wrenches are handy because of two end configurations.*

6.3.6 Socket Wrenches

Socket wrenches are box-like and are made with a tool head-socket (opening) that fits many types of handles (either solid bar or ratchet type). A typical socket wrench set contains various handles and a wide range of socket sizes, **Figure 6-27.** Many sets include both standard and metric sockets. Various types of socket openings are shown in **Figure 6-28.**

Figure 6-27. *A typical socket wrench and sockets. The wrench has a right- and left-hand ratchet mechanism.*

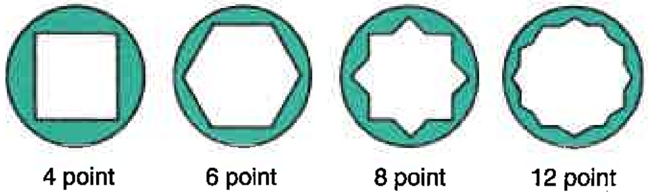


Figure 6-28. *Types of socket openings available. The 12-point socket can be used with both square and hex head fasteners.*

6.3.7 Spanner Wrenches

Spanner wrenches are special wrenches with drive lugs, and are designed to turn flush- and recessed-type threaded fittings. The fittings have slots or holes to receive the wrench end. They are usually furnished with machine tools and attachments. See **Figure 6-29.**

A ***hook spanner*** is equipped with a single lug that is placed in a slot or notch cut in the fitting. An ***end spanner*** has lugs on both faces of the wrench for better access to the fitting. The lugs fit notches or slots machined into the face of the fitting. On ***pin spanner wrenches,*** the lugs are replaced with pins that fit into holes on the fitting, rather than into notches.

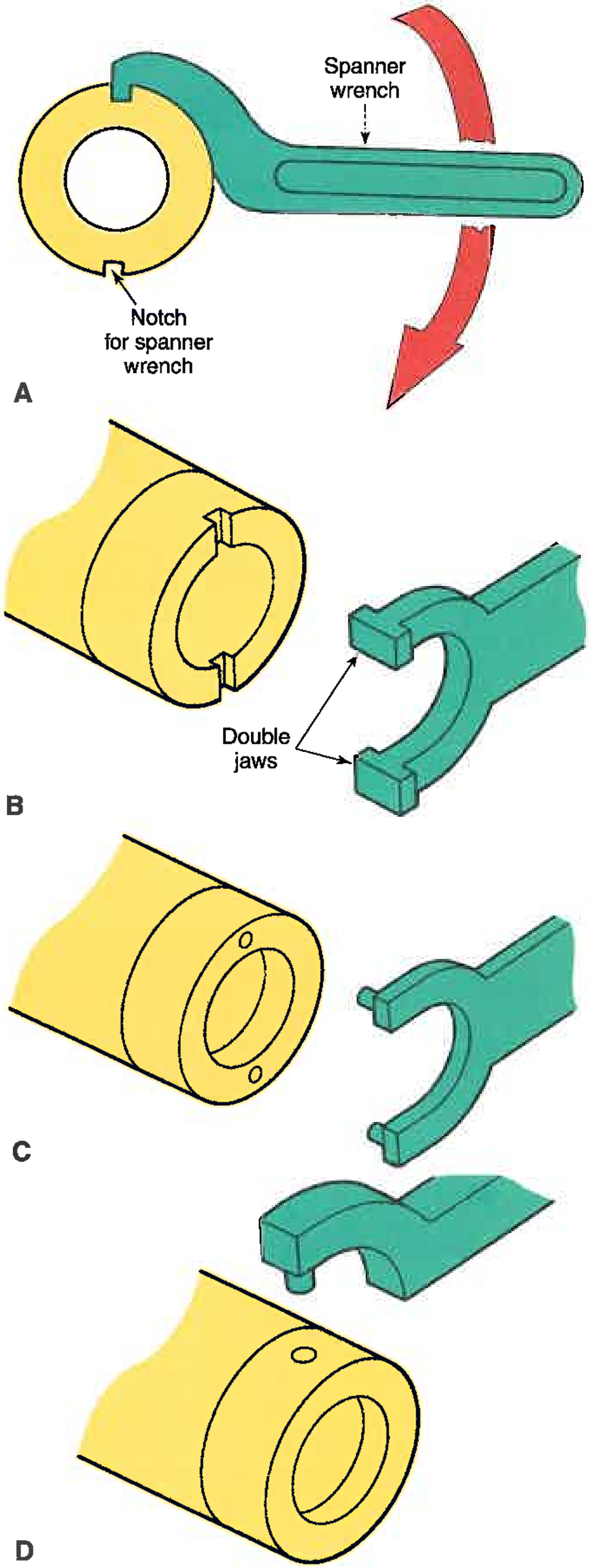


Figure 6-29. *Spanner wrenches. A—Hook-type spanner wrench. Some can be adjusted to fit different size fasteners. B—End spanner wrench. C and D—Two types of pin spanner wrenches.*

6.3.8 Allen Wrenches

The wrench that is used with socket-headed fasteners is commonly known as an ***Allen wrench,*** **Figure 6-30.** It is manufactured in many sizes to fit fasteners of various standard and metric dimensions.

Figure 6-30. Allen wrenches are used with socket-headed fasteners. They are made in both inch and metric sizes.

6.3.9 Wrench Safety

- Always *pull* on a wrench; never *push.* You have more control over the tool and there is less chance of injury.
- Select a wrench that fits properly. A loose-fitting wrench, or one with worn jaws, may slip and cause injury. It can also round off and ruin the bolt or nut on which it is being used.
- Never hammer on a wrench to loosen a stubborn fastener.
- Lengthening a wrench handle for additional leverage is a dangerous practice. Use a larger wrench.
- Before using a wrench, clean any grease or oil off the handle and the floor in the work area. This will reduce the possibility of your hands or feet slipping.
- Never try to use a wrench on moving machinery.

6.4 SCREWDRIVERS

Screwdrivers are manufactured with many different tip shapes, **Figure 6-31.** Each shape has been designed for a particular type of fastener. The standard and Phillips type screwdrivers are familiar to all shop workers. The other shapes may not be as well-known.

The ***Phillips screwdriver*** has an +-shaped tip for use with Phillips recessed head screws. Four sizes (#1, #2, #3, and #4) will handle the full range of this type fastener. They are manufactured in the same general styles as the standard screwdriver.

The ***Pozidriv® screwdriver*** tip is similar in appearance to the Phillips tip but has a slightly different shape. This type has been designed for Posidriv screws used extensively in the aircraft, automotive, electronic, and appliance industries.

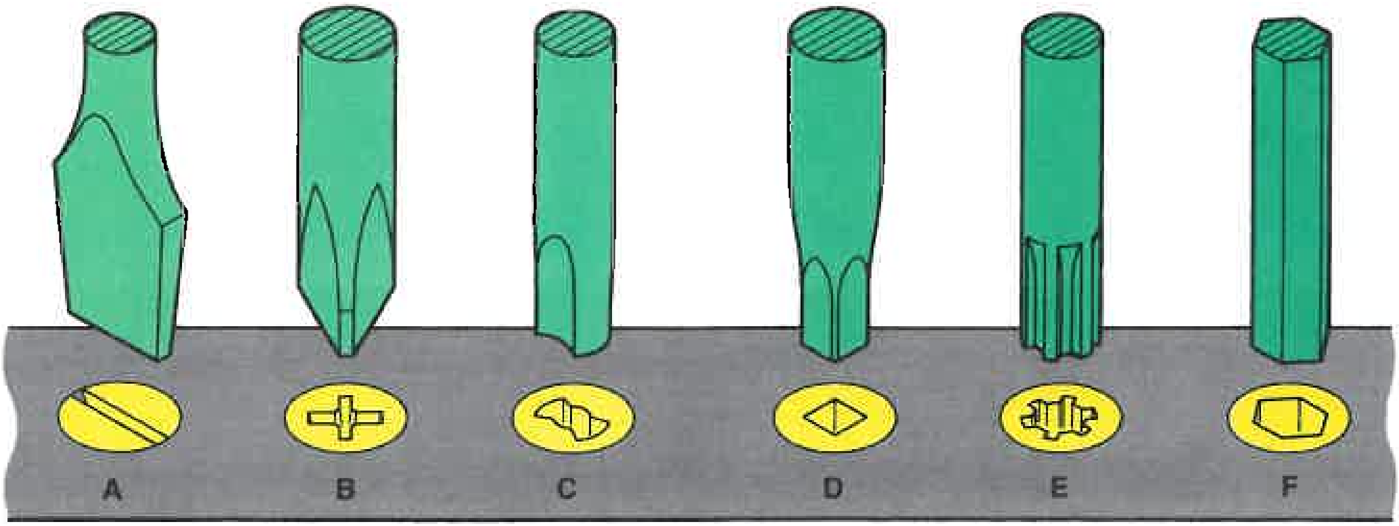


Figure 6-31. Types of screwdriver tips. A—Standard. B—Phillips. C—Clutch. D—Square. E—Torx. F—Hex.

The tip of this screwdriver has a black oxide finish to distinguish it from the Phillips tool. Using a Phillips tip will damage the opening in the head of the Pozidriv screw.

Clutch head, Robertson, Torx®, and ***hex screwdrivers*** are used for special industrial and security applications.

A ***standard screwdriver*** has a flattened wedge-shaped tip that fits into the slot in a screw head. This tool is made in 3" to 12" lengths. The shank diameter and the width and thickness of the tip are proportional with the length. Screwdriver *length* is measured from blade tip to the bottom of the handle. The blade is heat-treated to provide the necessary hardness and toughness to withstand the twisting pressures.

A few of the standard screwdriver types are shown in **Figure 6-32**. The ***double-end offset screwdriver*** can be used where there is not enough space for a conventional straight shank tool. The conventional ***straight shank screwdriver*** is widely used for a variety of work. The ***electrician's screwdriver*** has a long thin blade and an insulated handle.. The long thin blade will reach into tight areas. A ***heavy-duty screwdriver*** has a thick, square shank that permits a wrench to be applied for driving or removing large or stubborn screws. The ***stubby*** or ***close quarters screwdriver*** is designed for use where work space is limited. The ***ratchet screwdriver*** moves the screw on the power stroke, but not on the return stroke. It can be set for right-hand or left-hand operation.

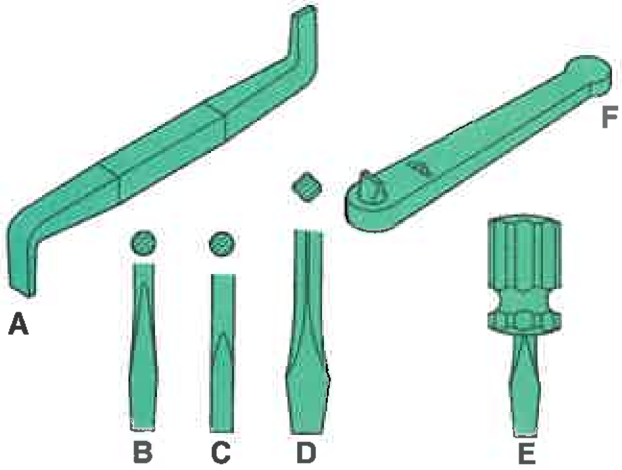


Figure 6-32*. Styles and types of standard screwdrivers. A—Double end offset. B—Conventional straight shank. C—Electrician's. D—Heavy-duty. E—Stubby or close quarter. F—Ratchet type offset.*

6.4.1 Using a Screwdriver

Always select the correct size screwdriver for the screw being driven, **Figure 6-33**. A poor fit will damage the screw slot and often will damage the tool's tip. Damaged screw heads are dangerous, and are often difficult to drive or remove. They should be replaced.

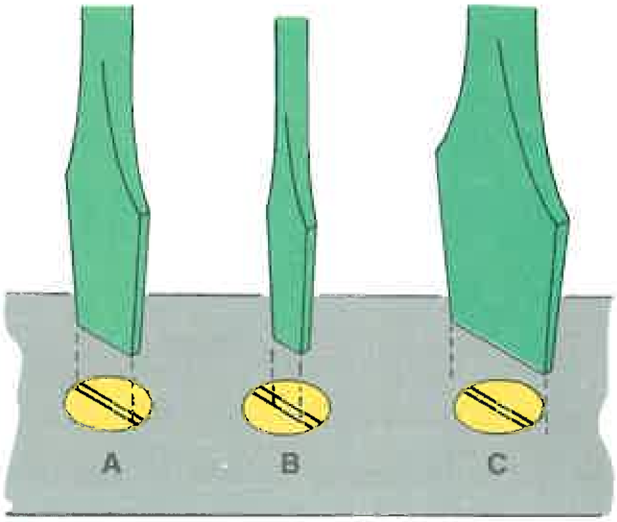


Figure 6-33*. Use the correct screwdriver tip for the job. Tip A is the correct width. Tip B is too narrow and will damage the screw head. Tip C is too wide and will damage the work.*

When driving or removing a screw, hold the screwdriver square with the fastener. Guide the tip with your free hand.

A worn screwdriver tip, such as the one shown at right in **Figure 6-34**, must be reground. A fine grinding wheel and light pressure is required. *Avoid overheating the tip during the grinding operation. It will destroy the tool.* Check the tip during the grinding operation by fitting it to a screw slot. A properly ground tip will fit snugly and hold the head firmly in the slot.

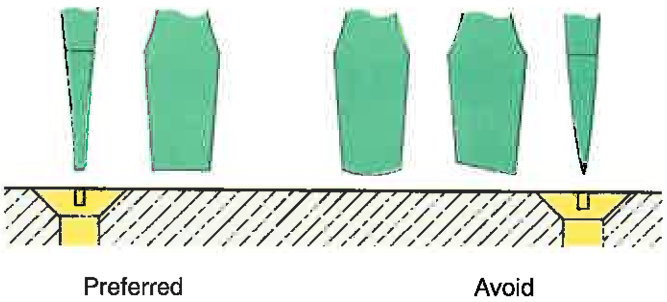


Figure 6-34. *Tips on the right are to be avoided. They are worn or improperly sharpened. The tip at left is ground correctly. Note that the sides are concave; this holds tip in slot when pressure is applied.*

6.4.2 Screwdriver Safety

- A screwdriver is not a substitute for a chisel, nor is it made to be hammered on or used as a pry bar.
- Wear safety goggles when regrinding screwdriver tips.
- Screws with burred heads are dangerous. They should be replaced or the burrs removed with a file or abrasive cloth.
- Always turn electric power off before working on electrical equipment. The screwdriver should have an insulated handle specifically designed for electrical work.
- Avoid carrying a screwdriver in your pocket. It is a dangerous practice that can cause injury to you or to someone else. It can also damage your clothing.

6.5 STRIKING TOOLS

The machinist's ball-peen hammer, **Figure 6-35,** is the most commonly used shop hammer. It has a hardened striking face and is used for all general purpose work that requires a hammer.

Figure 6-35. *The ball-peen hammer is most common type in a machine shop.*

Ball-peen hammer *sizes* are classified according to the weight of the head, without the handle. They are available in weights of 2, 4, 8, and 12 ounces, and 1, 1 1/2, 2, and 3 pounds.

A ***soft-face hammer*** or ***mallet*** permits heavy blows to be struck without damaging the part or surface. A steel face hammer would damage or mar the work surface. Soft-face hammers are especially useful for setting work tightly on parallels (steel bars) when mounting material in a vise.

Soft-face hammers are made of many different materials: copper, brass, lead, rawhide, and plastic. See **Figure 6-36.**

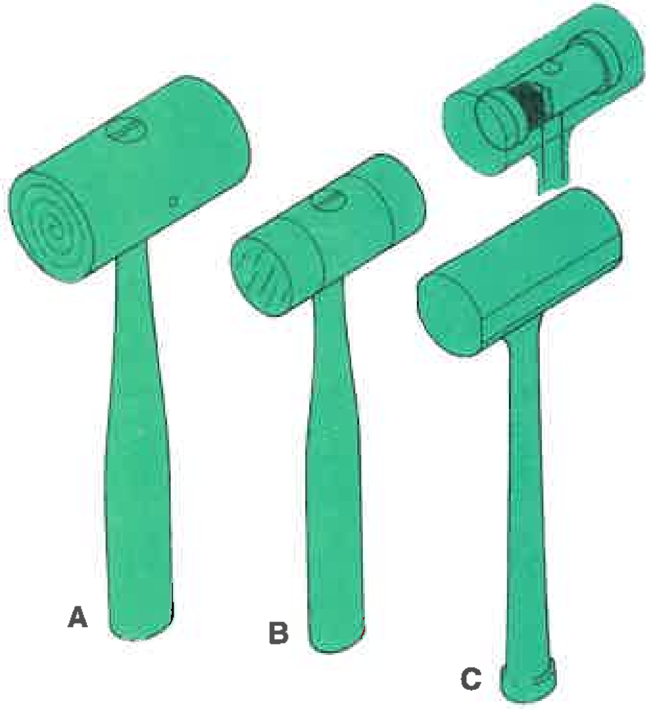


Figure 6-36. *Soft-face hammers and mallets. A—Rawhide mallet. B—Plastic-face hammer. C—Dead blow hammer. The head of the dead blow hammer contains tiny steel shot encased in plastic. This provides a hammer that has striking power but will not rebound (bounce back) as will other mallets and soft-face hammers.*

6.5.1 Striking Tool Safety

- Never strike two hammers together. The faces are very hard; the blow might cause a chip to break off and fly out at high speed.
- Do not use a hammer unless the head is on tightly and the handle is in good condition.
- Do not "choke up" too far on the handle when striking a blow, or you may injure your knuckles.
- Strike each blow squarely, or the hammer may glance off of the work and injure you or someone working nearby.
- Place a hammer on the bench carefully. A falling hammer can cause a painful foot injury, or damage precision tools on the bench.

6.6 CHISELS

Not all cutting in metalworking is done by machine. Chisels are one of several basic hand tools, such as hacksaws and files, that are considered cutting implements. These tools, when in good condition, sharp, and properly handled, are safe to use.

The ***chisel*** is used mostly to cut cold metal, hence the term "cold" chisel. The four chisels illustrated in **Figure 6-37** are the most common types. The general term ***cold chisel*** is used when referring to these chisels. Other chisels in this category are variations or combinations of these chisels.

The work to be cut will determine how the chisel should be sharpened, **Figure 6-38.** A chisel with a slightly curved cutting edge will work better when cutting on a flat plate. The curved edge will help prevent the chisel from cutting unwanted grooves in the surrounding metal, as when shearing rivet heads. If the chisel is to be used to shear metal held in a vise, the cutting edge should be straight.

The chisel is frequently employed to chip surplus metal from castings. Chipping is started by holding the chisel at an angle, as shown in **Figure 6-39**. The angle must be great enough to cause the cutting edge to enter the metal.

After the chisel cut has been started and the proper depth reached, the chisel angle can be decreased enough to keep the cutting action at the proper depth. Cut depth can be reduced by decreasing the chisel angle. However, if the cutting angle is decreased too much, the chisel will ride on the heel of the cutting edge and lift out of the cut.

When shearing metal in a vise, position it so the layout line is just below the vise jaws. This will leave sufficient metal to finish by filing or grinding. When cutting, it is usually best to hold the metal in a vise *without* using jaw caps. This provides a better shearing action between the vise jaws and chisel. Advance the chisel after each blow so the cutting is done by the center of the cutting edge.

A chisel is an ideal tool for removing rivets. The head can be sheared off and the rivet punched out. A variation of the conventional cold chisel for removing rivet heads is called a ***"rivet buster,"*** **Figure 6-40.**

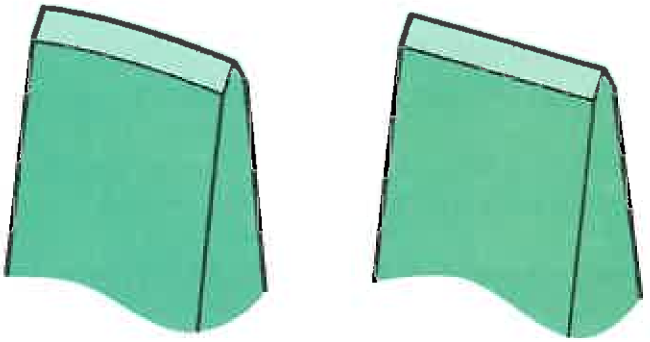


Figure 6-38. *The work to be done determines how a chisel should be sharpened. Left—Slightly rounded edge for cutting on flat plate. Right—Straight edge for shearing.*

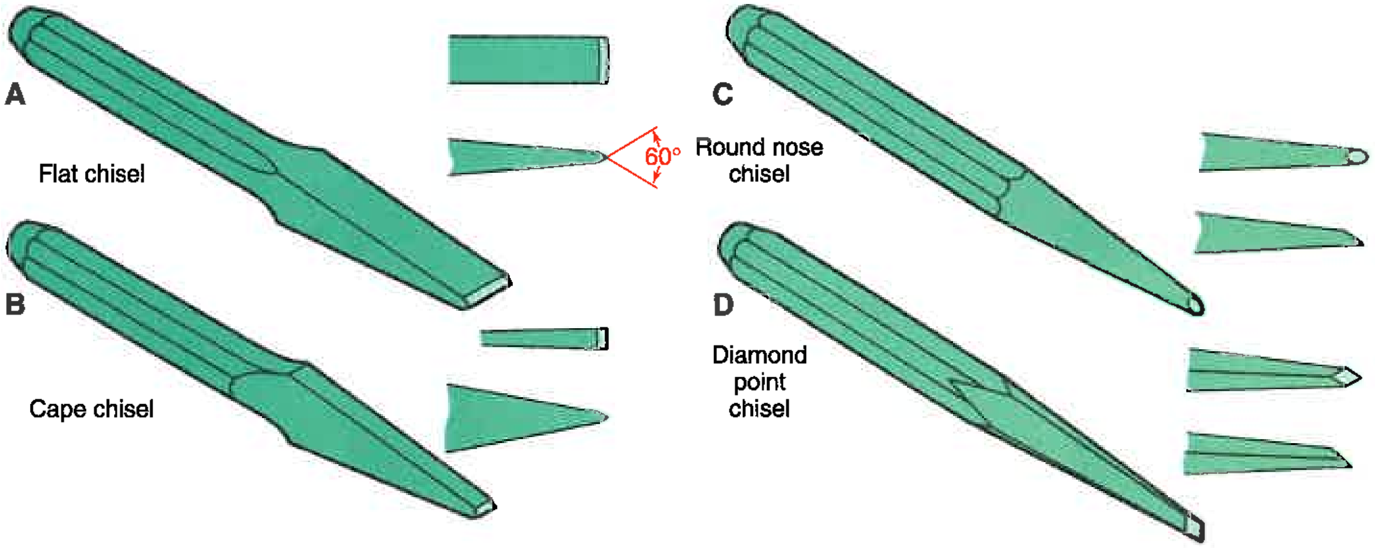


Figure 6-37. *Cold chisels. A—Flat chisel is used for general cutting and chipping work. B—Cape chisel has a narrower cutting edge than the flat chisel and is used to cut grooves. C—Round nose chisel can cut radii and round grooves. D—Diamond point chisel is principally used for squaring corners.*

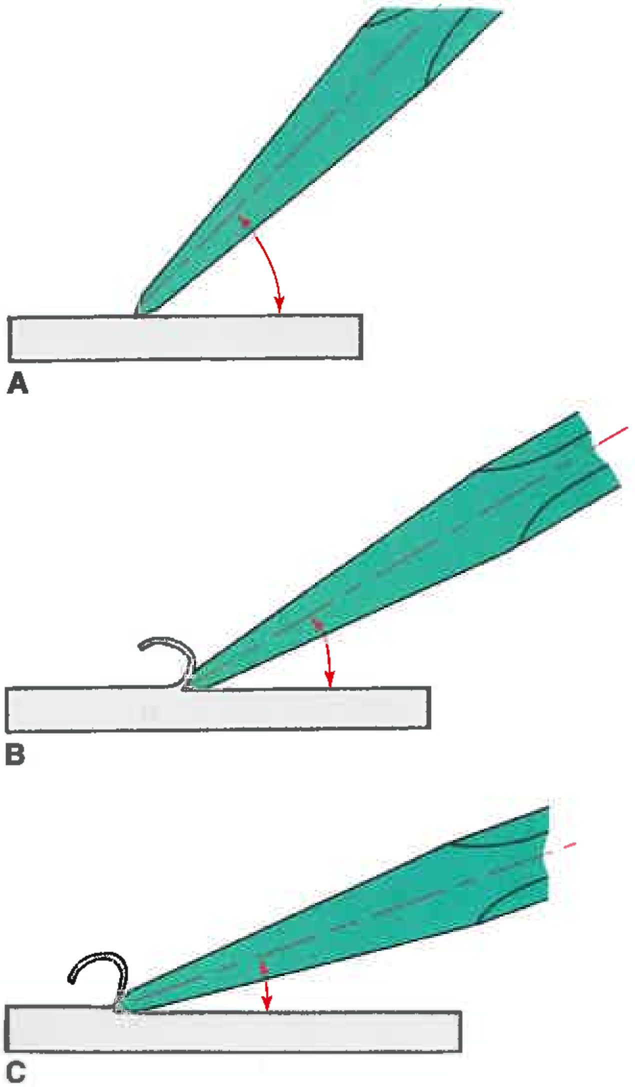


Figure 6-39. Proper chisel angles for various cutting operations. A—Starting the cut. B—Maintaining cut at desired depth. C—Reducing the cutting angle too much will cause chisel to lift out of cut.

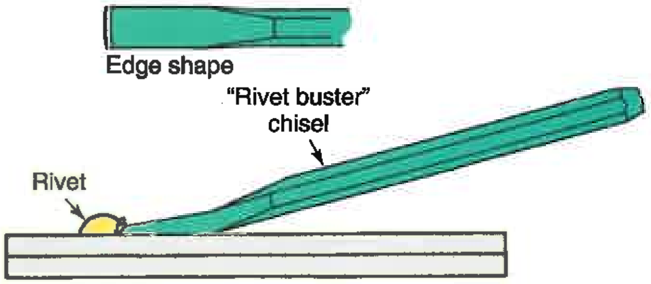


Figure 6-40. This variation of the flat chisel is often referred to as a "rivet buster." Upper drawing shows how it is sharpened.

When there is not enough room to swing a hammer with sufficient force to cut a rivet, an alternate procedure can be used, **Figure 6-41A.** Drill a hole, about the size of the rivet body, almost through the head. The head can then be removed easily with the chisel.

If the head is so large that it cannot be removed in one piece, make a saw cut almost through the head, then use the chisel to cut away half the head at a time. **Figure 6-41B** shows how this is done. Rivets also can be removed by drilling and using the narrow cape chisel, **Figure 6-41C.**

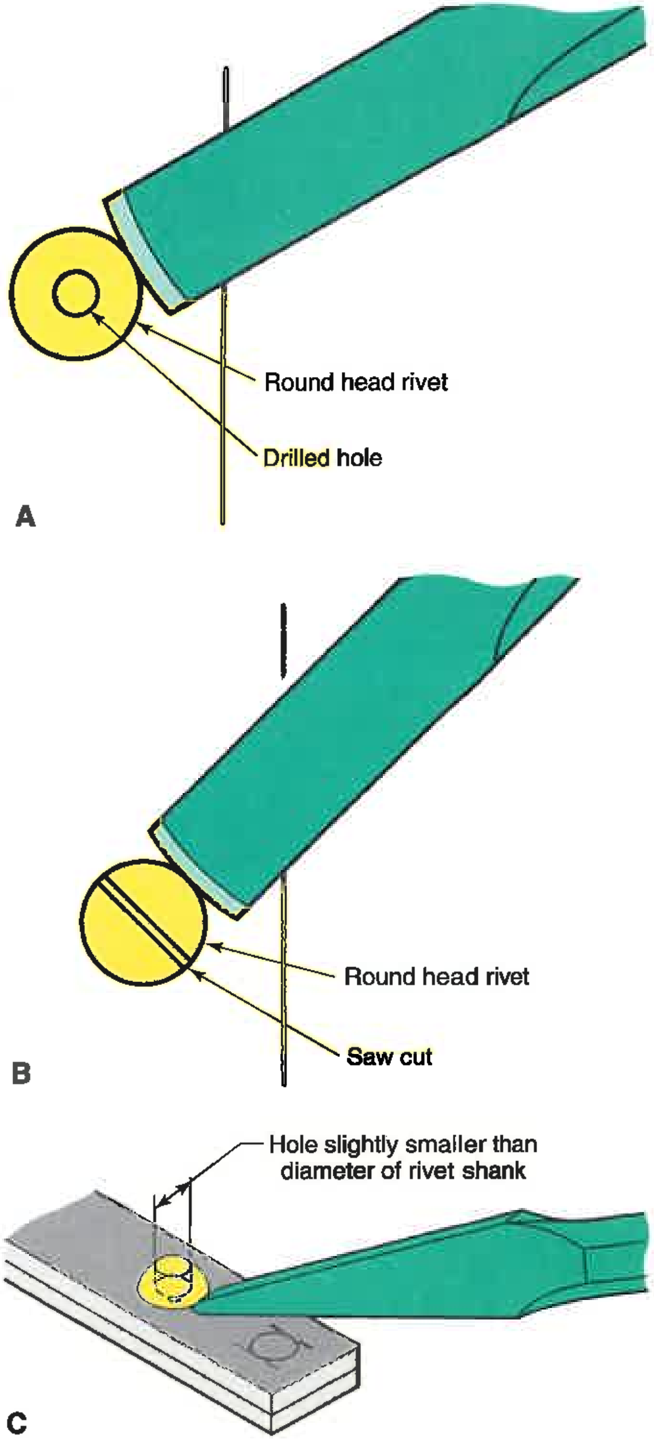


Figure 6-41. Alternate methods for removing rivet heads. A—When there is not enough room to swing hammer with sufficient force. B—When rivet head is too large to be removed as one piece. C—A cape chisel may also be used to remove rivets.

6.6.1 Chisel Safety

- Flying chips are dangerous! When cutting metal with a chisel, wear safety goggles and erect a shield around the work. These steps will protect you and people working nearby.
- Hold a chisel so that if you miss it with the hammer, you will not strike and injure your hand. If one is available, use a chisel holder.
- A **mushroomed chisel head** is extremely dangerous, since jagged metal can be knocked or chipped off and cause serious injury. Remove this hazardous condition by grinding away the excess metal. See **Figure 6-42.**
- Edges on metal cut with a chisel are sharp and can cause bad cuts. Remove them by grinding or filing.

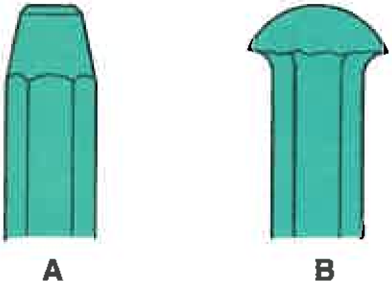


Figure 6-42. Chisel safety. A—Chisel head ground to a safe condition. B—A dangerous mushroomed head.

6.7 HACKSAW

The typical ***hacksaw*** is composed of a frame with a handle and a replaceable blade, **Figure 6-43.** Almost all hacksaws made today are adjustable to accommodate several different blade lengths. They are also made so the blade can be installed in either a vertical or horizontal position, **Figure 6-44.**

When placing a blade in the saw frame, make sure the frame is adjusted for the blade length being inserted. There should be sufficient adjustment remaining to permit tightening the blade until it "pings" when snapped with your finger. Frequently, a new blade must be retightened after a few strokes because it will stretch slightly from the heat produced while cutting.

Figure 6-43. A typical hacksaw.

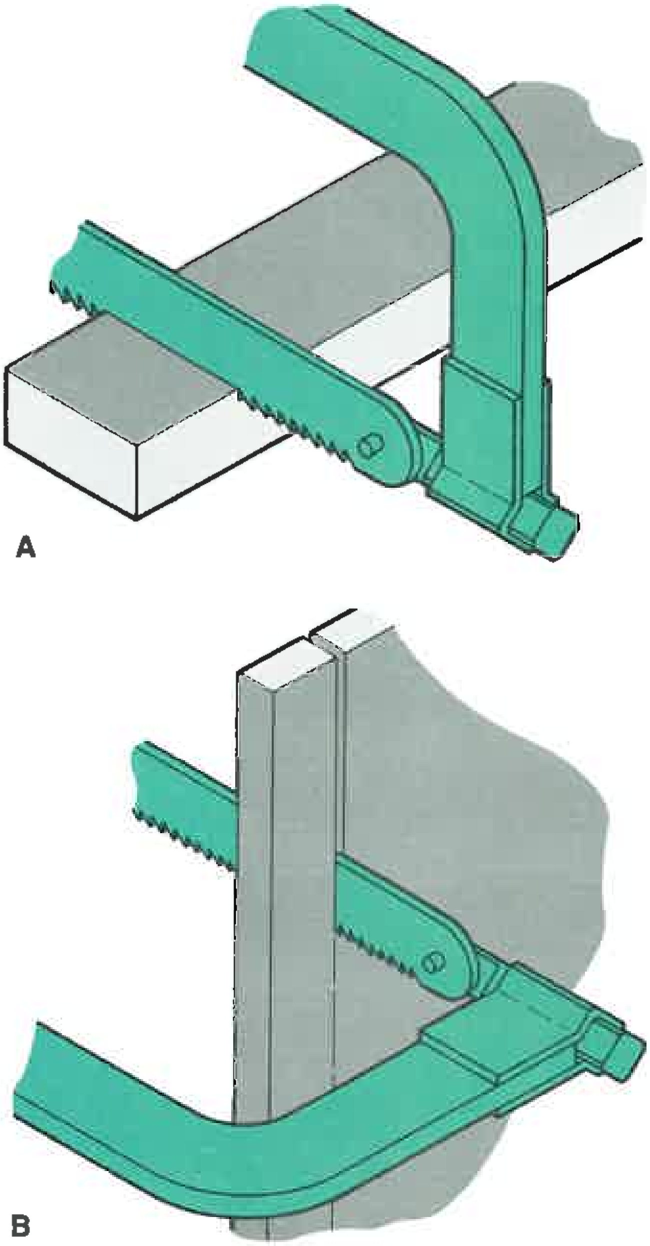


Figure 6-44. Blade positions. A—The blade is set to cut in a conventional vertical position. B—The blade has been pivoted 90º to cut a long narrow strip of stock.

The hacksaw blade must be positioned with the teeth pointing *away* from the handle, **Figure 6-45.** This will make it cut on the forward (push) stroke.

6.7.1 Holding Work for Sawing

The work must be held securely, with the point to be cut as close to the vise as practical. This helps to eliminates "chatter" and vibration that will dull the saw teeth.

Figure 6-45. A hacksaw blade must be inserted with the teeth pointing away from the handle. This positions it to cut on the forward stroke of the hacksaw.

Figure 6-46 shows some methods preferred for holding work that is irregular shape. The work is clamped so the cut is started on a flat side rather than on a corner or edge. This lessens the possibility of ruining the teeth or breaking the blade.

6.7.2 Starting a Cut

When starting a cut to a marked line, it is best to notch the work with a file, **Figure 6-47**. You can also use the thumb of your left hand to guide the blade until it starts the cut, **Figure 6-48.** Work carefully to avoid injury. Some hacksaw blades are manufactured with very fine teeth at the *front* to make starting a cut easier. Use enough pressure so the blade will begin to cut immediately.

6.7.3 Making the Cut

Grasp the hacksaw firmly by the handle and front of the frame. Apply enough pressure on the forward stroke to make the teeth cut. Insufficient pressure will permit the teeth to slide over the material, dulling the teeth. Also, lift the saw slightly on the return stroke.

Cut the *full length* of the blade and make about 40 to 50 strokes per minute. More strokes per minute may generate enough heat to draw the blade

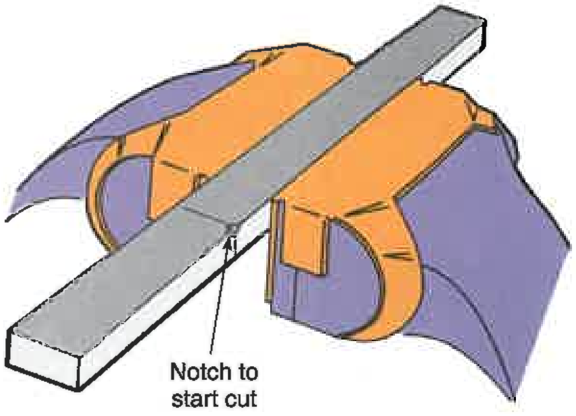


Figure 6-47. Using a file to notch or nick the edge of a piece to be cut permits easier starting of the hacksaw cut.

Figure 6-48. Using your thumb to guide the hacksaw blade as the cut is started.

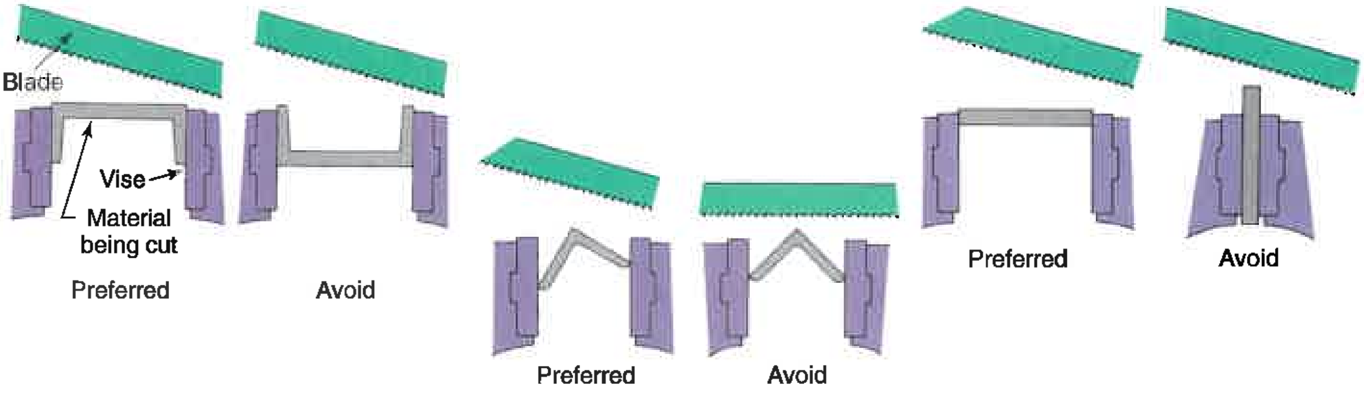


Figure 6-46. Preferred methods of holding irregular stock for sawing.

temper and dull the teeth. Keep the blade moving in a straight line. Avoid any twisting or binding, which can bend or break the blade.

Dulling or breaking a hacksaw blade. If you start a cut with an old blade and the blade breaks or dulls, do *not* continue in the same cut with a new blade. As a blade become dull, the ***kerf*** (slot made by blade) becomes narrower. If you try to continue the cut in the same slot, the new blade will usually bind and be ruined in the first few strokes. If possible, rotate the work and start a new cut on the other side.

6.7.4 Finishing a Cut

When the blade has cut almost through the material, saw carefully. Support the stock being cut off with your free hand to prevent it from dropping when the cut is completed.

6.7.5 Saw Blades

All hacksaw blades are heat-treated to provide the hardness and toughness needed to cut metal.

The shape and kind of material to be cut has an important bearing on blade choice, in terms of the number of teeth per inch, **Figure 6-49**.

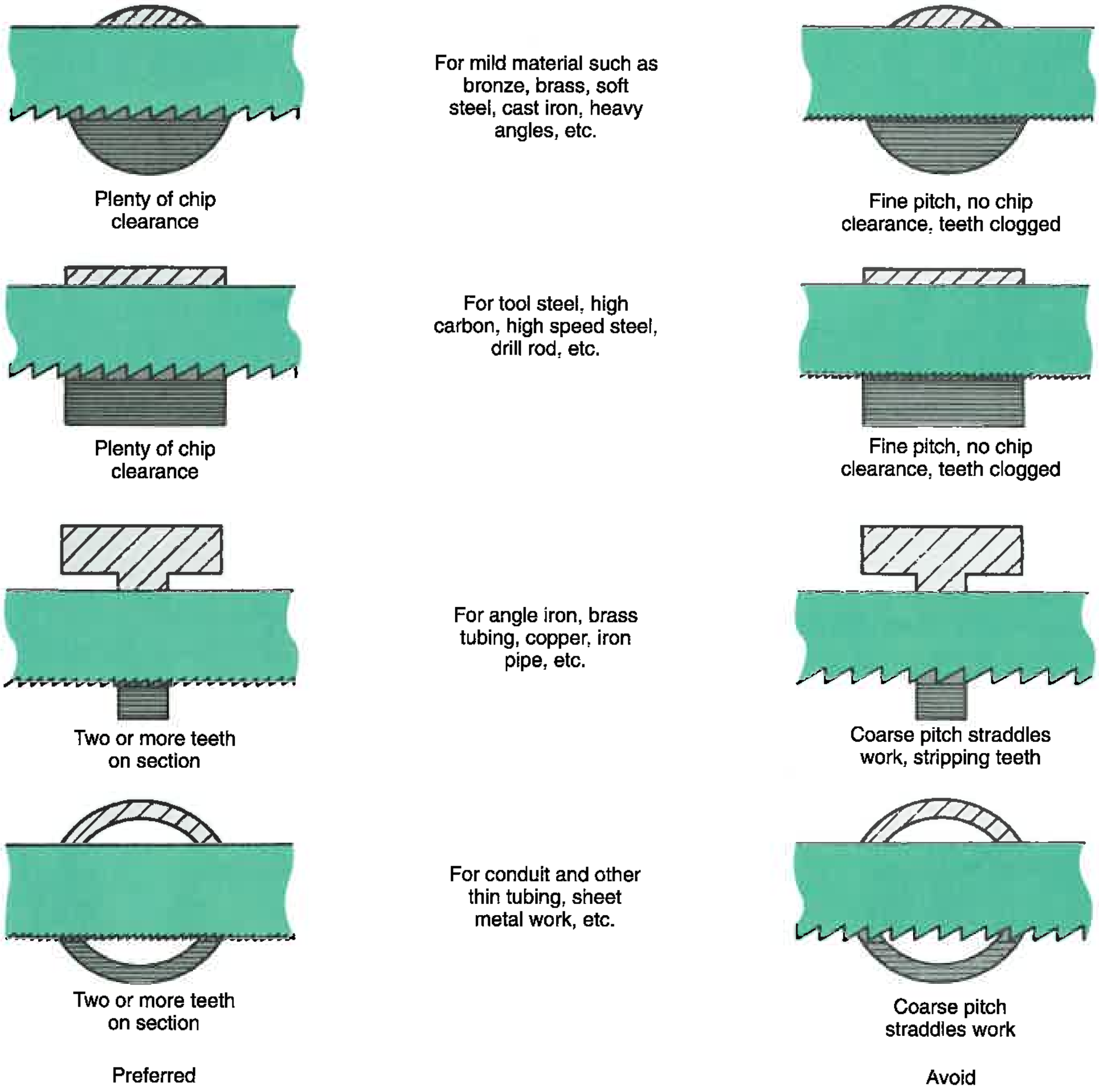


Figure 6–49. *The proper hacksaw blade should be used for each job to assure long blade life and rapid cutting action. Study recommendations.*

The ***flexible back blade*** has only the teeth hardened. The ***all-hard blade*** is hardened throughout. The hardness is reduced near the end holes, however, to reduce the possibility of breakage at these points.

Flexible back blades are best for sawing soft materials or materials with thin cross sections. An all-hard blade is best for cutting hard metals. It does not buckle when heavy pressure is applied.

Two or *three teeth* should be cutting at all times; otherwise, the teeth will straddle the section being cut and snap off when cutting pressure is applied.

The *set* of the blade provides the necessary clearance, and prevents the blade from binding in the cut. A blade may have one of three sets: ***undulating, raker,*** or ***alternate.*** These are shown in **Figure 6-50.**

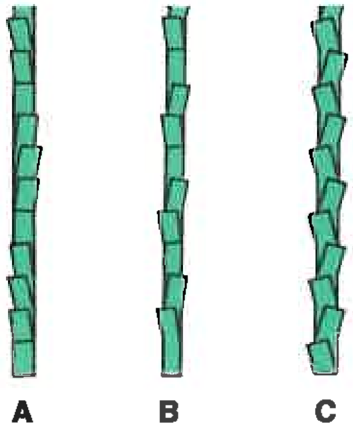


Figure 6-50. Types of sets in hacksaw teeth. A—Undulating. B—Raker. C—Alternate.

6.7.6 Unusual Cutting Situations

Cutting soft metal tubing can be a problem. The blade may bind and tear the tubing or the tubing may flatten. This can be eliminated by inserting a wood dowel of the proper size into the tubing. Then cut through both tubing and dowel. See **Figure 6-51.**

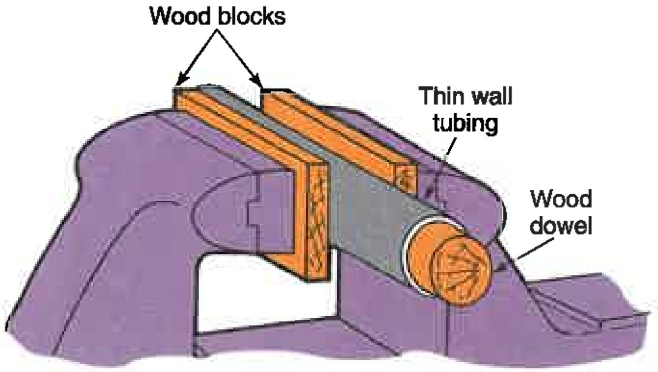


Figure 6-51. A snug-fitting dowel slid into thin-wall tubing will make cutting the tubing easier. If tubing is to be held in a vise for cutting, place soft wood blocks between the vise jaws and the work to prevent marring the exterior surface of the tubing.

Cutting a long narrow strip from thin metal can be done by setting the blade at right angles to the frame. Make the cut in the usual way, as in **Figure 6-52.** Strips of any width, up to the capacity of the saw frame, can be made in this manner.

Thin metal can be cut more easily and precisely by putting it between two pieces of wood, and cutting through both wood and metal, **Figure 6-53.**

6.7.7 Hacksaw Safety

- Never test the sharpness of a blade by running your fingers across its teeth.
- Store saws in a way that will prevent accidentally grasping the teeth when you pick up a saw.
- Burrs formed on the cut edge of metal are sharp and can cause a serious cut. Do not brush away chips with your hand; use a brush.
- Always wear safety goggles while using a hacksaw. All-hard blades can shatter and produce flying chips.
- Be sure the hacksaw blade is properly tensioned. If it should break while you are on the cutting stroke, your hand may strike the work, causing a painful injury.

6.8 FILES

A *file* is used for hand smoothing and shaping operations. The modern file is made from high-grade carbon steel and is heat-treated to provide the necessary hardness and toughness.

In manufacturing a file, the first production step is to cut the blank to approximate shape and size. See **Figure 6-54.** The tang and point are formed next. Then, the blank is annealed and straightened. The point and tang are trimmed after the sides and faces have been ground and the teeth cut. After another straightening, the file is heat-treated, cleaned, and oiled. Tests are made continually to assure a quality tool.

6.8.1 File Classifications

Files are classified by their shape. The shape is the general outline and cross section. The outline is either tapered or blunt, **Figure 6-55A.**

Files are also classified according to the cut of the teeth: ***single-cut, double-cut, rasp,*** and ***curved tooth,*** **Figure 6-55B,** and to the coarseness of the teeth: ***rough, coarse, bastard, second-cut, smooth,*** and ***dead smooth.***

6.8.2 File Care

A file should *never* be used without a handle. It is too easy to drive the unprotected tang into your hand.

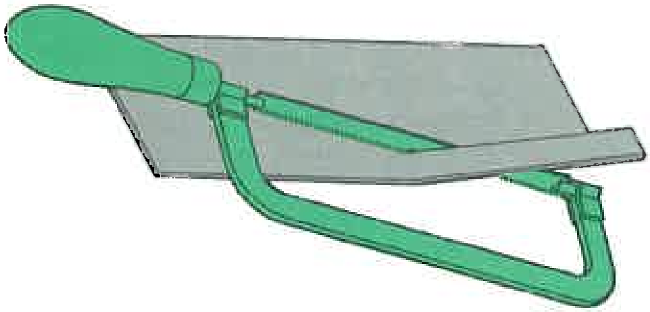


***Figure 6-52**. The hacksaw blade can be pivoted to a horizontal position for cutting long narrow strips. Best results can be obtained if strip is bent up slightly, as shown, during the sawing operation.*

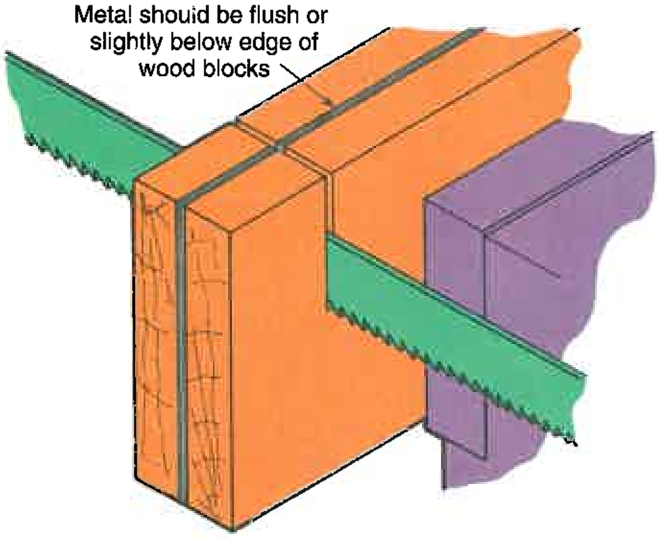


***Figure 6-53**. Sandwiching thin metal between two pieces of wood will make sawing easier and more precise.*

***Figure 6-54**. Steps in manufacturing a file. A—Steel bar cut to correct length. B—Bar forged to shape. C—Blank after it has been annealed. D—Annealed blank straightened and ground smooth. E—Teeth cut on blank. F—Blank trimmed and coated for heat treatment. G—Completed file cleaned and inspected. (Nicholson File Co.)*

Fit a handle to the file by drilling a hole in the handle. The hole should equal in diameter the width of the tang at its midpoint, **Figure 6-56**. Mate the file and handle by inserting the tang in the hole, then sharply striking the handle on a solid surface.

Store files so that they are always separated, **Figure 6-57;** *never* throw files together in a drawer or store them in a damp place.

Clean files frequently with a ***file card*** or ***brush,*** **Figure 6-58.** Some soft metals cause ***pinning,*** a condition in which the teeth become loaded or clogged with material the file has removed. Pinning will cause gouging and scratching of the work surface. The particles can be removed from the file with a pick or scorer. A file card combines the card, brush, and pick for file cleaning.

6.8.3 Selecting a File

The number of different kinds, shapes, and cuts of files that are manufactured is almost unlimited, **Figure 6-59**. For this reason, only the general classification of files will be covered.

Files have three distinct characteristics: *length,* kind or *type,* and *cut.* The file length is always measured from the heel to the point, **Figure 6-60**. The tang is not included in the measurement.

The file type refers to its *shape,* such as flat, mill, half-round, or square. The file cut indicates the relative *coarseness* of the teeth.

Single-cut files are usually used to produce a smooth surface finish. They require only light pressure to cut.

Double-cut files remove metal much faster than single-cut files. They require heavier pressure and they produce a rougher surface finish.

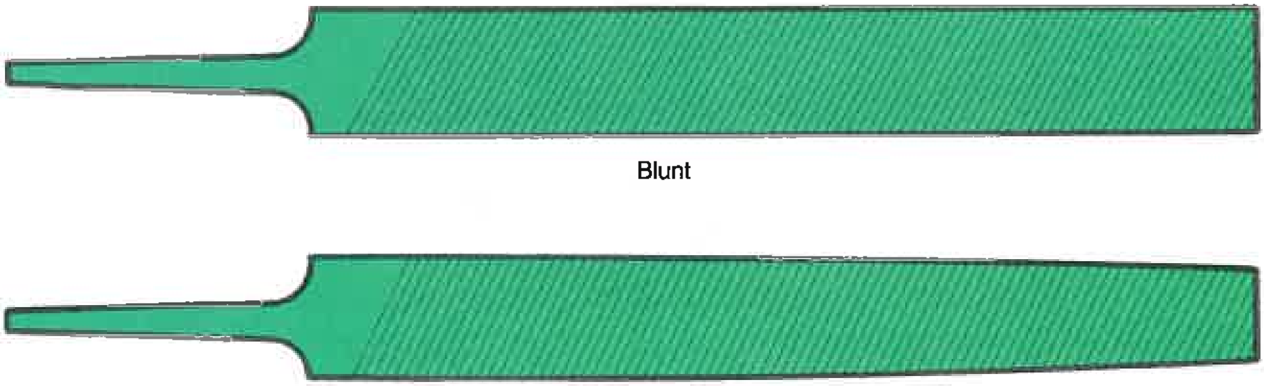


Figure 6-55. *File classifications. A—Blunt and tapered files. B—Single-cut, double-cut, rasp, and curved-tooth files.*

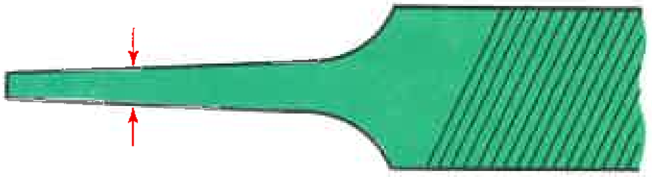


Figure 6-56. *File handle hole should be equal in diameter to width of file tang at the point indicated.*

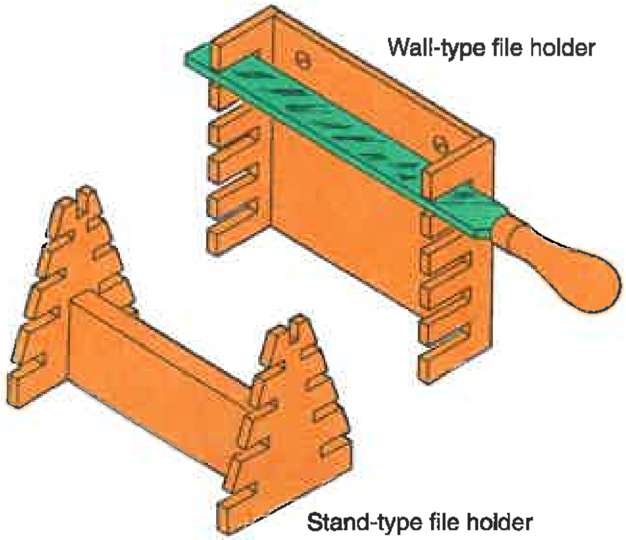


Figure 6-57. *Storing files properly, in holders like these, will greatly extend their useful life.*

Figure 6-58. *Using file card to clean a file.*

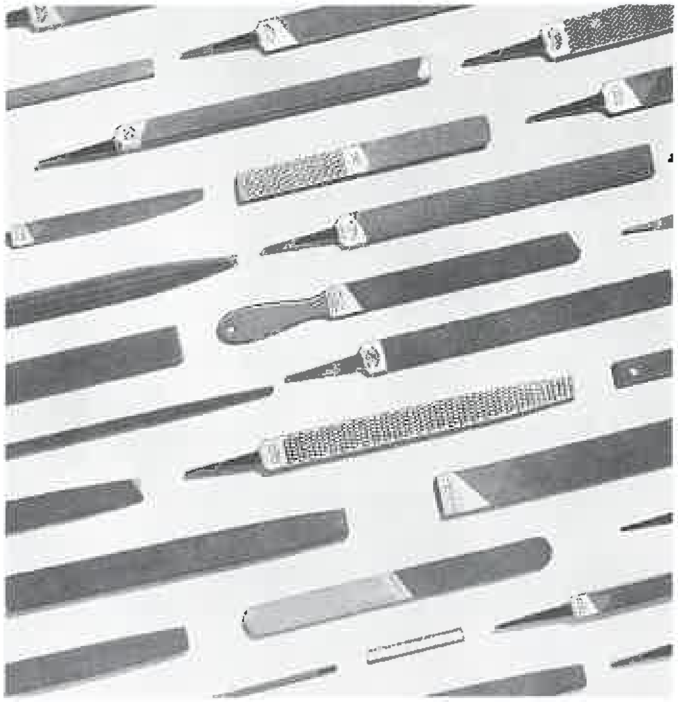


Figure 6-59 *A few of the many hundred different kinds of files. (Nicholson File Co.)*

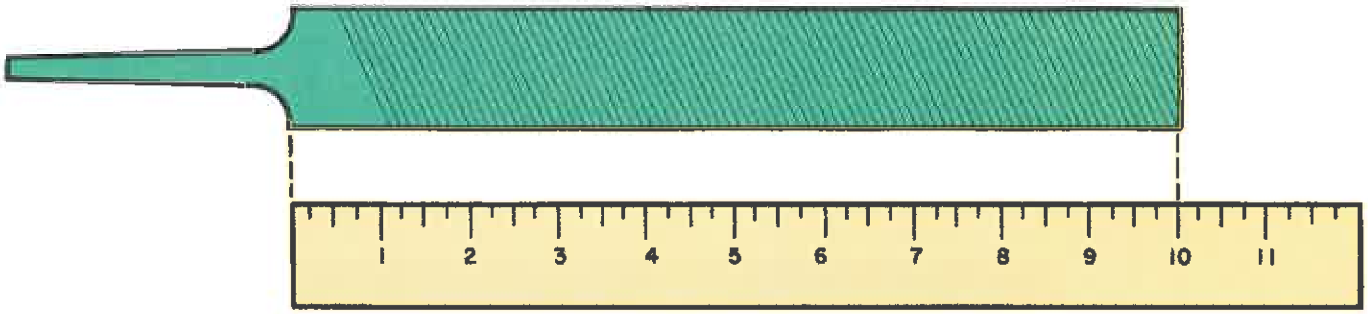


Figure 6-60. How a file is measured.(Nicholson File Co.)

Rasps are best for working wood or other soft materials where a large amount of stock must be removed in a hurry.

A ***curved-tooth file*** is used to file flat surfaces of aluminum and sheet steel.

Some files have ***safe edges***. This denotes that the file has one or both edges without teeth, **Figure 6-61.** This permits filing corners without danger to the portion of the work that is not to be filed.

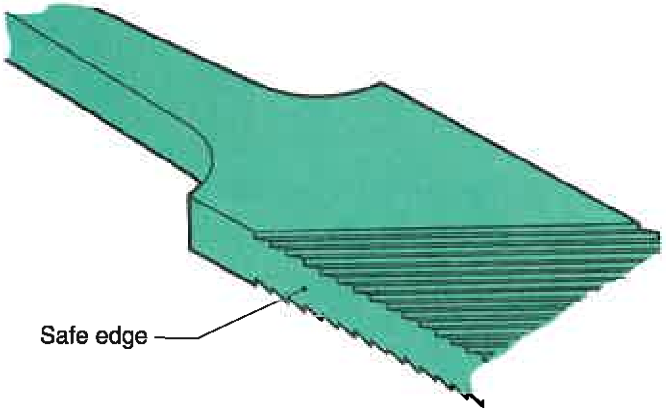


Figure 6-61. The safe edge of a file does not have teeth.

In selecting the file, many factors must be considered if maximum cutting efficiency is to be attained:

- The nature of the work (flat, concave, convex, notched, etc.).
- Kind of material.
- Amount of material to be removed.
- Surface finish and accuracy demanded.

Of the many file shapes available, the most commonly used are flat, pillar, square, 3-square, knife, half-round, crossing, and round, **Figure 6-62**. Each shape is available in many sizes and degrees of coarseness: rough, coarse, bastard, second-cut, smooth, and dead smooth, **Figure 6-63**. Note that a small (4″) rough cut file may be as fine as a large (16″) second-cut file.

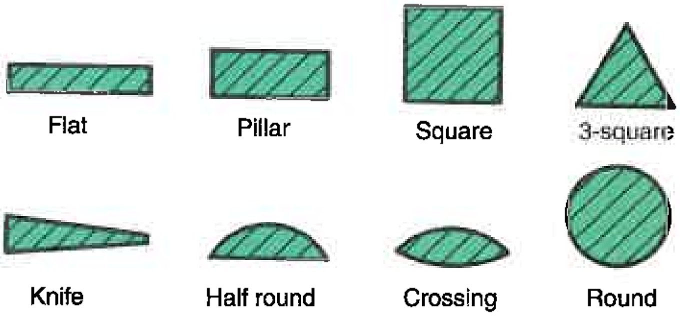


Figure 6-62. Cross-sectional views of the most widely used file shapes.

6.8.4 Types of Files

The wide variety of files can be divided into five general groups:

The ***machinists' file*** is used whenever metal must be removed rapidly, and the finish is of secondary importance. It is made in a large range of shapes and sizes, and is double-cut.

The ***mill file*** is a single-cut and tapers for the last third of its length away from the tang. It is suitable for general filing when a smooth finish is required.

Figure 6-63. Range in coarseness of a typical machinist flat bastard file. File sizes range from 4″ (100 mm) to 16″ (400 mm). (Nicholson File Co.)

A mill file works well for draw filing, lathe work, and working on brass and bronze.

Swiss pattern and ***jewelers' files*** are manufactured in more than a hundred different shapes. They are used primarily by tool-and-die makers, jewelers, and others who do precision filing.

The ***rasp*** has teeth that are individually formed and disconnected from each other. It is used for relatively soft materials (plastic for example) when large quantities of the material must be removed.

The ***special purpose files*** group includes those specifically designed to cut one type of metal or for one kind of operation. An example is the long-angle lathe file, **Figure 6-64**, which does an efficient filing job on the lathe.

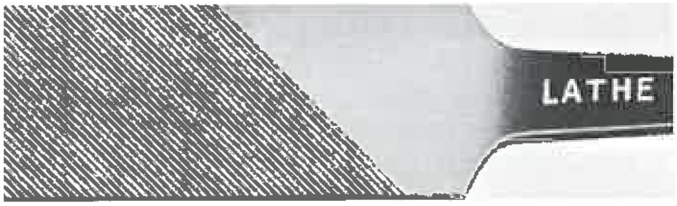


***Figure 6-64**. The long-angle lathe file is an example of a special-purpose file.*

6.8.5 Using a File

Efficient filing requires that the work be held solidly. Where practical, hold the work at about elbow height for general filing, **Figure 6-65**. If large quantities of metal must be removed by heavy filing, mount the work slightly lower.

Straight or ***cross filing*** consists of pushing the file lengthwise across the work, either straight ahead or at a slight angle. Grasp the file as shown in **Figure 6-66A**. Heavy-duty filing requires heavy pressure and can best be done if the file is held as in **Figure 6-66B.**

***Figure 6-65**. Mount work at elbow height for general filing.*

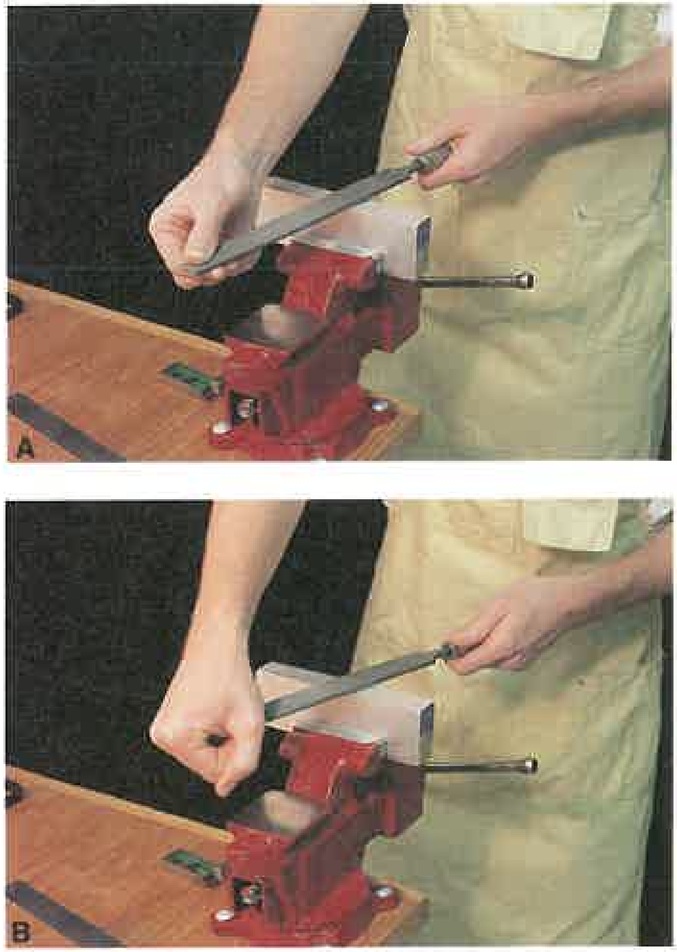


***Figure 6-66**. Holding a file. A—The proper way to hold a file for straight or cross filing. B—Holding method used to apply the additional pressure required when a considerable quantity of metal must be removed.*

A file can be ruined by using either *too much* pressure or *too little* pressure on the cutting stroke. Apply just enough pressure to permit the file to cut on the entire forward stroke. Too little pressure allows the file to slide over the work. This will dull the file. Too much pressure "overloads" the file, causing the teeth to clog and chip.

Lift the file from the work on the reverse stroke, except when filing soft metal. The pressure on the return stroke when filing soft metal should be no more than the weight of the file itself

Draw filing, when properly done, will produce a finer finish than straight filing. Hold the file as shown in **Figure 6-67**. Do not use a *short-angle* file for draw filing The short-angle file can cause scoring or scratching, instead of the desired shaving and shearing action, as the file is pushed and pulled across the work. Use a double-cut file to "rough down" the surface, then a single-cut file to produce the final finish.

Figure 6-67. Draw filing, when done properly, improves the surface finish.

6.8.6 File Safety

- Never use a file without a handle. Painful injuries may result!
- Clean files with a file card, not your hand. The chips can penetrate your skin and result in a painful infection.
- Do not clean a file by slapping it on the bench, since it may shatter.
- Files are very brittle. Never use one for prying tasks.
- Use a piece of cloth, not your bare hand, to clean the surface being filed. Sharp burrs are formed in filing and can cause serious cuts.
- Never hammer on or with a file. It can shatter, causing chips to fly in all directions.

6.9 REAMERS

A drill does not produce a smooth or accurate enough hole for a precision fit. *Reaming* is the operation that will produce smoothness and accuracy. Ordinarily, ***hand reaming*** is used only for final sizing of a hole.

6.9.1 Hand Reamer

A ***hand reamer*** has a square shank end so that it can be held in a tap wrench. See **Figure 6-68A.** Reamers may be made of high-speed steel or carbon steel, and are available in sizes from 1/8″ to 1 1/2″ (3.175 mm to 38.1 mm). The cutting end is ground with a slight taper to provide easy starting in the hole.

Straight-fluted reamers are suitable for most work. However, when reaming a hole with a keyway or other interruption, it is better to have a ***spiral-fluted reamer.***

When preparing a piece to be reamed by hand, 0.005″ to 0.010″ (0.15 mm to 0.25 mm) of stock should be left in the hole for removal by the reaming tool.

An ***expansion hand reamer*** is used when a hole must be cut a few thousandths inch over nominal size for fitting purposes, **Figure 6-68B**. Slots are cut into the center of the tool. The center opening is machined on a slight taper. The reamer is expanded by tightening a taper screw into this opening. The amount of expansion is limited; the reamer may be broken if expanded too much.

Because of the danger of producing *oversize holes,* do not use an expansion reamer instead of a solid reamer unless absolutely necessary.

The ***adjustable hand reamer*** is threaded its entire length and is fitted with tapered slots to receive the adjustable blades, **Figure 6-68C**. The blades are tapered along one edge to correspond with the taper slots in the reamer body, so that the cutting edges of the blades remain parallel.

Reamer diameter is set by loosening one adjusting nut and tightening the other. The blades can be moved in either direction. This type of reamer is manufactured in sizes ranging from 3/8″ to 3 1/2″ (9.5 mm to 85 mm). Each reamer has sufficient adjustment to increase its diameter to the next larger reamer size.

The ***taper reamer*** will finish a tapered hole accurately and with a smooth finish for taper pins, **Figure 6-68D**. Because of their long cutting edges, taper reamers are somewhat difficult to operate.

To provide for easier removal of surplus metal, a ***roughing reamer*** is first rotated into the hole. This reamer is slightly smaller (0.010″ or 0.25 mm) than the finish reamer. Left-hand spiral grooves are ground along the cutting edges to break up chips.

6.9.2 Using a Hand Reamer

A two-handle tap wrench is commonly used with a reamer, because it permits an even application of pressure. It is virtually impossible to secure a satisfactory hole using an adjustable wrench to turn the reamer.

To start reaming, rotate the tool slowly to allow it to align with the hole. It is desirable to check, at several points around the reamer's circumference, whether it has started square, **Figure 6-69.**

Feed should be steady and rapid. Keep the reamer cutting, or it will start to "chatter," producing a series of tool marks in the surface of the hole. This could also cause the hole to be out-of-round.

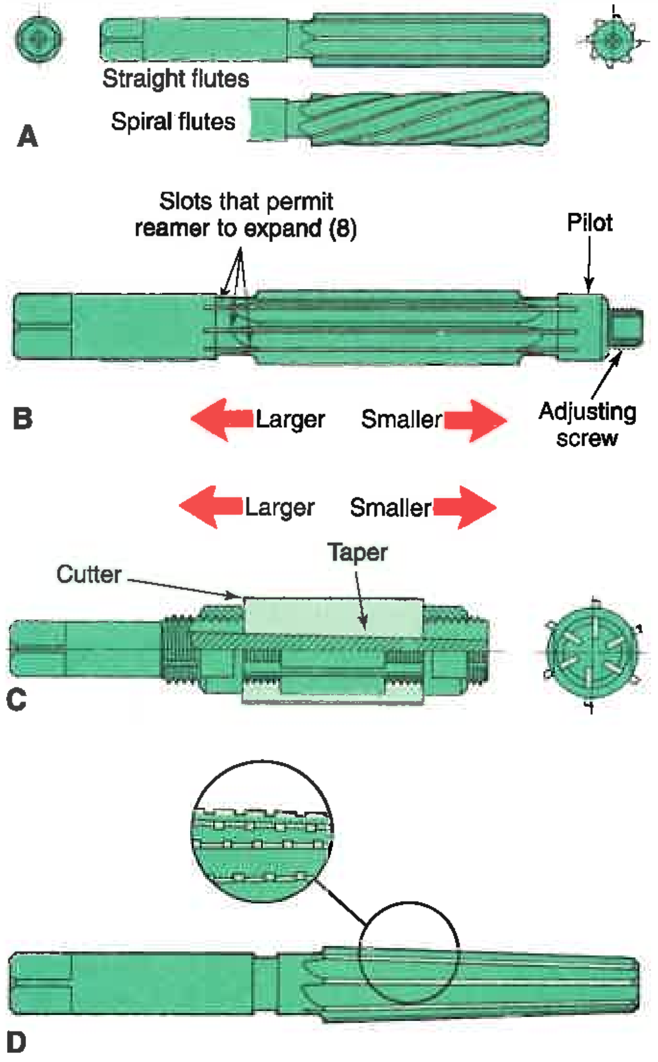


Figure 6-68. *Hand reamers. A—Straight flute and spiral flute solid reamers are used for different applications. B—The expansion hand reamer and how its size is adjusted. C—The adjustable hand reamer can be set for odd sizes. D—A taper hand reamer. Enlarged section shows how the cutting edges are notched on a roughing taper reamer.*

Turning pressure is applied evenly with both hands, *always* in a clockwise direction, **Figure 6-70.** Never turn a reamer in a *counterclockwise* direction, since this will dull the cutting edges.

Feed the reamer deeply enough into the hole to take care of the starting taper. The choice of cutting fluid to be applied will depend upon the metal being reamed.

6.9.3 Reaming Safety

- To prevent injury, remove all burrs from holes.
- Never use your hands to remove chips and cutting fluid from the reamer. Use a piece of cotton waste.

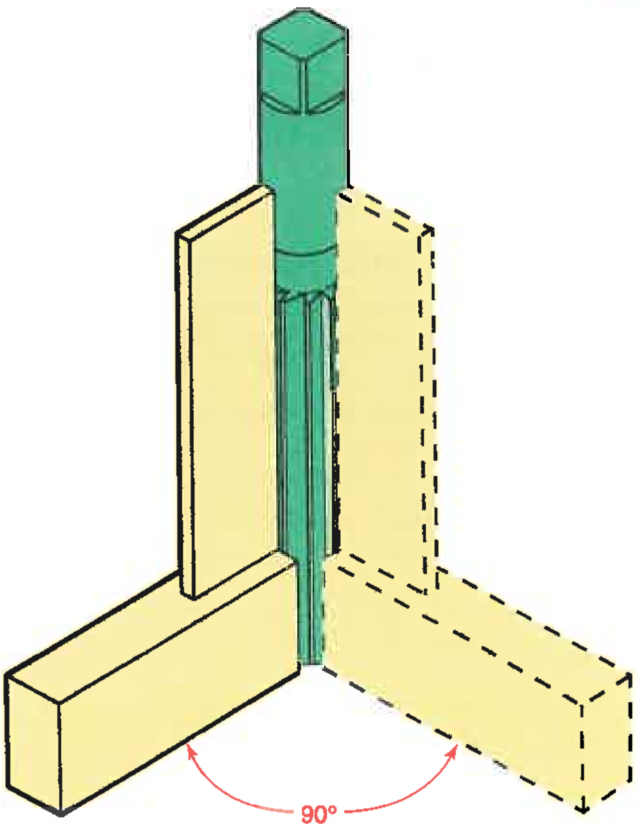


Figure 6-69. *Always make sure that the reamer is square with the work.*

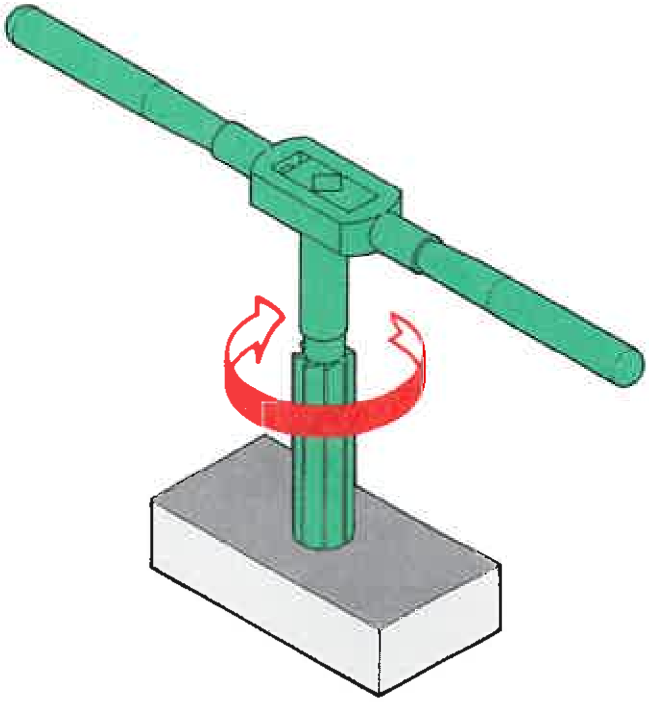
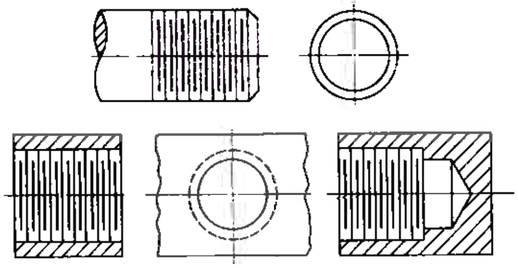


Figure 6-70. *Always turn a hand reamer in a clockwise direction.*

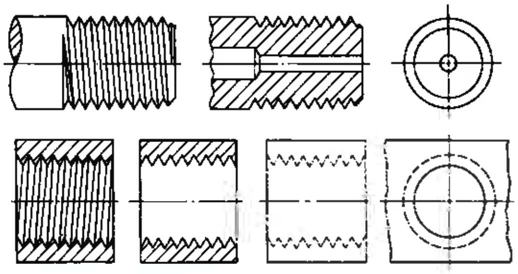
- Store reamers carefully so they do not touch one another. Never store reamers loose or throw them into a drawer with other tools.
- Clamp work solidly before starting to ream.
- Do not use compressed air to remove chips and cutting fluid or to clean a reamed hole.

6.10 HAND THREADING

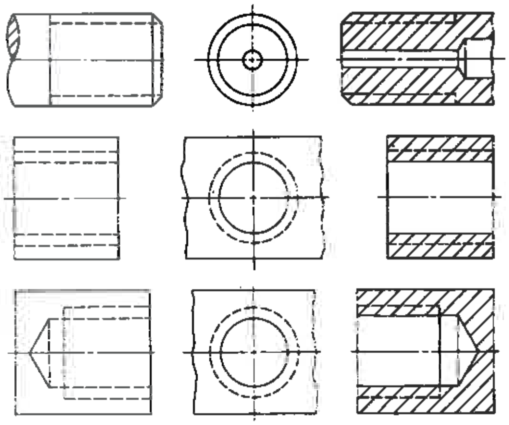
Threaded sections have many applications in our everyday life. A ***thread*** is a spiral or helical ridge found on nuts and bolts. When required on a job, threads are indicated on the plans and drawings in a special way, **Figure 6-71**. They are specified by diameter and number of threads per inch. Metric threads are specified by diameter and thread pitch is given in millimeters.



Schematic representation



Detailed representation



Simplified representation

Figure 6-71. *Methods used to visually depict threads on drawings. Only one type will be used on a given drawing.*

The ***American National Thread System*** was adopted in 1911. It is the common thread form used in the United States and is characterized by the 60° angle formed by the sides of the thread.

The ***National Coarse (NC) Thread*** is for general purpose work; the ***National Fine (NF) Thread*** is for precision assemblies. These are the most widely used thread groups in the American National series. The NF group has more threads per inch for a given diameter than the NC group.

A considerable amount of confusion resulted during World War II from the many different forms and kinds of threads used by the Allied nations. As a result, the powers that make up NATO (the North Atlantic Treaty Organization) adopted a standard thread form. It is referred to as the ***Unified System,*** **Figure 6-72**. It is very similar to the American National Thread System. It differs only in the thread shape. The thread root is rounded and the crest may be flat or rounded. The threads are identified by UNF and UNC (Unified National Fine and Unified National Coarse), **Figure 6-73**. Fasteners using this thread series are interchangeable with fasteners using the American National thread.

In addition to those just described, there are several other thread groups. Included are the ***Unified National Extra Fine*** (UNEF), ***Unified National 8 Series***, and ***Unified National 12 Series***. The 8 Series has 8 threads per inch and is used on diameters ranging from 1″ to 6″ in 1/8″ and 1/4″ increments. The 12 Series has 12 threads per inch and is used on diameters that range from 1″ to 6″.

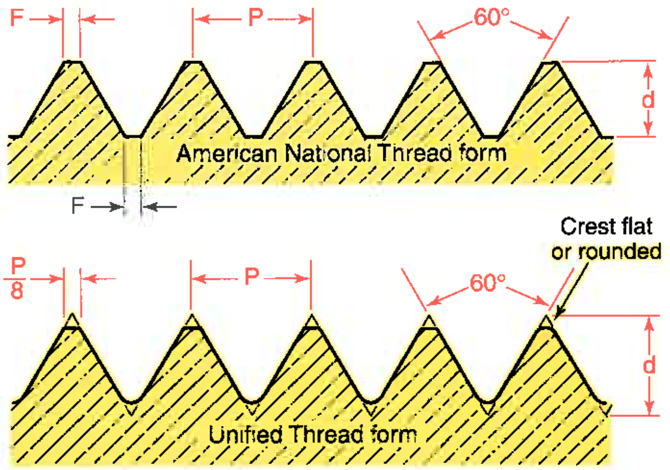


Figure 6-72. *Drawings illustrate similarities and differences of the American National Thread form and the Unified Thread*

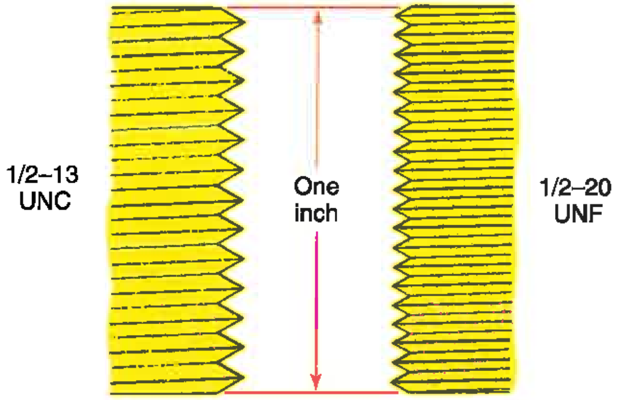


Figure 6-73. *Comparison of Unified Coarse (UNC), and Unified Fine (UNF) threads. Both have same geometric shape.*

Metric unit threads have the same shape as the Unified Thread, but are specified in a different manner, **Figure 6-74**. Metric threads and Unified National Series threads are not interchangeable. See **Figure 6-75**.

6.10.1 Thread Size

Threads of the Unified National system smaller than 1/4″ diameter are not measured as *fractional* sizes. They are given by ***number sizes*** that range from #0 (approximately 1/16″or 0.060″ diameter) to #12 (just under 1/4″or 0.216″ diameter). Both UNC and UNF series are available.

Care must be taken so the number denoting the thread diameter and the number of threads are not mistaken for a fraction. For example: a #8-32 UNC thread would be a thread that is a #8 (0.164″) diameter and 32 threads per inch, *not* an 8/32 (1/4″) diameter fastener with a UNC series thread.

6.10.2 Cutting Threads

Because thread dimensions have been standardized, the use of *taps* to cut *internal* threads, and dies to cut *external* threads have become universal practice whenever threads are to be cut by hand. See **Figure 6-76**.

6.10.3 Internal Threads

Internal threads are made with a tap, **Figure 6-77**. Taps are made of carbon steel or high-speed steel (HSS) and are carefully heat-treated for long life. Taps are quite brittle and are easily broken if not handled properly.

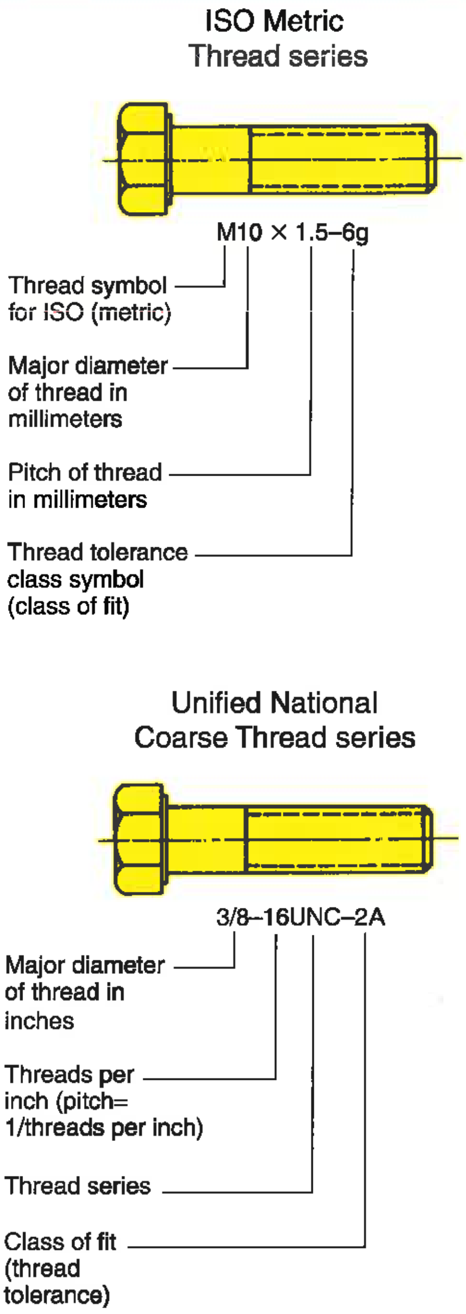


Figure 6-74. *How thread size is noted and what each term means.*

To meet demands for varying degrees of thread accuracy, it became necessary for industry to adopt standard working tolerances for threads. Working tolerances for threads have been divided into ***classes of fits,*** which are indicated by the last number on the thread description (1/2-1 3 UNC-2).

Fits for inch-based threads are:
Class 1 – Loose fit.
Class 2 – Free fit.
Class 3 – Medium fit.
Class 4 – Close fit.

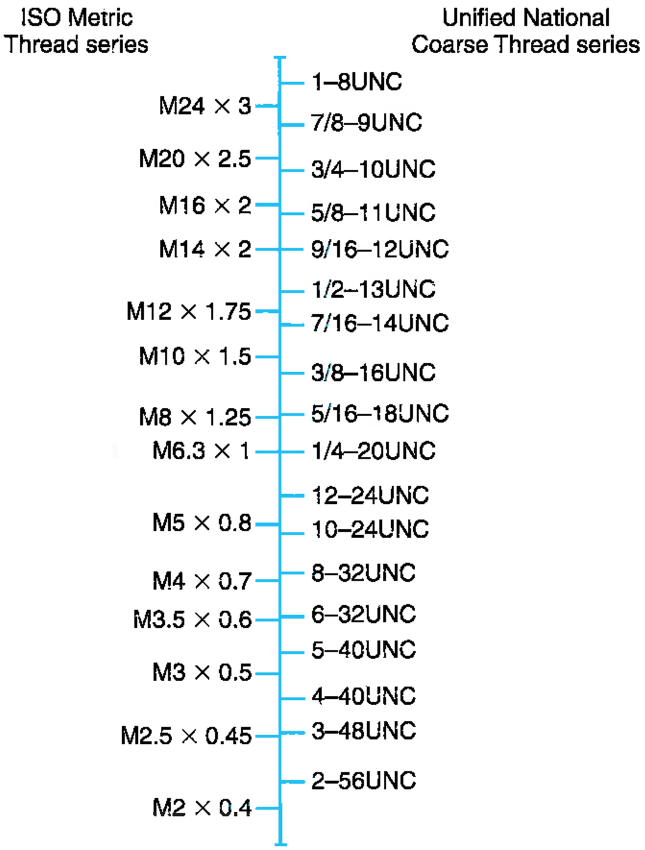


Figure 6-75*. While ISO Metric threads may appear to be similar in diameter to the Unified National Thread series, the two are not interchangeable.*

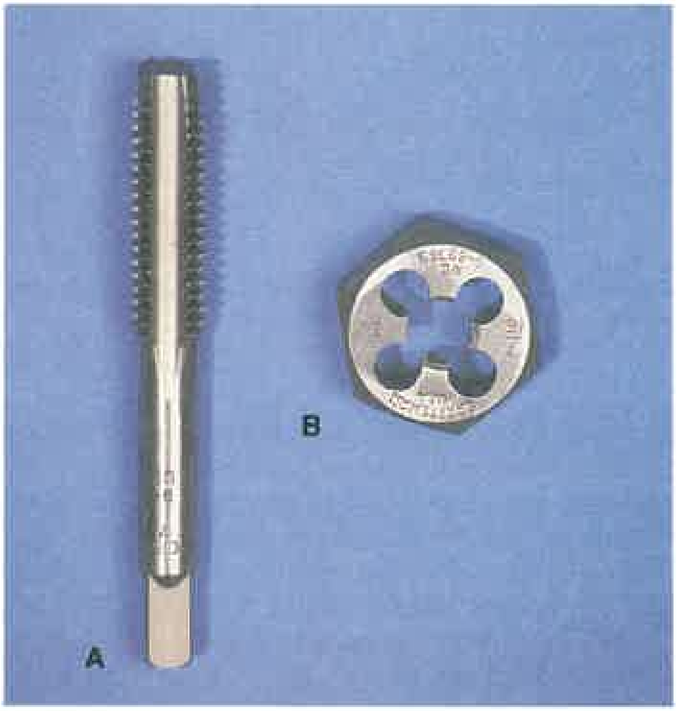


Figure 6-76. *Thread cutting. A—Tap is for cutting internal threads. B—Die is for cutting external threads.*

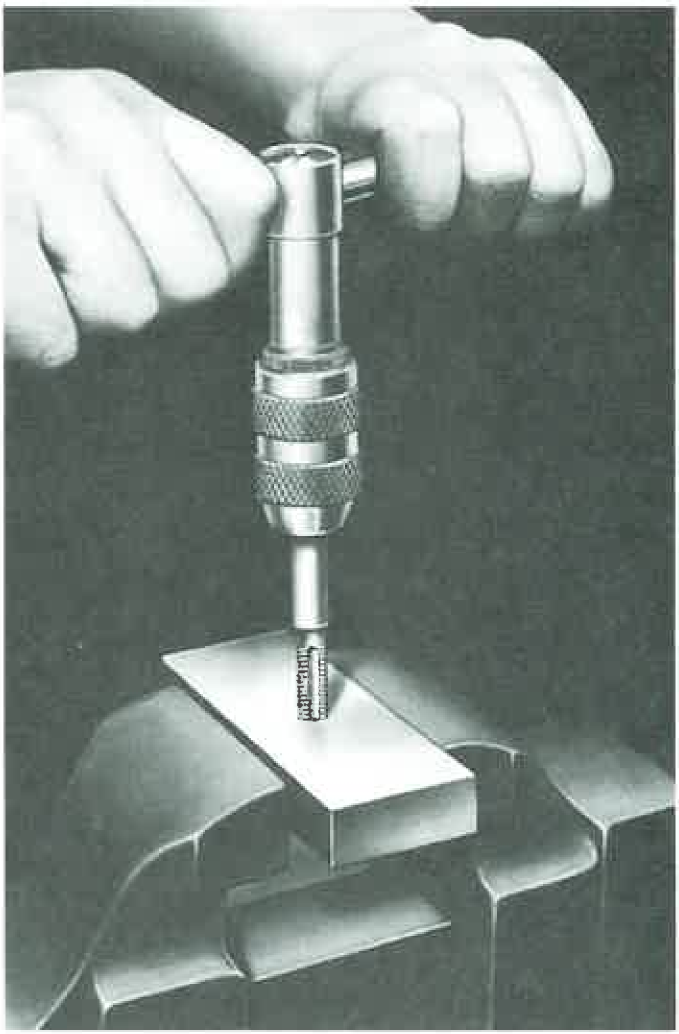


Figure 6-77*. A machinist is cutting internal threads with a tap.*

Under revised ISO standards, there are two classes of thread tolerances for external threads: 6g for ***general-purpose threads*** and 5g6g for ***close tolerance threads.*** There is only one tolerance class for internal threads, 6H. A *lower case* letter indicates the tolerance on a bolt and a *capital* letter is used for the nut.

6.10.4 Taps

Standard hand taps are made in sets of three. They are known as taper, plug, and bottoming taps. See **Figure 6-78.**

Threads are started with a ***taper tap.*** It is tapered back from the end 6 to 10 threads before full thread diameter is reached.

The ***plug tap*** is used after the taper tap has cut threads as far into the hole as possible. It tapers back 3 or 4 threads before full thread diameter is reached.

Figure 6-78. Standard hand taps. A—Taper for starting thread. B—Plug tap for continuing thread after taper tap has cut as far into hole as it can. C—Bottoming for continuing threads to bottom of a blind hole. (Threadwell Manufacturing Co.)

Threads are cut to the bottom of a ***blind hole*** (one that does not go through the part) with a ***bottoming tap.*** This tap tapers back 1 or 20 threads before full thread diameter is reached.It is necessary to use the full set of taps only when a blind hole is to be tapped, **Figure 6-79.**

Another tap used in the shop is a pipe tap. A ***pipe tap*** cuts a tapered thread, so there is a "wedging" action set up to make a leak-tight joint. The fraction that indicates pipe tap size may be confusing at first because it indicates pipe size and not the thread diameter. See **Figure 6-80.**

Figure 6-79. Cutaway of metal block illustrates three types of threaded holes. Left—Open or through hole. Center—Blind hole that has been drilled deeper than desired threads. Right—Blind hole with threads tapped to bottom.

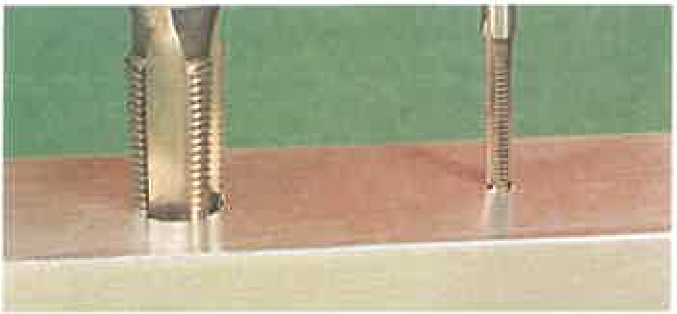


Figure 6-80. There is considerable difference between a 1/8" standard thread tap and a 1/8" pipe thread tap.

A ***pipe thread*** is indicated by the abbreviation ***NPT*** (National Pipe Thread). To obtain the wedging action needed for leakproof joints, the threads taper 3/4" per foot of length.

6.10.5 Tap Drill

The drill used to make the hole prior to tapping is called a ***tap drill.*** Theoretically, it should be equal in diameter to the minor diameter of the screw that will be fitted into the tapped hole. See **Figure 6-81.**

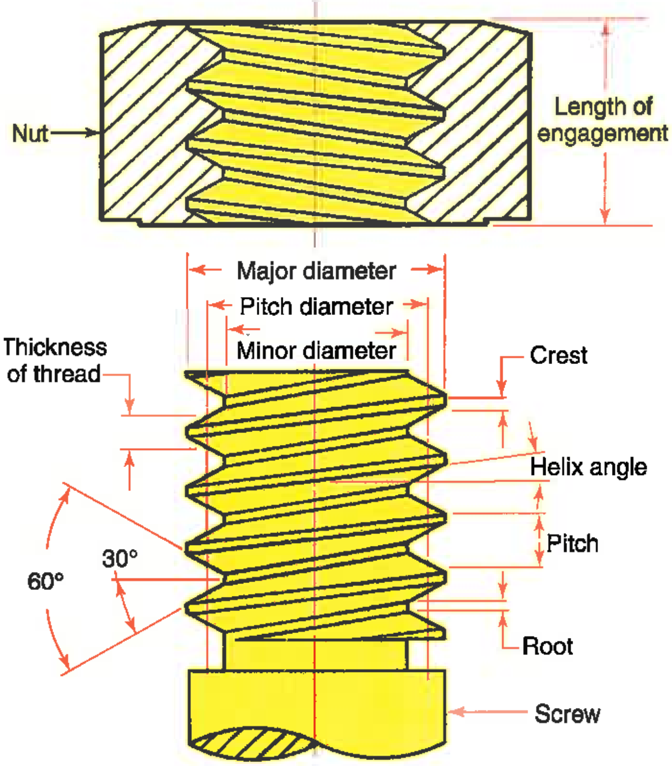


Figure 6-81. Nomenclature of a fastener thread.

This situation would cause the tap to cut a full thread, however. The pressure required to rotate the tap would be so great that tap breakage could occur. Full-depth threads are not necessary because three-quarter-depth threads are strong enough that the fastener usually breaks before the threads strip.

Drill sizes can be secured from a tap drill chart, **Figures 6-82** and **6-83.**

6.10.6 Tap Wrenches

Two types of tap wrenches are available, **Figure 6-84**. The type to be employed will depend upon tap size. A ***T-handle tap wrench*** should be used with all small taps. It allows a more sensitive "feel" when tapping. The ***hand tap wrench*** is best suited for large taps where more leverage is required.

When tapping by hand, the chief requirement is to make sure that the tap is started straight, and remains square during the entire tapping operation, **Figure 6-85**. The tap must be ***backed off*** (reversed in rotation) for one-half of a turn every one or two cutting turns. This will break the chips free and allow them to drop through the tap flutes. Backing off prevents chips from jamming the tap and damaging the threads.

National Coarse and National Fine threads and tap drills

Size	Threads per inch	Major dia.	Minor dia.	Pitch dia.	Tap drill 75% thread	Decimal equiv-alent	Clearance drill	Decimal equiv-alent
2	56	.0860	.0628	.0744	50	.0700	42	.0935
	64	.0860	.0657	.0759	50	.0700	42	.0935
3	48	.099	.0719	.0855	47	.0785	36	.1065
	56	.099	.0758	.0874	45	.0820	36	.1065
4	40	.112	.0795	.0958	43	.0890	31	.1200
	48	.112	.0849	.0985	42	.0935	31	.1200
6	32	.138	.0974	.1177	36	.1065	26	.1470
	40	.138	.1055	.1218	33	.1130	26	.1470
8	32	.164	.1234	.1437	29	.1360	17	.1730
	36	.164	.1279	.1460	29	.1360	17	.1730
10	24	.190	.1359	.1629	25	.1495	8	.1990
	32	.190	.1494	.1697	21	.1590	8	.1990
12	24	.216	.1619	.1889	16	.1770	1	.2280
	28	.216	.1696	.1928	14	.1820	2	.2210
1/4	20	.250	.1850	.2175	7	.2010	G	.2610
	28	.250	.2036	.2268	3	.2130	G	.2610
5/16	18	.3125	.2403	.2764	F	.2570	21/64	.3281
	24	.3125	.2584	.2854	I	.2720	21/64	.3281
3/8	16	.3750	.2938	.3344	5/16	.3125	25/64	.3906
	24	.3750	.3209	.3479	Q	.3320	25/64	.3906
7/16	14	.4375	.3447	.3911	U	.3680	15/32	.4687
	20	.4375	.3725	.4050	25/64	.3906	29/64	.4531
1/2	13	.5000	.4001	.4500	27/64	.4219	17/32	.5312
	20	.5000	.4350	.4675	29/64	.4531	33/64	.5156
9/16	12	.5625	.4542	.5084	31/64	.4844	19/32	.5937
	18	.5625	.4903	.5264	33/64	.5156	37/64	.5781
5/8	11	.6250	.5069	.5660	17/32	.5312	21/32	.6562
	18	.6250	.5528	.5889	37/64	.5781	41/64	.6406
3/4	10	.7500	.6201	.6850	21/32	.6562	25/32	.7812
	16	.7500	.6688	.7094	11/16	.6875	49/64	.7656
7/8	9	.8750	.7307	.8028	49/64	.7656	29/32	.9062
	14	.8750	.7822	.8286	13/16	.8125	57/64	.8906
1	8	1.0000	.8376	.9188	7/8	.8750	1- 1/32	1.0312
	14	1.0000	.9072	.9536	15/16	.9375	1- 1/64	1.0156
1-1/8	7	1.1250	.9394	1.0322	63/64	.9844	1- 5/32	1.1562
	12	1.1250	1.0167	1.0709	1- 3/64	1.0469	1- 5/32	1.1562
1-1/4	7	1.2500	1.0644	1.1572	1- 7/64	1.1094	1- 9/32	1.2812
	12	1.2500	1.1417	1.1959	1-11/64	1.1719	1- 9/32	1.2812
1-1/2	6	1.5000	1.2835	1.3917	1-11/32	1.3437	1-17/32	1.5312
	12	1.5000	1.3917	1.4459	1-27/64	1.4219	1-17/32	1.5312

Figure 6-82. *Thread and tap drill chart for Unified National threads.*

Nominal size	Internal thread minor diameter		Tap drill diameter
	Max.	Min.	
M1.6×0.35	1.321	1.221	1.25
M2×0.4	1.679	1.567	1.6
M2.5×0.45	2.138	2.013	2.05
M3×0.5	2.599	2.459	2.5
M3.5×0.6	3.010	2.850	2.9
M4×0.7	3.422	3.242	3.3
M5×0.8	4.334	4.134	4.2
M6.3×1	5.553	5.217	5.3
M8×1.25	6.912	6.647	6.8
M10×1.5	8.676	8.376	8.5
M12×1.75	10.441	10.106	10.2
M14×2	12.210	11.835	12.0
M16×2	14.210	13.885	14.0
M20×2.5	17.744	17.294	17.5
M24×3	21.252	20.752	21.0
M30×3.5	26.771	26.211	26.5
M36×4	32.370	31.670	32.0
M42×4.5	37.799	37.129	37.5
M48×5	43.297	42.587	43.0
M56×5.5	50.796	50.046	50.5
M64×6	58.305	57.505	58.0
M72×6	66.305	65.505	66.0
M80×6	74.305	73.505	74.0
M90×6	84.305	83.505	84.0
M100×6	94.305	93.505	94.0

Figure 6-83. *Thread and tap drill chart for metric threads.*

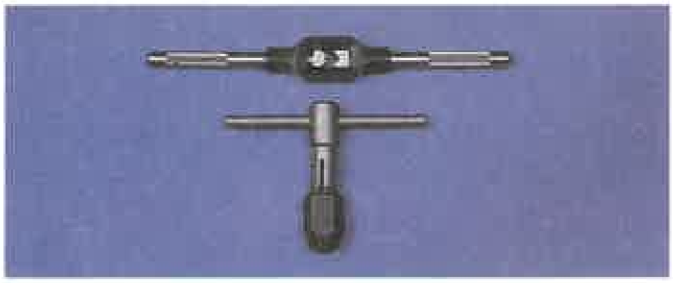


Figure 6-84. *Tap wrenches Top—Hand tap wrench. Bottom—T-handle tap wrench.*

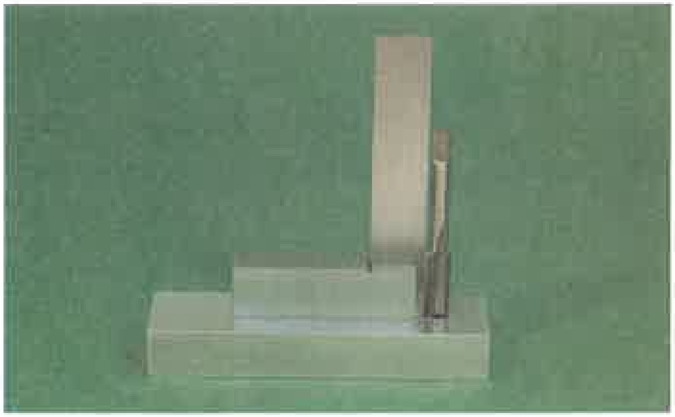


Figure 6-85. *A tap must be started squarely with the hole. A quick way to check this is to use a machinist's square.*

When tapping a blind hole, some machinists place a dab of grease, or a piece of grease pencil or wax crayon in the hole. As the tap cuts the threads, the grease is forced up and out of the hole, carrying the chips along.

Use a cutting fluid designed for the particular metal you are tapping.

6.10.7 Power Tapper

Tapping can also be done using an ***articulating arm power tapper,*** **Figure 6-86,** which can be fitted with an auto reverse, automatic tap lubrication, and a digital depth readout. Holes can be tapped quickly with little chance of tap breakage.

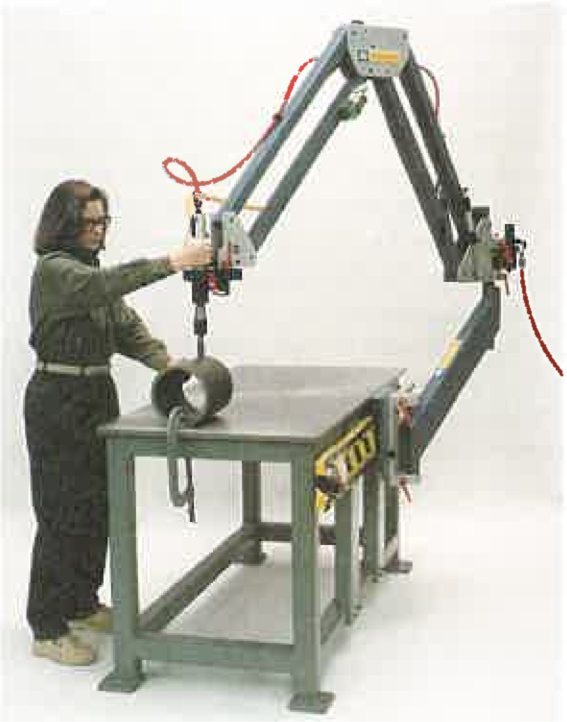


Figure 6-86. *Articulated arm power tapper. The machine can be fitted with an auto reverse system (the tap is run out of the hole automatically when the threads are at the proper depth), automatic lubrication of the tap, and a digital depth readout. (Procyon Machine Tools)*

6.10.8 Care in Tapping

Considerable care must be exercised when tapping:

- Use the correct size tap drill. Secure this information from a tap drill chart.
- Use a sharp tap and apply sufficient quantities of cutting fluid. With some cutting fluids, the area is flooded with fluid; with others, a few drops are sufficient. Read the container label.
- Start the taper tap square with the work.
- Do not force the tap to cut. Remove the chips using a piece of cloth or cotton waste, not your fingers.
- Avoid running a tap to the bottom of a blind hole and continuing to apply pressure. Do not allow the hole to fill with chips and jam the tap. Either condition can cause the tap to break (especially if the tap is small).
- Remove burrs on the tapped hole with a smooth file.

6.10.9 Dealing with Broken Taps

Taps sometime break off in a hole. Several tools and techniques have been developed for removing them without damaging the threads already cut. Remember that these methods do not always work and the part may have to be discarded.

Frequently, a tap will *shatter* in the hole. It may then be possible to remove the fragments with a pointed tool such as a scribe.

Broken carbon steel taps can sometimes be removed from steel if the work can be heated to annealing temperature. The tap can then be drilled out. This *cannot* be done with high-speed steel taps. If the HSS tap is large enough, it can be ground out with a hand grinder.

A ***tap extractor*** can sometimes be used to remove a broken tap. See **Figure 6-87**. Penetrating oil should be applied and allowed to "soak in" for a short time before the fingers of the tap extractor are fitted into the flutes of the broken tap. The collar on the extractor is slipped down flush with the work surface. A tap wrench is fitted on the extractor. The tap extractor is then carefully twisted back and forth to loosen the tap segments. After the broken parts have been loosened, it is a simple matter to remove them.

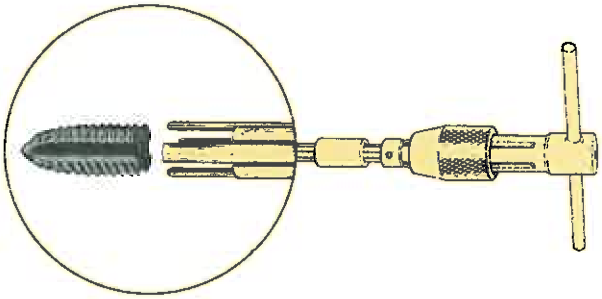


Figure 6-87. *Tap extractor will help remove a broken tap. Close-up shows fingers of extractor and how they fit into flutes of broken tap. It does not always work.*

In some shops, a ***tap disintegrator*** is used to remove broken taps. This device makes use of an electric arc to cause the tap to disintegrate. If used properly, it will break up the tap without affecting the metal surrounding the broken tool.

6.10.10 External Threads

External threads are cut with a die, **Figure 6-88.** ***Solid dies*** are not adjustable and for that reason are

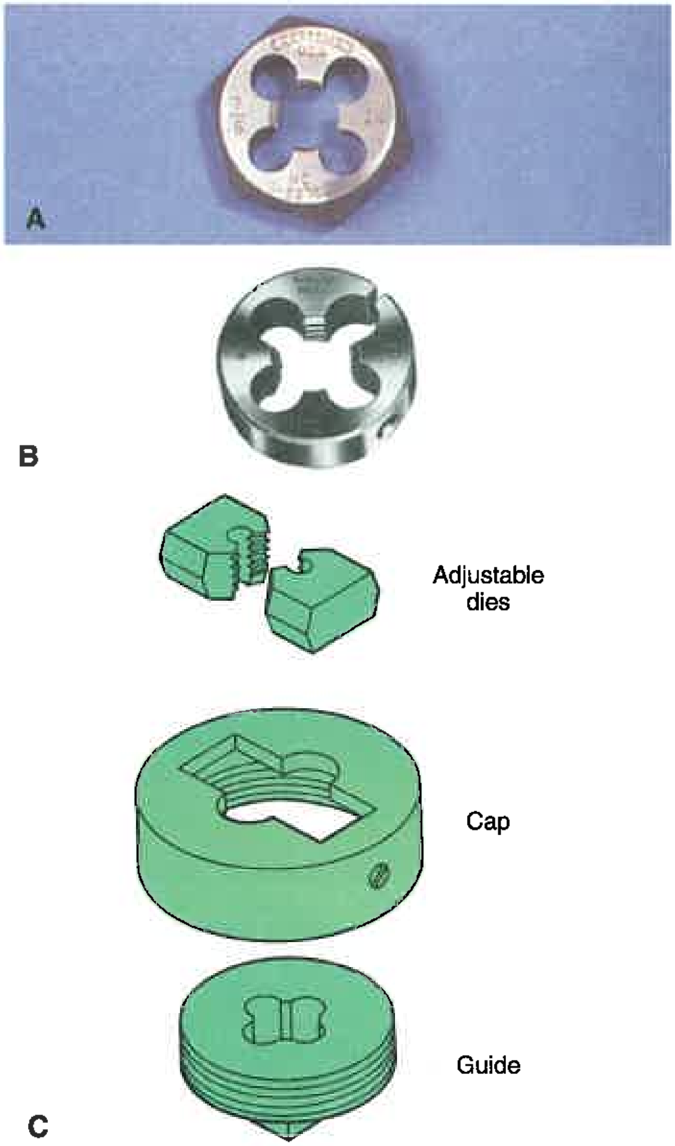


Figure 6-88. *Types of dies. A—Solid dies used to cut external threads by hand. They cannot be adjusted. B—A small screw on one side of the split permits small changes in size of an adjustable die. C—Construction of a multi-part adjustable die.*

not often used. The ***adjustable die,*** and the ***two-part adjustable die,*** are preferred. The two-part die has a wide range of adjustment and is fitted with guides to keep it true and square on the work. Dies are available for cutting most standard threads.

6.10.11 Die Stocks

A ***die stock*** holds the die and provides leverage for turning the die on the work. See **Figure 6-89.**

Figure 6–89. *The die is held in a die stock.*

When cutting external threads, it is necessary to remember:

- Material diameter is the *same size* as the desired thread diameter. That is, 1/2-13 UNC threads are cut on a 1/2" diameter shaft. What diameter shaft would be needed to cut 1/2-20 UNF threads?
- Mount work solidly in a vise.
- Set the die to the proper size. Make trial cuts on a piece of scrap until the proper adjustment is found.
- Grind a small chamfer on the shaft end, as shown in **Figure 6-90**. This permits a die to start easily.
- Start the cut with the tapered end of the die.
- Back off the die every one or two turns to break the chips.
- Use cutting oil. Place a paper towel down over the work to absorb excess cutting oil. The towel will also prevent the oil from getting on the floor.
- Remove any burrs from the finished thread with a fine cut file.

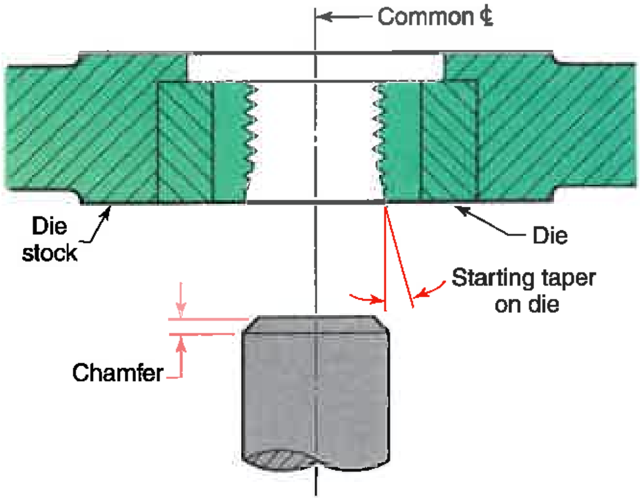


Figure 6-90. *A die will start more easily if a small chamfer is cut or ground on the end of the shaft to be threaded. Section through die and die stock shows proper way to start threads.*

6.10.12 Threading to a Shoulder

When a thread must be cut by hand to a shoulder, start and run the threads as far as possible in the usual manner, **Figure 6-91**. Remove the die. Turn it over with the taper up. Run the threads down again to the shoulder. *Never try this operation without first starting the threads in the usual manner.*

6.10.13 Problems in Cutting External Threads

The most common problem encountered when cutting external threads with a die is ***ragged threads.*** They are caused by:

- Applying little or no cutting oil.
- Dull die cutters.
- Stock too large for the threads being cut.
- Die not started square.
- One set of cutters upside down when using a two part die.

6.10.14 Hand Threading Safety

- If a tap or threaded piece must be cleaned of chips with compressed air, protect your eyes from flying chips by wearing goggles. Take care not to endanger persons working in the area near you!
- Chips produced by hand threading are sharp. Use a brush or piece of cloth, *not* your hand, to remove them!
- Newly-cut external threads are very sharp. Again, use a brush or cloth to clean them.
- Wash your hands after using cutting fluids or oils! Some cause skin rash. This can develop into a serious skin disorder if the oils are left on the hands for an extended period.
- Have cuts treated by a qualified person. Infections can occur when cuts and other injuries are not properly treated.
- Be sure the die is clamped firmly in the die stock. If not, it can fall from the holder and cause injury.
- Broken taps have very sharp edges and are very dangerous. Handle them as you would broken glass!

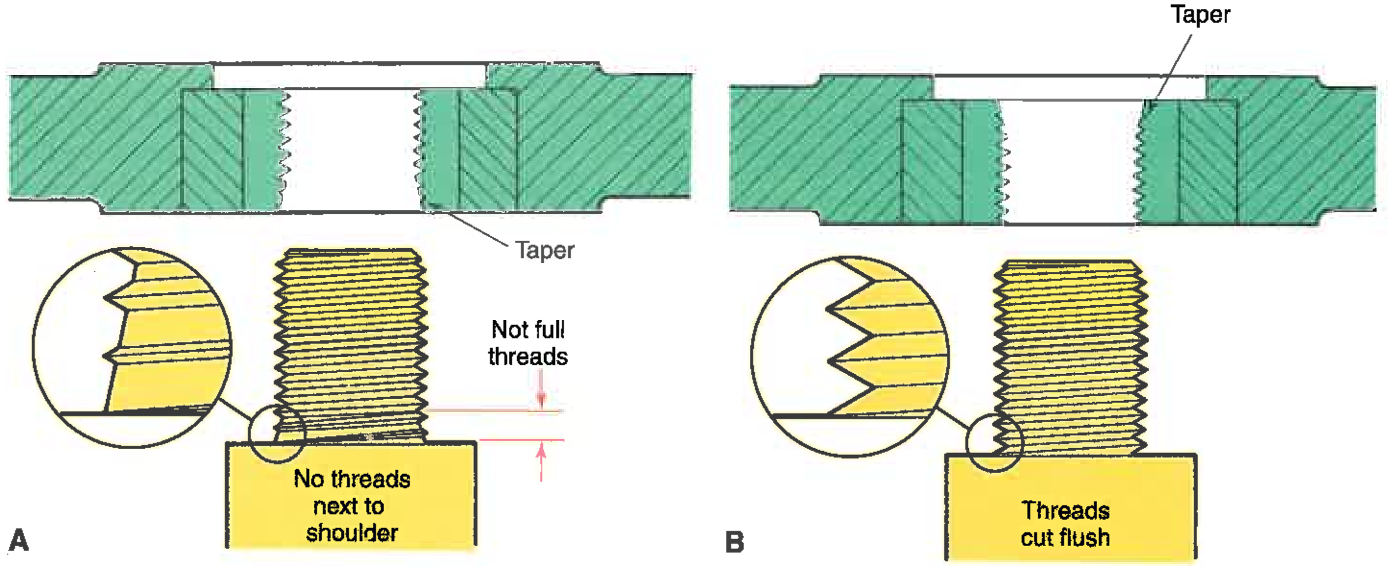


***Figure 6-91**. How to cut threads to a shoulder. After die has been run down as far as possible, the die is reversed. When rotated down the shaft, it will cut threads almost flush with shoulder. A—Running die down normally. B—Reversing die to cut flush.*

6.11 HAND POLISHING

An ***abrasive*** is commonly thought of as any hard substance that will wear away another material. The substance, grain size, backing material, and the manner in which the substance is bonded to the backing material determines the performance and efficiency of an abrasive.

6.11.1 Abrasive Materials

Emery is a natural abrasive. It is black in color and cuts slowly, with a tendency to polish.

Aluminum oxide has replaced emery as an abrasive when large quantities of metal must be removed. It is a ***synthetic*** (manufactured) material that works best on high-carbon and alloy steels. Aluminum oxide that is designed for use on metal is manufactured with a grain shape that is not as sharp as that made for woodworking.

Silicon carbide is the hardest and sharpest of the synthetic abrasives. Silicon carbide is greenish-black in color. It is superior to aluminum oxide in its ability to cut fast under light pressure. It is ideal for "sanding" metals like cast iron, bronze, and aluminum.

Crocus may be synthetic or natural iron oxide. It is bright red in color, very soft, and is used for cleaning and polishing when a minimum of stock is to be removed.

Diamonds are the hardest *natural* substance known. However, they can also be manufactured. Synthetic diamonds have no value as gems; they are used almost exclusively by industry for polishing and grinding. ***Diamond dust polishing compound*** is made by crushing synthetic diamonds. It is the only abrasive hard enough to polish the newer heat-treated, exotic alloy steels used by industry.

The table in **Figure 6-92** shows a comparison of abrasive grain size and indicates how the various abrasives are graded.

Technical grades		Simplified grades	Other grades
Mesh	Aluminum oxide Silicon carbide	Emery	Emery polishing
600			4/0
			3/0
500			2/0
400	10/0		0
360			
320	9/0		1/2
280	8/0		
240	7/0		1 G
220	6/0		2
180	5/0		3
150	4/0	Fine	
120	3/0		
100	2/0	Medium coarse	
80	0		
60	1/2		
50	1	Extra coarse	
40	1 1/2		
36	2		
30	2 1/2		
24	3		
20	3 1/2		
16	4		
12	4 1/2		

***Figure 6-92**. A comparative grading chart for abrasives. (Coated Abrasive Manufacturers Institute)*

6.11.2 Coated Abrasives

A ***coated abrasive*** is cloth or paper with abrasive grains bonded to one surface. Because of its flexibility, cloth is used as a backing material for abrasives found in metalworking or machining. It is available in 9″ × 11″ (210 mm × 280 mm) sheets. It is also available in rolls starting at 1/2″ (12.5 mm) in width, and is called ***abrasive cloth.***

6.11.3 Using Abrasive Cloth

Abrasive cloth is quite expensive. Use only what you need. Tear the correct amount from the sheet or roll. Do not discard abrasive cloth unless it is completely worthless. Used cloth is excellent for polishing.

If a job has been filed properly, only a fine-grain cloth will be needed to polish the surface. However, if scratches are deep, start the polishing operation by using a coarse-grain cloth first. Change to a medium-grain cloth next, and finally a fine-grain abrasive. A few drops of oil will speed the operation. For a high polish, leave the oil on the surface after the scratches have been removed. Reverse the cloth and rub the smooth backing over the work.

Abrasive cloth must be properly supported to work efficiently. To get such support, wrap the cloth around a block of wood or file, **Figure 6-93.** Apply pressure and rub the abrasive back and forth in a straight line, parallel to the long side of the work.

Avoid using abrasives on machined surfaces.

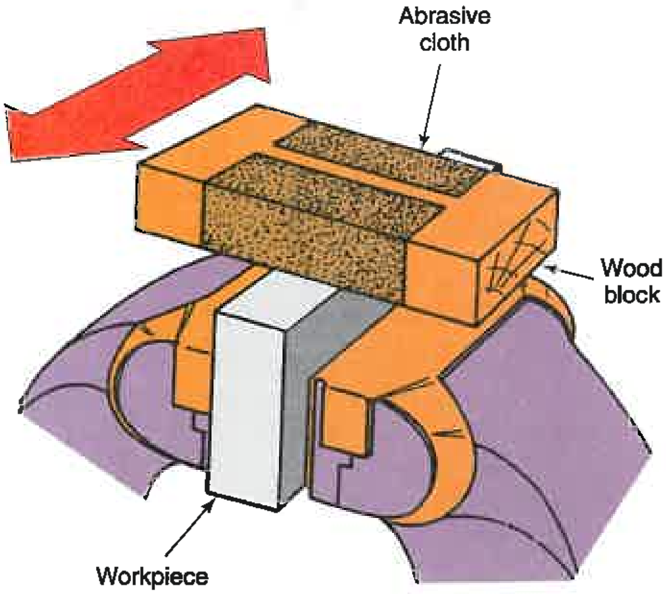


***Figure 6-93.** Abrasive cloth should be supported with a block of wood or a file. Do not support it with your fingers alone.*

6.11.4 Abrasives Safety

- Avoid rubbing your fingers or hand over polished surfaces or surfaces to be polished. Burrs on the edges of the metal can cause painful cuts.
- Wash your hands thoroughly after polishing operations.
- Treat all cuts immediately, no matter how small!
- Place all oily rags in a closed metal container. Never put them in your apron/shop coat or in a locker!
- Wipe up any oil dropped on the floor during polishing operations.
- If a lathe is used for polishing operations, make sure the machine is protected from the abrasive grains that fall from the polishing cloth. Stop the machine when inspecting your work.

TEST YOUR KNOWLEDGE

Please do not write in the text. Write your answers on a separate sheet of paper.

1. List two variations of the machinist's vise.
2. How is vise size determined?
3. Work held in a vise can be protected from damage by the jaw serrations if _____ are placed over the jaws.
4. To prevent injuries, what should be avoided when mounting work in a vise?
5. Work is often held together with a _____ and/or _____ while being machined or worked on.
6. How do combination pliers have an advantage over many other types of pliers?
7. Why are the cutting edges on diagonal pliers set at an angle?
8. List three ways of extending the working life of pliers.
9. What are adjustable clamping pliers?
10. Of what use are torque-limiting wrenches?
11. Do torque-limiting wrenches give a more accurate reading when they are pushed or when they are pulled?
12. Several different wrenches can be classified as adjustable wrenches. Name three.

13. List three points that should be observed when using an adjustable wrench.
14. Round work can be gripped with a _____ wrench. Its main disadvantage is that the jaws will probably _____ the work.
15. Describe socket wrenches.
16. What wrenches are employed to turn flush and recessed types of threaded fasteners? The fasteners have slots or holes to receive the wrench lugs.
17. Rather than lengthen the wrench handle for additional leverage, it is better to use a _____ wrench.
18. List five safety precautions that should be observed when using a wrench.
19. What is the difference between a standard screwdriver tip and a Phillips screwdriver tip?

Match each phrase with the correct screwdriver name listed below.

20. _____ Pozidriv®.
21. _____ Standard.
22. _____ Electrician.
23. _____ Heavy-duty.
24. _____ Stubby.
25. _____ Ratchet.
    a. Has a flattened wedge-shaped tip.
    b. Moves the fastener on the power stroke, but not on the return stroke.
    c. Has a square shank to permit additional force to be applied with a wrench.
    d. Useful when handling small screws.
    e. Tip is similar to that of a Phillips head screwdriver.
    f. Has an insulated handle.
    g. Is short and is used when space is limited.
26. List three safety precautions that should be observed when using a screwdriver.
27. How is the size of a ball-peen hammer determined?
28. Why are soft-face hammers and mallets used in place of a ball-peen hammer?
29. List three safety precautions that should be observed when using striking tools.
30. There are few things more dangerous than a chisel with a head that has become _____ from use. This danger can be removed by _____ .
31. The chisel is an ideal tool for _____ .
32. List the four general types of cold chisels.
33. The standard hacksaw is designed to accommodate _____ .
34. A hacksaw cuts best at about _____ to _____ strokes per minute.
35. Why should the work be mounted solidly and close to the vise before cutting with a hacksaw?
36. If a blade breaks or dulls before completing a cut, you should not continue in the same cut with a new blade. Why?
37. The number of teeth per inch on a hacksaw blade has an important bearing on the shape and kind of metal being cut. At least _____ or _____ should be cutting at all times, otherwise _____.
38. What is the best way to hold thin metal for hacksawing?
39. What is the best way to hold thin wall tubing for hacksawing?
40. Files are cleaned with a _____, never with _____.
41. Files are classified according to the cut of their teeth. List the four cuts.
42. What are the most commonly used file shapes?
43. List three safety precautions that should be observed when files are used.
44. When is reaming done?
45. How much stock should be left in a hole for hand reaming?
46. A _____ is used to cut internal threads. External threads are cut with a _____ .
47. The hole to be tapped must be:
    a. The same diameter as the desired thread.
    b. A few thousandths larger than the desired thread.
    c. A few thousandths (0.003″–0.004″) smaller than the threads.
    d. All of the above.
    e. None of the above.

48. The drill used to make the hole prior to threading is called a _____.
49. How does the UNC thread series differ from the UNF thread series?
50. List the correct sequence taps should be used to form threads the full depth of a blind hole.
51. Should a shaft be larger or smaller than the finished size if external threads are to be cut on it?
52. Taps are turned in with a _____ . A _____ is used with dies.
53. What is an abrasive?
54. _____ surfaces are never polished with an abrasive.

Since many products make use of threaded fasteners for assembly, fast and very accurate drilling and tapping of holes in components is necessary. For larger volumes of parts, automated equipment like this CNC drill and tap center are often used. Note that tools are mounted on an 8-spindle turret for quick tool changes. The workpiece is held stationary on a fixed position worktable, with all movement in the X, Y, and Z axes made by the traveling column holding the tool turret. (Sugino Corp.)

Chapter 7

Fasteners

LEARNING OBJECTIVES

After studying this chapter, you will be able to:

- ❍ Identify several types of fasteners.
- ❍ Explain why inch-based fasteners are not interchangeable with metric-based fasteners.
- ❍ Describe how some fasteners are used.
- ❍ Select the proper fastening technique for a specific job.
- ❍ Describe chemical fastening techniques.

IMPORTANT TERMS

adhesives
assembly
cyanoacrylate quick setting adhesives
fastener
keyway
machine bolts
permanent assemblies
setscrews
threaded fasteners
washers

A ***fastener*** is any device used to hold two objects or parts together. This definition would include bolts, nuts, screws, pins, keys, rivets, and even chemical bonding agents or adhesives. The most common types of fasteners will be explained and illustrated in this chapter.

It is critical to choose the proper fasteners for each job, **Figure 7-1**. A poorly selected fastener can greatly reduce the safety and dependability originally designed into a product. Choosing improper fasteners could increase assembly costs and result in an inferior or faulty product. To improve quality, several different fastening techniques are often employed in the same or related assemblies. For example, one auto manufacturer uses more than 11,000 kinds and sizes of fasteners.

7.1 THREADED FASTENERS

Threaded fasteners make use of the *wedging action* of the screw thread to clamp parts together.

Figure 7-1. *Complex products, such as this large earthmover and the flatbed truck being used to transport it, require the use of many types and sizes of fasteners. Reliability of the product, and the safety of persons using it, can be greatly affected if improper fasteners are selected in the design or assembly phases.*

To achieve maximum strength, a threaded fastener should screw into its mating part at least a distance equal to one and one-half times the thread diameter. See **Figure 7-2.**

Threaded fasteners vary in cost from thousands of dollars for special bolts that attach the wings to the fuselage of large aircraft, to a fraction of a penny for small machine screws. See **Figure 7-3.**

Most threaded fasteners are available in metric sizes. Many American manufacturers now use metric-sized fasteners in their products, which has led

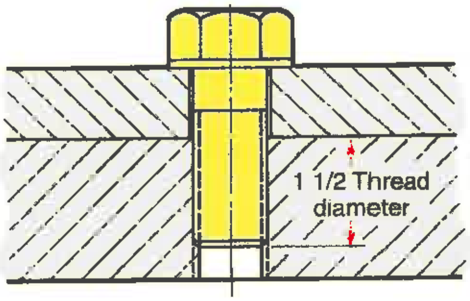


Figure 7-2. *For maximum strength, a threaded fastener must screw into the mating part a distance equal to 1 1/2 times the diameter of the thread.*

Figure 7-3*. There is a wide range of threaded fasteners available, from tiny machine screws used in precision instruments to large bolts used in building construction. This 1" anchor bolt is being used to mount a steel column on a concrete foundation pier.*

to some problems. Metric threads and the common unified (inch-based) threads have the same ***basic profile*** (shape), but are *not* interchangeable. See **Figure 7-4**.

Until a complete changeover to metric sizes is made, and products already made with unified threads wear out or are discarded, some easy-to-use method must be devised to distinguish between metric threaded fasteners and inch-based fasteners. While no foolproof method has yet been contrived, **Figures 7-5** and **7-6** illustrate two possible solutions.

7.1.1 Machine Screws

Machine screws are widely used in general assembly work. They have slotted or recessed heads, and are made in a number of head styles, **Figure 7-7**.

Machine screws are available in body diameters ranging from #0000 (0.021") to 3/4" (0.750") and in lengths from 1/8" (0.125") to 3". Metric sizes are also manufactured. Nuts, in either square or hexagonal shapes, are purchased separately.

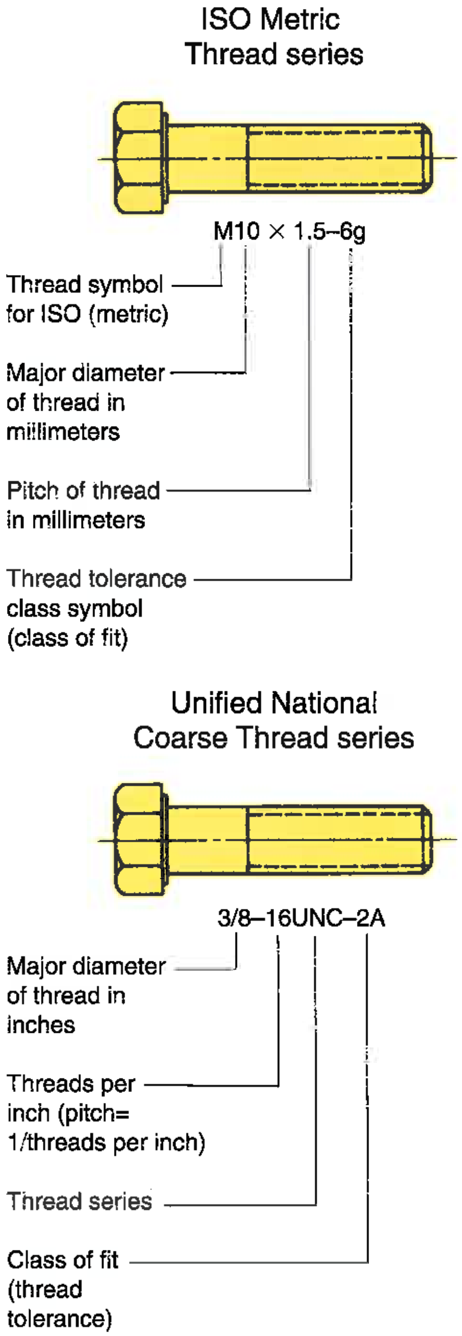


Figure 7-4*. Metric threads have the same basic profile (shape) as the Unified Thread series; however, the Unified and Metric threads are* not *interchangeable.*

7.1.2 Machine Bolts

Machine bolts are employed to assemble parts that do not require close tolerances. They are manufactured with square and hexagonal heads, in diameters ranging from 1/2" to 3". The nuts are similar in shape to the bolt head. They are usually furnished with the machine bolts. Tightening the nut produces a clamping action to hold parts together, **Figure 7-8**.

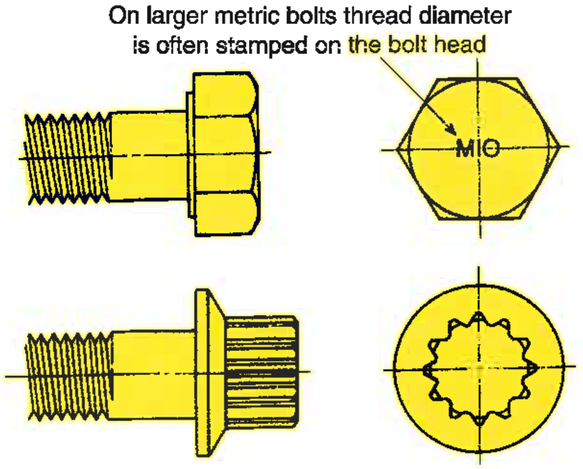


Figure 7-5. *Metric fasteners are manufactured in the same variety of head shapes as inch-based fasteners. However, there is a problem in finding an easy way to distinguish between the two fastener types. Top—Some larger size hex-head metric fasteners have the size stamped on the head. Bottom—A twelve-spline flange head is under consideration for use on eight sizes of metric fasteners: 5, 6.3, 8, 10,13, 14, 16, and 20 mm.*

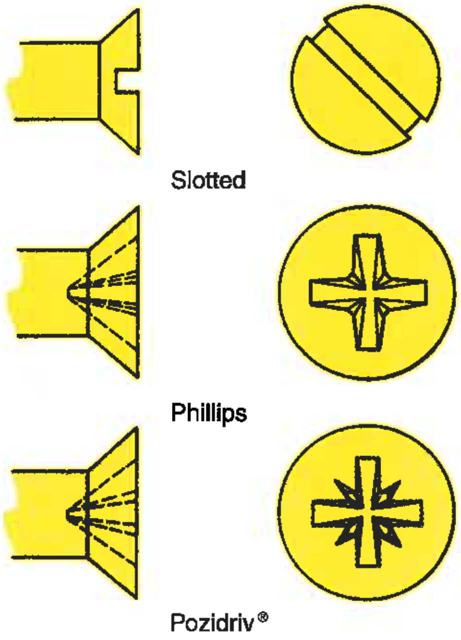


Figure 7-6. *The Pozidriv® cross-recess head has been suggested as a means of identifying metric screws. It would replace the Phillips cross-recess head, which would be used only with inch-based screws.*

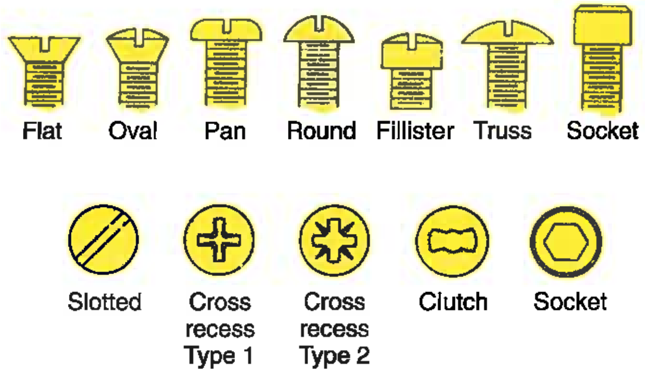


Figure 7-7. *A sampling of the many types of machine screws available.*

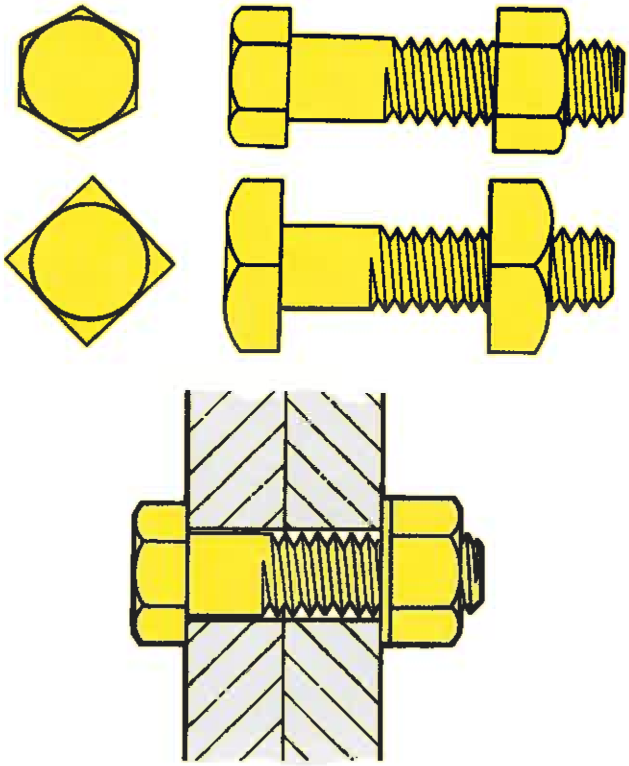


Figure 7-8. *Machine bolts use a nut to produce clamping force.*

7.1.3 Cap Screws

Cap screws are found in assemblies requiring a higher quality and a more finished appearance, **Figure 7-9**. Instead of tightening a nut to develop clamping action, as with the machine bolt, the cap screw passes through a clearance hole in one of the pieces and screws into a threaded hole in the other part. Clamping action is accomplished by tightening the bolt into the threaded part.

Cap screws are held to much closer tolerances in their manufacture than machine screws. They are provided with a machined or semifinished bearing surface under the head. Some type of cap screws are heat-treated.

The application determines the strength of the cap screw to be used. Required strength is indicated on the print. Since all steel hex head cap screws are similar in appearance, a series of markings on the bolt head indicate strength capabilities. See **Figure 7-10**. The stronger the cap screw, the more expensive it is.

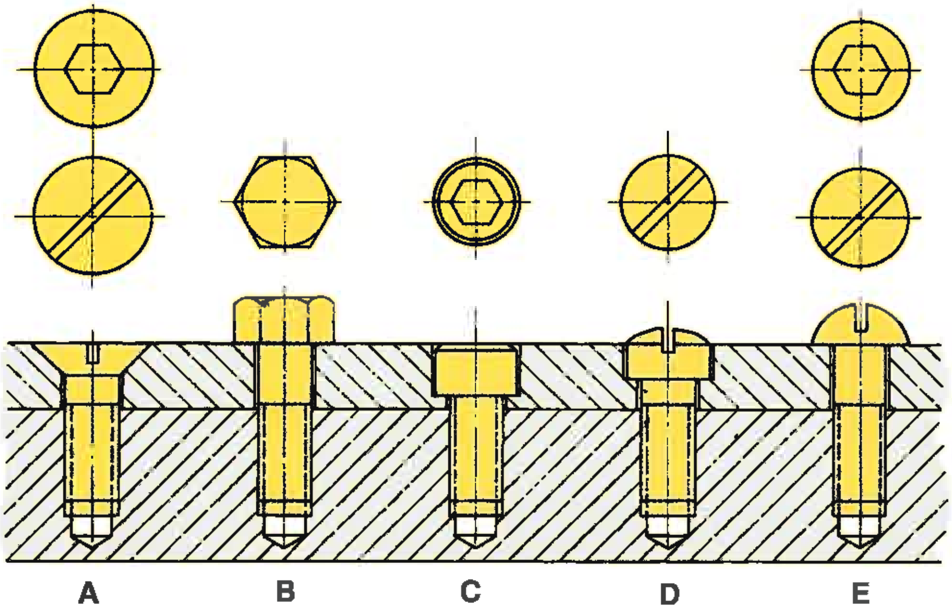
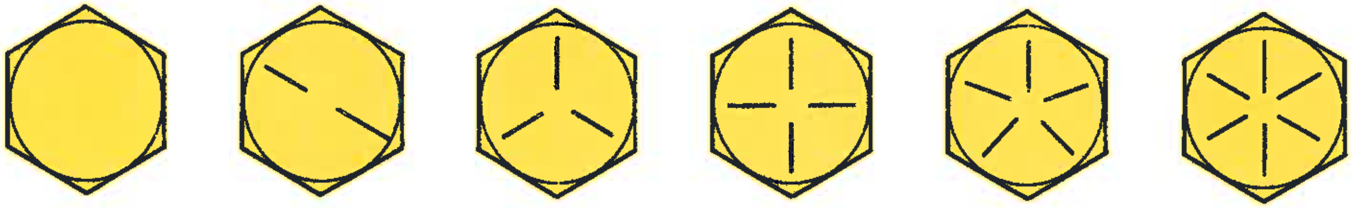
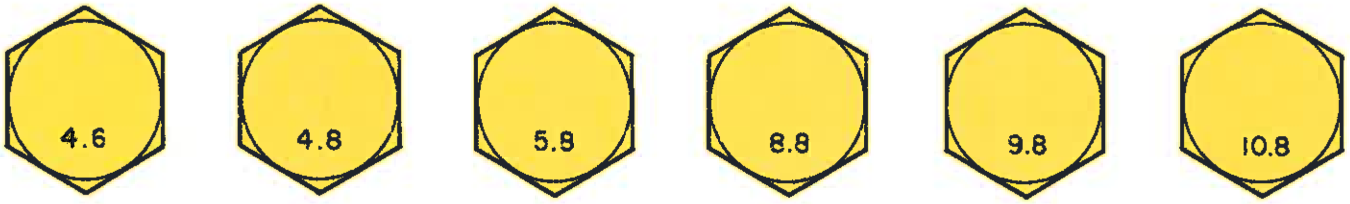


Figure 7-9. Cap screws are manufactured in various types. A—Flat head. B—Hexagonal head. C—Socket head. D—Fillister head. E—Button or round head.



Customary (inch) bolts



Metric bolts

Figure 7-10. Identification marks (inch size) and class numbers (metric size) are used to indicate the relative strength of hex head cap screws. As identification marks increase in number, or class numbers become larger, increasing strength is indicated.

Cap screws are stocked in coarse and fine thread series and in diameters from 1/4″ to 2″. Lengths from 3/8″ to 10″ are available. Metric sizes can also be supplied.

7.1.4 Setscrews

Setscrews are semipermanent fasteners that are used for such applications as preventing pulleys from slipping on shafts, holding collars in place on assemblies, and positioning shafts on assemblies. See **Figure 7-11**. Setscrews are usually made of

Figure 7-11. A typical setscrew application, to hold a gear onto a shaft.

heat-treated steel. They are classified in two ways, by head style, and by point style, **Figure 7-12**.

The ***thumbscrew*** is a variation of the setscrew that can be turned by hand. It is typically used in place of a setscrew for assemblies that require rapid or frequent disassembly, **Figure 7-13**. Thumbscrews are available with points similar to those on setscrews.

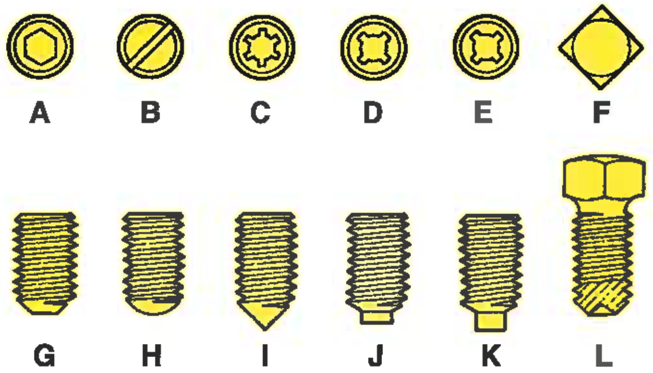


Figure 7-12. *Setscrew head and point designs. A—Socket head. B—Slotted head. C, D, and E—Fluted head. F—Square head. G—Flat point. H—Oval point. I—Cone point. J—Half dog point. K—Full dog point. L—Cut point.*

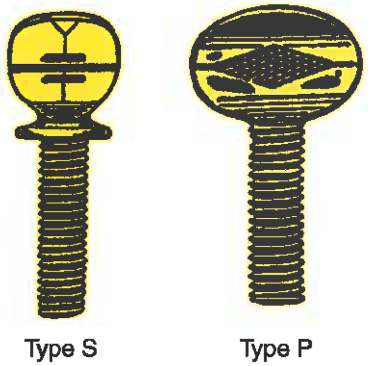


Figure 7-13. *Thumb screws can be removed or installed by hand. (Parker-Kalon)*

7.1.5 Stud Bolts

Stud bolts are headless bolts that are threaded for their entire length, or (more commonly) on both ends, **Figure 7-14**. One end is designed for semipermanent installation in a tapped hole; the other end is threaded for standard nut assembly to clamp the pieces together.

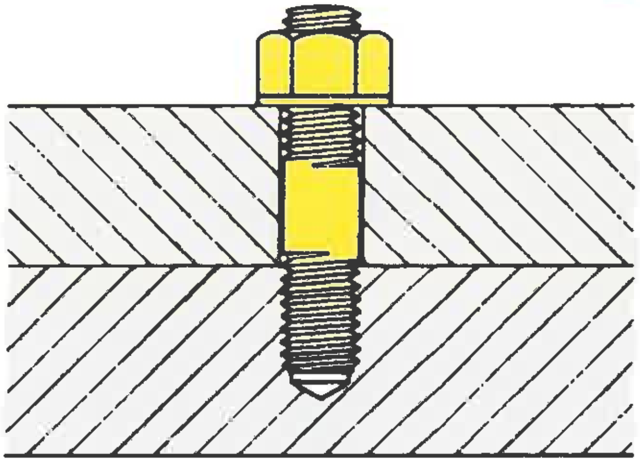


Figure 7-14. *One end of stud bolt usually threads into the part, while other end accepts a nut.*

Figure 7-15. *The Drill-Out Power Extractor™ combines a drill with an adjustable extractor collar. The drill makes the required-size hole, then the power drill is reversed. This causes the extractor to grip the broken bolt and torque it out. (Alden Corp.)*

7.1.6 Removing Broken or Sheared Bolts

Bolts that have broken or sheared off can be hard to remove without proper tools. The ***drill-out power extractor***, **Figure 7-15**, is available in several sizes. It is used with a 3/8 in. capacity variable speed/reversing power drill. The built-in drill cuts the proper size hole for the extractor unit to fit. After the hole is drilled and the extractor is placed into position, drill speed is reduced and reversed. This will remove the broken bolt.

The type of extractor shown in **Figure 7-16** is also available in several sizes. A chart furnished with the extractor indicates drill size to be used. After the hole has been drilled, the extractor is inserted and turned counterclockwise with an appropriate size tap wrench.

Extractors of the type shown in **Figure 7-17** are designed to remove sheared machine screws. The blade is made of heat-treated tool steel. A hole is drilled in the screw, then the tapering blade is

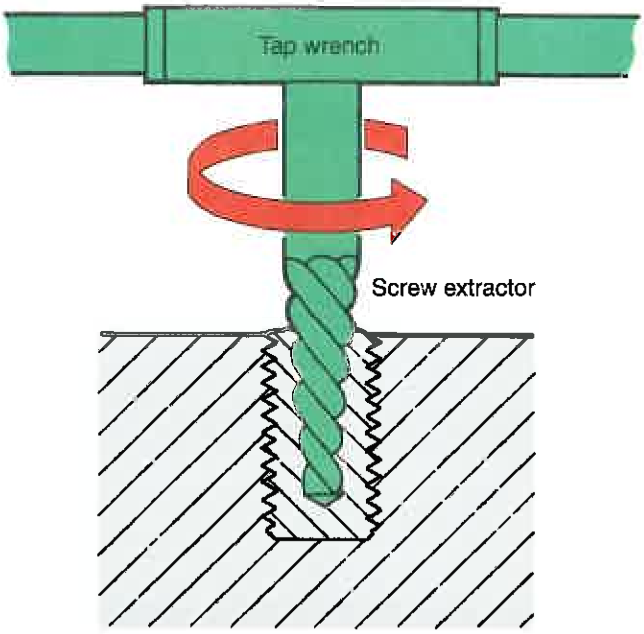


Figure 7-16. Spiral-flute broken bolt extractor. A hole of proper size is drilled, a tap wrench applied, and the broken bolt is turned out.

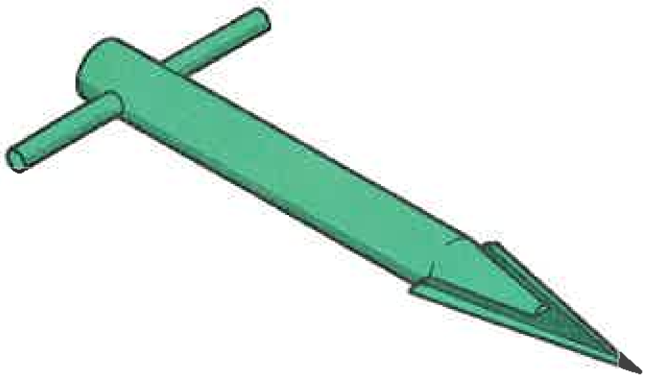


Figure 7-17. An extractor designed to remove machine screws. A hole is drilled in the broken-off screw. The extractor blade is tapped into place and carefully turned to remove broken screw.

lightly driven into the hole. The extractor is turned counterclockwise to remove the screw.

Treatment with penetrating oil will often make it easier to remove stubborn sheared bolts and machine screws.

7.1.7 Nuts

For most threaded fasteners, ***nuts*** with hexagonal or square shapes are used with bolts having the same-shape head. They are available in various degrees of finish. Nuts are usually manufactured of the same materials as their mating bolts.

A ***regular nut*** is ***unfinished*** (not machined), except on the threads, **Figure 7-18**.

A ***regular semifinished nut*** is machined on the bearing face to provide a truer surface for the washer, **Figure 7-19.**

A ***heavy semifinished nut*** is identical in finish to the regular semifinished nut. However, the body is thicker for additional strength, **Figure 7-20**.

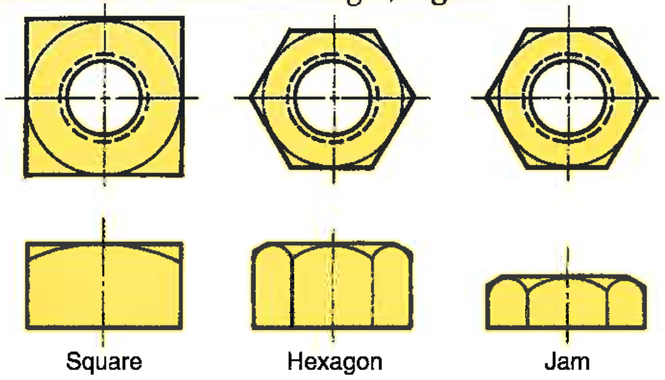


Figure 7-18*. Only the threads of regular nuts are machined.*

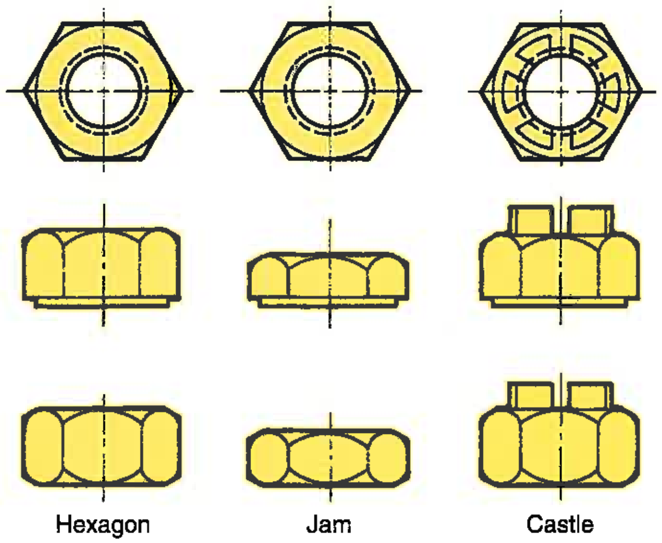


Figure 7-19*. Regular semifinished nuts have a machined bearing surface.*

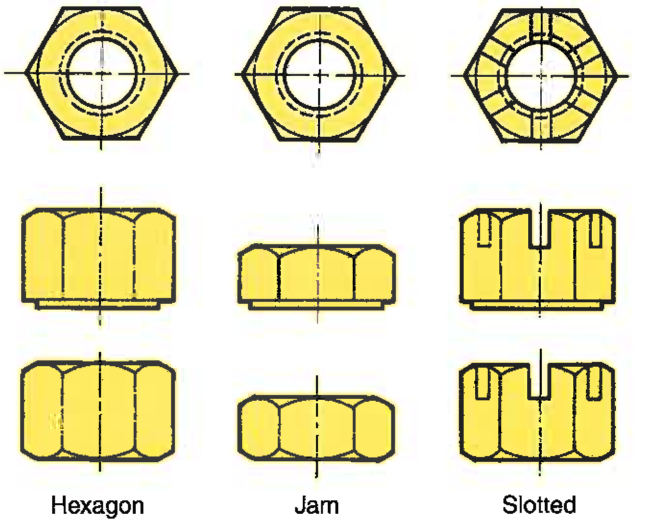


Figure 7-20*. Heavy semifinished nuts are thicker than regular nuts.*

The *jam nut* is thinner than the standard nut. It is frequently used to lock a full-size nut in place.

Castellated and ***slotted nuts*** have slots across the flats so they can be locked in place with a cotter pin or safety wire. A hole is drilled in the bolt or stud, and a cotter pin or wire is inserted through the slot and hole to prevent the nut from turning loose.

These types of locking nuts are being replaced on many applications by ***self-locking nuts***, **Figure 7-21.** Self-locking nuts are slightly deformed to produce a friction fit, or have a nylon insert, so they cannot vibrate loose. No hole through the bolt is required when self-locking nuts are employed in an assembly.

In critical assemblies, use a *new* self-locking nut to replace one that has been removed for any reason. The used nut may not have adequate locking action remaining and may loosen in service.

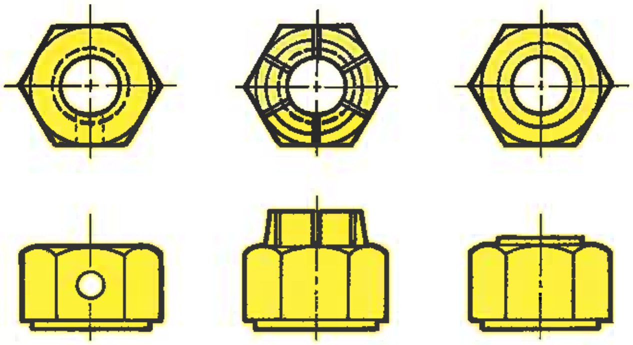


Figure 7-21. *Self-locking nuts.*

Acorn nuts are used when appearance is of primary importance, or where projecting threads must be protected. They are available in high or low crown styles. See **Figure 7-22.**

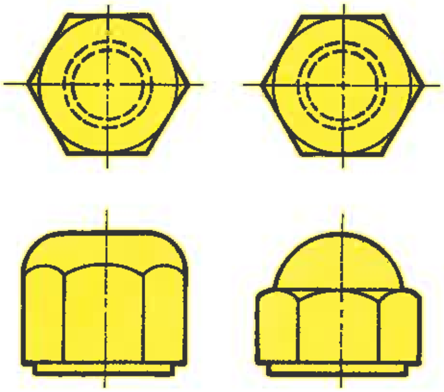


Figure 7-22. *Acorn or cap nuts look good and will protect threads.*

The ***wing nut*** is found where frequent adjustment or frequent removal is necessary. It can be loosened and tightened rapidly without the need of a wrench. Refer to **Figure 7-23.**

Figure 7-23. *Wing nut looks as if it has "wings." Like the thumbscrew, it is turned by hand. (Parker-Kalon)*

7.1.8 Inserts

An ***insert*** is a special form of nut or internal thread. Inserts are designed to provide higher strength threads in soft metals and plastics. The types shown in **Figure 7-24** are frequently used to replace damaged or stripped threads. The threaded hole is drilled and tapped. The insert is then screwed into the hole. Its internal thread is standard size and form. For optimum results, inserts must be installed according to the manufacturer's instructions.

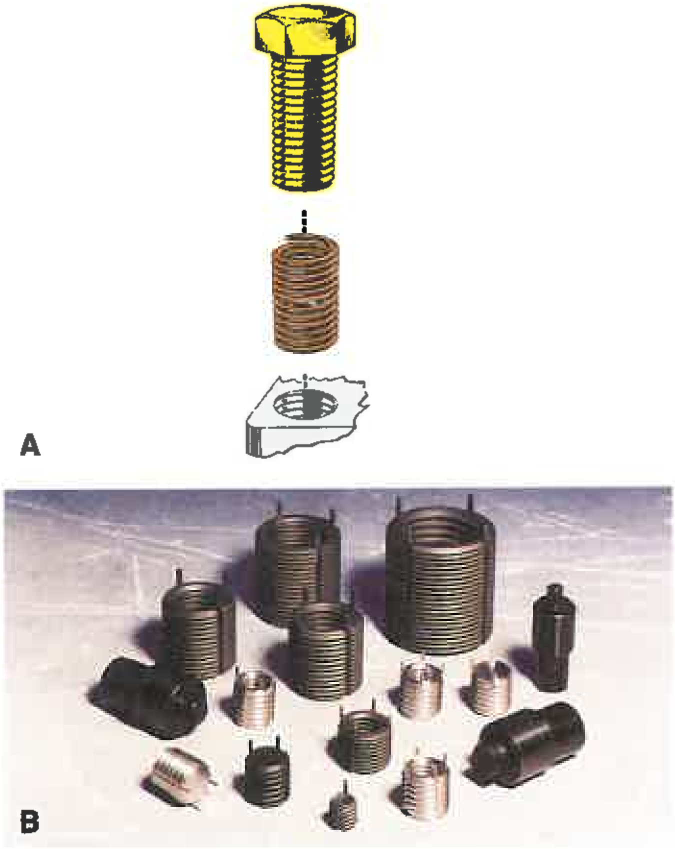


Figure 7-24. *Thread repair inserts. A—An insert is frequently used to replace damaged or stripped threads in a part. (Heli-Coil Corp.) B—These keylocking inserts can be easily installed or removed without special tools. They are available in both carbon steel and stainless steel. (Jergens, Inc.)*

7.1.9 Washers

Washers provide an increased bearing surface for bolt heads and nuts, distributing the load over a larger area. They also prevent surface marring. The ***standard washer*** is produced in light, medium, heavy-duty, and extra heavy-duty series. See **Figure 7-25**.

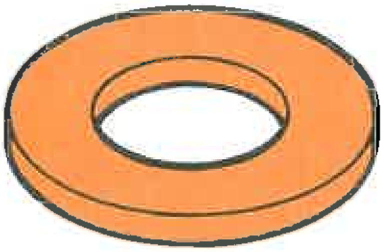


Figure 7-25. The standard flat washer provides a bearing surface for a fastener.

7.1.10 Lock Washers

A ***lock washer*** will prevent a bolt or nut from loosening under vibration. The ***split-ring lock washer*** is rapidly being replaced by the ***tooth-type lock washer,*** which has greater holding power on most applications. See **Figure 7-26.**

Preassembled lock washer and screw nuts and ***lock washer and nut units*** have a washer mounted on the nut. They are employed to lower assembly time and reduce waste in the mass-assembly market, **Figure 7-27.**

Figure 7-27. Lock washer and screw units and lock washer and nut units are frequently used to simplify assembly. (Shakeproof Div., Illinois Tool Works, Inc.)

Figure 7-28. There are many liquid thread locks available. They prevent a bolt, nut, or screw from vibrating loose, but allow easy removal should disassembly be required.

7.1.11 Liquid Thread Lock

Nuts, bolts, and machine screws can be prevented from loosening due to vibration though use of a liquid thread lock, **Figure 7-28.** Although the thread lock material will prevent fasteners from vibrating loose, it allows easy removal of the fastener should disassembly be necessary. When using a liquid thread lock, follow the manufacturer's recommendations for maximum effectiveness.

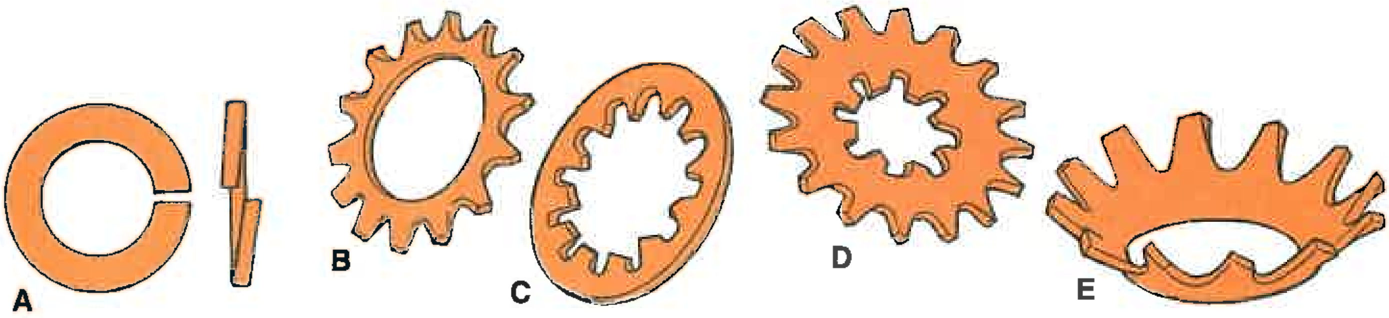


Figure 7-26. Lock washer variations. A—Split-ring type. B—External type. This type should be used whenever possible, because it provides greatest resistance to turning. C—Internal type. It is used with small head screws and to hide teeth, either for appearance or to prevent snagging. D—Internal-external type. It is employed when mounting holes are oversize. E—Countersunk type. It is used with flat or oval head screws.

7.1.12 Thread-Forming Screws

Thread-forming screws produce a thread in the part as they are driven, **Figure 7-29.** This feature eliminates a costly tapping operation. A variation of the thread-forming screw eliminates expensive hole-making (drilling or punching) and aligning operations because the screw drills its own hole as it is driven into place. See **Figure 7-30.**

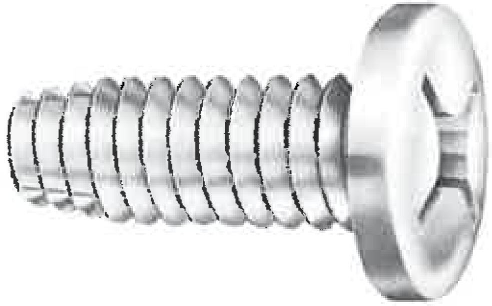


Figure 7-29. *One type of thread-forming screw.*

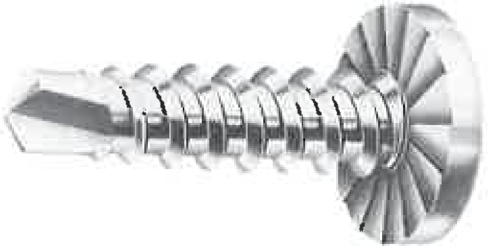


Figure 7-30. *A self-drilling screw. This type is also known as TEKS®. (USM Corp., Fastener Group)*

7.1.13 Thread Cutting Screws

Thread cutting screws differ from thread forming screws because they actually cut threads into the material when driven. Refer to **Figure 7-31.** Thread cutting screws are hardened. They are employed to join heavy-gage sheet metal, and to thread into nonferrous metal assemblies.

7.1.14 Drive Screws

Drive screws are simply hammered into a drilled or punched hole of the proper size. A permanent assembly results, **Figure 7-32.**

7.2 NONTHREADED FASTENING DEVICES

Nonthreaded fasteners comprise a large group of mechanical holding devices. These include dowel pins, cotter pins, retaining rings, rivets, and keys. Each has its advantages.

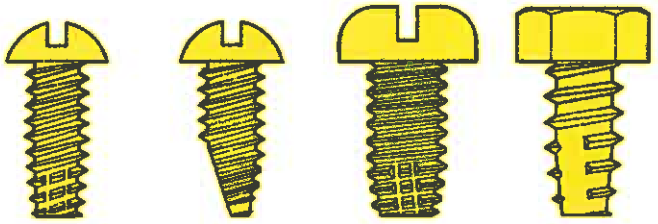


Figure 7-31. *Variations among thread cutting screws.*

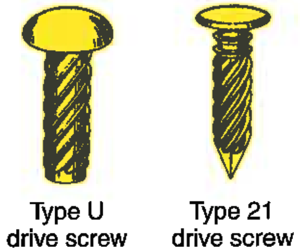


Figure 7-32. *Drive screws are hammered or forced into place in a presized hole. (Parker-Kalon)*

7.2.1 Dowel Pins

Dowel pins are made of heat-treated alloy steel and are found in assemblies where parts must be accurately positioned and held in absolute relation to one another. See **Figures 7-33** and **7-34.** They assure perfect alignment and facilitate quicker disassembly and reassembly of parts in exact relationship to each other. They are fitted into reamed holes and are available in diameters from 1/16″ to 1″. They are also available in metric sizes.

Figure 7-33. *Types of dowel pins. They are made in a wide range of sizes and types. (Driv-lok Inc.)*

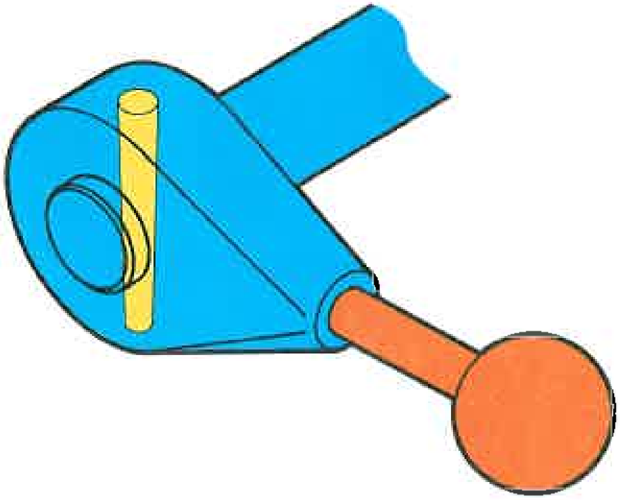


Figure 7-34. A taper pin is often used to lock a handle to its shaft.

Dowel pins are normally 0.0002" (0.005 mm) oversize (identified by a plain steel finish) but are available in 0.001" (0.025 mm) oversize (identified by a black finish) for repairs.

Taper pins are made with a uniform taper of 1/4" per foot in lengths up to 6", with diameters as small as 5/32" at the large end.

7.2.2 Cotter Pins

A ***cotter pin*** is fitted into a hole drilled crosswise through a shaft, **Figure 7-35**. The pin prevents parts from slipping or rotating off. Other types of retaining devices are replacing the cotter pin.

7.2.3 Retaining Rings

The ***retaining ring,*** **Figure 7-36,** has been developed for both internal and external applications. Retaining rings reduce both cost and weight of the product on which they are employed. While most retaining rings must be seated in grooves, **Figure 7-37**, a *self-locking* type does not require this special recess. Special pliers are needed for rapid installation and removal of the retaining rings, **Figure 7-38.**

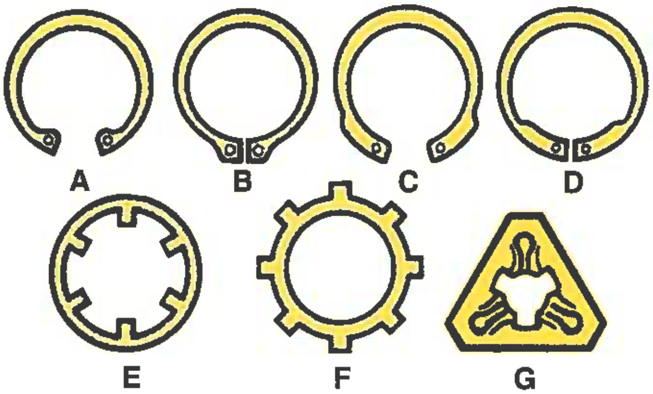


Figure 7-36. Retaining rings. A—Basic internal ring. B—Basic external ring. C—Inverted internal ring. D—Inverted external ring. E—External self-locking ring. F—Internal self-locking ring. G—Triangular self-locking ring.

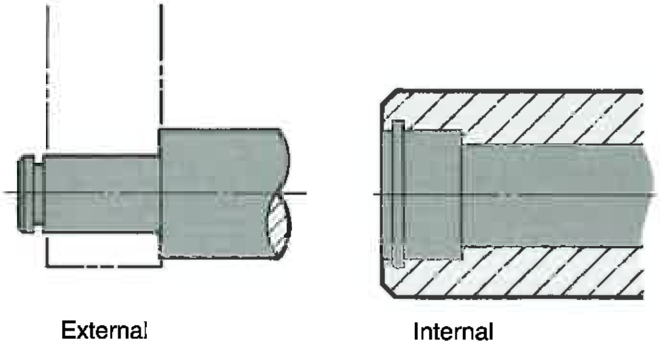


Figure 7-37. Grooves are machined into parts to receive retaining rings. They eliminate many other expensive machining operations.

7.2.4 Rivets

Permanent assemblies (those that do not have to be taken apart) can be made with rivets, **Figure 7-39**. Solid rivets can be *set*, or deformed to become larger on one end, by hand or machine methods.

Blind rivets are mechanical fasteners that have been developed for applications where the joint is accessible from only one side. They require special tools for installation, **Figure 7-40**. Blind rivet types are shown in **Figure 7-41**.

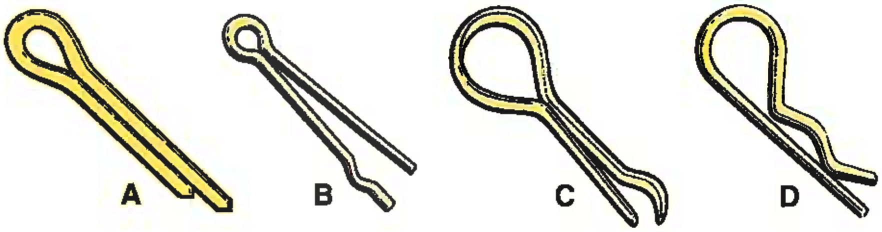


Figure 7-35. Types of cotter pins. A—Standard. B—Humped. C—Clinch. D—Hitch.

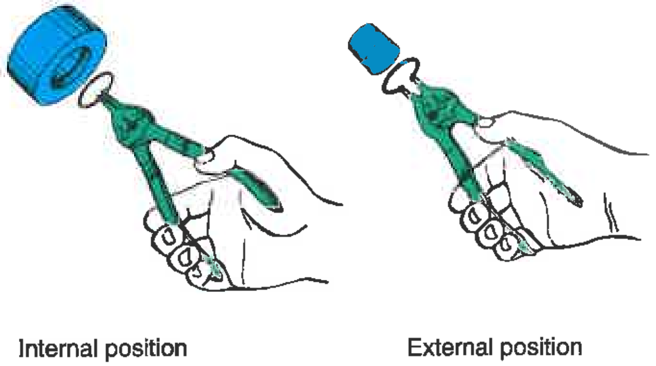


Figure 7-38. Special pliers are used to install some types of retaining rings. (Waldes Kohinoor Inc.)

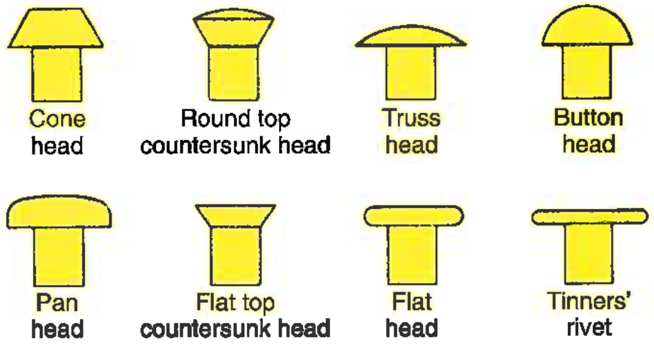


Figure 7-39. Rivet head styles.

Figure 7-40. A pliers-type rivet gun is used to insert one type of blind rivet.

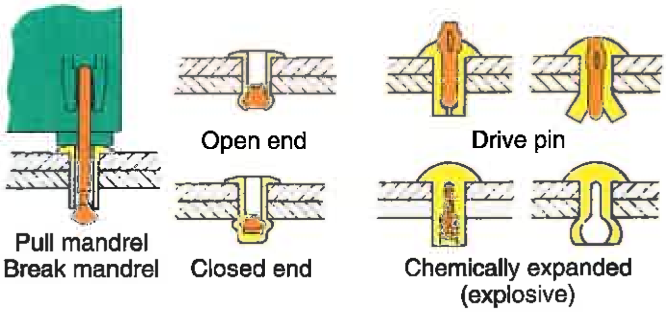


Figure 7-41. Types of blind rivets.

7.2.5 Keys

A ***key*** is a small piece of metal that prevents a gear or pulley from rotating on its shaft. One-half of the key fits into a ***keyseat*** on the shaft while the other half of the key fits into a ***keyway*** in the hub of the gear or pulley, **Figure 7-42.** Commonly used keys are shown in **Figure 7-43.**

A ***square key*** is usually one-fourth the shaft diameter. It may be slightly tapered on the top to make it easier to install.

The ***Pratt & Whitney key*** is similar to the square key, but is rounded at both ends. It fits into a keyseat of the same shape.

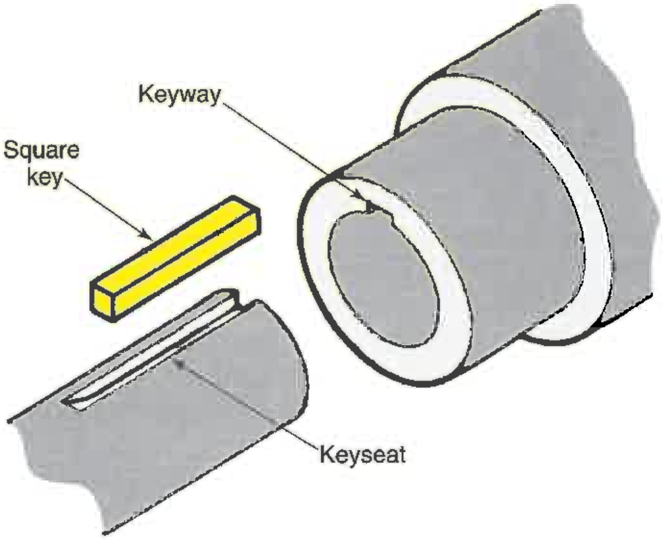


Figure 7-42. A square key is used to prevent a pulley or gear from rotating on its shaft.

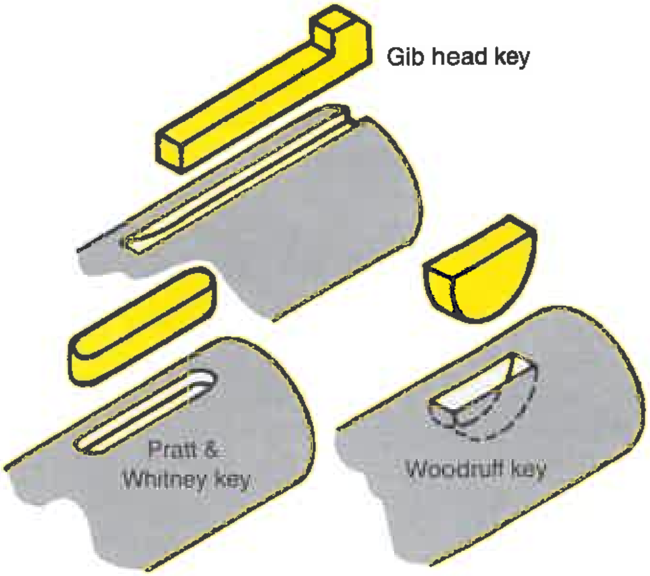


Figure 7-43. Three types of keys.

The ***gib head key*** is interchangeable with the square key. The head design permits easier removal from the assembly.

A ***Woodruff key*** is semicircular and fits into a keyseat of the same shape. The top of the key fits into the keyway of the mating part.

7.3 ADHESIVES

Adhesives provide one of the newer ways to join metals and to keep threaded fasteners from vibrating loose. In some applications, the resulting joints are stronger than the metal itself. Adhesive bonded joints do not require costly and time-consuming operations such as drilling, countersinking, riveting, etc.

The major drawback to the use of adhesives is heat. While some adhesives retain their strength at temperatures as high as 700°F (371°C), most should not be used for assemblies that will be exposed to temperatures above 150–200°F (66–93°C).

Adhesives for locking threaded fasteners in place are made in a number of chemical formulations. The desired permanence of the threaded joint will determine the type of adhesive to be employed.

Adhesive-bonded assemblies offer many advantages over other fastening techniques:

- The load is distributed evenly over the entire joined area, **Figure 7-44.**
- There is continuous contact between the mating surfaces, **Figure 7-45.**

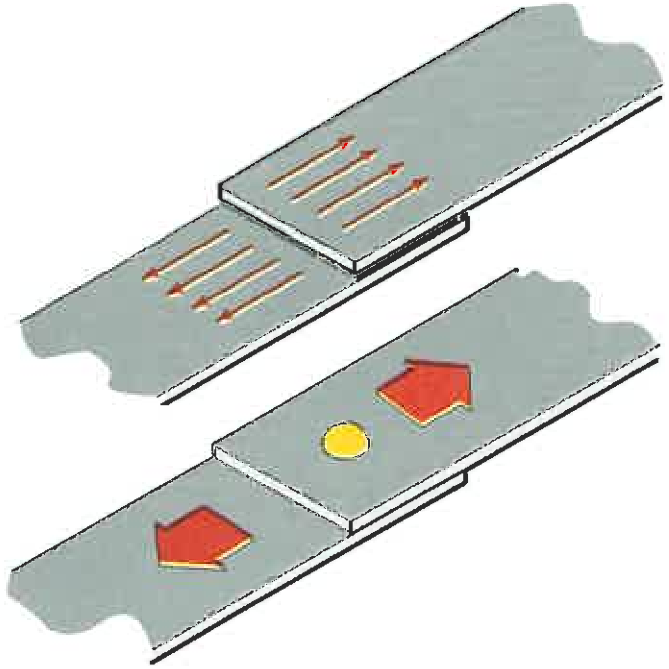


Figure 7-44. *When an adhesive is used to join metal, the load is distributed evenly over the entire joint. A rivet or conventional threaded fastener localizes the load in a small area.*

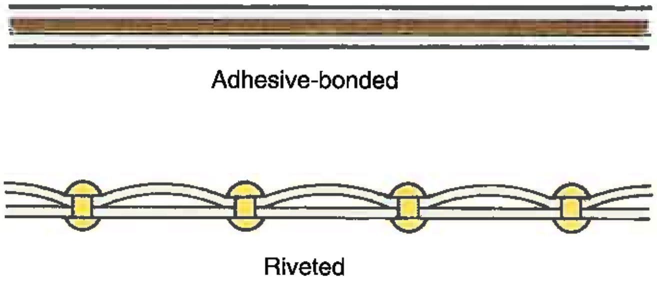


Figure 7-45. *On parts joined with an adhesive, the mating surfaces are in continuous contact.*

- Full strength of the mating parts is maintained, since holes do not have to be made to insert fasteners. The extreme heat required for joining methods like welding is not necessary with adhesive bonding. This means there is no danger of the work becoming distorted or having its heat treatment affected.
- Smooth surfaces result from adhesive bonding—there are no external projections (as with rivets or bolts), and the surface is not marred by the heat and pressure necessary to join pieces with spot welds.

Many commercial adhesives are sold in small quantities. They are suitable for use in training areas and the home, **Figure 7-46.**

Figure 7-46. *Adhesives for joining metal to metal and metal to other materials are available in good hardware stores. They are similar to those found in industry.*

Adhesives are available in liquid, paste, or solid form. Many can be applied directly from the container. Others must be mixed with a catalyst or hardener. A few pressure-sensitive adhesives are manufactured in sheet form.

One type of adhesive has found growing use in machining technology to make temporary bonds. ***Cyanoacrylate quick setting adhesives*** (known by such trade names as Eastman 910™, Super Glue™, and Crazy Glue™) are used to hold matching metal sections together while they are being machined. Round stock too small for existing collets can be glued into larger stock for turning, milling, or grinding. Some fragile parts have been glued to backup blocks for machining, and parts like the gun sight casting shown in **Figure 7-47** have been glued to a fixture for such operations as machining the sighting and bottom grooves.

After machining, the parts can be removed from the holding device by an application of heat (175°F or 79°C maximum). Very small parts can be removed by applying a cyanoacrylate debonder.

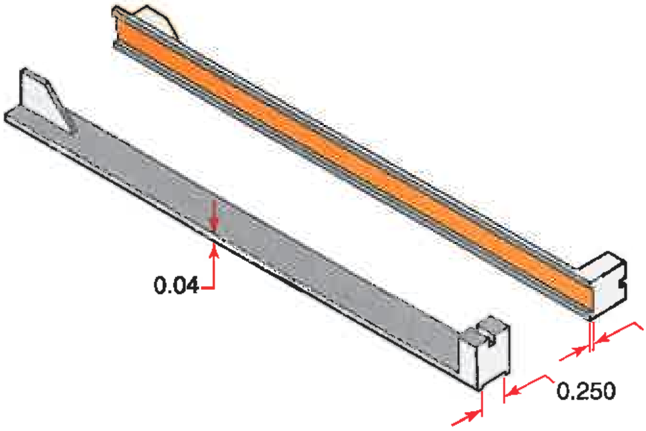


Figure 7-47*. This gunsight was held on a fixture with cyanoacrylate adhesive to allow machining. The thin base section of the part made it difficult and expensive to mount to a fixture by other means.*

For successful use of cyanoacrylate adhesives, the part and mounting surface must be prepared according to the adhesive manufacturer's directions.

When using cyanoacrylate adhesives, always wear approved eye protection and keep fingers away from your eyes and mouth. Since this adhesive can instantly bond fingers to each other or to other surfaces, always have a debonder available for immediate use. Should you get adhesive in your eyes, see a physician immediately.

7.3.1 Using Adhesives

Most adhesives require following a five-step process to produce solidly bonded joints:

1. Surface preparation is critical! All adhesives require clean surfaces to produce full-strength bonds. Preparation may range from simply wiping surfaces with a solvent to performing multistage cleaning and chemical treatment.
2. Adhesive preparation must be done properly. Mixing, delivery to the work area, setting up equipment, etc. must all be done according to the manufacturer's directions.
3. Adhesive application may be done by brushing, rolling, spraying, dipping, or methods designed for a specific assembly. See **Figure 7-48**.
4. Assembly involves positioning of materials to be joined. This often requires the use of jigs or fixtures for alignment.

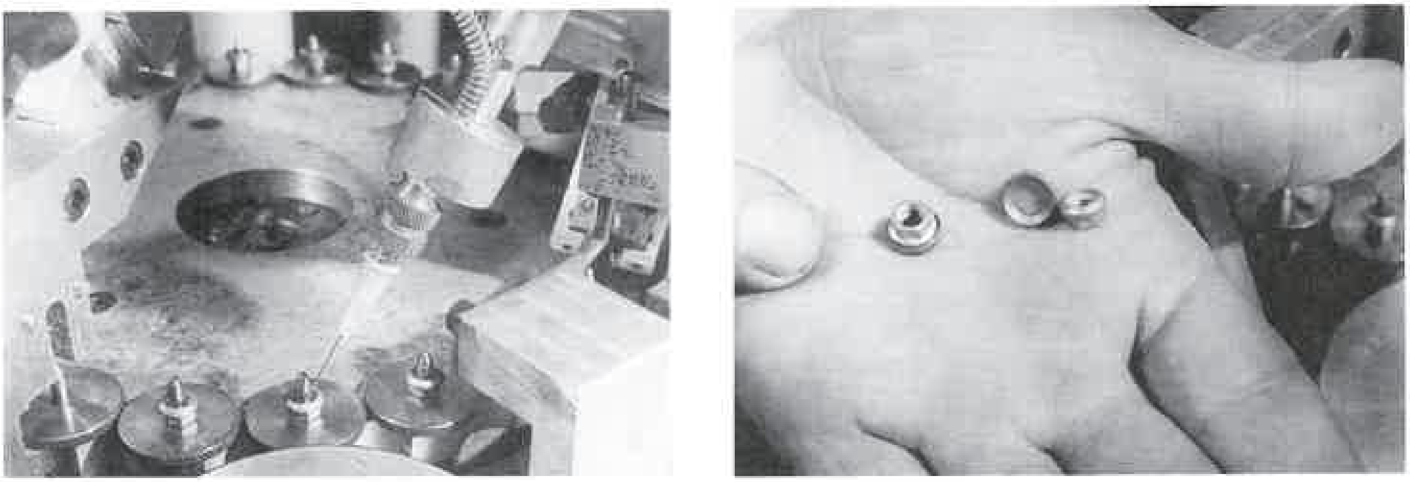


Figure 7-48*. Self-aligning nuts used in aircraft are assembled with a cyanoacrylate adhesive. The two part fasteners are assembled automatically and drop onto an indexing table. Setup produces 3600 assemblies an hour. Left—A needle-tip applicator applies a precise amount of adhesive before parts are brought together. Right—Parts after and before assembly. (Loctite Corp.)*

5. Bond development is the process of evaporation of solvents and curing of the adhesive. It may involve application of pressure and/or heat. See **Figure 7-49**.

***Figure 7-49.** This autoclave is used to bond large assemblies, with steam applying heat and pressure simultaneously. The assembly to be bonded is wrapped in a rubber blanket to protect the bond from moisture. (3M Co.)*

7.4 FASTENER SAFETY

- Wear approved eye protection when drilling, punching, or countersinking openings for fasteners.
- Never use your hands to remove metal chips from holes drilled for fasteners! Do not use your fingers to check whether burrs have been removed. Burrs around the opening can cause nasty cuts. Use a brush.
- If you use compressed air to clean drilled or tapped holes, wear approved eye protection. Make sure that there is no danger of flying chips injuring nearby workers.
- Do not apply adhesives near areas where there are open flames. Solvents used in some adhesives are highly flammable and/or toxic. Apply them only in well-ventilated areas, and wear a suitable respirator.
- The chemicals in adhesives for joining metals and other materials can cause severe skin irritation. To be safe, wear disposable plastic gloves when preparing or applying all types of adhesives.
- Carefully follow all instructions on the adhesive container when mixing and using adhesives. Only mix the amount you will need. Promptly remove any adhesive from your skin by washing in water.
- Cyanoacrylate adhesives cure in 5 to 15 seconds. Do not allow any of this adhesive to get onto your fingers, since it will bond skin together. Unless a suitable solvent is available, surgery might be needed to separate the joined fingers.

TEST YOUR KNOWLEDGE

Please do not write in the text. Write your answers on a separate sheet of paper.

1. For maximum strength, a threaded fastener should screw into its mating part a distance equal to _____ times the diameter of the thread.
2. There are many ways of joining material. List four types of threaded fasteners. Describe how each is used.
3. _____ screws are used for general assembly work.
4. How is the strength of hex-head cap screws indicated?
5. When removing stubborn sheared bolts, what can be done to make their removal easier?
6. To prevent a pulley from slipping on a shaft, a _____ is often employed.
7. The _____ bolt is threaded at both ends.
8. _____ or _____ are employed when the parts are to be joined permanently.
9. Why are lock washers used?
10. While most _____ must be seated in grooves, a self-locking type does not require the special recess.
11. When is a jam nut employed?
12. The shape of the _____ nut permits it to be loosened and tightened without a wrench.

Match each word in the left column with the most correct sentence in the right column. Place the appropriate letter in the blank.

13. _____ Rivet.

14. _____ Jam nut.

15. _____ Drive screw.

16. _____ Thread-cutting screw.

17. _____ Acorn nut.

18 _____ Dowel pin.

19. _____ Blind rivet.

20 _____ Keyway.

21. _____ Keyseat.

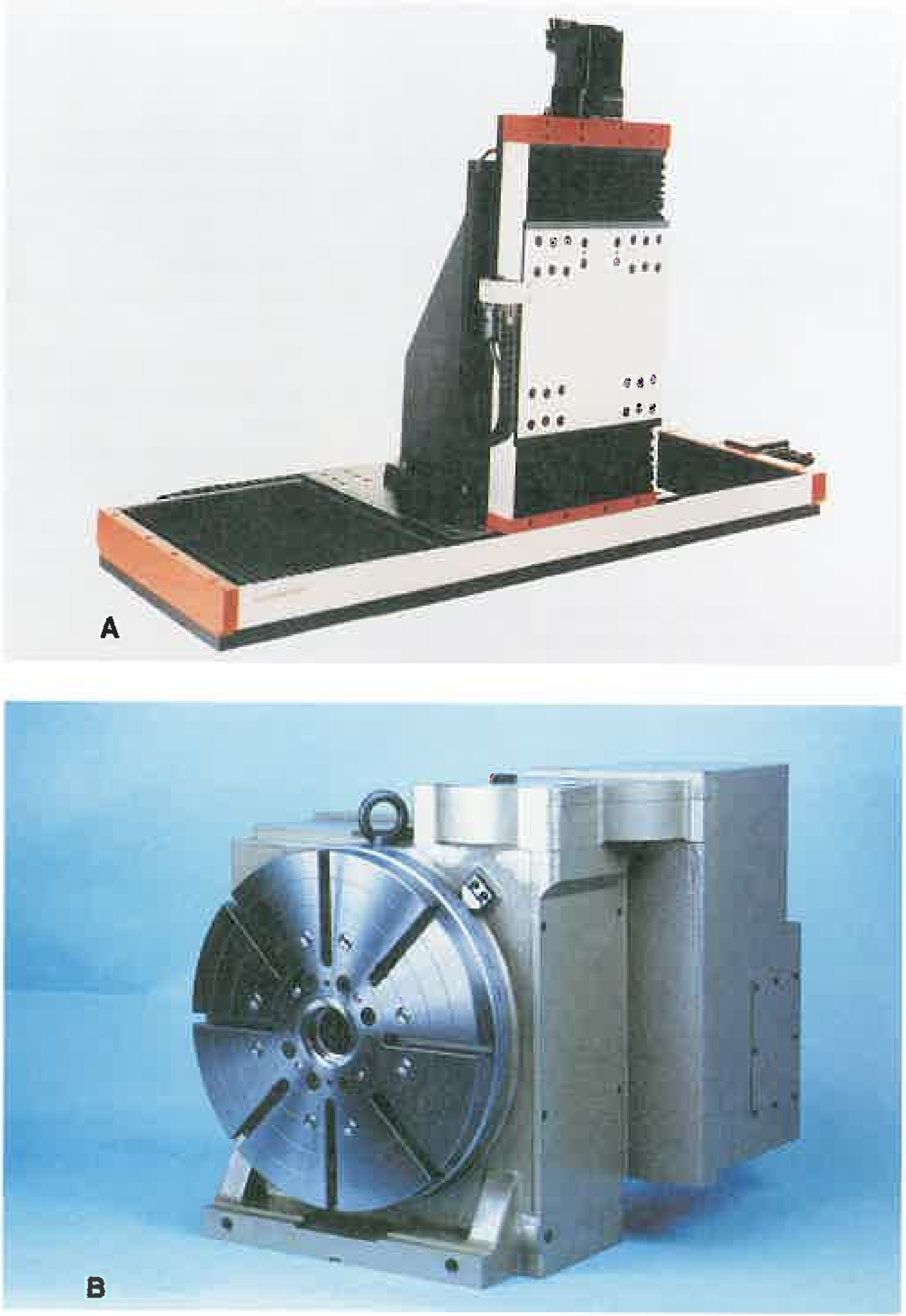
22. _____ Key.

23. List the steps, in their proper sequence, that must be used to join metals with adhesives.

24. List at least five safety precautions that must be observed when using fasteners.

a. Developed for use in confined area, where a joint is only accessible from one side.
b. Used where parts must be aligned accurately and held in absolute relation with one another.
c. Prevents a pulley or gear from slipping on a shaft.
d. Protects projecting threads.
e. Is hammered into a drilled or punched hole.
f. Used to make permanent assemblies.
g. Slot cut in gear or pulley to receive "c."
h. Locks a regular nut in place.
i. Eliminate costly tapping operations.
j. Slot cut in shaft to receive "c."

The Bell 609 is the latest design for transporting people in and out of inner city locations. After a helicopter-like take-off, the blades transition into a horizontal position and the 609 flies like a conventional aircraft. The blades are returned to an upright position for a vertical landing. In addition to the use of conventional-type fasteners, high-strength adhesives are used extensively in its manufacture.(Bell Helicopter/ A Subsidiary of TEXTRON, Inc.)



Positioning tables are used with fixtures and jigs to rapidly and accurately move the workpiece into the proper relationship with a cutting tool. A—This linear table moves in both the horizontal and vertical axes, using precision-ground ballscrews and guide rails to achieve positioning as accurate as ± 10 microns per 300 mm. (Schneeberger, Inc.) B—Rotary tables can be mounted vertically or horizontally, depending upon the application. They offer precise and repeatable indexing of the workpiece through a full 360° rotation. (Yukiwa Seiko USA, Inc.)

Chapter 8

Jigs and Fixtures

LEARNING OBJECTIVES

After studying this chapter, you will be able to:

- ○ Explain why jigs and fixtures are used.
- ○ Describe a jig.
- ○ Describe a fixture.
- ○ Elaborate on the classifications of jigs and fixtures.

IMPORTANT TERMS

box jig
bushings
closed jig
drill template
fixture
fixture holding devices
jig
open jig
plate jig
slip bushings

Jigs and fixtures are devices that are used extensively in production machine shops to hold work while machining operations are performed. They position the work and guide the cutting tool or tools so that all of the parts produced are uniform and within specifications. When large numbers of identical and interchangeable parts must be produced, the use of jigs and fixtures helps to reduce manufacturing costs. The use of jigs and fixtures is often justified when limited production is required, because they allow relatively unskilled workers to operate the machines.

Jigs and fixtures are also employed in assembly work for operations, such as welding or riveting. They position and hold work while standardized parts are being fabricated.

8.1 JIGS

A *jig* is a device that holds a workpiece in place and guides the cutting tool during a machining operation such as drilling, reaming, or tapping. Hardened steel ***bushings*** are used to guide the drill or cutting tool, **Figure 8-1**.

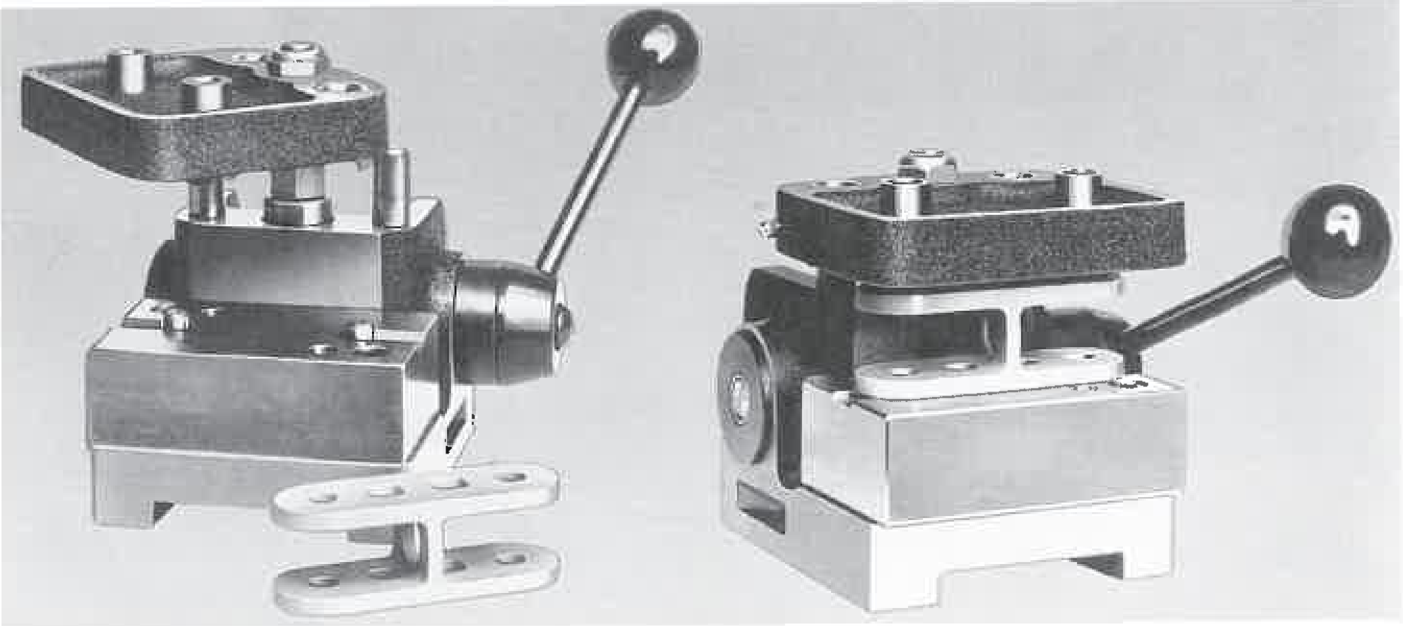


Figure 8-1. *A lift-type drill jig. Left—Jig is open to receive the part shown in foreground. Right—Jig is closed, with the part in place, ready for drilling. (Ex-Cell-O Corp.)*

The jig is seldom mounted solidly to the drill press table. For safety, however, it is usually nested between guide bars that *are* mounted solidly to the table, **Figure 8-2**.

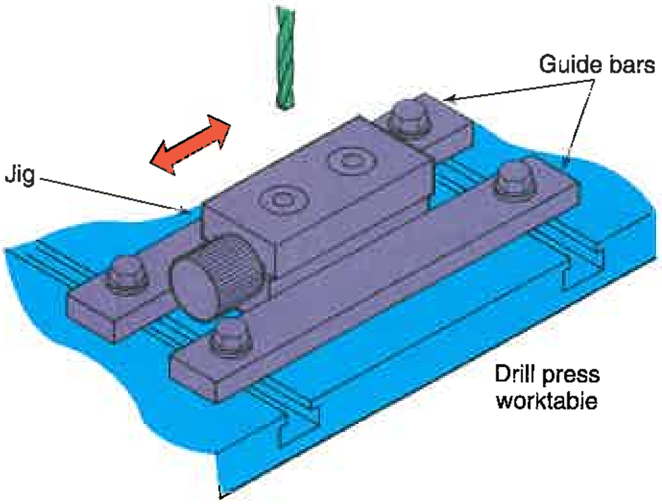


***Figure 8-2**. A drill jig is nested between guide bars to prevent dangerous and undesirable "merry-go-round" rotation.*

8.1.1 Jig types

Drill jigs fall into two general types: open jigs and box (closed) jigs.

The ***drill template*** or ***plate jig*** is the simplest form of the ***open jig***. It consists of a plate with holes to guide the drill. The jig fits over the work, **Figure 8-3**.

In a more elaborate form of open-type drill jig, **Figure 8-4,** clamps are used to hold the work in place. Drill jigs may be fitted with a base plate to provide clearance for the drill as it breaks through the work. Such a base plate is used on the circular jig shown in **Figure 8-5**. This jig is a variation of the plate jig.

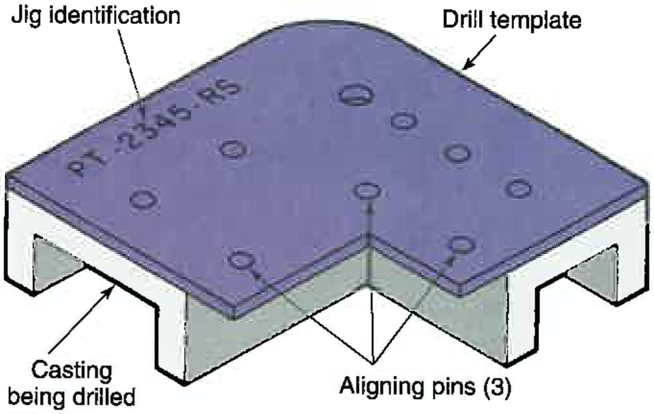


***Figure 8-3**. A simple drill template. Identification numbers on jigs and fixtures allow these devices to be located easily when stored away between uses.*

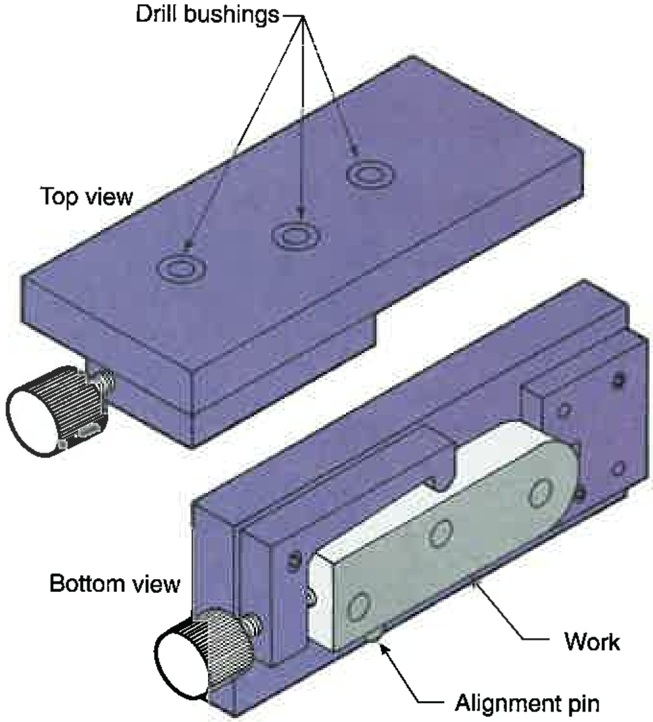


***Figure 8-4**. This open-type jig has a clamp to hold the work in position for drilling. A V-notch at one end and an alignment pin at the other end position the work properly in the jig.*

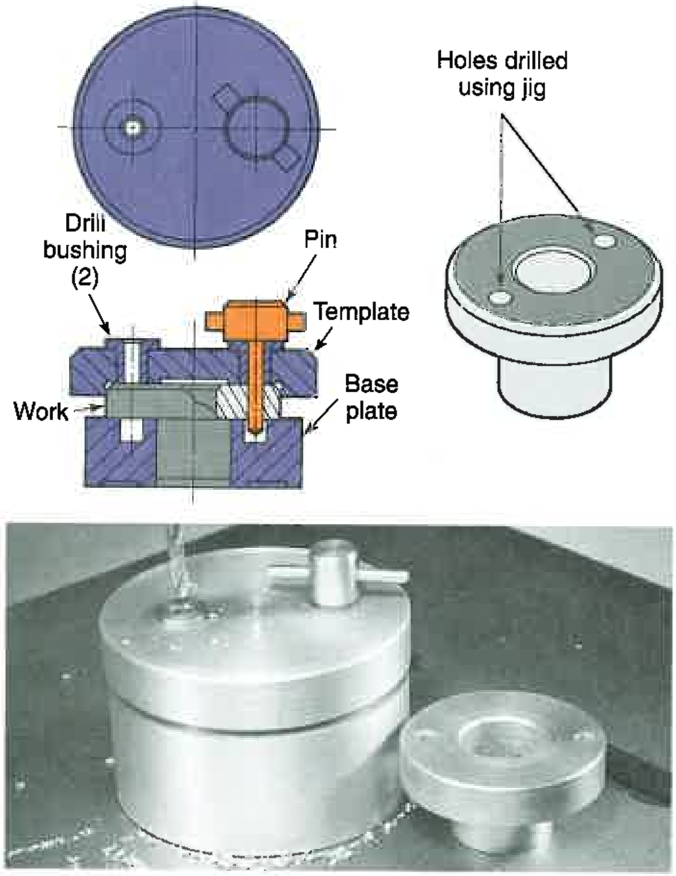


***Figure 8-5**. With this circular type drill jig, a pin is placed in first hole after it is drilled. This holds the workpiece in position when drilling the second hole. Note the base plate that provides clearance for the drill as it breaks through the workpiece.*

The **box jig** or **closed jig** encloses the work, **Figure 8-6**. This type is more costly to make than an open jig, but is often used when holes must be drilled in several directions. **Figure 8-7** illustrates a box jig in its simplest form. The work is fitted into the jig through a hinged or swinging cover. The clamps that hold the work in place are permanently mounted to the jig. A more complex type of box jig is shown in **Figure 8-8**.

When several different operations must be performed on a job, a combination of open and box jigs is often used. ***Slip bushings*** are utilized to guide the drills. They are then removed for subsequent operations such as reaming, tapping, countersinking, counterboring, or spot facing.

8.2 FIXTURES

A ***fixture*** is employed to position and hold a workpiece while machining operations are performed on it, **Figure 8-9**. Unlike a jig, a fixture does *not* guide the cutting tool.

Fixtures fall into many classifications. The class is determined by the type of machine tool on which the fixture is used, such as a machining center,

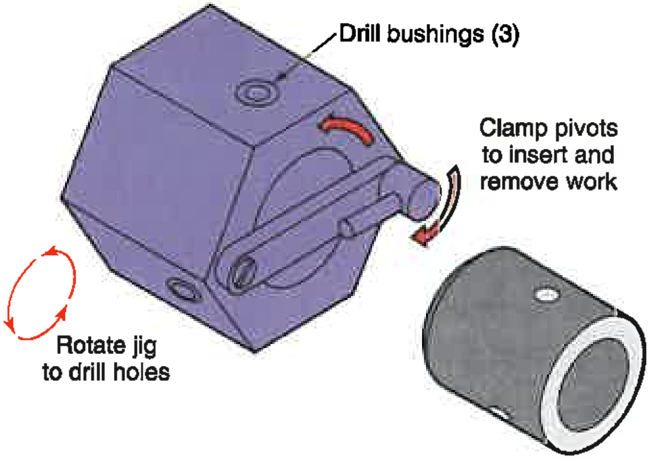


Figure 8-7. *A light closed (box) jig used to drill three equally spaced holes in a base end cap. Since only a limited production run was required, it was not necessary to construct a more elaborate jig.*

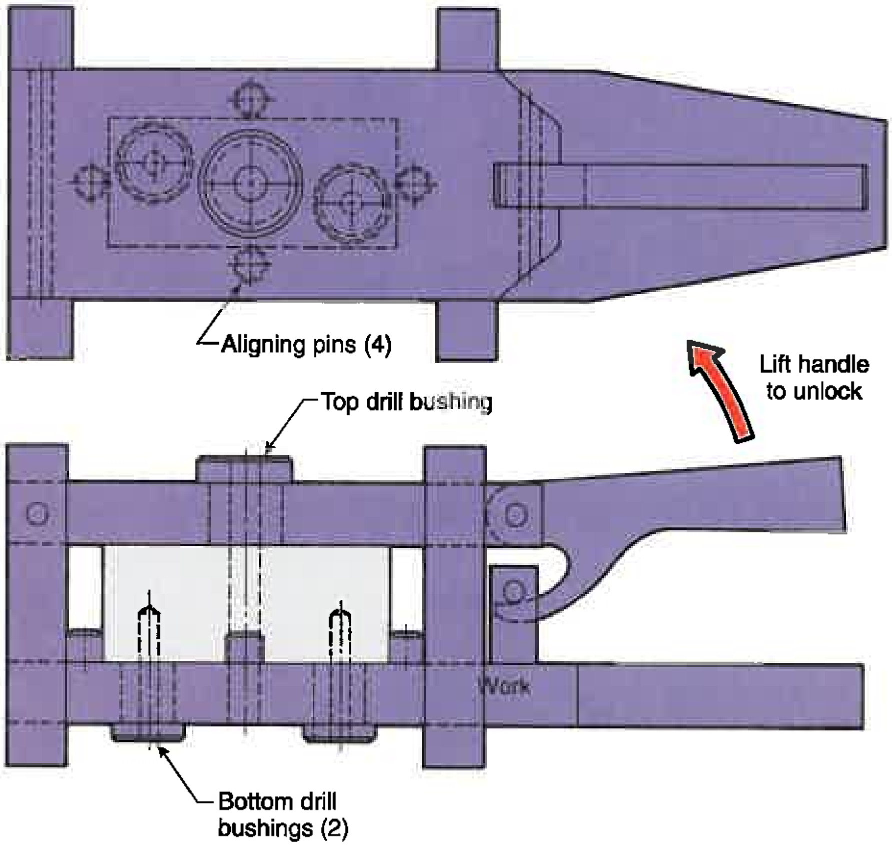


Figure 8-6. *A box type drill jig. Lowering the handle locks work in the jig.*

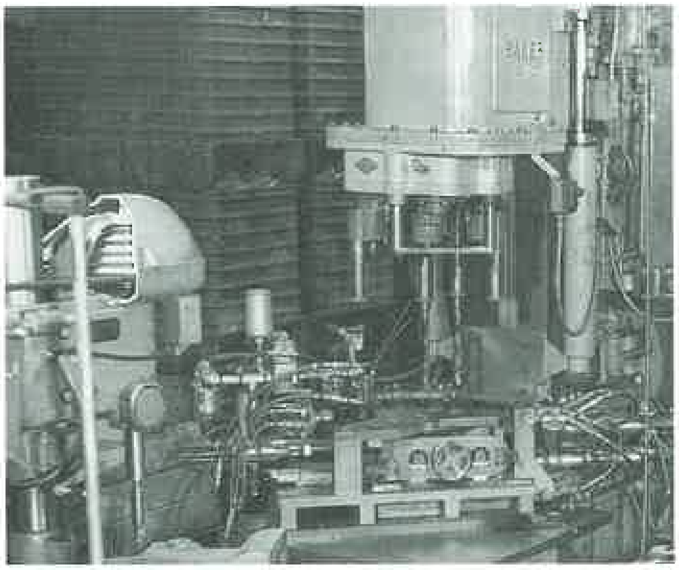


Figure 8-8. To hold a drill head casting in place for machining, a fairly complex jig is used. (Clausing Industrial, Inc.)

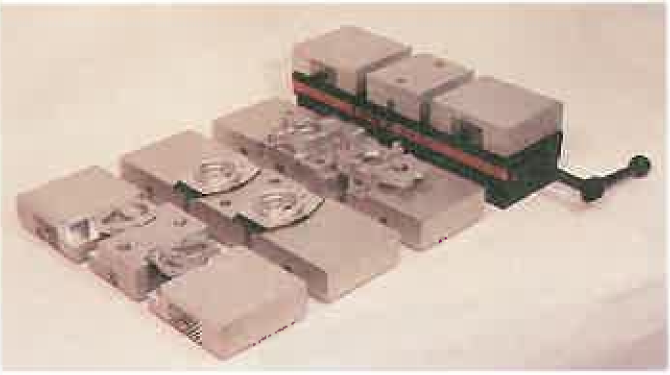


Figure 8-9. Machining centers often require special fixture holding devices. The pockets to hold the work are machined directly into the body of the jaws. Three examples are shown in the foreground. The jaws, which snap on and off the workholding system, are shown in the background. New setups can be made in a very short period of time. (Chick Machine Tool, Inc.)

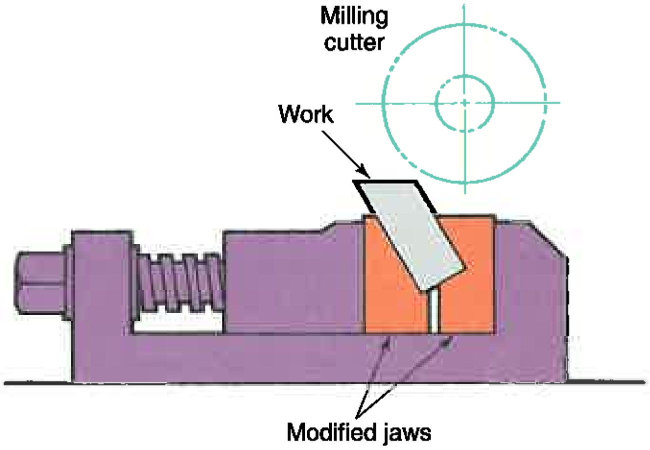


Figure 8-10. This simple fixture consists of vise jaws that have been modified to position a workpiece so that an angular cut can be made on it.

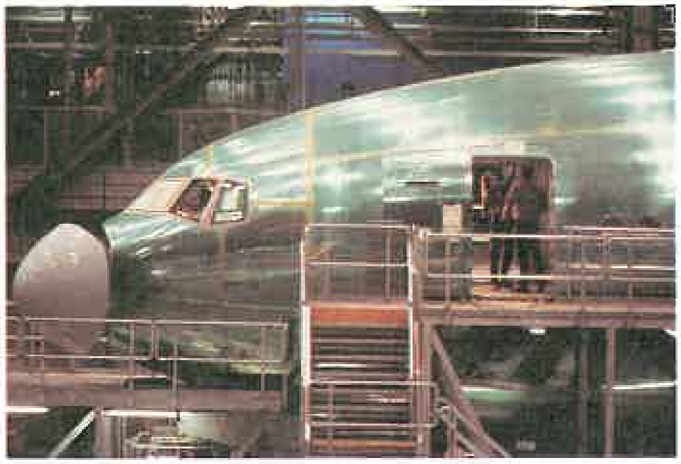


Figure 8-11. Many fixtures are used in the assembly of an aircraft fuselage. In fixture construction, extreme accuracy is critical, requiring use of lasers to assure precise alignment. (The Boeing Co.)

milling machine (vertical or horizontal), lathe, band saw, or grinder. Fixture designs range from simple vise jaw modifications, **Figure 8-10**, to very large and complex devices used by the aerospace industry, **Figure 8-11**.

8.3 JIG AND FIXTURE CONSTRUCTION

Jigs and fixtures are devices that are designed for *specific* jobs. Their complexity is determined by the number of pieces to be produced, the degree of accuracy required, and the kind of machining operations that must be performed.

The body of a jig or fixture may be built-up, welded, or cast. Commercial components are available in a wide range of sizes, types, and shapes. See **Figure 8-12.** Special ***fixture holding devices*** have been developed for machining centers and other CNC machine tools that permit multiple setups. See **Figures 8-13** and **8-14.**

***Figure 8-12**. Standard cast iron shapes are machined parallel and square to save time and money in both designing and building jigs and fixtures. Sections of different shapes can be bolted together to form complex holding devices. Two completed units are shown at bottom of illustration. (Ex-Cell-O Corp.)*

***Figure 8-13**. Special fixture holding devices for machining centers and other CNC machine tools. Workholding pockets are cut directly into the jaw blocks. Pivoting the vertical setup will bring the next set of workpieces into position. (Chick Machine Tool, Inc.)*

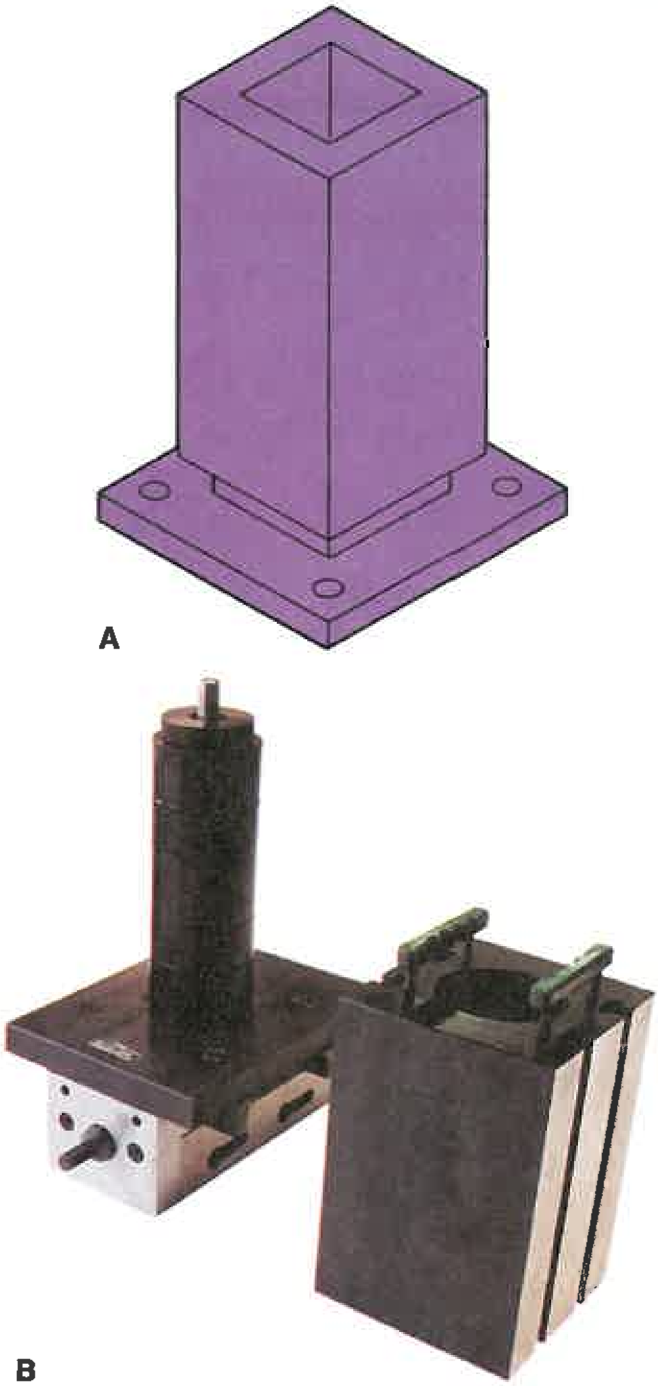
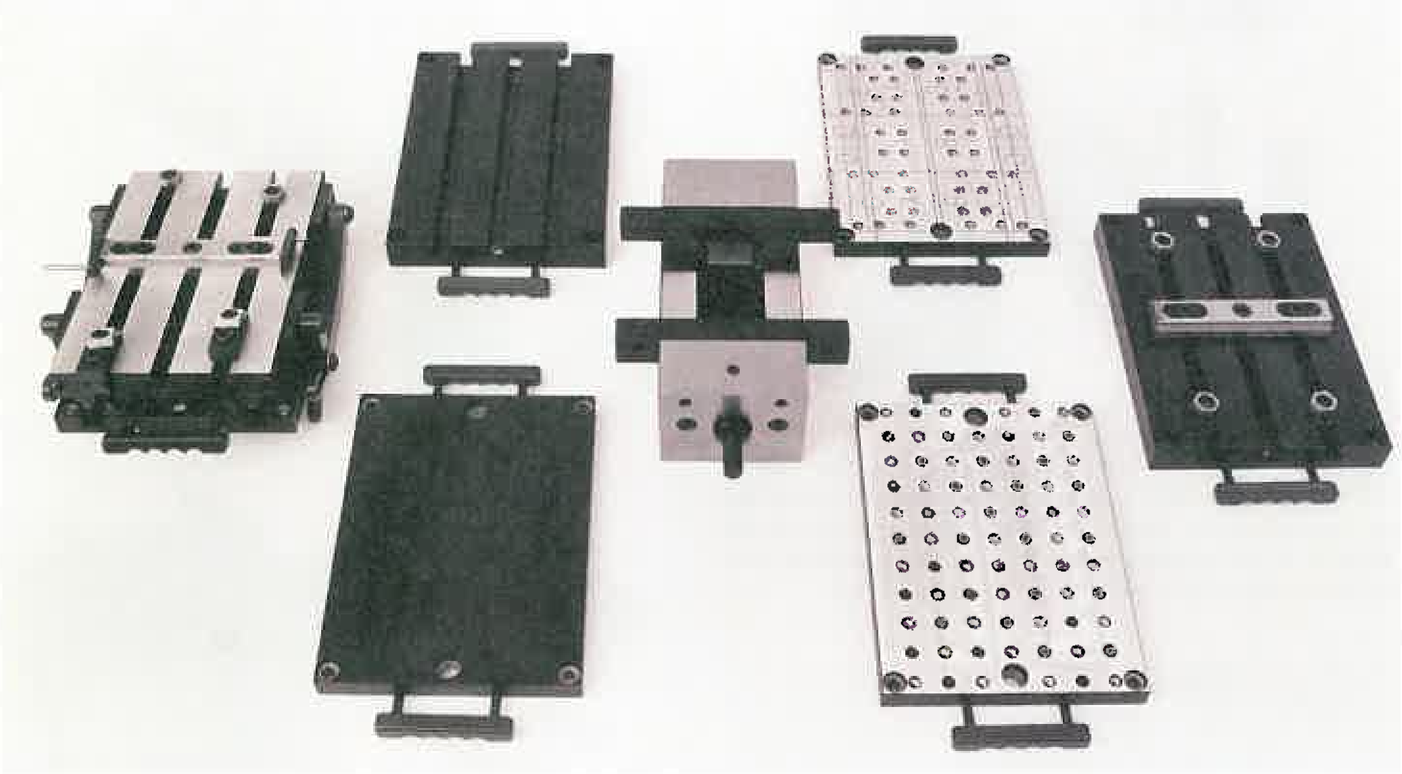


***Figure 8-14**. "Tombstones" are a common fixture-holding device used with machining centers and other CNC machine tools. They are made from heavy castings and are precisely machined. A—A typical tombstone intended for mounting on a machine's worktable. Fixtures are mounted directly onto the tombstones. The tombstone may be blank, as shown, have drilled and tapped mounting holes, or use T-slots for mounting fixtures. B—A modular tombstone system consisting of a receiver module and mating tombstone modules. An expansion mechanism on the receiver's centerpost allows the tombstone module to be rotated, then locked in position. The assembly at the bottom of the receiver module is a dovetail adapter that allows the assembly to be locked in a standard machine vise. The receiver module can also be bolted directly to the worktable. (Interlen Products Corporation)*

TEST YOUR KNOWLEDGE

Please do not write in the text. Write your answers on a separate sheet of paper.

1. Jigs and fixtures are devices used in _____ to _____ while machining operations are performed.
2. Why are jigs and fixtures used?
3. What is a jig?
4. Jigs fall into two general types. List and briefly describe each type.
5. A combination of the two jig types listed in question 4 is often used when _____.
6. What is a fixture?



Special quick-change pallets have been developed to lock into a receiver that mounts on a standard machine vise. Shown are some of the interchangeable pallets, which come in a variety of T-slot and grid-hole patterns for mounting different workpieces and fixtures. (Interlen Products corporation)

Chapter 9

Cutting Fluids

LEARNING OBJECTIVES

After studying this chapter, you will be able to:

- ○ Understand why cutting fluids are necessary.
- ○ List the types of cutting fluids.
- ○ Describe each type of cutting fluid.
- ○ Discuss how cutting fluids should be applied.

IMPORTANT TERMS

chemical cutting fluids
contaminants
cutting fluids
emulsifiable oils
gaseous fluid
lubricating
mineral oils
misting
noncorrosive
semichemical cutting fluids

Cutting fluids are required to do many things simultaneously. These functions include:

- Cooling the work and cutting tool, **Figure 9-1**.
- Improving surface finish quality.
- ***Lubricating*** to reduce friction and cutting forces, thereby extending tool life.
- Minimizing material buildup on tool cutting edges.
- Protecting machined surface against corrosion.
- Flushing away chips, **Figure 9-2**.

In addition, cutting fluids must comply with all federal, state, and local regulations for human safety, air and water pollution, waste disposal, and shipping restrictions.

Figure 9-1. *For maximum results, coolant should flood the area being machined and cutting tool to provide the most efficient removal of the heat generated. (Kesel/JRM International, Inc.)*

9.1 TYPES OF CUTTING FLUIDS

Cutting fluids fall into four basic types:

- Mineral oils.
- Emulsifiable oils.
- Chemical and semichemical fluids.
- Gaseous fluids.

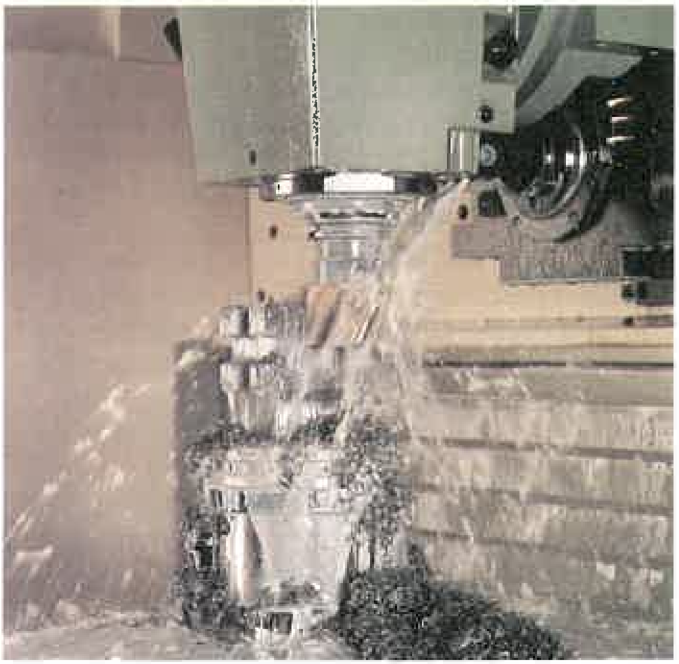


Figure 9-2. *An additional function of cutting fluids is to flush away chips from the area where cutting is taking place. (EROWA Technology, Inc.)*

9.1.1 Mineral Cutting Oils

Cutting oils made from mineral oil may be used straight or combined with additives. Straight ***mineral oils*** are best suited for light-duty (low speed, light feed) operations where high levels of cooling and lubrication are not required.

They are ***noncorrosive*** and are usually used with high machinability metals, such as aluminum, magnesium, brass, and free-cutting steel.

Mineral oils are often combined with animal and vegetable oils and contain sulfur, chlorine, and/or phosphorus. Their use is limited by high cost, operator health problems, and danger from smoke and fire. Mineral oils also stain some metals. They have a tendency to become rancid, so the tank containing them must be cleaned periodically and the fluid replaced.

When working in situations where cutting fluid mists or vapors are present, always wear an approved respirator. A simple dust mask is not sufficient protection.

9.1.2 Emulsifiable Oils

Emulsifiable oils are also known as *soluble oils.* They are composed of oil droplets that are suspended in water by blending the oil with emulsifying agents and other materials. Emulsifiable oils range in appearance from milky to translucent. They are available in many variations for metal removal applications that generate considerable heat.

Emulsifiable oils offer a number of advantages over straight cutting oils. They provide increased cooling capacity in some applications. They are cleaner to work with than other cutting fluids, and provide cooler and cleaner parts for the machinist to handle. These oils reduce the ***misting*** and fogging that are health hazards for machine operators. Because they are diluted with water, they offer increased economy and present no fire hazard.

Emulsifiable cutting oils can be used in most light- and moderate-duty machining operations. For economy and best machining results, these oils must be mixed according to the manufacturer's recommendations. These take into account the material being machined and the machining operation performed. Fluid maintenance must be performed on a routine basis to control rancidity.

Water-based cutting fluids must never be used when machining magnesium.

9.1.3 Chemical and Semichemical Cutting Fluids

Chemical cutting fluids generally contain no oil. They have various rates of dilution depending upon use. A wetting agent is often added to provide moderate lubricating qualities.

Semichemical cutting fluids may have a small amount of mineral oil added to improve the fluid's lubricating qualities. Semichemical cutting fluids incorporate the best qualities of both chemical and emulsifiable cutting fluids.

Chemical and semichemical cutting fluids offer the following advantages:

- Fluids dissipate heat rapidly.
- They are clean to use.
- After machining, residue is easy to remove.
- The fluids are easy to mix and do not become rancid.

Their disadvantages are:

- Some formulas have minimal lubricating qualities.
- Fluids may cause skin irritation in some workers.
- When they become contaminated with other oils, disposal can be a problem.

9.1.4 Gaseous Fluids

Compressed air is the most commonly used ***gaseous fluid*** coolant. It cools by forced convection.

In addition to cooling the workpiece and tool, compressed air also blows chips away at high velocity. Workers in surrounding areas must be shielded from the flying chips.

9.2 APPLICATION OF CUTTING FLUIDS

Machining and grinding applications require a continuous flooding of fluid around the cutting tool and work to provide efficient removal of the heat generated, **Figure 9-3.** Coolant nozzles must be positioned carefully so that, in addition to cooling the work area, the cutting fluid will also carry the chips away. In some machining operations, a conveyor system, **Figure 9-4,** is used to remove chips and cutting fluid from the cutting area. The cutting fluid is filtered to remove ***contaminants*** and returned to the machine's coolant tank for reuse.

Figure 9-3. *Coolant fluid must surround the cutting area for maximum effect. Shown is a modular hose system for applying air and liquids. The units snap together and can be shaped to fit any job. (Lockwood Products, Inc.)*

Figure 9-4. *This powered conveyor system moves chips away from a machine's cutting area. A rotating drum filter is used to separate chips and metal particles as fine as 50 microns from the cutting fluid. The filtered fluid is recycled to the machine at rates of up to 30 gallons per minute. (Jorgensen Conveyors, Inc.)*

9.2.1 Evaluation of Cutting Fluids

It is not possible in this text's limited space to cover all cutting fluids, nor does space permit recommending specific cutting fluids for every machining operation. This information can be obtained from data published by cutting fluid manufacturers. Recommendations for cutting fluid use are included in the chapters of this text dealing with each type of machine tool. In general, however, cutting fluids (*gaseous fluids* excepted) are compatible with HSS (high-speed steel) and carbide tooling. See **Figure 9-5.** Since carbide tooling operates at higher cutting speeds and generates higher cutting temperatures, cutting fluids that have high cooling rates should be used in such applications.

Machining with ceramic tooling is usually accomplished without the use of cutting fluids, **Figure 9-6.**

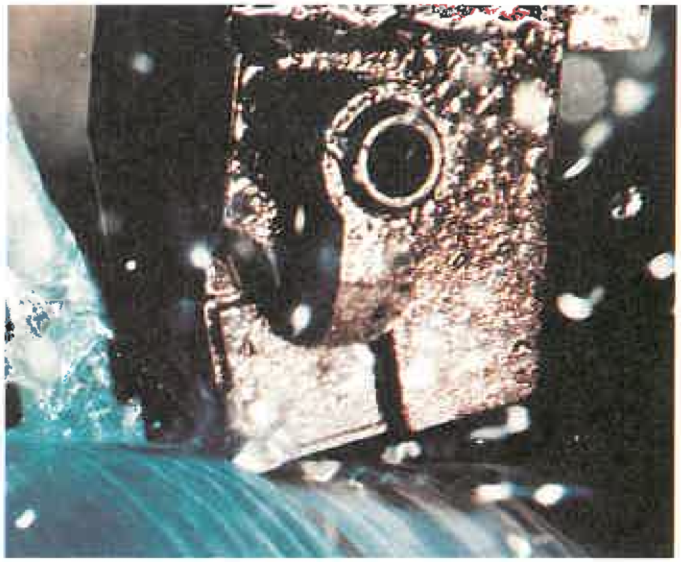


Figure 9-5. *Carbide cutting inserts are widely used on machine tools, but require coolants that can remove heat at higher rates. This tungsten carbide insert is coated with a thin film of diamond material for longer life when cutting highly abrasive materials. (Kennametal, Inc.)*

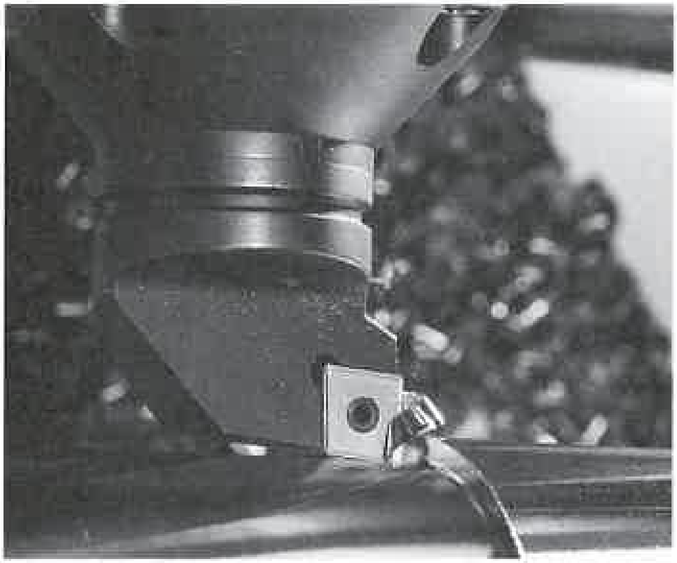


Figure 9-6. *This ceramic cutting insert is being used to turn a titanium workpiece. Cutting fluids are not necessary with ceramic tooling. (Kennametal, Inc.)*

TEST YOUR KNOWLEDGE

Please do not write in the text. Write your answer on a separate sheet of paper.

1. Cutting fluids must do many things simultaneously. What does this include?
2. List the four basic types of cutting fluids.
3. What type cutting oil is recommended for machining aluminum, magnesium, brass, and free-machining steels?
4. Why does the above type of cutting fluid have limited use?
5. _____ cutting fluids are also known as soluble oils.
6. What advantages do the emulsifiable oil cutting fluids have over the cutting fluids indicated in Question 3?
7. _____ cutting fluids contain no oils.
8. When small amounts of mineral oil are added to the cutting fluid described in Question 7, it is known as _____ cutting fluid.
9. What are the advantages of the cutting fluids indicated in Questions 7 and 8?
10. What is dangerous about using compressed air to cool the area being machined?

Some machine tools are designed with built-in fluid delivery systems that surround the cutting area, flooding it with coolant to remove heat and flush away chips. In other machines, the delivery system consists of separate nozzles and tubing. (Sharnoa Corp.)

Chapter 10

Drills and Drilling Machines

LEARNING OBJECTIVES

After studying this chapter, you will be able to:

- ○ Select and safely use the correct drills and drilling machine for a given job.
- ○ Make safe setups on a drill press.
- ○ Explain the safety rules that pertain to drilling operations.
- ○ List various drill series.
- ○ Sharpen a twist drill.

IMPORTANT TERMS

blind hole
center finder
countersinking
drill point gage
flutes
lip clearance
machine reamer
multiple spindle drilling machines
spotfacing
twist drills

10.1 DRILLING MACHINES

A ***drilling machine*** is a power-driven machine that holds the material and cutting tool and brings them together so a round hole is made in the material. Many different types of drilling machines are used in industry. The type of machine used depends on the operation being performed, size of the workpiece, and the variety of operations required of the machine.

The most common drilling machine is the ***drill press,*** **Figure 10-1.** A drill press operates by rotating a cutting tool, or drill, against the material with sufficient pressure to cause the drill to penetrate the material. See **Figure 10-2.**

The size of a drill press is determined by the largest diameter circular piece that can be drilled on center, **Figure 10-3.** A 17″ drill press can drill to the center of a 17″ diameter piece. The centerline of the drill is 8 1/2″ from the column.

Figure 10-1. *A light 15″ variable speed drill press. (Wilton Corp.)*

Bench drill presses can be used to drill holes in small workpieces, **Figure 10-4.** These presses do not have as many capabilities as the floor models.

Electric hand drills are used to drill small holes in relatively thin material. They are reasonably priced and convenient to use. See **Figure 10-5.**

A ***radial drill press*** is designed to handle very large drilling work. The drill head is mounted in a way that allows it to be moved back and forth on an

Figure 10-2. *Drilling is the operation most often performed on a drill press. A—Completed hole. B—Both rotating force and a downward pushing force are needed for drilling. (Clausing Industrial, Inc.)*

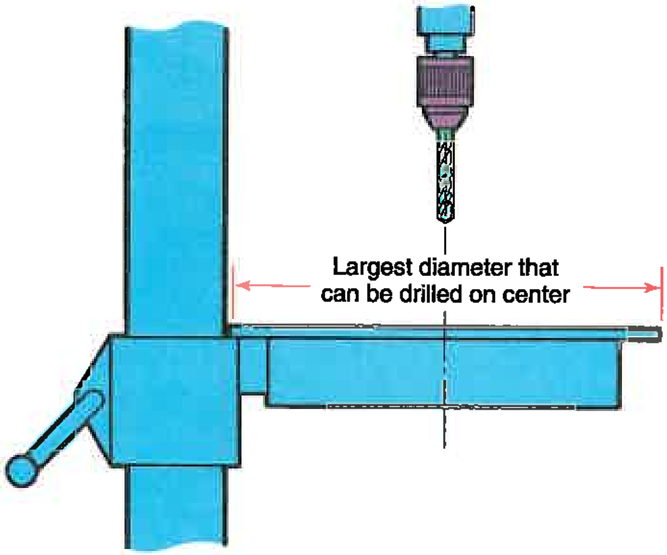


Figure 10-3. *How drill press size is determined.*

Figure 10-4. *A small bench drill press can be very useful.*

arm that extends from the massive machine column. The arm can be moved up and down and pivoted on the column, **Figure 10-6.**

Often, a large pit is located along one side of the machine to permit the positioning of large, odd-shaped work. The pit is covered when not in use. Holes up to 3 1/2″ (90 mm) in diameter can be drilled with this machine.

Figure 10-5. *Portable electric drills are manufactured in a wide range of sizes. This model is battery powered.*

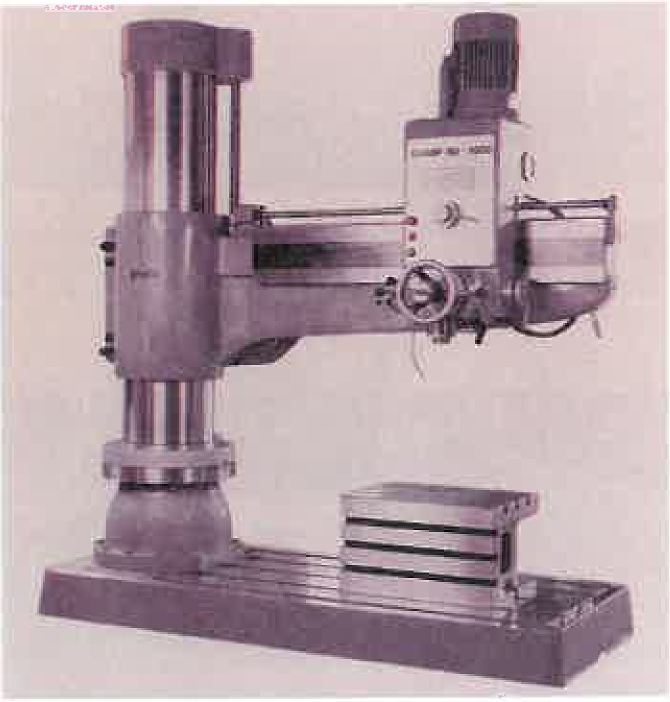


Figure 10-6. *This radial drill press can drill large diameter holes in large workpieces. (Sharp Industries, Inc.)*

Smaller ***bench radial drill presses*** are used to drill smaller holes. See **Figure 10-7.** These units are not as expensive as a full-size radial drill press.

Portable magnetic drills can be used in the shop and on the job site. These machines can be positioned in an upright, horizontal, or vertical position when drilling. See **Figure 10-8.**

Figure 10-7. *Bench radial drill press. (DoALL Co.)*

Figure 10-8. *Portable magnetic drill. The magnetic base locks the machine to ferrous metals and enables it to be used in situations where a conventional drill press cannot be employed. (Hougen Manufacturing, Inc.)*

Gang drilling machines consist of several drill assemblies, **Figure 10-9.** The workpiece is moved from one assembly to another. A different operation is performed at each stage.

Multiple spindle drilling machines have several drilling heads. Several operations can be performed without changing drills. See **Figure 10-10.**

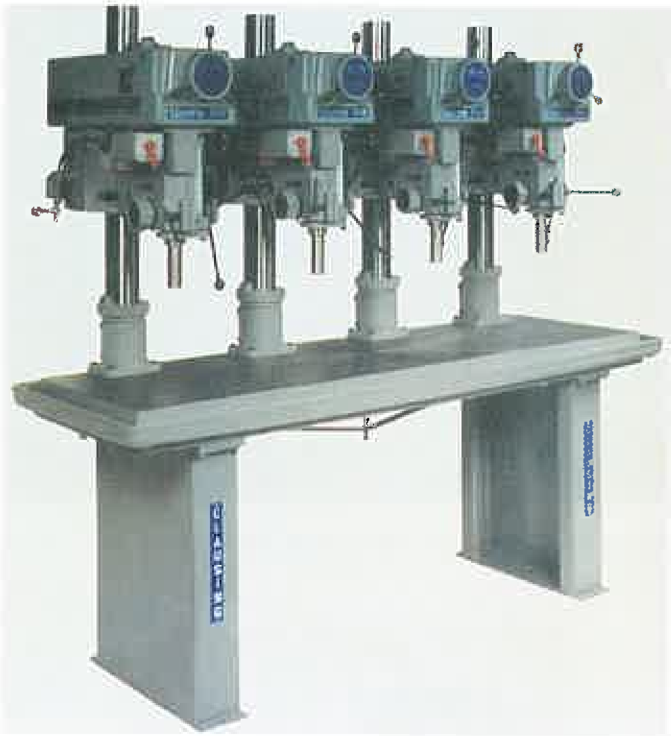


Figure 10-9. This gang drilling machine has four drills working together. Each machine is fitted with a different cutting tool. The work is held in a drill jig that moves from position to position as each operation is performed. (Clausing Industrial, Inc.)

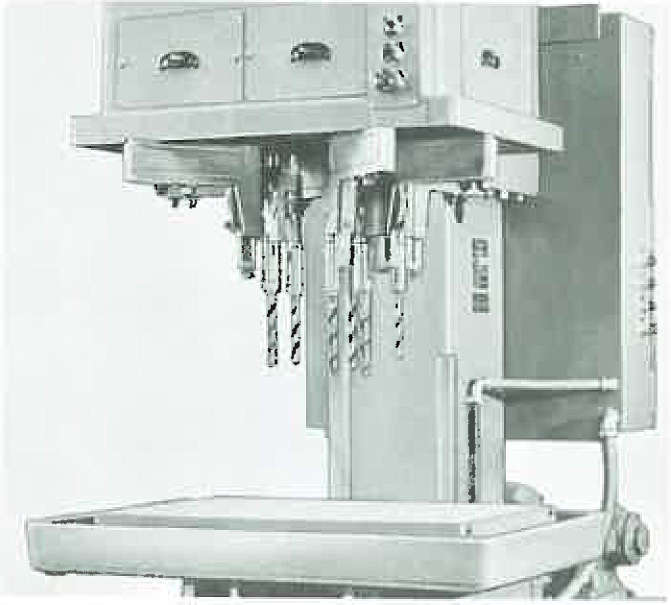


Figure 10-10. This heavy-duty multiple spindle drill press has drilling heads that can be positioned to meet various drilling requirements. (Deka-Drill, South Bend Lathe)

Machining centers operate very efficiently and accurately. The center operates under computer numerical control (CNC). These systems are only used in industrial applications. See **Figure 10-11.**

Robotic drilling machines are basically programmable, mobile drilling machines, **Figure 10-12.** The machine is programmed to move along one workpiece or between several workpieces, drilling at specified locations. With a standard drill press, the drill is stationary and the workpiece is moved. With robotic drilling machines, the workpiece is stationary and the machine moves.

Figure 10-11. Vertical machining center can be programmed to drill holes as part of a machining sequence. No drill jigs are required. Computer numerical control (CNC) is used to set drill position and speed. (Sharnoa Corp.)

Figure 10-12. This robotic drilling machine moves between workstations on rails. (Northrop-Grumman Corp.)

10.1.1 Uses of Drilling Machines

Drilling machines are primarily used for cutting round holes. They are also used for many different machining operations, including the following:

- Reaming—An operation performed on an existing hole. The hole is enlarged and finished as the tool removes material from the internal surface of the hole. See **Figure 10-13**.
- Countersinking—Enlarging a hole at the workpiece surface along an angle to allow a screw head to be flush with the surface. See **Figure 10-14**.
- Counterboring—Similar to countersinking, this operation cuts a cylindrical enlargement at the surface of a hole to allow bolt heads to be flush with the surface of the workpiece. See **Figure 10-15**.
- Spotfacing—Putting a smooth finish on a raised area surrounding a hole. See **Figure 10-16**.
- Tapping—This operation cuts screw threads into an existing hole.

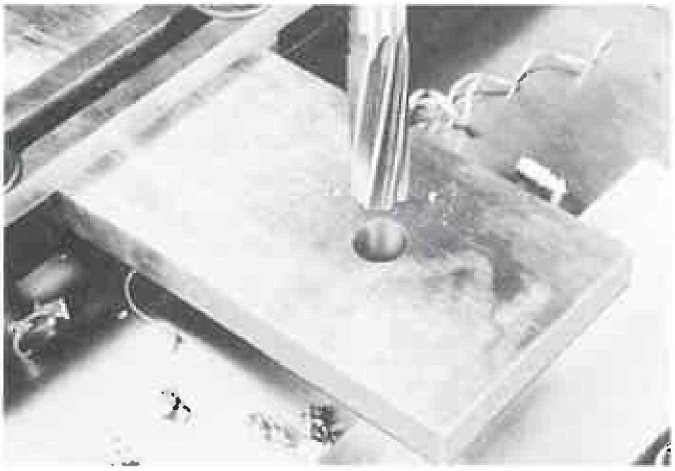


Figure 10-13. *Reaming is being done on this drill press. (Clausing Industrial, Inc.)*

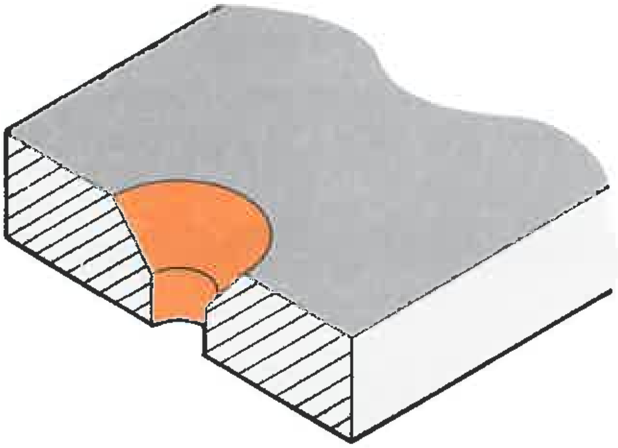


Figure 10-14. *Cross section of a drilled hole that has been countersunk.*

10.2 DRILL PRESS SAFETY

- Wear goggles when working on a drill press.
- Remove any jewelry and tuck in loose clothing so they do not become entangled in the rotating drill.

Figure 10-15. *Counterboring is done to prepare a hole to receive a fillister- or socket-head screw. (Clausing Industrial, Inc.)*

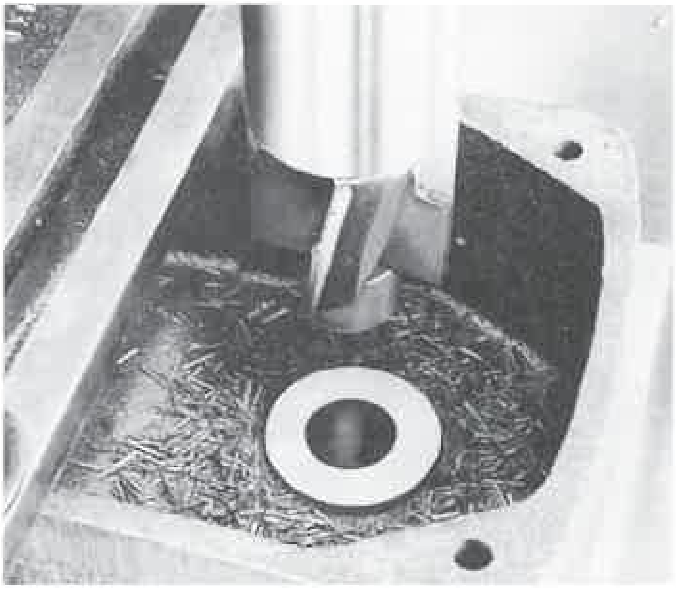


Figure 10-16. *Spotfacing is machining a surface to permit a nut or bolt head to bear uniformly. (Clausing Industrial, Inc.)*

- Check the operation of the machine.
- Be sure all guards are in place.
- Clamp the work solidly. Do not hold work with your hand.
- When removing a drill, place a piece of wood below it. Small drills can be damaged if dropped and larger drills can cause injuries.
- Never attempt to operate a drilling machine while your senses are impaired by medication or other substances.
- Use sharp tools.
- Always remove the key from the chuck before turning on the power.
- Let the drill spindle come to a stop after completing the operation. Do not stop it with your hand.
- Clean chips from the work with a brush, not your hands.
- Keep the work area clear of chips. Place them in an appropriate container. Do not brush them onto the floor.
- Wipe up all cutting fluid that spills on the floor right away.
- Place all oily and dirty waste in a closed container when the job is finished.

10.3 DRILLS

Common drills are known as ***twist drills*** because most are made by forging or milling rough flutes and then twisting them to a spiral shape. After twisting, the drills are milled and ground to approximate size, **Figure 10-17.** Then, they are heat-treated and ground to exact size.

Most drills are made of ***high-speed steel*** (HSS) or ***carbon steel.*** High-speed steel drills can be operated at much higher cutting speeds than carbon steel drills without danger of burning and drill damage.

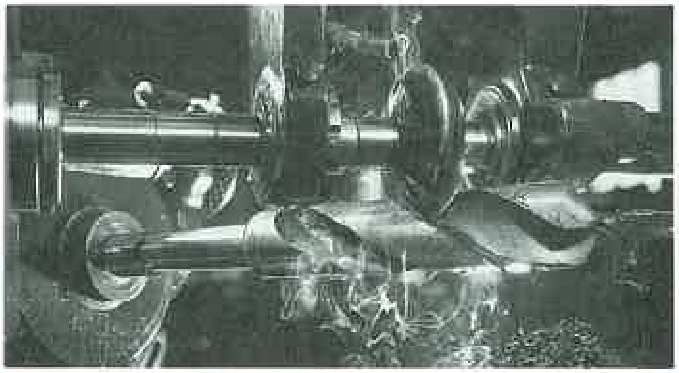


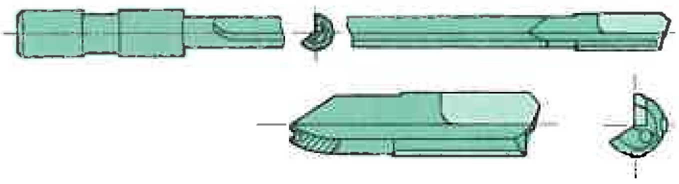
Figure 10-17. *Close-up of flute milling operation in the manufacture of a large drill. (Chicago-Latrobe)*

Most drills are available with straight or taper shanks and with tungsten carbide tips. Coating drills with titanium nitride greatly increases tool life.

10.3.1 Types of Drills

Industry uses special drills to improve the accuracy of the drilled hole, to speed production, and to improve drilling efficiency.

The ***straight-flute gun drill*** is designed for ferrous and nonferrous metals, **Figure 10-18.** It is usually fitted with a carbide cutting tip.

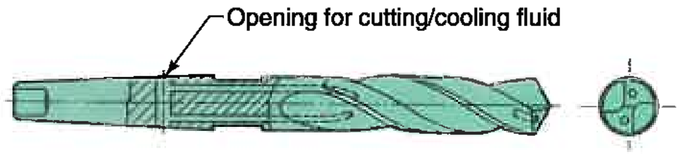


Carbide-tipped gun drill

Figure 10-18. *Gun drill. The tip is shown in larger scale. The light-colored portion is tungsten carbide. The larger area does the cutting; the smaller sections act as wear surfaces.*

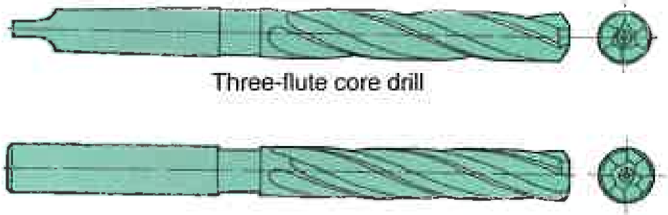
An ***oil-hole drill*** has coolant holes through the body, which permit fluid or air to remove heat from the point, **Figure 10-19.** The pressure of the fluid or air also ejects the chips from the hole while drilling.

Three- and four-flute core drills are used to enlarge core holes in a casting. See **Figure 10-20.**



Coolant hole drills

Figure 10-19. *A taper shank twist drill with holes to direct coolant to the cutting edges.*



Three-flute core drill

Four-flute core drill

Figure 10-20. *Three- and four-flute core drills.*

Special ***step drills*** eliminate drilling operations in production work. **Figure 10-21** illustrates this type drill. A ***combination drill and reamer*** also speeds up production by eliminating one operation, **Figure 10-22.**

Microdrills, **Figure 10-23,** have diameters smaller than 0.0135″. They require special drilling equipment.

Half-round straight-flute drills, **Figure 10-24,** are designed for producing holes in brass, copper alloys, and other soft nonferrous materials. Heavy duty carbide-tipped versions are available for drilling hardened steels. These drills are manufactured in fractional and number sizes.

Indexable-insert drills, **Figure 10-25,** are capable of drilling at much higher speeds than high-speed steel drills. The low-cost carbide inserts with multiple cutting edges eliminate costly sharpening. The entire drill does not need to be replaced when the cutting edges are worn—only the insert is changed.

Indexable-insert drills do have limitations. Hole depth is limited to approximately four times the hole diameter. The smallest size available is 5/8″ diameter.

Spade drills have replaceable cutting tips that are normally made of tungsten carbide, **Figure 10-26.** These drills are available in sizes from 1″ to 5″ (25 mm to 125 mm). They are less expensive than twist drills of the same size.

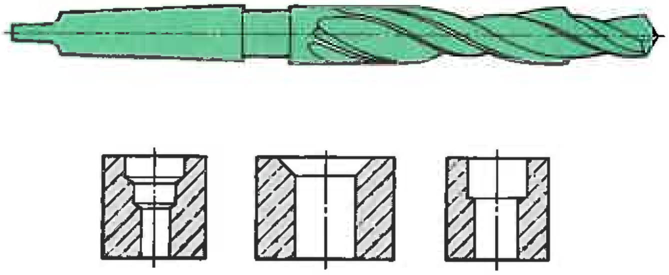


Figure 10-21. Step drill.

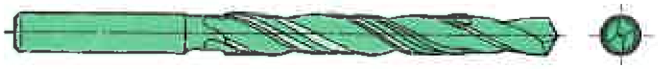


Figure 10-22. Combination drill and reamer.

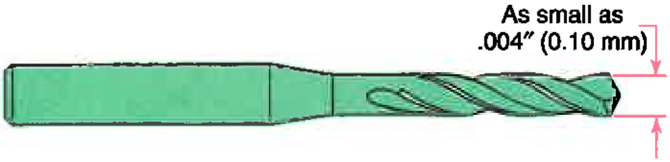


Figure 10-23. Microdrills are smaller than the #80 drill (0.0135″ diameter).

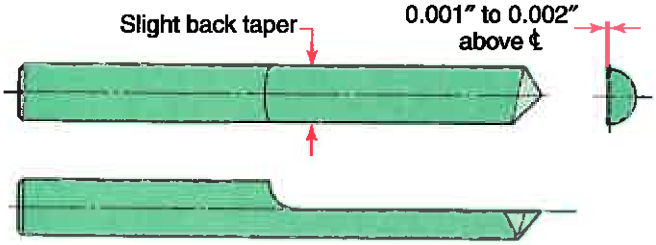


Figure 10-24. A half-round drill.

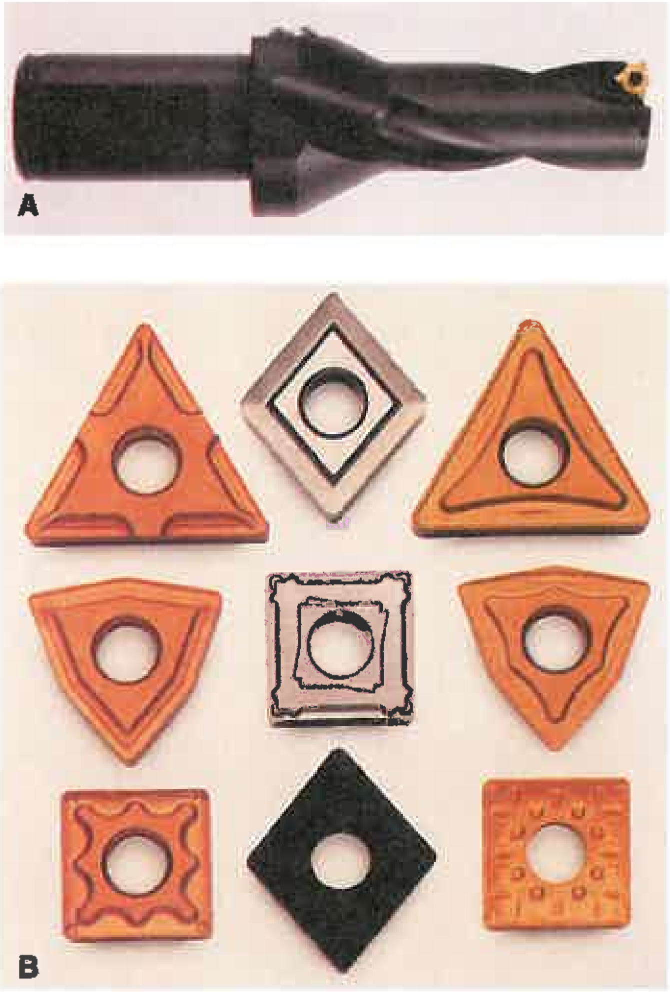


Figure 10-25. Indexable-insert drill. A—When the carbide insert becomes worn, it can be replaced with a new insert. B—Different insert shapes are used with different materials and operations.
(Iscar Metals, Inc. and Hartel Cutting Technologies, Inc.)

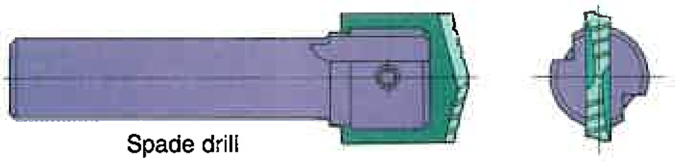


Figure 10-26. *This spade drill has an interchangeable cutting tip.*

10.3.2 Drill Size

Drill sizes are expressed by the following series:

- *Numbers*—#80 to #1 (0.0135″ to 0.2280″ diameters).
- *Letters*—A to Z (0.234″ to 0.413″ diameters).
- *Inches and fractions*—1/64″ to 3 1/2″ diameters.
- *Metric*—0.15 mm to 76.0 mm diameters.

The drill size chart will give an idea of this vast array of drill sizes, **Figure 10-27.**

10.3.3 Drill Measurements

Most drills, with the exception of small drills in the number series, have the diameter stamped on the shank. These figures frequently become obscured, making it necessary to determine the diameter by measuring.

When a micrometer is used for measuring, the measurement is made across the drill margins. However, if the drill is worn, the measurement is made on the shank at the end of the flutes. See **Figure 10-28** for both techniques.

Diameter can also be checked with a ***drill gage,*** **Figure 10-29.** Drill gages are made for various drill series; however, 1/2″ drills are the largest that can be checked. New drills are checked at the points, worked drills are checked at the end of the flutes.

Always check the drill diameter before using it. Using the wrong size drill can be a very expensive and time-consuming mistake.

Inch	mm	Wire gage	Decimals of an inch
		80	.0135
		79	.0145
1/64			.0156
	.4		.0157
		78	.0160
		77	.0180
	.5		.0197
		76	.0200
		75	.0210
	.55		.0217
		74	.0225
	.6		.0236
		73	.0240
		72	.0250
	.65		.0256
		71	.0260
	.7		.0276
		70	.0280
		69	.0293
	.75		.0295
		68	.0310
1/32			.0313
	.8		.0315
		67	.0320
		66	.0330
	.85		.0335
		65	.0350
	.9		.0354
		64	.0360
		63	.0370
	.95		.0374
		62	.0380
		61	.0390
	1		.0394
		60	.0400
		59	.0410
	1.05		.0413
		58	.0420
		57	.0430
	1.1		.0433
	1.15		.0453
		56	.0465

Inch	mm	Wire gage	Decimals of an inch
3/64			.0469
	1.2		.0472
	1.25		.0492
	1.3		.0512
		55	.0520
	1.35		.0531
		54	.0550
	1.4		.0551
	1.45		.0571
	1.5		.0591
		53	.0595
	1.55		.0610
1/16			.0625
	1.6		.0630
		52	.0635
	1.65		.0650
	1.7		.0669
		51	.0670
	1.75		.0689
		50	.0700
	1.8		.0709
	1.85		.0728
		49	.0730
	1.9		.0748
		48	.0760
	1.95		.0768
5/64			.0781
		47	.0785
	2		.0787
	2.05		.0807
		46	.0810
		45	.0820
	2.1		.0827
	2.15		.0846
		45	.0860
	2.2		.0866
	2.25		.0886
		43	.0890
	2.3		.0906
	2.35		.0925
		42	.0935

Inch	mm	Wire gage	Decimals of an inch
3/32			.0938
	2.4		.0945
		41	.0960
	2.45		.0966
		40	.0980
	2.5		.0984
		39	.0995
		38	.1015
	2.6		.1024
		37	.1040
	2.7		.1063
		36	.1065
	2.75		.1083
7/64			.1094
		35	.1100
	2.8		.1102
		34	.1110
		33	.1130
	2.9		.1142
		32	.1160
	3		.1181
		31	.1200
	3.1		.1220
1/8			.1250
	3.2		.1260
	3.25		.1280
		30	.1285
	3.3		.1299
	3.4		.1339
		29	.1360
	3.5		.1378
		28	.1405
9/64			.1406
	3.6		.1417
		27	.1440
	3.7		.1457
		26	.1470
	3.75		.1476
		25	.1495
	3.8		.1496
		24	.1520
	3.9		.1535
		23	.1540

Inch	mm	Wire gage	Decimals of an inch
5/32			.1563
		22	.1570
	4		.1575
		21	.1590
		20	.1610
	4.1		.1614
	4.2		.1654
		19	.1660
	4.25		.1673
	4.3		.1693
		18	.1695
11/64			.1719
		17	.1730
	4.4		.1732
		16	.1770
	4.5		.1772
		15	.1800
	4.6		.1811
		14	.1820
		13	.1850
	4.7		.1850
	4.75		.1870
3/16			.1875
	4.8		.1890
		12	.1890
		11	.1910
	4.9		.1929
		10	.1935
		9	.1960
	5		.1969
		8	.1990
	5.1		.2008
		7	.2010
13/64			.2031
		6	.2040
	5.2		.2047
		5	.2055
	5.25		.2067
	5.3		.2087
		4	.2090
	5.4		.2126
		3	.2130
	5.5		.2165
7/32			.2188
	5.6		.2205
		2	.2210
	5.7		.2244
	5.75		.2264
		1	.2280
	5.8		.2883

Figure 10-27. *Decimal equivalents of drill sizes.*

Inch	mm	Letter sizes	Decimals of an inch	Inch	mm	Letter sizes	Decimals of an inch	Inch	mm	Decimals of an inch	Inch	mm	Decimals of an inch
	5.9		.2323	21/64			.3281	35/64		.5469	1		1.0000
		A	.2340		8.4		.3307		14	.5512		25.5	1.0039
15/64			.2344			Q	.3320	9/16		.5625	1 1/64		1.0156
	6		.2362		8.5		.3346		14.5	.5709		26	1.0236
		B	.2380		8.6		.3386	37/64		.5781	1 1/32		1.0313
	6.1		.2402			R	.3390		15	.5906		26.5	1.0433
		C	.2420		8.7		.3425	19/32		.5938	1 3/64		1.0469
	6.2		.2441	11/32			.3438	39/64		.6094	1 1/16		1.0625
		D	.2460		8.75		.3345		15.5	.6102		27	1.0630
	6.25		.2461		8.8		.3465	5/8		.6250	1 5/64		1.0781
	6.3		.2480			S	.3480		16	.6299		27.5	1.0827
1/4		E	.2500		8.9		.3504	41/64		.6406	1 3/32		1.0938
	6.4		.2520		9		.3543		16.5	.6496		28	1.1024
	6.5		.2559			T	.3580	21/32		.6563	1 7/64		1.1094
		F	.2570		9.1		.3583		17	.6693		28.5	1.1220
	6.6		.2598	23/64			.3594	43/64		.6719	1 1/8		1.1250
		G	.2610		9.2		.3622	11/16		.6875	1 9/64		1.1406
	6.7		.2638		9.25		.3642		17.5	.6890		29	1.1417
17/64			.2656		9.3		.3661	45/64		.7031	1 5/32		1.1562
	6.75		.2657			U	.3680		18	.7087		29.5	1.1614
		H	.2660		9.4		.3701	23/32		.7188	1 11/64		1.1719
	6.8		.2677		9.5		.3740		18.5	.7283		30	1.1811
	6.9		.2717	3/8			.3750	47/64		.7344	1 3/16		1.1875
		I	.2720			V	.3770		19	.7480		30.5	1.2008
	7		.2756		9.6		.3780	3/4		.7500	1 13/64		1.2031
		J	.2770		9.7		.3819	49/64		.7656	1 7/32		1.2188
	7.1		.2795		9.75		.3839		19.5	.7677		31	1.2205
		K	.2810		9.8		.3858	25/32		.7812	1 15/64		1.2344
9/32			.2812			W	.3860		20	.7874		31.5	1.2402
	7.2		.2835		9.9		.3898	51/64		.7969	1 1/4		1.2500
	7.25		.2854	25/64			.3906		20.5	.8071		32	1.2598
	7.3		.2874		10		.3937	13/16		.8125	1 17/64		1.2656
		L	.2900			X	.3970		21	.8268		32.5	1.2795
	7.4		.2913			Y	.4040	53/64		.8281	1 9/32		1.2813
		M	.2950	13/32			.4063	27/32		.8438	1 19/64		1.2969
	7.5		.2953			Z	.4130		21.5	.8465		33	1.2992
19/64			.2969		10.5		.4134	55/64		.8594	1 5/16		1.3125
	7.6		.2992	27/64			.4219		22	.8661		33.5	1.3189
		N	.3020		11		.4331	7/8		.8750	1 21/64		1.3281
	7.7		.3031	7/16			.4375		22.5	.8858		34	1.3386
	7.75		.3051		11.5		.4528	57/64		.8906	1 11/32		1.3438
	7.8		.3071	29/64			.4531		23	.9055		34.5	1.3583
	7.9		.3110	15/32			.4688	29/32		.9063	1 23/64		1.3594
5/16			.3125		12		.4724	59/64		.9219	1 3/8		1.3750
	8		.3150	31/64			.4844		23.5	.9252		35	1.3780
		O	.3160		12.5		.4921	15/16		.9375	1 25/64		1.3906
	8.1		.3189	1/2			.5000		24	.9449		35.5	1.3976
	8.2		.3228		13		.5118	61/64		.9531	1 13/32		1.4063
		P	.3230	33/64			.5156		24.5	.9646		36	1.4173
	8.25		.3248	17/32			.5313	31/32		.9688	1 27/64		1.4219
	8.3		.3268		13.5		.5315		25	.9843		36.5	1.4370
								63/64		.9844			

Figure 10-27. *(continued)*

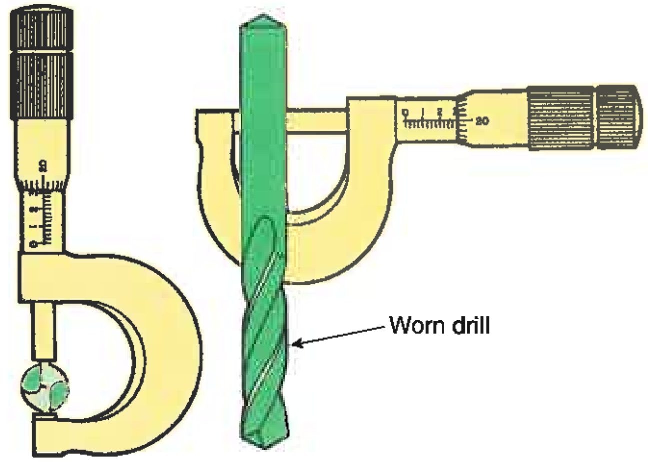


Figure 10-28. *Measuring drill size with a micrometer.*

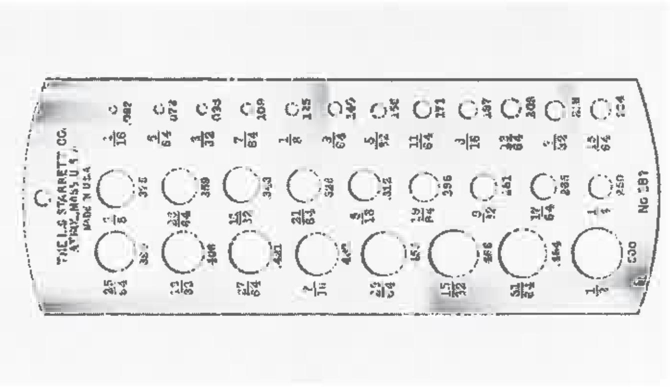


Figure 10-29. *This drill gage is used to measure fractional size drills. Similar gages are available for measuring letter, number, and millimeter size drills.*

10.3.4 Parts of a Drill

The twist drill is an efficient cutting tool. It is composed of three principal parts: point, shank, and body, **Figure 10-30.**

The ***point*** is the cone-shaped end that does the cutting. The point consists of the following components:

- ***Dead center*** refers to the sharp edge at the extreme tip of the drill. This should always be in the *exact center* of the drill axis.
- The ***lips*** are the cutting edges of the drill.
- The ***heel*** is the portion of the point back from the lips.
- ***Lip clearance*** is the amount by which the surface of the point is relieved back from the lips.

10.3.5 Shank

The ***shank*** is the portion of the drill that mounts into the chuck or spindle. Twist drills are made with shanks that are either straight or tapered, **Figure 10-31.** Straight shank drills are used with a chuck. Taper shank drills have self-holding tapers (No. 1 to No. 5 Morse taper) that fit directly into the drill press spindle.

A ***tang*** is on the taper shank; it fits into a slot in the spindle, sleeve, or socket, and assists in driving the tool. The tang also provides a means of separating the taper from the holding device.

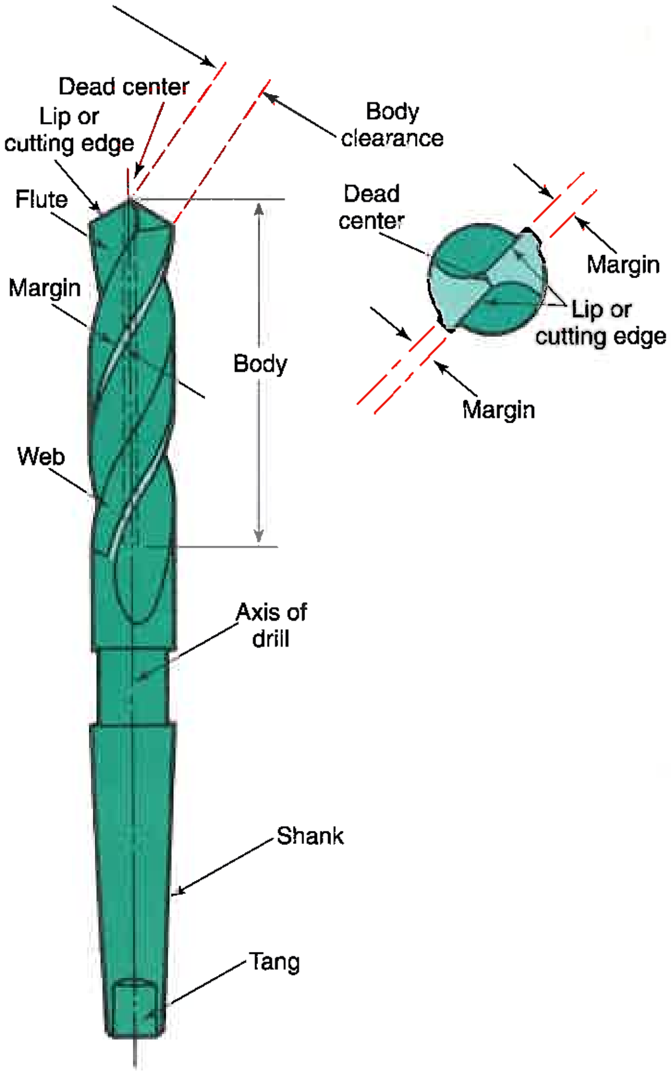


Figure 10-30. *Parts of a twist drill.*

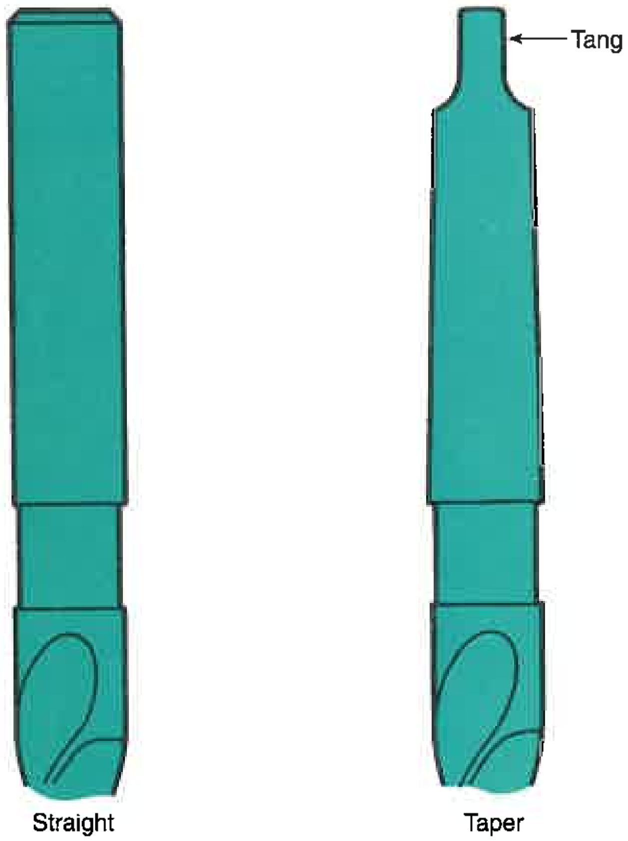


Figure 10-31. *Types of twist drill shanks.*

10.3.6 Body

The ***body*** is the portion of the drill between the point and the shank. It consists of the following:

- The ***flutes*** are two or more spiral grooves that run along the length of the drill body. The flutes serve four purposes:
  - Help form the cutting edges of the drill point.
  - Curl the chip tightly for easier removal.
  - Form channels through which the chips can escape as the hole is drilled.
  - Allow coolant and lubricant to flow down to the cutting edges.
- The ***margin*** is the narrow strip extending back along the entire length of the drill body.

- ***Body clearance*** refers to the part of the drill body that has been reduced in order to lower friction between the drill and the wall of the hole.
- The ***web*** is the metal column that separates the flutes. It gradually increases in thickness toward the shank for added strength.

10.4 DRILL-HOLDING DEVICES

A drill is held in the drill press by either of these methods:

- *Chuck:* A movable jaw mechanism for drills with straight shanks. See **Figure 10-32.**
- *Tapered spindle*: A tapered opening for drills with taper shanks, **Figure 10-33.**

Figure 10-32. *A drill chuck can be tightened with a key to hold the drill. Keyless chucks are also available. (Yukiwa Seiko USA, Inc.)*

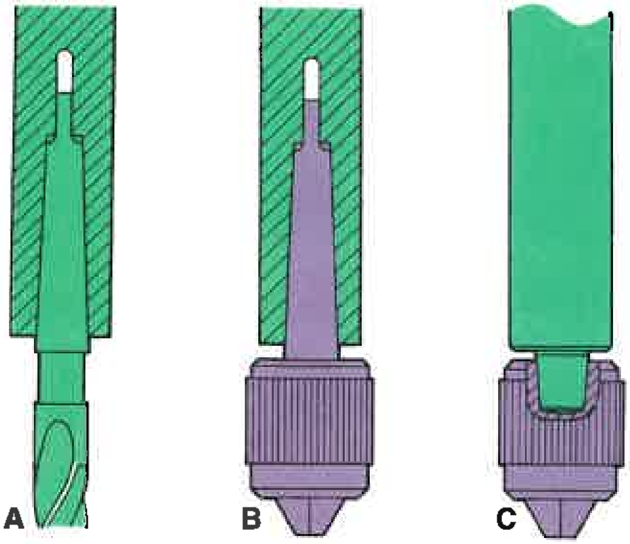


Figure 10-33. *Tapered spindles. A—Taper shank drill. B—Drill chuck with a taper shank. C—This is a solid spindle with a short external taper that fits into a drill chuck. The chuck is permanently attached to spindle.*

A drill chuck with a taper shank makes it possible to use straight-shank drills when the drill press is fitted with a tapered spindle.

When using a chuck, first insert the drill and tighten the chuck jaws by hand. If the chuck is centered and running true, tighten the chuck with a ***chuck key.*** Always remove the key from the chuck before turning on the drill press.

Taper shank drills must be wiped clean before inserting the shank into the spindle. Nicks in the shank must be removed with an oilstone; otherwise, the shank will not seat properly.

Never attempt to use a taper shank drill mounted in a drill chuck.

Most drill press spindles are made with a No. 2 or No. 3 Morse taper (often indicated as "MT"). A drill with a shank smaller than the spindle taper must be enlarged to fit by using a sleeve, **Figure 10-34.** Drills with shanks larger than the spindle opening can be fit by using a socket, **Figure 10-35.** The taper opening in the socket is larger than the taper on its shank.

Figure 10-34. *A drill sleeve is needed when the drill is too small for the spindle.*

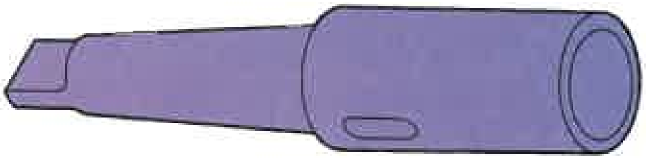


Figure 10-35. *A drill socket is used when the spindle is too small for the drill.*

A socket should only be used when a larger drill press is not available. It is dangerous to overstress a drill press by using a drill larger than the machine's rated capacity.

Sleeves, sockets, and taper shank drills are separated with a ***drift,*** **Figure 10-36.** To use a drift, insert it in the slot with the round edge up, **Figure 10-37.** A sharp rap with a lead hammer will cause the parts to separate.

Never use a file tang in place of a drift. It will damage the drill shank and machine spindle. Then

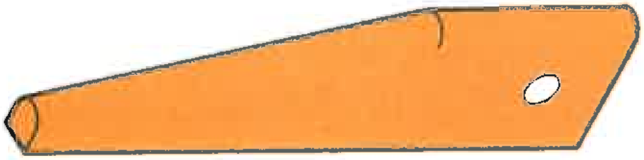


Figure 10-36. A drift is used to remove taper shank tools from the drill spindle. Never use a file tang.

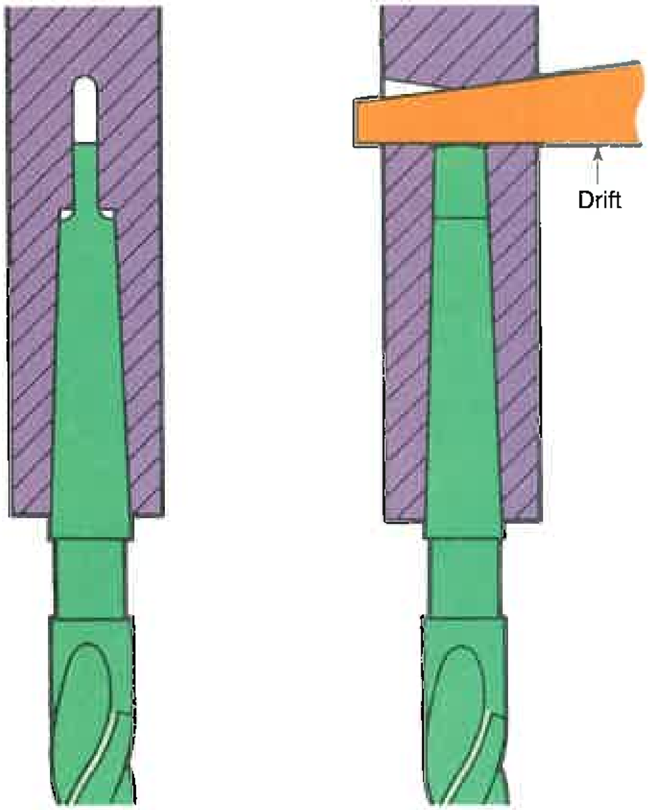


Figure 10-37. Using a drift. A—The drill is locked in the spindle. B—Using a drift to remove the drill from the spindle.

other drills will not fit properly. The file could also shatter.

When removing a drill from the spindle, hold the drill to prevent it from falling to the floor. Dropping the drill may damage the drill point. Wrap a piece of clean cloth around the drill to protect your hand from metal chips.

10.5 WORK-HOLDING DEVICES

Work must be mounted solidly on the drilling machine. If work is mounted improperly, it may spring or move, causing drill damage or breakage.

Serious injury can result from work that becomes loose and spins on a drill press. This dangerous situation is nicknamed a "merry-go-round."

10.5.1 Vises

Vises are widely used to hold work, **Figure 10-38.** For best results, the vise must be bolted to the drill table.

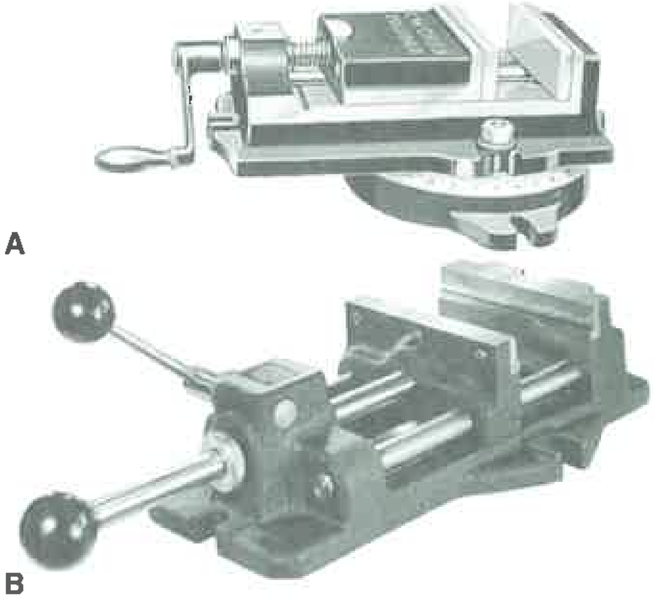


Figure 10-38. Typical vises used on drill press. A—The swivel base permits the vise body to pivot 180°. B—This quick-acting vise jaw slides to lock parts quickly. (L-W Chuck Co.)

Parallels are often used to level the work and raise it above the vise base, **Figure 10-39.** This will permit the drill to come through the work and not damage the vise. Parallels can be made from stock steel bars or from special heat-treated steel. Heat-treated parallels are ground to size.

Seat the work on the parallels by tightening the vise and tapping the work with a mallet until the parallels do not move. Loose parallels indicate that the work is not seated properly.

An ***angular vise*** permits angular drilling without tilting the drill press table. See **Figure 10-40.**

A ***cross-slide*** permits rapid alignment of the work. Some cross-slides are fitted with a vise, **Figure 10-41.** Others have a series of tapped holes for mounting a vise or another work-holding device.

10.5.2 V-Blocks

V-blocks support round work for drilling, **Figure 10-42.** These blocks are made in many sizes. Some are fitted with clamps to hold the work. Larger sizes must be clamped with the work, **Figure 10-43.**

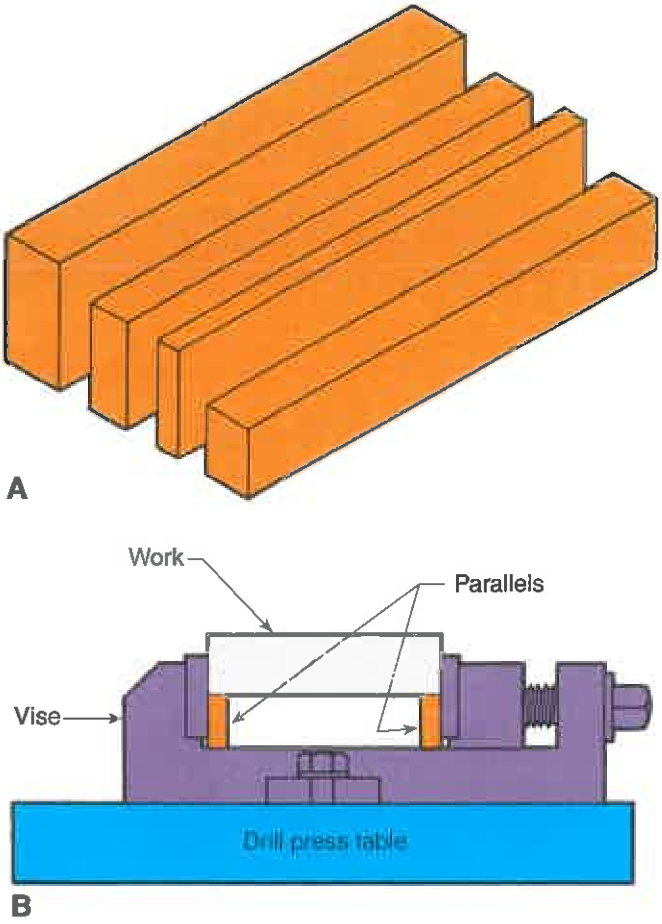


Figure 10-39. *Parallels. A— Steel parallels are available in a large variety of sizes. B—Parallels are often used to raise work above the vise base. This will prevent the drill from cutting into the vise as it goes through the work.*

10.5.3 T-Bolts

T-bolts fit into the drill press table slots and fasten the work or clamping devices to the machine, **Figure 10-44.** A washer should always be used between the nut and the holding device. For convenience, it is desirable to have an assortment of different length T-bolts. To reduce the chance of a setup working loose, place the bolts as close to the work as possible. See **Figure 10-45.**

10.5.4 Strap Clamps

Strap clamps, **Figure 10-46,** make the clamping operation easier. The elongated slot permits some adjustment without removing the washer and nut. Use a strip of copper or aluminum to protect a machined surface that must be clamped.

A *U-strap clamp* is used when the clamp must bridge the work. It can straddle the drill and not interfere with the drilling operation. The small, round section that projects from a *finger clamp* permits the use of small holes or openings in the work for clamping.

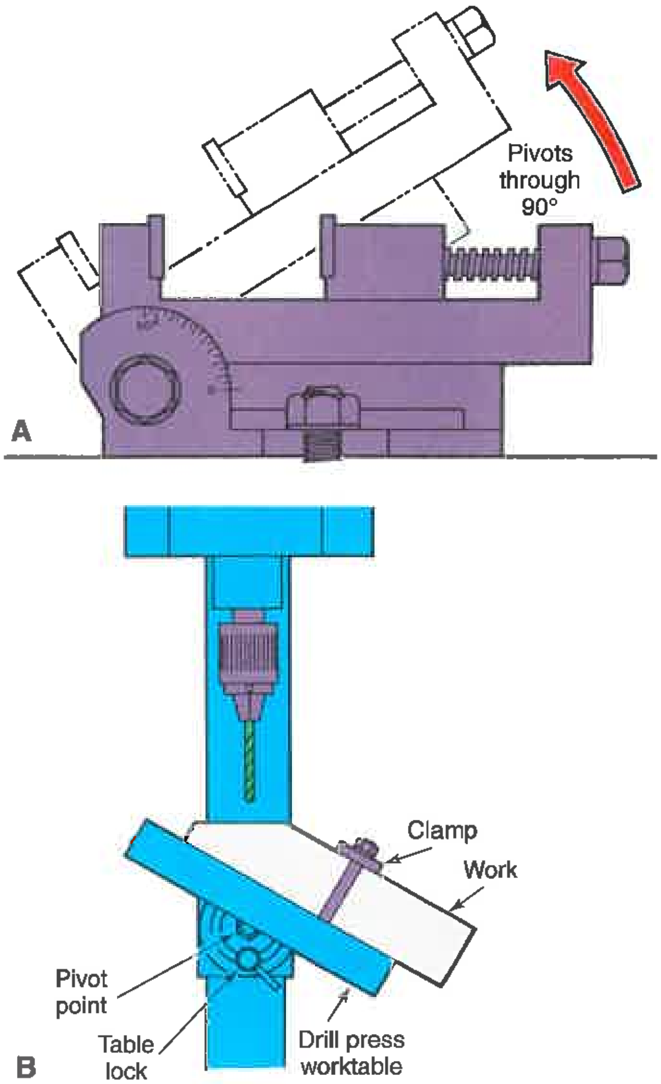


Figure 10-40. *Angular vise. A—An angular vise can be adjusted through 90° to permit drilling on an angle without tilting the entire vise or drill table. B—Angular drilling can also be done by tilting the drill table. Be sure the table is locked tightly before starting to drill.*

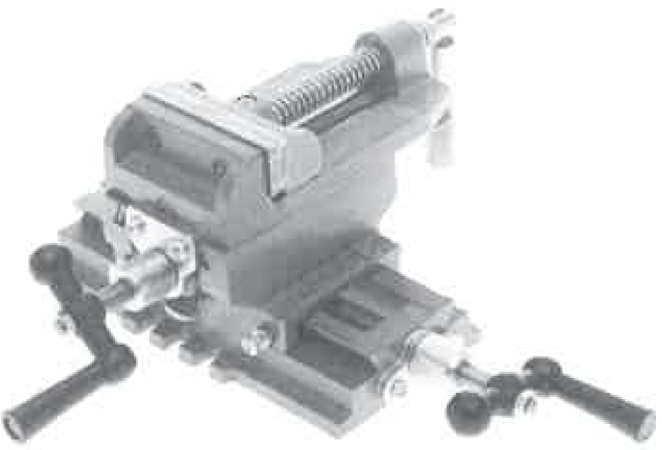


Figure 10-41. *A cross-slide vise permits rapid alignment of work for drilling.*

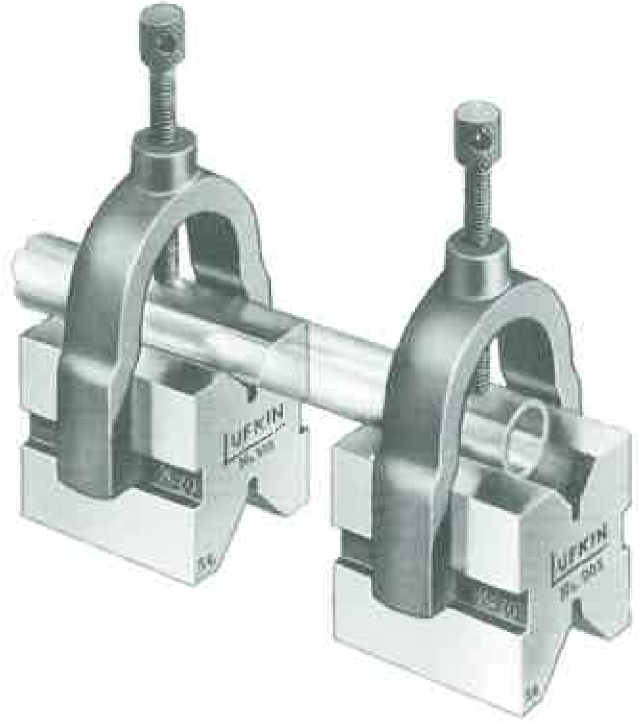


Figure 10-42. V-blocks supporting round work for drilling.

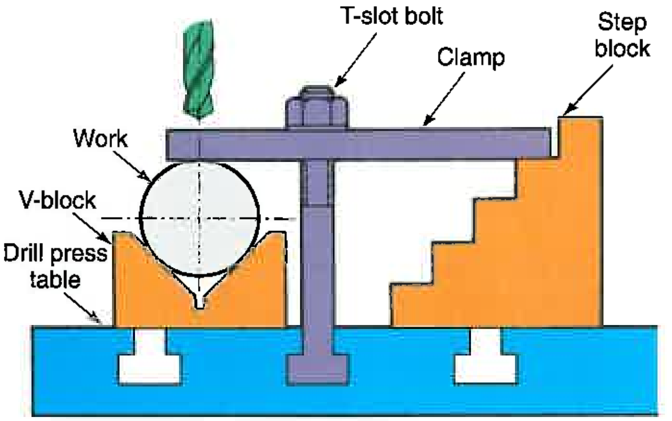


Figure 10-43. One method of clamping large diameter stock for drilling. Always check to be sure that drill will clear the V-block when it comes through work.

Figure 10-44. A few of the many types and sizes of T-bolts available.

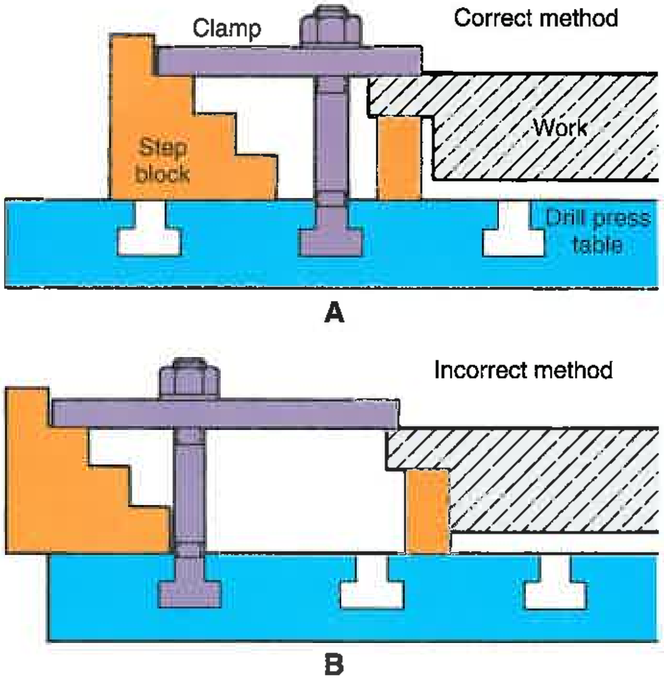


Figure 10-45. Examples of clamping techniques. A—Correct clamping technique. Note that clamp is parallel to work. Clamp slippage can be reduced by placing a piece of paper between the work and the clamp. B—Incorrect clamping technique. T-bolt is too far from work. This allows the clamp to spring under pressure.

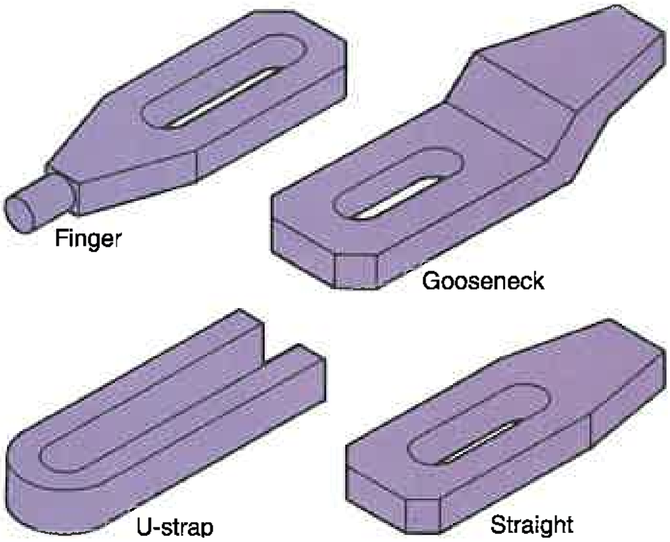


Figure 10-46. Types of strap clamps.

10.5.5 Step Blocks

A ***step block*** supports the strap clamp opposite the work, **Figure 10-47.** The steps allow the adjustments necessary to keep the strap parallel with the work.

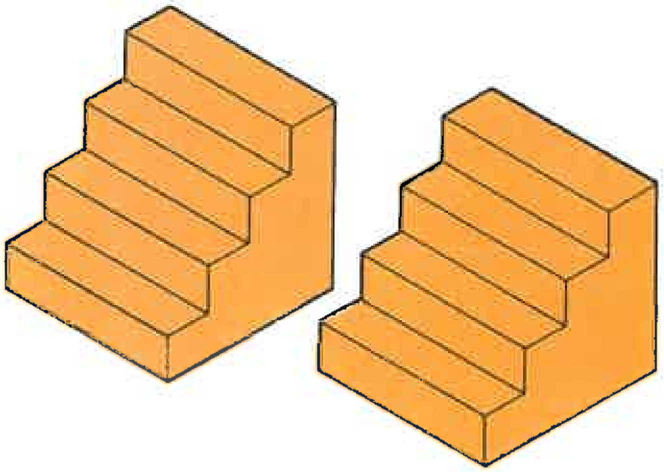


Figure 10-47. Step blocks are used to support strap clamps.

10.5.6 Angle Plate

An ***angle plate,*** **Figure 10-48,** is often used when work must be clamped to a support. The angle plate is then bolted to the machine table, **Figure 10-49.**

10.5.7 Drill Jig

A ***drill jig*** permits holes to be drilled in a number of identical pieces, **Figure 10-50.** This clamping device supports and locks the work in the proper position. With the use of drill bushings, it guides the drill to the correct location. This makes it unnecessary to lay out each individual piece for drilling.

10.6 CUTTING SPEEDS AND FEEDS

The ***cutting speed*** is the speed at which the drill rotates. The ***feed*** is the distance the drill is moved into the work with each revolution. Both are important considerations because they determine the time required to produce the hole.

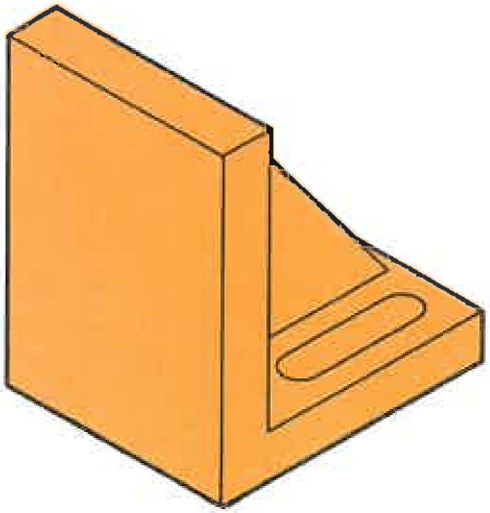


Figure 10-48. Angle plate is used to support work.

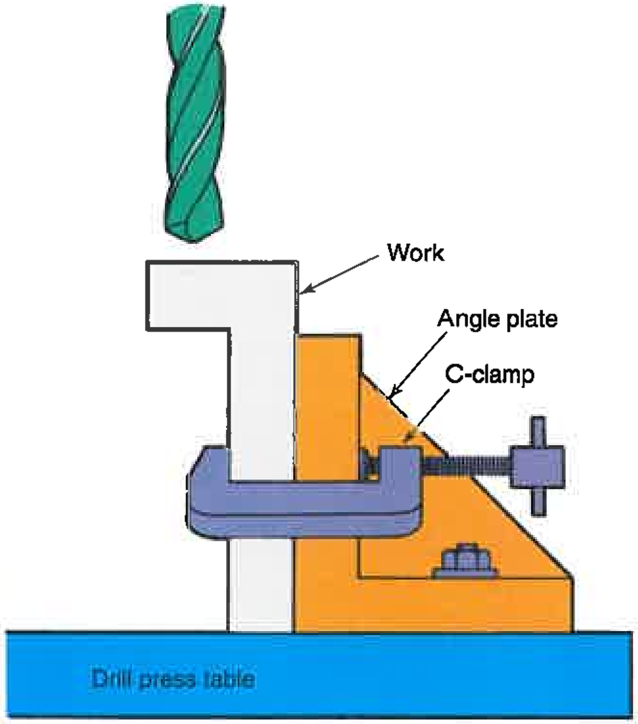


Figure 10-49. Work must sometimes be mounted against an angle plate for adequate support for drilling.

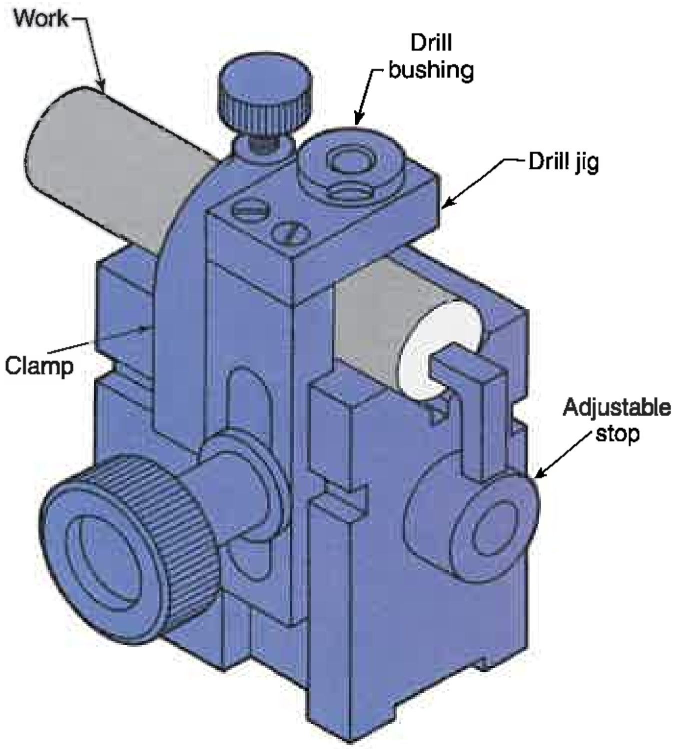


Figure 10-50. A typical drill jig for holding round stock for drilling through center.

Drill cutting speed, also known as ***peripheral speed,*** does not refer to the revolutions per minute (rpm) of the drill, but rather to the distance that the drill cutting edge circumference travels per minute.

10.6.1 Feed

Contrary to popular belief, the spiral shape of a drill flute does not cause the drill to pull itself into the work. Constant pressure must be applied and maintained to advance the drill point at a given rate. This advance is called *feed* and is measured in either decimal fractions of an inch or millimeters.

Because so many variables affect results, there can be no hard and fast rule to determine the exact cutting speed and feed for a given material. For this reason, the ***drill speed and feed table*** indicates only recommended speeds and feeds, **Figure 10-52.** They are a starting point and can be increased or decreased for optimum cutting.

Feed cannot be controlled accurately on a hand-fed drill press. A machinist must be aware of the cutting characteristics (such as uniform chips) that indicate whether the drill is being fed at the correct rate.

A feed that is too *light* will cause the drill to scrape, "chatter," and dull rapidly. Chipped cutting edges, drill breakage, and drill heating (despite the application of coolant) usually indicate that the rate is too *great.*

10.6.2 Speed Conversion

A problem arises in setting a drill press to the correct speed because its speed is given in revolutions per minute (rpm), while recommended drill cutting speed (CS) is given in feet or meters per minute (fpm or mpm).

The simple formula

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

determines the rpm to operate any diameter drill (D) at any specified speed.

Drill speed problem: At what speed (rpm) must a 1/2" diameter high-speed steel drill rotate when drilling aluminum?

To solve this problem:

1. Refer to the speed and feed table, **Figure 10-51.** It gives the recommended cutting speed for aluminum (250 fpm).
2. Convert drill diameter (1/2") to decimal fraction (0.5).
3. Substitute values into the formula

$$\begin{aligned}\text{rpm} &= \frac{4 \times \text{CS}}{\text{D}}\\ &= \frac{250 \times 4}{0.5}\\ &= 2000 \text{ rpm}\end{aligned}$$

Metric problems are solved in a similar manner using the following formula:

$$\text{rpm} = \frac{\text{CS} \times 1000}{\text{D} \times \pi}$$

Where:

CS = Cutting speed (mpm)
D = Drill diameter (mm)
π = 3 (rounded)

10.6.3 Drill Press Speed Control Mechanisms

With some drill presses, it is possible to set a dial to the desired rpm, **Figure 10-52.** However, on most conventional drilling machines, it is not possible to set the machine at the exact speed desired. The machinist must settle for the available speed nearest the desired speed.

The number of speed settings is limited by the number of pulleys in the drive mechanism, **Figure 10-53.** A decal or an engraved metal chart showing spindle speeds at various settings is attached to many machines. Information on spindle speeds can be found in the operator's manual, or it can be calculated if motor speed and pulley diameters are known.

10.7 CUTTING COMPOUNDS

Drilling at the recommended cutting speeds and feeds generates considerable heat at the cutting point. This heat must be dissipated (carried away) as fast as it is generated, or it will destroy the drill's temper and cause it to dull rapidly.

Cutting compounds are applied to absorb the heat. They cool the cutting tool, serve as a lubricant to reduce friction at the cutting edges, and minimize the tendency for the chips to weld to the lips. Cutting compounds also improve hole finish and aid in the rapid removal of chips from the hole.

There are many kinds of cutting fluids and compounds. Many cutting compounds must be applied liberally. However, some newer compounds should be applied sparingly. Read the instructions on the container for the compound being employed.

Material	Cutting Fluid	Speed feet per minute	Feeds per revolution Over .040 diameter** Under 1/8	1/8 to 1/4	1/4 to 1/2	1/2 to 1	Over 1
Aluminum & aluminum alloys	Sol. oil, ker. & lard oil, lt. oil	200-300	.0015	.003	.006	.010	.012
Aluminum & bronze	Sol. oil, ker. & lard oil, lt. oil	50-100	.0015	.003	.006	.010	.012
Brass, free machining	Dry, sol. oil, ker. & lard oil, lt. min. oil	150-300	.0025	.005	.010	.020	.025
Bronze, common	Dry, sol. oil, lard oil, min. oil	200-250	.0025	.005	.010	.020	.025
Bronze, soft and medium hard	Min. oil with 5%-15% lard oil	70-300	.0025	.005	.010	.020	.025
Bronze, phosphor, 1/2 hard	Dry, sol. oil, lard oil, min. oil	110-180	.0015	.003	.006	.010	.012
Bronze, phosphor, soft	Dry, sol. oil, lard oil, min. oil	200-250	.0025	.005	.010	.020	.025
Cast iron, soft	Dry or airjet	100-150	.0025	.005	.010	.020	.025
Cast iron, medium	Dry or airjet	70-120	.0015	.003	.006	.010	.012
Cast iron, hard	Dry or airjet	30-100	.001	.002	.003	.005	.006
*Cast iron, chilled	Dry or airjet	10-25	.001	.002	.003	.005	.006
*Cast steel	Soluble oil, sulphurized oil, min. oil	30-60	.001	.002	.003	.005	.006
Copper	Dry, soluble oil, lard oil, min. oil	70-300	.001	.002	.003	.005	.006
Magnesium & magnesium alloys	Mineral seal oil	200-400	.0025	.005	.010	.020	.025
Manganese copper, 30% mn	Soluble oil, sulphurized oil	10-25	.001	.002	.003	.005	.006
Malleable iron	Dry, soluble oil, soda water, min. oil	60-100	.0025	.005	.010	.020	.025
Monel metal	Sol. oil, sulphurized oil, lard oil	30-50	.0015	.003	.006	.010	.012
Nickel, pure	Sulphurized oil	60-100	.001	.002	.003	.005	.006
Nickel, steel 3-1/2%	Sulphurized oil	40-80	.001	.002	.003	.005	.006
Plastics, thermosetting	Dry or airjet	100-300	.0015	.003	.006	.010	.012
Plastics, thermoplastic	Soluble oil, soapy water	100-300	.0015	.003	.006	.010	.012
Rubber, hard	Dry or airjet	100-300	.001	.002	.003	.005	.006
Spring steel	Soluble oil, sulphurized oil	10-25	.001	.002	.003	.005	.006
Stainless steel, free mach'g	Soluble oil, sulphurized oil	60-100	.0025	.005	.010	.020	.025
Stainless steel, tough mach'g	Soluble oil, sulphurized oil	20-27	.0025	.005	.010	.020	.025
Steel, free machin'g SAE 1100	Soluble oil, sulphurized oil	70-120	.0015	.003	.006	.010	.012
Steel, SAE-AISI, 1000-1025	Soluble oil, sulphurized oil	60-100	.0015	.003	.006	.010	.012
Steel, .30-.60% carb., SAE 1000-9000							
Steel, annealed 150-225 Brinn.	Soluble oil, sulphurized oil	50-70	.0015	.003	.006	.010	.012
Steel, heat treated 225-283 Brinn.	Sulphurized oil	30-60	.0025	.005	.010	.020	.025
Steel, tool hi. car. & hi. speed	Sulphurized oil	25-50	.0025	.005	.010	.020	.025
Titanium	Highly activated sulphurized oil	15-20	.0025	.005	.010	.020	.025
Zinc, alloy	Soluble oil, kerosene & lard oil	200-250	.0015	.003	.006	.010	.012

*Use specially constructed heavy duty drills.

**For drill under .040, feeds should be adjusted to produce chips and not powder with ability to dispose of same without packing.

A

Drill Speeds in R.P.M.

Diameter of drill	Soft metals 300 fpm	Plastics and hard rubber 200 fpm	Annealed cast from 140 fpm.	Mild steel 100 fpm	Malleable iron 90 fpm	Hard cast iron 80 fpm	Tool or hard steel 60 fpm	Alloy steel cast steel 40 fpm
1/16 (No. 53 to 80)	18320	12217	8554	6111	5500	4889	3667	2445
3/32 (No. 42 to 52)	12212	8142	5702	4071	3666	3258	2442	1649
1/8 (No. 31 to 41)	9160	6112	4278	3056	2750	2445	1833	1222
5/32 (No. 23 to 30)	7328	4888	3420	2444	2198	1954	1465	977
3/16 (No. 13 to 22)	6106	4075	2852	2037	1833	1630	1222	815
7/32 (No. 1 to 12)	5234	3490	2444	1745	1575	1396	1047	698
1/4 (A to E)	4575	3055	2139	1527	1375	1222	917	611
9/32 (G to K)	4071	2712	1900	1356	1222	1084	814	542
5/16 (L, M, N)	3660	2445	1711	1222	1100	978	733	489
11/32 (O to R)	3330	2220	1554	1110	1000	888	666	444
3/8 (S, T, U)	3050	2037	1426	1018	917	815	611	407
13/32 (V to Z)	2818	1878	1316	939	846	752	563	376
7/16	2614	1746	1222	873	786	698	524	349
15/32	2442	1628	1140	814	732	652	488	326
1/2	2287	1528	1070	764	688	611	458	306
9/16	2035	1357	950	678	611	543	407	271
5/8	1830	1222	856	611	550	489	367	244
11/16	1665	1110	777	555	500	444	333	222
3/4	1525	1018	713	509	458	407	306	204

Figures are for high-speed drills. The speed of carbon drills should be reduced one-half. Use drill speed nearest to figure given.

B

Figure 10-51. *A—Drill speed and feed table. B—Drill speeds in rpm. These tables are starting points for drilling different materials. Feeds and speeds should be increased or decreased depending upon the specific metal being drilled and the condition of the drill press. (Chicago-Latrobe)*

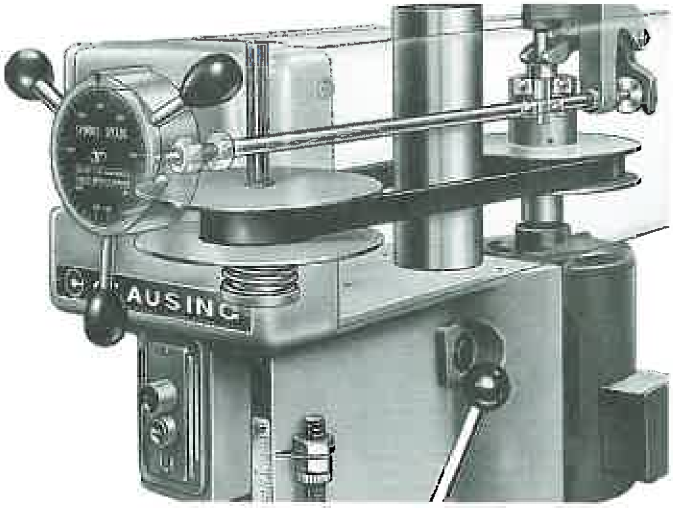


Figure 10-52. This split-pulley speed control mechanism allows speed adjustment by turning the dial in front. (Clausing Industrial, Inc.)

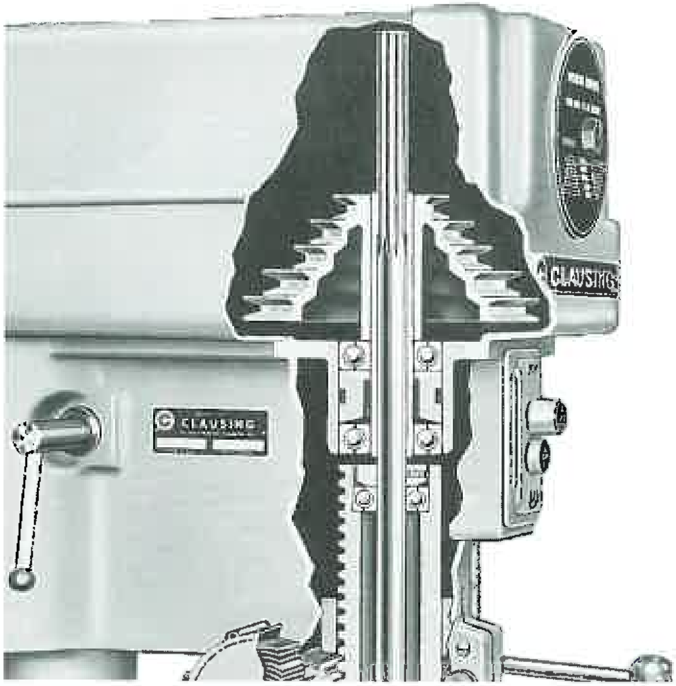


Figure 10-53. With step-pulley speed control, the belt is transferred to different pulley ratios to change drill speed. (Clausing Industrial, Inc.)

Avoid using cutting fluids and compounds when drilling cast iron or other brittle materials. The fluids tend to cause the chips to pack and glaze the opening. Compressed air, used with care, will work when drilling these materials.

Always wash your hands thoroughly with soap and warm water after using cutting compounds and fluids. Before using the cutting compound, check the container to determine what should be done if you get any in your eye.

10.8 SHARPENING DRILLS

A drill becomes dull with use and must be resharpened. Continued use of a dull drill may result in drill breakage or burning. Improper sharpening will cause the same problems.

Remove the entire point if it is badly worn or if the margins are burned, chipped, or worn off near the point. If the drill becomes overheated during grinding, do not plunge it into water. Allow it to cool in still air. The shock of sudden cooling may cause it to crack.

Three factors must be considered when repointing a drill: lip clearance, length and angle of the lips, and proper location of dead center.

Lip clearance. The two cutting edges, or lips, are comparable to chisels, **Figure 10-54.** To cut effectively, the heel (part of the point back of the cutting edge) must be relieved. Without this ***lip clearance,*** it is impossible for the lips to cut. If there is too much clearance, the cutting edges will be weakened. Too little clearance results in the drill point merely rubbing without penetration into the material.

Gradually increase lip clearance toward the center until the line across dead center stands at an angle of 120° to 135° with the cutting edge. See **Figure 10-55.**

Length and angle of lips. The material being drilled determines the proper point angle, **Figure 10-56.** The angles, in relation to the axis, must be the same (a 59° angle is satisfactory for most metals). If the angles are unequal, only one lip will cut and the hole will be oversized.

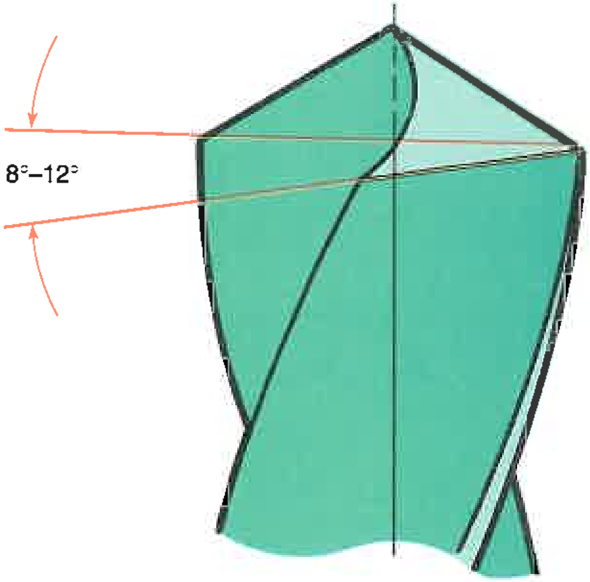


Figure 10-54. Lip clearance of 8° to 12° is satisfactory for most drilling.

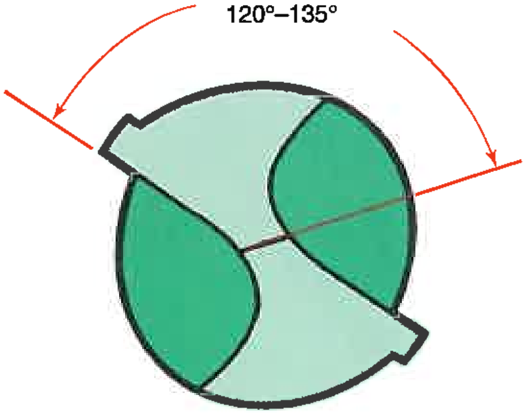


Figure 10-55. *Proper angle of drill dead center.*

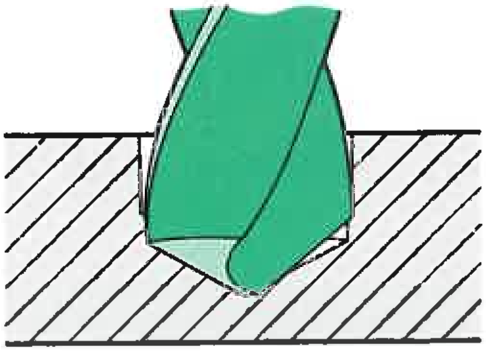


Figure 10-56. *Unequal drill point angles will produce a drilled hole that is oversize.*

Proper location of dead center. The drill dead center must be accurate. Lips of different lengths will result in oversized holes, causing "wobble." This places tremendous pressures on the drill press spindle and bearings. See **Figure 10-57.**

A combination of lip and dead center faults can result in a broken drill. If the drill is very large, permanent damage to the drilling machine can result. The hole produced will be oversized and often out-of-round. Refer to **Figure 10-58.**

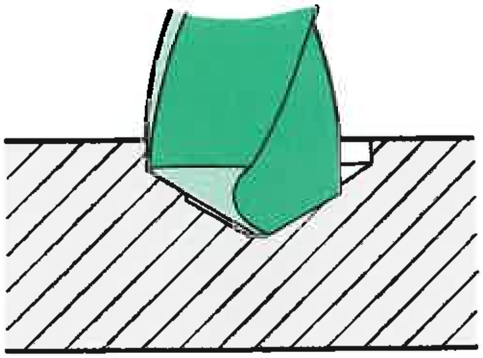


Figure 10-57. *Hole produced when drill is sharpened off center.*

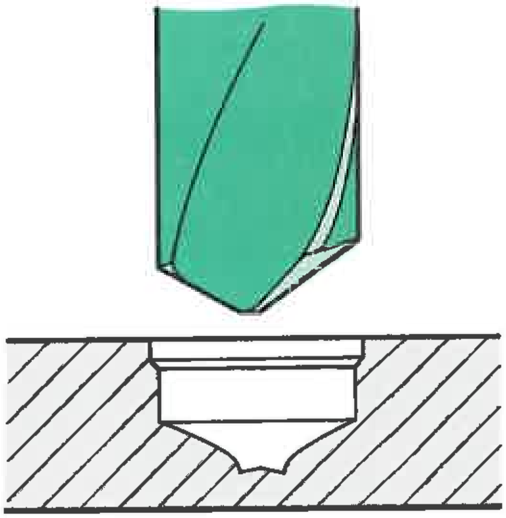


Figure 10-58. *This is the type of hole produced when the drill point has unequal point angles and is sharpened off center.*

The web of a drill increases in thickness toward the shank, **Figure 10-59.** When a drill has been shortened by repeated grindings, the web must be thinned to minimize the pressure required to make the drill penetrate the material. The thinning must be done equally on both sides of the web and care must be taken to ensure that the web is centered.

A ***drill point gage*** is used to check a drill point while sharpening. Its use is shown in **Figure 10-60.**

10.8.1 Drill Sharpening Procedures

Use a coarse grinding wheel for roughing out the drill point if a large quantity of metal must be removed. Complete the operation on a fine wheel.

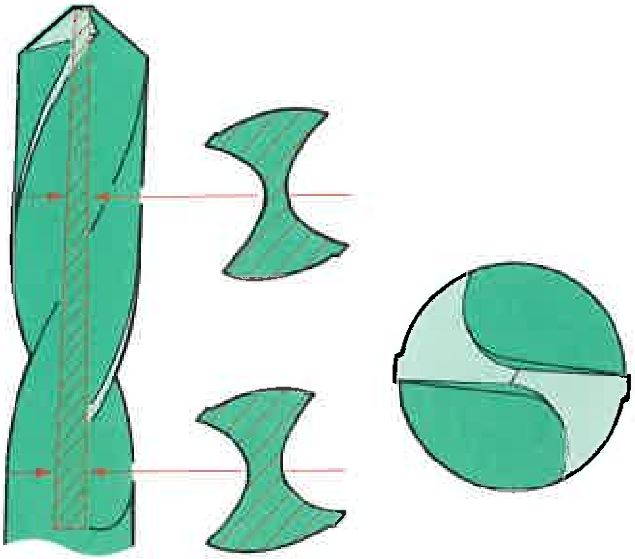


Figure 10-59. *The drill web tapers down toward the tip. The point is sometimes relieved to improve cutting action.*

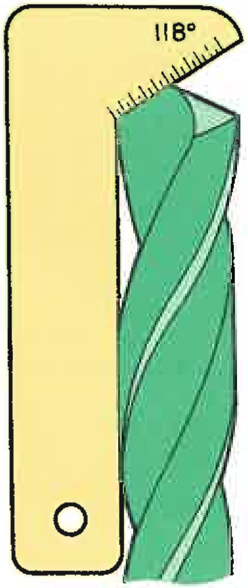


Figure 10-60. Using a drill point gage will help ensure proper drill sharpening. For general drilling, an included angle of 118° is used.

Many hand sharpening techniques have been developed. The following technique is suggested:

1. Grasp the drill shank with your right hand and the rest of the drill with your left hand. See **Figure 10-61.**

Figure 10-61. One recommended way to hold a drill when it is being sharpened.

2. Place your left-hand fingers that are supporting the drill on the grinder tool rest. The tool rest should be slightly below center (about 1" down on a 7" diameter wheel, for example).
3. Stand so the centerline of the drill will be at a 59° angle to the centerline of the wheel, **Figure 10-62.** Lightly touch the drill lip to the wheel in a horizontal position.
4. Use your left hand as a pivot point and slowly lower the shank with your right hand. Increase pressure as the heel is reached to ensure proper clearance.

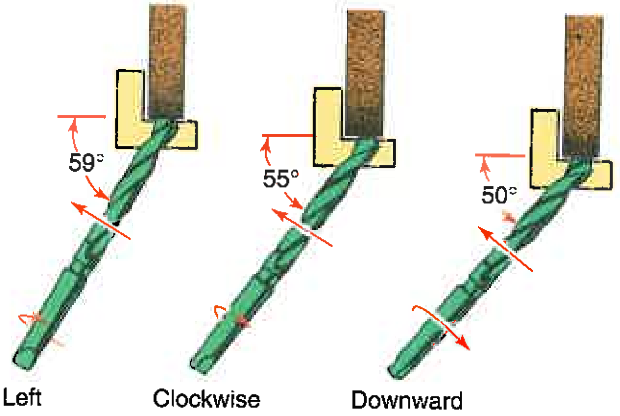


Figure 10-62. One drill sharpening technique. Hold the point lightly against the rotating wheel and use three motions of the shank: to the left, clockwise rotation, and downward.

5. Repeat the operation on each lip until the drill is sharpened. Do not quench high-speed steel drills in water to cool them. Allow them to cool in air.
6. Check the drill tip frequently with a drill point gage to ensure a correctly sharpened drill.

Sharpening a drill is not as difficult as it may first appear. However, before attempting to sharpen a drill, secure a properly sharpened drill and run through the motions explained above. When you have acquired sufficient skill, sharpen a dull drill.

To test, drill a hole in soft metal and observe the chip formation. When properly sharpened, chips will come out of the flutes in curled spirals of equal size and length. Tightness of the chip spiral is governed by the rake angle, **Figure 10-63.**

A standard drill point has a tendency to stick when used to drill brass. When brass is drilled, sharpen the drill as shown in **Figure 10-64.**

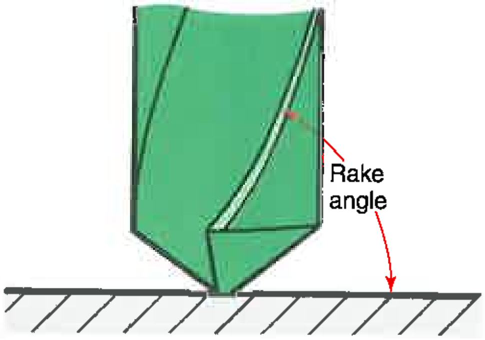


Figure 10-63. Rake angle of the drill.

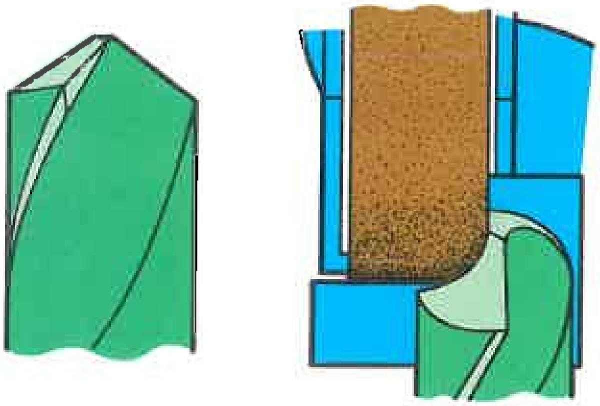


Figure 10-64. Modified rake angle for drilling brass.

10.8.2 Drill Grinding Attachments

A drill sharpening device is shown in **Figure 10-65.** An attachment for conventional grinders is shown in **Figure 10-66.** In the machine shop where a high degree of hole accuracy is required and a large amount of sharpening must be done, these devices are a must.

10.9 DRILLING

Obeying a few simple rules will help you drill accurately. Use the following procedure:

1. Carefully study the drawing to determine hole locations. Lay out the positions and mark the intersecting lines with a prick punch.
2. Secure a drill and check its size.
3. Mount work solidly on the drilling machine. Never hold the work by hand. The workpiece could whip out of your hand and cause serious injuries.
4. Insert a ***wiggler*** or ***center finder*** in the drill chuck to position the point to be drilled directly under the chuck or spindle. Turn on the power and center the wiggler point with your fingers. Position the work until the revolving wiggler point does not wiggle when it is lightly dropped into the punched hole location and removed. If there is any point movement, additional alignment is necessary because the work is not positioned properly, **Figure 10-67**.

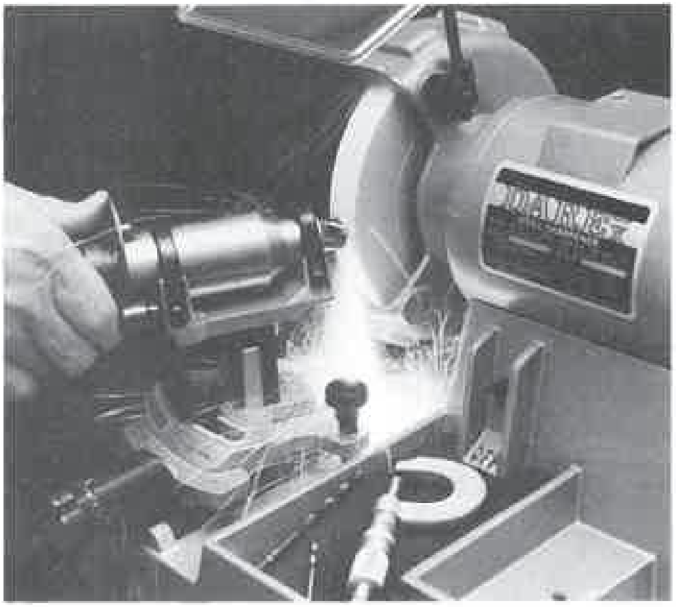


Figure 10-65. A drill sharpening grinder attachment ensures that cutting edges of the drill will be uniform. (Darex Corp.)

Figure 10-66. A drill and tool grinder with built-in point splitting and web thinning capabilities. The user can choose from a wide variety of drill points. Machines of this type are available in semiautomatic and automatic versions. (Rush Machinery, Inc.)

Figure 10-67. It is difficult to align a workpiece with centerlines by eye. To assist in this job, a center finder, or wiggler, is used.

5. Remove the wiggler and insert a center drill. Hand tighten the chuck, **Figure 10-68**. Check to be sure the drill runs true. If it does, tighten the chuck with a chuck key. Remember to remove the key before starting the machine.

Figure 10-68. After alignment with the wiggler, center drilling will ensure that the drill will make the hole in the proper location.

6. After center drilling, replace the tool with the required drill. Hand tighten it in the chuck. Turn on the machine. If it does not run true, the drill may be bent or may have been placed in the chuck off center. Also check that it will drill to the required depth.
7. Calculate the correct cutting speed and feed if you plan to use a power feed. Adjust the machine to operate as closely as possible to this speed.
8. Turn on the power and apply cutting fluid. Start the cut. Even pressure on the feed handle will keep the drill cutting freely.
9. Watch for the following signs that indicate a poorly cutting drill:
   - A dull drill will squeak and overheat. Chips will be rough and blue, and cause the machine to slow down. Small drills will break.
   - Infrequently, a chip will get under the dead center and act as a bearing, preventing the drill from cutting. Remove it by raising and lowering the drill several times.
   - Chips packed in the flutes will cause the drill to bind and slow the machine or cause the drill to break. Remove the drill from the hole and clean it with a brush that has been dipped in cutting fluid. Do not use cutting fluid when drilling cast iron.
10. Clear chips and apply cutting fluid as needed.
11. The most critical time of the drilling operation occurs when the drill starts to break through the work. Ease up on feed pressure at this point to prevent the drill from "digging in."
12. Remove the drill from the hole and turn off the power. Never try to stop the chuck with your hand. Clear the chips with a brush. Unclamp the work and use a file to remove all burrs.
13. Clean chips and cutting fluid from the machine. Wipe it down with a soft cloth. Return equipment to storage after cleaning.

Observe extreme care in positioning the piece for drilling. A poorly planned setup may permit the drill to cut into the vise or drill table when it breaks through the work.

If a hole must be located precisely, certain additional precautions can be taken to ensure that the hole will be drilled where it is supposed to be drilled. After the center point has been determined, a series of ***proof circles*** are scribed, **Figure 10-69A.** They will serve as reference points to help check whether the drill remains on center as it starts to penetrate the material.

Even when work is properly centered, the drill may "drift" when starting a hole. Various factors can cause this, such as hard spots in the metal or an improperly sharpened drill. The drill cannot be brought back on center by moving the work, because it will still try to follow the original hole. This condition, **Figure 10-69B,** must be corrected before the full diameter of the drill is reached.

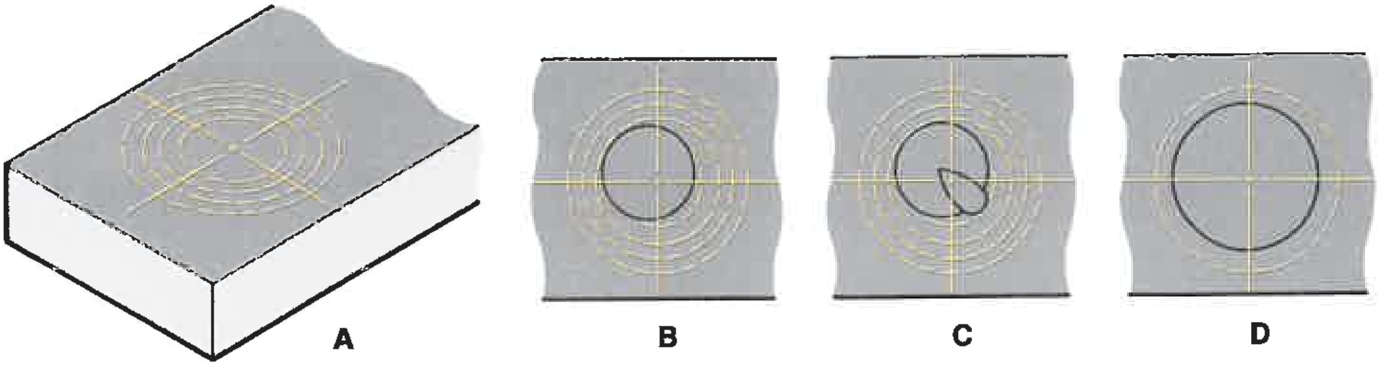


Figure 10-69. How to bring a drill back on center. A—Proof circles. B—Drill has been started off center (exaggerated). C—Groove cut to bring drill back on center. D—Drill back on center. This operation will only work if the drill has not begun to cut to its full diameter.

The drill is brought back on center by using a round-nose cape chisel to cut a groove on the side of the hole where the drill must be drawn, **Figure 10-69C.** This groove will "pull" the drill point to the center. Repeat the operation until the hole is centered in the proof circles, **Figure 10-69D.**

10.9.1 Drilling Larger Holes

Drills larger than 1/2" (12.5 mm) diameter require considerable power and pressure to get started. Even then, they may run off center. The pressure can be greatly reduced and accuracy improved by first drilling a ***pilot hole*** (***lead hole***) that is smaller in diameter than the final hole. See **Figure 10-70.**

The small pilot hole permits pressure to be exerted directly on the cutting edges of the large drill, causing it to drill faster. The diameter of the pilot hole should be as large as, or slightly larger than, the width of the dead center.

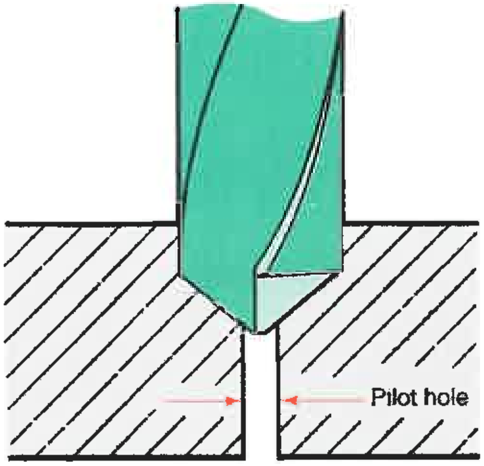


Figure 10-70. A pilot hole makes drilling a large hole much easier.

10.9.2 Drilling Round Stock

Holes are more difficult to drill in the curved surface of round stock. Many difficulties can be eliminated by holding the round material in a V-block, **Figure 10-71.** A V-block can be held in a vise or clamped directly to the table.

Use the following procedure to center round stock in a V-block:

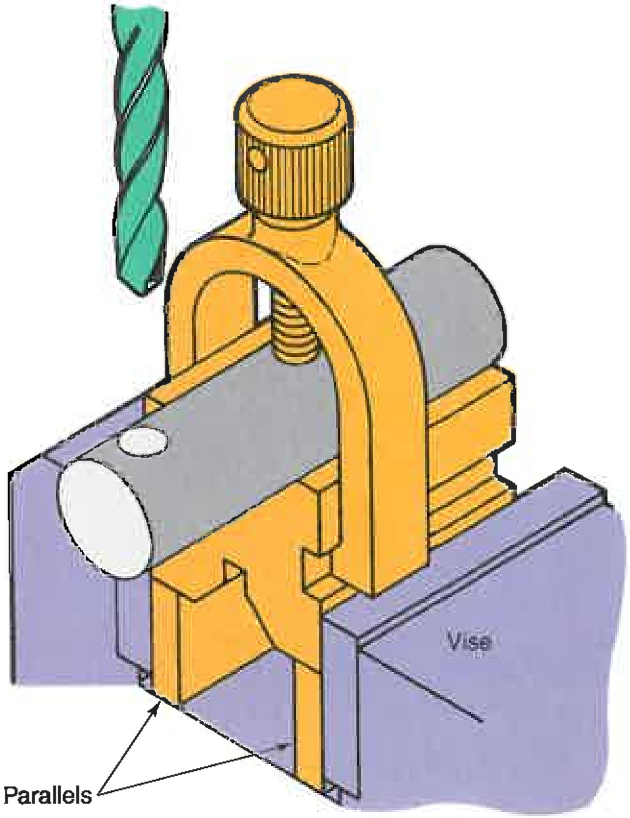


Figure 10-71. The V-block eliminates many difficulties when drilling round stock. Be sure the drill will clear the V-block when it comes through material.

1. Locate the hole position on the stock. Prick punch the intersection of the layout lines. Place the stock in a V-block. If the hole is to go through the piece, make certain that the drill will clear the V-block. Also be sure there is ample clearance between the clamp and drill chuck.
2. To align the hole for drilling through exact center, place the work and V-block on the drill press table or on a surface plate. Rotate the punch mark until it is upright. Place a steel square on the flat surface with the blade against the round stock as shown in **Figure 10-72.** Measure from the square blade to the punch mark, and rotate the stock until the measurement is the same when taken from both sides of the stock.
3. From this point, the drilling sequence is identical to that previously described.

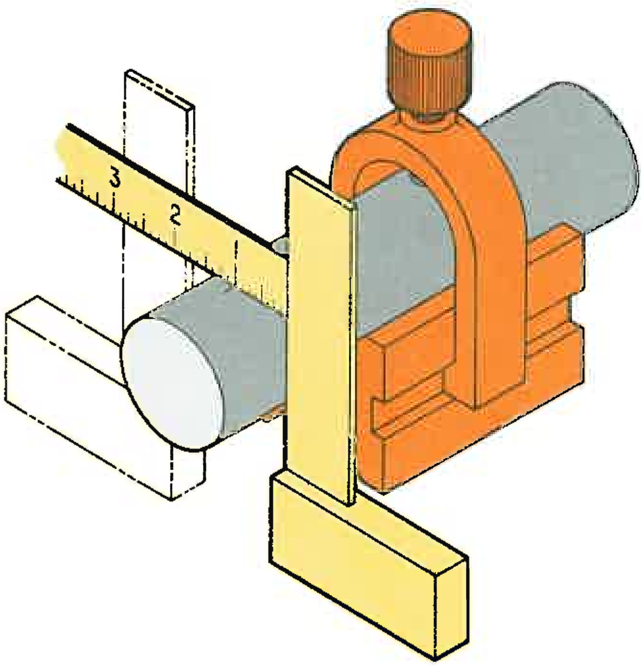
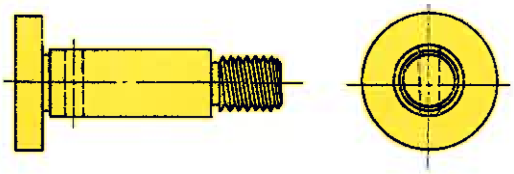


Figure 10-72. *Using a square to center round stock in a V-block.*

If a large number of identical parts must be drilled, it may be desirable to make a drill jig, **Figure 10-73.** The drill jig automatically positions and centers each piece for drilling.



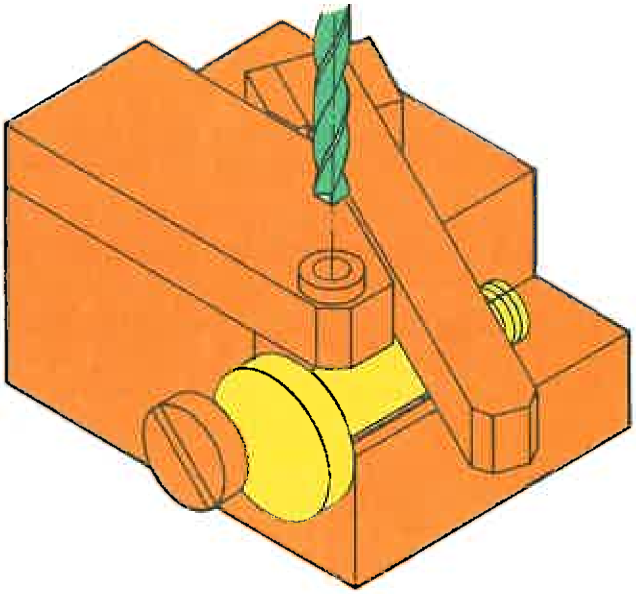


Figure 10-73. *This typical drill jig has an arm that lifts to allow easy insertion and removal of the part being drilled.*

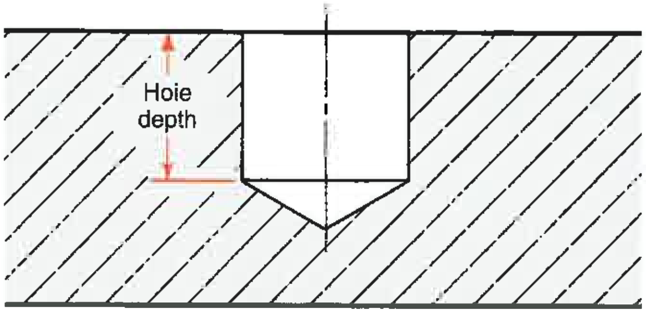


Figure 10-74. *Measuring the depth of a blind hole.*

10.9.3 Blind Holes

A ***blind hole*** is a hole that is not drilled all the way through the work. ***Hole depth*** is measured by the distance the *full hole diameter* goes into the work, **Figure 10-74.** Using a drill press fitted with a ***depth stop*** or ***depth gage*** is the quickest means of achieving proper depth when drilling blind holes, **Figure 10-75.**

10.10 COUNTERSINKING

Countersinking is the operation that cuts a chamfer in a hole to permit a flat-headed fastener to be inserted with the head flush to the surface, **Figures 10-76.**

The tool used to machine countersinks is called a ***countersink,*** **Figure 10-77.** Countersinks are available with cutting edge angles of 60°, 82°, 90°, 100°,

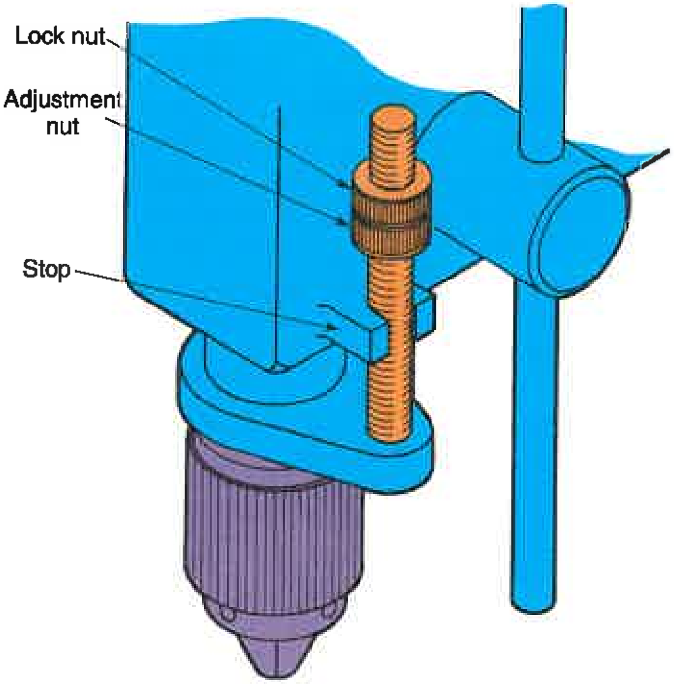


Figure 10-75. The depth gage attachment provides easy adjustment of how far the drill moves into the work.

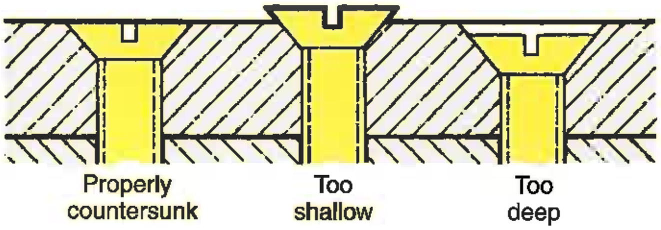


Figure 10-76. Correctly and incorrectly countersunk holes. The countersink angle must match the fastener head angle.

Figure 10-77. Countersinks come in various sizes. (Greenfield Tap & Die)

110°, and 120° included angles. Countersinks are also used for deburring holes.

Countersinks with indexable carbide inserts, **Figure 10-78,** are available in a number of sizes and point angles. They have two cutting edges per insert and do not require resharpening. Cutting speeds are five to ten times higher than with HSS countersinks.

A single cutting edge countersink, **Figure 10-79,** is free cutting and produces minimum chatter. Chips produced by the cutting edge pass through the hole and are ejected.

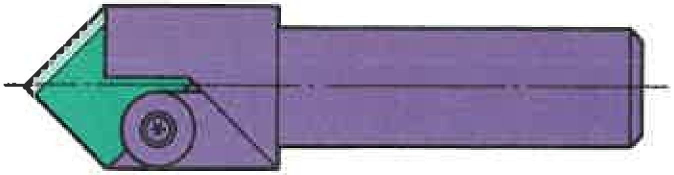


Figure 10-78. Countersinks with indexing carbide inserts have a life five to ten times longer than similar HSS countersinks.

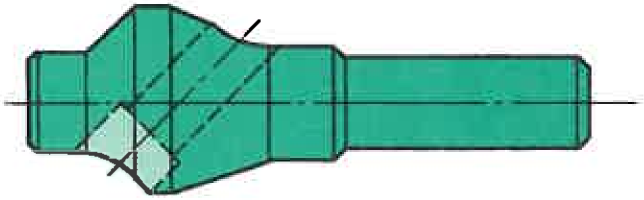


Figure 10-79. Countersink with a single cutting edge and pilot.

10.10.1 Using a Countersink

1. The cutting speed should be about one-half that recommended for a similar size drill. This will minimize the probability of chatter.
2. Feed the tool into the work until the chamfer is large enough for the fastener head to be flush.
3. Use the **depth** stop on the drill press if a number of similar holes must be countersunk.

10.11 COUNTERBORING

The heads of fillister-head and socket-head screws are usually set below the work surface. A ***counterbore*** is used to enlarge the drilled hole to the proper depth and machine a square shoulder on the bottom to secure maximum clamping action from the fastener, **Figure 10-80.**

The counterbore tool has a guide, called a pilot, which keeps it positioned correctly in the hole. Solid counterbores are available. However, counterbores with interchangeable pilots and cutters are commonly used, **Figure 10-81.** They can be changed easily from one size cutter or pilot to another size. A drop of oil on the pilot will prevent it from binding in the drilled hole.

Counterbores with indexable carbide inserts, **Figure 10-82,** are also available. When the cutting edges become dull, new edges can be indexed into

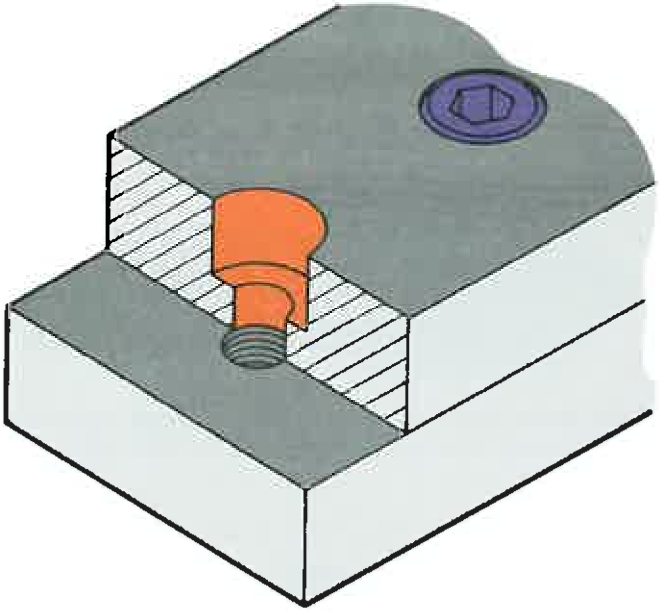


Figure 10-80. *A sectional view of a hole that has been drilled and counterbored to receive a socket-head screw.*

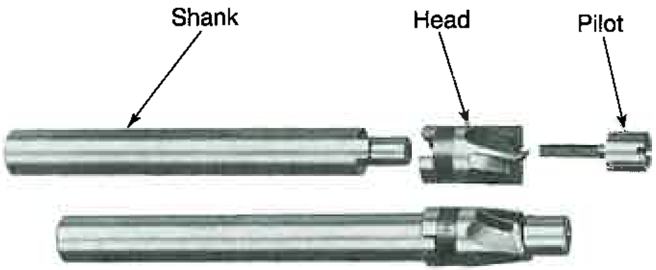


Figure 10-81. *A straight shank interchangeable counterbore. Head and pilot can be quickly changed.*

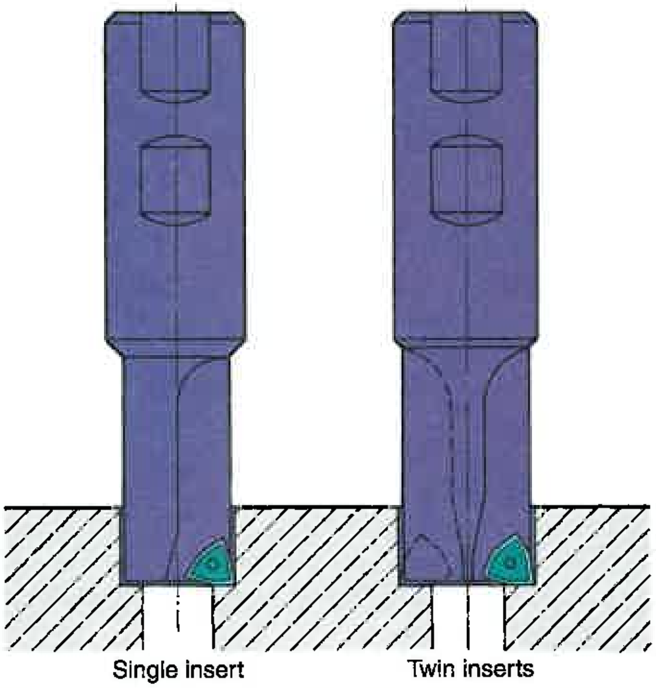


Figure 10-82. *Counterbores with carbide indexable inserts. The inserts are rotated when a cutting edge becomes dull.*

place without affecting opening diameter. Costly sharpening is eliminated.

10.12 SPOTFACING

Spotfacing is the operation during which a circular spot is machined on a rough surface (such as a casting or forging) to provide a bearing surface for the head of bolt, washer, or nut. A counterbore may be used for spotfacing, although a special tool manufactured for inverted spotfacing is available, **Figure 10-83.**

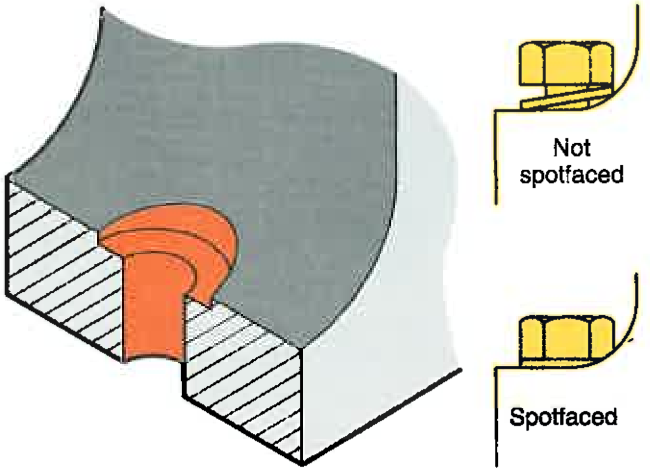


Figure 10-83. *Sectional view of a casting with a mounting hole that has been spotfaced. Profile drawings show the casting before and after spotfacing. The bolt head cannot be drawn down tightly until the mounting hole is spotfaced.*

Special ***backspotface*** and ***backcounterbore*** tools are required to perform operations in areas where conventional tools cannot be used. See **Figure 10-84.** The cutting point is lifted up into the workpiece, rather than being pushed down into it. See **Figure 10-85.**

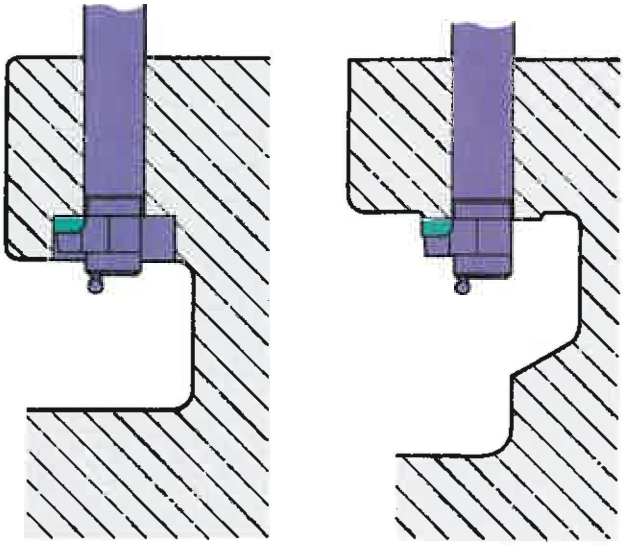
Large diameter openings can be counterbored, spotfaced, or drilled with the APT Multi-Tool™, **Figure 10-86.** A pilot hole is drilled with a conventional twist drill. The required size pilot and blade is inserted and the opening is made. For very large openings, it may be necessary to use a smaller-size blade before machining the specified size. Multi-Tool blades are available in sizes from 1 1/8″ diameter to 4″ diameter.

10.13 TAPPING

Tapping may be done by hand on a drill press using the following steps:

1. Drill the correct size hole for the tap, **Figure 10-87.**
2. With the work clamped in the machine, insert a small 60° center in the chuck. The center holds the tap vertically.

Figure 10-84. *Special backspotface and backcounterbore tools are used in situations where conventional tools cannot be inserted. Blade setting can be manual or automatic. Standard tools are available for bores and spotfaces of 0.250 and larger. (Parlec, Inc.)*



Counterboring

Spotfacing

Figure 10-85. *These section drawings show how backcounterboring and backspotfacing tools work.*

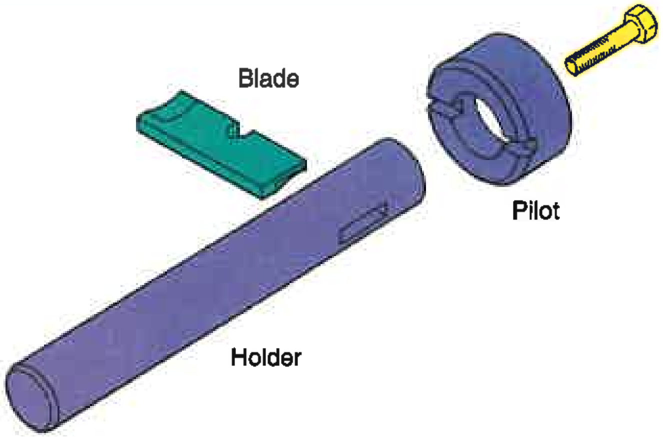


Figure 10-86. *Interchangeable drilling, spotfacing, and counterboring tool. Blades for producing holes up to 4″ diameter are available.*

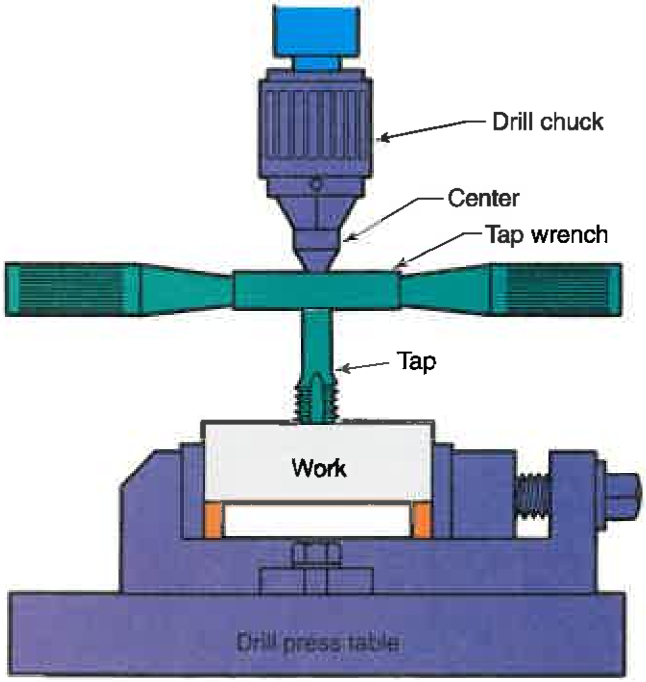


Figure 10-87. *Setup for hand tapping on a drill press. Work must be mounted solidly.*

3. Place the center point in the tap's center hole.
4. Feed the tap into the work by holding down on the feed handle and turning the tap with a ***tap wrench***.

Never insert a tap into the drill chuck and attempt to use the drill press power to run the tap into the work. The tap will shatter when power is applied. Turn the tap by hand.

Tapping can only be done with power through the use of a ***tapping attachment,*** **Figure 10-88.** This device fits the standard drill press. It has reducing gears that slows the tap to about one-third of the drill press speed. A table provided with the attachment gives recommended spindle speeds for tapping.

A clutch arrangement drives the tap until it reaches the predetermined depth, at which time the tap stops rotating. Raising the feed handle causes the tap to reverse direction and back out of the hole.

Specially designed ***microdrilling machines,*** **Figure 10-89,** are required for drilling holes as small as 0.0016″ (0.04 mm) with close tolerances. These drills have very accurate spindles and collets to reduce drill flexing and breakage. Many of the microdrilling machines are controlled by computer numerical control (CNC) systems.

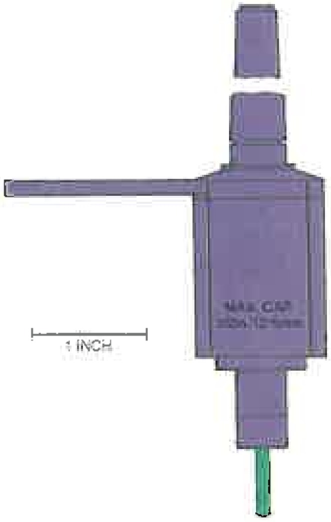


Figure 10-88. *Tapping attachment on a drill press.*

Figure 10-89. *CNC microdrilling machine with a point-to-point programmable control system. The table has 2″ × 2″ (50 mm × 50 mm) travel with an accuracy of 0.0001″ (0.0025 mm). Variable speed control for the drill head motor is controlled directly from the computer. (Minitool Inc.)*

Microdrilling uses a "pecking" technique to cut these small diameter holes. In this technique, the drill is repeatedly inserted and removed from the hole. The drills have flutes to pull chips out of the hole, but because they are so small, pecking is necessary for chip removal. The depth of each peck is determined by the drill diameter and the material being drilled.

Small-size holes with other geometric shapes (such as square, rectangular, or hexagonal) are made by Electrical Discharge Machining (EDM). This topic is discussed in Chapter 27, Electromachining Processes.

10.14 REAMING

Reaming produces holes that are extremely accurate in diameter and have an exceptionally fine surface finish. Machine reamers are made in a variety of sizes and styles. They are usually manufactured from high-speed steel. Some are fitted with carbide cutting edges. Descriptions of a few of the more common machine reamers follow. Refer to **Figure 10-90.**

- A ***jobber's reamer,*** also called a ***machine reamer,*** is identical to a hand reamer except that a taper shank is available and the tool is designed for machine operation.
- A ***chucking reamer*** is manufactured with both straight and taper shanks. It is similar to a jobber's reamer but its flutes are shorter and deeper. It is available with straight or spiral flutes.
- A ***rose chucking reamer*** is designed to cut on its end. The flutes provide chip clearance and are ground to act only as guides. This type of reamer is best used when considerable metal must be removed and the finish is not critical.
- A ***shell reamer*** is mounted on a special arbor that can be used with several reamer sizes. The arbor can have straight or spiral flutes and is also made in the rose style. The arbor shank may be straight or tapered. A hole in the reamer is tapered to fit the arbor, which is fitted with drive lugs.
- An ***expansion chucking reamer*** is available with straight flutes and either a straight or taper shank. Slots are cut into the body to permit the reamer to expand when an adjusting screw in the end is tightened.

A regular expansion reamer has several drawbacks. The slots, which are necessary for the reamer

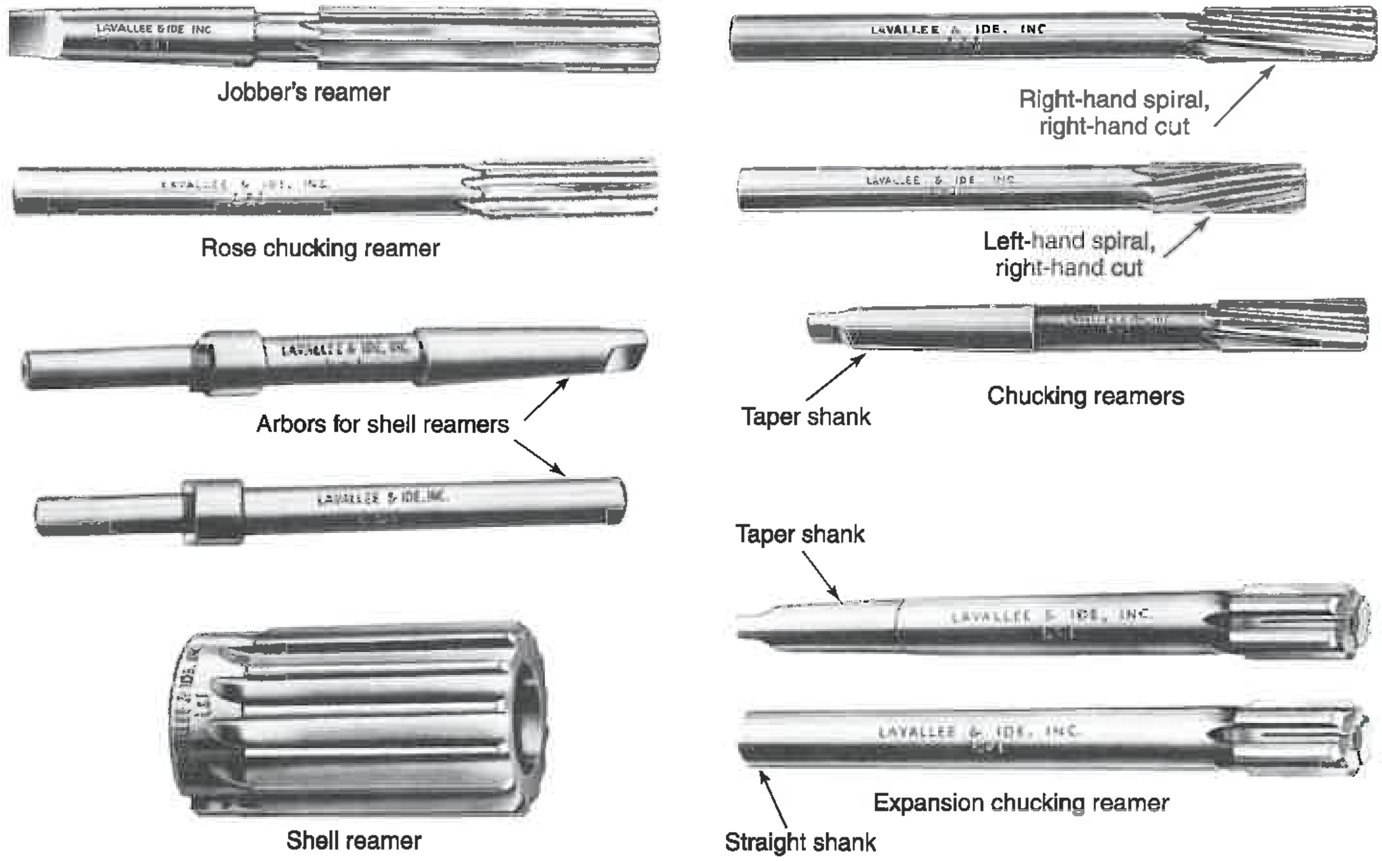


Figure 10-90. *Each type of reamer is suited for a different application.*

to expand, reduce tool rigidity. This diminishes accuracy and surface finish. Also, cutting-edge clearance is reduced as the reamer expands, creating a "drag." This often causes the tool to chatter with a resulting decrease in finish quality.

A *solid* expansion reamer provides rigidity and accuracy not possible with conventional expansion reamers. See **Figure 10-91.** To expand this type, a tapered plug is forced into the reamer end. The tool body expands well beyond the tip, and ensures uniform parallel expansion across the full length of the carbide cutting lips. Clearance is automatically provided. The plug can be removed for shimming to a larger size. Once expanded, the reamer diameter cannot be reduced without grinding.

Figure 10-91. *This solid expansion reamer has tungsten carbide cutting edges for extended cutting life. (Standard Tool Co.)*

10.14.1 Using Machine Reamers

Reamers are expensive precision tools. The quality of the finish and accuracy of the reamed hole will depend on how the tool is used. Obey the following reaming rules:

- Carefully check the reamer diameter before use. If the hole diameter is critical, drill and ream a hole in a piece of similar material to check tool accuracy.
- Use a sharp reamer.
- Mount the reamer solidly.
- Cutting speed for a high-speed steel reamer should be about two-thirds that of a similar size drill.
- Feed should be as high as possible while still providing a good finish and accurate hole size.
- Allow enough material in the drilled hole to permit the reamer to cut rather than burnish (smooth and polish). The following allowances are recommended:
  - Up to 1/4" (6.3 mm) diameter, allow 0.010" (0.25 mm).
  - 1/4" to 1/2" (6.3 mm to 12.5 mm) diameter allow 0.015" (0.4 mm).
  - 1/2" to 1.0" (12.5 mm to 25.0 mm) diameter allow 0.020" (0.5 mm).
  - 1.0" to 1.5" (25.0 mm to 38.0 mm) diameter allow 0.025" (0.6 mm).
- Use an ample supply of cutting fluid.

- Remove the reamer from the hole before stopping the machine.
- When not being used, reamers should be stored in separate containers or storage compartments. This will minimize chipping and dulling of the cutting edges.

TEST YOUR KNOWLEDGE

Please do not write in the text. Write your answers on a separate piece of paper.

1. A twist drill works by:
   a. Being forced into material.
   b. Rotating against material and being pulled through by the spiral flutes.
   c. Rotating against material with sufficient pressure to cause penetration.
   d. All of the above.
   e. None of the above.
2. How is drill press size determined?
3. Drills are made from:
   a. High-speed steel.
   b. Carbon steel.
   c. Both of the above.
   d. Neither of the above.
4. Drill sizes are expressed by what four series?
5. What are two techniques used to determine a drill's size?
6. List the two types of drill shanks.
7. _____ shank drills are used with a chuck.
8. _____ shank drills fit directly into the drill press spindle.
9. The spiral grooves that run the length of the drill body are called _____.
10. The spiral grooves in a drill body are used to:
    a. Help form the cutting edge of the drill point.
    b. Curl chips for easier removal.
    c. Form channels through which the chips can escape from the hole.
    d. All of the above.
    e. None of the above.
11. Name the device employed to enlarge a taper shank drill so it will fit the spindle opening.
12. The device used to permit a drill with a taper shank too large to fit the spindle opening is called a(n)_____.
13. What is the name of the tool used to separate a taper shank drill from the above devices?
14. Cutting compounds or fluids are used to:
    a. Cool the drill.
    b. Improve the finish of a drilled hole.
    c. Aid in the removal of chips.
    d. All of the above.
    e. None of the above.
15. List the three factors that must be considered when repointing a drill.
16. What occurs when the cutting lips of a drill are not sharpened to the same lengths?
17. The _____ should be used frequently when sharpening to ensure a correctly sharpened drill.
18. The included angle of a drill point sharpened for general drilling is _____ degrees.
19. What coolant should be used when drilling cast iron?
20. Large drills require a considerable amount of power and pressure to get started. They also have a tendency to drift off center. These conditions can be minimized by first drilling a _____ hole. This hole should be as large as, or slightly larger than, the width of the _____ of the drill point.
21. What is a blind hole?
22. How is the depth of a drilled hole measured?
23. The _____ is almost identical to the hand reamer except that the shank has been designed for machine use.
24. A(n) _____ expansion reamer provides rigidity and accuracy not possible with conventional expansion reamers.
25. How should a reamer be removed from a finished hole?
26. The cutting speed for a high-speed reamer is approximately _____ that for a similar-sized drill.
27. What is the name of the operation employed to cut a chamfer in a hole to receive a flat-head screw?
28. The operation used to prepare a hole for a fillister or socket head screw is called _____.
29. _____ is the operation that machines a circular spot on a rough surface for the head of a bolt or nut.

Chapter 11

Offhand Grinding

LEARNING OBJECTIVES

After studying this chapter, you will be able to:

- ○ Identify the various types of offhand grinders.
- ○ Dress and true a grinding wheel.
- ○ Prepare a grinder for safe operation.
- ○ Use an offhand grinder safely.
- ○ List safety rules for offhand grinding.

IMPORTANT TERMS

abrasive belt grinding machines
bench grinder
concentricity
flexible shaft grinders
pedestal grinder
precision microgrinder
reciprocating hand grinder
temper
tool rest
wheel dresser

Grinding is an operation that removes material by rotating an abrasive wheel or belt against the work, **Figure 11-1.** It is used for the following tasks:

- Sharpening tools.
- Removing material too hard to be machined by other techniques.
- Cleaning the parting lines from castings and forgings.
- Finishing and polishing molds used in die casting of metals and injection molding of plastics.

11.1 ABRASIVE BELT GRINDERS

Abrasive belt grinding machines are heavy-duty versions of the belt and disc sanders found in woodworking, **Figure 11-2.** A wide variety of abrasive belts permits these machine tools to be used for grinding to a line, finishing cast and forged parts, deburring, contouring, and sharpening. See **Figure 11-3.**

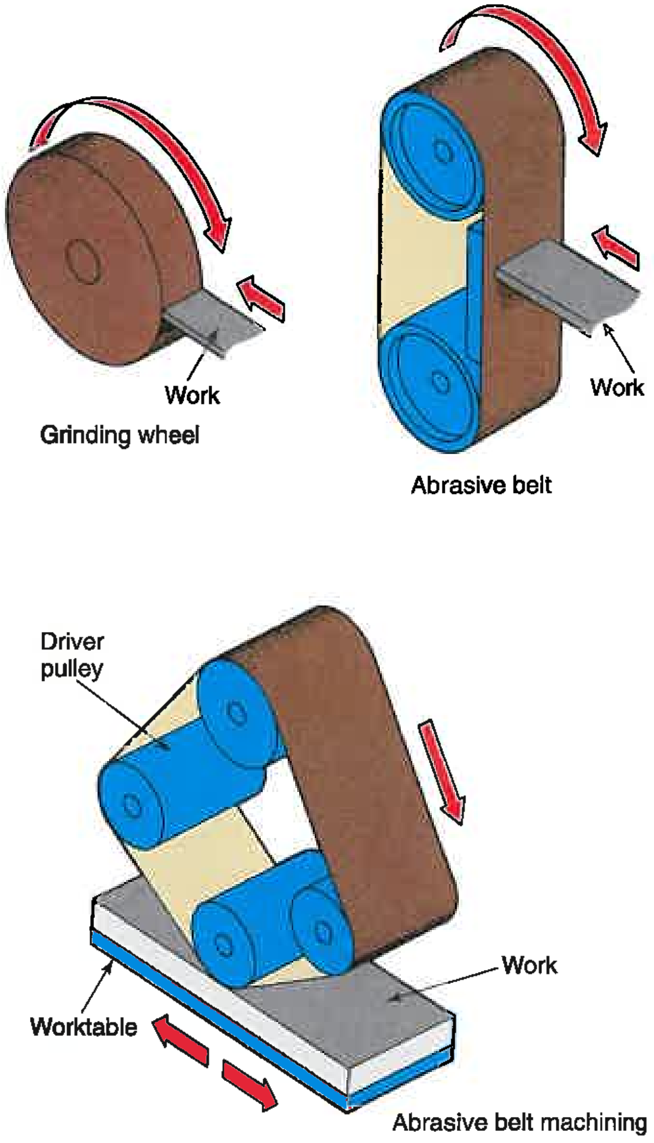


Figure 11-1. *Principles of how typical grinding machines work.*

11.2 BENCH AND PEDESTAL GRINDERS

The bench grinder and pedestal grinder are the simplest and most widely used grinding machines.

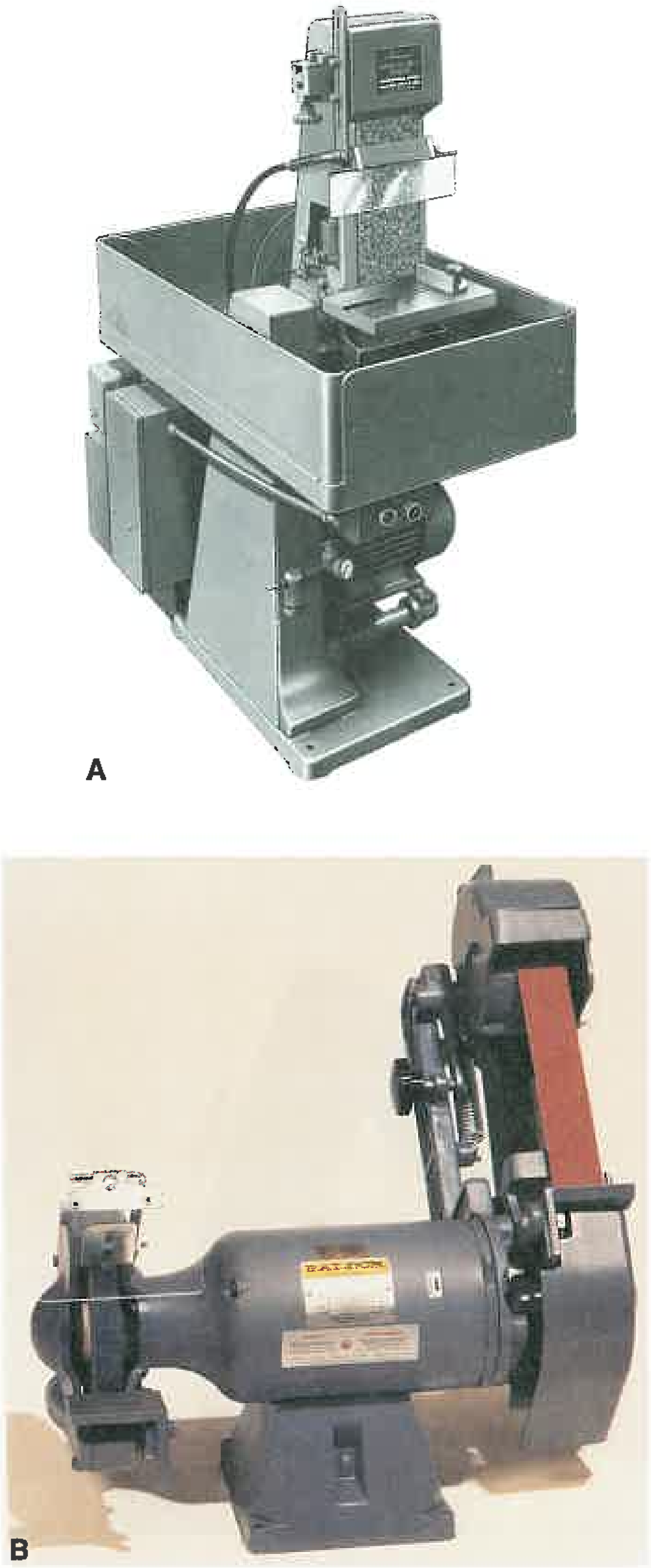


Figure 11-2. *Abrasive belt grinding machines. A—Wet-type abrasive belt platen grinder sprays water on the belt to cool the work. It can be quickly changed from vertical to horizontal. B—Flexible-belt grinder adapts to three-dimensional contours. (Hammond Machinery, Inc. and Baldor)*

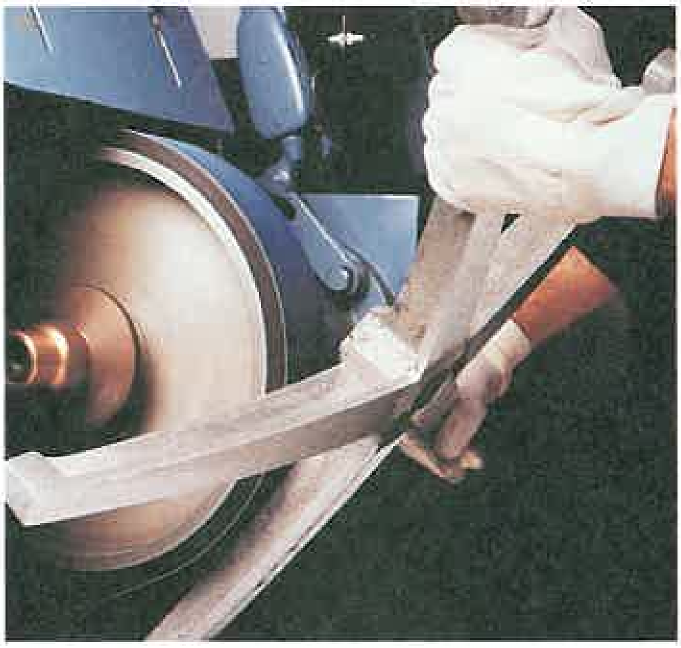


Figure 11-3. *Abrasive belt being used to clean up an aluminum casting of a base for an office chair. (American Foundrymen's Society)*

Grinding done on a bench, pedestal, or belt grinder is called ***offhand grinding.*** This type of work does not require great accuracy. The part is held in your hands and manipulated until it is ground to the desired shape.

The ***bench grinder*** is one that has been fitted to a bench or table, **Figure 11-4.** The grinding wheels mount directly onto the motor shaft. Normally, one wheel is ***coarse,*** for roughing, and the other is ***fine,*** for finish grinding.

Figure 11-4. *A bench grinder can be used for many tasks. Never operate a bench or pedestal grinder unless all safety devices are in place and in sound condition. The tool rest must be properly spaced and eye protection must be worn, even though the grinder has eye shields. (Baldor)*

A *pedestal grinder* is usually larger than the bench grinder and is equipped with a pedestal (base) fastened to the floor. See **Figure 11-5.** The ***dry-type pedestal grinder*** has no provisions for cooling the work during grinding other than a water container. The part is dipped into the water. A ***wet-type pedestal grinder,* Figure 11-6,** has a coolant system built into the grinder. This system keeps the wheels constantly flooded with fluid. The coolant washes away particles of loose abrasive material and metal and cools the work. Cooling prevents localized heat buildup, which can ruin tools and "burn" areas of other types of work.

Wear safety glasses and be sure the grinder eye shield is in place before doing any grinding.

The ***tool rest*** is provided to support the work being ground. It is recommended that the rest be adjusted to within 1/16" (1.5 mm) of the wheel, **Figure 11-7.** This will prevent the work from being wedged between the rest and the wheel. After adjusting the rest, turn the wheel by hand to be sure there is sufficient clearance.

Do not make tool rest adjustments while the grinding wheels are revolving.

Figure 11-5. *This pedestal grinder has a roughing wheel on the left and a finishing wheel on the right. (Baldor)*

Figure 11-6. *This grinder has a coolant attachment that keeps coolant dripping on the tool being sharpened. (Baldor)*

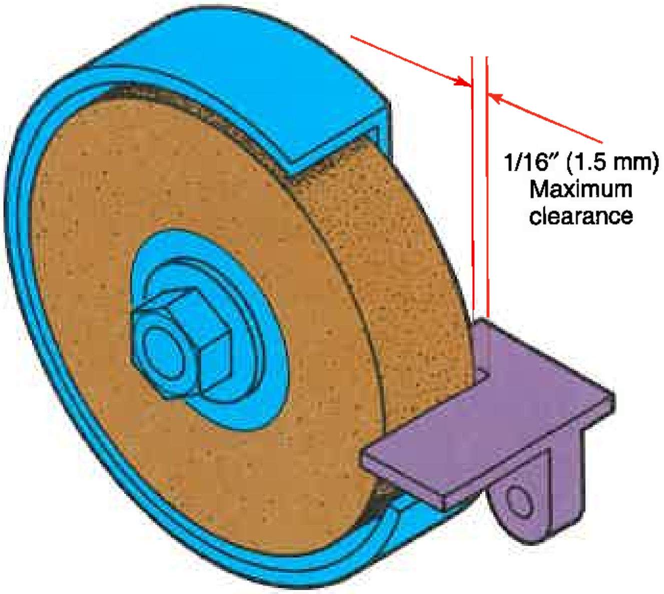


Figure 11-7. *The tool rest must be spaced properly for safety. Maximum safe clearance is 1/16" (1.5 mm).*

11.3 GRINDING WHEELS

Grinding wheels can be a source of danger and should be examined frequently for ***concentricity*** (running true), roundness, and cracks. A new wheel can be tested by suspending it on a string or wire, or holding it lightly with one finger, and tapping the side lightly with a metal rod or screwdriver handle. A solid wheel will give off a clear ringing sound. A wheel that does not give off a clear sound should be assumed to have a fault.

11.3.1 Wheel Dresser

The grinding wheels must run true and be balanced on the shaft. A ***wheel dresser,*** **Figure 11-8,** is used to true the wheel and remove any glaze that may have formed during grinding operations.

The wheel dresser is supported on the tool rest and is held firmly against the wheel with both hands. It is moved back and forth across the wheel face to remove a thin layer of stone. See **Figure 11-9.**

Figure 11-8. *A mechanical wheel dresser will true stone face. (Hammond Machinery Inc.)*

The appearance of the grinding wheel surface indicates the amount of glaze. A wheel dresser should be used to remove the glaze. **Figure 11-10** shows grinding wheel conditions.

11.3.2 Grinding Rules

To obtain maximum efficiency from a grinder, the following recommendations should be observed:

- Grind using the face of a wheel, not the sides.
- Move the work back and forth across the wheel face. This will wear the wheel evenly and prevent grooves from forming.
- Keep the wheel dressed and the tool rests properly adjusted.
- Soft metals (aluminum, brass, and copper) tend to load (clog) grinding wheels. When possible, these metals should be ground on an abrasive belt grinder.

11.4 ABRASIVE BELT AND GRINDER SAFETY

- Make sure the tool rest is properly adjusted.
- Wear goggles or a face shield when performing grinding operations, even though the machines are fitted with eye shields.
- Never attempt to operate an offhand grinding machine while your senses are impaired by medication or other substances.
- Check the machine thoroughly before using it. Lubricate the machine only as recommended by the manufacturer.

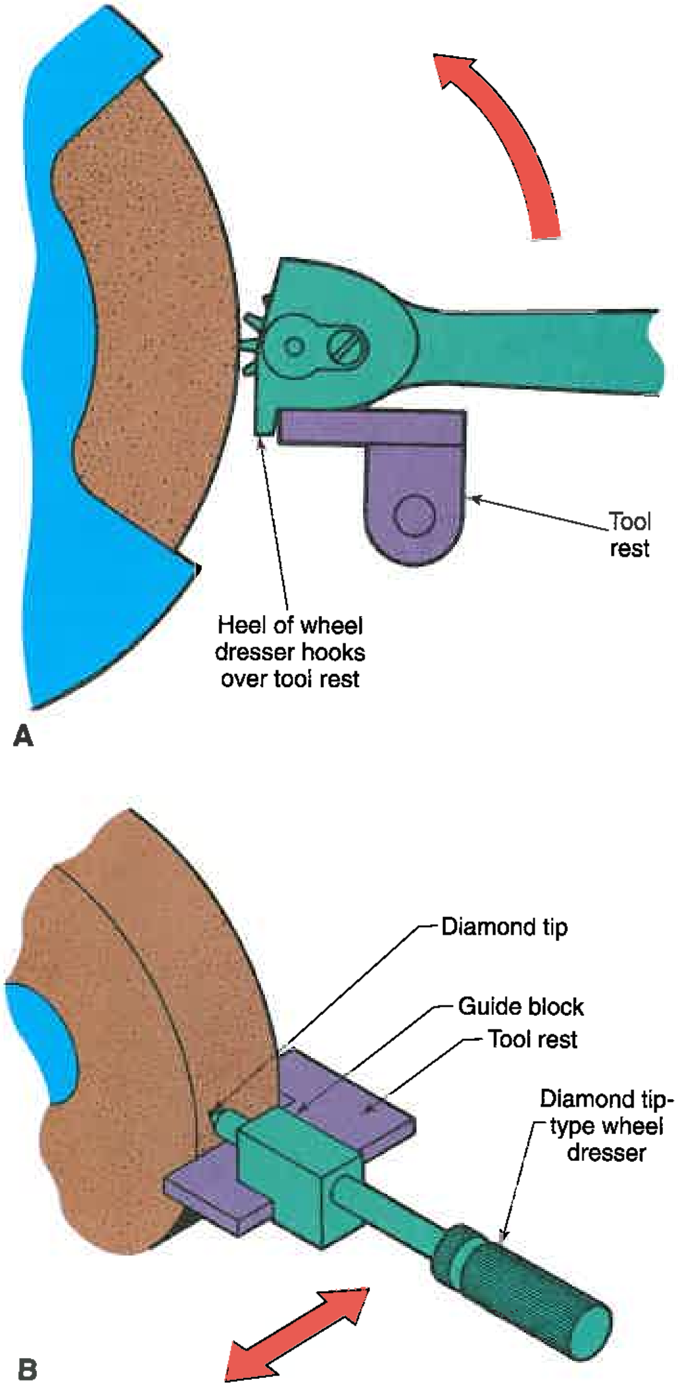


Figure 11-9. *The proper way to use a mechanical wheel dresser. A—Move the tool back and forth over the face of the stone. Wear a dust mask, eye protection, and an apron when dressing wheels on grinders. B—Industrial diamonds are also used to dress and true grinding wheels. The guide block is used for grinders with slotted tool rests.*

- Check a grinding wheel for soundness before putting it on the grinder. Destroy wheels that are not sound or that have a worn center hole.
- Do not use a wheel that is glazed or loaded with metal.

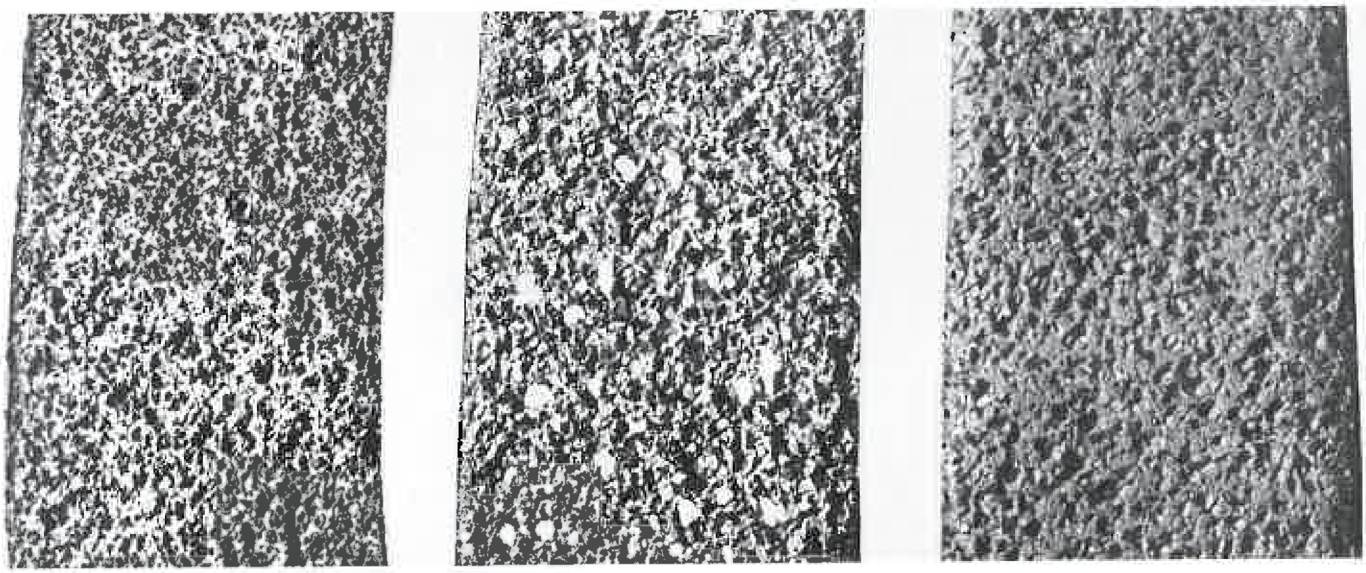


Figure 11-10. Grinding wheels in various conditions. A—Properly dressed. B—Loaded. C—Glazed. (Norton Co.)

- Be sure all wheel guards and safety devices are in place before attempting to use a grinder or abrasive belt machine.
- If the grinding operation is to be performed dry, be sure to hook up all exhaust attachments before starting.
- Stand to one side of the machine during operation. Do not stand directly in front of the wheel.
- Hold small work in a clamp or hand vise. Under no condition should work be held with a cloth.
- Avoid work pressure on the side of the grinding wheel.
- Keep your hands clear of the rotating wheel.
- Never operate a grinding wheel at speeds higher than those recommended by the manufacturer.
- Have injuries caused by rotating grinding wheels treated immediately.
- Allow the wheels or belt to stop completely before attempting to make any machine adjustments.

11.5 USING A DRY-TYPE GRINDER

After examining the grinder and making the necessary adjustments, turn on the machine. Be sure that you wear safety glasses whenever you are in the shop. Stand to one side until the grinder has reached operating speed.

Place the work on the tool rest and slowly push it against the grinding wheel. If too much pressure is applied, the work will begin to "burn" or discolor. Overheating can be minimized by dipping the work into the water container from time to time. Care must be taken when grinding edge tools because excessive heat will "draw" (remove) the ***temper*** (hardness) and ruin the tool.

Keep the work moving across the wheel face to prevent the formation of grooves or ridges. Dress and retrue the wheel as neccessary for maximum efficiency.

Pieces of cloth should never be used to hold work while it is being ground. Serious injuries can result if the cloth is pulled into the wheel. Hold the work, especially small lathe cutter bits, in hand vises specially designed for that purpose, **Figure 11-11.**

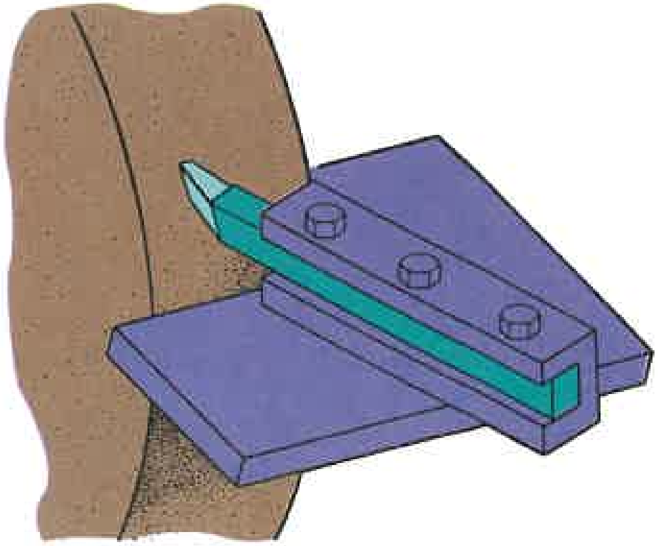


Figure 11-11. A hand vise is used to hold cutter bits while they are sharpened.

11.6 USING A WET-TYPE GRINDER

The wet-type grinder is primarily used to grind carbide-tipped tools. Since a carbide tool is often brazed onto a steel shank, both steel and carbide must be ground away when these tools are sharpened. Aluminum oxide wheels should be used to grind the steel shank and silicon carbide or diamond-impregnated wheels to grind the carbide tip.

Wet-type grinders should normally have a flat face, but a slightly crowned face should be used when grinding carbide-tipped tools, **Figure 11-12.** The crown minimizes the contact between the wheel and the work. This reduces the possibility of the tip being damaged by excessive heat.

The coolant attachment must be adjusted to keep a full flow of liquid directed on the tool at all times. Adjust the tool table rest to obtain the correct clearance angle, **Figure 11-13.** A protractor guide is helpful when compound clearance angles are required.

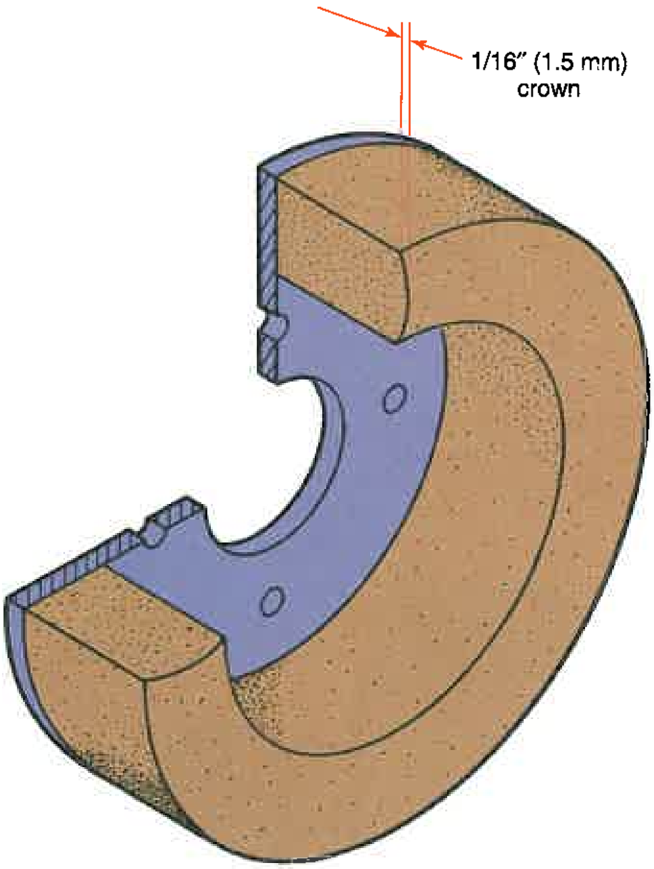


Figure 11-12. *A slight crown on the wheel face minimizes the amount of contact between the work and the wheel. A crown reduces the possibility of heat destroying the carbide tip of a tool.*

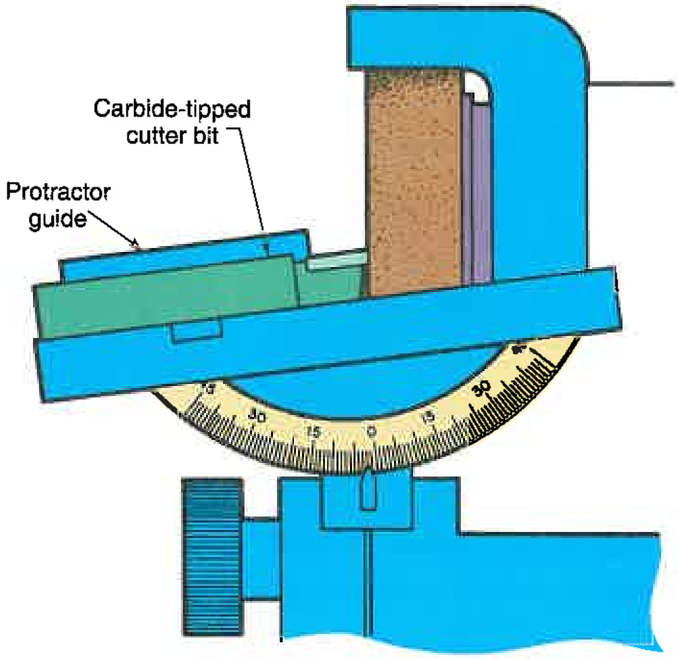


Figure 11-13. *The tool rest can be adjusted to any desired angle using a table protractor and protractor guide.*

Use the entire face of the wheel. Keep the tool in continuous motion to minimize wheel wear. Dress and retrue the wheel as needed.

11.7 PORTABLE HAND GRINDERS

Many grinding jobs, from light deburring to die-polishing operations, are done with small portable hand grinders. ***Flexible shaft grinders*** and ***precision microgrinders***, **Figure 11-14,** are used to perform a variety of toolroom and production jobs. They can be powered by electricity or air.

A ***reciprocating hand grinder,*** **Figure 11-15,** is used to finish dies. Using various attachments, this tool can polish dies to a mirror finish, **Figure 11-16.**

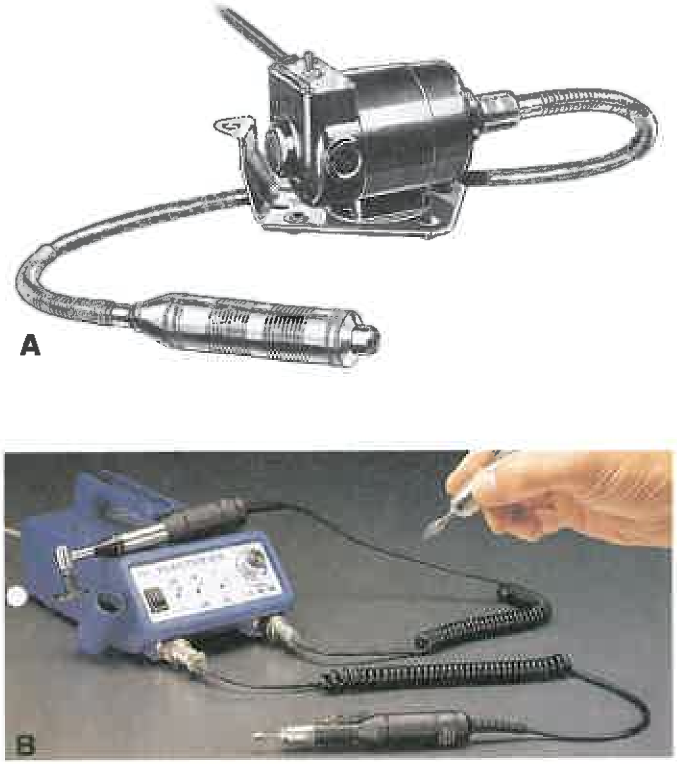


Figure 11-14. *Portable hand grinders. A—A flexible shaft hand grinding unit is helpful for working in recessed areas on parts. B—A precision electric microgrinder. (Dumore Co. and NSK America)*

Figure 11-15. *A pneumatic reciprocating finishing grinder. (NSK America)*

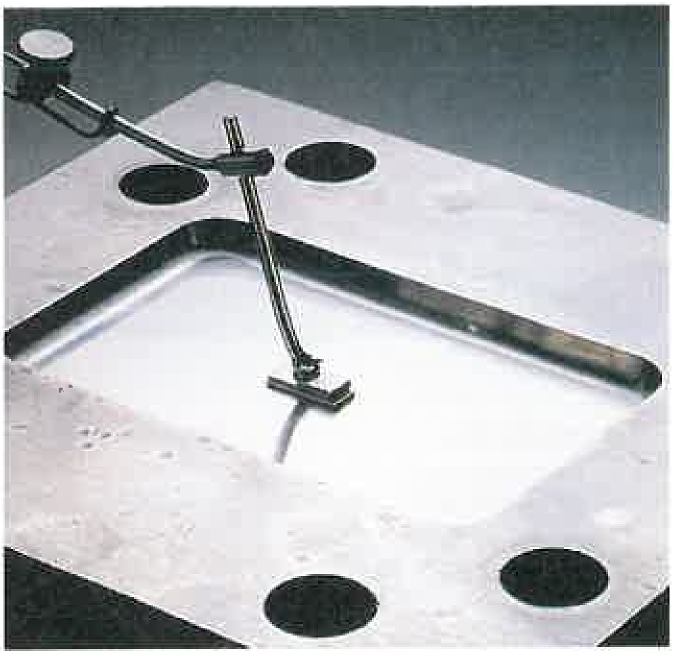


Figure 11-16. *A reciprocating finishing grinder being used to polish a die casting mold. The abrasive stone can be replaced with files and hones of various shapes. (NSK America)*

TEST YOUR KNOWLEDGE

Please do not write in this text. Write your answers on a separate sheet of paper.

1. Describe the grinding operation.
2. How do abrasive belt grinders differ from abrasive wheel grinders?
3. Bench and pedestal grinders are used to do _____ grinding.
4. The grinding technique referred to in the preceding statement is so named because:
    - a. It can only do external work.
    - b. Work is too hard to be machined by other methods.
    - c. Work is manipulated with fingers until desired shape is obtained.
    - d. All of the above.
    - e. None of the above.
5. Name the two types of pedestal grinders. How do they differ?

6. The tool rest should be about _____ inches or _____ mm away from the grinding wheel or belt for safety. This prevents the possibility of work being _____ between the tool _____ and _____.
7. How can grinding wheel soundness be checked?
8. Since a grinding wheel cannot be checked each time the grinder is used, it is recommended that the operator:
   a. Not use the grinder.
   b. Check with the instructor whether the wheel is sound.
   c. Stand to one side of the grinder when using the machine.
   d. All of the above.
   e. None of the above.
9. Work will _____ if it is forced against the wheel with too much pressure.
10. Carbide-tipped tools are usually sharpened on a _____ grinder.
11. The face of the wheel on a wet-type grinder is _____ slightly. Why is this done?
12. Never mount a grinding wheel on a grinder without _____.
13. List four safety precautions to be observed when operating a grinder.

Chapter 12

Sawing and Cutoff Machines

LEARNING OBJECTIVES

After studying this chapter, you will be able to:

- ○ Identify the various types of sawing and cutoff machines.
- ○ Select the correct machine for the job to be done.
- ○ Mount a blade and prepare the machine for use.
- ○ Position the work for the most efficient cutting.
- ○ Safely operate sawing and cutoff machines.

IMPORTANT TERMS

all-hard blade
cold circular saw
dry abrasive cutting
flexible-back blades
friction saw
gravity feed
horizontal band saw
raker set
three-tooth rule
wet abrasive cutting

12.1 METAL-CUTTING SAWS

The first step in most machining jobs is to cut the stock to required length. This can be done using power saws, **Figure 12-1.**

Figure 12-1. *The first step in most machining jobs is to cut the stock to the desired length. Measure the cutoff length carefully and observe all safety precautions. (DoALL Co.)*

There are three principal types of metal-cutting saws, **Figure 12-2.** Reciprocating power saws use a back-and-forth (reciprocating) cutting action. The cutting is done on the backstroke. The blade is similar to that found on a hand hacksaw, only larger and heavier. Band-type power saws have a continuous blade that moves in one direction. Circular-type power saws have a round, flat blade that rotates into the work. A toothed blade, friction blade, or abrasive blade may be used, depending on the material and the operation.

12.2 RECIPROCATING POWER HACKSAW

A ***reciprocating power hacksaw,*** **Figure 12-3,** uses a blade that moves back and forth across the work. The blade cuts on the backstroke. There are several types of feeds available.

Positive feed produces an exact depth of cut on each stroke. The pressure on the blade varies with the number of teeth in contact with the work.

Definite pressure feed yields a pressure on the blade that is uniform regardless of the number of teeth in contact with the work. The depth of the cut varies with the number of teeth contacting the work. This condition prevails with ***gravity feed.***

Feed can be adjusted to meet varying conditions. For best performance, the blade and feed must be selected to permit high-speed cutting and heavy feed pressure with minimum blade bending and breakage.

Standard reciprocating metal cutting saws are available in sizes from 6″ × 6″ (150 mm × 150 mm) to 24″ × 24″ (900 mm × 900 mm). The saws can be fitted with many accessories, including quick-acting vises, power stock feed, power clamping of work, and automatic cycling of the cutting operation. The latter moves the work out the required distance, clamps it, and makes the cut automatically. The cycle is repeated upon completion of the cut.

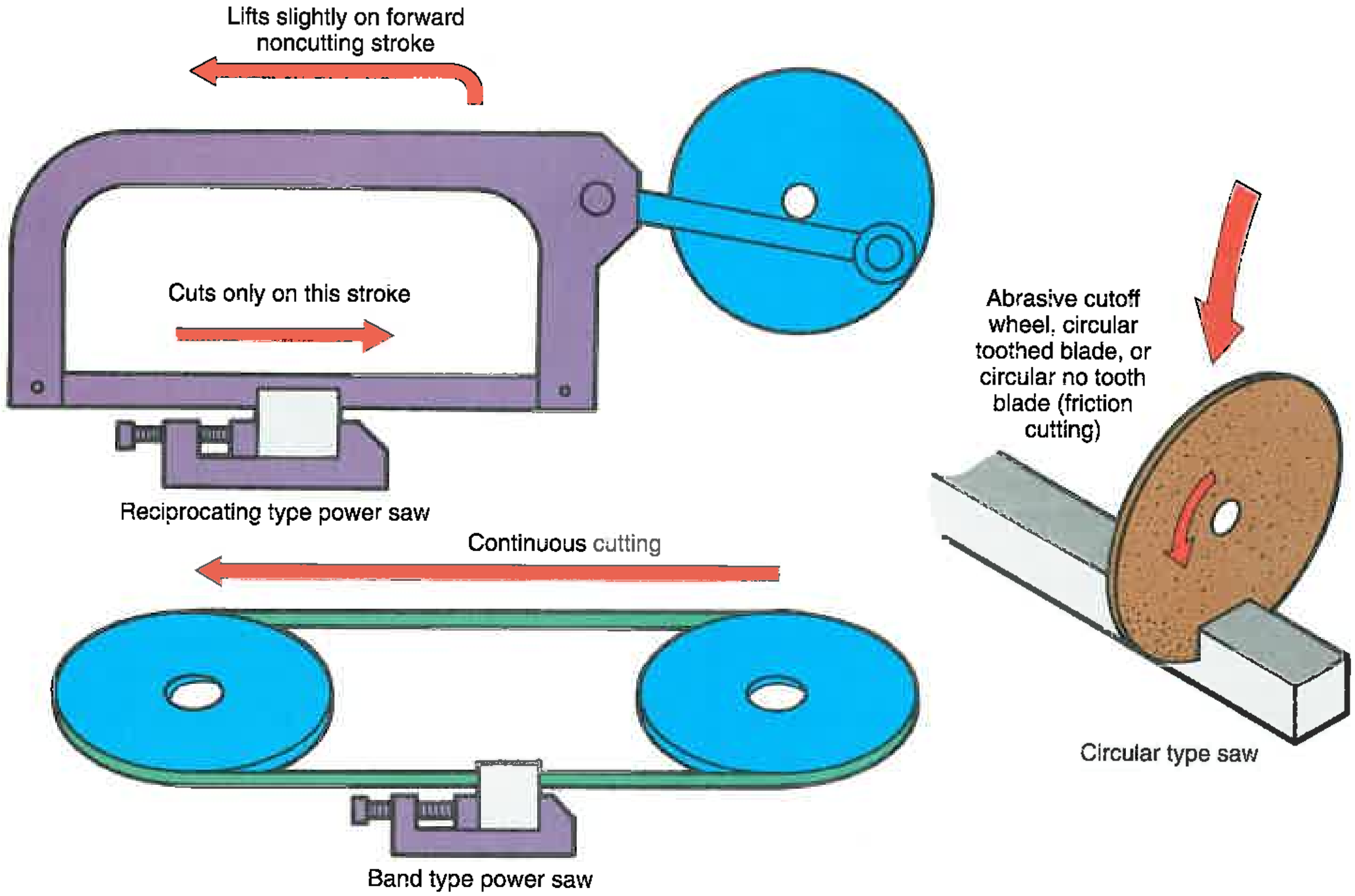


Figure 12-2. *The three principal types of cutoff saws.*

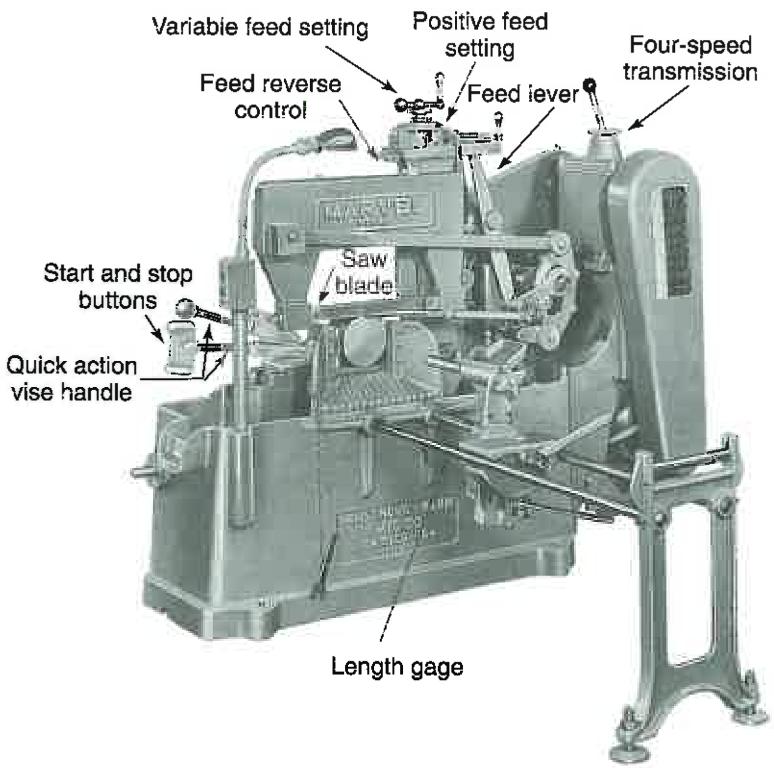


Figure 12-3. *An industrial reciprocating power hacksaw. (Armstrong-Blum Mfg. Co.)*

High-speed cutting requires use of a coolant. Coolant reduces friction, increases blade life, and prevents chip-clogged teeth. Cast iron and some brass alloys, unlike most materials, do not require coolant.

A swivel vise permits angular cuts to be made quickly. See **Figure 12-4.**

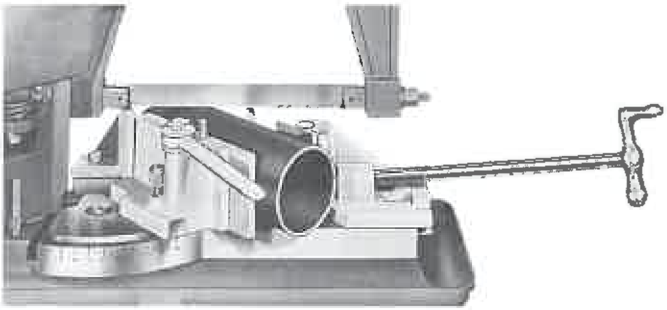


Figure 12-4. *A swivel vise permits angular cuts.*

12.2.1 Selecting a Power Hacksaw Blade

Proper blade selection is important. Use the ***three-tooth rule***—at least three teeth must be in contact with the work. Large sections and soft materials require a coarse-tooth blade. Small or thin work and hard materials require a fine-tooth blade.

For best cutting action, apply heavy feed pressure on hard materials and large work. Use light feed pressure on soft materials and work with small cross sections, **Figure 12-5.**

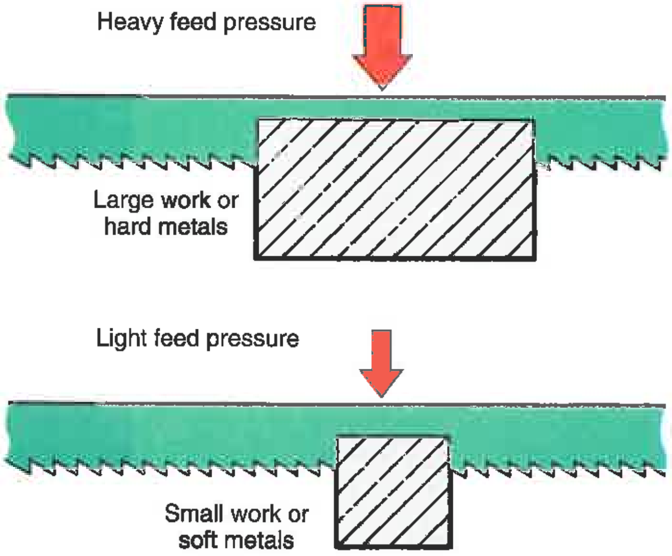


Figure 12-5. *Apply heavy feed pressure on hard metals and large work. Use light pressure on soft metals and work with small cross sections.*

Blades are made in two principal types: flexible-back and all-hard. The choice depends upon use.

Flexible-back blades should be used where safety requirements demand a shatterproof blade. These blades should also be used for cutting odd-shaped work if there is a possibility of the work coming loose in the vise.

For a majority of cutting jobs, the ***all-hard blade*** is best for straight, accurate cutting under a variety of conditions.

When starting a cut with an all-hard blade, be sure the blade does not drop on the work when cutting starts. If it falls, the blade could shatter and flying pieces cause injuries.

Blades are also made from tungsten and molybdenum steels, and with tungsten carbide teeth on steel alloy backs. The following "rule-of-thumb" can be followed for selecting the correct blade:

- Use a 4-tooth blade for cutting large sections or readily machined metals.
- Use a 6-tooth blade for cutting harder alloys and miscellaneous cutting.
- Use 10- and 14-tooth blades primarily on light duty machines where work is limited to small sections requiring moderate or light feed pressure.

12.2.2 Mounting a Power Hacksaw Blade

The blade must be mounted to cut on the power (back) stroke. The blade must also lie perfectly flat against the mounting plates, **Figure 12-6.** If long life and accurate cuts are to be achieved, the blade must be properly tensioned.

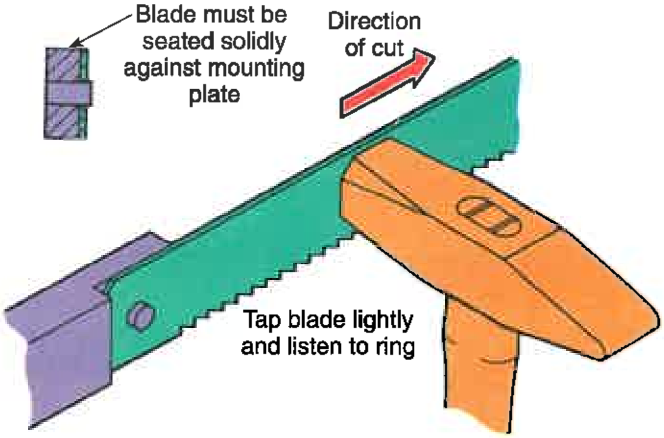


Figure 12-6. *The blade must be adjusted to cut on the back stroke. Make sure it is perfectly flat against the mounting plates before tensioning. Tighten the blade until a low musical ring is heard when the blade is tapped with a small hammer. Since blades have a tendency to stretch slightly after making a few cuts, tension should be checked and, if necessary, adjusted.*

Many techniques have been developed for properly mounting and tensioning blades. Use a torque wrench and consult the manufacturer's literature. If the information (proper torque for a given blade on a given machine) is not available, the following methods can be used:

- Tighten the blade until a low musical ring is heard when the blade is tapped lightly. A high-pitched tone indicates that the blade is too tight. A dull thud means the blade is too loose.
- The shape of the blade pin hole can serve as an indicator of whether the blade is tensioned properly. When proper tension is achieved, the pin holes will become slightly elongated, **Figure 12-7**.

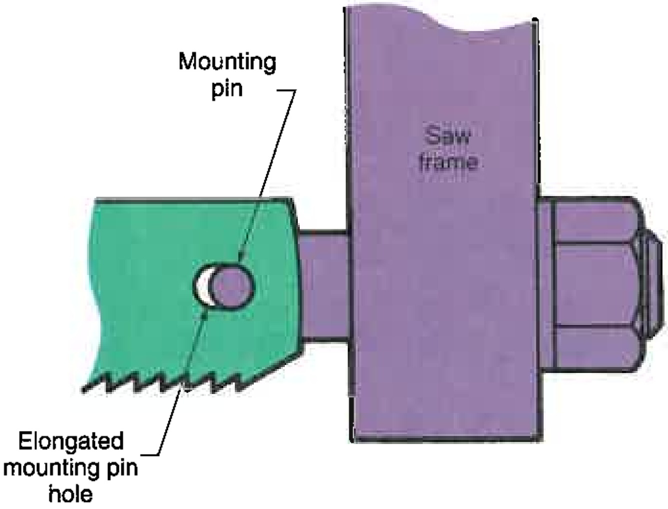


Figure 12-7. *Pin holes on a properly tensioned blade will be slightly elongated, rather than round.*

The blade will become more firmly seated after the first few cuts and will stretch slightly. The blade will require ***retensioning*** (retightening) before further cutting can be done.

12.2.3 Cutting with a Power Hacksaw

Measure off the distance to be cut. Allow ample material for facing if the work order does not specify the length of cut. Mark the stock and mount the work firmly on the machine, **Figure 12-8.**

If several sections are to be cut, use a ***stop gage,*** **Figure 12-9.** Apply an ample supply of coolant if the machine has a built-in coolant system.

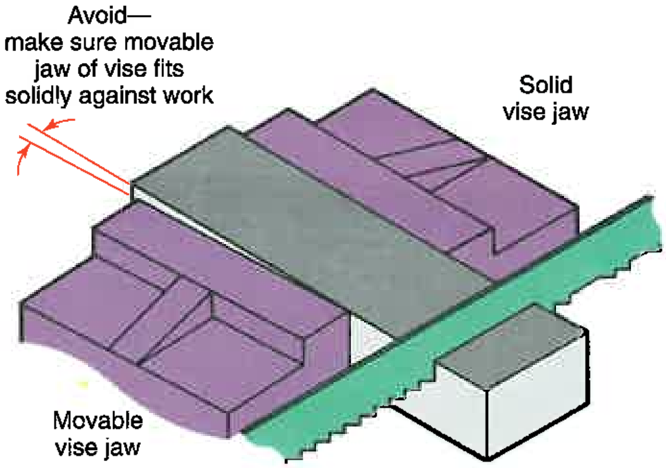


Figure 12-8. *If the work is not clamped solidly, it will twist and the blade will bind and be ruined in the first few seconds of use.*

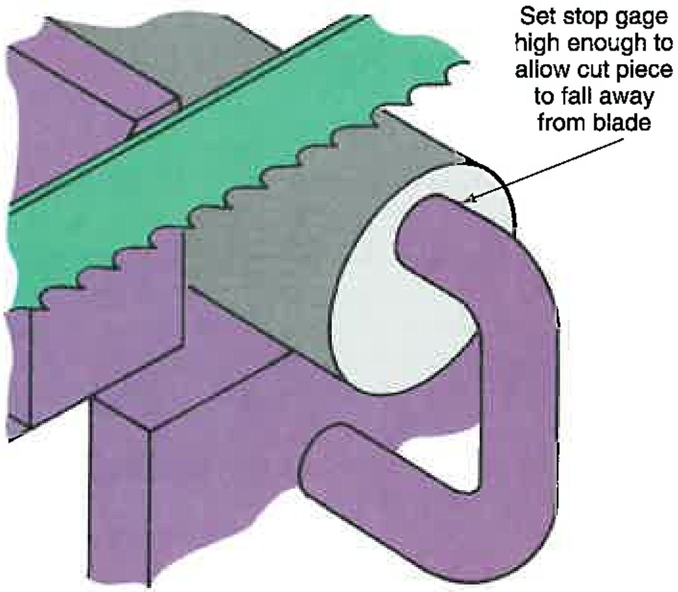


Figure 12-9. *A stop gage is used when several pieces of the same length must be cut. Set it high to permit the work to fall free when completely cut.*

12.3 POWER BAND SAW

The ***horizontal band saw,*** **Figure 12-10,** is frequently referred to as the cutoff machine. It offers three advantages over the reciprocating hacksaw:

- *Greater precision*—The blade on a band saw can be guided more accurately than the blade on the reciprocating power saw. It is common practice to cut directly "on the line" when band sawing, because finer blades can be used.

Figure 12-10. *Band sawing machines. A—Typical band cutoff saw with built-in coolant system, manually controlled blade tension, and automatic end-of-cut shutoff. B—Band cutoff machines are also available with tilting frame. The frame is capable of tilting up to 45°. (W. F. Wells)*

- *Faster speed*—The long, continuous blade moves in only one direction, so cutting is also continuous. The blade can run at much higher speeds because it rapidly dissipates the cutting heat.
- *Less waste*—The small cross section of the band saw blade makes smaller and fewer chips than the thicker reciprocating blade, **Figure 12-11**.

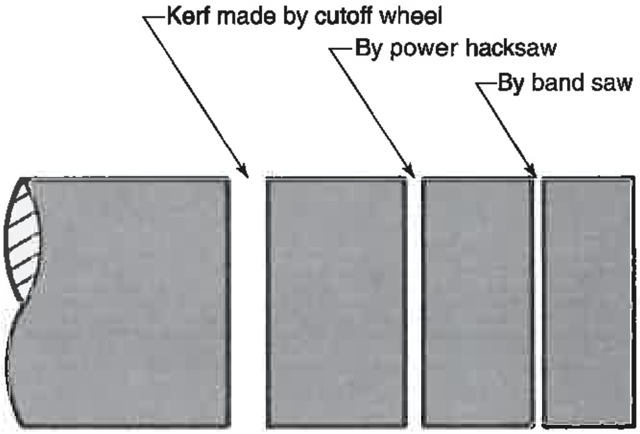


Figure 12-11. *Differences in the amount of metal converted to chips (waste) by each cutoff machine.*

12.3.1 Selecting a Band Saw Blade

Band saw blades are made with raker teeth or wavy teeth, **Figure 12-12.** Most manufacturers also make variations of these sets. The ***raker set*** is preferred for general use.

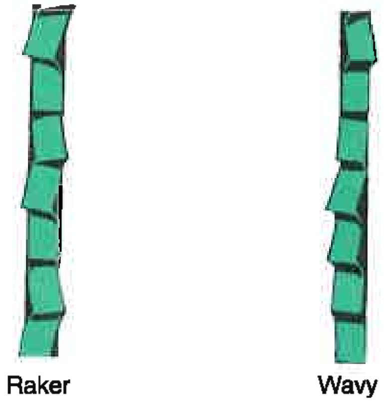


Figure 12-12. *Saw blades commonly have raker or wavy teeth. Raker teeth are preferred for general use, cutting large solid sections, and cutting thick plate.*

Tooth pattern determines the efficiency of a blade in various materials. The ***standard tooth*** blade pattern is best suited for cutting most ferrous metals. A ***skip tooth*** blade pattern is preferred for cutting aluminum, magnesium, copper, and soft brasses. The ***hook tooth*** blade pattern also is recommended for most nonferrous metallic materials. See **Figure 12-13.**

For best results, consult the blade manufacturer's chart or manual for the proper blade characteristics (set, pattern, and number of teeth per inch) for the particular material being cut.

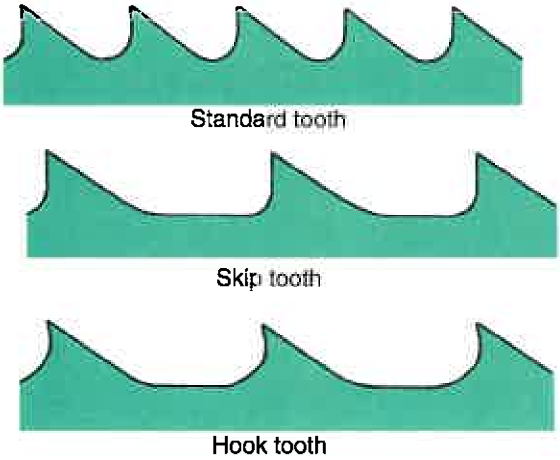


Figure 12-13. *Standard tooth blades, with rounded gullets, are usually best for most ferrous metals, hard bronzes, and hard brasses. Skip tooth blades provide for more chip clearance without weakening the blade body. They are recommended for cutting aluminum, magnesium, copper, and soft brasses. Hook tooth blades offer two advantages over skip tooth blades—easier feeding and less "gumming up."*

12.3.2 Installing a Band Saw Blade

If the saw is to work at top efficiency, the blade must be installed carefully. Wear heavy leather gloves to protect your hands when installing a band saw blade.

Blade guides should be adjusted to provide adequate support, **Figure 12-14.** Proper blade support is required to cut true and square with the holding device.

Follow the manufacturer's instructions for adjusting blade tension. Improper blade tension ruins blades and can cause premature failure of bearings in the drive and idler wheels.

Cutting problems encountered with the band saw are similar to those of the reciprocating hack saw. Most problems are caused by poor machine condition. They can be kept to a minimum if a maintenance program is followed on a regular basis. This typically includes checking wheel alignment, guide alignment, feed pressure, and hydraulic systems.

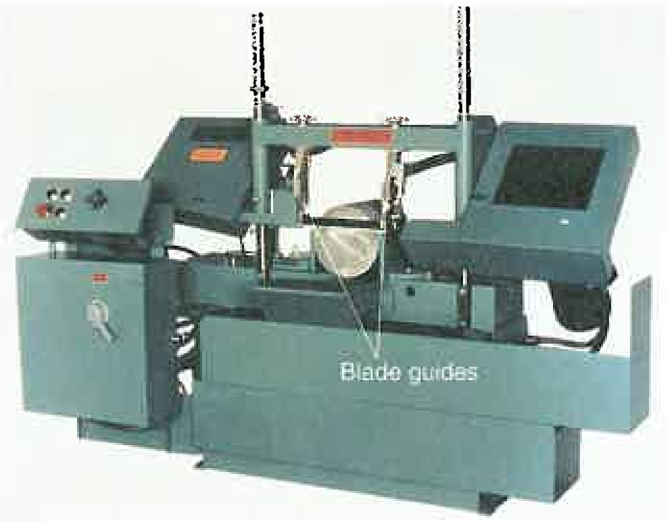


Figure 12-14. *Adjust blade guides to provide adequate blade support; otherwise, blade will not cut true. (W.F. Wells)*

12.4 USING RECIPROCATING AND BAND SAWS

Most sawing problems can be prevented by careful planning and observing a few rules. These apply to both reciprocating and band saws.

12.4.1 Blades Breaking

Blades are normally broken when they are dropped on the work. A loose blade or excessive feed can cause the blade to fracture. Loose work can also cause blade damage, as will making a cut on a corner or sharp edge where the three-tooth rule is not observed. Broken blades can normally be avoided with proper machine setup.

12.4.2 Crooked Cutting

This problem is usually the result of a worn blade. Remember to reverse the work after replacing a blade, and start a new cut on the opposite side. See **Figure 12-15.** A loose blade or a blade rubbing on a clamping fixture will cause the same problem. It can also be caused by excessive blade pressure on the work or by worn saw guides.

12.4.3 Blade Pin Holes Breaking Out

This reciprocating blade problem can be caused by dirty mounting plates or too much tension on the blade. Worn mounting plates can cause a blade to twist and strain in such a way that the pin hole will break out.

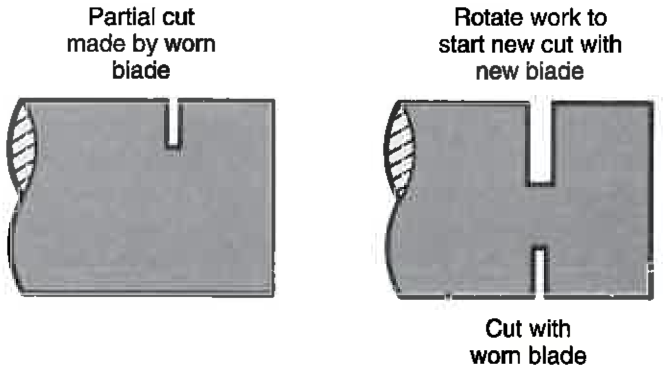


Figure 12-15. *Never attempt to start a new blade in a cut made by a worn blade. Reverse the work and start another cut on the opposite side. Cut through to the old cut.*

12.4.4 Premature Blade Tooth Wear

When this problem occurs, the teeth become rounded and dull quickly. Insufficient feed pressure (indicated by light, powdery chips) is one of the major causes of this condition. Excessive pressure (indicated by burned chips) causes the same problem.

Insufficient pressure can be corrected by increasing cutting pressure until a full curled chip is produced. If too much pressure is the culprit, reduce feed pressure until a full curled chip is formed.

Lack of coolant or a poorly adjusted machine can also cause rapid wear. Correct by following the manufacturer's recommendations.

12.4.5 Teeth Strip Off

This failure results when the teeth snap off the blade. Starting a cut on a sharp corner is a major cause of this problem. A machine setup with a flat starting surface will greatly reduce tooth stripping. Be sure the work is clamped securely; loose work can also cause the teeth to strip, **Figure 12-16.**

Check the manufacturer's chart to determine the proper blade for the job to be done. A blade with teeth too fine will clog (load) and jam, causing the teeth to shear off. A blade that is too coarse (less than three teeth cutting) will cause the same problem. Make sure the blade is properly mounted and cutting on the power stroke.

12.5 CIRCULAR METAL-CUTTING SAWS

Metal-cutting circular saws are found in many areas of metalworking. Primarily production machines, these saws are divided into three classifications:

- Abrasive cutoff saw.
- Cold circular saw.
- Friction saw.

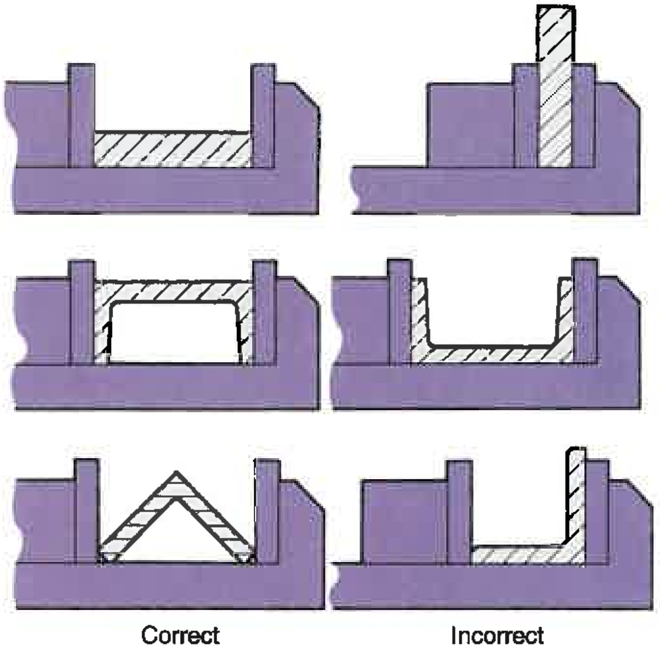


Figure 12-16. *Recommended ways to hold sharp-cornered work for cutting. A carefully planned setup will ensure that at least three teeth will be cutting, greatly extending blade life.*

An ***abrasive cutoff saw,*** **Figure 12-17,** cuts material using a rapidly revolving, thin abrasive wheel. Most materials—glass, ceramics, and metals—can be cut to close tolerances. Hardened steel does not require annealing to be cut. Special heat-resistant abrasive wheels are available for high-speed cutoff of hot stock.

Abrasive cutting falls into two classifications, dry and wet. ***Wet abrasive cutting,*** while not quite as rapid as dry cutting in some applications, produces a finer surface finish and permits cutting to close tolerances. The cuts are burn-free and have few or no burrs. ***Dry abrasive cutting*** does not use a coolant and is used for rapid, less-critical cutting.

A ***cold circular saw,*** **Figure 12-18,** makes use of a circular, toothed blade capable of producing very accurate cuts. Large cold circular saws can sever round metal stock up to 27" (675 mm) in diameter.

A ***friction saw*** blade may or may not have teeth. The saw operates at very high speeds (20,000 surface feet per minute or 6000 m per minute) and actually melts its way through the metal.

Teeth are used primarily to carry oxygen to the cutting area. These machines find many applications in steel mills to cut red-hot ***billets*** (sections of semifinished steel).

12.6 POWER SAW SAFETY

- Never attempt to operate a sawing machine while your senses are impaired by medication or other substances.

Figure 12-17. *An industrial abrasive cutoff saw. (Everett Industries, Inc.)*

- Get help when you are lifting and cutting heavy material.
- Clean oil, grease, and coolant from the floor around the work area.
- Burrs on cut pieces are sharp. Use special care when handling pieces with burrs.
- Follow the manufacturer's instructions for tensioning a blade. Too much tension can shatter the blade.
- Handle band saw blades with extreme care. They are long and springy and can uncoil suddenly.
- Be sure the work is mounted solidly before starting a cut.
- Be sure all guards are in place before using the saw.
- Always wear a dust mask and full face shield when cutting stock with a dry-type abrasive cutoff saw.
- Avoid standing directly in line with the blade when operating a circular cutoff saw.
- Use a brush to clean chips from the machine. Do not use your hands. Wait for the machine to come to a complete stop before cleaning.

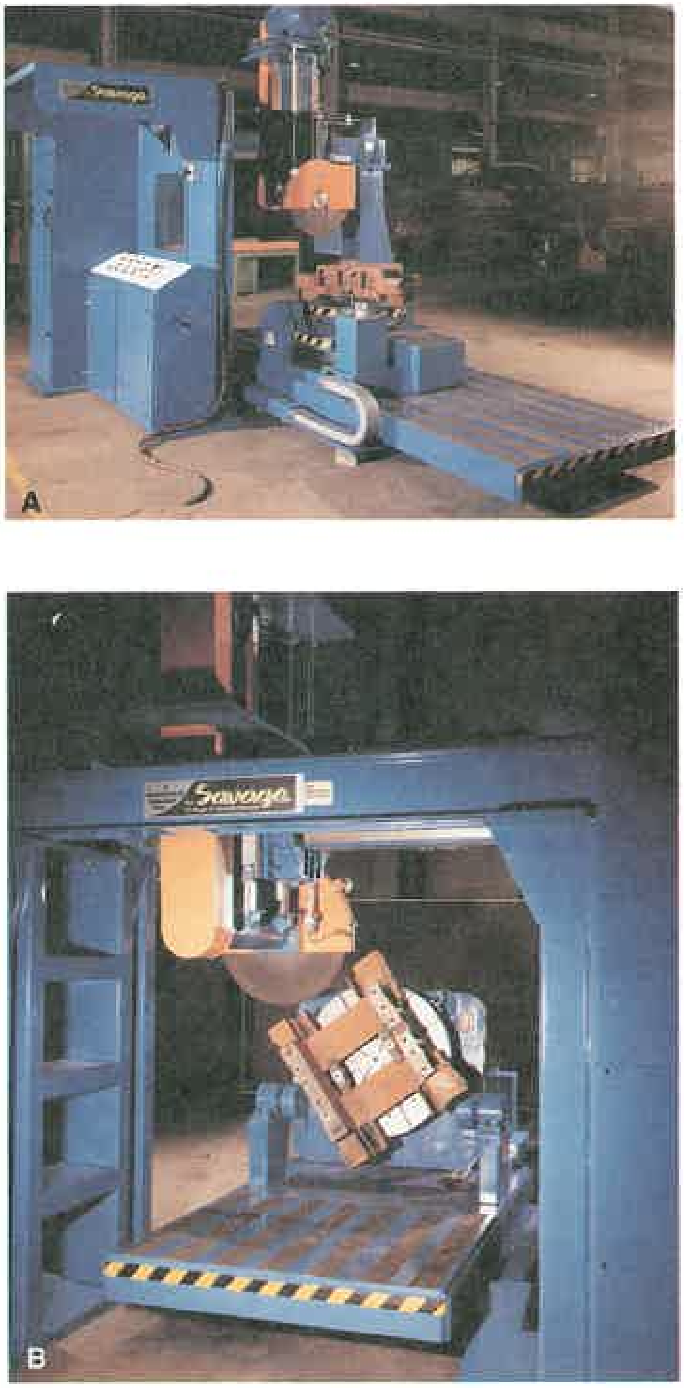


Figure 12-18. *Cold circular saws. A—This automated cold circular saw can accept a piece up to 2500 pounds. A laser guide light marks the position of the cut. B—This machine can be fitted with carbide-tipped blades or abrasive disks. (W.J. Savage Co.)*

- Keep your hands out of the way of moving parts.
- Stop the machine before making adjustments.
- Have all cuts, bruises, and scratches, even minor ones, treated immediately.

TEST YOUR KNOWLEDGE

Please do not write in the text. Write your answers on a separate sheet of paper.

1. List the three basic types of metal-cutting saws.
2. The _____ type saw has a back-and-forth cutting action. However, it only cuts on the _____ stroke.
3. What is the "three-tooth rule" for sawing?
4. When using a power sawing machine, with which materials should you *not* use coolant?
5. Hacksaw blades are manufactured in two principal types. Name them.
6. The following "rule-of-thumb" should be followed for selecting the correct blade:
   a. _____ teeth per inch for cutting large sections or readily machined materials.
   b. _____ teeth per inch for cutting harder alloys and miscellaneous cutting.
   c. _____ teeth per inch for cutting on the majority of light-duty machines, where work is limited to small sections and moderate to light feed pressures.
7. List three methods used to put proper tension on a power hacksaw blade.
8. When is a stop gage used?
9. What three advantages does the continuous band sawing machine offer over other types of power saws?
10. Band saw blades are made with two types of teeth. Name them.
11. The tooth pattern of a blade determines the efficiency of a blade in various materials.
    a. The _____ tooth is best suited for cutting most ferrous metals.
    b. The _____ tooth pattern is preferred for cutting aluminum, magnesium, copper, and soft brass.
    c. The _____ tooth is also recommended for most nonferrous metallic materials.
12. List the three types of circular metal-cutting saws.
13. List five safety precautions to be observed when operating a power saw.

Grooving and parting are operations frequently performed on the lathe. This grooving tool uses a replaceable insert that clamps into a special thin holder for performing deep face grooving. The insert can be reversed, allowing a quick change to a new cutting edge when one becomes dull or damaged. Inserts for grooves ranging from 2 mm to 6 mm in width are available. (Iscar Metals, Inc.)

Chapter 13

The Lathe

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.
- Sharpen lathe cutting tools.

IMPORTANT TERMS

compound rest
cross-slide
depth of cut
facing
headstock
indexable insert cutting tools
plain turning
single-point cutting tool
tailstock
tool post

The ***lathe*** operates on the principle of the work being rotated against the edge of a cutting tool, **Figure 13-1**. 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 13-2**.

13.1 LATHE SIZE

Lathe size is determined by the *swing* and the *length of the bed,* **Figure 13-3**. The ***swing*** indicates 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 13-3**.

Figure 13-1. Metal-cutting lathes. A—The basic lathe. All controls are operated manually. Most machinists begin their training on this type of lathe. (Jet Equipment & Tools) B—Metal-cutting lathes are some of the most versatile machine tools made. There are many variations of the basic lathe for differing applications. This one is CNC controlled. (Harrison/REM Sales Inc.)

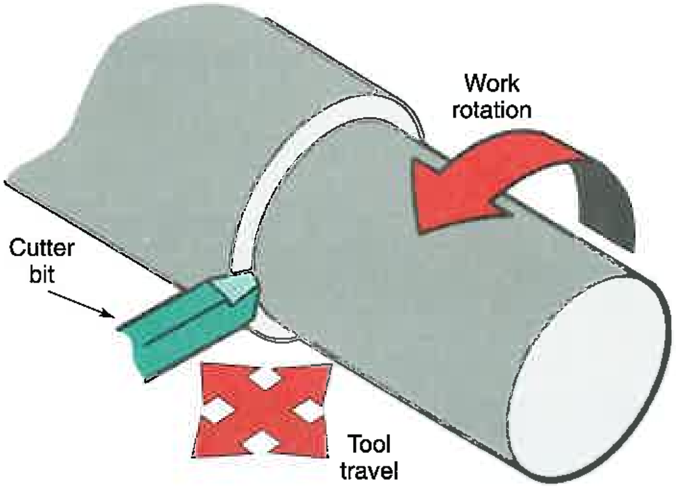


Figure 13-2. Operating principle of the lathe. The cutting tool is fed into the revolving work.

As an example, consider the capacity and clearance of a modern 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)

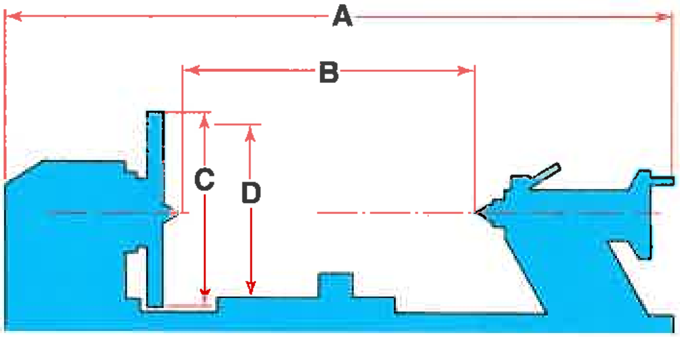


Figure 13-3. Lathe measurements. A—Length of bed. B—Distance between centers. C—Diameter of work that can be turned over the ways. D—Diameter of work that can be turned over the cross-slide.

13.2 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 13-4** can be assigned to one of these three categories:

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

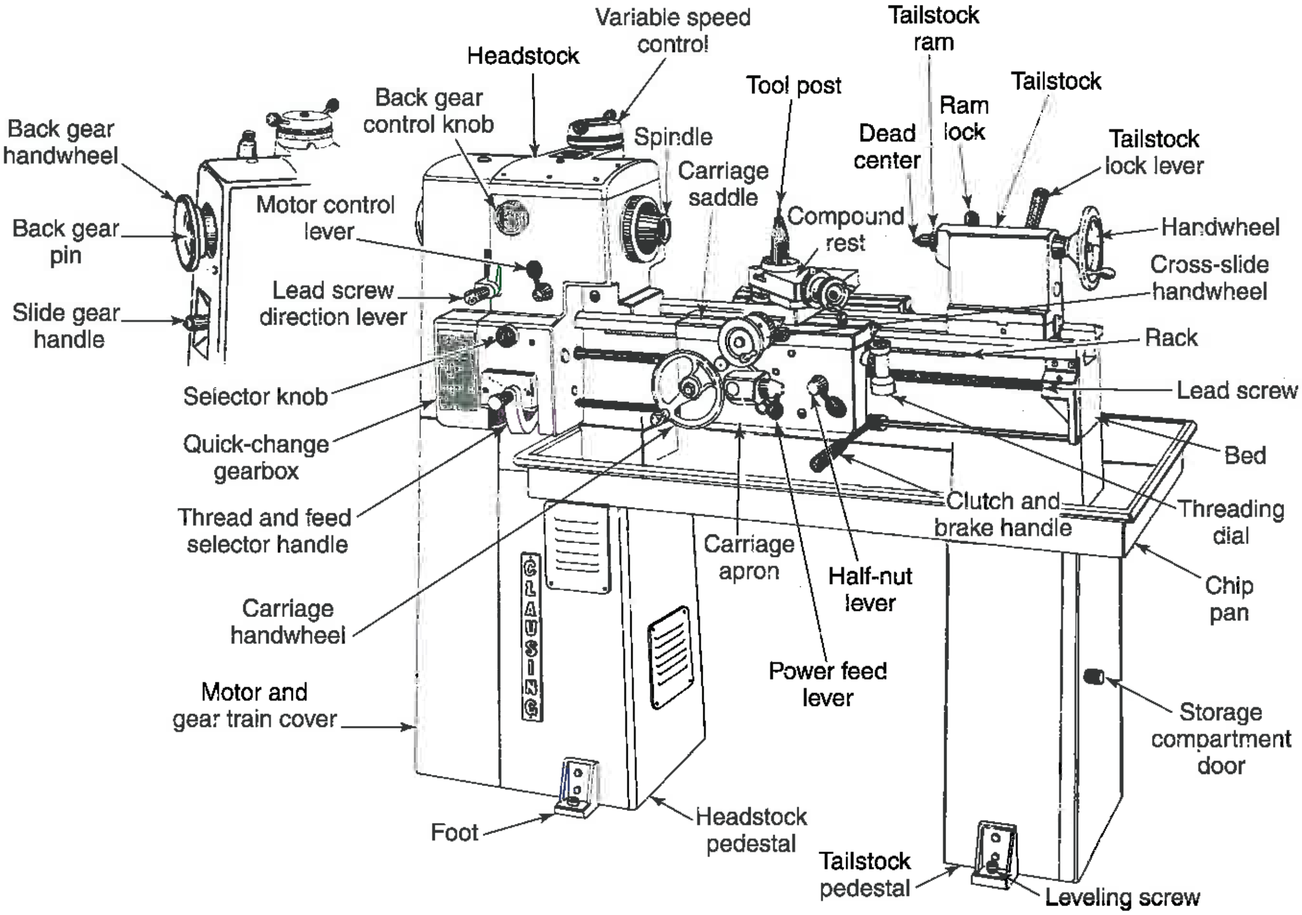


Figure 13-4. The engine lathe and its major parts. (Clausing Industrial, Inc.)

13.2.1 Driving the lathe

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

- Shifting to a different gear ratio, **Figure 13-5.**
- Adjusting a split pulley to another position, **Figure 13-6.**
- 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.*** See **Figure 13-7.** To avoid damaging the lathe's drive system, *do not engage the back gear while the spindle is rotating.*

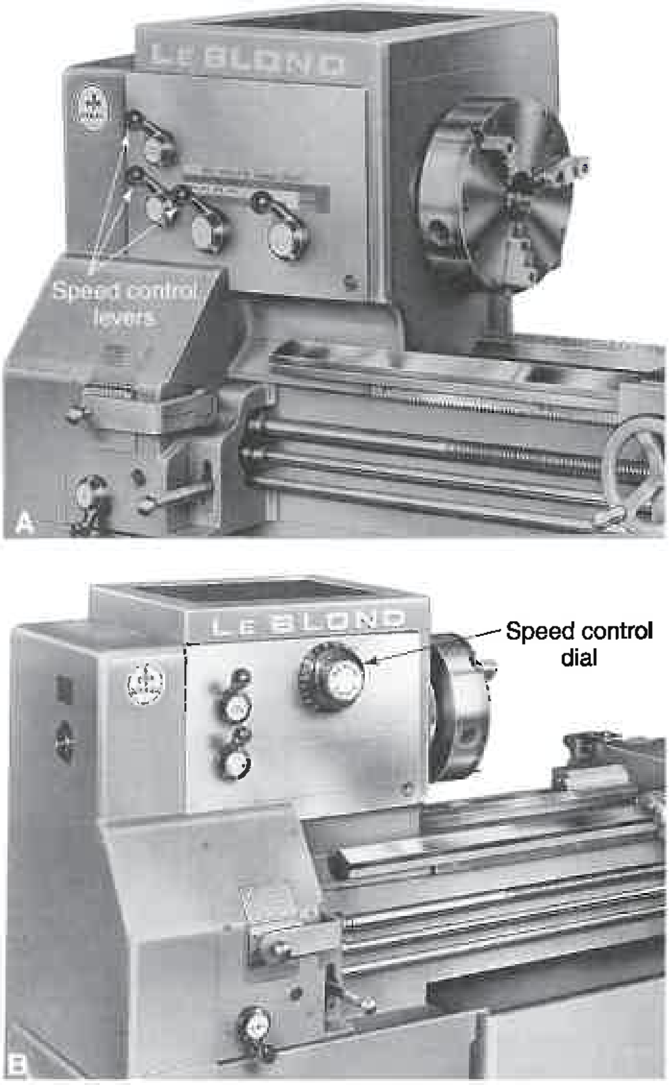


Figure 13-5. Spindle speed control. A—Speed is increased or decreased by shifting to different gear ratios. B—On this machine, spindle speed is controlled by an automatic transmission. Desired speed is dialed in. (LeBlond Makino Machine Tool Co.)

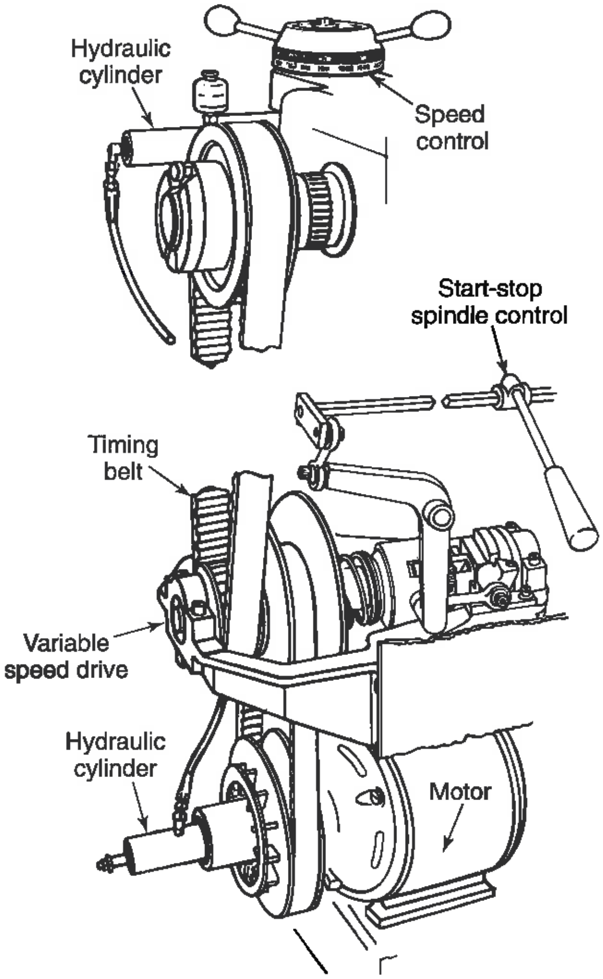


Figure 13-6. This split pulley is hydraulically actuated from the top of the machine by a speed control. A split pulley is used to control spindle speeds on many lathes. (Clausing Industrial, Inc.)

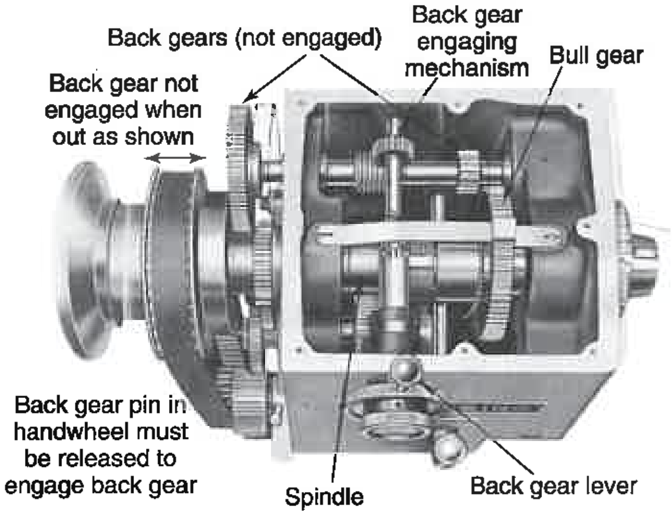


Figure 13-7. The back gear mechanism is clearly visible in this view of a lathe headstock. Direct drive is disengaged before back gear slides into position. (Clausing Industrial, Inc.)

13.2.2 Holding and rotating the work

The ***headstock*** contains the spindle to which the various work-holding attachments are fitted, **Figure 13-8**. 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 13-9**. 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.

Figure 13-8. *The headstock is the driving end of lathe. (Clausing Industrial, Inc.)*

On the front end, a spindle may be threaded externally or fitted with one of two types of tapered spindle noses to receive work-holding attachments. . See **Figure 13-10.** A ***threaded spindle*** nose is seldom used on modern lathes. It permits mounting an attachment by screwing it directly on 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*** has a protruding long taper and key that fits into a corresponding taper and keyway in the back of the work-holding device.

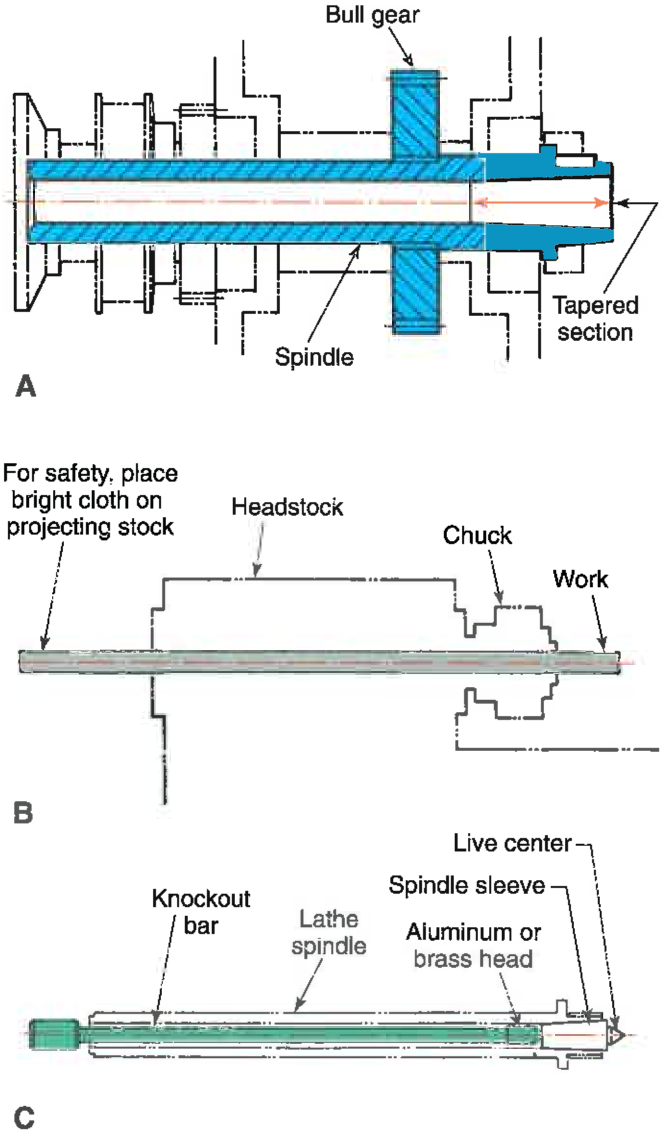


Figure 13-9. *Lathe spindle. A— Hollow construction of the spindle allows long stock to be turned without dangerous overhang. B—To prevent accidents that could cause injury, some sort of flag should be tied to the portion of stock that projects from the rear of the spindle. C—A knockout bar is used to tap tapered shank lathe accessories out of the spindle.*

To mount a work-holding device (a chuck or faceplate), the spindle is rotated until the key is on top. See **Figure 13-10C.** 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, then tightened.

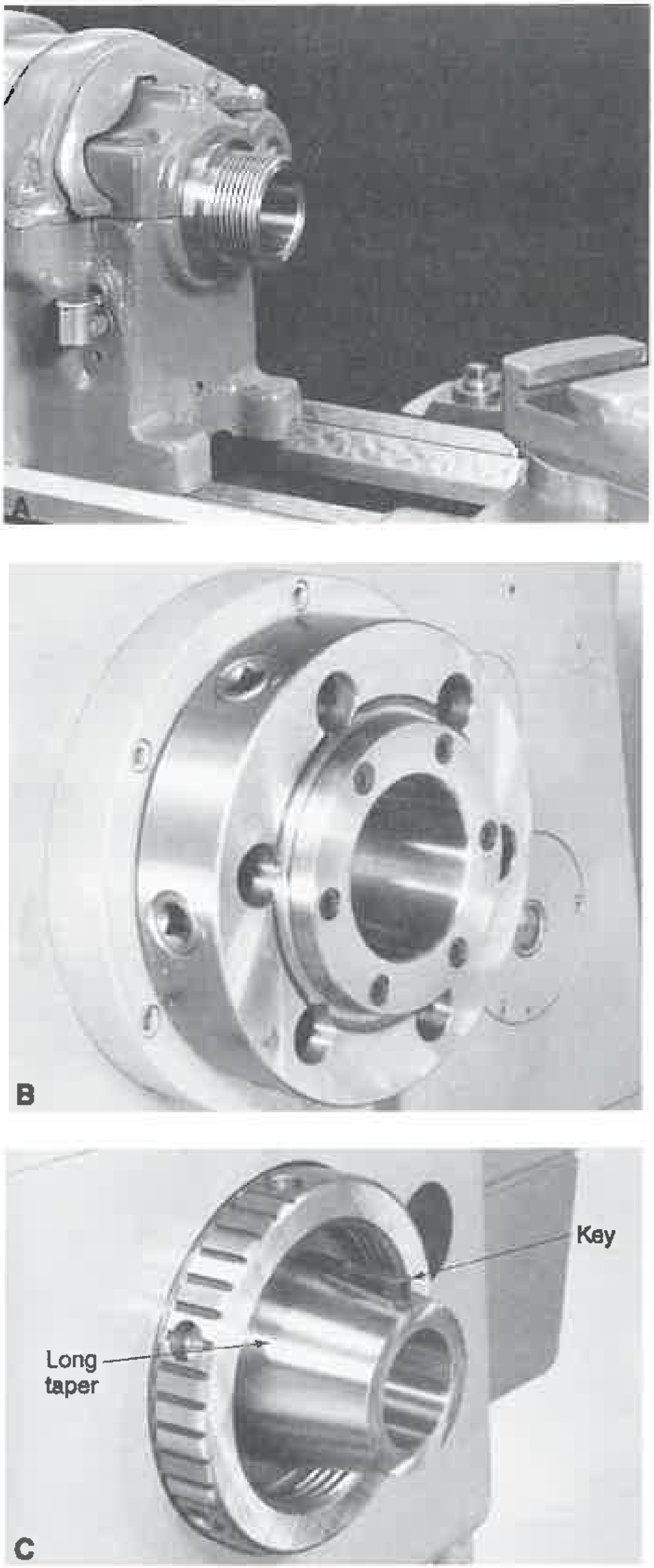


Figure 13-10. *Spindle noses. A—Threaded spindle nose is seldom used today. (South Bend Lathe Corps.) B—A cam-lock spindle nose. (Clausing Industrial, Inc.) C—A long taper key spindle nose. (Clausing Industrial, Inc.)*

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 13-11**. The tailstock can be adjusted along the ways to accommodate different lengths of work.

The tailstock is used to mount the "dead" center, or can be fitted with tools for drilling, reaming, and threading. It can also 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 and can be locked in position by tightening a binding lever.

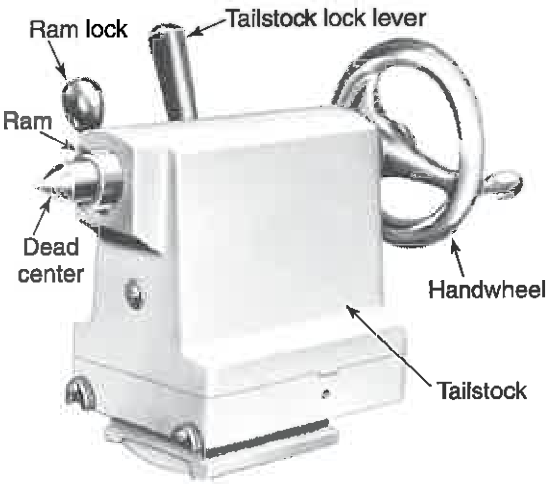


Figure 13-11 *Parts of the tailstock. (Clausing Industrial, Inc.)*

13.2.3 Holding, moving, and guiding the cutting tool

The ***bed,*** **Figure 13-12,** 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, **Figure 13-13.**

The ***carriage*** controls and supports the cutting tool, and is composed of a number of 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.

Power is transmitted to the carriage through the ***feed mechanism,*** which is located at the left (headstock) end of the lathe, **Figure 13-14**. Power is transmitted through a train of gears to the ***quick-change gearbox***. This device, **Figure 13-15,** 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 is controlled in the same manner.

Figure 13-12. *The bed is the foundation of the lathe.*

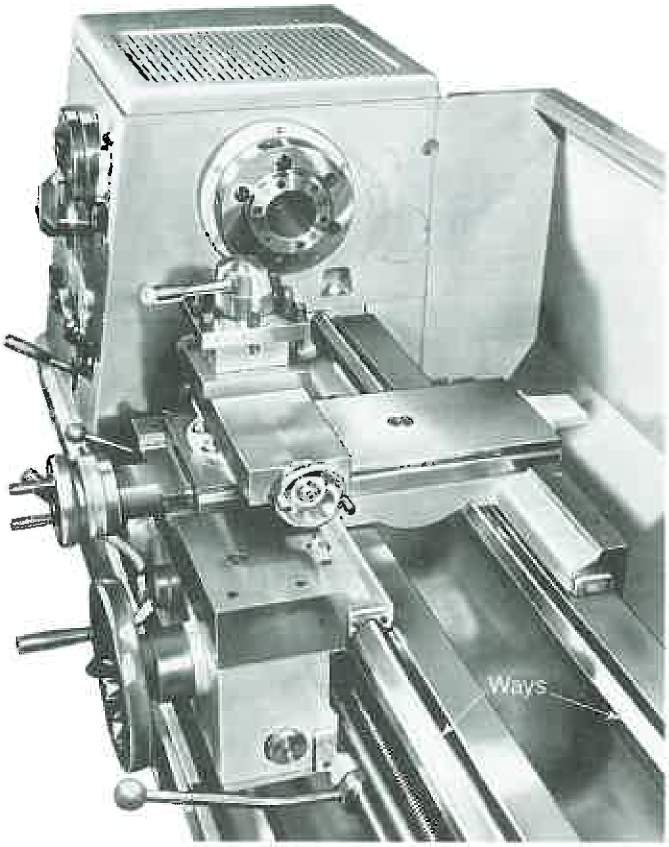


Figure 13-13. *V-shaped ways guide carriage. The cutting tool is mounted on the carriage.*

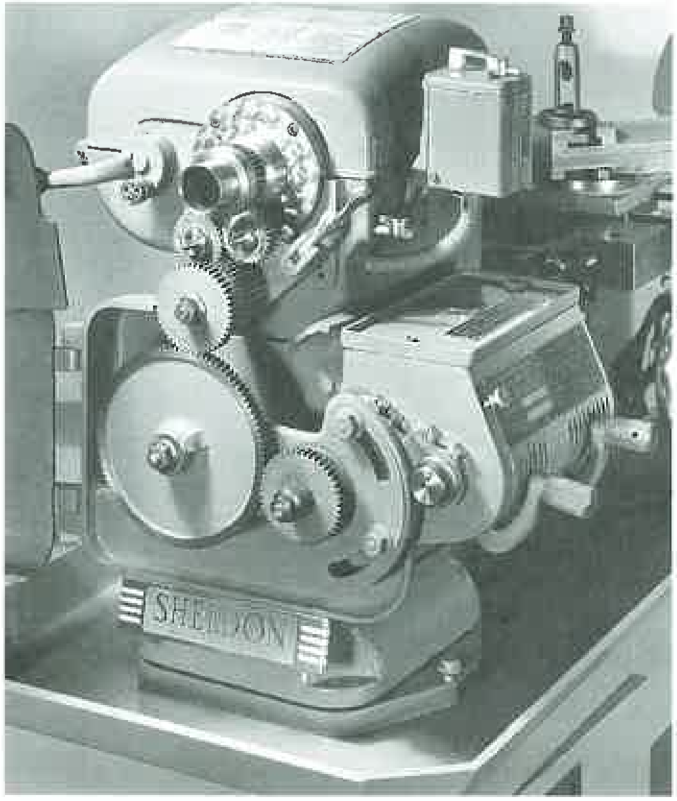


Figure 13-14. *A feed mechanism is used to move the carriage along the ways.*

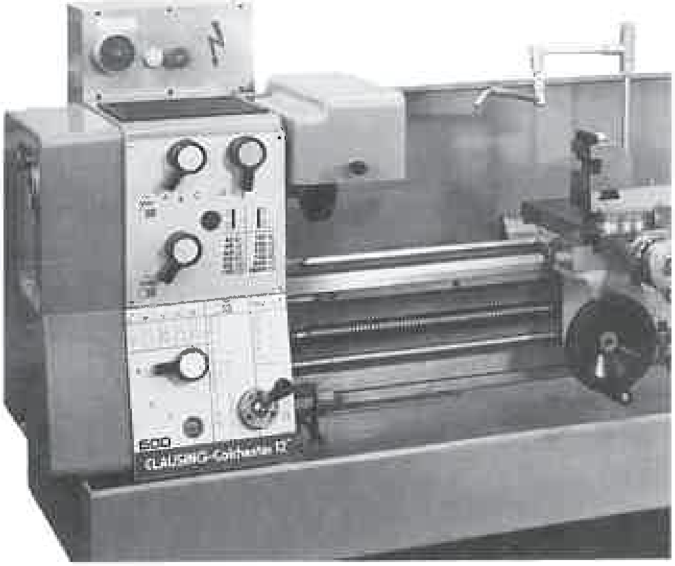


Figure 13-15. A quick-change gearbox for cutting both inch and metric size threads. (Clausing Industrial, Inc.)

An ***index plate*** provides instructions on how to set the lathe shift levers for various threadcutting and feed combinations, **Figure 13-16.** It is located on the face of the gearbox. The ***large numbers*** on the index plate at left indicate the number of threads that can be cut per inch or pitch of metric threads. The ***smaller figures*** indicate the carriage longitudinal movement, in thousandths of an inch or in mm for each spindle revolution.

The ***lead screw*** transmits power to the carriage through a gearing and clutch arrangement in the carriage apron, **Figure 13-17.** ***Feed change levers*** on the apron control the operation of power longitudinal feed and power cross-feed, **Figure 13-18.**

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 power feed and half-nuts simultaneously. The half-nuts are engaged *only* for thread cutting and are *not* used as an "automatic feed" for regular turning.

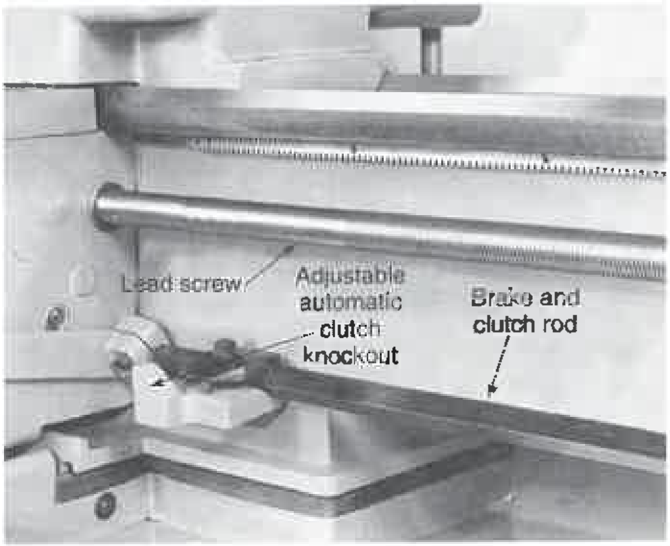


Figure 13-17. The lead screw.

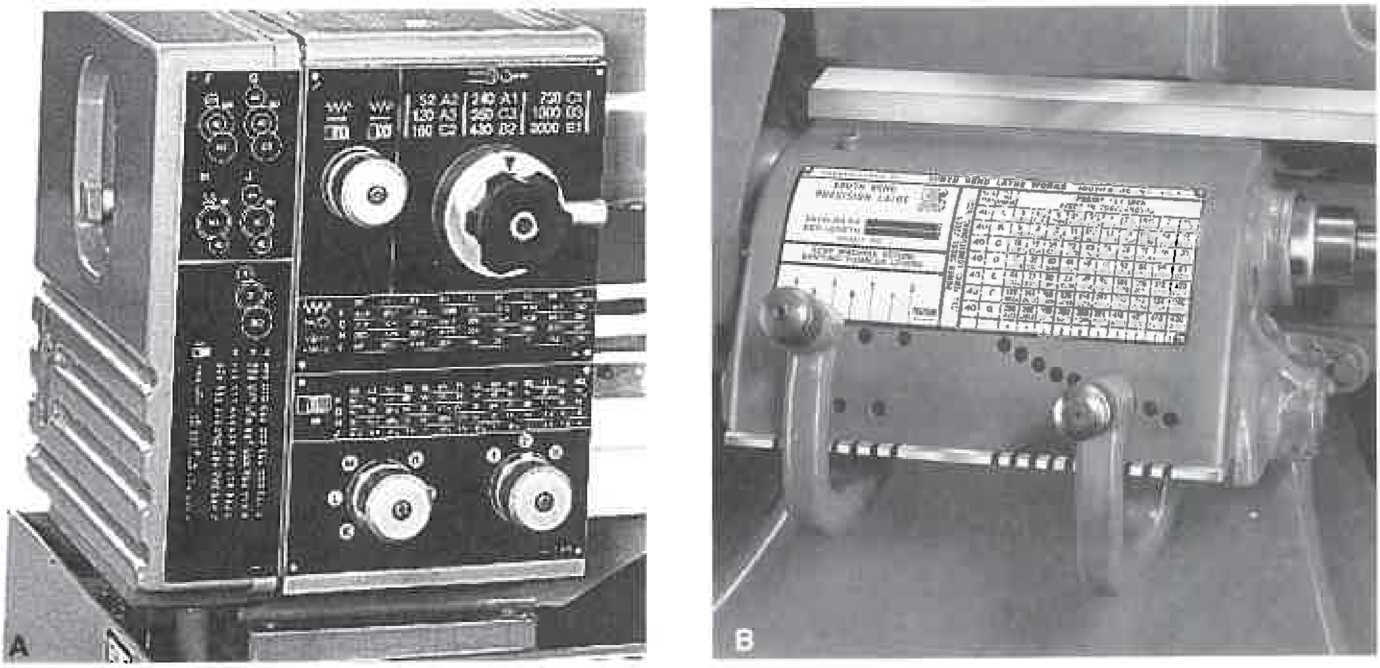


Figure 13-16. Index plates. A—An index plate showing lever positions for inch and metric feeds and threads. (Clausing Industrial, Inc.) B—An index plate for inch feeds and threads. (South Bend Lathe Corp.)

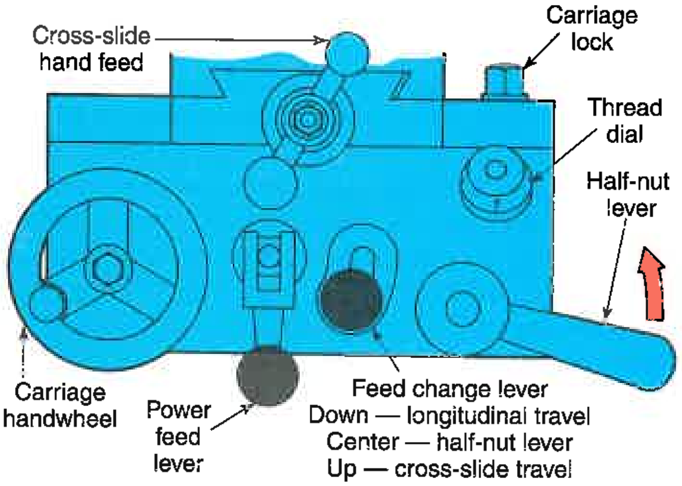


Figure 13-18. *Power feed functions are controlled by the feed change levers on the apron. The half-nut lever is engaged only for thread cutting, sometimes called "thread chasing."*

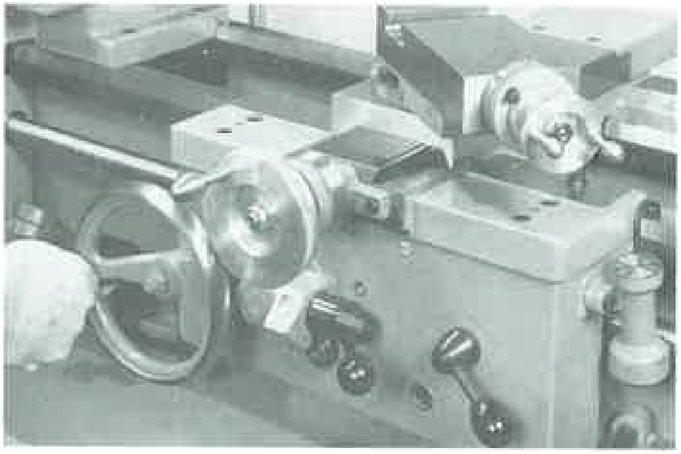


Figure 13-19. *The carriage should be checked for binding by moving it along the ways. Correct any binding before attempting to use lathe. (Clausing Industrial, Inc.)*

13.3 PREPARING 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 nor engaged in back gear (unless you intend to use back gear).
- Move the carriage along the ways, **Figure 13-19**. There should be no binding.
- Check cross-slide movement. If there is too much play, adjust the gibs. See **Figure 13-20**.
- Mount the desired work-holding attachment. Clean the spindle nose with a soft brush. A threaded nose spindle should have a drop of lubricating oil applied before the chuck or faceplate is attached.
- Adjust the drive mechanism for the desired speed and feed.
- If the tailstock is used, check it for proper alignment, **Figure 13-21**.

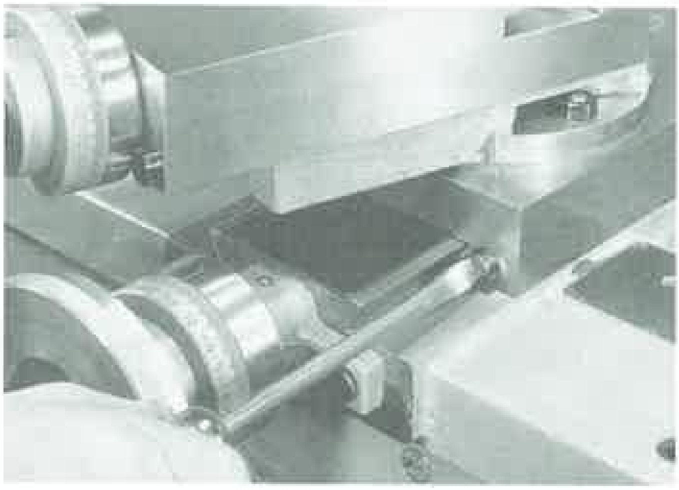


Figure 13-20. *If there is too much play in the cross-slide, adjust gibs according to instructions in manufacturer's handbook. (Clausing Industrial, Inc.)*

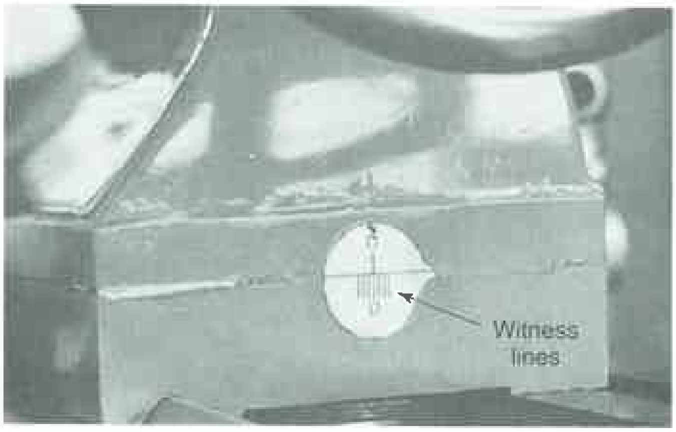


Figure 13-21. *Witness lines on tailstock indicate whether the tailstock is aligned properly with the headstock.*

- Clamp the cutter bit into an appropriate toolholder and mount it in the tool post. Do not permit excessive compound rest overhang, since this often causes tool "chatter" and results in a poorly machined surface, **Figure 13-22**.
- Mount the work. Check for adequate clearance between the work and the various machine parts.

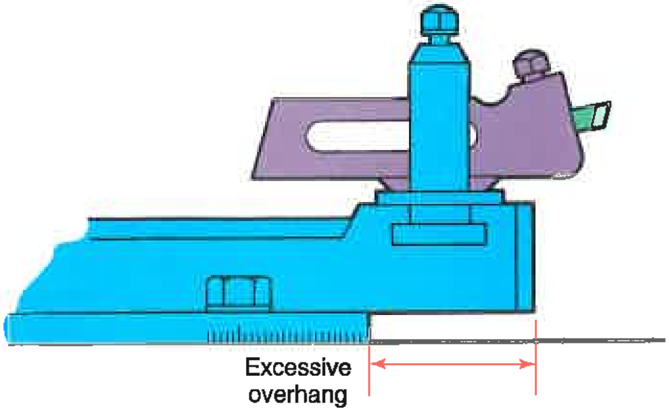


Figure 13-22*. Excessive overhang of the compound rest usually causes tool "chatter," resulting in a surface that is poorly machined.*

In addition to the above procedures, the operator must take some safety precautions. Sleeves should be rolled up and all jewelry removed before beginning to use the lathe.

A lathe board will aid in organizing and holding the tools and measuring instruments needed for the job. See **Figure 13-23.** Loose tools must never be placed on the lathe ways or carriage.

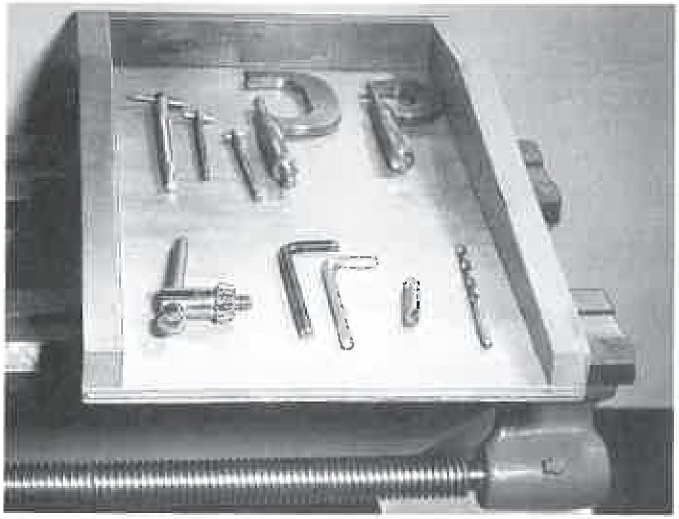


Figure 13-23*. A lathe board is easily made in the school. It keeps tools within easy reach and away from chips and turnings.*

13.4 CLEANING THE LATHE

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

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, **Figure 13-24.** With the lead screw revolving, permit the cord to feed along the thread. Hold the cord just tightly enough to remove the accumulated dirt. *Never* wrap the cord around your hand. The cord could catch and cause serious injury.

13.5 LATHE SAFETY

- 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.

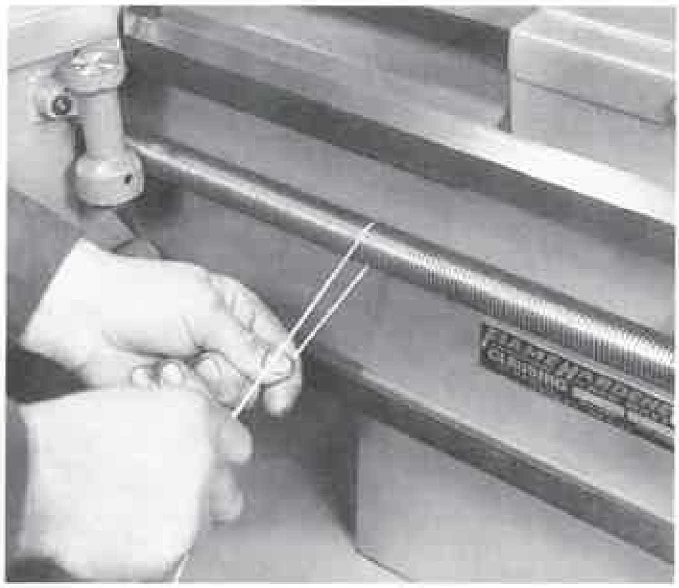


Figure 13-24*. This machinist is cleaning lead screw with a piece of cord. The lead screw should rotate at low speed.* Do not *wrap the cord around your fingers or hands.*

- 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, and 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. Stop the machine and remove them with pliers, **Figure 13-25**.
- 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/or break. See **Figure 13-26**.

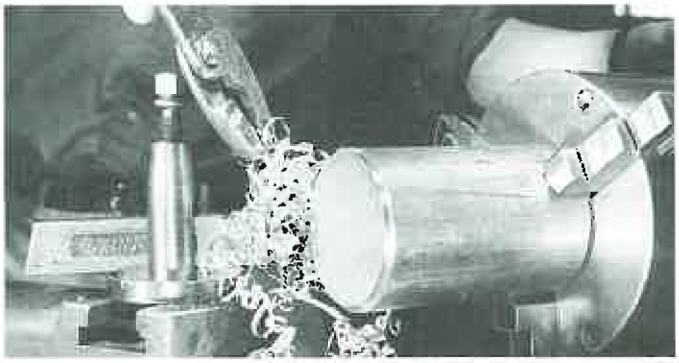


Figure 13-25. *To avoid injury, always remove stringy chips with pliers; never with your hands.*

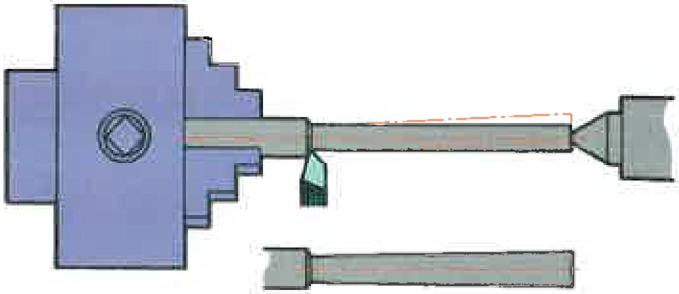


Figure 13-26. *If a small-diameter workpiece is not properly supported by a tailstock center, it will spring away from cutting tool and be machined on a slight taper.*

- 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 doesn't 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.
- Avoid talking to anyone while running a lathe! Do not permit anyone to fool around with 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 dangerous practice to stop such a lathe by reversing the direction of rotation. The chuck could spin off and cause serious injury to you.
- 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.
- Tools must not be placed on the lathe ways. Use a tool board or place them on the lathe tray, **Figure 13-27**.
- 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.

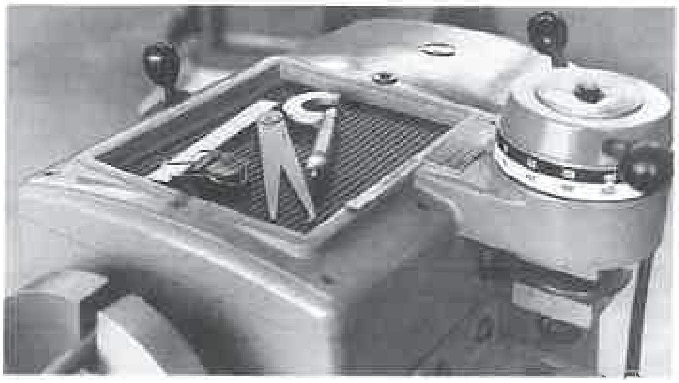


Figure 13-27. Keep tools on a lathe tray; never on the machine ways or in the chip pan.

- 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.

13.6 CUTTING TOOLS AND TOOL HOLDERS

To operate a lathe efficiently, the machinist must have a thorough knowledge of cutting tools and know how they must be shaped to machine various materials. The ***cutting tool*** is held in contact with the revolving work to remove material from the work. In most applications, you will be using a ***single-point cutting tool*** of high-speed steel (HSS).

The square cutter bit body is inserted in a lathe ***toolholder*****, Figure 13-28.** Toolholders are made in straight, right-hand, and left-hand models. 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 holder points to the right, the left-hand toolholders points to the left. A ***turret holder*** may also be utilized. 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.

13.6.1 High-speed steel cutting tool shapes

Figure 13-29 shows the parts of the cutter bit, and the correct terminology for those parts.

To get 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 one only 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 13-30.**

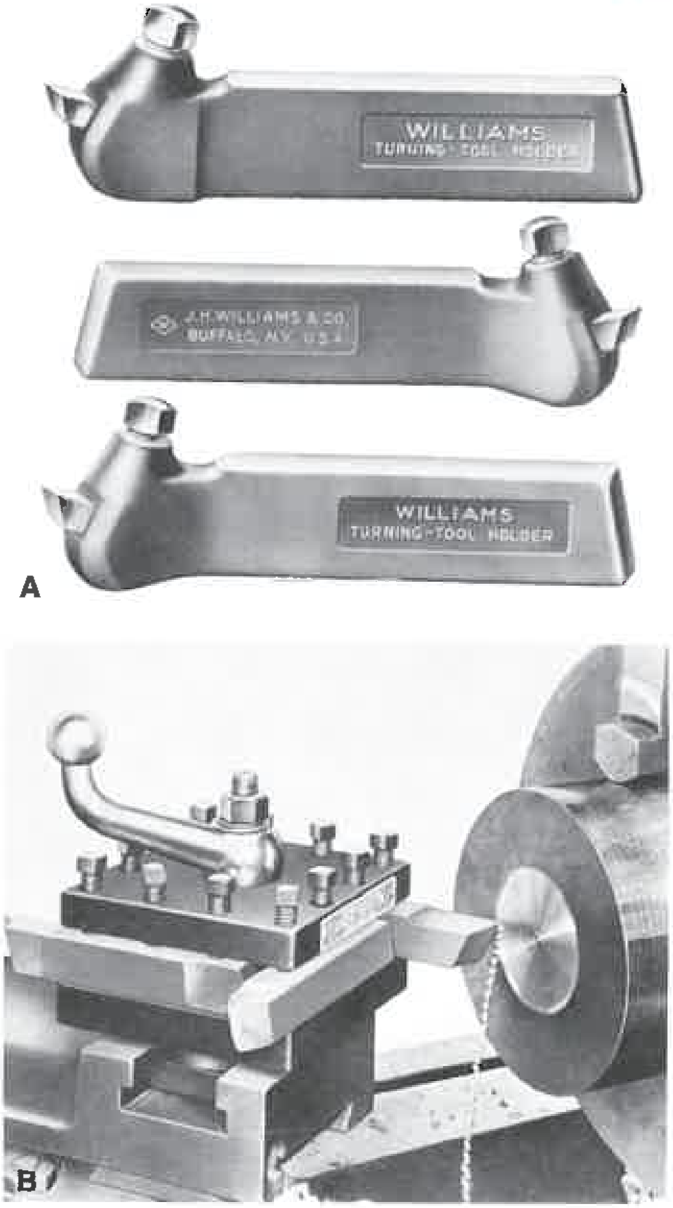


Figure 13-28. Toolholders. A—Straight, left-hand, and right-hand toolholders. (J.H.Williams and Co.) B—A turret-type toolholder with four cutter bits. (ENCO Mfg. Co.)

13.6.2 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

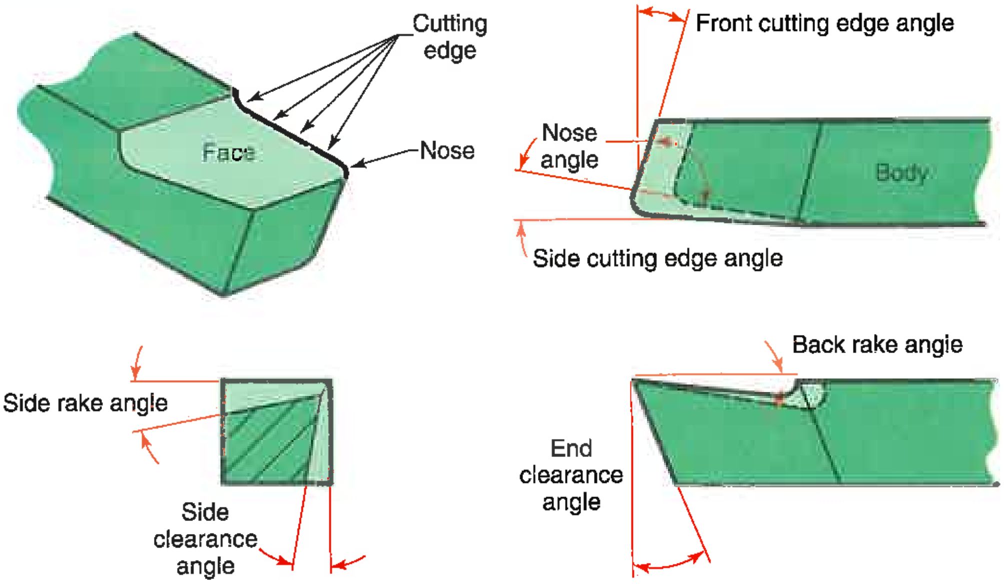


Figure 13-29. *Cutter bit nomenclature.*

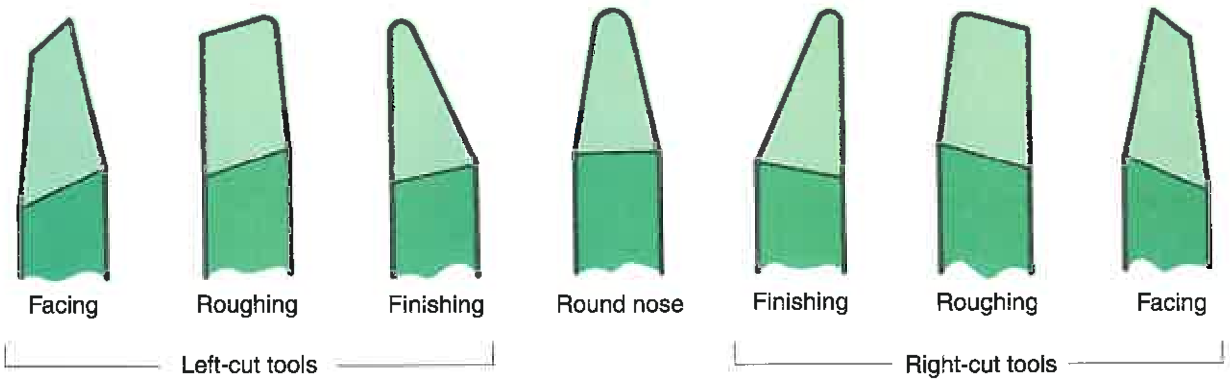


Figure 13-30. *Standard HSS cutting tool shapes.*

small rounded nose. This shape permits deep cuts at heavy feeds. 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, right to left. See **Figure 13-31A.**

13.6.3 Finishing tools

The nose of a ***finishing tool*** is more rounded than the nose of the roughing tool. See **Figure 13-31B.** If the cutting edge is honed with a fine oil stone after grinding, a finishing tool will produce 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.

13.6.4 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 work face with the point leading slightly. See **Figure 13-32A.**

13.6.5 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 13-32B.** A slight variation of the round-nose tool, with a negative rake ground on the face, is excellent for machining brass, **Figure 13-33.**

Machining *aluminum* requires a tool with a considerably different shape from those previously described. As shown in **Figure 13-34,** the tool is set slightly above center to reduce any tendency to

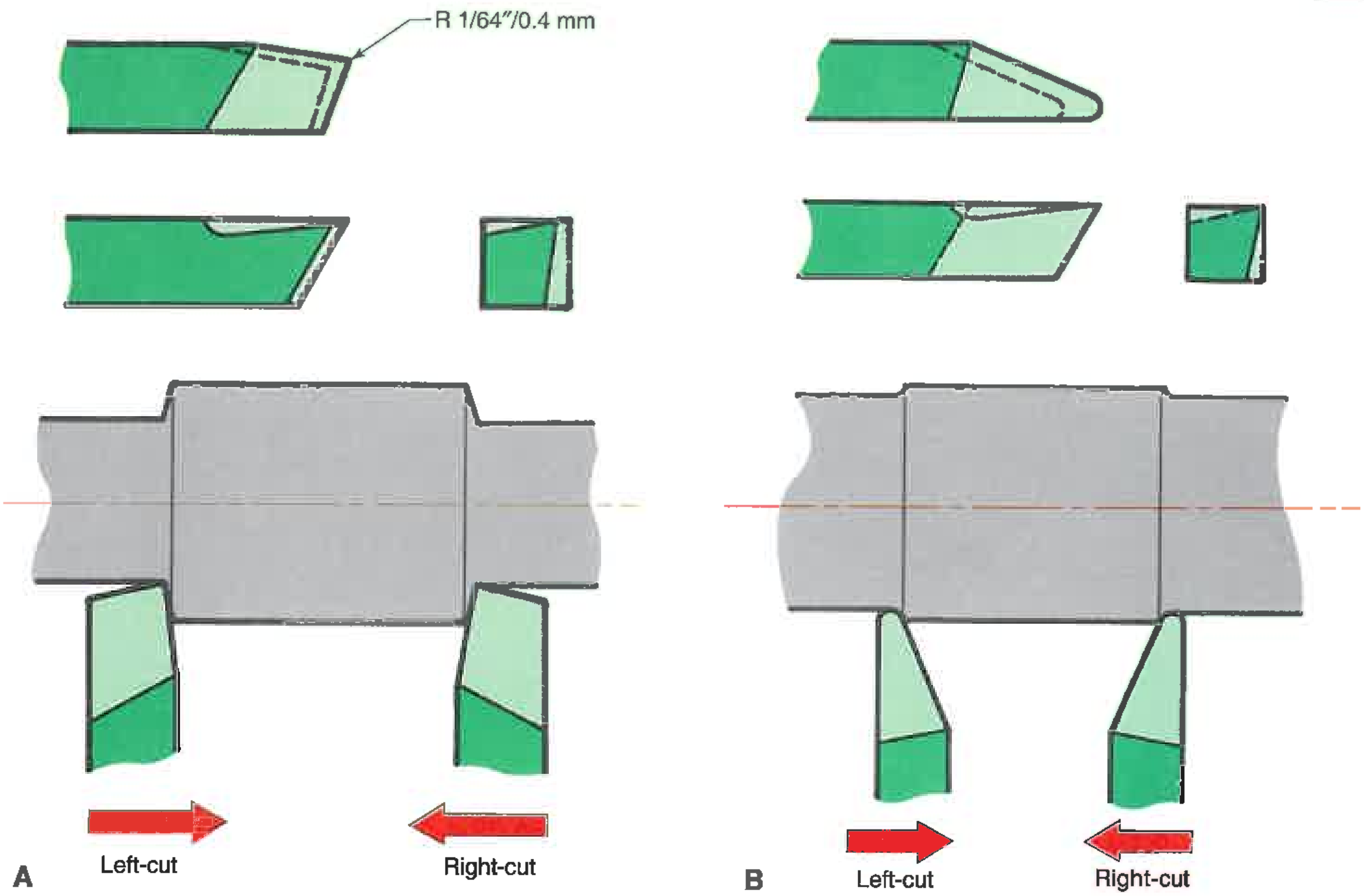


***Figure 13-31**. Lathe tools. A—The roughing tool is used for rapid material removal. B—The finishing tool will produce a smooth surface.*

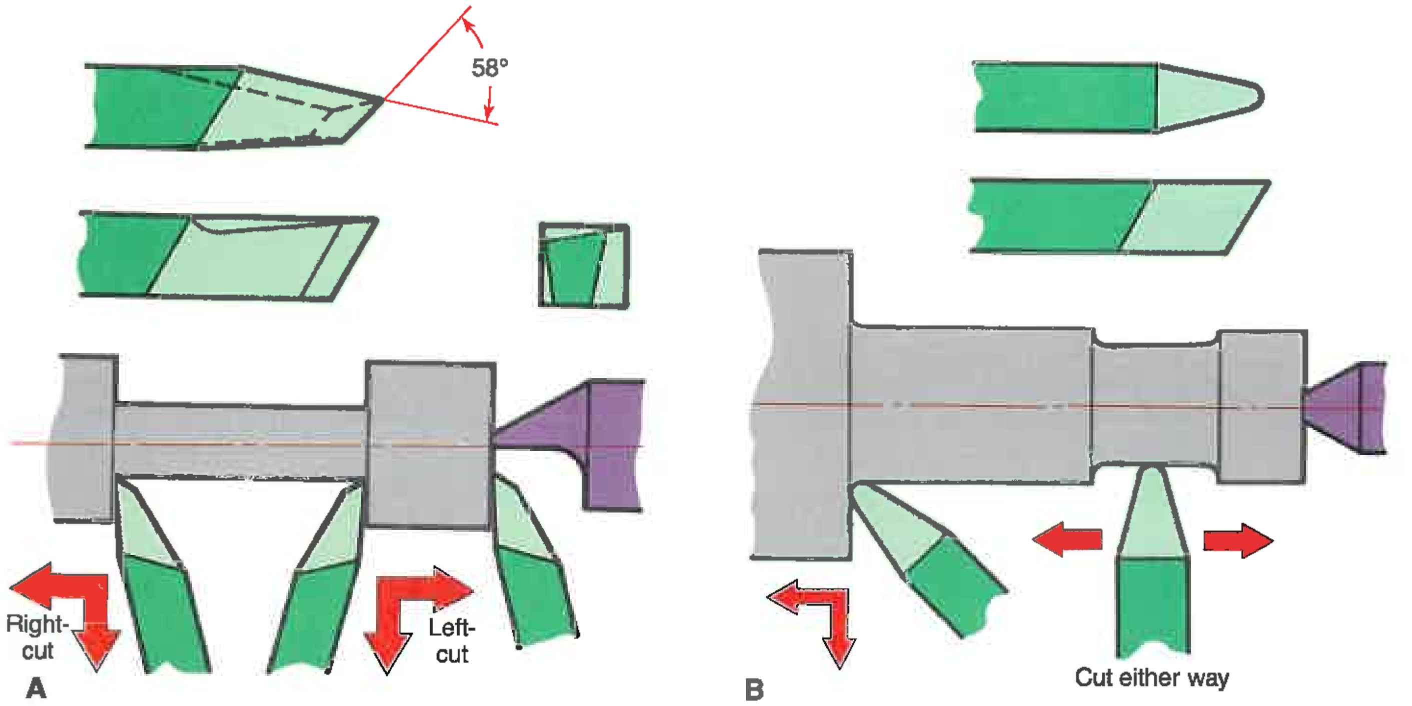


***Figure 13-32**. Lathe tools. A—A facing tool is used to machine surfaces perpendicular to the spindle centerline. B—A round-nose tool will produce fillets. Its shape permits it to cut either left or right.*

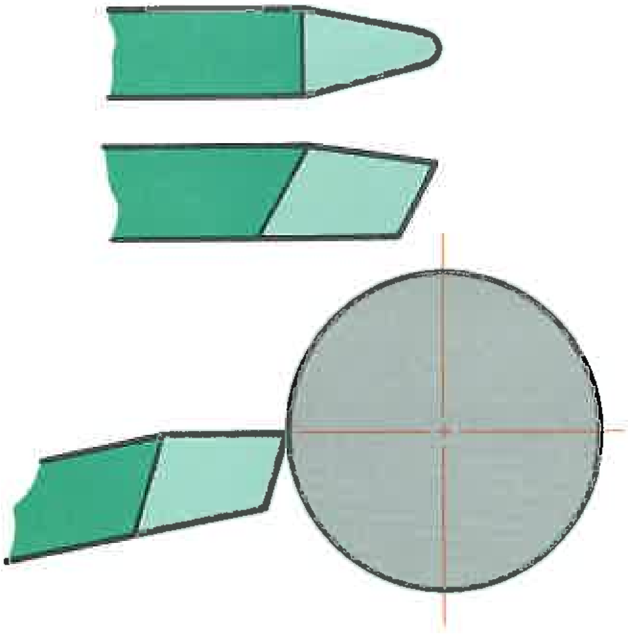


Figure 13-33. Cutter nose shape for machining brass. Note the negative rake.

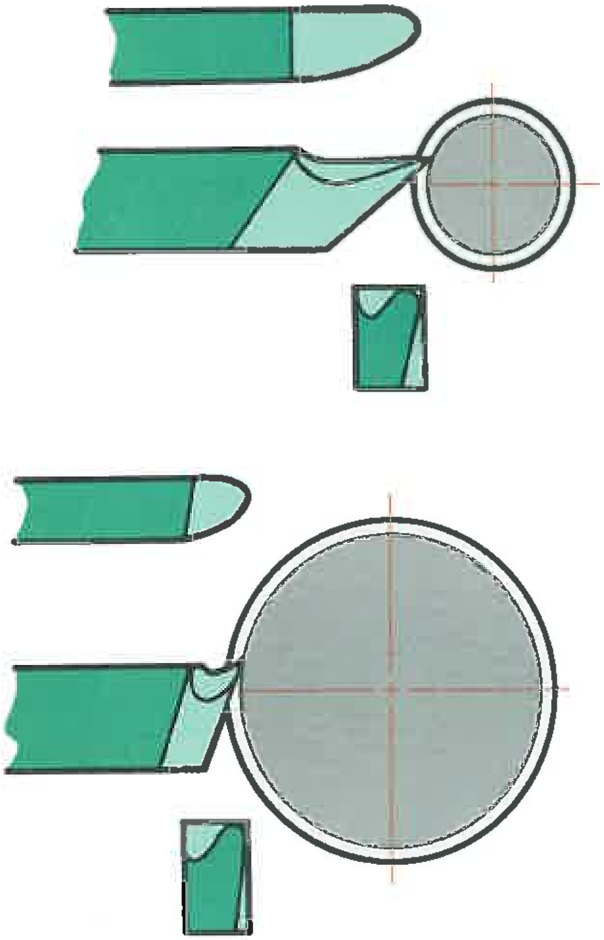


Figure 13-34. For machining aluminum, cutter bit shapes different from those used for other metals are necessary. The tool is set slightly above center for a smoother cut.

chatter (vibrate rapidly). The tool designs illustrated are typical of cutting tools used to machine aluminum alloys.

13.6.6 Grinding high-speed cutter bits

When first attempting to grind a cutter bit, it may be best if you first practice on square sections of cold finished steel rod. You may also want to use chalk or bluing and draw the desired tool shape on the front portion of the blank, as shown in **Figure 13-35.** The lines will serve as guides for grinding.

Figure 13-36 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 13-37**.

Figure 13-35. This cutter blank has been laid out and marked in preparation for grinding.

13.6.7 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 material, **Figure 13-38.** 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

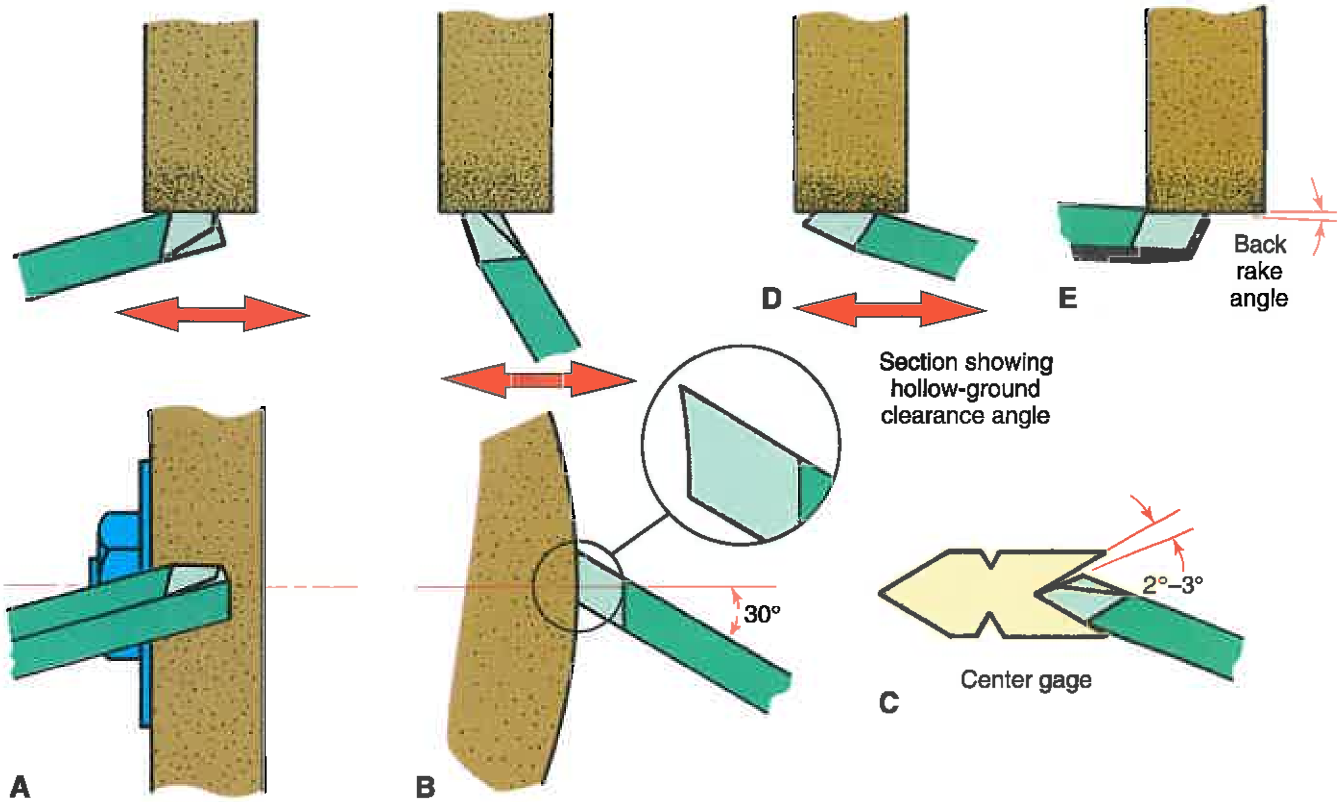
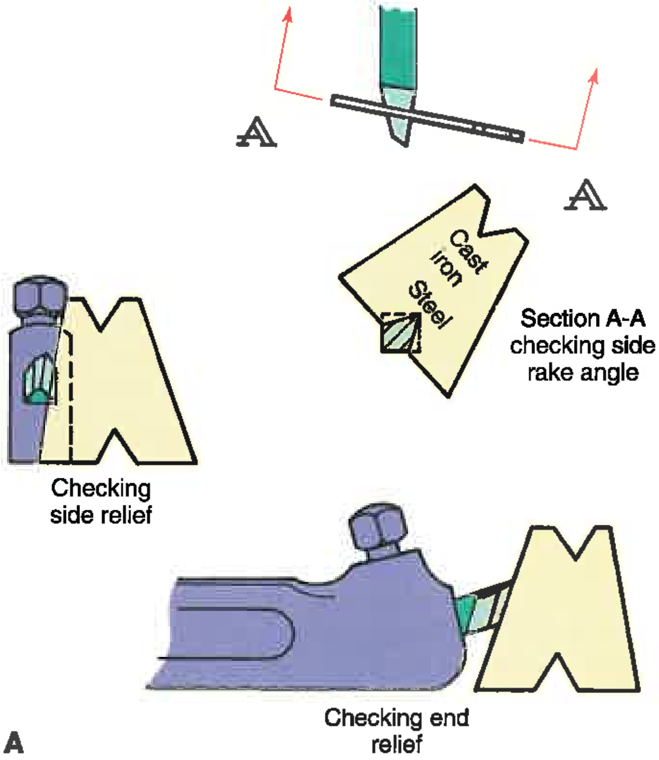


Figure 13-36. *Grinding sequence for a cutter bit. A—Two views showing how to position a cutter bit blank on the grinding wheel to shape side clearance angle and side cutting edge angle. B—Shaping end clearance angle and front cutting edge angle. C—Center gage being used to check nose angle. D—Grinding other side clearance angle, when required. E—Grinding back/side rake angles. Accuracy of clearance angles can be checked with cutter bit gage.*



Rake and Clearance Angle for Lathe Tools
(High-speed steel)

	Cast iron	Low-carbon steel	High-carbon steel
Back rake	6–8°	8–12°	4–6°
Side rake	10–12°	14–18°	8–10°
Clearance*	6–9°	8–10°	6–8°
	Alloy steels	Soft brass	Aluminum
Back rake	5–8°	0–2°	25–50°
Side rake	10–15°	0–2°	10–20°
Clearance*	6–8°	10–15°	7–10°
	Copper		
Back rake	10–12°		
Side rake	20–25°		
Clearance*	6–8°		

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

B

Figure 13-37. *Cutter bit gage. A—Bit gage being used to check accuracy after grinding cutter tip. B—This table provides rake and clearance angles for lathe tools to machine different metals.*

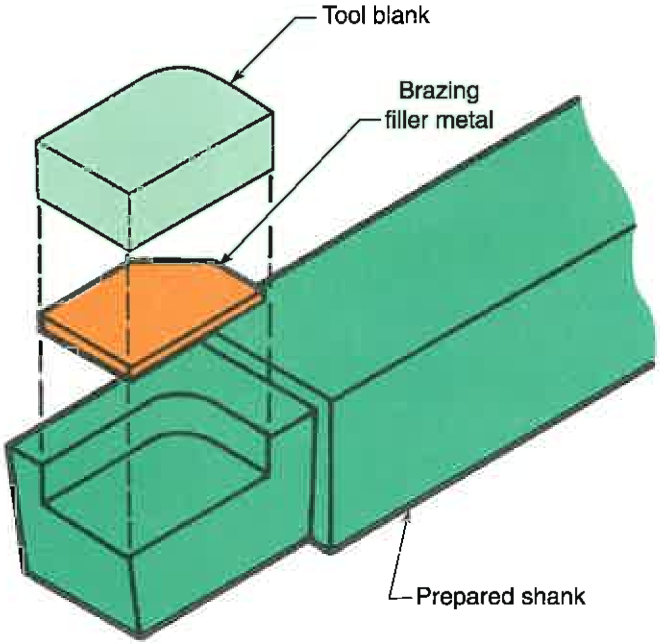


Figure 13-38. Brazing a carbide tool blank into place on a prepared shank. The brazing must be done properly or the tool blank will not be solidly attached, causing it to wear rapidly. Tungsten carbide tool blanks are available in a wide selection of shapes, sizes, and degrees of hardness.

be used, **Figure 13-39.** (Also see Chapter 11, Offhand Grinding.)

Cutting tools designed for machining steel are chamfered 0.003" to 0.002" (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.

13.6.8 Carbide tipped straight turning tools

The cutting tools shown in **Figure 13-40** are general purpose tools for facing, turning, and boring. The ***square nose*** shape permits machining to a square shoulder. Note that the clearance angles of the carbide tools described are not as great as those required for high speed steel cutting tools.

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

13.6.9 Indexable insert cutting tools

Brazed-tip cutting tools are being replaced by mechanically clamped ***indexable insert cutting tools,*** **Figure 13-41.** Indexable insert cutting tools are widely used for turning and milling operations. The inserts are manufactured in a number of shapes and sizes, **Figure 13-42,** for different turning geometries. Six of the most commonly used standard shapes are shown in order of increasing and decreasing strengths. See **Figure 13-43.**

Figure 13-39. A grinder designed for sharpening carbide, cermet, cubic boron nitride, and polycrystalline diamond (PCD) cutting tools. It uses special diamond charged wheels and is fitted with a microscope inspection system.

Indexable inserts clamp to special tool holders, **Figure 13-44.** 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 13-45.** Carbide inserts are given increased versatility (higher abrasion resistance, chemical stability, and lubricity) when coated with various combinations of ***titanium carbide (TiC), titanium nitride (TiN),*** and ***alumina.***

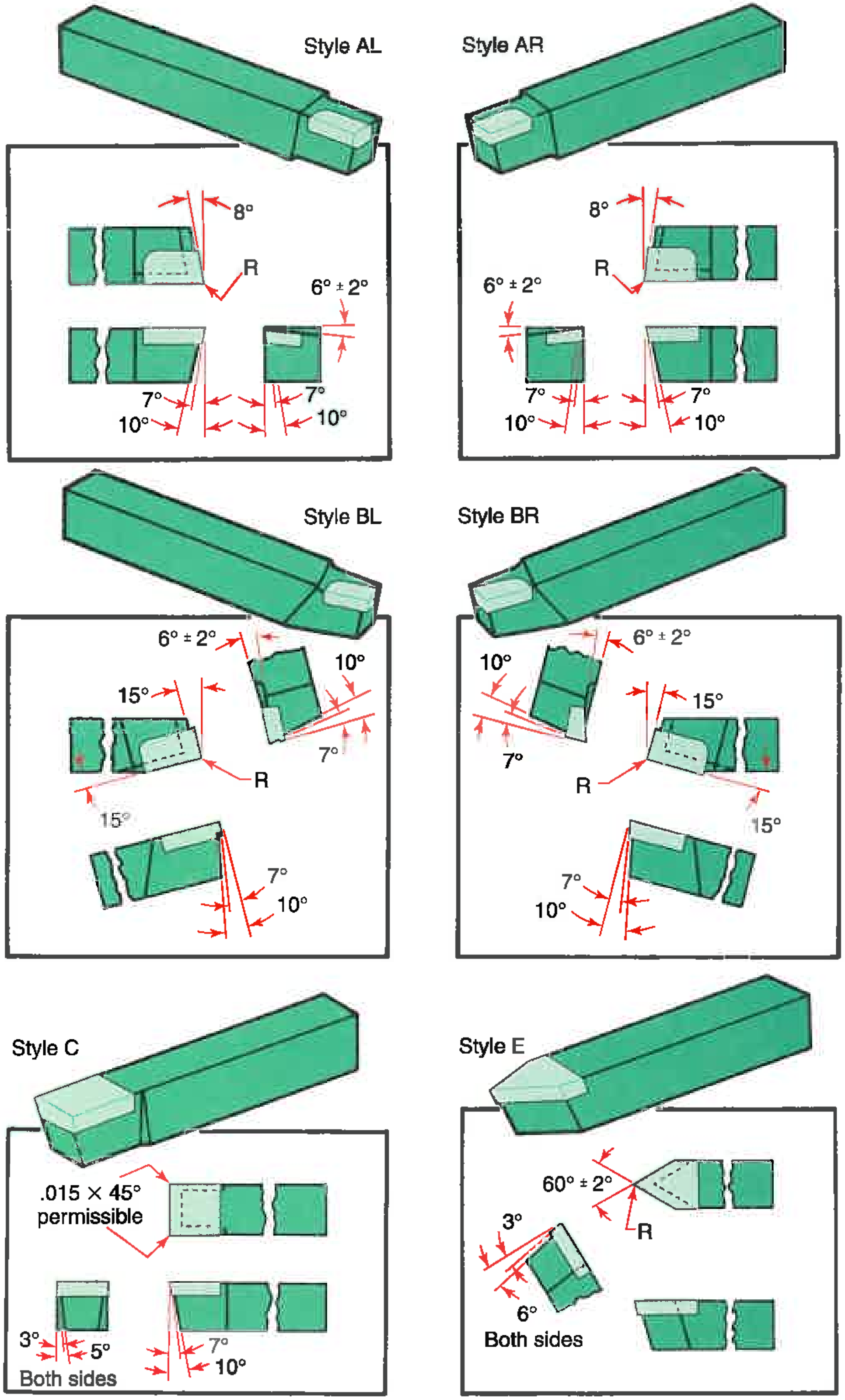


Figure 13-40. *Typical standard cemented-carbide single-point tools. Style E is a carbide tipped threading tool. (Carboloy, Inc.)*

Chipbreakers

When some metals are machined, long continuous chips will be created, unless some method is employed to break the chips into smaller pieces. This is accomplished by a small step or groove, called a ***chipbreaker,*** that is located on the top of the

Figure 13-41*. Indexable insert cutting tools of carbide or sintered oxides (often referred to as cermets) are mechanically clamped into tool holders to perform cutting tasks.This insert is being used to machine stainless steel. (Sandvik Coromant Co.)*

Figure 13-44. *A selection of typical holders 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. (Carboloy, Inc.)*

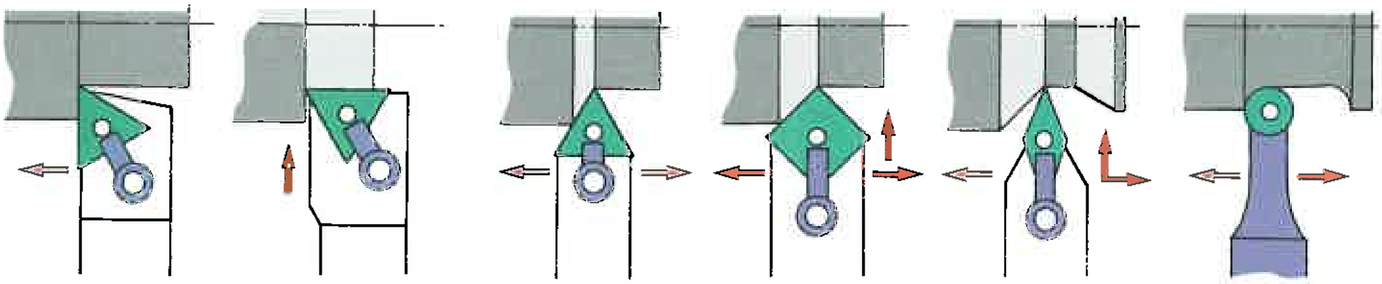


Figure 13-42*. Indexable inserts are manufactured in a number of different shapes and sizes for different turning operations.*

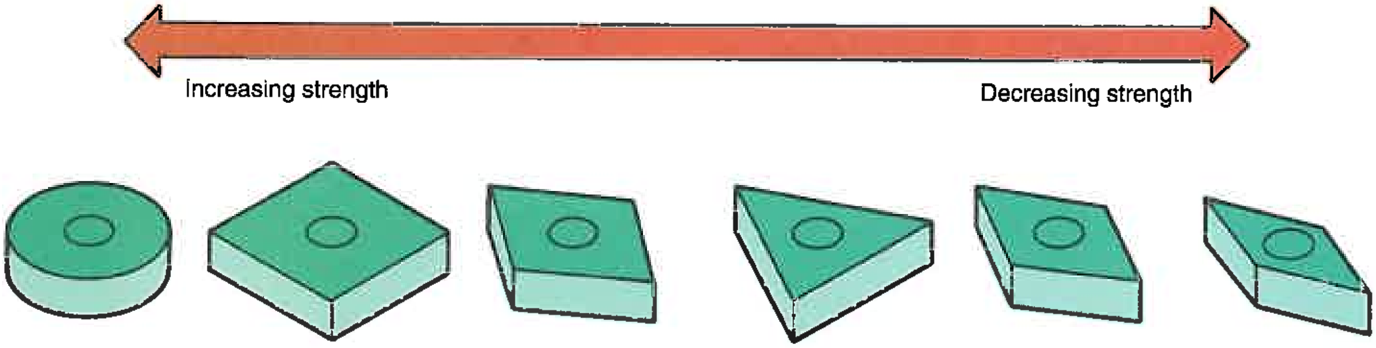


Figure 13-43. *Most commonly used indexable insert shapes are shown in order of increasing and decreasing strengths.*

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 operations on irons, steels, stainless steels, and aluminum alloys.
Ceramic-Hot/Cold Pressed	High abrasion resistance High-speed capabilities 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 thermal 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 38Rc Limited applications High cost	Hardened work materials in 45-70 Rockwell C 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.

Figure 13-45. *The nine basic categories of cutting tool materials. (Valenite, Inc.)*

cutter at the cutting edge. Most inserts manufactured today have molded-in chipbreakers. Other single-point cutting tools must have a chipbreaker ground into the top face of the tool, **Figure 13-46.**

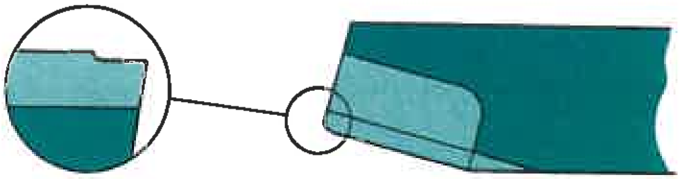


Figure 13-46. *Typical chipbreaker on a single-point tool.*

13.6.10 Other types of cutting tools

Diamonds, both natural and manufactured, are employed 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.***

13.7 CUTTING SPEEDS AND FEEDS

The matter of cutting speed and feed is most important, since these factors govern the length of time required to machine the work and the quality of the surface finish.

Cutting speed indicates 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 this 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 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 amount of feed:

- Material used for the cutting tool.
- Kind of material being machined.
- Desired finish.
- 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 13-47** is a chart for use with high-speed steel cutter bits. Cutting speeds and feeds 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 employed.

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
Aluminum*	600	183	1000	300

The speeds for rough turning are offered as a starting point. It should be all the machine and work will withstand. The finishing feed depends upon the finish quality desired.

* The speeds for turning aluminum will vary greatly according to the alloy being machined. The softer alloys can be turned at speeds upward of 1600 fpm (488 mpm) roughing to 3500 fpm (106 mpm) finishing. High silicon alloys require a lower cutting speed.

Figure 13-47. *Cutting speeds and feeds suggested for turning various metals with high-speed steel tools.*

13.7.1 Calculating cutting speeds

The cutting speeds shown in **Figure 13-47** should be considered as only a starting point. Depending upon machine condition, they may have to be increased or decreased until optimum cutting conditions are obtained.

Cutting speed (CS), as noted, 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:

Inch-based:

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

Where rpm = Revolutions per minute.
CS = Cutting speed recommended for the particular material being machined (steel, aluminum, etc.) in feet per minute.
D = Diameter of work in inches. Convert all fractions to decimals.

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

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

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

$$= 1000 \text{ rpm}$$

CS = Table recommends a cutting speed of 1000 fpm for finish-turning aluminum alloy.
D = 4"

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

Metric-based:

$$\text{rpm} = \frac{\text{CS} \times 1000}{\text{D} \times \pi}$$

Where rpm = Revolutions per minute.
CS = Cutting speed recommended for particular material being machined (steel, aluminum, etc.) in meters per minute (mpm).
D = Diameter of work in millimeters (mm).
π = 3 (Since cutting speeds are approximate, π has been rounded off to 3 from 3.1416 to simplify calculation.)

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

$$\text{rpm} = \frac{\text{CS} \times 1000}{\text{D} \times 3}$$

$$= \frac{300 \times 1000}{100 \times 3}$$

$$= 1000 \text{ rpm}$$

CS = Table recommends a cutting speed of 300 m/pm for finish-turning aluminum alloy.
D = 100 mm
π = 3

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

13.7.2 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.

13.7.3 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 employed.

13.7.4 Depth of cut

The ***depth of cut*** refers to the distance the cutter is fed into the work surface. The depth of cut, like feed, varies greatly with lathe condition, material hardness, speed, feed, 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 13-48.**

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″ or 0.04 mm. See **Figure 13-49**. This must not be forgotten or *twice* as much material as specified will be removed.

Some lathes, however, have a micrometer dial set up so that the number of graduations the cutter

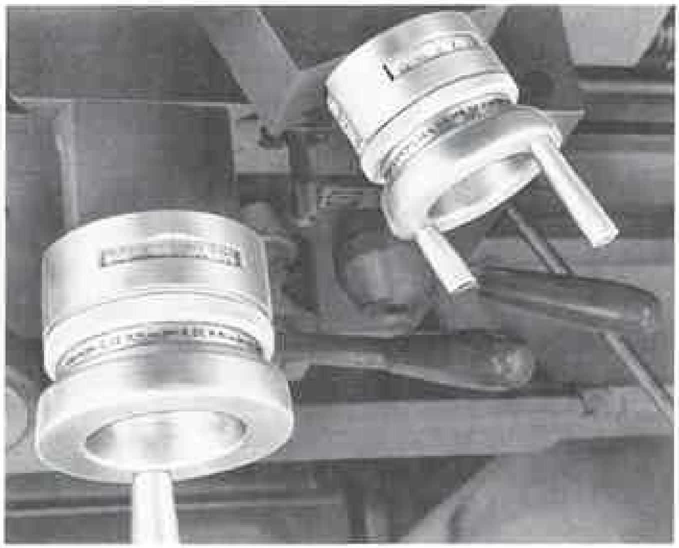


Figure 13-48. *Combination inch/metric-graduated micrometer dials on the cross-slide and compound rest handwheels of a lathe. (Clausing Industrial, Inc.)*

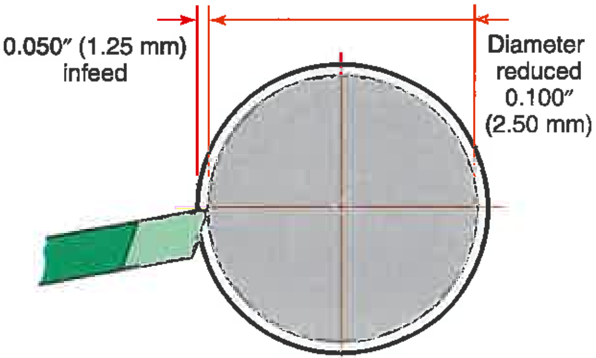


Figure 13-49. *Remember that material is removed from the work on each cut at two times the infeed distance.*

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 employs.

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.

13.8 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 and/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 methods shown in **Figure 13-50**:

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

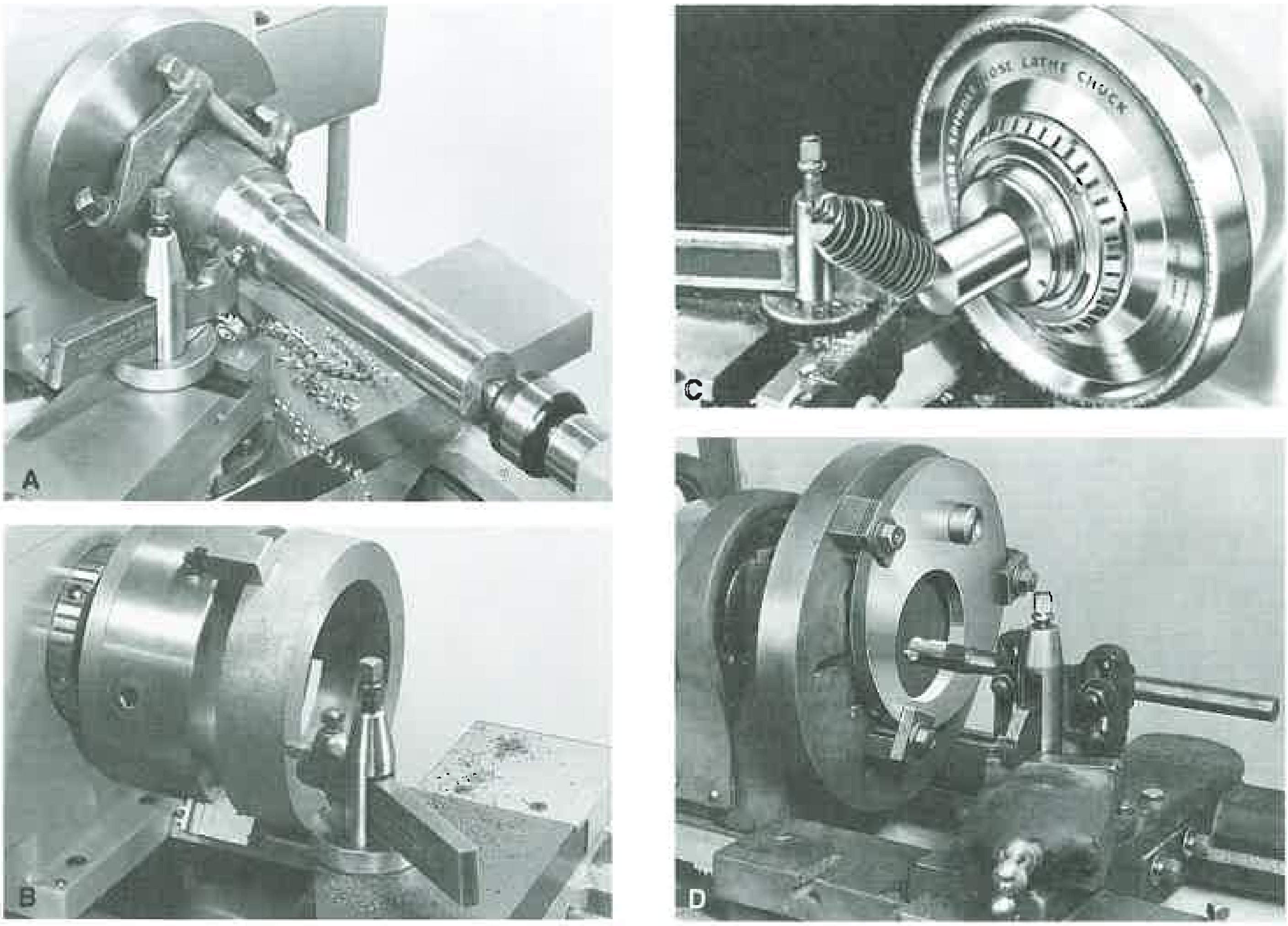


Figure 13-50. *Work-holding methods. A—Work being machined between centers. B—Work held in a chuck for machining. (Clausing Industrial, Inc.) C—Work being machined while held in a collet. D—Work bolted to a faceplate for machining.*

13.9 TURNING WORK BETWEEN CENTERS

Considerable lathe work is done with the workpiece supported between centers. For this operation, a faceplate, **Figure 13-51,** is attached to the spindle nose. A sleeve and live center are inserted into the spindle opening, **Figure 13-52.**

Either a nonrevolving ***dead center*** or a heavy duty ***ball bearing center*** is fitted into the tailstock spindle to support one end of the work. See **Figure 13-53.** The ends of the stock are drilled to fit over the center points.

A lathe dog, **Figure 13-54,** is clamped to one end of the material. Three types of ***lathe dogs*** are shown in **Figure 13-55:**

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

Figure 13-51. *Lathe faceplates come in various sizes.*

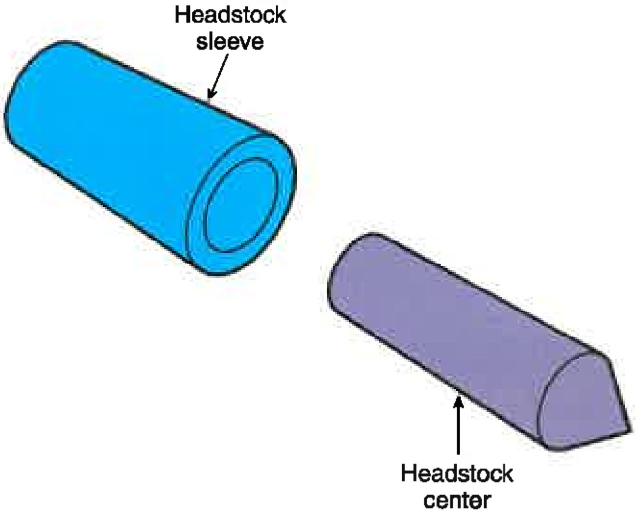


Figure 13-52. *Sleeve and headstock center.*

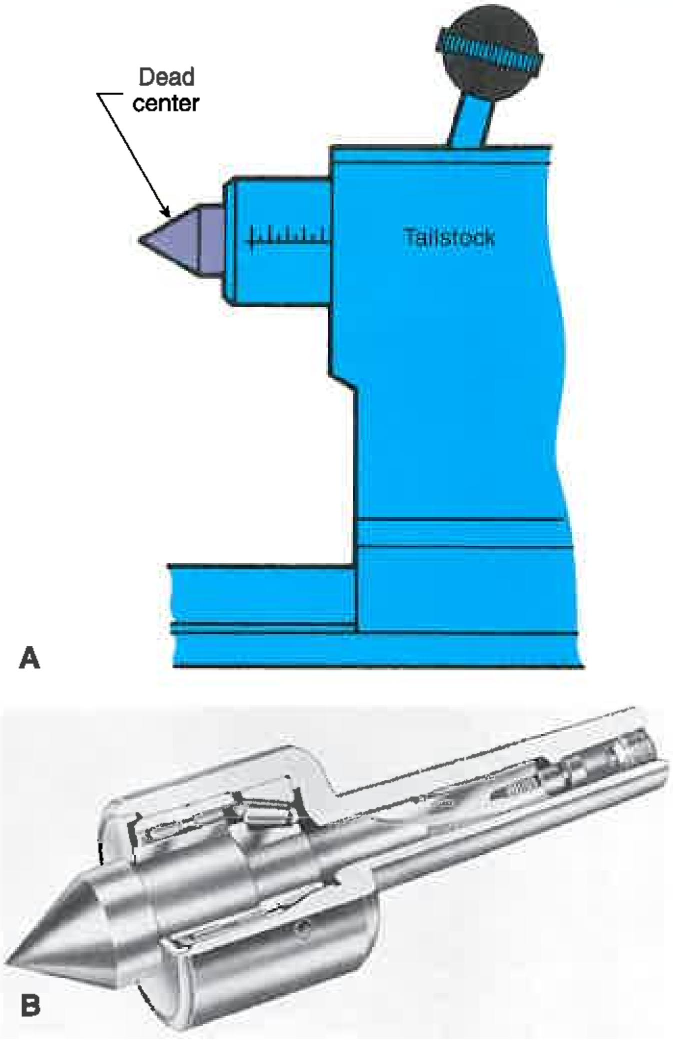


Figure 13-53. *Tailstock centers. A—A dead center does not rotate. It is fitted into the tailstock spindle. B—Cutaway shows construction of a heavy-duty ball bearing center.*

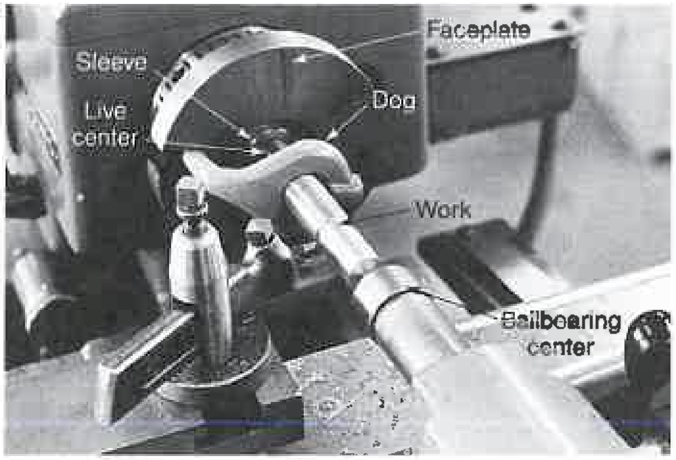


Figure 13-54. *Machining work mounted between centers.*

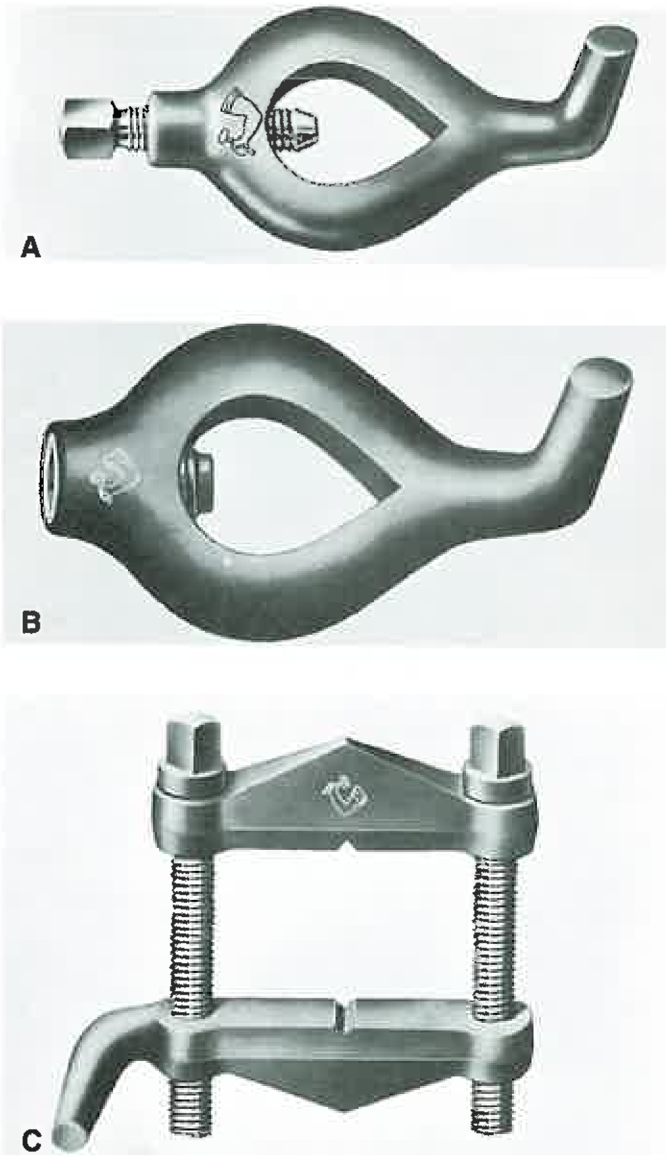


Figure 13-55. Lathe dogs. A—Bent-tail standard dog. B—Bent-tail safety dog. C—Clamp-type dog. (Armstrong Bros. Tool Co.)

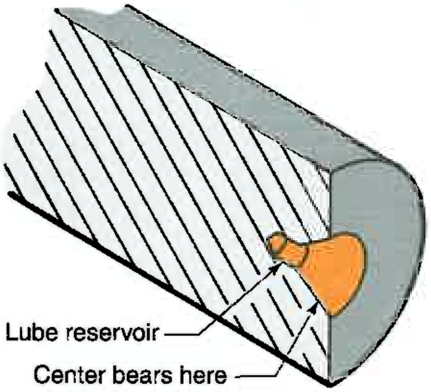


Figure 13-56. Tail center rides in the drilled and countersunk center hole. A supply of lubricant is placed in reservoir. The lubricant will expand and lubricate the center as metals heats up.

13.9.1 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 13-56**. Several methods for locating the center of round stock are shown in **Figure 13-57.**

Center holes are usually drilled with a ***combination drill and countersink***, **Figure 13-58**. The ***drill angle*** is identical to that of the center point.

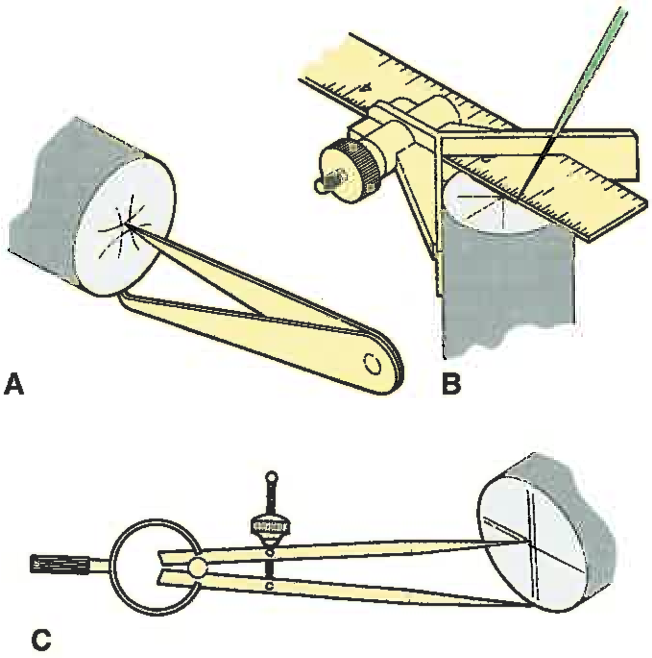
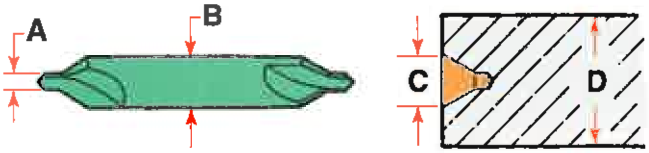


Figure 13-57. Several ways to locate the center of round stock. A—With a hermaphrodite caliper. B—With center head and rule of combination set (recommended method). C—With dividers.



Combination drill and countersink no.	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

Figure 13-58. This size chart contains information needed to select correct-size center drill. Combination drill and countersink makes the hole and countersinks it in one operation.

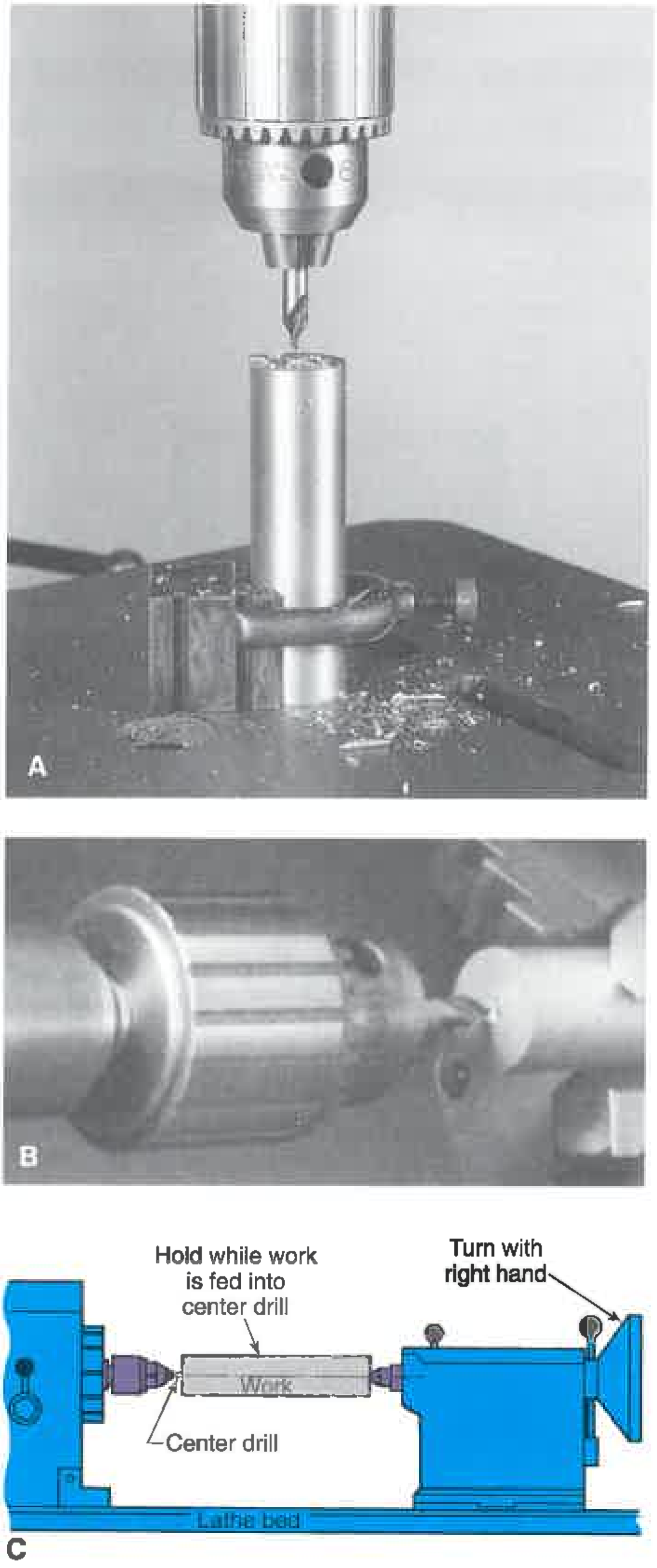


__Figure 13-59.__ Drilling center holes. A—Holes can be drilled on a drill press. Mount work in a V-block for support. B—Some work can be held in a lathe chuck for center drilling. C—Center holes can be drilled in large stock by mounting a Jacobs chuck in the headstock. Locate center point of each end and center punch. Support one end on tail center and feed other into center drill mounted in the Jacobs chuck. Repeat operation on second end.

The ***straight drill*** provides clearance for the center point and serves as a reservoir for a lubricant. The chart provides information needed to select the correct size center drill.

The center holes can be drilled on a drill press, on the lathe with the work centered in the chuck, or on the lathe with the center drill held in the headstock. See **Figure 13-59.**

The center holes should be drilled deep enough to provide adequate support, **Figure 13-60.**

13.9.2 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

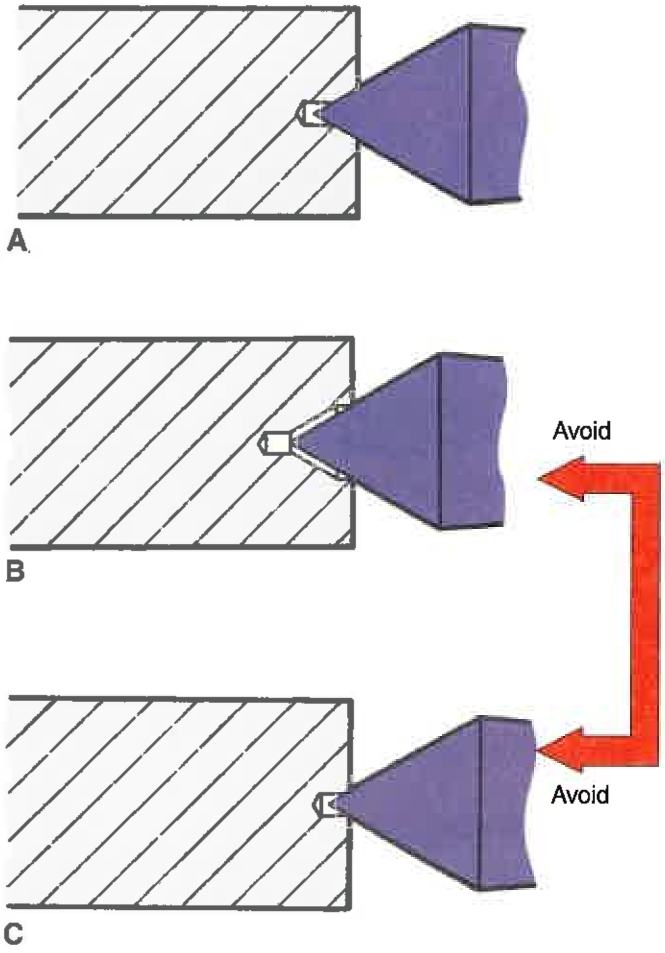


__Figure 13-60.__ Correctly and incorrectly drilled center holes. A—Properly drilled center hole. B—Hole drilled too deep. C—Hole not drilled deep enough. Does not provide enough support; if used with a dead center, the center point will burn off.

run true. If it does *not,* the diameters will be ***eccentric*** (not aligned on the same center line), **Figure 13-61.** 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 will be needed to true a hardened center.

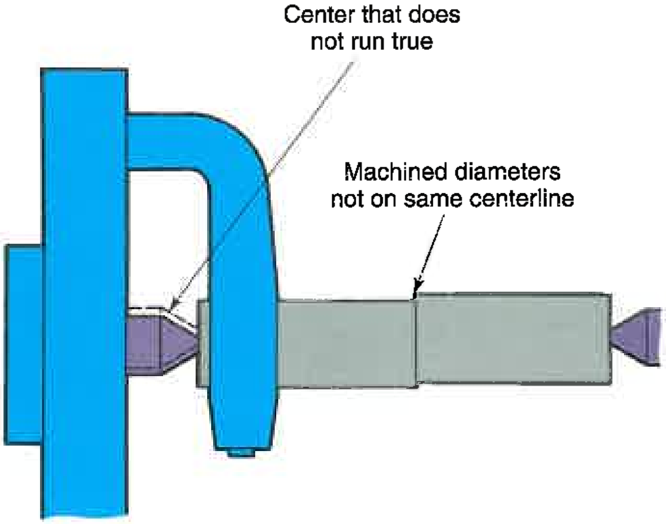


Figure 13-61*. The workpiece must be reversed in the lathe dog so it can be machined for its entire length. If the live center does not run true, eccentric diameters will result.*

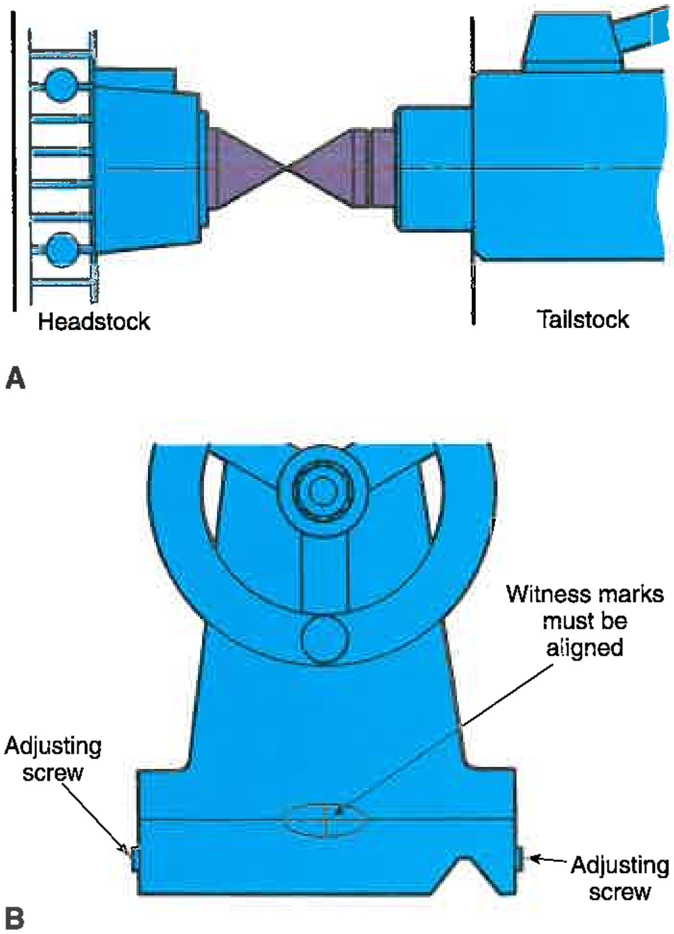


Figure 13-62*. Checking center alignment. A—Checking alignment by bringing center points together. View is looking down on top of centers. B—Alignment can be determined by checking witness lines on base of tailstock.*

Approximate alignment can be determined by checking centers visually by bringing their points together, or by checking the witness lines on the base of the tailstock for alignment. See **Figure 13-62.**

A more precise method for checking alignment is needed if close tolerance work is to be done. Several such methods are described below:

- Make a light trial cut across a few inches of the material. Check the diameter at each end with a micrometer, **Figure 13-63**. The centers are aligned if the readings are identical.
- Use a steel test bar and dial indicator, **Figure 13-64**. Mount the test bar between centers and position the dial indicator in the tool post and 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.

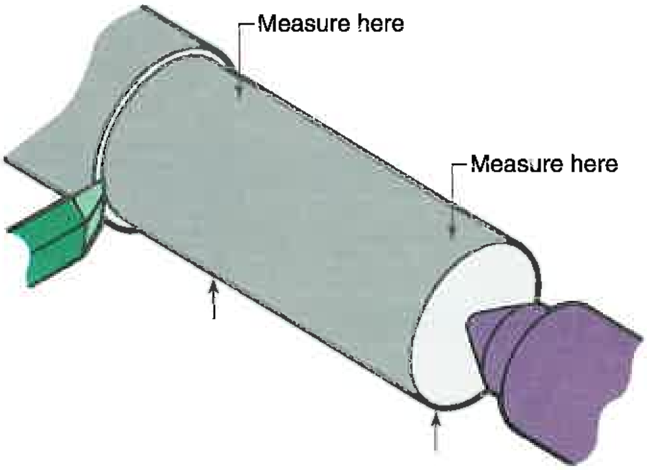


Figure 13-63*. Make a light cut on stock and measure diameter at two points to check alignment. Measurements must be equal.*

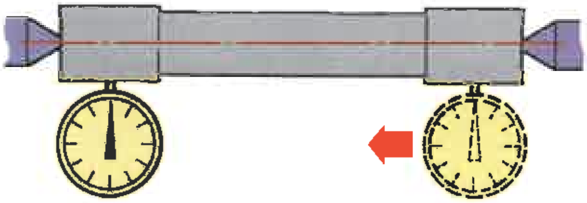


Figure 13-64. Using a test bar and dial indicator to check center alignment.

- Machine a section of scrap, as shown in **Figure 13-65**. 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 this is needed. Make adjustments gradually. See **Figure 13-66**.

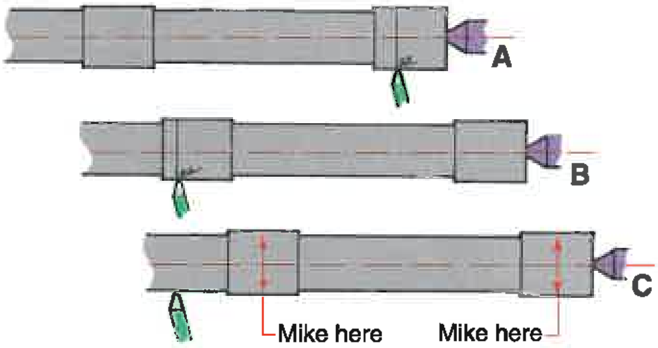


Figure 13-65. Checking for center alignment. A—Machine two shoulders on a test piece. B—Keep same tool setting and make a cut on both shoulders. C—Measure resulting diameters.

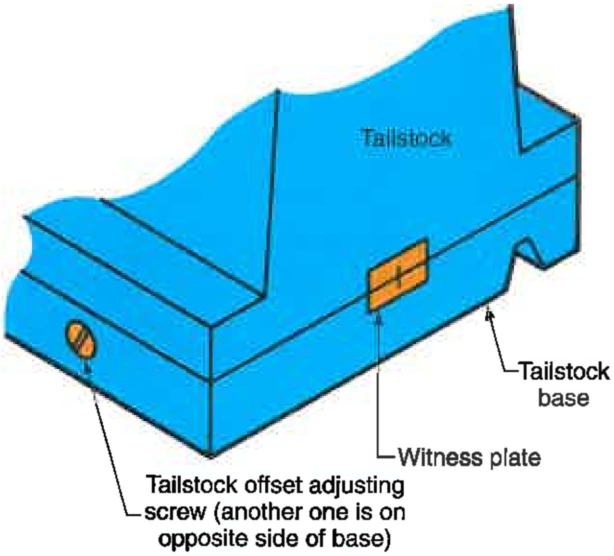


Figure 13-66. Adjusting screws are located on both sides of tailstock base. Their primary use is to set over or shift the tailstock for taper turning.

13.9.3 Mounting work between centers

Clamp a dog to one end of the work. Place a ***lubricant*** (white lead, 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, since heat generated by the machining process will cause the work to expand. Using a ball bearing 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 13-67**. This can cause the work to be pulled off center. When machined, this will produce a surface that is not concentric with the center hole. If binding is occurring, use a different faceplate.

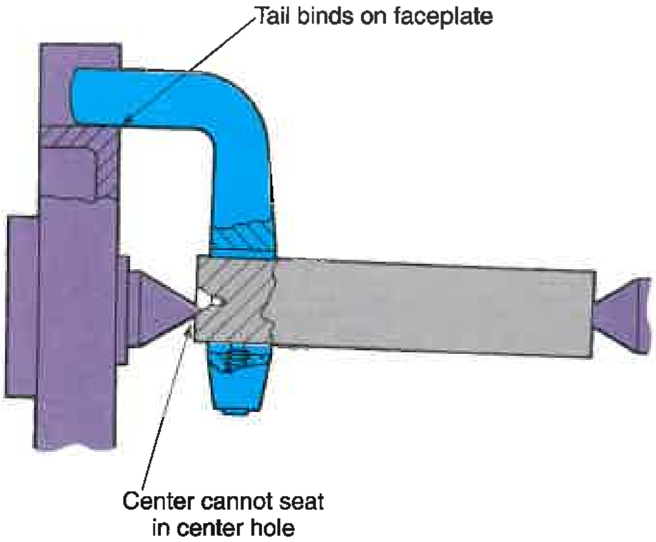


Figure 13-67. Diameter of the turned surface will not be concentric with the center holes if center hole is not seated properly on the live center. A binding lathe dog is a common cause of this problem.

13.9.4 Facing work held between centers

Facing is an operation that machines the end of the work square and reduces it to a specific length. 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. A ***right-cut facing tool*** will be needed. The 58° point on this tool provides a slight clearance between the center point and the work face, **Figure 13-68**. 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— it does not provide an adequate bearing surface for general work and will not hold lubricant.

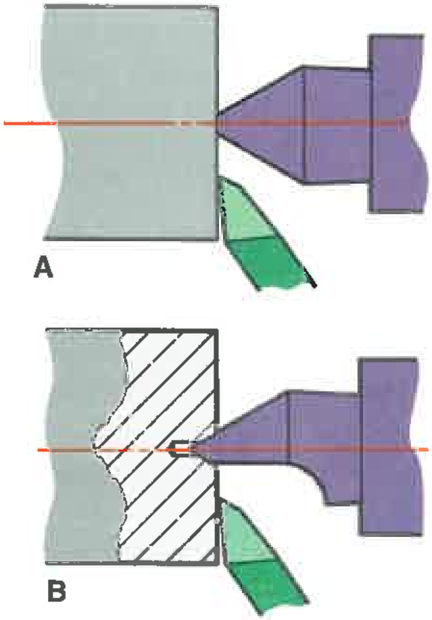


Figure 13-68. *Facing stock. A—Relationship of cutter bit to work face when making a facing cut. B—Using a half center will give more clearance when facing end of stock.*

13.9.5 Facing to length

It is standard machining practice to cut stock slightly longer than needed to permit its ends to be machined square. A steel rule may be employed 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.

Set the compound rest at 30°, **Figure 13-69**. Bring the cutting tool up until it just touches the surface to be machined, then lock the carriage. Remove material from each end of the stock until the specified length is attained.

13.9.6 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 employing deep cuts and coarse feeds.

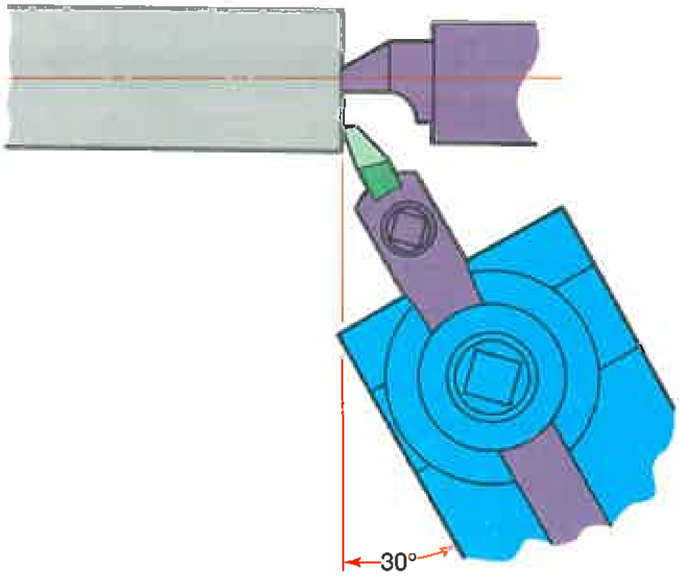


Figure 13-69. *Recommended compound rest setting when facing stock to length.*

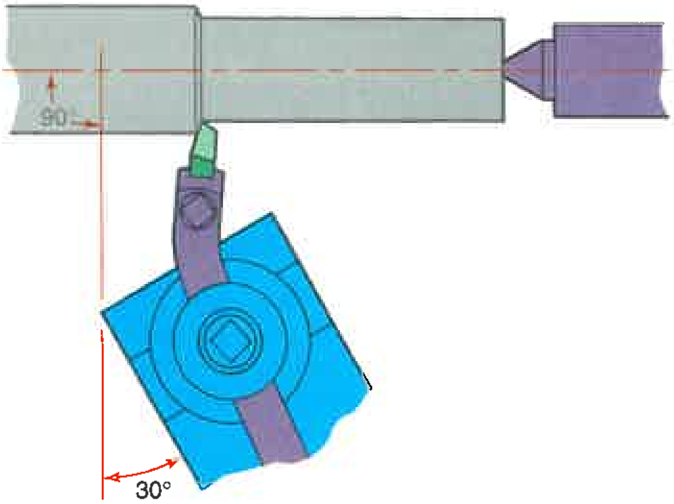


Figure 13-70. *Compound rest setting used for rough turning.*

Set the compound rest at 30° from a right angle to the work, **Figure 13-70**. 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.

CAUTION: Always check the maximum distance that 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 left as possible in the compound rest T-slot. Avoid excessive tool overhang, **Figure 13-71**.

Locate the cutting edge of the tool about 1/16" (1.5 mm) above work center for *each inch* of diameter, **Figure 13-72**. 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.

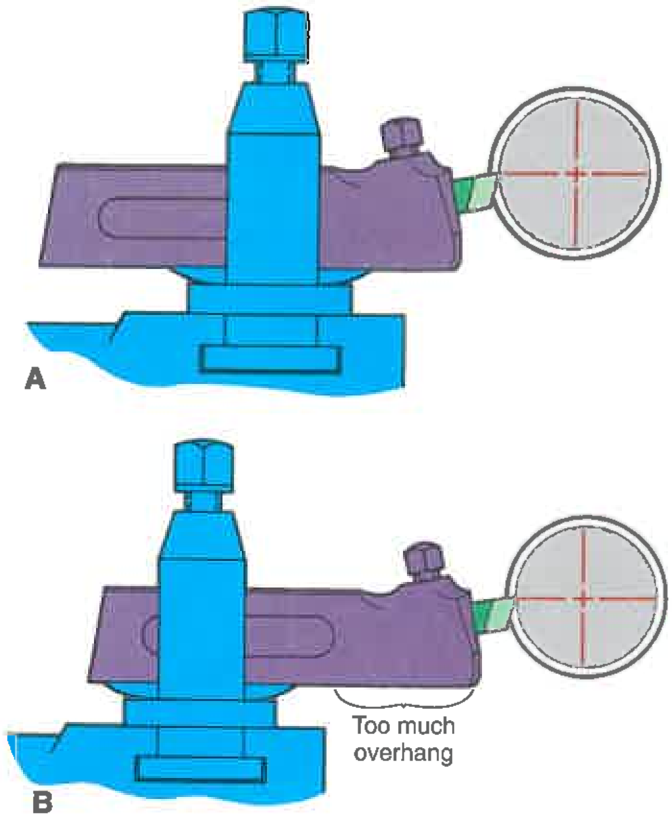


Figure 13-71. *Correct and incorrect mounting of tool holder. A—Tool holder and cutter bit in proper position. B—Too much overhang will cause the tool to "chatter" and produce a rough machined surface.*

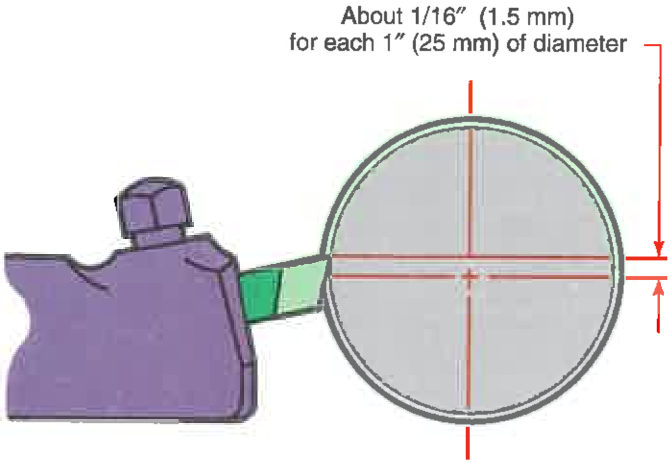


Figure 13-72. *Correct location of cutter bit point when rough turning. Lower the point as work diameter decreases in size.*

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 will pivot away from the work. See **Figure 13-73.**

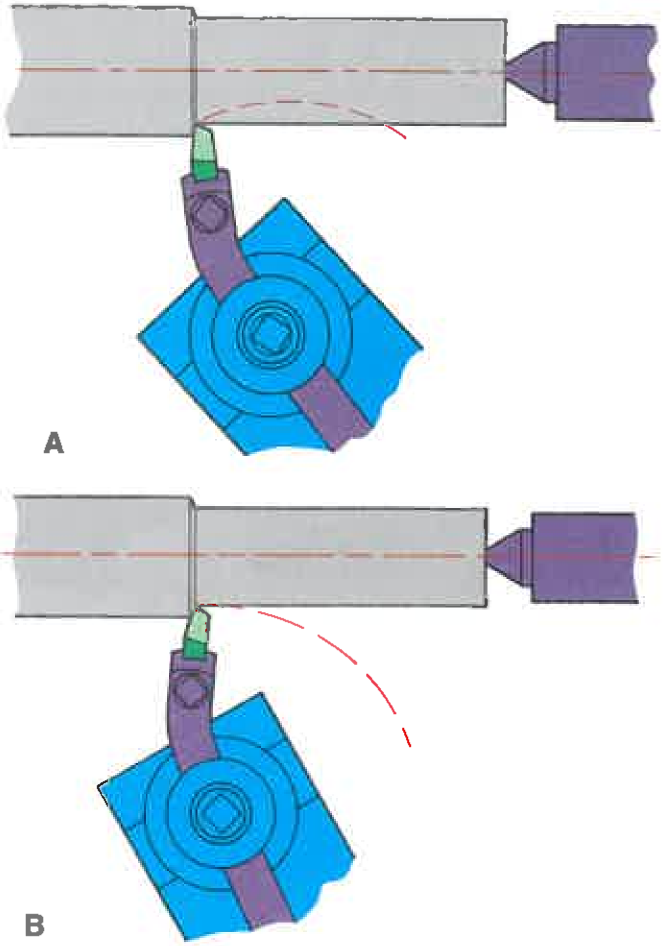


Figure 13-73. *Toolholder positioning. A—An incorrectly positioned tool will cut deeper into work if toolholder slips in the tool post. B—A correctly positioned tool will swing clear of work if tool holder slips.*

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 tool with one that is properly sharpened.

After each cut, measure work diameter to prevent excess metal removal. 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."

13.9.7 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 13-74.**

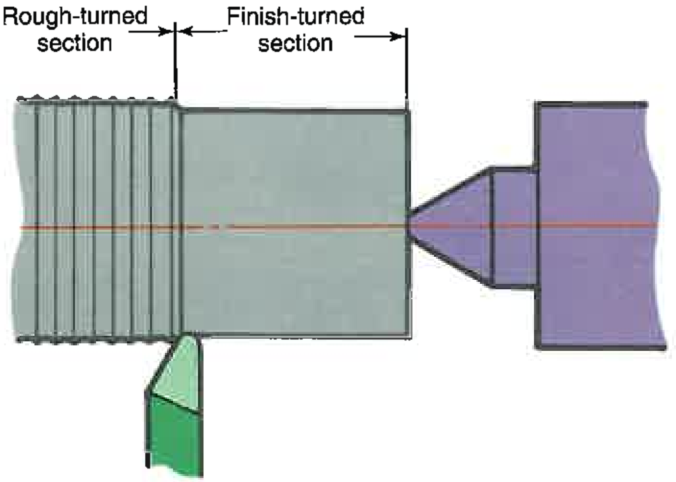


Figure 13-74. *After roughing work to approximate size, turn it to required size with a finishing tool. Cut should be made from right to left.*

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 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. Never reverse a lathe; brake it to a stop!

Do not interfere with the cross-slide setting. "Mike" 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 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 of the finished surface by the lathe dog setscrew. Insert a small piece of soft aluminum or copper sheet between the setscrew and the workpiece.

13.9.8 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 which has been set to the required length, **Figure 13-75.**

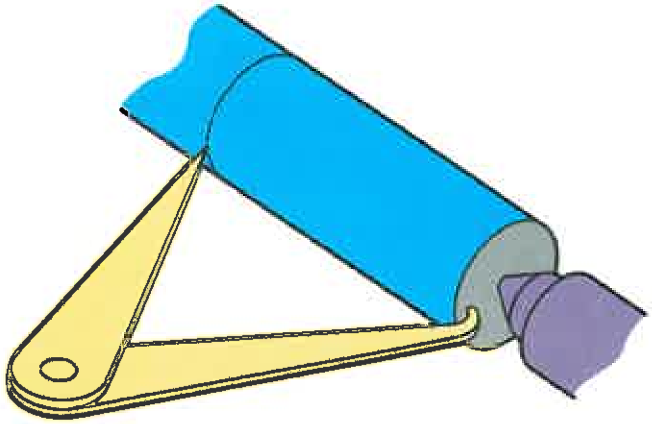


Figure 13-75. *Scribing reference lines on a workpiece with a hermaphrodite caliper.*

Machining is done as previously described, with the exception of cutting the ***shoulder,*** (the point where the diameters change). **Figure 13-76** shows the four types of shoulders:

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

A *right-cut tool* is used to make the square and angular type shoulders. See **Figures 13-77** and **13-78.** For machining a filleted shoulder, a ***round nose tool*** is ground to the required radius using a fillet or radius gage to check radius accuracy. See **Figure 13-79.**

13.9.9 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 pieces, **Figure 13-80.** As any recess cut into a surface has a tendency to weaken a shaft, it is better to make the groove *round,* rather than square.

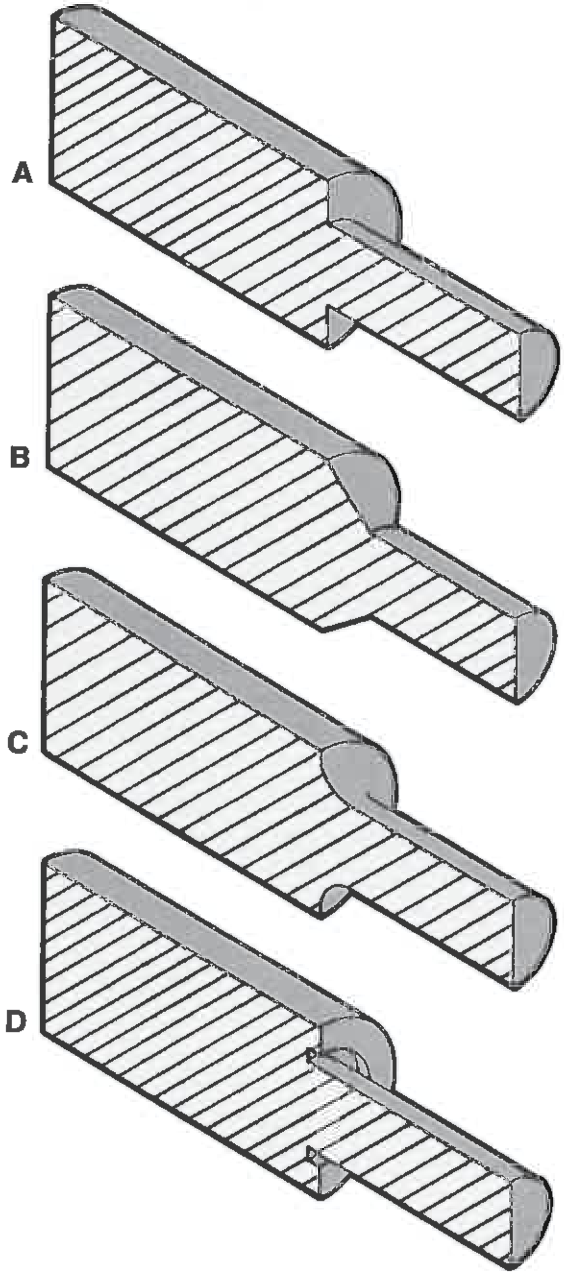


Figure 13-76. Four types of shoulders. A—Square. B—Angular. C—Filleted. D—Undercut.

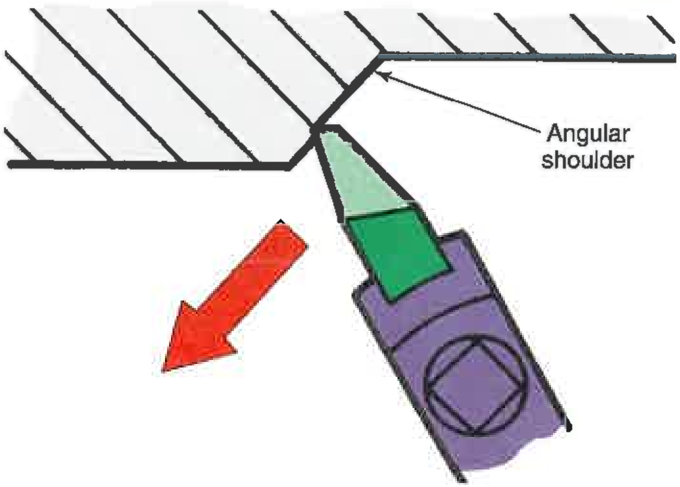


Figure 13-77. To machine an angular shoulder, cut is made from smaller diameter to larger diameter.

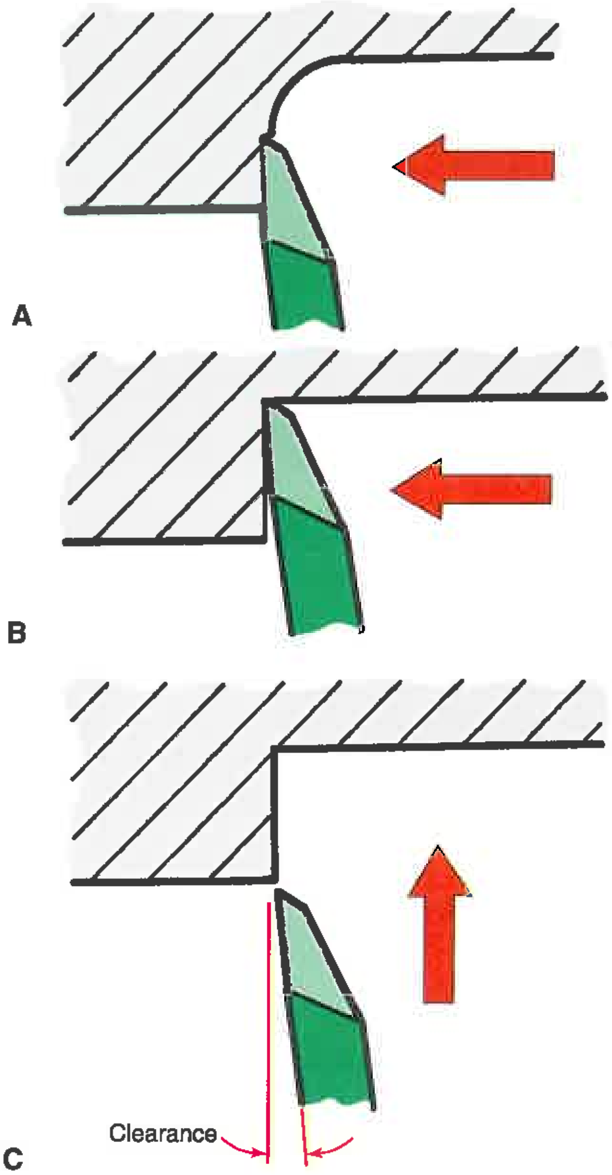


Figure 13-78. Machining sequence used for cutting a square shoulder. A—First cut. B—Second cut. C—Facing cut.

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 the tool in the required number of thousandths/millimeters for the specified depth. Square grooves can be machined with a *parting tool.*

13.10 USING LATHE CHUCKS

The chuck is another device for holding work in a lathe. *Chucking* is the most rapid method of mounting work; for that reason, it is widely preferred. Other operations, such as drilling, boring,

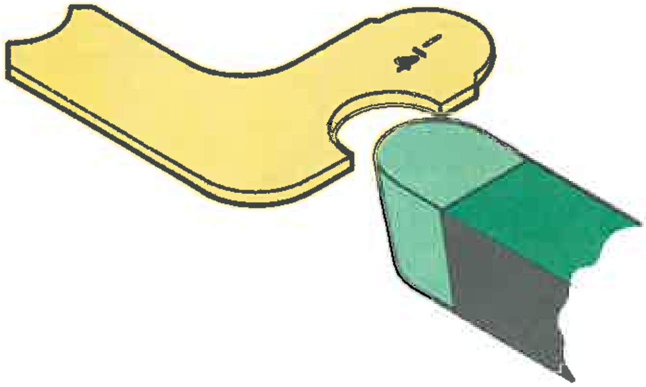


Figure 13-79. The radius ground on a cutter bit can be checked with a fillet gage.

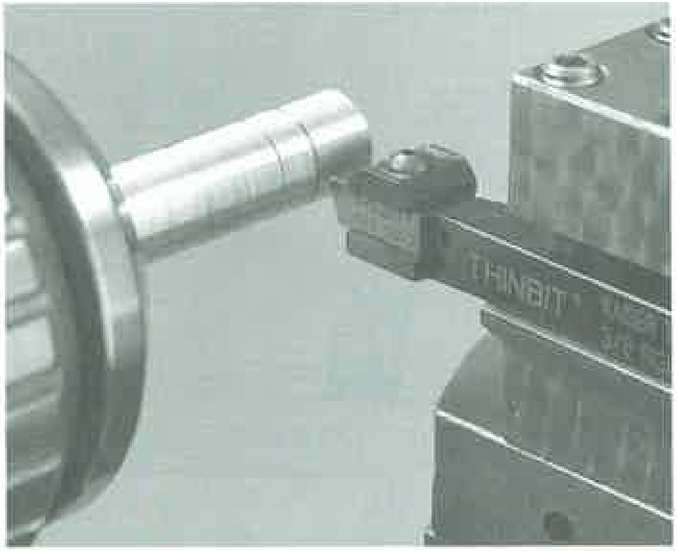


Figure 13-80. A groove or neck can be cut into a shaft with a grooving or parting tool. Shape of the groove may be square, angular (square with sloping sides), or round. (Kaiser Tool Co., Inc.)

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.

The common types of chucks are:

- 3-jaw universal.
- 4-jaw independent.
- Jacobs.
- Draw-in collet.

13.10.1 3-jaw universal chuck

The ***3-jaw universal chuck*** is designed so that all jaws operate at the same time, **Figure 13-81**. It will automatically center round or hexagonal stock.

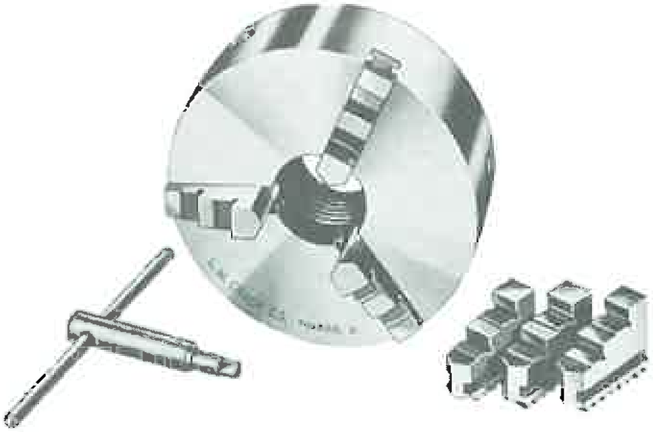


Figure 13-81. The 3-jaw universal chuck automatically centers round or hexagonal stock. (L-W Chuck Co.)

Two sets of jaws are supplied with each universal chuck. One set is used to hold large-diameter work; the other set is for small-diameter work, **Figure 13-82.**

The jaws in each set are numbered 1, 2, and 3, as are the slots in which they are fitted. The ***jaw number*** must correspond with the ***slot number*** if the work is to be centered. 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!

13.10.2 Installing chuck jaws

Before installing jaws, clean the jaws, jaw slots, and ***scroll*** (spiral thread seen 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. Now, 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.

Remove the chuck key when you finish using it; if left in the chuck, it could become a dangerous missile when the lathe is turned on. 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.

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

13.10.3 4-jaw independent chuck

Each of the jaws of ***4-jaw independent chuck***, **Figure 13-83,** operates individually, instead of being

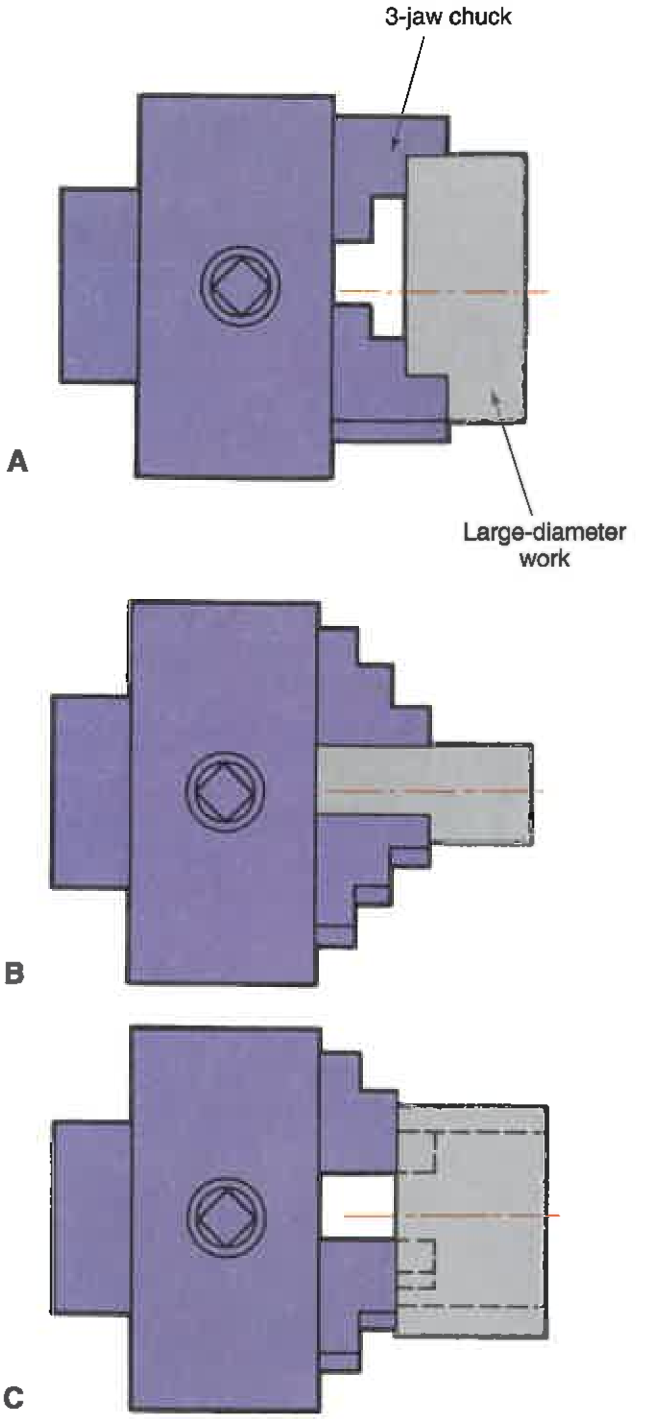


***Figure** 13-82. Chuck jaws. A—One of the sets of jaws supplied with a 3-jaw universal chuck is used to mount large-diameter work. B—Holding work using the set of jaws supplied for smaller-size workpieces. C—Another method of mounting work in the chuck.*

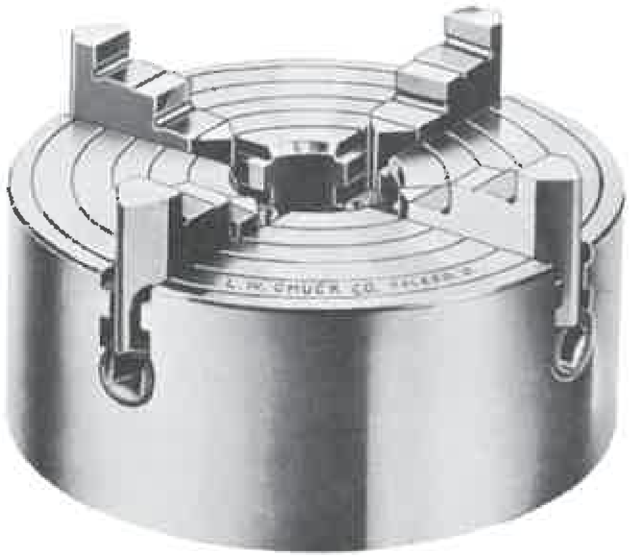


***Figure** 13-83. A 4-jaw independent chuck. The jaws on this type chuck are reversible. (L-W Chuck Co.)*

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 the 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 smaller-diameter work when reversed, **Figure 13-84.**

Unlike the 3-jaw chuck, the 4-jaw type is not self-centering. The most accurate way to center round work in this type 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 13-85**. 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.

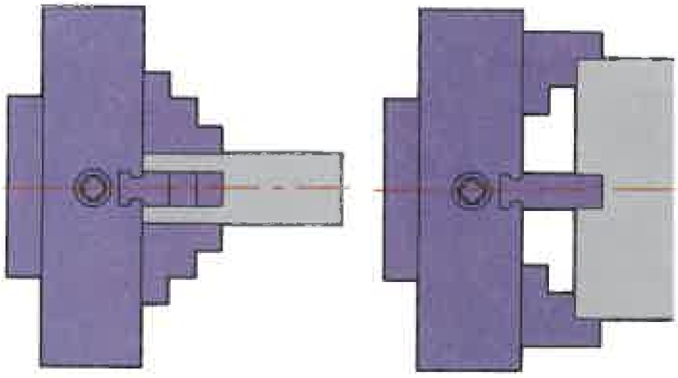


***Figure** 13-84. Reversing feature of jaws in a 4-jaw independent chuck makes it possible to turn work having extreme differences in diameter without difficulty.*

Another centering method uses chalk, **Figure 13-86**. Rotate the work while bringing the chalk into contact with it. 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.

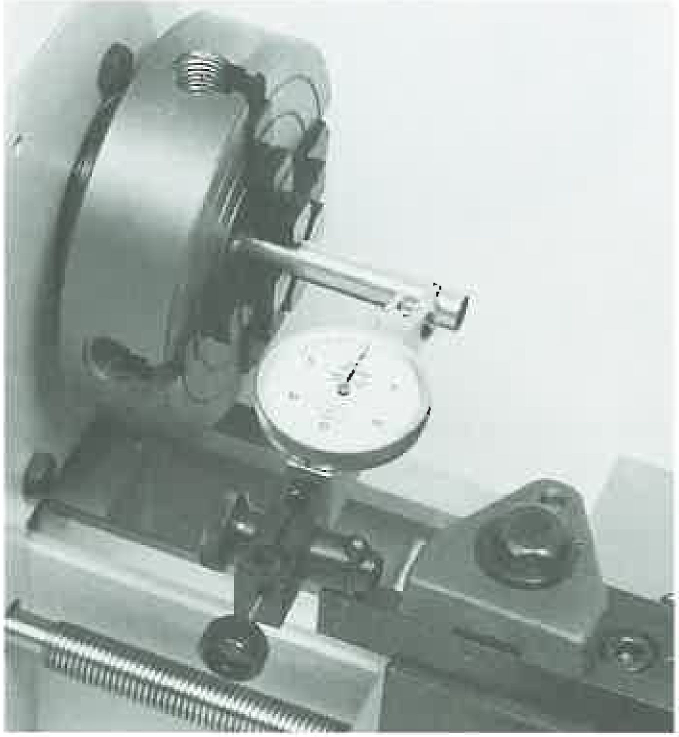


Figure 13-85. *Centering work in a 4-jaw chuck using a dial indicator. The machine shown is a small modelmaker's lathe.*

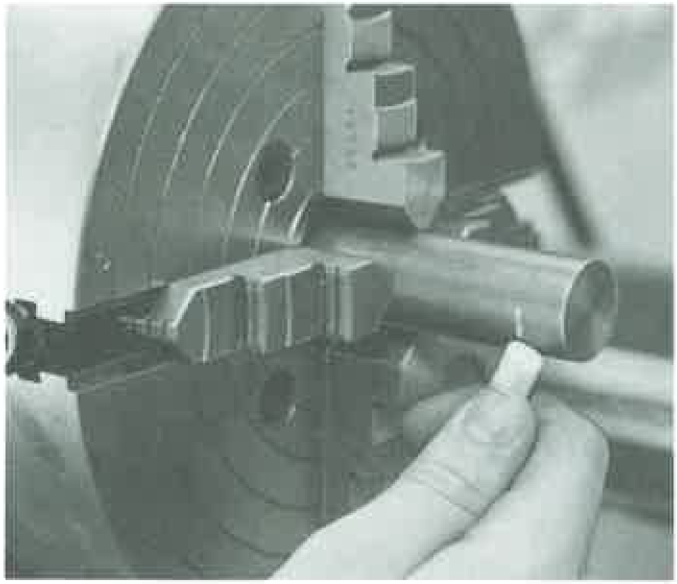


Figure 13-86. *Using chalk to center work in a 4-jaw chuck. The chalk mark indicates the "high point." Gradual adjustments, rather than large movements, are most effective in centering work.*

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

13.10.4 Jacobs chuck

When turning small-diameter work, such as screws or pins, the ***Jacobs chuck*** can be utilized. This chuck, **Figure 13-87,** is better suited for such work than the larger universal or independent chuck.

A ***standard Jacobs chuck*** is normally 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 they are fitted together.

A ***headstock spindle Jacobs chuck*** is similar to the standard Jacobs chuck, but is designed to fit directly onto a threaded spindle nose, **Figure 13-88.** The chuck has the advantage of not interfering with the compound rest, making it possible to work very close to the chuck.

13.10.5 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 13-89.**

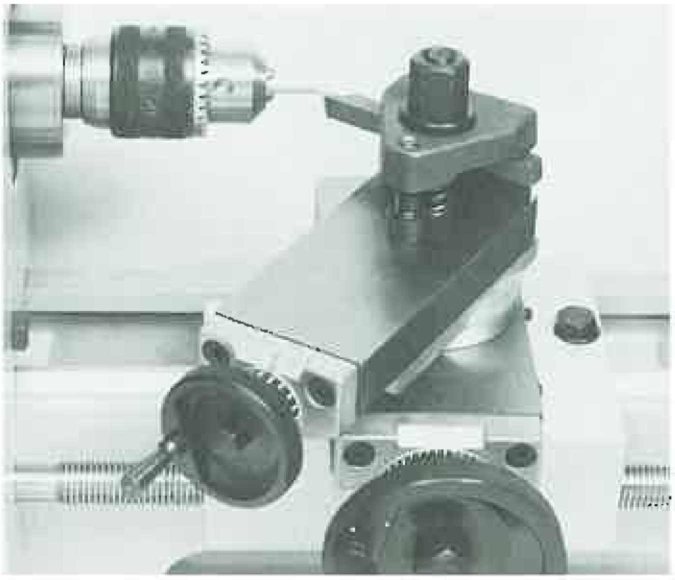


Figure 13-87. *Turning small-diameter work in a Jacobs chuck.*

Figure 13-88. Jacobs chuck that can be mounted on a lathe with a threaded spindle nose. (Jacobs Mfg. Co.)

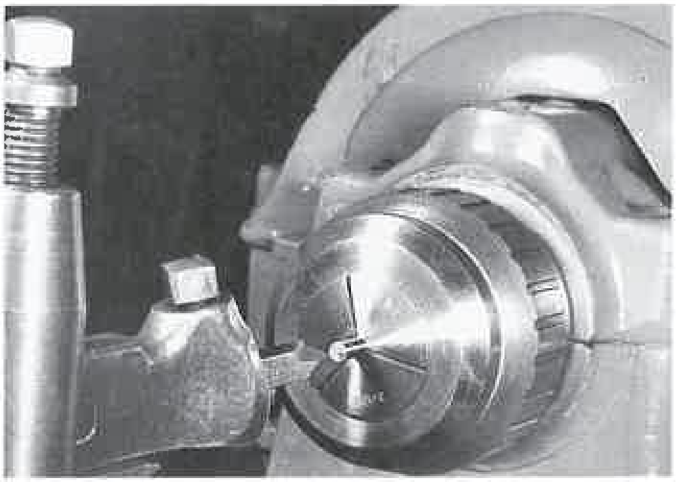


Figure 13-89. A draw-in type collet chuck.

Collets are accurately made sleeves with one end threaded and the other split into three even sections. The slots are cut slightly more than half the length of the collet and permit the jaws to spring in and clamp the work, **Figure 13-90.**

Figure 13-90. A variety of collets is necessary to clamp different stock sizes. (Maswerks, Inc.)

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.

A collet chuck using 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.

13.10.6 Mounting and removing chucks

If a chuck is not installed on the spindle nose correctly, its accuracy will be 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 13-91.**

With the 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. Pins on the cam-lock spindle are fitted into place and locked.

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.

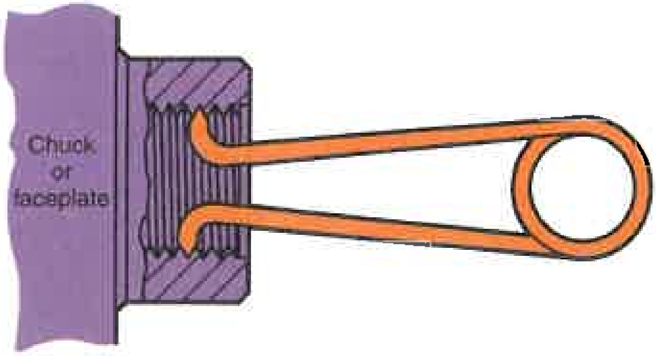


Figure 13-91. A spring cleaner is used to clean the threads in a chuck before mounting it on a threaded spindle nose.

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.

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

13.10.7 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 cradle across the ways beneath the chuck for support. Then use one of the techniques shown in **Figure 13-92.**

- 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 wood between the rear lathe ways and one of the chuck jaws. Engage the back gear and give the drive pulley a quick rearward turn.

13.10.8 Removing a chuck from other spindle noses

Little difficulty should be encountered when removing a chuck from tapered and cam lock spindle noses. 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.

Place a wooden cradle under the chuck before attempting to remove it from the spindle. Removal will be easier and hand injuries will be avoided.

13.11 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. The compound rest is pivoted 30° to the right. The toolholder is set to less than 90° to face the work, and the cutter bit is exactly on center. The carriage is then moved into position and locked to the way. See **Figure 13-93A.**

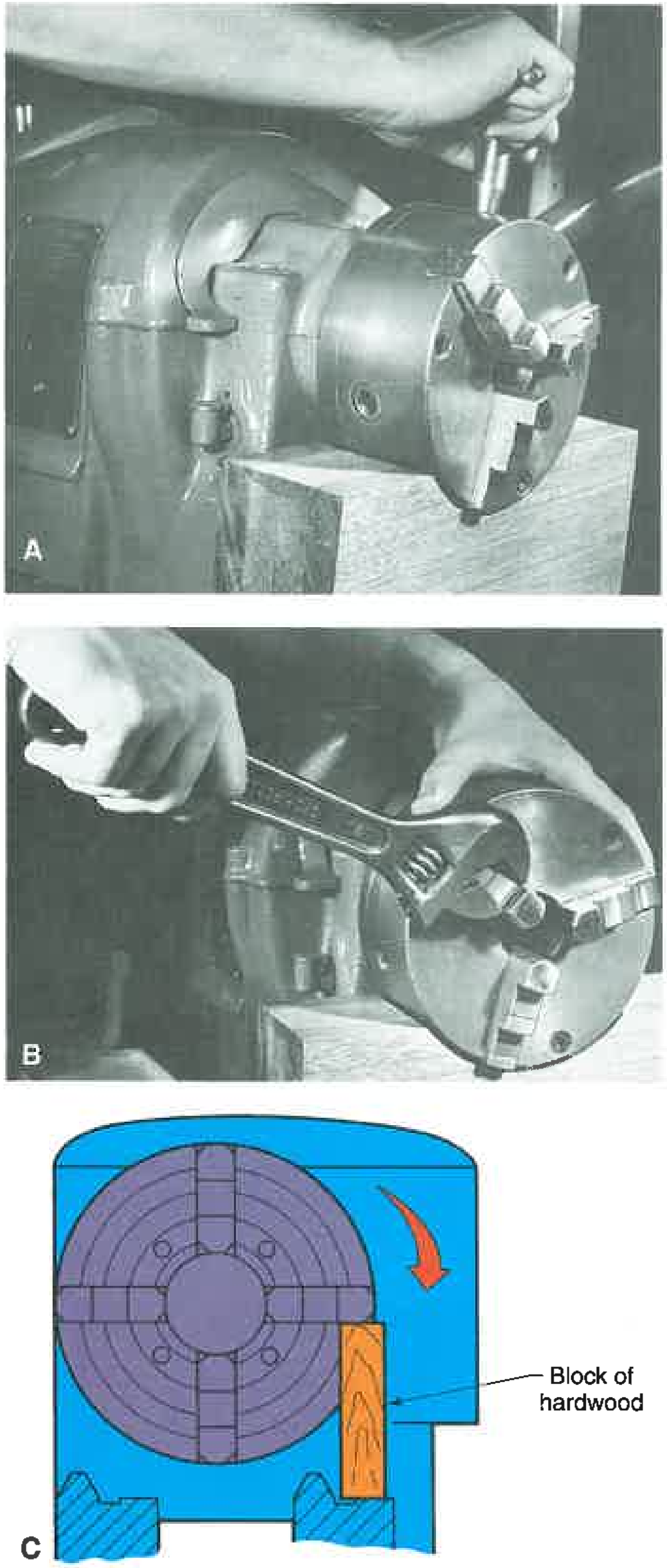


Figure 13-92. *Removing a chuck. A— Using a chuck wrench to loosen the chuck. Note the wooden cradle placed under the chuck for support. B—Fitting an adjustable wrench to one of the jaws may help you loosen a stubborn chuck. C—In truly stubborn cases, reversing the chuck against a block of wood is often used.*

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 employed.

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 13-93B**. A ***square-shoulder "nubbin"*** indicates that the cutter is below center, **Figure 13-93C**. Reposition the tool and repeat the operation if either condition is seen.

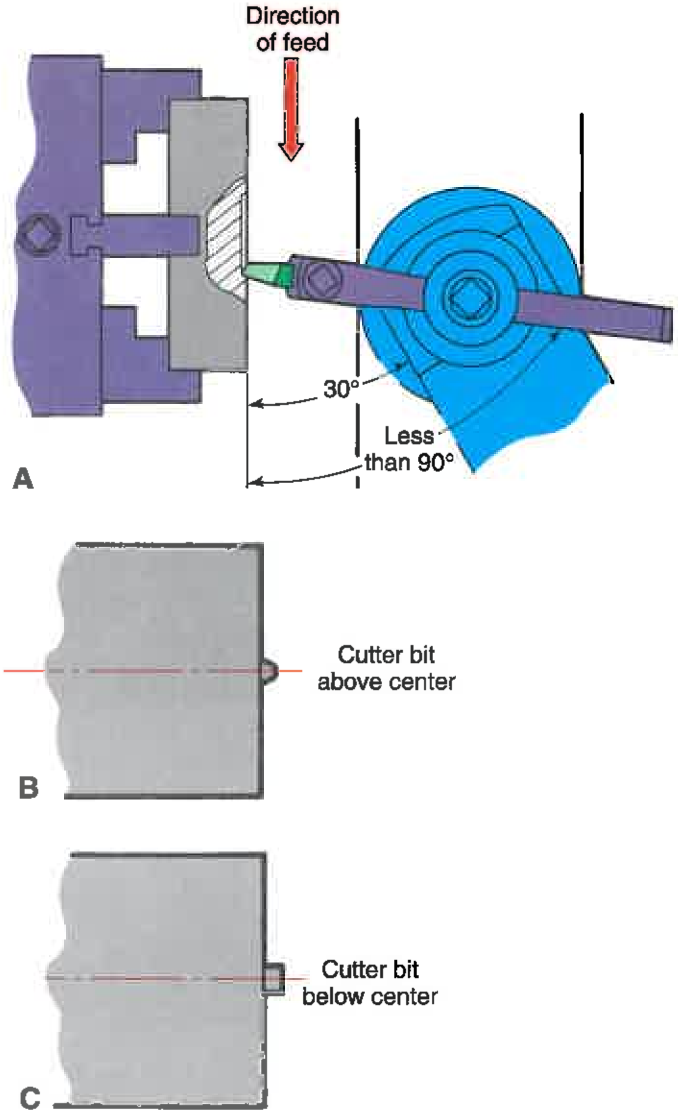


Figure 13-93. *Facing in a chuck. A—Correct tool and tool holder positions for facing. B—Rounded nubbin left by above-center cutter. C—Square-shoulder nubbin left by below-center cutter.*

13.12 PLAIN TURNING AND TURNING TO A SHOULDER

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 13-94.**

13.13 PARTING OPERATIONS

Parting is the operation of cutting off material after it has been machined, **Figure 13-95.** This is one of the more difficult operations performed on a lathe.

Cutting tools for parting or grooving are held in a straight or offset toolholder, **Figure 13-96.** 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 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.

The cutoff blade is set at *exactly* 90° to the work surface, **Figure 13-97**. 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

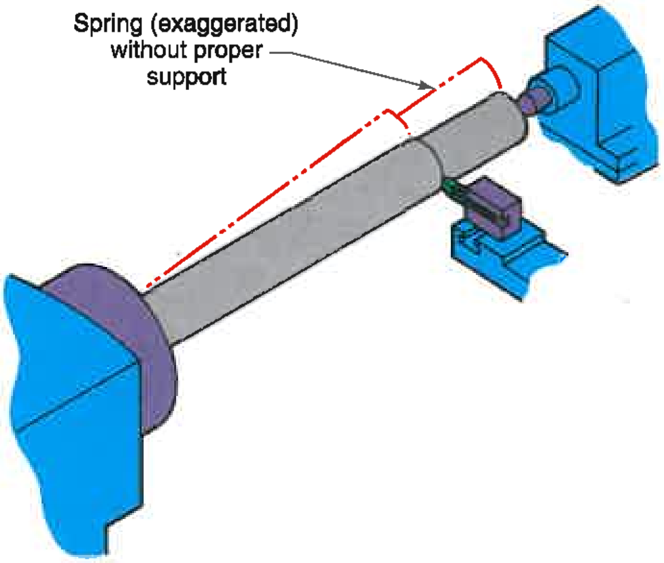


Figure 13-94. *For accurate turning, long work must be supported with tailstock center.*

Figure 13-95. *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. (Iscar Metals, Inc.)*

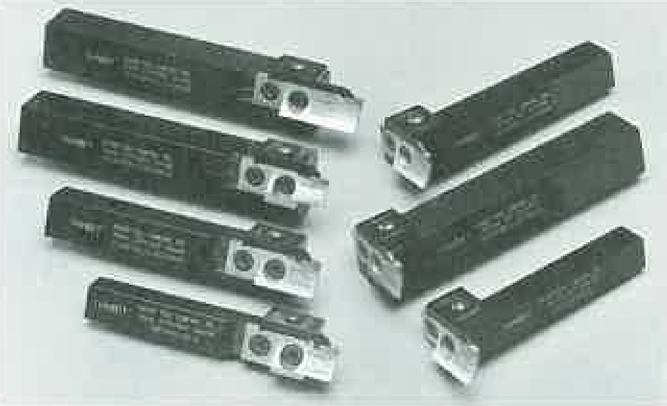


Figure 13-96. *Typical straight and offset tool holders for parting and grooving. They use replaceable carbide inserts, which are more wear resistant than conventional high speed tools. (Kaiser Tool Co., Inc.)*

work diameter is reduced, unless the center of the piece has been drilled out.

Spindle speed is about one-third that employed for conventional turning. The compound rest and cross-slide must be tightened to prevent play. 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 surface of the metal for a revolution or two, then bite suddenly. If the machine is in good condition, automatic cross-feed may be employed.

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

Never attempt to part work that is held between centers. Serious trouble will be encountered. See **Figure 13-98.**

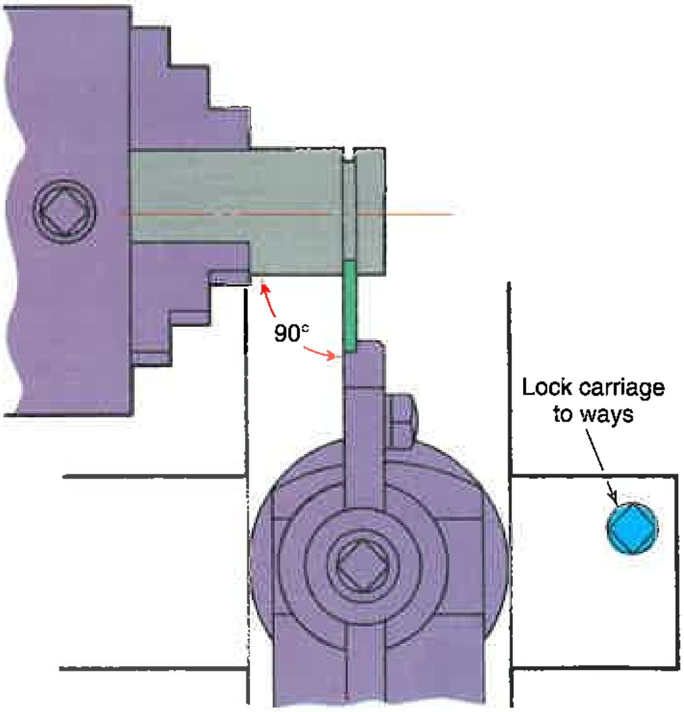


Figure 13-97. *Work is held close in chuck for the parting operation. Parting tool blade is set at a 90º angle to cut, and carriage is locked to the ways.*

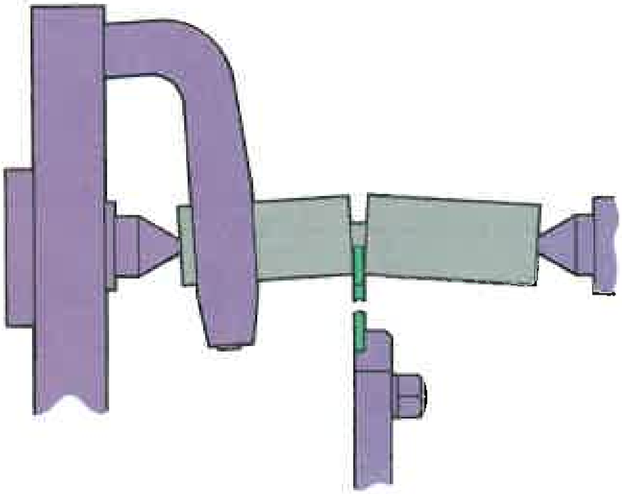


Figure 13-98. *Work cannot be parted safely while being held between centers.*

TEST YOUR KNOWLEDGE

Please do not write in the text. Write your answers on a separate sheet of paper.

1. The lathe operates on the principle of:
   a. The cutter revolving against the work.
   b. The cutting tool, being controllable, can be moved vertically across the work.
   c. The work rotating against the cutting tool, which is controllable.
   d. All of the above.
   e. None of the above.
2. The size of a lathe is determined by the _____ and the _____of the _____.
3. The largest piece that can be turned between centers is equal to:
   a. The length of the bed minus the space taken up by the headstock.
   b. The length of the bed minus the space taken up by the tailstock.
   c. The length of the bed minus the space taken up by the headstock and the tailstock.
   d. All of the above.
   e. None of the above.
4. Into which of the following categories do the various parts of the lathe fall?
   a. Driving the lathe.
   b. Holding and rotating the work.
   c. Holding, moving, and guiding the cutting tool.
   d. All of the above.
   e. None of the above.
5. Explain the purpose of ways on the lathe bed.
6. Power is transmitted to the carriage through the feed mechanism to the quick-change gearbox which regulates the amount of _____ per _____.
7. The carriage supports and controls the cutting tool. Describe each of the following parts:
   a. Saddle.
   b. Cross-slide.
   c. Compound rest.
   d. Tool post.
8. Accumulated metal chips and dirt are cleaned from the lathe with a _____, *never* with _____.
9. Which of the following actions are considered dangerous when operating a lathe?
   a. Wearing eye protection.
   b. Wearing loose clothing and jewelry.
   c. Measuring with work rotating.
   d. Operating lathe with most guards in place.
   e. Using compressed air to clean machine.
10. In most lathe operations, you will be using a single-point cutting tool made of _____.
11. Cutting speeds can be increased 300% to 400% by using _____ tools.
12. What does cutting speed indicate?
13. _____ is used to indicate the distance that the cutter moves in one revolution of the work.
14. Calculate the cutting speeds for the following metals. The information furnished is sufficient to do so.
   a. Formula: $\text{rpm} = \dfrac{CS \times 4}{D}$
   b. CS = Cutting speed recommended for material being machined.
   c. D = Diameter of work in inches.

   Problem A: 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.

   Problem B: 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.
15. Calculating the cutting speed for metric-size material requires a slightly different formula.
   a. Formula: $\text{rpm} = \dfrac{CS \times 1000}{D \times 3}$
   b. CS = Cutting speed recommended for particular material being machined (steel, aluminum, etc.) in meters per minute (mpm).
   c. D = Diameter of work in millimeters (mm).

   Problem: What spindle speed is required to finish-turn 200 mm diameter aluminum alloy? Recommended cutting speed for the material is 300 mpm.

16. Most work is machined while supported by one of four methods. List them.
17. Sketch a correctly drilled center hole.
18. A tapered piece will result, when the work is turned between centers, if the centers are not aligned. Approximate alignment can be determined by two methods. What are they?
19. Describe one method for checking center alignment if close tolerance work is to be done between centers.
20. It is often necessary to turn to a shoulder or to a point where the diameters of the work change. One of four types of shoulders will be specified. Make a sketch of each.
    a. Square shoulder.
    b. Angular shoulder.
    c. Filleted shoulder.
    d. Undercut shoulder.
21. What are the four types of lathe chucks most commonly used? Describe the characteristics of each.
22. When using the parting tool, the spindle speed of the machine is about _____ the speed used for conventional turning.
23. Why is a concave rake ground on top of the cutter when used for parting operations?
24. There are many safety precautions that must be observed when operating a lathe. List what *you* consider the five most important.

Chapter 14

Cutting Tapers and Screw Threads on the Lathe

LEARNING OBJECTIVES

After studying this chapter, you will be able to:

- ○ Describe how a taper is turned on a lathe.
- ○ Calculate tailstock setover for turning a taper.
- ○ Safely set up and operate a lathe for taper turning.
- ○ Describe the various forms of screw threads.
- ○ Cut screw threads on a lathe.

IMPORTANT TERMS

external threads
internal threads
major diameter
minor diameter
offset tailstock method
pitch diameter
setover
taper attachment
thread cutting stop
three-wire method of measuring threads

14.1 TAPER TURNING

A section of material is considered to be ***tapered*** when it increases or decreases in diameter at a uniform rate, **Figure 14-1**. A cone is an example of a taper. The "wedging" action of a taper makes it ideal as a means for driving drills, milling arbors, end mills, and centers. In addition, it can be assembled and disassembled easily, and will automatically align itself in a similarly tapered hole each time. Taper can be stated in taper per inch, taper per foot, degrees, millimeters per 25 mm of length, or as a ratio, **Figure 14-2**.

There are five principal methods of machining tapers on a lathe. Each has its advantages and disadvantages. The five methods are listed in **Figure 14-3.**

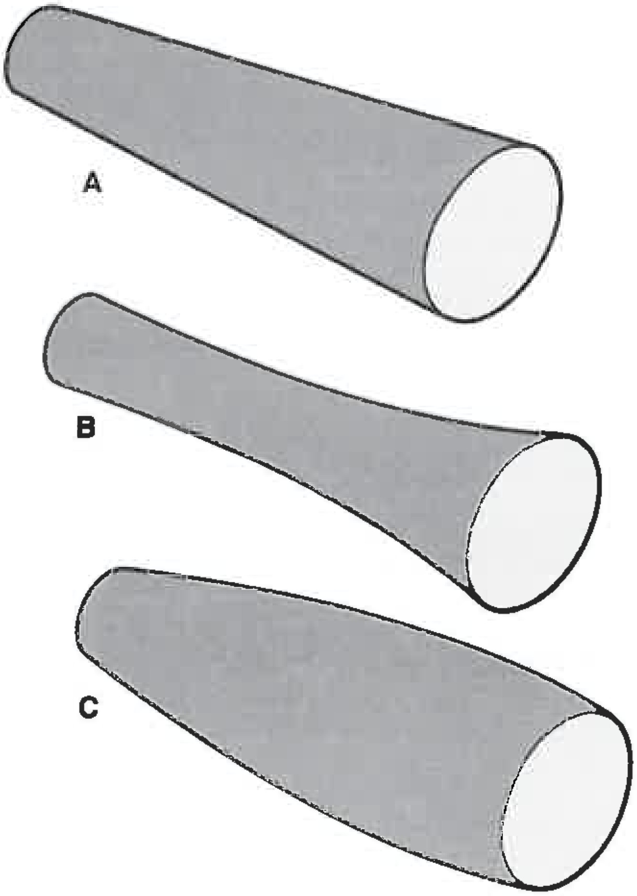


Figure 14-1*. Taper. A—The diameter of a taper increases or decreases at a uniform rate. B and C—These pieces are "bell shaped," rather than tapered.*

14.1.1 Taper turning with compound rest

The compound rest method of turning a taper is the easiest. Either internal or external tapers can be machined, as shown in **Figure 14-4**.

Taper length is limited, however, by the movement of the compound rest. Because the compound rest base is graduated in degrees, **Figure 14-5**, the taper must be converted to degrees. A conversion table may be used. See **Figure 14-6.**

A careful study of the print will show whether the angle given is from center, or is the included angle. **Figure 14-7** shows the difference in methods of measuring angles. If an ***included angle*** is given, it must be divided by two to obtain the ***angle from the centerline.***

With the lathe's centerline representing 0°, pivot the compound rest to the desired angle and lock it in position. It is the usual practice to turn a taper from the smaller diameter to the larger diameter. Refer to **Figure 14-4A.**

As will be the case when turning all tapers, the cutting tool must be set on *exact center*. A toolholder that will provide ample clearance should be selected.

To machine a taper, bring the cutting tool into position with the work and lock the carriage to prevent it from shifting during the turning operation.

Since there is no power feed for the compound rest, the cutting tool must be fed evenly with both

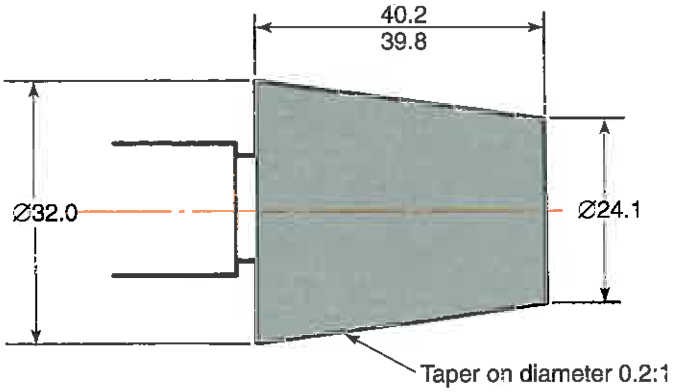


Figure 14-2*. Taper may be stated as a ratio (0.2:1, in the example above), taper per inch, taper per foot, degrees, or in millimeters per 25 mm.*

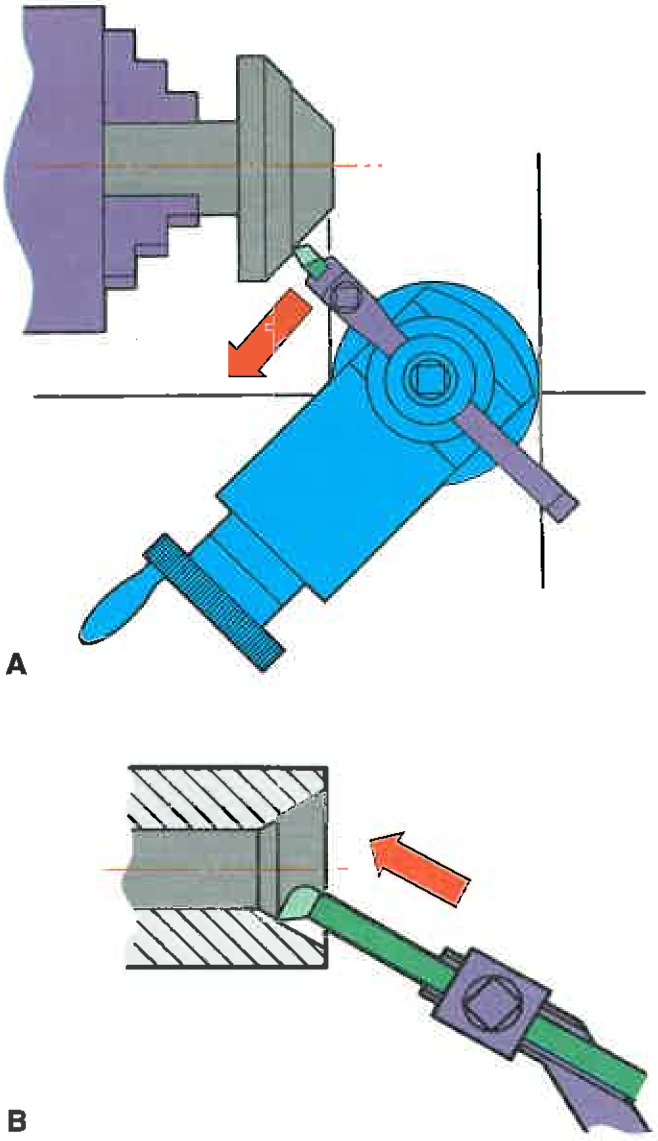


Figure 14-4*. Cutting tapers using the compound rest. A—External taper. Note that the cut is being made from small diameter to large diameter. B—Internal taper being turned with the compound rest.*

Ways of Machining Tapers		
Method	**Advantages and disadvantages**	**Information needed**
1. Compound	Length of taper limited. Will cut external and internal taper.	Must know the taper angle.
2. Offset tailstock	External taper only. Must work between centers.	Taper per inch or taper per foot.
3. Taper attachment	Best method to use.	Angle or taper per inch or foot.
4. Tool bit	Very short taper.	Taper angle.
5. Reamer	Internal only.	Taper number.

Figure 14-3*. Methods by which tapers can be turned on a lathe.*

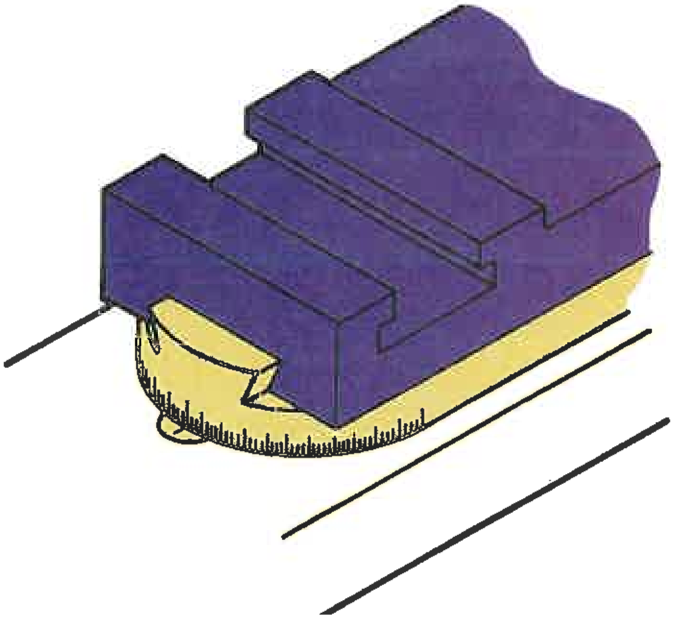


Figure 14-5. *Base of the compound rest is marked in degrees to aid in precise positioning.*

Taper per Foot with Corresponding Angles		
Taper per foot	Included angle	Angle with centerline
1/16	0° 17′ 53″	0° 8′ 57″
1/8	0° 35′ 47″	0° 17′ 54″
3/16	0° 53′ 44″	0° 26′ 52″
1/4	1° 11′ 38″	0° 35′ 49″
5/16	1° 29′ 31″	0° 44′ 46″
3/8	1° 47′ 25″	0° 53′ 42″
7/16	2° 5′ 18″	1° 2′ 39″
1/2	2° 23′ 12″	1° 11′ 36″
9/16	2° 41′ 7″	1° 20′ 34″
5/8	2° 58′ 3″	1° 29′ 31″
11/16	3° 16′ 56″	1° 38′ 28″
3/4	3° 34′ 48″	1° 47′ 24″
13/16	3° 52′ 42″	1° 56′ 21″
7/8	4° 10′ 32″	2° 5′ 16″
15/16	4° 28′ 26″	2° 14′ 13″
1	4° 46′ 19″	2° 23′ 10″

Figure 14-6. *You can use this table to convert taper per foot into corresponding angles for adjustment of the compound rest.*

hands to achieve a smooth finish. The entire cut must be made without stopping the cutting tool. The compound rest is moved back to the starting point and positioned with the cross-slide for the next cut.

When tapers are cut with a compound rest, the work can be mounted between centers or held in a chuck. A suitable boring bar is needed when machining internal tapers. Some internal tapers are finished to size with a taper reamer.

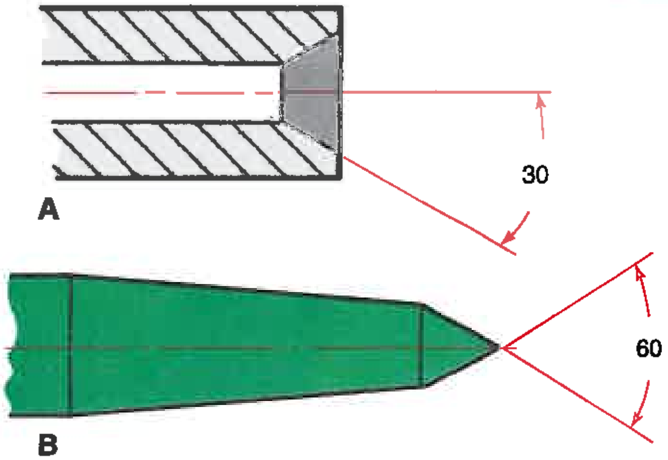


Figure 14-7. *The two methods used to measure angles. A—Angle measured from the centerline of the workpiece. B—Measurement of the included angle.*

14.1.2 Taper turning by offset tailstock method

The *offset tailstock method,* also known as the *tailstock setover method,* is also employed for taper turning, **Figure 14-8.** Jobs that can be turned between centers may be taper turned by this technique. Only *external* tapers can be machined in this way, however.

Most lathe tailstocks consist of two parts, which permits the upper portion to be shifted off center, **Figure 14-9.** This movement, referred to as *setover,* is accomplished by loosening the anchor bolt that locks the tailstock to the ways, then making the proper adjustments with screws on the tailstock. After the setover has been made, the screws are drawn up *snug,* but not tight.

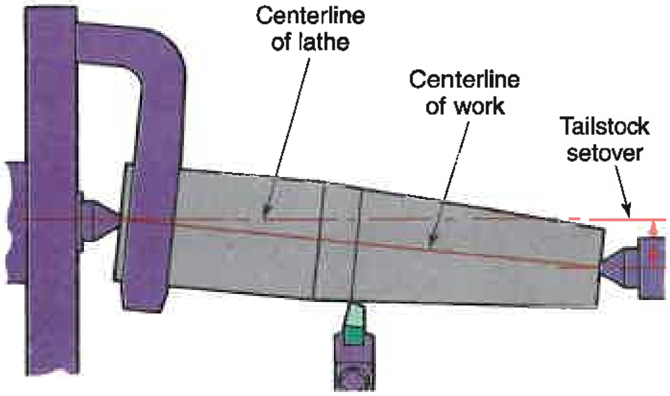


Figure 14-8. *Machining a taper using the offset tailstock method.*

Figure 14-9. The tailstock is usually constructed in two parts. This allows the section mounting the center to be shifted relative to the lathe's centerline. The distance off center, or setover, can be checked by observing the witness lines. (Rockwell International)

14.2 CALCULATING TAILSTOCK SETOVER

Taper turning by this technique is *not* a precise method and requires some "trial and error" adjustments to produce an accurate tapered section. The approximate setover can be calculated when certain basic information is known.

Offset must be calculated for each job, because the length of the piece plays an important part in the calculations. When lengths of the pieces vary, different tapers will be produced with the same tailstock offset, **Figure 14-10**.

The following terms are used with calculating tailstock setover. See **Figure 14-11**.

D = Diameter at large end
d = Diameter at small end
ℓ = Length of taper
L = Total length of piece
TPI = Taper per inch
TPF = Taper per foot

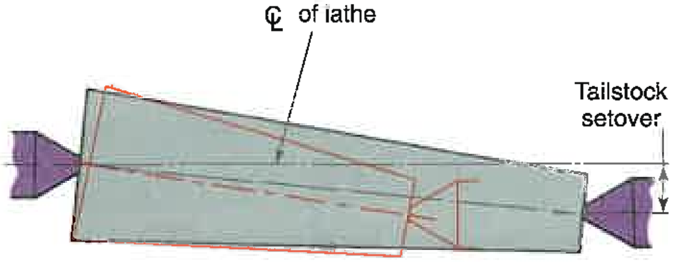


Figure 14-10. Length of work causes taper to vary even though tailstock offset remains the same.

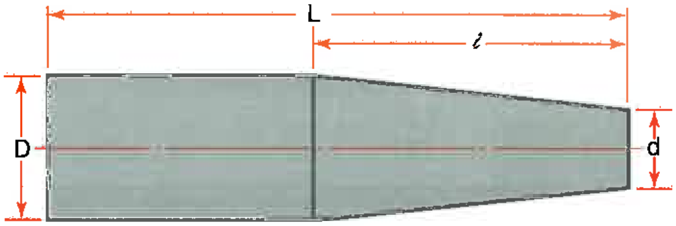


Figure 14-11. Basic taper information. D = diameter at large end of taper; d = diameter at small end of taper; ℓ = length of taper; L = total length of piece.

14.2.1 Calculating setover when taper per inch is known

Information needed:
TPI = Taper per inch
L = Total length of piece

Formula used: Offset $= \dfrac{\text{L} \times \text{TPI}}{2}$

Example: What will be the tailstock setover for the following job?

$$\begin{aligned} \text{Taper per inch} &= 0.0125 \\ \text{Total length of piece} &= 8.000 \\ \text{Offset} &= \frac{\text{L} \times \text{TPI}}{2} \\ &= \frac{8.000 \times 0.125}{2} \\ &= 0.500'' \end{aligned}$$

Note: The same procedure would be followed when using metric units. However, all dimensions would be in millimeters.

14.2.2 Calculating setover when taper per foot is known

When taper per foot (TPF) is known, it must be converted to taper per inch (TPI). The following formula takes this into account:

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

14.2.3 Calculating setover when dimensions of tapered sections are known but TPI or TPF is not given

Plans often do not specify TPI or TPF, but do give other pertinent information. Calculations will be easier if all fractions are converted to decimals. All dimensions must either be in inches or in millimeters.

Information needed:

D = Diameter at large end
d = Diameter at small end
ℓ = Length of taper
L = Total length of piece

Formula used: $\text{Offset} = \frac{L \times (D\text{–}d)}{2\ell}$

Example: Calculate the tailstock setover for the following job.

$$D = 1.250''$$
$$d = 0.875''$$
$$\ell = 3.000''$$
$$L = 9.000''$$

$$\begin{aligned} \text{Offset} &= \frac{L \times (D\text{–}d)}{2\ell} \\ &= \frac{9.000 \times (1.250\text{–}0.875)}{2 \times 3.000} \\ &= \frac{9.000 \times 0.375}{6} \\ &= 0.562'' \end{aligned}$$

14.2.4 Calculating setover when taper is given in degrees

The space available in this text does not permit the introduction of basic trigonometry, which is necessary to make these calculations. However, any good machinist's handbook will provide this information. At least one such book should be part of every machinist's toolbox.

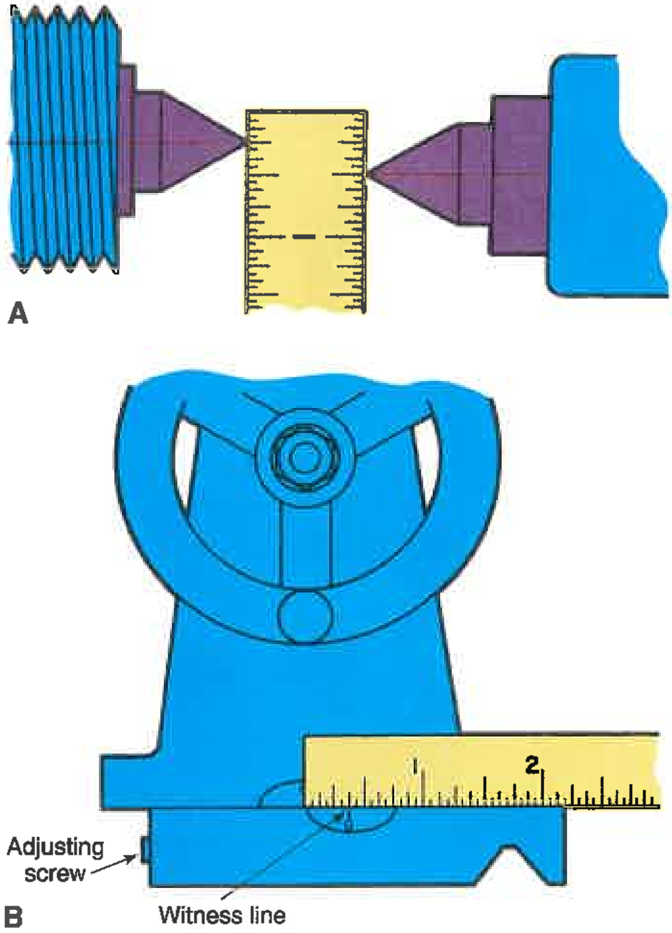


Figure 14-12. Measuring setover. A—Approximate tailstock setover can be determined by measuring distance between center points. B—Approximate setover can also be determined by measuring distance between witness lines on the tailstock.

14.3 MEASURING TAILSTOCK SETOVER

When an ample tolerance is allowed, (±0.015" or 0.05 mm), the setover can be measured with a steel rule. There are two ways to measure:

- Place a rule that has graduations on both edges between the center points, **Figure 14-12A**. Measure the distance between the center points.
- Measure the distance between the two witness lines on the tailstock base, **Figure 14-12B**.

Accurate work requires care in making the tailstock setover. An additional factor enters into the calculations—the distance that the center point enters the piece. Typically, 1/4" (6.5 mm) is an ample allowance; it must be subtracted from the total length of the piece.

Use the appropriate method to calculate the offset. A precise setover may be made using the *micrometer collar* on the lathe cross-slide. See **Figure 14-13.**

1. Clamp the toolholder in a reverse position in the tool post.
2. Turn the cross-slide screw back to remove all play.
3. Turn in the compound rest until the toolholder can be felt with a piece of paper between the toolholder and tailstock spindle.
4. Use the micrometer collar and turn out the cross-slide screw the distance the tailstock is to be set over.
5. Move the tailstock over until the spindle touches the paper in same manner described in Step 3.
6. Check the setting again after "snugging up" the adjusting screws.

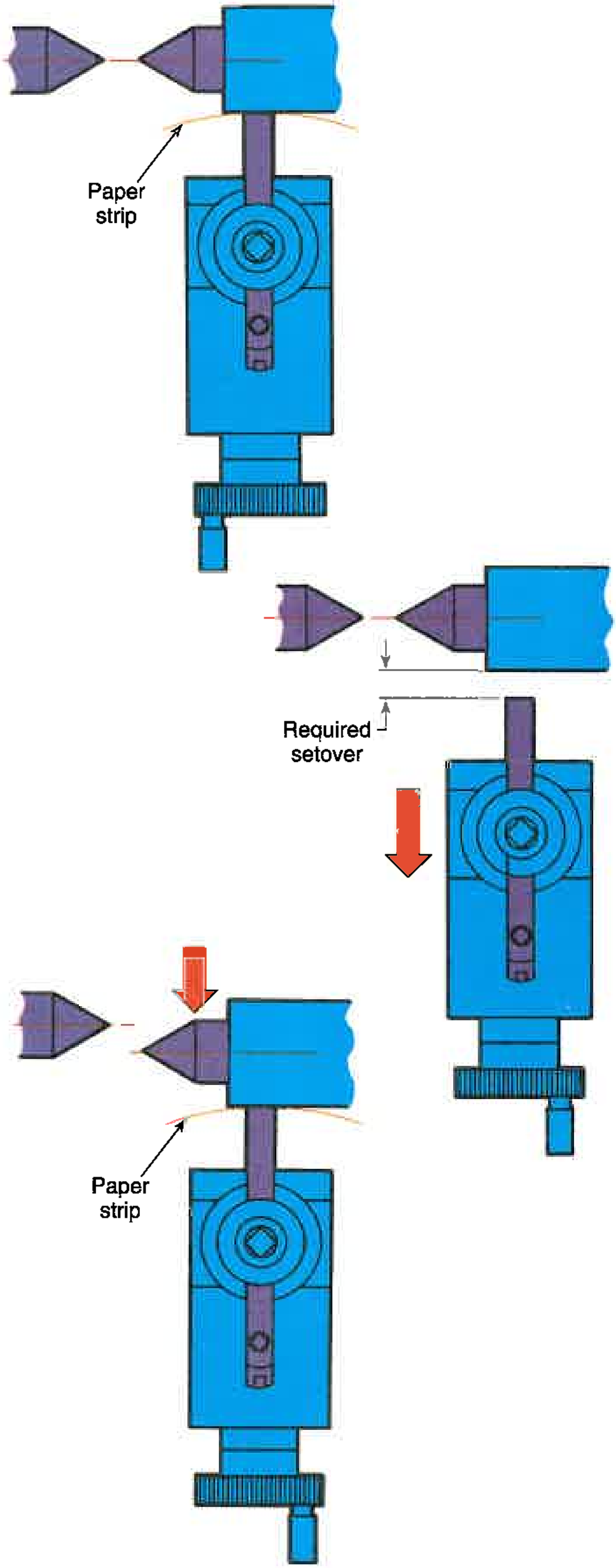


Figure 14-13. *Using the micrometer collar of the compound rest to make the setover measurement.*

In place of the toolholder and paper strip, a *dial indicator* can be employed to establish the offset. See **Figure 14-14.**

1. Mount the dial indicator in the tool post.
2. Position it with the cross-slide until the indicator reads zero when in contact with the tailstock spindle. There should be *no* "play" in the cross-slide.
3. Set the tailstock over the required distance using the dial indicator to make the measurement.
4. Recheck the reading after "snugging up" the adjusting screws. Make additional adjustments if any deviation in the indicator reading occurs.

14.4 CUTTING A TAPER

When cutting a taper, additional strain is imposed on the centers because they are out-of-line and do not bear true in the center holes. Because the pressures imposed are uneven, the work is more apt to heat up than when doing conventional turning between centers. It must be checked frequently for binding. A bell-type center drill offers some advantage in reducing strain. Some machinists prefer a center with a ball tip to produce an improved bearing surface. See **Figure 14-15.**

Make the cuts as in conventional turning. However, cutting should start at the small end of the taper.

14.4.1 Turning a taper with a taper attachment

A ***taper attachment*** is a guide that can be attached to most lathes. It is an accurate way to cut tapers and offers advantages over other methods of machining tapers.

Both internal and external tapers can be cut. This helps assure an accurate fit for mating parts. Once the attachment has been set, the taper can be machined on material of various lengths. Work can be held by any conventional means. One end of the taper attachment ***swivel bar*** is graduated in total taper in inches per foot. The other end is graduated to indicate the included angle of the taper in degrees.

The lathe does not have to be altered. The machine can be used for straight turning by locking out the taper attachment. No realignment of the lathe is necessary.

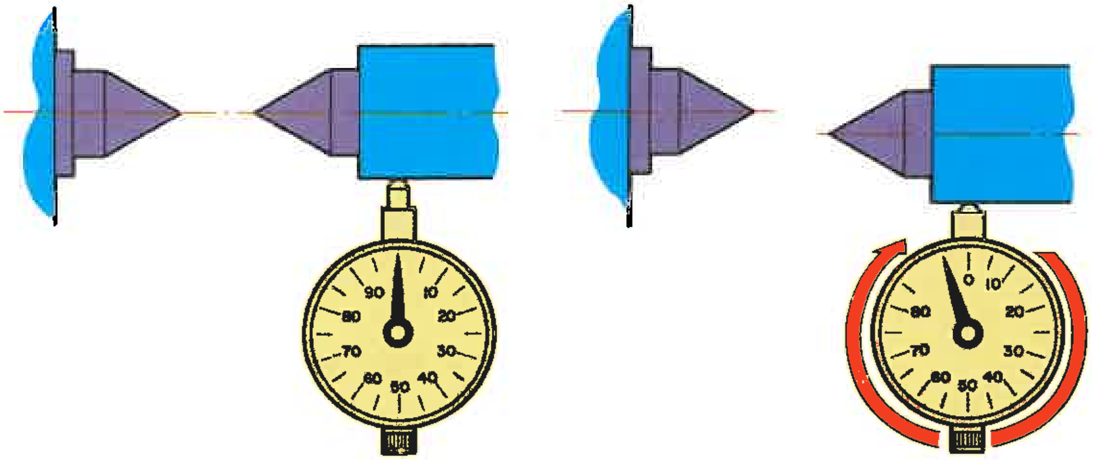


Figure 14-14. *A dial indicator can also be used to measure amount of setover.*

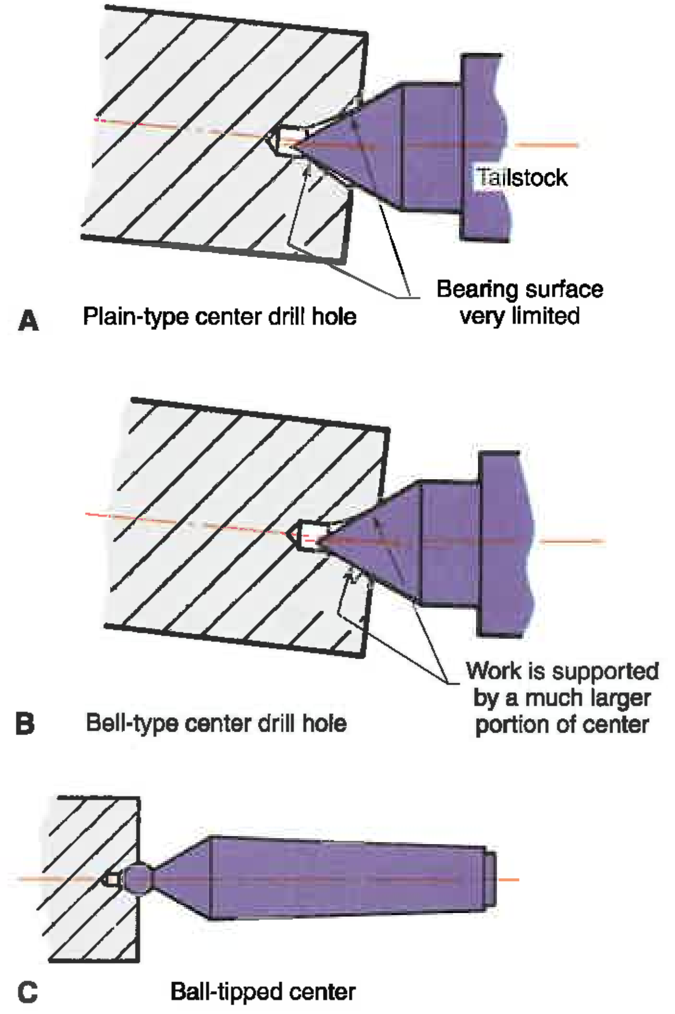


Figure 14-15. *Taper turning done by the offset tailstock method is hard on the tailstock center. A—Center point does not bear evenly in conventional center hole. B—A center hole drilled with a bell-type center drill reduces the problem by providing more bearing surface. C—A ball-tipped center lessens pressure on tail center when turning tapers.*

14.4.2 Types of taper attachments

There are two types of taper attachments, plain and telescopic. See **Figure 14-16**. The ***plain taper attachment*** requires disengaging the cross-slide screw from the cross-slide feed nut. The cutting tool is advanced by using the compound rest feed screw.

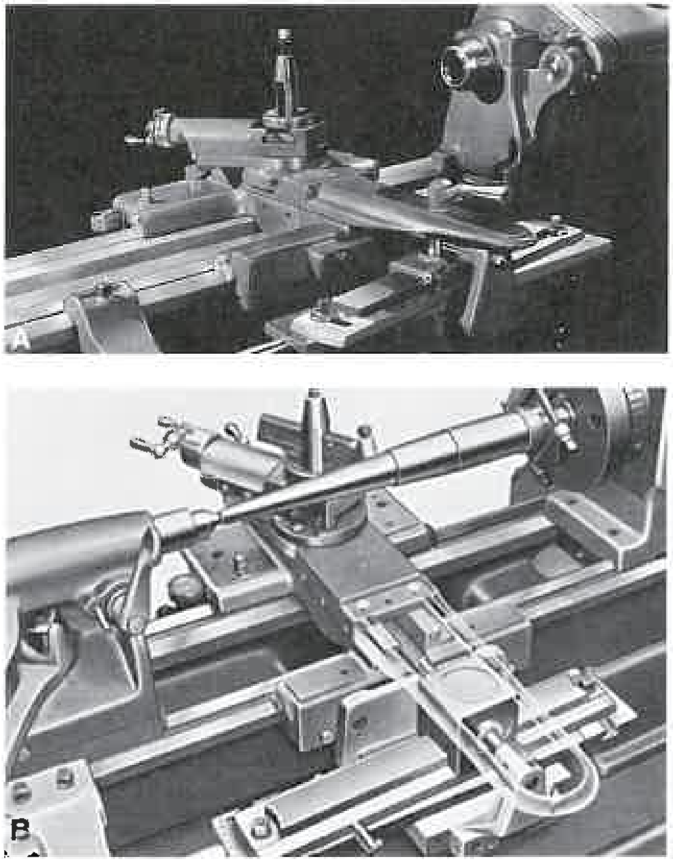


Figure 14-16. *Taper attachments. A—Plain taper attachment. (South Bend Lathe Corp.) B—Telescopic taper attachment. (Clausing Industrial, Inc.)*

The *telescopic taper attachment* is made in such a way that it is not necessary to disconnect the cross-slide feed nut. The tool can be advanced into the work with the cross-slide screw in the usual manner.

14.4.3 Setting a taper attachment

1. Study the plans and, if necessary, calculate the taper. Set the swivel bar as specified from the calculations.
2. Mount the work in the machine.
3. Slide the taper attachment unit to a position that will permit the cutting tool to travel the full length of the taper. Lock it to the ways.
4. Move the carriage to the right until the cutting tool is about 1" (25 mm) away from the end of the work. This will permit any play to be taken up before the tool starts to cut.
5. If the machine is fitted with a plain taper attachment, tighten the binding screw that engages the cross-slide feed to the attachment.
6. Oil the bearing surfaces of the taper attachment and make a trial cut. If necessary, readjust until the taper is being cut to specifications. Complete the cutting operation.

14.4.4 Turning a taper with a square-nose tool

Using a square-nose tool is a taper technique limited to the production of *short tapers*, **Figure 14-17**. The cutter bit is ground with a square nose and set to the correct angle with the protractor head and blade of a combination set.

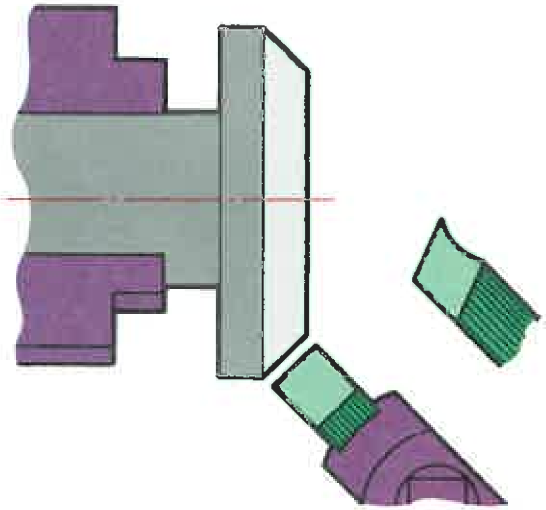


Figure 14-17. *A short taper can be turned with a square-nose tool.*

The tool is positioned on center and fed into the revolving work. "Chatter" can be minimized by running the work at a slow spindle speed. The carriage must be locked to the ways.

Before using any of the taper-turning techniques on work mounted between centers, it is very important that centers be ***"zeroed in"*** (put in perfect alignment). Then the necessary adjustments (tailstock setover, taper attachment adjustment) can be made.

14.5 MEASURING TAPERS

There are two basic methods of testing the accuracy of machined tapers. One is a ***comparison method;*** the other involves ***direct taper measurement.***

14.5.1 Measuring tapers by comparison

Taper plug gages and ***taper ring gages*** serve two purposes, **Figure 14-18.** They measure the basic diameter of the taper as well as the angle of slope. The angle is checked by applying ***bluing*** (usually a liquid known as "Prussian blue") to the machined surface or plug gage. The blued section is inserted into the mating part and slowly rotated. If the bluing rubs off *evenly*, it indicates that the taper is correct. If the bluing rubs off *unevenly*, **Figure 14-19**, the remaining material will show where the taper is incorrect and indicate what machine adjustments are needed.

Gages are also provided with notches to indicate the specified tolerance in taper diameter. The indentations show the *go* and *no-go* limits, **Figure 14-20.**

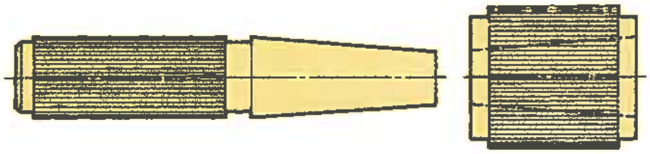


Figure 14-18. *Left—Plug gage. Right—Ring gage.*

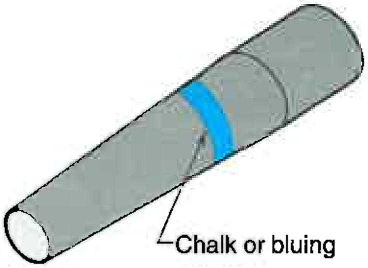


Figure 14-19. *When chalk or bluing does not rub off evenly, it indicates that taper does not fit properly and additional machine adjustments will have to be made.*

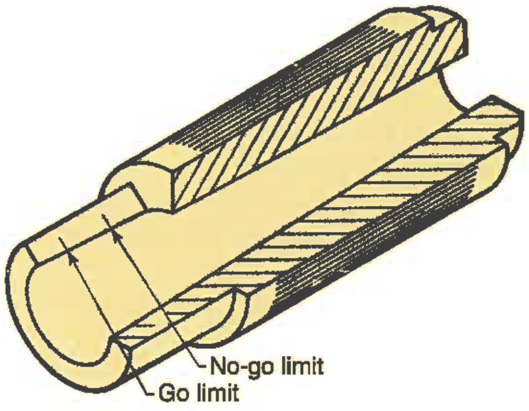


***Figure 14-20.** Typical go and no-go ring gage for measuring tapers.*

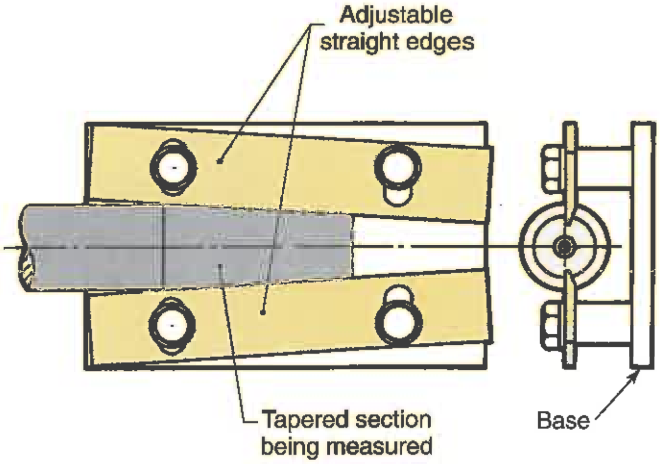


***Figure 14-21.** A taper test gage can be set for different tapers.*

14.5.2 Direct measurement of tapers

A ***taper test gage*** is sometimes employed to check taper accuracy, **Figure 14-21**. It consists of a base with two adjustable straight edges. Slots in the straight edges permit adapting the gage to check different tapers. The taper test gage is set by using two discs of known size which are located the correct distance apart.

Another technique for checking and/or measuring tapers is to set the tapered section on a surface plate. Two gage blocks or ground parallels of the same height are placed on opposite sides of the taper. Two cylindrical rods (sections of drill rod are satisfactory) of the same diameter are placed on the blocks. See **Figure 14-22**. The distance across the rods is then measured with a micrometer.

Blocks 1″, 3″, or 6″ (25, 75, or 150 mm) taller than those used for the first reading are substituted. The rods are the same diameter as those used to make the first reading. A second reading is made, **Figure 14-22B.** The taper per foot then can be determined. First, subtract to find the difference between the two measurements. Then multiply it by twelve (if the readings were made 1″ apart), by four (if they were made 3″ apart), or by two (if they were made 6″ apart).

A ***sine bar*** is a very accurately machined bar with edges that are parallel, **Figure 14-23**. The bar is used in conjunction with gage blocks and sine tables to precisely measure angles.

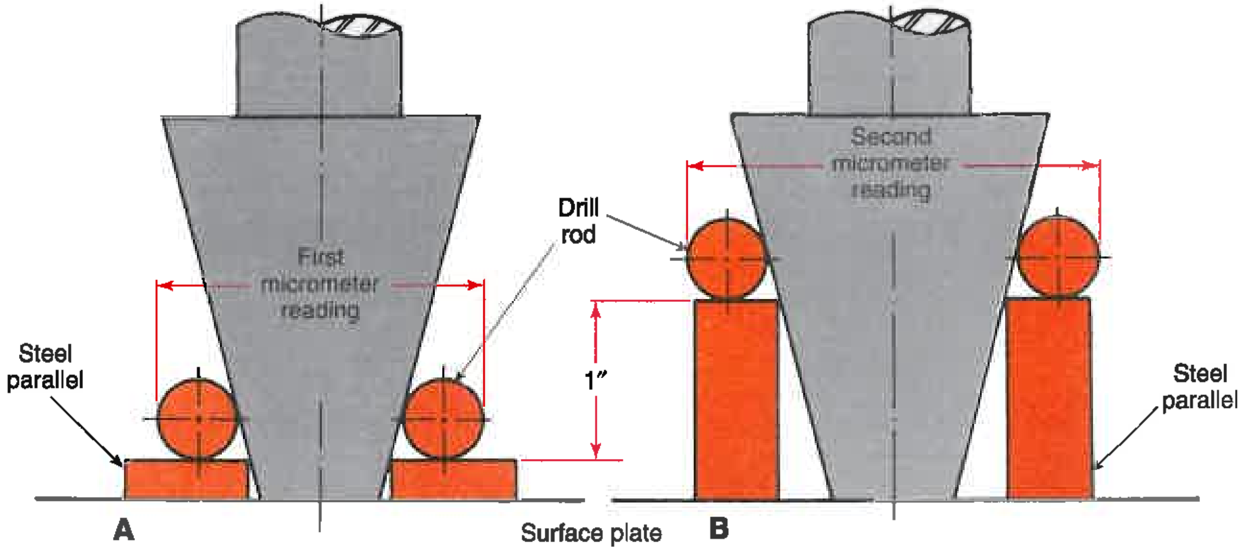


***Figure 14-22.** Measuring a taper using parallels, drill rod, micrometer, and a surface plate. A—Setup for first measurement. B—Setup for second measurement.*

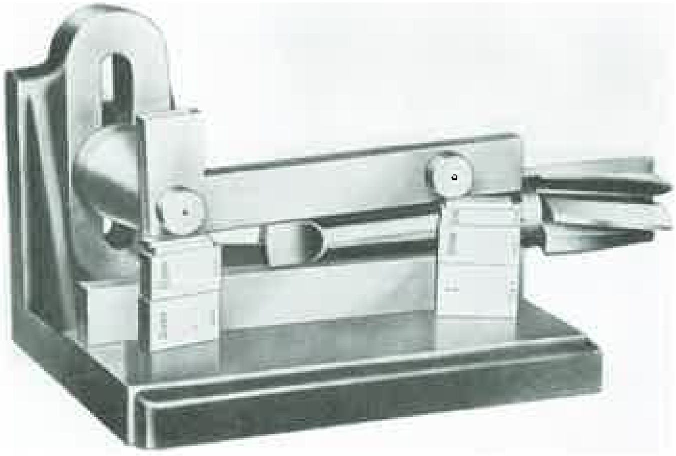


Figure 14-23. *A sine bar and precision gage blocks can also be used to measure a taper. (C.E. Johannson Co.)*

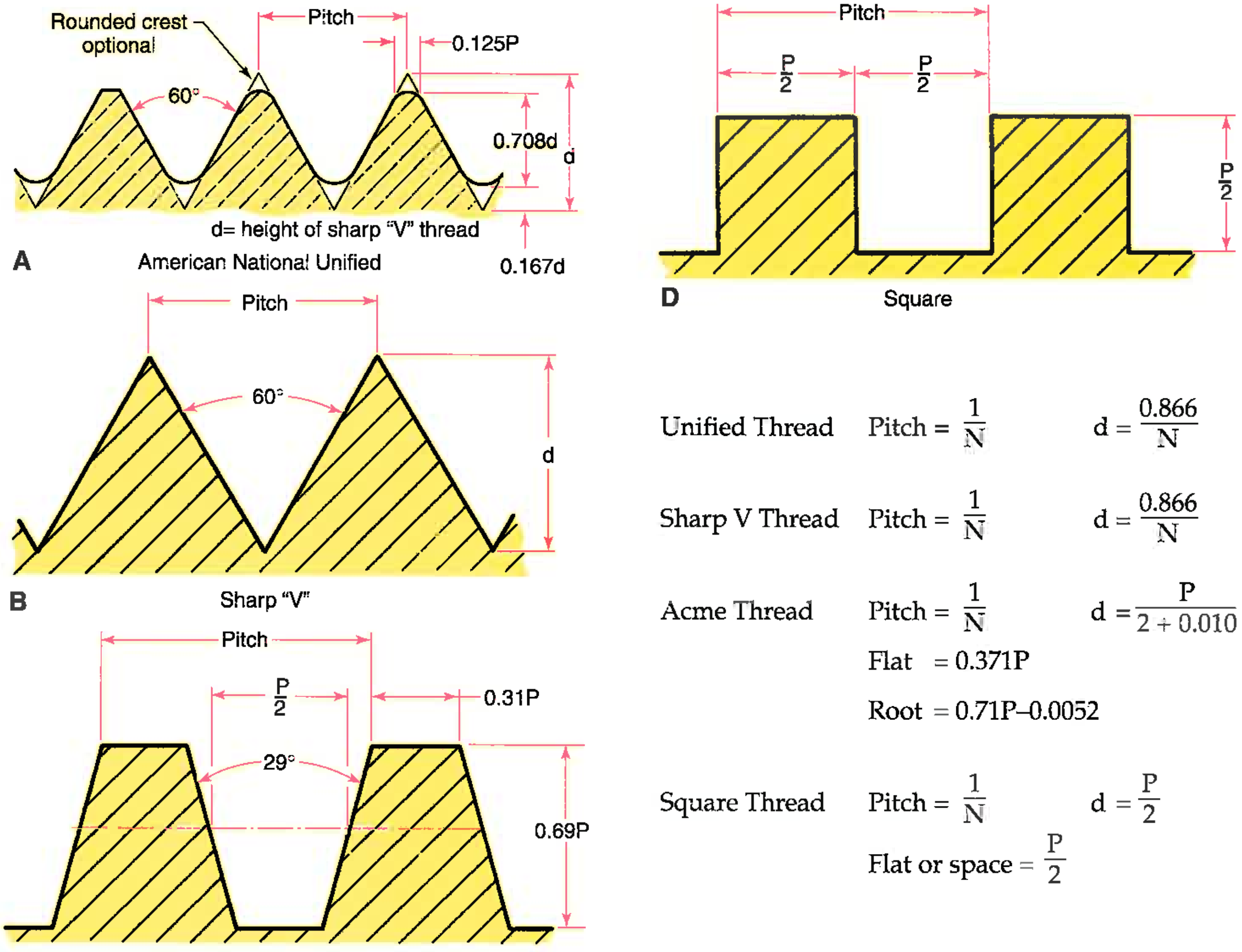
14.6 CUTTING SCREW THREADS ON THE LATHE

Screw threads are utilized for many applications. The more important are:

- Making adjustments (cross-feed on a lathe).
- Assembling parts (nuts, bolts, and screws).
- Transmitting motion (lead screw on a lathe).
- Applying pressure (clamps).
- Making measurements (micrometer).

14.6.1 Screw thread forms

The first screw threads cut by machine were square in cross-section. Since that time, many different thread forms have been developed, including American National, Unified, Sharp V, Acme, Worm threads, and others. Each thread form has a specific use and a formula for calculating its shape and size. See **Figure 14-24**. More than 75% of all threads cut in the United States are of the Unified (UN) 60° type.



Unified Thread	Pitch = $\frac{1}{N}$	$d = \frac{0.866}{N}$
Sharp V Thread	Pitch = $\frac{1}{N}$	$d = \frac{0.866}{N}$
Acme Thread	Pitch = $\frac{1}{N}$	$d = \frac{P}{2 + 0.010}$
	Flat = 0.371P	
	Root = 0.71P–0.0052	
Square Thread	Pitch = $\frac{1}{N}$	$d = \frac{P}{2}$
	Flat or space = $\frac{P}{2}$	

Figure 14-24. *Common thread forms. A—Unified thread form, interchangeable with American National Thread. B—Sharp "V" thread form. C—Acme thread form. D—Square thread form. Note: In formulas above, N = Number of threads per inch; P = Pitch; d = depth of thread.*

The following terms relate to *screw threads,* as shown in **Figure 14-25**:

- ***External threads*** are cut on the outside surface of piece.
- ***Internal threads*** are cut on the inside surface of piece.
- ***Major diameter*** is the largest diameter of the thread.
- ***Minor diameter*** is the smallest diameter of the thread.
- ***Pitch diameter*** is the diameter of an imaginary cylinder that would pass through threads at such points to make width of thread and width of the spaces at these points equal.
- ***Pitch*** is the distance from one thread point to the next thread point, measured parallel to the thread axis. Pitch of inch-based threads is equal to 1 divided by the number of threads per inch.
- ***Lead*** is the distance that a nut will travel in one complete revolution of the screw. On a single thread, the lead and pitch are the same. Multiple thread screws have been developed to secure an increase in lead without weakening the thread. See **Figure 14-26.**

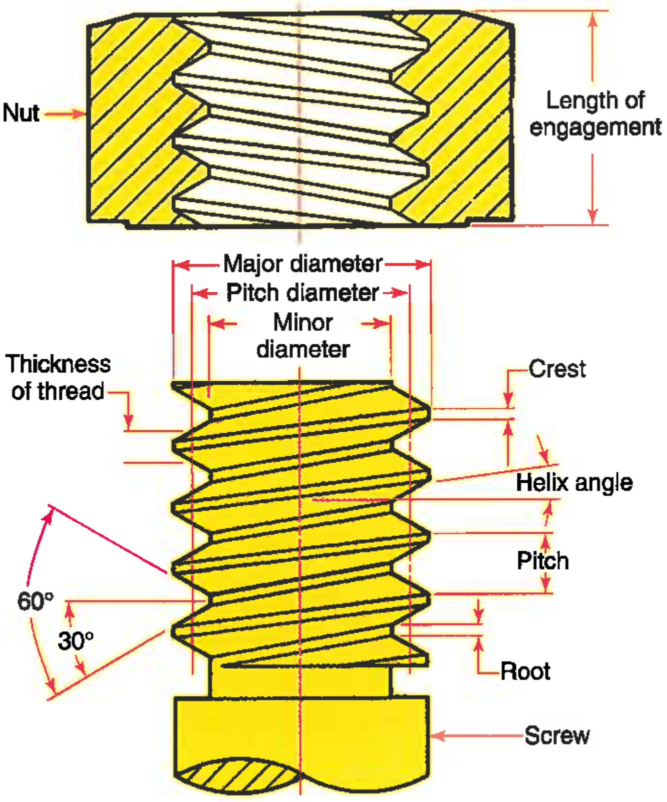


Figure 14-25. *Nomenclature of a thread.*

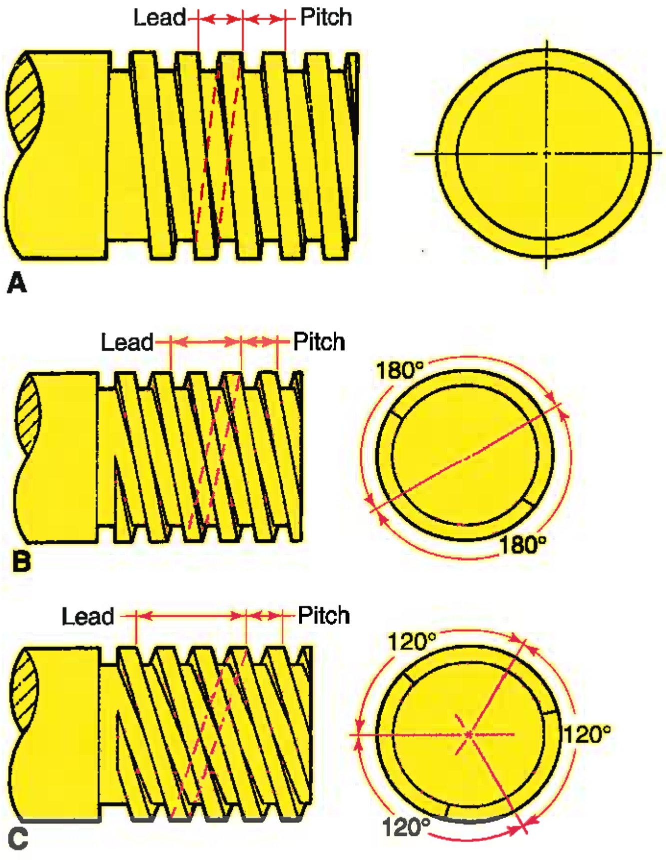


Figure 14-26. *The difference between lead and pitch. A—Single thread screw, the pitch and lead are equal. B—Double thread screw, the lead is twice the pitch. C—Triple thread screw, the lead is three times the pitch.*

14.6.2 Preparing to cut 60° threads on a lathe

Sharpen the cutting tool to the correct shape, including the proper clearance. The top is ground flat with no side or back rake, **Figure 14-27**. An oilstone is used to touch up the cutting edges and form the radius on the tip.

A *center gage* is used for grinding and setting the tool bit in position, **Figure 14-28.** The gage is often referred to as a *fishtail.*

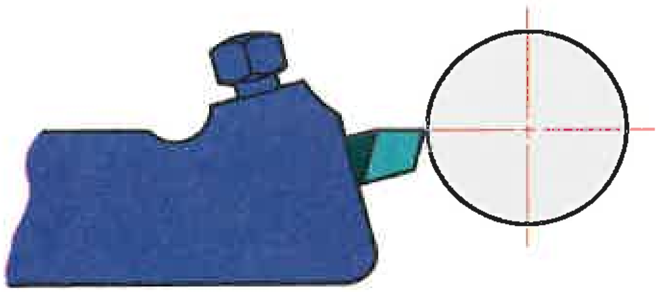


Figure 14-27. *Cutting tool positioned for cutting 60° threads. The tool is set on center as shown.*

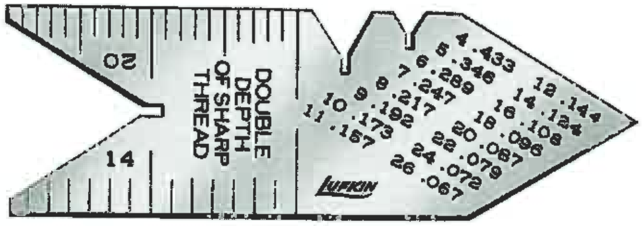


Figure 14-28. *A center gage or "fishtail." (Lufkin Rule Co.)*

The work is set up in the same manner as for straight turning. If mounted between centers, the centers must be precisely aligned; otherwise, a tapered thread will be produced. If this occurs, the thread will not be usable unless it is cut excessively deep at one end. The work must also run true with no "wobble." The tail of the lathe dog must have no play in the face plate slot.

A groove is frequently cut at the point where the thread is to terminate, **Figure 14-29**. The ***thread end groove*** is cut equal to the minor diameter of the thread and serves two purposes:

- It provides a place to stop the threading tool at the end of its cut.
- It permits a nut to be run up to the end of the thread.

Several methods may be employed to terminate a thread, as shown in **Figure 14-29**. Ordinarily, the beginner should use a groove until sufficient experience has been gained. However, the design of some parts does not permit a groove to be used. In such a case, the threads must be terminated by another method. They require perfect coordination and very rapid operation of the cross-slide to get the tool out of position at the end of the cut.

The gearbox is adjusted to cut the correct number of threads. Make apron adjustments to permit the half-nuts to be engaged. After the proper apron and gear adjustments have been made, pivot the compound rest to 29° to the right, **Figure 14-30.** Then set the threading tool in place.

It is essential that the tool be set on center with the tool axis at 90° to the centerline of the work.

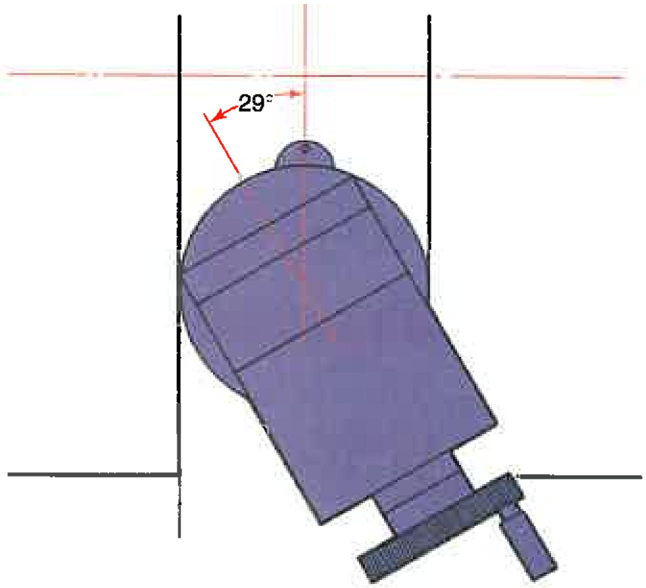


Figure 14-30. *The compound rest is set up for machining right-hand external threads.*

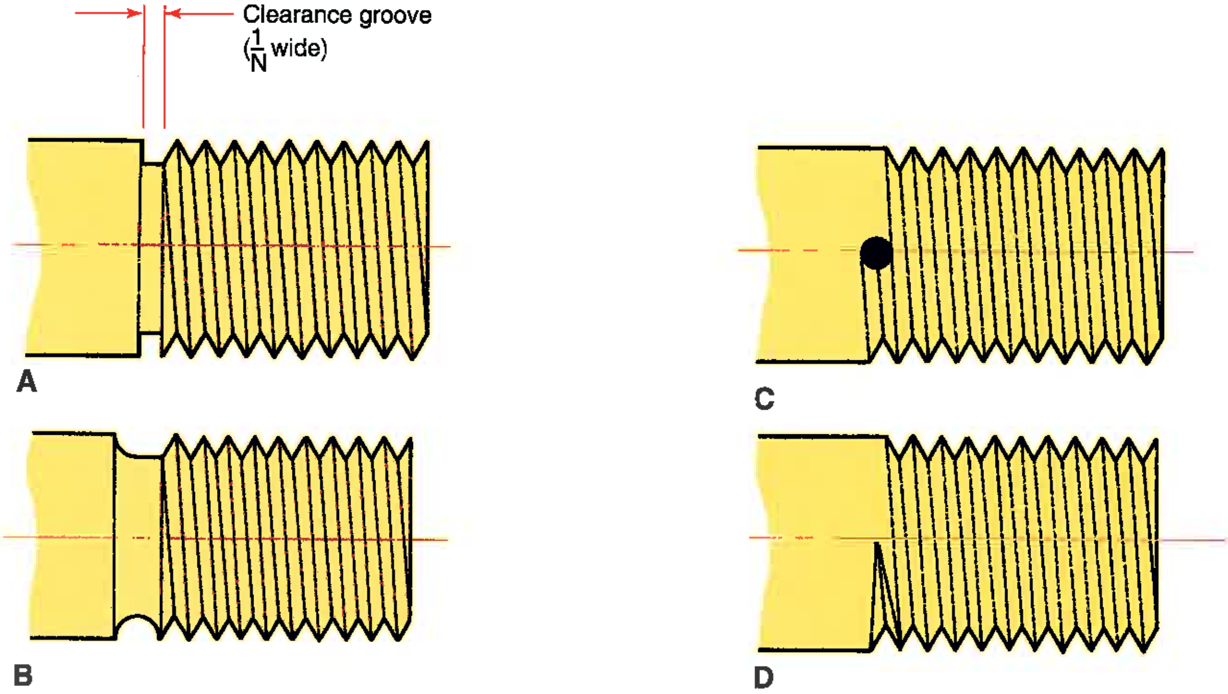


Figure 14-29. *Techniques for terminating a screw thread. A—Square groove. B— Round groove. C—Small shallow hole. D—Tool withdrawn from thread at end of cut.*

This is done with the aid of a center gage. Place the gage against the work while the tool is set into a V, **Figure 14-31**. Tool height can be set by using the centerline scribed on the tailstock spindle or with the center point.

The compound rest is set at 29° to permit the tool to shear the chip better than if it were fed straight into the work, **Figure 14-32**. Since the angle of the tool is 30° and it is fed in at an angle of 29°, the slight shaving action that results will produce a smooth finish on the right side of the thread. At the same time, not enough metal is removed to interfere with the main chip that is removed by the left edge of the tool.

Since the tool must be removed from the work after each cut and repositioned before the next cut can be started, a ***thread cutting stop*** may be used. After the point of the tool is set to just touch the work, lock the stop to the saddle dovetail with the adjusting screw just bearing on the stop, **Figure 14-33**.

After a cutting pass has been made, move the tool back from the work with the cross-slide screw. Move the carriage back to start another cut. Feed the tool into the work until the adjusting screw again bears against the thread cutting stop. By turning the compound rest in a distance of 0.002" to 0.005" (0.05 mm to 0.12 mm), the tool will be positioned for the next cut.

A thread dial that meshes with the lead screw is fitted to the carriage of most lathes, **Figure 14-34**. The ***thread dial*** is used to indicate when to engage the half-nuts, which permit the tool to follow exactly in the original cut. The thread dial eliminates the need to reverse spindle rotation after each cut to bring the tool back to the starting point.

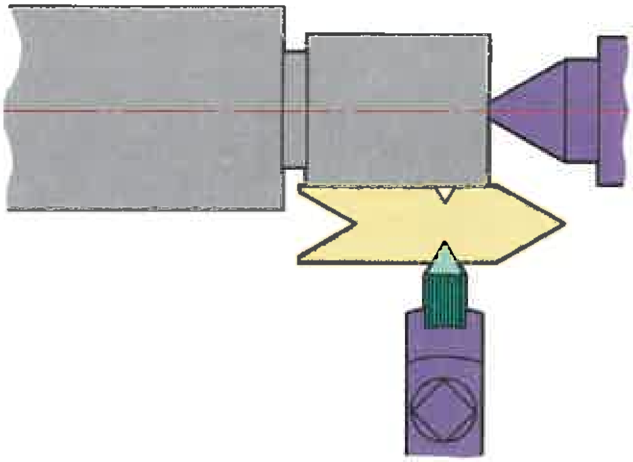


Figure 14-31. Positioning a cutting tool for machining threads, using a center gage.

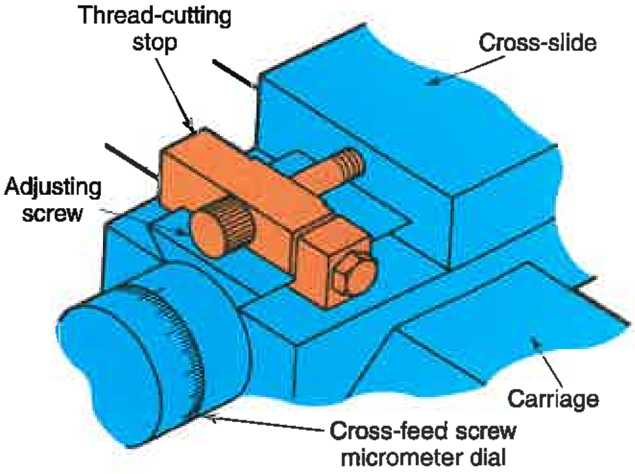


Figure 14-33. After being properly adjusted, the thread cutting stop will let you start next cut in same location.

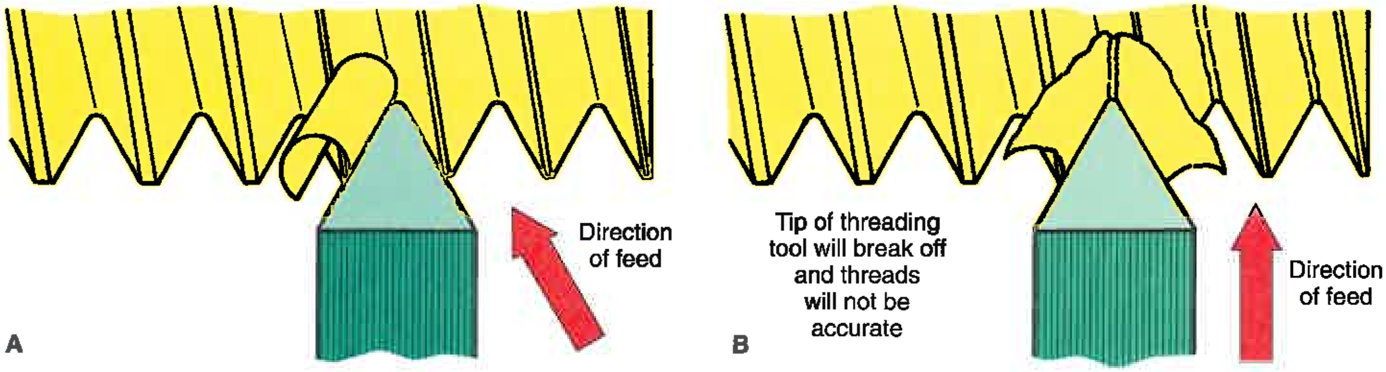


Figure 14-32. Cutting action of tool. A—When the tool is fed in at 29° angle, note that only one edge is cutting, and that the cutting load is distributed evenly across the edge. B—When fed straight in, note that both edges are cutting and weakest part of tool, the point, is doing hardest work.

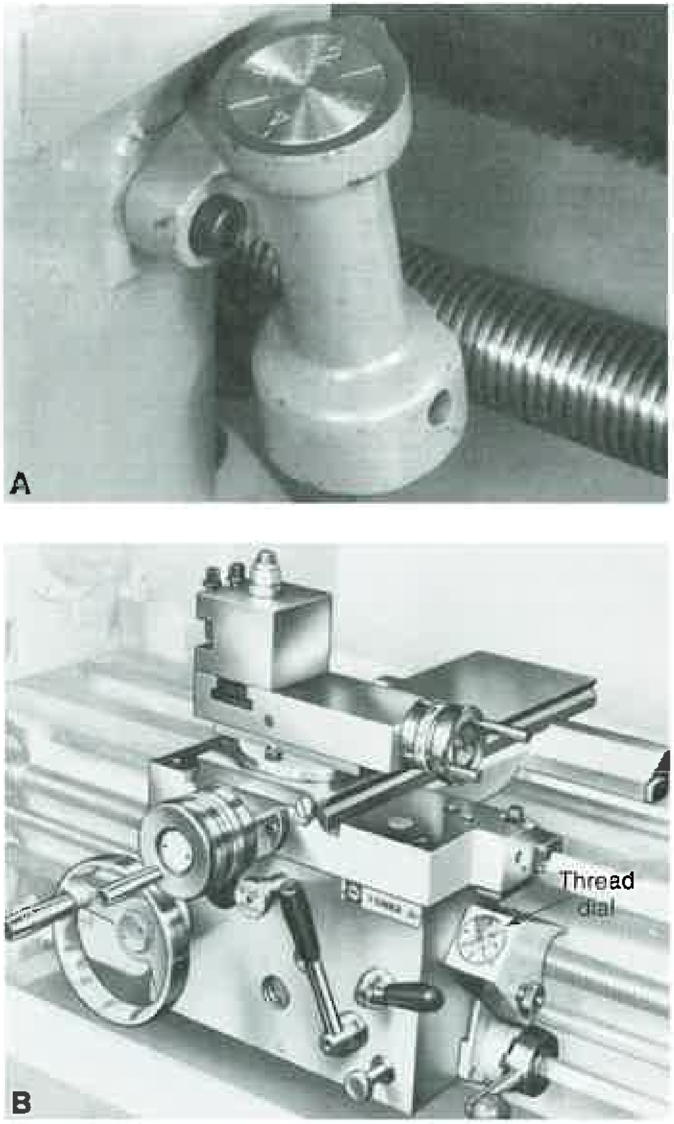


Figure 14-34. Thread dials. A—Thread dial for cutting inch-based threads. B—Dial used for cutting either inch-based or metric-based threads. The housing contains a series of gears, with gear selection depending upon threads being cut. (Clausing Industrial, Inc.)

The face of the thread dial, **Figure 14-35,** rotates when the half-nuts are *not* engaged. When the desired graduation moves into alignment with the index line, the half-nuts can be engaged.

The thread dial is used as follows for all inch-based threads:

- For all *even-numbered* threads, close the half-nuts at any line on the dial.
- For all *odd-numbered* threads, close the half-nuts at any numbered line on the dial.
- For all threads involving *one-half of a thread in each inch* (such as 11 1/2), close the half-nuts at any odd numbered line.

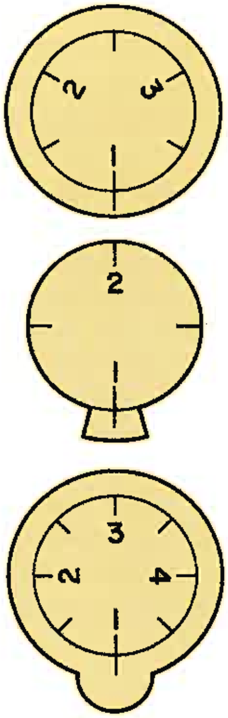


Figure 14-35. Typical thread dial faces.

- For all threads involving *one-fourth of a thread in each inch* (such as 4 3/4), return to the original starting line before closing the half-nuts.

On lathes that have been *converted* to metric threading capability, the thread dial cannot be used. When thread cutting with such a lathe, the half-nuts (once closed) must not be opened until the thread is completely cut. The spindle rotation must be reversed after each cut to return the tool to its starting position.

The thread dial *can* be used, however, on lathes with full metric capabilities. The thread dial will vary with the lathe manufacturer and must be considered individually. To be sure of correct thread dial procedure, consult the manufacturer's handbook for the machine.

14.6.3 Making the cut

Set the spindle speed to about one-fourth the speed that is used for conventional turning. Feed in the tool until it just touches the work. Then, move the tool beyond the right end of the work and adjust it to take a 0.002" (0.05 mm) cut.

Turn on the power and engage the half-nuts when indicated by the thread dial. This cut is made to check whether the lathe is producing the correct threads. Thread pitch can be checked with a rule or with a screw pitch gage, **Figure 14-36**. When

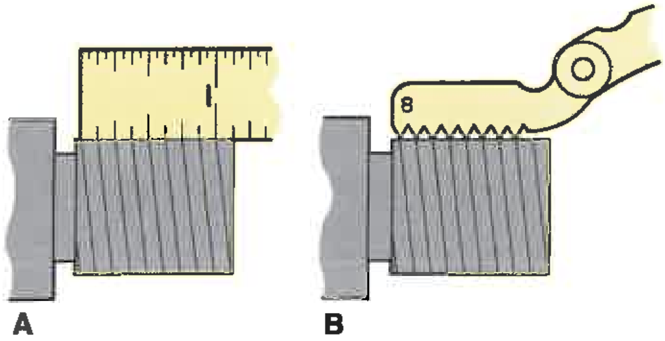


Figure 14-36. *Always check thread pitch after first light cut has been made. A—Checking with a rule. B—Checking with a screw pitch gage.*

everything checks, make additional cuts, working in 0.005" (0.12 mm) increments, until the thread is almost to size. The last few cuts should be no more than 0.002" (0.05 mm) deep. Note that all advances of the cutting tool are made with the compound rest feed screw.

A liberal application of cutting oil, before each cut, will help to obtain a smooth finish.

14.6.4 Resetting tool in thread

It is sometimes necessary to replace a broken cutting tool, or to resharpen it for the finish cuts. After replacing the tool, you must realign it with the portion of the thread already cut. This can be done as follows:

1. Set the tool on center and position it with a center gage.
2. Engage the half-nuts at the proper thread dial graduation.
3. Move the tool back from the work and rotate the spindle until the tool reaches a position about halfway down the threaded section.
4. Using the compound rest screw and the cross-slide screw, align the tool in the existing thread. Reset the thread cutting stop after the tool has been aligned.

14.6.5 Cutting threads with insert-type cutting tools

There are two basic types of 60° threading inserts, the partial profile insert and the full-profile insert.

Partial-profile inserts, **Figure 14-37,** are most commonly used because they can cut a range of thread pitches. However, the major diameter (OD) of the thread must be cut to size prior to threading.

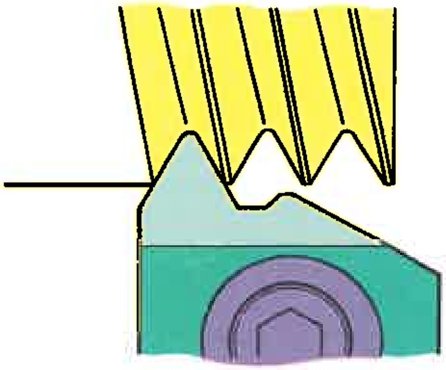


Figure 14-37. *Cutting threads with a partial-profile insert. The major (outside) diameter of the thread must be cut to size before using this type insert.*

Deburring may be required when cutting threads on most metals.

Full-profile inserts, **Figure 14-38,** produce the best thread form and finish. The tool cuts the leading flank, the root, and the trailing flank simultaneously. The machinist needs only to check the pitch diameter to determine if the major and minor diameters of the thread are to size. No deburring is necessary since the insert trims the thread crest. The disadvantage of the full-profile insert is that a separate insert is required for each thread pitch.

14.6.6 Measuring threads

Measure threads at frequent intervals during the machining operation to assure accuracy. The easiest way to check thread size is to try fitting the threaded piece into a threaded hole or nut of the proper size. If the piece does not fit, it is too large and further machining is necessary. This technique is not very accurate, but is usually satisfactory when close tolerances are not specified.

A ***thread micrometer*** can be used to make quick, accurate thread measurements. It has a pointed

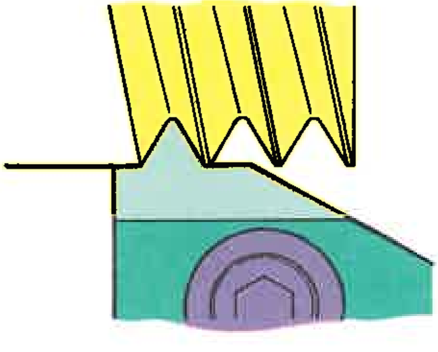


Figure 14-38. *Using a full-profile insert to cut a thread. A separate insert is required for each thread pitch.*

spindle and a double-V anvil to engage the thread. See **Figure 14-39.**

The micrometer reading given is the true pitch diameter. It equals the outside diameter of the screw minus the depth of one thread. Each micrometer is designed to read a limited number of thread pitches and is available in both inch and millimeter graduations.

Figure 14-39. *A thread micrometer can be used to check cut threads precisely. (L.S. Starrett Co.)*

The ***three-wire method of measuring threads*** has proven to be quite satisfactory. As shown in **Figure 14-40A,** three wires of a specific diameter are fitted into the threads and a micrometer measurement is made over the wires. The formula in **Figure 14-40B** will provide the information necessary to calculate the correct measurement over the wires.

A three-wire thread measuring system has been developed to simplify and speed up the measuring process. It consists of a digital micrometer mounted in a special fixture that holds the threaded workpiece and the three wires. See **Figure 14-41.**

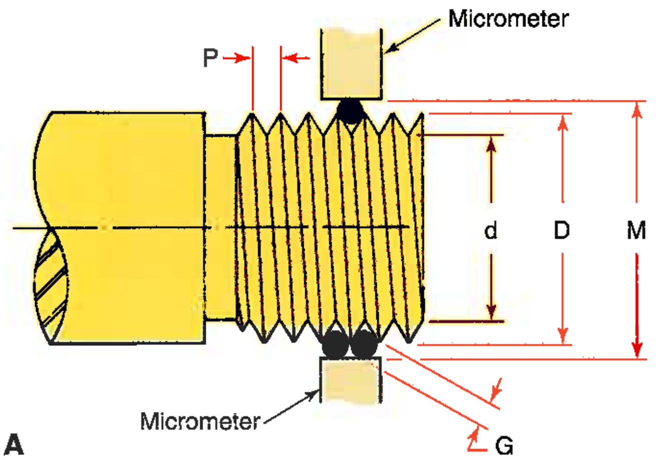
14.6.7 Cutting left-hand threads

Left-hand threads are cut in basically the same manner as right-hand threads. The major differences involve pivoting the compound to the *left* and changing the lead screw rotation so the carriage travels toward the tailstock (left to right), **Figure** 14-42.

14.6.8 Cutting square threads

Square threads are employed to transmit motion. They are more difficult to cut than 60° threads.

To cut a square thread, first calculate the width of the required tool bit (0.5 × thread pitch). If the



$$M = D + 3G - \frac{1.5155}{N}$$

Where: M = Measurement over the wires
D = Major diameter of thread
d = Minor diameter of thread
G = Diameter of wires

$$P = \text{Pitch} = \frac{1}{N}$$

N = Number of threads per inch

The smallest wire size that may be used for a given thread:

$$G = \frac{0.560}{N}$$

The largest wire size that can be used for a given thread:

$$G = \frac{0.900}{N}$$

The three-wire formula will work only if "G" is no larger or smaller than the sizes determined above. Any wire diameter between the two extremes may be used. All wires must be the same diameter.

B

Figure 14-40. *Three-wire method of measuring screw threads. A—Arrangement of the workpiece, wires, and micrometer. B—The three-wire thread measuring formula.*

Figure 14-41. *Thread measurements can be made in a fraction of the time normally needed with this new three-wire measuring system. Wires are mounted in individual holders that fit into the clamping fixture. (Mitutoyo/MTI Corp.)*

square thread is fairly coarse, a roughing tool is ground 0.010″ to 0.015″ (0.2 mm to 0.4 mm) smaller than the thread groove width. The cutting point of the finishing tool is ground 0.002″ to 0.003″ (0.05 mm to 0.08 mm) wider than the calculated groove width. Be sure adequate clearance is ground on the cutting tool, **Figure 14-43.**

14.6.9 Cutting Acme threads

On the *Acme thread,* the top and bottom are flat, but the sides have a 29° included angle. It was originally developed to replace the square thread. Its advantages are the strength and ease with which it can be cut, compared to the square thread. The thread form is employed in machine tools for precise control of component movement.

The *Acme screw thread gage* is the standard for grinding and setting Acme thread cutting tools. The tool angle is ground to fit a V in the thread gage. The width of the flat section varies with the pitch of the thread. This width is obtained by grinding back the tool point until it fits into the notch appropriate for the thread being cut. See **Figure 14-44.**

In cutting the threads, the groove is usually roughed out with a square nosed tool to approximate depth, then finished with an Acme-shaped tool. The compound rest is set to 14° and the tool is positioned using the thread gage, **Figure 14-45.** Other than this, Acme threads are cut in the same manner as the Sharp V thread.

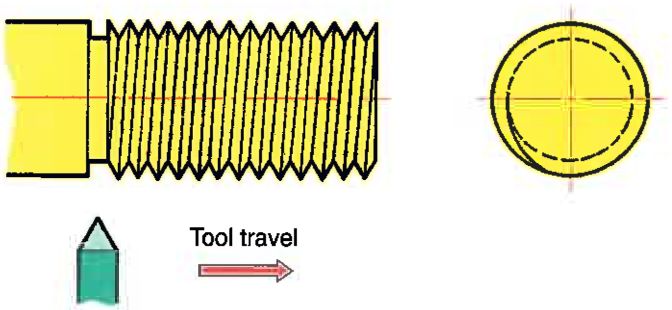


Figure 14-42. Direction of tool travel for cutting left-hand threads.

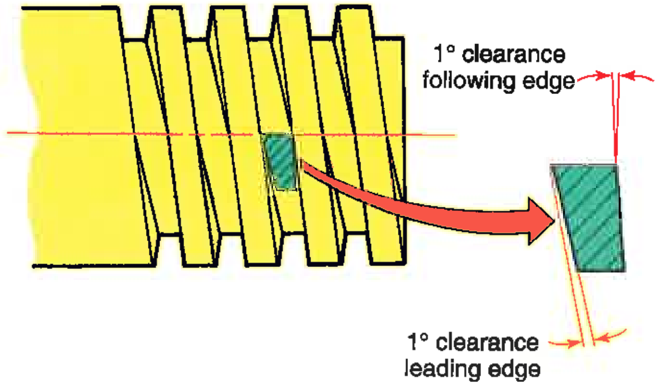


Figure 14-43. Allow adequate side clearance when sharpening a tool to cut square threads.

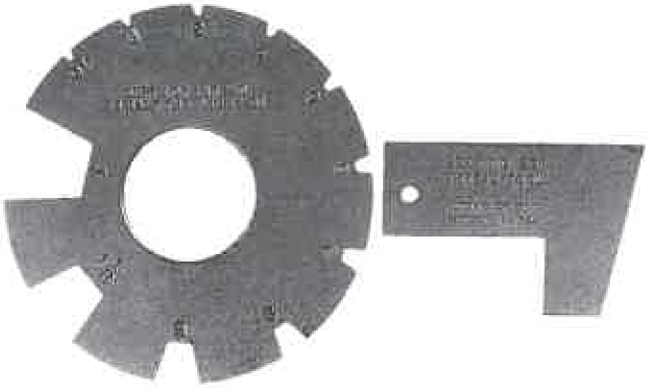


Figure 14-44. The Acme screw thread gage and tool setup gage will allow you to check lathe settings.

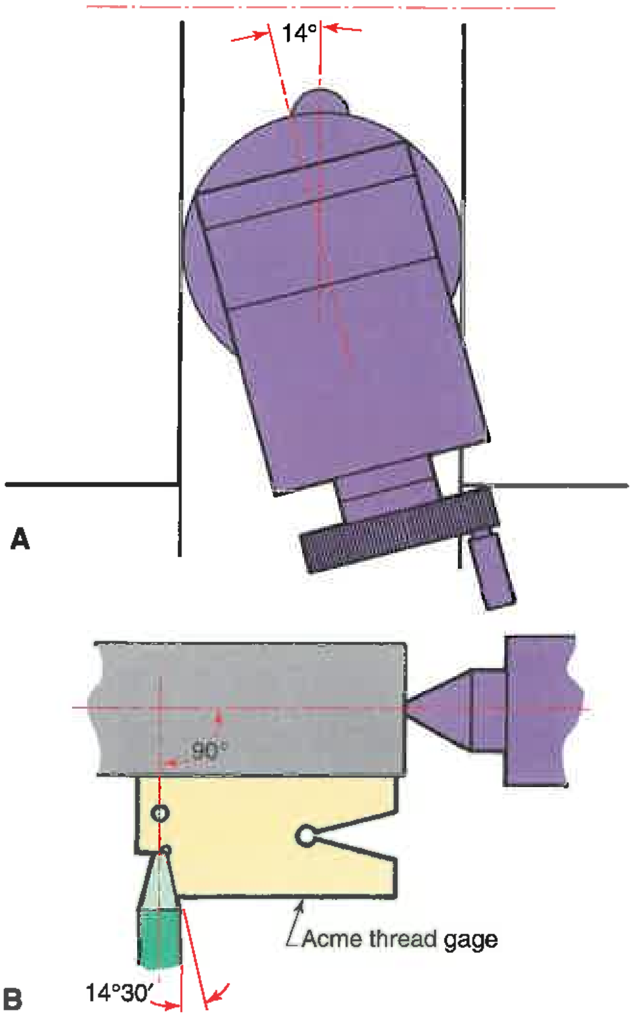


Figure 14-45. Cutting Acme threads. A—Compound setting for cutting Acme threads. B—Cutting tool is positioned with an Acme thread gage.

14.6.10 Cutting internal threads

Internal threads, **Figure 14-46,** are made on the lathe with a conventional boring bar and a cutting tool sharpened to the proper shape.

Before internal threads can be machined, the work must be prepared. A hole is drilled and bored to correct size for the thread's minor diameter. A recess is then machined with a square-nosed tool at the point where the thread terminates, **Figure 14-47.** The diameter of the recess is equal to the major diameter of the thread.

To cut right-hand internal threads, pivot the compound rest 29° to the *left,* as shown in **Figure 14-48.** Mount the tool on center and align it, using a center gage, **Figure 14-49.**

Bring the tool up until it just touches the work surface. Adjust the micrometer collar on the cross-slide to zero with the tool in position. Using the compound rest screw, adjust the cutter to make a cut of 0.002" (0.05 mm).

Remember that, when cutting *internal* threads, tool infeed and removal from the cut are the reverse of those used when cutting external threads.

A problem may arise in trying to determine when the tool has traveled far enough into the hole so the half-nuts can be disengaged. One method makes use of a line that has been lightly scribed in a blued area on the flat way of the lathe bed. The tool will have advanced far enough when the carriage reaches this point.

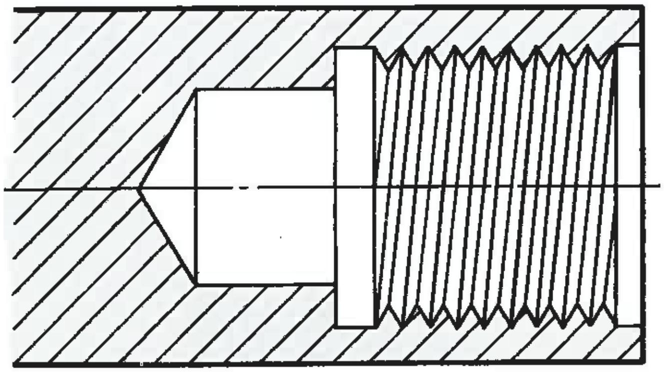


Figure 14-46. Internal threads.

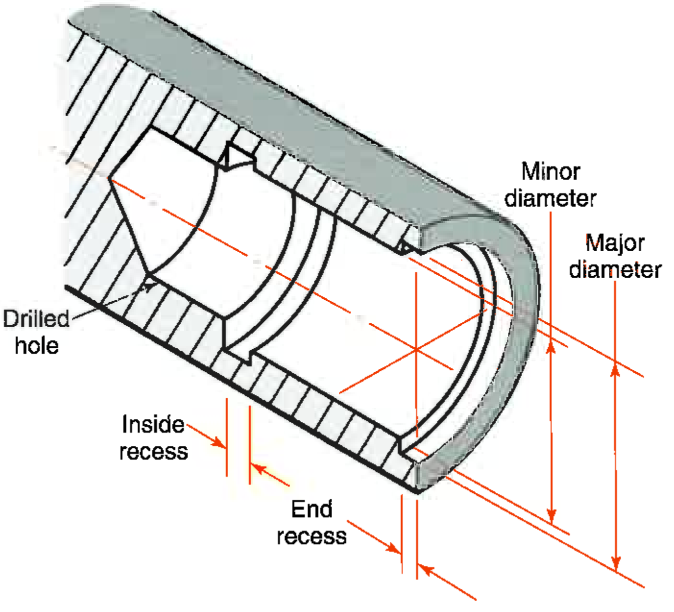


Figure 14-47. Opening for internal screw threads has been drilled and grooves machined to the major diameter.

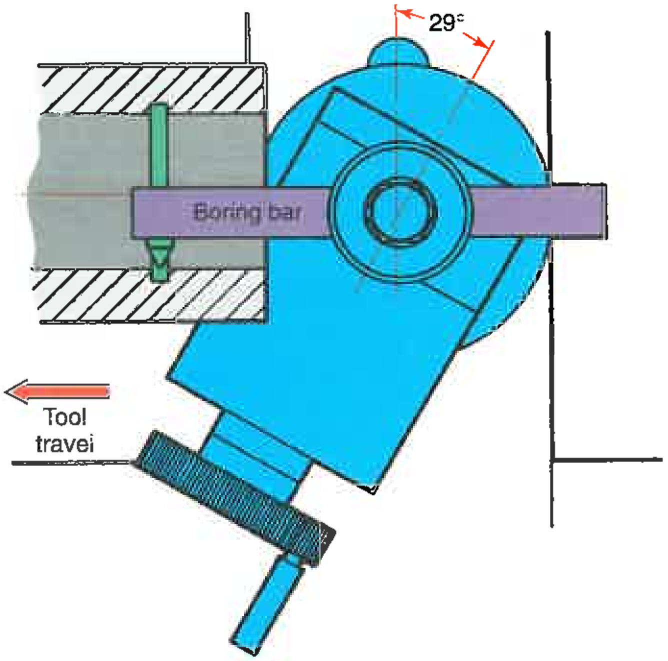


Figure 14-48. Compound setting for cutting internal right-hand screw threads.

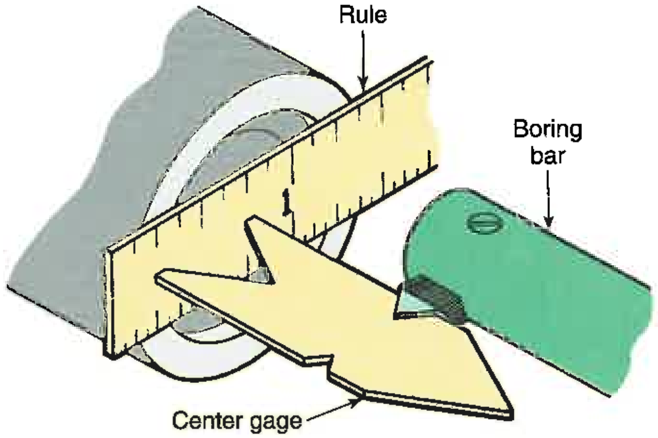


Figure 14-49. How to position cutting tool for machining internal screw threads.

Another technique allows you to start at the back of the hole when cutting internal threads. Pivot the compound rest 29° to the *right*. Place the threading tool to the rear of the boring bar with the cutting edge up. See **Figure 14-50**.

The lathe spindle is run in reverse. To prevent the tool from being placed too far into the hole to start the cut, mount a micrometer carriage stop on the ways. See **Figure 14-51**. The carriage is returned until it touches the stop. For cutting the threads, follow the same general procedure previously described.

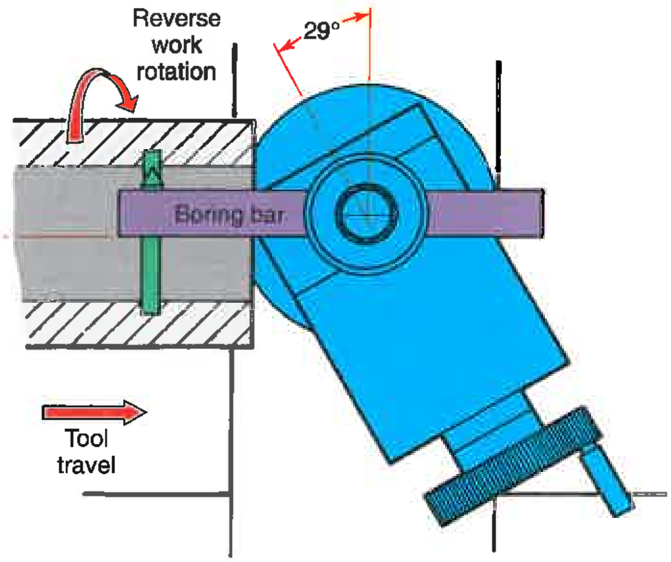


Figure 14-50. *An alternative setup for cutting internal right-hand threads. The work rotates in a direction opposite that of normal turning operations.*

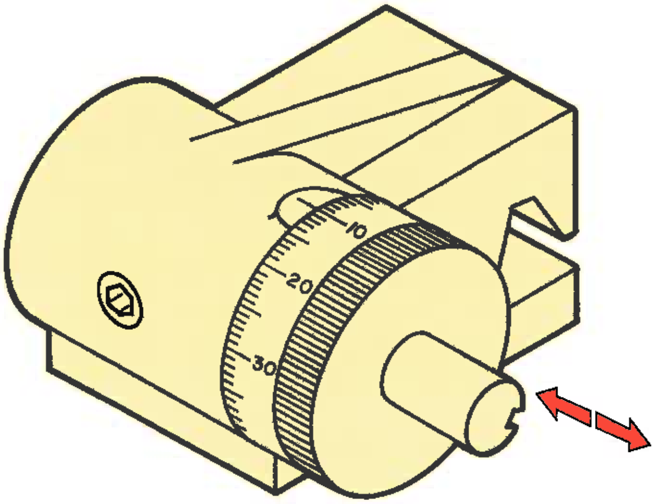


Figure 14-51. *When using alternate technique for cutting internal right-hand threads, mount a micrometer carriage stop on the ways. Adjust it to prevent the tool from being placed too far into hole when starting each cut.*

Continue making additional cuts until the threads are finished. Because the toolholder is not as rigid, lighter cuts must be taken when cutting internal threads than when machining external threads,. Keep the work flooded with cutting fluid.

14.6.11 Cutting threads on a taper surface

Tapered threads must be cut, at times, to obtain a fluid- or gas-tight joint. When this situation arises, the threading tool must be positioned in relation to the centerline of the taper rather than to the taper itself, **Figure 14-52**.

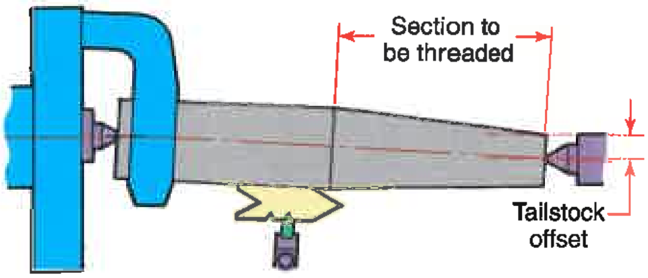


Figure 14-52. *Tool setup method for machining screw threads on a taper. Note that the tool is* not *positioned on the taper.*

TEST YOUR KNOWLEDGE

Please do not write in the text. Write your answers on a separate sheet of paper.

1. There are five ways of machining tapers on a lathe. List them, with their advantages and disadvantages.
2. When is a section of material considered tapered?
3. Machine adjustments must be calculated for each tapering job. The information given below will enable you to calculate the necessary tailstock setover for the problems given.

   Formula: When taper per inch is known,

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

   When taper per foot is known,

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

   When dimensions of tapered section are known but TPI or TPF is not given,

   $$\text{Offset} = \frac{L \times (D-d)}{2 \times \ell}$$

Where:

TPI = Taper per inch
TPF = Taper per foot
D = Diameter at large end of taper
d = Diameter at small end of taper
ℓ = Length of taper
L = Total length of piece

Note: These formulas, except for the TPF formula, can be used when dimensions are in mm.

Problem A: What will the tailstock setover be for the following job?

Taper per inch = 0.125″
Total length of piece = 4.000″

Problem B: What will the tailstock setover be for the following job?

D = 2.50″
d = 1.75″
ℓ = 6.00″
L = 9.00″

Problem C: What will the tailstock setover be for the following job?

D = 45.0 mm
d = 25.0 mm
ℓ = 175.0 mm
L = 275.0 mm

4. Screw threads are used for many reasons. List five or more important uses.

The following questions are of the matching type. Place the letter of the correct explanation on your paper.

5. _____ External thread.
6. _____ Internal thread.
7. _____ Major diameter.
8. _____ Minor diameter.
9. _____ Pitch diameter.
10. _____ Pitch.
11. _____ Lead.
    a. Smallest diameter of thread.
    b. Largest diameter of thread.
    c. Distance from one point on a thread to a corresponding point on next thread.
    d. Cut on outside surface of piece.
    e. Diameter of imaginary cylinder that would pass through threads at such points as to make width of thread and width of space at these points equal.
    f. Cut on inside surface of piece.
    g. Distance a nut will travel in one complete revolution of screw.

12. A groove is cut at the point where a thread is to terminate. It is cut to the depth of the thread and serves to:
    a. Provide a place to stop the threading tool after it makes a cut.
    b. Permits a nut to be run up to the end of the thread.
    c. Terminate the thread.
    d. All of the above.
    e. None of the above.

13. The tip of a cutting tool to cut a Sharp V thread is sharpened using a _____ to check that it is the correct shape. This tool is frequently called a _____.

14. The _____ is fitted to many lathe carriages. It meshes with the lead screw and is used to indicate when to engage the half-nuts to permit the thread cutting tool to follow exactly in the original cut.

15. The compound rest is set at _____ when cutting threads to permit the cutting tool to shear the material better than if it were fed straight into the work.

16. The three-wire thread measuring formula for inch-based threads is:

$$M = D + 3G - \frac{1.5155}{N}$$

Where: G = Wire diameter
D = Major diameter of thread (Convert to decimal size)
M = Measurement over the wires
N = Number of threads per inch

Problem: Calculate the correct measurement over the wires for the following threads. Use the wire size given in the problem.

a. 1/2-20 UNF (wire size 0.032″)
b. 1/4-20 UNC (wire size 0.032″)
c. 3/8-16 UNC (wire size 0.045″)
d. 7/16-14 UNC (wire size 0.060″)